GE Fanuc Automation

Computer Numerical Control Products

Series 16 / 18 / 160 / 180 – Model B

Connection Manual (Function)

GFZ-62443E-1/03

January 1995

GFL-001

Warnings, Cautions, and Notes as Used in this Publication

Warning

Warning notices are used in this publication to emphasize that hazardous voltages, currents, temperatures, or other conditions that could cause personal injury exist in this equipment or may be associated with its use.

In situations where inattention could cause either personal injury or damage to equipment, a Warning notice is used.

Caution

Caution notices are used where equipment might be damaged if care is not taken.

Note

Notes merely call attention to information that is especially significant to understanding and operating the equipment.

This document is based on information available at the time of its publication. While efforts have been made to be accurate, the information contained herein does not purport to cover all details or variations in hardware or software, nor to provide for every possible contingency in connection with installation, operation, or maintenance. Features may be described herein which are not present in all hardware and software systems. GE Fanuc Automation assumes no obligation of notice to holders of this document with respect to changes subsequently made.

GE Fanuc Automation makes no representation or warranty, expressed, implied, or statutory with respect to, and assumes no responsibility for the accuracy, completeness, sufficiency, or usefulness of the information contained herein. No warranties of merchantability or fitness for purpose shall apply.

Export Control Information

This document and the information contained in this document are classified with an Export Control Classification Number of 2E201. It is restricted and controlled under US export regulations. If you intend to export (or reexport), the document, directly or indirectly, or technical information relating thereto supplied hereunder or any portion thereof, it is your responsibility to ensure compliance with U.S. export control regulations and, if appropriate, to secure any required export licenses in your own name.

PREFACE

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. General
Describes feature of the function. Refer to Operator's manual as requied.

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

3. Parameters
Describes parameters related with a function.

4. Alarms and messages
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

The models covered by this manual, and their abbreviations are :

Product Name	Abbreviations	
FANUC Series 16–TB	16–TB	T series or T series (two–path control) *1
FANUC Series 160–TB	160–TB	
FANUC Series 16–MB	16–MB	M series or M series (two–path control) *1
FANUC Series 160–MB	160–MB	
FANUC Series 18–TB	18–TB	T series or T series (two–path control) *1
FANUC Series 180–TB	180–TB	
FANUC Series 18–MB	18–MB	M series
FANUC Series 180–MB	180–MB	

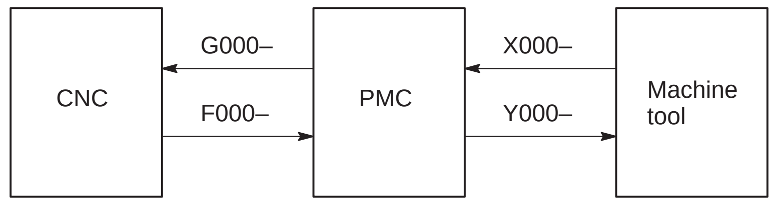
*1) In the case of two–path control is added.

> **Note**
> Some functions described in this manual may not be applied to some products.
> For details, refer to the DESCRIPTIONS manual (B–62442E).

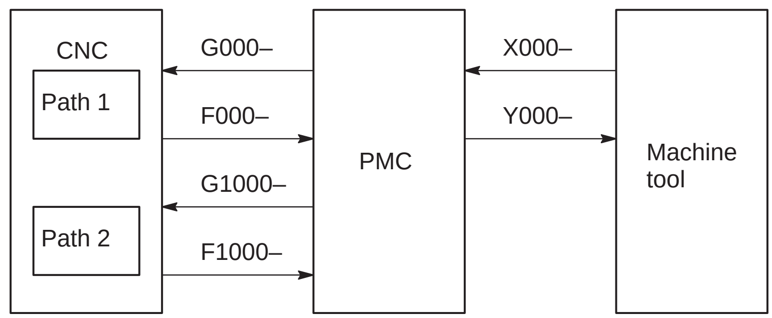
Signal description

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

[For one–path control]



[For two–path control]



Notes

1 In two–path control, the signals of the same functions are prepared for both of path 1 and path 2. These signals have suffix #1 and #2 to their signal names on path 1 and path 2, respectively.
When a signal is common to both paths, the signal is prepared only to path 1 and the suffix #1 and #2 are not attached.

2 In the context, signals are described on path 1 only. Refer to Appendix A.1.2 List of addresses for two–path control for signals on path 2.

- **Expression of signals**

One address accommodates eight signals.

Address / Symbol (#0 to #7 indicates bit position)

Address	#7	#6	#5	#4	#3	#2	#1	#0
F000	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 T series or M series does not have this signal. Upper part is for T series and lower part is for M series.

[Example 1]

Signal EXLM, ST is a common signal, STLK is for T series only and RLSOT and RVS are for M series only.

	#7	#6	…	#2	#1	#0	
G007	RLSOT	EXLM	…	ST	STLK	RVS	T series M series

Parameter description

Parameters are classified by data type as follows :

Dta type	Valid data range	Remarks
Bit	0 or 1	
Bit axis		
Byte	0 – ±127 0 – 255	In some parameters, signs are ignored.
Byte axis		
Word	0 – ±32767	
Word axis		
2–word	0 – ±99999999	
2–word axis		

Notes

1 For the bit type and bit axis type parameters, a single data number is assigned to 8 bits. Each bit has a different meaning.

2 The axis type allows data to be set separately for each control axis.

3 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.

- **Notation of bit type and bit axis type parameters**

Data No. / Data (#0 to #7 indicates bit position)

Data No.	#7	#6	#5	#4	#3	#2	#1	#0
0000			SEQ			INI	ISO	TVC

- **Notation of parameters other than bit type and bit axis type**

Data No.	Data
1023	Servo axis number of a specific axis

> **Notes**
> In an item where both T series and M series are described, parameters having different meanings between the T series and M series and parameters that are valid only for the T or M series are indicated in two levels as shown below. Parameters left blank are unavaliable.

[Example 1]

Parameter 5010 has different meanigs for the T series and M series.

5010	Tool nose radius compensation . . .	T series
	Cutter compensation C . . .	M series

[Example 2]

DPI is a parameter common to the M and T series, but GSB and GSC are parameters valid only for the T series.

	#7	#6			#0	
3401	GSC	GSB			DPI	T series
					DPI	M series

[Example 3]

The following parameter is provided only for the M series.

1450		T series
	F1 digit feed . . .	M series

Related Manuals

The table below lists manuals related to MODEL B of Series 16, Series 18, Series 160 and Series 180.
In the table, this manual is marked with an asterisk(*).

Table 1 Related manuals

Manual name	Specification Number	
DESCRIPTIONS	B–62442E	
CONNECTION MANUAL (HARDWARE)	B–62443E	

Table 1 Related manuals

Manual name	Specification Number	
CONNECTION MANUAL (FUNCTION)	B–62443E–1	*
OPERATOR'S MANUAL	B–62444E	
OPERATOR'S MANUAL	B–62454E	
MAINTENANCE MANUAL	B–62445E	
PARAMETER MANUAL	B–62450E	
PROGRAMMING MANUAL (Macro Compiler / Macro Executer)	B–61803E–1	
FAPT MACRO COMPILER PROGRAMMING MANUAL	B–66102E	
FANUC Super CAP T OPERATOR'S MANUAL	B–62444E–1	
FANUC Super CAP M OPERATOR'S MANUAL	B–62154E	
FANUC Super CAP M PROGRAMMING MANUAL	B–62153E	
CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION I FOR LATHE (Series 18–TB) OPERATOR'S MANUAL	B–61804E–1	
CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION FOR LATHE (Series 15–MODEL B, Series 16 CAP II) OPERATOR'S MANUAL	B–61804E–2	

Manuals related to the Control Motor α series

The table below lists manuals related to the Control Motor α series.

Table 2 Manuals related to the Control Motor α series

Document name	Document number	Major contents	Major usage
FANUC AC SERVO MOTOR α series DESCRIPTIONS	B–65142E	● Specification ● Characteristics ● External dimensions ● Connections	● Selection of motor ● Connection of motor
FANUC AC SPINDLE MOTOR α series DESCRIPTIONS	B–65152E	● Specification ● Characteristics ● External dimensions ● Connections	
FANUC CONTROL MOTOR AMPLIFIER α series DESCRIPTIONS	B–65162E	● Specifications and functions ● Installation ● External dimensions and maintenance area ● Connections	● Selection of amplifier ● Connection of amplifier
FANUC CONTROL MOTOR α series MAINTENANCE MANUAL	B–65165E	● Start up procedure ● Troubleshooting ● Maintenance of motor	● Start up the system (Hardware) ● Troubleshooting ● Maintenance of motor
FANUC AC SERVO MOTOR α series PARAMETER MANUAL	B–65150E	● Initial setting ● Setting parameters ● Description of parameters	● Start up the system (Software) ● Turning the system (Parameters)
FANUC AC SPINDLE MOTOR α series PARAMETER MANUAL	B–65160E	● Initial setting ● Setting parameters ● Description of parameters	

Table of Contents

1. AXIS CONTROL 1

1.1 CONTROLLED AXES 2
1.2 SETTING EACH AXIS 4
1.2.1 Name of Axes 4
1.2.2 Increment System 6
1.2.3 Specifying the Rotation Axis 9
1.2.4 Controlled Axes Detach 12
1.2.5 Outputting the Movement State of an Axis 15
1.2.6 Mirror Image 17
1.2.7 Follow–up 20
1.2.8 Servo Off (Mechanical handle) 22
1.2.9 Position Switch 24
1.3 ERROR COMPENSATION 27
1.3.1 Stored Pitch Error Compensation 27
1.3.2 Backlash Compensation 35
1.3.3 Straightness Compensation 37
1.4 SETTINGS RELATED TO SERVO–CONTROLLED AXES 41
1.4.1 Parameters Related to Servo 41
1.4.2 Absolute Position Detection 46
1.5 SETTINGS RELATED WITH COORDINATE SYSTEMS 47
1.5.1 Machine Coordinate System 47
1.5.2 Workpiece Coordinate System/Addition of Workpiece Coordinate System Pair 48
1.5.3 Rotary Axis Roll Over 54
1.6 SIMPLE SYNCHRONOUS CONTROL 56
1.7 TANDEM CONTROL (M SERIES) 64
1.8 SYNCHRONOUS CONTROL 73
1.9 SYNCHRONOUS CONTROL AND COMPOSITE CONTROL (T SERIES (TWO–PATH CONTROL)) 83
1.9.1 Overview 83
1.9.2 Synchronous Control 86
1.9.2.1 Setting 87
1.9.2.2 Programming 87
1.9.2.3 Signal operation 87
1.9.2.4 Examples of applications 87
1.9.2.5 Reference position return and its check during synchronous control 90
1.9.2.6 Out–of–synchronization detection 90
1.9.2.7 Move command after switching between independent control and synchronous control 91
1.9.2.8 Automatic setting of a workpiece coordinate system 91
1.9.3 Composite Control 93
1.9.3.1 Setting 93
1.9.3.2 Programming 93
1.9.3.3 Signal operation 93
1.9.3.4 Examples of applications 94
1.9.3.5 Spindle control 95
1.9.3.6 Tool offset during composite control 95
1.9.3.7 Reference position return during composite control 95
1.9.3.8 Move commands after the control mode is switched between independent control and composite control 95
1.9.4 Superimposed Control 96

1.9.4.1 Setting . . . 96
1.9.4.2 Programming . . . 96
1.9.4.3 Signal operation . . . 96
1.9.4.4 Examples of applications . . . 97
1.9.4.5 Feedrate . . . 97
1.9.4.6 Differences between superimposed control and ordinary synchronous control . . . 98
1.9.5 Signal . . . 98
1.9.6 Parameter . . . 100
1.9.7 Alarms and Messages . . . 110
1.9.7.1 P/S alarms . . . 110
1.9.7.2 Servo alarms . . . 110
1.9.8 Cautions . . . 111
1.9.8.1 Items common to synchronous, composite, and superimposed control . . . 111
1.9.8.2 Items related only to synchronous control . . . 112
1.9.8.3 Restrictions imposed during synchronous, composite, and superimposed control . . . 112
1.9.8.4 Reading the coordinates during synchronous, composite, or superimposed control . . . 114
1.9.8.5 Terminating synchronous, composite, or superimposed control . . . 114
1.9.8.6 Status output signals for an axis under synchronous, composite, or superimposed control . . . 116
1.9.9 Examples of Applications . . . 117
1.9.9.1 Independent control and synchronous control of the Z1– and Z2–axes . . . 117
1.9.9.2 Independent control and interpolation for the X1– and Z2–axes . . . 119
1.9.9.2.1 Using synchronous control . . . 120
1.9.9.2.2 Using composite control . . . 121
1.9.9.3 Independent control and interpolation between the X1– and Z2–axes and between the X2– and Z1–axes . . . 124
1.9.9.4 Independent control and superimposed control for the Z1– and Z2–axes . . . 128
1.9.9.5 Miscellaneous . . . 130
1.9.10 Troubleshooting . . . 131

2. PREPARATIONS FOR OPERATION . . . 135

2.1 EMERGENCY STOP . . . 136
2.2 CNC READY SIGNAL . . . 139
2.3 OVERTRAVEL CHECK . . . 141
2.3.1 Overtravel Signal . . . 141
2.3.2 Stored Stroke Limit 1 . . . 144
2.3.3 Stored Stroke Limit 2, 3 . . . 147
2.3.4 Chuck/Tailstock Barrier (T series) . . . 154
2.3.5 Tool Post Interference Check (T series, Two–path Control) . . . 160
2.4 ALARM SIGNAL . . . 165
2.5 START LOCK/INTERLOCK . . . 167
2.6 MODE SELECTION . . . 171
2.7 PATH SELECTION/DISPLAY OF OPTIONAL PATH NAMES . . . 178
2.8 STATUS OUTPUT SIGNAL . . . 180
2.9 VRDY OFF ALARM IGNORE SIGNAL . . . 182
2.10 ABNORMAL LOAD DETECTION . . . 184

3. MANUAL OPERATION . . . 194

3.1 JOG FEED/INCREMENTAL FEED . . . 195
3.2 MANUAL HANDLE FEED . . . 202

3.3 MANUAL HANDLE INTERRUPTION 208
3.4 TOOL AXIS DIRECTION HANDLE FEED FUNCTION (M SYSTEM) 210

4. REFERENCE POSITION ESTABLISHMENT 214

4.1 MANUAL REFERENCE POSITION RETURN 215
4.2 SETTING THE REFERENCE POSITION WITHOUT DOGS 227
4.3 REFERENCE POSITION SHIFT 234
4.4 REFERENCE POSITION RETURN 237
4.5 2ND REFERENCE POSITION RETURN/3RD, 4TH REFERENCE POSITION RETURN 239
4.6 FLOATING REFERENCE POSITION RETURN 242

5. AUTOMATIC OPERATION 245

5.1 CYCLE START/FEED HOLD 246
5.2 RESET AND REWIND 251
5.3 TESTING A PROGRAM 255
5.3.1 Machine Lock 255
5.3.2 Dry Run 258
5.3.3 Single Block 261
5.4 MANUAL ABSOLUTE ON/OFF 263
5.5 OPTIONAL BLOCK SKIP/ADDITION OF OPTIONAL BLOCK SKIP 266
5.6 SEQUENCE NUMBER COMPARISON AND STOP 269
5.7 PROGRAM RESTART 270
5.8 TOOL RETRACTION AND RETURN (M SERIES) 273
5.9 EXACT STOP/EXACT STOP MODE/TAPPING MODE/CUTTING MODE (M SERIES) 277
5.10 BALANCE CUT(2–PATH CONTROL FOR T SYSTEM) 279
5.11 DNC OPERATION 281
5.12 MANUAL INTERVENTION AND RETURN 283

6. INTERPOLATION FUNCTION 284

6.1 POSITIONING 285
6.2 LINEAR INTERPOLATION 287
6.3 CIRCULAR INTERPOLATION 290
6.4 THREAD CUTTING 295
6.4.1 Thread Cutting 295
6.4.2 Thread Cutting Cycle Retract (T series) 300
6.5 SINGLE DIRECTION POSITIONING (M SERIES) 302
6.6 HELICAL INTERPOLATION 304
6.7 INVOLUTE INTERPOLATION (M SERIES) 306
6.8 POLAR COORDINATE INTERPOLATION 307
6.9 CYLINDRICAL INTERPOLATION 309
6.10 POLYGONAL TURNING (T SERIES) 312
6.10.1 Polygonal Turning 313
6.10.2 Polygonal Turning with Two Spindles 318
6.11 NORMAL DIRECTION CONTROL (M SERIES) 334

7. FEEDRATE CONTROL/ACCELERATION AND DECELERATION CONTROL 338

7.1 FEEDRATE CONTROL 339

7.1.1 Rapid Traverse Rate . . . 339
7.1.2 Cutting Feedrate Clamp . . . 342
7.1.3 Feed per Minute . . . 344
7.1.4 Feed per Revolution/Manual Feed per Revolution . . . 347
7.1.5 F1-digit Feed (M series) . . . 349
7.1.6 Feedrate Inverse Time Specification (M series) . . . 352
7.1.7 Override . . . 353
7.1.7.1 Rapid traverse override . . . 353
7.1.7.2 Feedrate override . . . 356
7.1.7.3 Second feedrate override . . . 357
7.1.7.4 Override cancel . . . 358
7.1.8 Automatic Corner Override (M series) . . . 359
7.1.9 External Deceleration . . . 363
7.1.10 Feed Stop Function . . . 365
7.1.11 Feedrate Clamping By Arc Radius . . . 366
7.1.12 Automatic Corner Deceleration (M series) . . . 368
7.1.13 Look–Ahead Control . . . 376
7.1.14 High–precision Contour Control by RISC (M series) . . . 384
7.1.14.1 Look–ahead acceleration/deceleration before interpolation . . . 386
7.1.14.2 Automatic feedrate control function . . . 397
7.1.14.3 Signal . . . 404
7.1.14.4 Parameter . . . 405
7.1.14.5 Alarm and message . . . 413
7.1.14.6 Note . . . 414
7.2 ACCELERATION/DECELERATION CONTROL . . . 416
7.2.1 Automatic Acceleration/Deceleration . . . 416
7.2.2 Bell–shaped Acceleration/Deceleration for Rapid Traverse . . . 421
7.2.3 Linear Acceleration/Deceleration after Interpolation for Cutting Feed . . . 424
7.2.4 Bell Shaped Acceleration/Deceleration after Cutting Feed . . . 428
7.2.5 Linear Acceleration/Deceleration of Cutting Feed before Interpolation . . . 431
7.2.6 Corner Control . . . 436
7.2.6.1 In–position check . . . 436
7.2.6.2 In–position check independently of feed/rapid traverse . . . 437
7.2.6.3 Error detect (T series) . . . 439
7.2.7 Feed Forward in Rapid Traverse . . . 441

8. AUXILIARY FUNCTION . . . 442

8.1 MISCELLANEOUS FUNCTION/2ND AUXILIARY FUNCTION . . . 443
8.2 AUXILIARY FUNCTION LOCK . . . 454
8.3 MULTIPLE M COMMANDS IN A SINGLE BLOCK . . . 456
8.4 HIGH–SPEED M/S/T/B INTERFACE . . . 459
8.5 WAITING M CODE (TWO–PATH CONTROL) . . . 464
8.6 M CODE LOOP CHECK FUNCTION . . . 466

9. SPINDLE SPEED FUNCTION . . . 470

9.1 SPINDLE SPEED FUNCTION (S CODE OUTPUT) . . . 471
9.2 SPINDLE SERIAL OUTPUT/SPINDLE ANALOG OUTPUT . . . 472
9.3 SPINDLE SPEED CONTROL . . . 480
9.4 SPINDLE SPEED CONTROL FOR TWO–PATH LATHE . . . 507

9.5 CONSTANT SURFACE SPEED CONTROL . . . 517
9.6 SPINDLE SPEED FLUCTUATION DETECTION . . . 522
9.7 ACTUAL SPINDLE SPEED OUTPUT . . . 527
9.8 SPINDLE POSITIONING (T SERIES) . . . 528
9.9 CS CONTOUR CONTROL . . . 548
9.10 MULTI-SPINDLE CONTROL (T SERIES) . . . 563
9.11 RIGID TAPPING . . . 574
9.11.1 General . . . 574
9.11.2 Connection among Spindle, Spindle Motor, and Position Coder . . . 576
9.11.3 Rigid Tapping Specification . . . 581
9.11.4 Display Data on the Diagnosis Screen . . . 582
9.11.5 Command format . . . 586
9.11.6 Signal . . . 590
9.11.6.1 Signals for the rigid tapping function . . . 590
9.11.6.2 Signals related to S code output . . . 591
9.11.6.3 Signals related to gear switching . . . 592
9.11.6.4 Signals related to second spindle rigid tapping (T series only) . . . 594
9.11.6.5 Signal addresses . . . 596
9.11.6.6 Notes on interface with the PMC . . . 596
9.11.7 Timing Charts for Rigid Tapping Specification . . . 599
9.11.7.1 When M29 is specified before G84 (G74) . . . 600
9.11.7.2 M29 and G84 (G74) are specified in the same block . . . 604
9.11.7.3 Specifying G84 (G74) for rigid tapping by parameters . . . 608
9.11.7.4 Timing to cancel rigid tapping mode . . . 612
9.11.8 Parameter . . . 614
9.11.9 Alarm and Message . . . 627
9.11.10 Notes . . . 629
9.11.11 Reference Item . . . 633
9.12 SPINDLE SYNCHRONOUS CONTROL . . . 634
9.13 SPINDLE ORIENTATION . . . 638
9.14 SPINDLE OUTPUT SWITCHING . . . 640

10. TOOL FUNCTIONS . . . 642

10.1 TOOL FUNCTION . . . 643
10.2 TOOL COMPENSATION VALUE/TOOL COMPENSATION NUMBER/ TOOL COMPENSATION MEMORY . . . 646
10.3 TOOL LIFE MANAGEMENT . . . 653
10.4 CUTTER COMPENSATION . . . 661
10.4.1 Cutter Compensation B, C (M series) . . . 661
10.4.2 Tool Nose Radius Compensation (T series) . . . 665

11. PROGRAM COMMAND . . . 668

11.1 DECIMAL POINT PROGRAMMING/POCKET CALCULATOR TYPE DECIMAL POINT PROGRAMMING . . . 669
11.2 G CODE SYSTEM (T SERIES) . . . 671
11.3 PROGRAM CONFIGURATION . . . 675
11.4 INCH/METRIC CONVERSION . . . 678
11.5 HIGH SPEED CYCLE CUTTING . . . 683
11.6 CUSTOM MACRO . . . 690

11.6.1 Custom Macro 690
11.6.2 Interruption Type Custom Macro 700
11.6.3 Custom Macro Variables Common To Two–path Control (T series (Two–path control)) 703
11.7 CANNED CYCLE (M SERIES) / CANNED CYCLE FOR HOLE MACHINING (T SERIES) . . . 705
11.8 EXTERNAL MOTION FUNCTION (M SERIES) 715
11.9 CANNED CYCLE (T SERIES)/MULTIPLE REPETITIVE CANNED CYCLE (T SERIES) 717
11.10 MIRROR IMAGE FOR DOUBLE TURRETS (T SERIES) 724
11.11 INDEX TABLE INDEXING FUNCTION (M SERIES) 726
11.12 SCALING (M SERIES) 734
11.13 COORDINATE SYSTEM ROTATION 738
11.14 THREE–DIMENSIONAL COORDINATE CONVERSION (M SERIES) 741
11.15 RETRACE (M SERIES) 743
11.16 MACRO COMPILER/EXECUTER 747
11.17 SMALL HOLE PECK DRILLING CYCLE (M SERIES) 748

12. DISPLAY / SET / EDIT 755

12.1 DISPLAY/SET 756
12.1.1 Clock Function 756
12.1.2 Displaying Operation History 757
12.1.3 Help Function 759
12.1.4 Displaying Alarm History 760
12.1.5 Servo Tuning Screen 761
12.1.6 Spindle Tuning Screen 762
12.1.7 Waveform Diagnosis Display 763
12.1.8 Self–diagnosis 765
12.1.9 Display of Hardware and Software Configuration 766
12.1.10 Position Display Neglect 767
12.1.11 Run Hour and Parts Count Display 769
12.1.12 Graphic Display/Dynamic Graphic Display 774
12.1.13 Displaying Operating Monitor 781
12.1.14 Stamping the Machining Time 783
12.1.15 Software Operator’s Panel 785
12.1.16 Multi–language Display 795
12.1.17 Remote Diagnosis 796
12.1.18 External Operator Message Logging and Display 802
12.2 EDIT 803
12.2.1 Part Program Storage Length 803
12.2.2 No. of Registered Programs 804
12.2.3 Memory Protection Key 805
12.2.4 Password Function 807
12.2.5 Background Editing 809
12.2.6 Playback 810
12.2.7 Conversational Programming with Graphic Function 811

13. INPUT/OUTPUT OF DATA 812

13.1 READER/PUNCHER INTERFACE 813
13.2 REMOTE BUFFER 820

13.3 DNC1 INTERFACE 821
13.4 DNC2 INTERFACE 829
13.5 EXTERNAL I/O DEVICE CONTROL 830
13.6 SIMULTANEOUS INPUT AND OUTPUT OPERATIONS (M SERIES) 837
13.7 EXTERNAL PROGRAM INPUT 839

14. MEASUREMENT 843

14.1 TOOL LENGTH MEASUREMENT (M SERIES) 844
14.2 AUTOMATIC TOOL LENGTH MEASUREMENT (M SERIES) / AUTOMATIC TOOL OFFSET (T SERIES) 845
14.3 SKIP FUNCTION 851
14.3.1 Skip Function 851
14.3.2 High–speed Skip Signal 854
14.3.3 Multi–step Skip (T series) 858
14.3.4 Torque Limit Skip (T series) 862
14.4 ENTERING COMPENSATION VALUES 865
14.4.1 Input of Offset Value Measured A (T series) 865
14.4.2 Input of Tool Offset Value Measured B (T series) 867
14.4.3 Direct Input of Workpiece Zero Point Offset Value Measured 878

15. PMC CONTROL FUNCTION 879

15.1 PMC AXIS CONTROL/PMC AXIS SPEED CONTROL FUNCTION 880
15.2 EXTERNAL DATA INPUT 918
15.3 EXTERNAL WORKPIECE NUMBER SEARCH 927
15.4 SPINDLE OUTPUT CONTROL BY THE PMC 930
15.5 EXTERNAL KEY INPUT 937
15.6 DIRECT OPERATION BY PMC/MMC 942

APPENDIX

A. INTERFACE BETWEEN CNC AND PMC 945

A.1 LIST OF ADDRESSES 946
A.1.1 List of Addresses (One–path Control) 946
A.1.2 List of Addresses (Two–path Control) 964
A.2 LIST OF SIGNALS 1001
A.2.1 List of Signals (In Order of Functions) 1001
A.2.2 List of Signals (In Order of Symbols) 1017
A.2.3 List of Signals (In Order of Addresses) 1032

1 AXIS CONTROL

1.1 CONTROLLED AXES

General

Series 16, Series 160

Item		M series	T series
No. of basic controlled axes	1–path	3 axes	2 axes
	2–path	3 axes per path	2 axes per path
Controlled axes expansion (total)	1–path	Max. 8 axes (Including the Cs axis)	Max. 8 axes (Including the Cs axis)
	2–path	Max. 7 axes per path (Feed 6 axes + Cs axis)	Max. 7 axes per path (Feed 6 axes + Cs axis)
Basic simultaneously controlled axes	1–path	2 axes	2 axes
	2–path	2 axes per path	2 axes per path
Simultaneously controlled axes expansion (total)	1–path	Max. 6 axes	Max. 6 axes
	2–path	Max. 6 axes per path	Max. 6 axes per path

Series 18, Series 180

Item		M series	T series
No. of basic controlled axes	1–path	3 axes	2 axes
	2–path	—	2 axes per path
Controlled axes expansion (total)	1–path	Max. 6 axes (Including the Cs axis)	Max. 6 axes (Including the Cs axis)
	2–path	—	Max. 5 axes per path (Feed 4 axes + Cs axis)
Basic simultaneously controlled axes	1–path	2 axes	2 axes
	2–path	—	2 axes per path
Simultaneously controlled axes expansion (total)	1–path	Max. 4 axes	Max. 4 axes
	2–path	—	Max. 4 axes per path

Parameter

1010	Number of CNC–controlled axes

[Data type] Byte

[Valid data range] 1, 2, 3, ..., the number of controlled axes

Set the maximum number of axes that can be controlled by the CNC.

[Example] Suppose that the first axis is the X axis, and the second and subsequent axes are the Y, Z, A, B, and C axes in that order, and that they are controlled as follows:

X, Y, Z, and A axes: Controlled by the CNC and PMC
B and C axes: Controlled by the PMC

Then set this parameter to 4 (total 4: X, Y, Z, and A)

Alarm and message

Number	Message	Description
015	TOO MANY AXES COMMANDED (M series)	The number of the commanded axes exceeded that of simultaneously controlled axes. Correct the program.
	P/S ALARM (T series)	An attempt was made to move the machine along the axes, bu the number of the axes exceeded the specified number of axes controlled simultaneously. Alternatively, in a block where the skip function activated by the torque–limit reached signal (G31 P99/P98) was specified, either moving the machine along an axis was not specified, or moving the machine along multiple axes was specified. Specify movement only along one axis.

Note

Note 1 When the 9” CRT is fitted, the overall position display screen and the position display screen for manual handle interrupt can display up to eight axes. The positions of the 9th and 10th axes are not displayed on these screens when used with 2–path control having nine or more axes.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.2.1	Controlled Axes
OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.2.1	Controlled Axes

1.2 SETTING EACH AXIS

1.2.1 Name of Axes

General

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.
With 2–path control, the name of basic axis for one path is fixed; the names of additional axes can be optionally selected from axes names except axes names of basic axes by using parameter No. 1020. For one path, the same axis name cannot be assigned to multiple axes, but the same axis name can be used with the other path.

Parameter

1020	Name of the axis used for programming for each axis

[Data type] Byte axis

Set the name of the program axis for each control axis, with one of the values listed in the following table:

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

Note 1 In the T series, when G code system A is used, neither U, V, nor W can be used as an axis name. Only when G code system B or C is used, U, V, and W can be used as axis names.

Note 2 The same axis name cannot be assigned to more than one axis.

Note 3 When the secondary auxiliary function is provided, address B cannot be used as an axis name. In the T series, when CCR, #4 of parameter 3405, is set to 1, address A and C may not be used with functions such as chamfering, corner R, or direct drawing dimensions programming.

Note

Note 1 With 2–path control, when information (such as the current position) about each axis is displayed on the CRT screen, an axis name may be followed by a subscript to indicate a path number (e.g.,X1 and X2). This is axis name to help the user to easily understand which path an axis belongs to. When writing a program, the user must specify X, Y, Z, U, V, W, A, B, and C without attaching a subscript.

Reference item

OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.2.2	NAMES OF AXES
OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.2.2	NAMES OF AXES

1.2.2
Increment System

General

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.
The increment system is classified into IS–B and IS–C (Tables 1.2.2(a) and 1.2.2 (b)). Select IS–B or IS–C using bit 1 (ISC) of parameter 1004. When selecting IS–C, the option of increment system 1/10 is necessary.

Table 1.2.2 (a) Increment system IS–B

		Least input increment	Least command increment
Metric system machine	mm input	0.001mm(Diameter)	0.0005mm
		0.001mm(Radius)	0.001mm
		0.001deg	0.001deg
	inch input	0.0001inch(Diameter)	0.0005mm
		0.0001inch(Radius)	0.001mm
		0.001deg	0.001deg
Inch system machine	mm input	0.001mm(Diameter)	0.00005inch
		0.001mm(Radius)	0.0001inch
		0.001deg	0.001deg
	inch input	0.0001inch(Diameter)	0.00005inch
		0.0001inch(Radius)	0.0001inch
		0.001deg	0.001deg

Table 1.2.2 (b) Increment system IS–C

		Least input increment	Least command increment
Metric system machine	mm input	0.0001mm(Diameter)	0.00005mm
		0.0001mm(Radius)	0.0001mm
		0.0001deg	0.0001deg
	inch input	0.00001inch(Diameter)	0.00005mm
		0.00001inch(Radius)	0.0001mm
		0.0001deg	0.0001deg
Inch system machine	mm input	0.0001mm(Diameter)	0.000005inch
		0.0001mm(Radius)	0.00001inch
		0.0001deg	0.0001deg
	inch input	0.00001inch(Diameter)	0.000005inch
		0.00001inch(Radius)	0.00001inch
		0.0001deg	0.0001deg

Note 1 Diameter programming is used only for T series. Whether diameter programming or radius programming is used is selected by parameter DIAx (No. 1006#3) on each axis. Also, parameter IPR (No. 1004#7) can make the least input increment of IS–B and IS–C ten times the least command increment on each axis.

Parameter

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

Setting entry is acceptable.

[Data type] Bit

INI Unit of input
0 : In mm
1 : In inches

	#7	#6	#5	#4	#3	#2	#1	#0
1001								INM

Note 1 When this parameter is set, the power must be turned off before operation is continued.

[Data type] Bit

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						ISC	
	IPR						ISC	ISA

[Data type] Bit

ISA, ISC The least input increment and least command increment are set.

IS–C	IS–A	Least input increment and least command increment	Symbol
0	0	0.001mm, 0.001deg, or 0.0001inch	IS–B
0	1	0.01mm, 0.01deg, or 0.001inch	IS–A
1	0	0.0001mm, 0.0001deg, or 0.00001inch	IS–C

Note 1 IS–A cannot be used at present.

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 and IS–C.
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	0.01mm, 0.01deg, or 0.001inch
IS–C	0.0001mm, 0.0001deg, or 0.00001inch

Note 1 For IS–A, the least input increment cannot be set to a value 10 times as large as the least command increment.

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

Note 1 When this parameter is changed, turn off the power before continuing operation.

[Data type] Bit axis

DIAx Either a diameter or radius is set to be used for specifying the amount of travel on each axis.
0 : Radius
1 : Diameter

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.2.3	Increment System
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.2.3	Increment System

1.2.3 Specifying the Rotation Axis

General

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.

The roll–over function prevents coordinates for the rotation axis from overflowing. The roll–over function is enabled by setting bit 0 of parameter 1008 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 set in parameter No. 1260, and rounded by the angle corresponding to one rotation. The tool moves in the direction in which the final coordinates are closest when bit 1 of parameter No. 1008 is set to 0. Displayed values for relative coordinates are also rounded by the angle corresponding to one rotation when bit 2 of parameter No. 1008 is set to 1.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

Note 1 When this parameter is changed, turn off the power before continuing operation.

[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. (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 and #2. (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.

ROSx	ROTx	Meaning
1	0	Setting is invalid (unused)
1	1	Rotation axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values is linear axis type. (Is not rounded in 0 to 360°) Absolute coordinate values and relative coordinate values are rounded or not rounded by parameter No. 1008#0 and #2. (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)

	#7	#6	#5	#4	#3	#2	#1	#0
1008						RRLx	RABx	ROAx

[Data type] Bit axis

ROAx The roll–over function of a rotation axis is
0 : Invalid
1 : Valid

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

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.

Note 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 1 After setting the parameter, turn off the power once and turn it on again to operate the machine.

[Data type] Two–word

[Unit of data]

Increment system	Unit of data	Standard value
IS–A	0.01 deg	36000
IS–B	0.001 deg	360000
IS–C	0.0001 deg	3600000

[Valid data range] 1000 – 9999999

Set the amount of a shift per one rotation of a rotation axis.

Note 2 This parameter is valid only when ROAx = 1.

Note

Note 1 Rotary axis roll–over function cannot be used together with the indexing function of the index table.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.21.1	Rotary Axis Roll–over
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.20.2	Rotary Axis Roll–over

1.2.4 Controlled Axes Detach

General

These signals release the specified control axes from control.
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.

Signal

Controlled axis detach signals DTCH1 – DTCH8 <G124>

[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.

DTCH 1

1 ..... The 1st axis is detached.
2 ..... The 2nd axis is detached.
: :
: :
8 ..... The 8th 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 is effective for the axis, but do not execute the command. The command is accepted but the operation is restrained, because the axis interlock is 0. In an automatic operation, the execution may stop and hold at at block.
5) Position display also displays the position of the detached axis.

Controlled axis detach status signals MDTCH1 – MDTCH8 <F110>

[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.

MDTCH 1
- 1 ..... The 1st axis is detached.
- 2 ..... The 2nd axis is detached.
- : :
- : :
- 8 ..... The 8th axis is detached.

[Output condition] 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
G124	DTCH8	DTCH7	DTCH6	DTCH5	DTCH4	DTCH3	DTCH2	DTCH1
F110	MDTCH8	MDTCH7	MDTCH6	MDTCH5	MDTCH4	MDTCH3	MDTCH2	MDTCH1

Parameter

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

Setting entry is acceptable.

[Data type] Bit axis

RMVx Releasing the assignment of the control axis for each axis
0 : Not released
1 : Released

Note 1 RMVx is valid when RMBx in parameter 1005 is 1.

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

[Data type] Bit axis

RMBx Releasing the assignment of the control axis for each axis (signal input and setting input)
0 : Invalid
1 : Valid

MCCx When an axis is released from control, control for the MCC signal for the corresponding servo amplifier is
0 : Disabled
1 : Enabled

Note 1 If the servo motor for an axis is connected to a 2–axis or other multiaxis amplifier, releasing the axis from control causes servo alarm 401 (V ready off) to be output. This alarm can be disabled by this parameter. When the servo motor is disconnected from the CNC, however, servo alarm 401 is output, regardless of the value of the parameter, due to the nature of multiaxis amplifier.

Note

Note 1 Controlled axis detach signals DTCH1 <G124#0>, DTCH2 <G124#1>, and DTCH3 <G124#2> 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 no. 1005#7 must be set, indicating the axes are detachable.

Note 3 Setting parameter no. 0012#7 from the MDI panel detaches the axes in the same way as these signals.

Note 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 224 to be output (the alarm can be disabled by setting bit 0 (ZRNx) of parameter 1005).

Note 5 When a 2–axis or 3–axis amplifier is used, releasing only one axis from control results in the output of servo alarm 401 (V ready off). Use 1–axis amplifiers for those axes to be released from control, e.g., by replacing the rotary table.

1.2.5 Outputting the Movement State of an Axis

General

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

Signal

Axis moving signals MV1 – MV8 <F102>

[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.

MV 1

1 ..... The 1st axis is moving.
2 ..... The 2nd axis is moving.
3 ..... The 3rd axis is moving.
: :
: :

[Output condition] 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:

. Corresponding axis has stopped moving to be in-positioned.

. When the move command for the corresponding axis has been distributed (when bit 6 (MVX) of parameter 3003 is 0)

. When deceleration for the corresponding axis has been completed and the axis is set to the in–position condition. If in–position check is not performed, when the deceleration for the corresponding axis is completed. (When bit 6 (MVX) of parameter 3003 is 1)

Setting 1 in bit 7 (MVG) of parameter 3003 prevents these signals from being output during drawing in dynamic graphics mode (drawing without movement of the machine).

Axis moving direction signals MVD1 – MVD8 <F106>

[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.

MVD 1

- 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 condition] “1” indicates the corresponding axes are moving in the minus direction, and “0” indicates they are moving in the plus direction.

Note 1 These signals maintain their existing condition during a stop, indicating the direction of the axes’ movement before being stopped.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F102	MV8	MV7	MV6	MV5	MV4	MV3	MV2	MV1
F106	MVD8	MVD7	MVD6	MVD5	MVD4	MVD3	MVD2	MVD1

Parameter

- **Setting the output format of the axis–in–movement signal**

	#7	#6	#5	#4	#3	#2	#1	#0
3003	MVG	MVX						

[Data type] Bit

MVX The axis–in–movement signal is set to 0 when:
0: Distribution for the axis is completed. (The signal is set to 0 in deceleration.)
1: Deceleration of the axis is terminated, and the current position is in the in–position.

MVG While drawing using the dynamic graphics function (with no machine movement), the axis–in–movement signal is:
0: Output
1: Not output

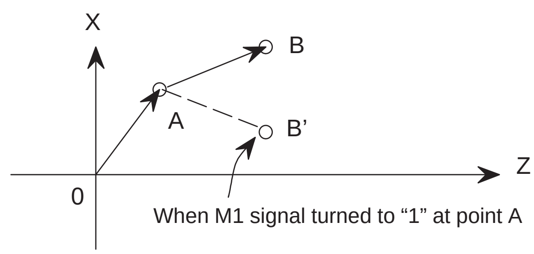
Note

Note 1 These signals are output in both automatic and manual operations.

1.2.6
Mirror Image

General

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 T series)

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 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 signal
MI1 – MI8 <G106>

[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.

MI 1

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:

a) During offset cancel;

b) When the CNC is in the automatic operation stop state and not in the feed hold state.

Mirror image check signal MMI1 – MMI8<F108>

[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.

MMI 1

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 condition] 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
G106	MI8	MI7	MI6	MI5	MI4	MI3	MI2	MI1
F108	MMI8	MMI7	MMI6	MMI5	MMI4	MMI3	MMI2	MMI1

Parameter

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

Setting entry is acceptable.

[Data type] Bit axis

MIRx Mirror image for each axis
0 : Mirror image is off.
1 : Mirror image is on.

Note

Note 1 When programmable mirror image and ordinary mirror image are specified at the same time, programmable mirror image is applied first.

Note 2 No programmable mirror image affects mirror image check signals MMI1 to MMI8 <F108>.

Note 3 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 MMI8 <F108>.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.4.7	Mirror Image
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.4.7	Mirror Image

1.2.7 Follow–up

General

When position control is disabled for the controlled axes (when the servo is off, during emergency stop, or during a servo alarm), if the machine is moved, a positional error occurs. Follow–up is a function for changing the current position of the CNC and reset the error counter to zero, assuming a command corresponding to the error has been specified.
You can select whether to perform follow–up for axes for which the servo is turned off.
Follow–up is always performed during emergency stop or a servo alarm.

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

When 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 *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.

Signal

Follow–up signal *FLWU <G007#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
G007			*FLWU					

Parameter

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

[Data type] Bit axis

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 1 When the index table indexing function (M series) is used, be sure to set FUPx of the 4th axis to 1.

Reference item

CONNECTION MANUAL (This manual)	1.2.8	Servo Off (Mechanical handle)

1.2.8 Servo Off (Mechanical handle)

General

Place the controlled axes in the servo off state; that is, they 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 signal SVF1 – SVF8 <G126>

[Classification] Input signal

[Function] Select whether to place each axis in the servo off state.

[Operation] These signals place those axes for which the signals are 1 in the servo off state (the current to the servo motor is stopped), thus disabling position control. However, the position detection feature continues to function, so the current position is not lost.
These signals are provided for the controlled axes on a one–to–one basis. A number appended to a signal represents a controlled axis number.

* SVF 1

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

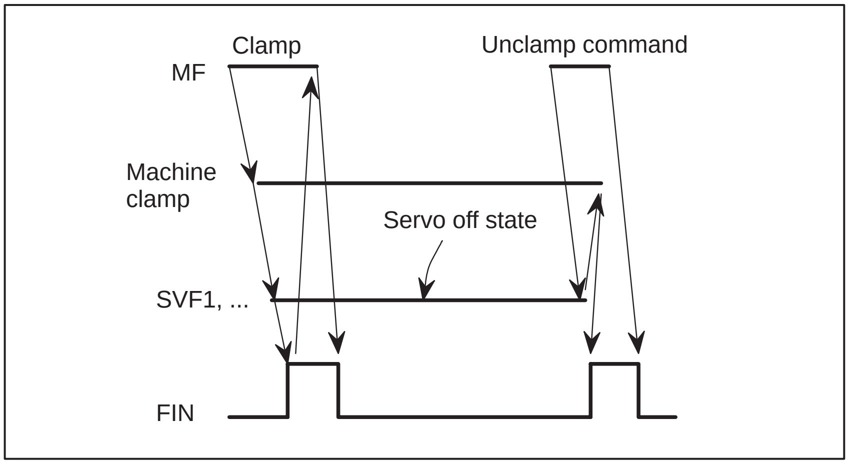
Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G126	SVF8	SVF7	SVF6	SVF5	SVF4	SVF3	SVF2	SVF1

Note

Note 1 In general, interlock is applied to an axis while the servo off signal for that axis is 1.

Note 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”.



Reference item

CONNECTION MANUAL (This manual)	1.2.7	Follow–up

1.2.9 Position Switch

General

Position switch signals can be output to the PMC while the machine coordinates along a controlled axes are within a specified ranges.

Signal

Position switch signal PSW01 – PSW10 <F070#0 – F071#1>

[Classification] Output signal

[Function] Notifies that the machine coordinates along the controlled axes specified by parameters (6910 to 6919) are within the ranges specified by parameters (6930 to 6939 and 6950 to 6959). Up to ten position switch signals can be output.

[Output condition] 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
F070	PSW08	PSW07	PSW06	PSW05	PSW04	PSW03	PSW02	PSW01
F071							PSW10	PSW09

Parameter

- **Setting the correspondence between the position switch signals and the controlled axes**

6910	Axis corresponding to the first position switch
6911	Axis corresponding to the second position switch
6912	Axis corresponding to the third position switch
6913	Axis corresponding to the fourth position switch
6914	Axis corresponding to the fifth position switch
6915	Axis corresponding to the sixth position switch
6916	Axis corresponding to the seventh position switch
6917	Axis corresponding to the eighth position switch
6918	Axis corresponding to the ninth position switch
6919	Axis corresponding to the tenth position switch

[Data type] Byte

[Valid data range] 1, 2, 3, . . . , control axis count

These parameters specify the control–axes numbers corresponding to the first through tenth position switch functions. A corresponding position switch signal is output to PMC when the machine coordinate value of a corresponding axis is within the range that is set using a parameter.

Note 1 Set 0 for those position switch numbers that are not to be used.

- **Setting the machine coordinate ranges for which the position switch signals are output**

- **Maximum operation range**

6930	Maximum operation range of the first position switch
6931	Maximum operation range of the second position switch
6932	Maximum operation range of the third position switch
6933	Maximum operation range of the fourth position switch
6934	Maximum operation range of the fifth position switch
6935	Maximum operation range of the sixth position switch
6936	Maximum operation range of the seventh position switch

6937	Maximum operation range of the eighth position switch
6938	Maximum operation range of the ninth position switch
6939	Maximum operation range of the tenth position switch

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 0 to ±99999999

These parameters set the maximum operation range of the first through tenth position switches.

- **Minimum operation range**

6950	Minimum operation range of the first position switch
6951	Minimum operation range of the second position switch
6952	Minimum operation range of the third position switch
6953	Minimum operation range of the fourth position switch
6954	Minimum operation range of the fifth position switch
6955	Minimum operation range of the sixth position switch
6956	Minimum operation range of the seventh position switch
6957	Minimum operation range of the eighth position switch
6958	Minimum operation range of the ninth position switch
6959	Minimum operation range of the tenth position switch

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 0 to ±99999999

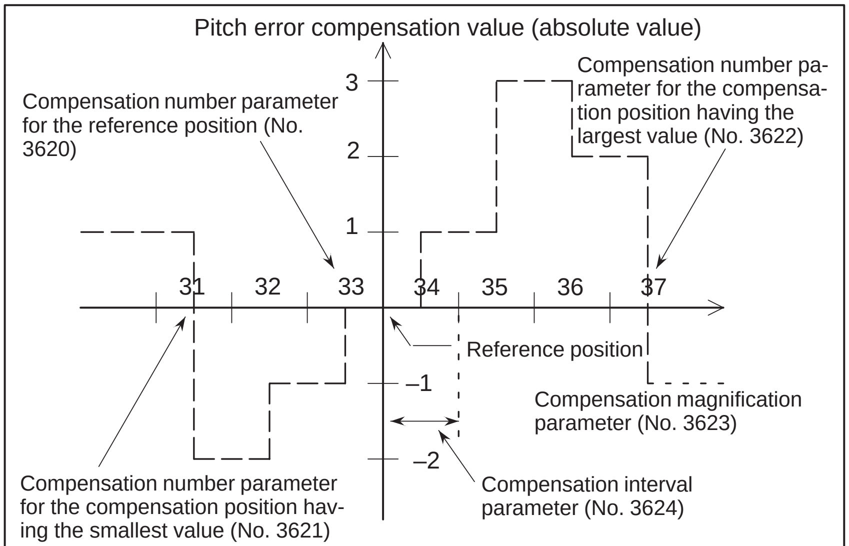
These parameters set the minimum operation range of the first through tenth position switches.

1.3 ERROR COMPENSATION

1.3.1 Stored Pitch Error Compensation

General

If pitch error compensation data is specified, pitch errors of each axis can be compensated in detection unit per axis.
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 written directly with the MDI panel.
The following parameters must be set for pitch error compensation. Set the pitch error compensation value for each pitch error compensation position number set by these parameters.
In the following example, 33 is set for the pitch error compensation number at the reference position.



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

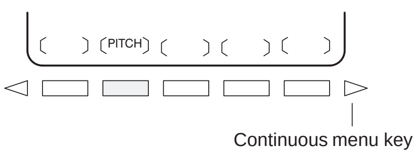
· Number of the pitch error compensation position at the reference position (for each axis): Parameter 3620
· Number of the pitch error compensation position having the smallest value (for each axis): Parameter 3621
· Number of the pitch error compensation position having the largest value (for each axis): Parameter 3622
· Pitch error compensation magnification (for each axis): Parameter 3623

· Interval of the pitch error compensation positions (for each axis): Parameter 3624

Procedure for displaying and setting the pitch error compensation data

1 Set the following parameters:

- Number of the pitch error compensation position at the reference position (for each axis): Parameter 3620
- Number of the pitch error compensation position having the smallest value (for each axis): Parameter 3621
- Number of the pitch error compensation position having the largest value (for each axis): Parameter 3622
- Pitch error compensation magnification (for each axis): Parameter 3623
- Interval of the pitch error compensation positions (for each axis): Parameter 3624



2 Press function key [SYSTEM].

3 Press the continuous menu key [▷], then press chapter selection soft key **[PITCH]**.
The following screen is displayed:

```
PIT–ERROR SETTING                   O0000 N00000

      NO.     DATA    NO.     DATA    NO.     DATA
      0000    0       0010    0       0020    0
      0001    0       0011    0       0021    0
      0002    0       0012    0       0022    0
      0003    0       0013    0       0023    0
   (X)0004    0       0014    0       0024    0
      0005    0       0015    0       0025    0
      0006    0       0016    0       0026    0
      0007    0       0017    0       0027    0
      0008    0       0018    0       0028    0
      0009    0       0019    0       0029    0
 > _
 MEM **** *** ***       16:05:59
 [ NO.SRH ][ ON:1 ][ OFF:0 ][ +INPUT ][ –INPUT ]
```

4 Move the cursor to the compensation point number to be set in either of the following ways:

- Enter the compensation point number and press the [NO.SRH] soft key.
- Move the cursor to the compensation point number using the page keys, [↑ PAGE] and [PAGE ↓], and cursor keys, [↑], [↓], [←], and [→].

5 Enter a value with numeric keys and press the [INPUT] soft key.

Explanations

● Specifying the compensation point

To assign the compensation positions for each axis, specify the positive direction or the negative direction in reference 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.

● Compensation point number

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

Examples

● For linear axis

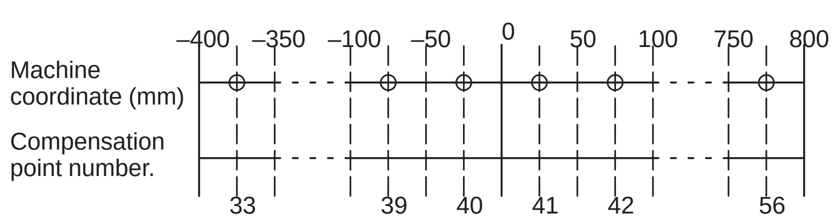
· Machine stroke: –400 mm to +800 mm
· Interval between the pitch error compensation positions: 50 mm
· No. of the compensation position of the reference position: 40
If the above is specified, the No. of the farthest compensation position in the negative direction is as follows:

> No. 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

No. of the farthest compensation position in the positive direction is as follows:

> No. 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:



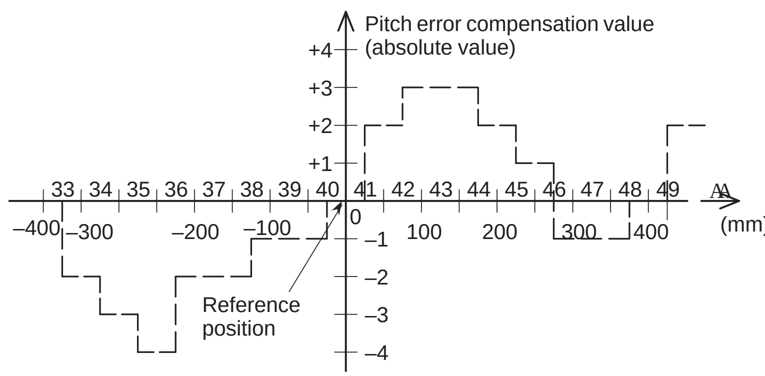
Compensation values are output at the positions indicated by ○.

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.

No	33	34	35	36	37	38	39	40	41	42	43	44	45	46	47	48	49	56
Compensation value	+2	+1	+1	–2	0	–1	0	–1	+2	+1	0	–1	–1	–2	0	+1	+2	1

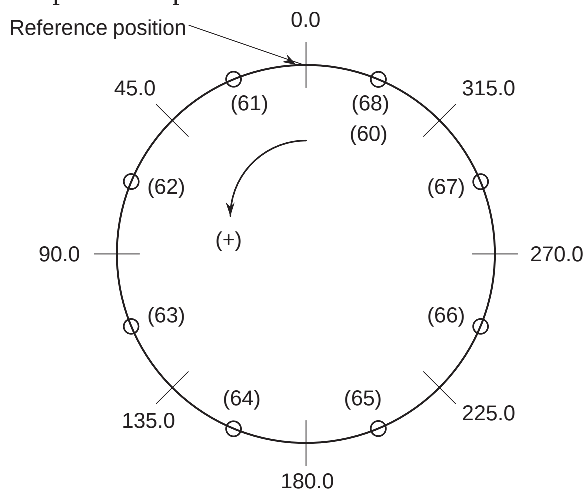


- **For rotary axis**

·Amount of movement per rotation: 360°
· Interval between pitch error compensation positions: 45°
· No. of the compensation position of the reference position: 60
If the above is specified, the No. of the farthest compensation position in the negative direction for the rotating axis is always equal to the compensation position No. of the reference position.
The No. of the farthest compensation position in the positive direction is as follows:

No. 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:



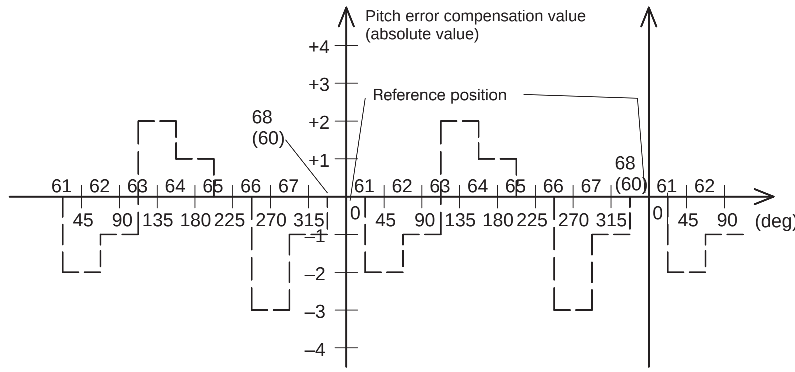
Compensation values are output at the positions indicated by ○ .

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.
Therefore, set the parameters as follows:

Parameter	Setting value
3620 : Compensation number for the reference position	60
3621 : Smallest compensation position number	60
3622 : Largest compensation position number	68
3623 : Compensation magnification	1
3624 : Interval between pitch error compensation positions	45000

The following is an example of compensation amounts.

No	60	61	62	63	64	65	66	67	68
Compensation value	+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

[Data type] Word axis

[Unit of data] Number

[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

[Data type] Word axis

[Unit of data] Number

[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

[Data type] Word axis

[Unit of data] Number

[Valid data range] 0 to 1023

Set the number of the pitch error compensation position at the extremely positive position for each axis.

Note 1 This value must be larger than set value of parameter (No. 3620).

3623	Magnification for pitch error compensation for each axis

[Data type] Byte axis

[Unit of data] 1

[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.

3624	Interval between pitch error compensation positions for each axis

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 0 to 99999999

The pitch error compensation positions are arranged with equally spaced. 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)/3750

Unit: mm, inches, deg

[Example]

When the maximum rapid traverse rate is 15000 mm/min, the minimum interval between pitch error compensation positions is 4 mm.

Note

- **Compensation value range**

Compensation values can be set within the range from –7 x compensation magnification (detection unit) to +7 x compensation magnification (detection unit). The compensation magnification can be set for each axis within the range from 0 to 100 in parameter 3623.

- **Intervals of compensation positions**

The pitch error compensation positions are arranged with equally spaced. Set the space between two adjacent positions for each axis to the parameter (No. 3624).

- **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.

- **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

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III 8.6.3	Inputting pitch error compensation data
	III 8.6.4	Outputting pitch error compensation data
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III 8.6.3	Inputting pitch error compensation data
	III 8.6.4	Outputting pitch error compensation data

1.3.2 Backlash Compensation

General

- **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.

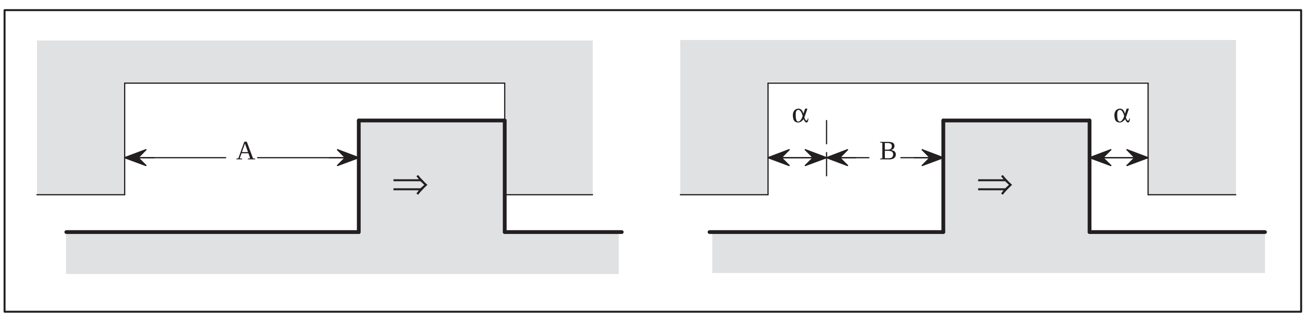
- **Backlash compensation for each rapid traverse and cutting feed**

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	¿α	¿ (−α)
Opposite direction	¿A	¿B	¿ (B+α)	¿ (B+α)

- $\alpha = (A–B)/2$
- 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				

[Data type] Bit

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

1851	Backlash compensating value for each axis

[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

[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.

Note

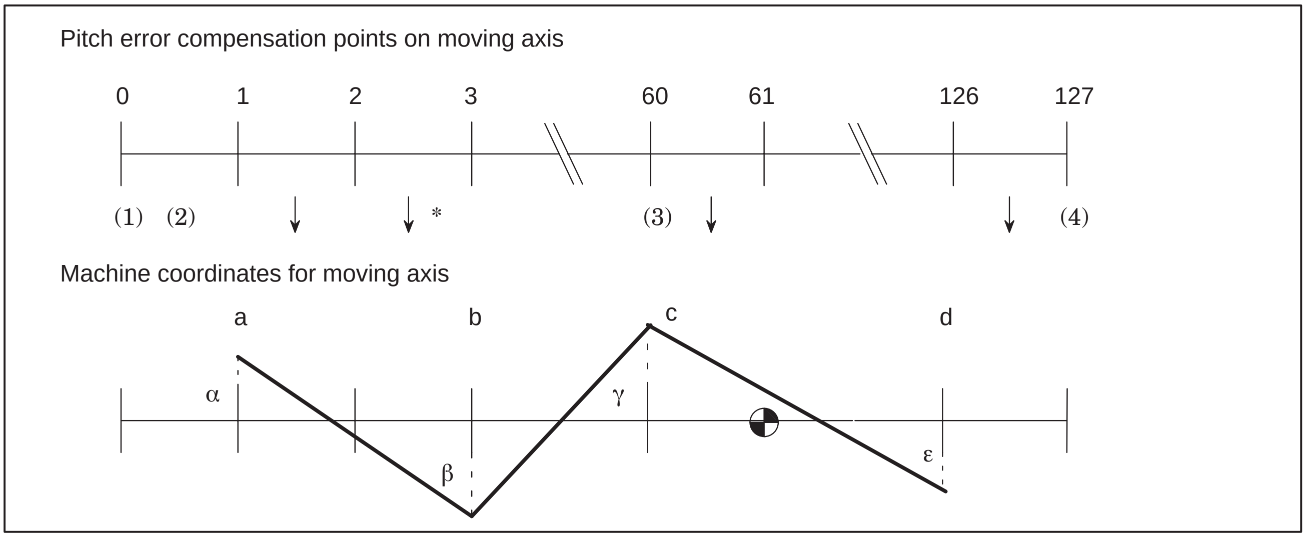
Note 1 When backlash compensation is applied separately for cutting feed and rapid traverse, jog feed is regarded as cutting feed.

Note 2 The backlash compensation for each 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 and a cutting feed.

1.3.3 Straightness Compensation

General

For a machine tool with a long stroke, deviations in straightness between axes may deteriorate 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 Nos. 5711 to 5713) moves, the corresponding compensation axis (parameter Nos. 5721 to 5723) is compensated.
That is, the compensation axis is compensated at the pitch error compensation point (See 1.3.1) of the moving axis.



●

a, b, c, d Compensation point numbers of the moving axis (parameter No. 5501 to 5524)
α, β, γ, ε Compensation for compensation axis (parameter No. 5551 to 5574)

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

Parameter

5711	Axis number of moving axis 1
5712	Axis number of moving axis 2
5713	Axis number of moving axis 3

[Classification] Input

[Data type] Byte

[Unit of data] Axis number (When 0, compensation is not performed.)

[Valid data range] 1 to Number of controlled axes

Set the axis numbers of moving axes.

5721	Axis number of compensation axis 1 for moving axis 1
5722	Axis number of compensation axis 2 for moving axis 2
5723	Axis number of compensation axis 3 for moving axis 3

[Classification] Input

[Data type] Byte

[Unit of data] Axis number (When 0, compensation is not performed.)

[Valid data range] 1 to Number of controlled axes

Set the axis numbers of compensation axes.

5731	Compensation point number a of moving axis 1
5732	Compensation point number b of moving axis 1
5733	Compensation point number c of moving axis 1
5734	Compensation point number d of moving axis 1
5741	Compensation point number a of moving axis 2
5742	Compensation point number b of moving axis 2
5743	Compensation point number c of moving axis 2
5744	Compensation point number d of moving axis 2
5751	Compensation point number a of moving axis 3
5752	Compensation point number b of moving axis 3
5753	Compensation point number c of moving axis 3
5754	Compensation point number d of moving axis 3

[Classification] Input

[Data type] Word

[Unit of data] Number
(Compensation point numbers in stored pitch error compensation)

[Valid data range] 0 to 128 × number of controlled axes

Set four compensation points for each moving axis.

5761	Compensation corresponding to compensation point number a of moving axis 1
5762	Compensation corresponding to compensation point number b of moving axis 1
5763	Compensation corresponding to compensation point number c of moving axis 1
5764	Compensation corresponding to compensation point number d of moving axis 1
5771	Compensation corresponding to compensation point number a of moving axis 2
5772	Compensation corresponding to compensation point number b of moving axis 2
5773	Compensation corresponding to compensation point number c of moving axis 2
5774	Compensation corresponding to compensation point number d of moving axis 2
5781	Compensation corresponding to compensation point number a of moving axis 3
5782	Compensation corresponding to compensation point number b of moving axis 3
5783	Compensation corresponding to compensation point number c of moving axis 3
5784	Compensation corresponding to compensation point number d of moving axis 3

[Classification] Input

[Data type] Word

[Unit of data] Detection unit

[Valid data range] –32768 to +32767

Note 1 Set compensation for each compensation point.

Alarm and message

Number	Message	Description
5046	ILLEGAL PARAMETER (ST. COMP)	Parameters related to straightness compensation have been erroneously specified. Possible causes are as follows: 1. Invalid axis numbers have been assigned to move or compensation axes. 2. The number of pitch–error compensation points between the maximum positive and maximum negative points exceeds 128. 3. Straightness compensation point numbers have been assigned in other than ascending order. 4. Straightness compensation points could not be located between the maximum positive and maximum negative pitch–error compensation points. 5. The amount of compensation per compensation point is too large or too small.

Note

Note 1 The straightness compensation function can be used after a moving axis and its compensation axis have returned to the reference position.

Note 2 After setting parameters for straightness compensation, be sure to turn off the NC power.

Note 3 Set parameters for straightness compensation according to the following conditions:

- The compensation at a compensation point must be within the range –128 to 127.
- Compensation points must be set so that “a ≦ b ≦ c ≦ d” is satisfied.
- Compensation points must exist between the compensation point with the largest positive value and that with the largest negative value in the stored pitch error compensation data for each axis. Four compensation points can be set to 0 at a time. In this case, compensation is not performed.

Note 4 To add the straightness compensation function option, the stored pitch error compensation option is needed.
In this case, the number of compensation points of each axis between the compensation point 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 128.

Note 5 Straightness compensation data is superposed on stored pitch error compensation data and output.

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

General

Explanation of terms frequently used in CNC

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

Notes
The relations among the least command increment, detection unit, CMR, and DMR are as specified below.
Least command increment = CMR × detection unit

$$\text{Detection unit} = \frac{\text{Move amount per revolution of motor}}{\text{DMR} \times \text{number of pulses of detector per revolution}}$$

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	

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	

Note 1 When this parameter has been set, the power must be turned off before operation is continued.

[Data type] Bit axis

OPTx Position detector
0: A separate pulse coder is not used.
1: A separate pulse coder is used.

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

Note 1 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.

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				

Note 1 When this parameter has been set, the power must be turned off before operation is continued.

[Data type] Bit axis

DM1x to DM3x Setting of detection multiply

Set value			Detection multiply
DM3x	DM2x	DM1x	
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 1 When the flexibly feed gear is used, do not use these parameters. Set the numerator and denominator of DMR to an appropriate values in parameters 2084 and 2085 respectively.

1820	Command multiply for each axis (CMR)

Note 1 When this parameter has been set, the power must be turned off before operation is continued.

[Data type] Byte axis

Set a command multiply indicating the ratio of the least command increment to the detection unit for each axis.

Least command increment = detection unit x command multiply

Relationship between the increment system and the least command increment

	Least command increment			
Increment system	**IS–A**	**IS–B**	**IS–C**	**Unit**
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

The value set in the parameter is obtained as follows:

(1) When command multiply is 1/2 to 1/27

$$\text{Set value} = \frac{1}{(\text{Command multiply})} + 100$$

Valid data range: 102 to 127

(2) When command multiply is 1 to 48

Set value = 2 $\times$ command multiply
Valid data range: 2 to 96

Note 1 When command multiply is 1 to 48, the set value must be determined so that an integer can be set for command multiply.

1821	Reference counter size for each axis

[Data type] Two–word axis

[Valid data range] 0 to 99999999

Set the size of the reference counter.

Note 1 When this parameter has been set, the power must be turned off before operation is continued.

1825	Servo loop gain for each axis

[Data type] Word axis

[Unit of data] 0.01 s^{-1}

[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:

$$\text{Positioning deviation} = \frac{\text{feedrate}}{60} \times (\text{loop gain})$$

Unit: Positioning deviation mm, inches, or deg
Feedrate: mm/min, inches/min, or deg/min
loop gain: s^{-1}

1828	Positioning deviation limit for each axis in movement

[Data type] Two–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 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

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

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

1832	Feed stop positioning deviation for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Set the feed stop positioning deviation for each axis.

If the positioning deviation exceeds the feed stop positioning deviation during movement, pulse distribution and acceleration/deceleration control are stopped temporarily. When the positioning deviation drops to the feed stop positioning deviation or below, pulse distribution and acceleration/deceleration control are resumed.

The feed stop function is used to reduce overshoot in acceleration/deceleration mainly by large servo motors.
Generally, set the middle value between the positioning deviation limit during movement and the positioning deviation at rapid traverse as the feed stop positioning deviation.

1850	Grid shift for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] 0 to ±99999999

A grid shift is set for each axis.

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.

Note 1 When this parameter has been set, the power must be turned off before operation is continued.

1.4.2 Absolute Position Detection

General

Even when the power to the NC is turned off, a battery–powered pulse coder stores the current position. No reference position return is required when the power to the NC is turned on next.

Parameter

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

Note 1 When this parameter has been set, the power must be turned off before operation is continued.

APCx Position detector
0: Other than absolute position detector
1: Absolute position detector (absolute pulse coder)

1.5 SETTINGS RELATED WITH COORDINATE SYSTEMS

1.5.1 Machine Coordinate System

General

Machine coordinate system is a coordinate system set with a zero point proper to the machine system.
A coordinate system in which the reference position becomes the parameter-preset (No. 1240) coordinate value when manual reference position return is performed, is set. With G53 command, the machine coordinate system is selected and the axis is moved at rapid traverse to the position expressed by the machine coordinates.

Parameter

1240	Coordinate value of the reference position on each axis in the machine coordinate system

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

Set the coordinate values of the reference positions in the machine coordinate system.

Note

Note 1 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.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.8.1	MACHINE COORDINATE SYSTEM
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.8.1	MACHINE COORDINATE SYSTEM

1.5.2 Workpiece Coordinate System/Addition of Workpiece Coordinate System Pair

General

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).

Setting a workpiece coordinate system

A workpiece coordinate system can be set using one of three methods:

(1) Method using G92 (G50 for G code system A)
A workpiece coordinate system is set by specifying a value after G92 (G50) in the program.

(2) Automatic setting
If bit 0 (ZPR) of parameter No. 1201 is set beforehand, a workpiece coordinate system is automatically set when manual reference position return is performed.

(3) Input using the CRT/MDI panel
Six workpiece coordinate systems can be set beforehand using the CRT/MDI panel.

Selecting a workpiece coordinate system

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

(1) Selecting a workpiece coordinate system set by G92 (G50) or automatic workpiece coordinate system setting
Once a workpiece coordinate system is selected, absolute commands work with the workpiece coordinate system.

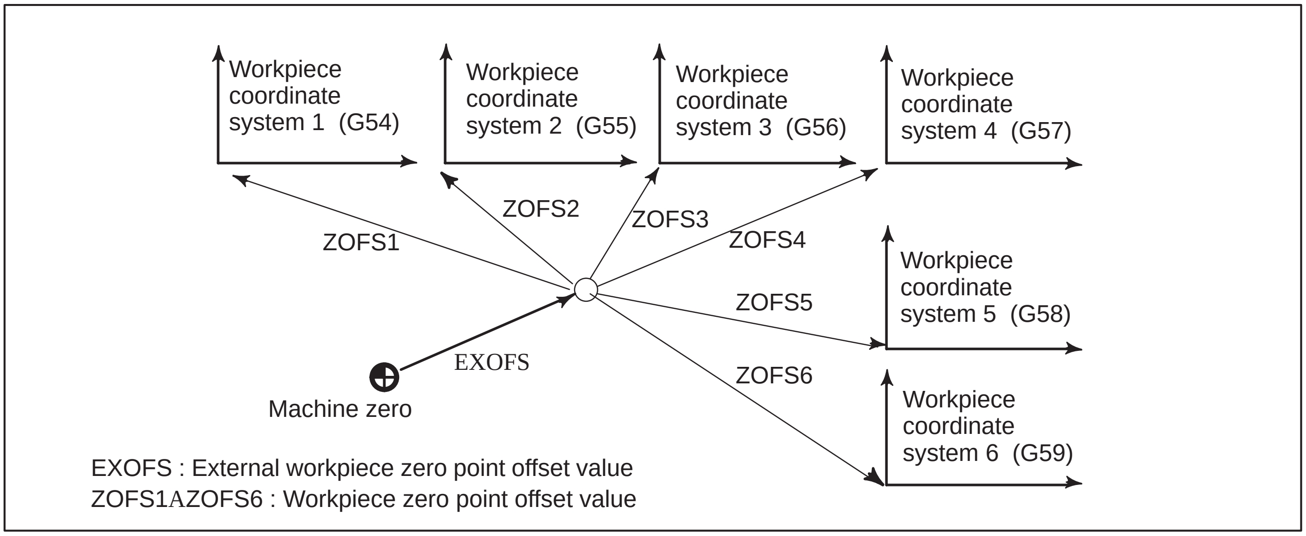
(2) Choosing from six workpiece coordinate systems set using the CRT/MDI panel
By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.

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.

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 CRT/MDI panel
(2) Programming by G10 or G92 (G50)
(3) Changing an external workpiece zero point offset value

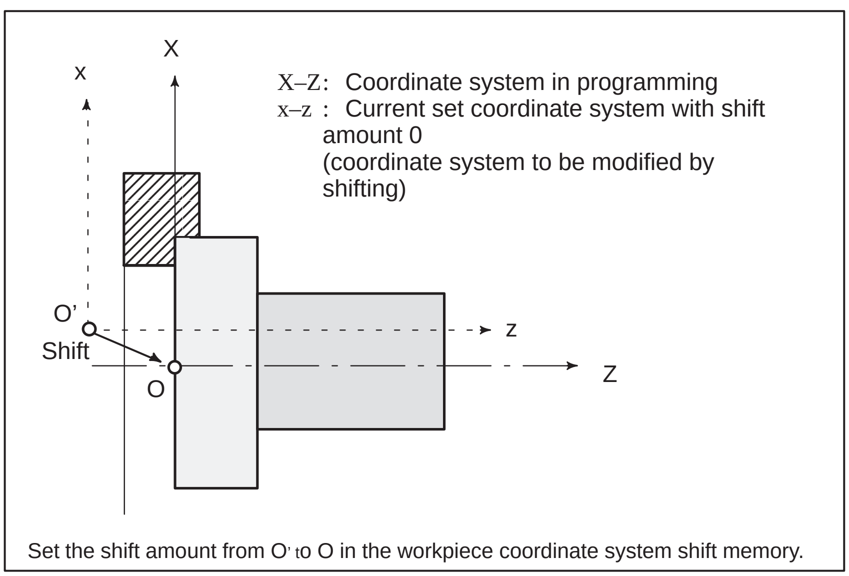


Changing an external workpiece zero point offset value or workpiece zero point offset value

Workpiece coordinate system shift (T series)

When the coordinate system actually set by the G92 (G50) command or the automatic coordinate system setting deviates from the programmed workpiece coordinate, the set coordinate system can be shifted.
Set the desired shift amount in the workpiece coordinate system shift memory.

Explanations



Workpiece Coordinate System shift

Addition of workpiece coordinate system pair (M series)

Besides the six workpiece coordinate systems (standard workpiece coordinate systems) selectable with G54 to G59, 48 or 200 additional workpiece coordinate systems (additional workpiece coordinate systems) can be used.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1201			AWK				ZPI	ZPR

[Data type] Bit

ZPR Automatic setting of a coordinate system when the manual reference position return is performed
0 : Not set automatically
1 : Set automatically

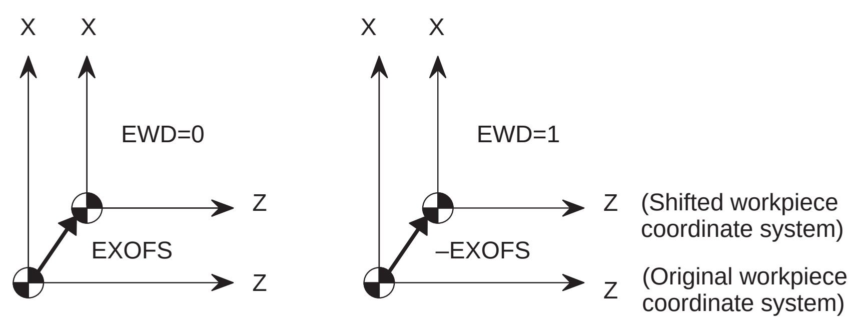
ZPI Coordinates at the reference position when a coordinate system is set automatically
0 : Value set in parameter No. 1250 is used.
1 : For input in mm, the value set in parameter 1250 is used, or for input in inches, the value set in parameter No. 1251 is used.

AWK Action taken after the workpiece zero point offset value is changed
0 : The absolute coordinate value is changed when the first automatic operation is performed.
1 : The absolute coordinate value is changed immediately.

	#7	#6	#5	#4	#3	#2	#1	#0
1202						G50	EWS	EWD

[Data type] Bit

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



EXOFS : External workpiece zero point offset value

EWS Shift value of the workpiece coordinate system and external workpiece zero point offset value are
0 : Stored in the separate memory areas.
1 : Stored in the same memory area, that is, the shift and the offset values are the same.

G50 When the CNC has commands G54 to G59 specifying workpiece coordinate systems (optional function), if the G50 command for setting a coordinate system (or the G92 command in G command system B or C) is specified,

0 : The G50 (or G92) command is executed without an alarm.

1 : P/S alarm No. 010 is issued and the G50 (or G92) command is not executed.

1220	External workpiece zero point offset value

[Data type] Two–word axis

[Unit of data]

Input increment	IS–A	IS–B	IS–C	Unit
Linear axis (input in mm)	0.01	0.001	0.0001	mm
Linear axis (input in inches)	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –7999 to 7999

This is one of the parameters that give the position 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. Usually, this parameter is set automatically when it is input from the machine (external data input).

1221	Workpiece zero point offset value in workpiece coordinate system1 (G54)
1222	Workpiece zero point offset value in workpiece coordinate system2 (G55)
1223	Workpiece zero point offset value in workpiece coordinate system3 (G56)
1224	Workpiece zero point offset value in workpiece coordinate system4 (G57)
1225	Workpiece zero point offset value in workpiece coordinate system5 (G58)
1226	Workpiece zero point offset value in workpiece coordinate system6 (G59)

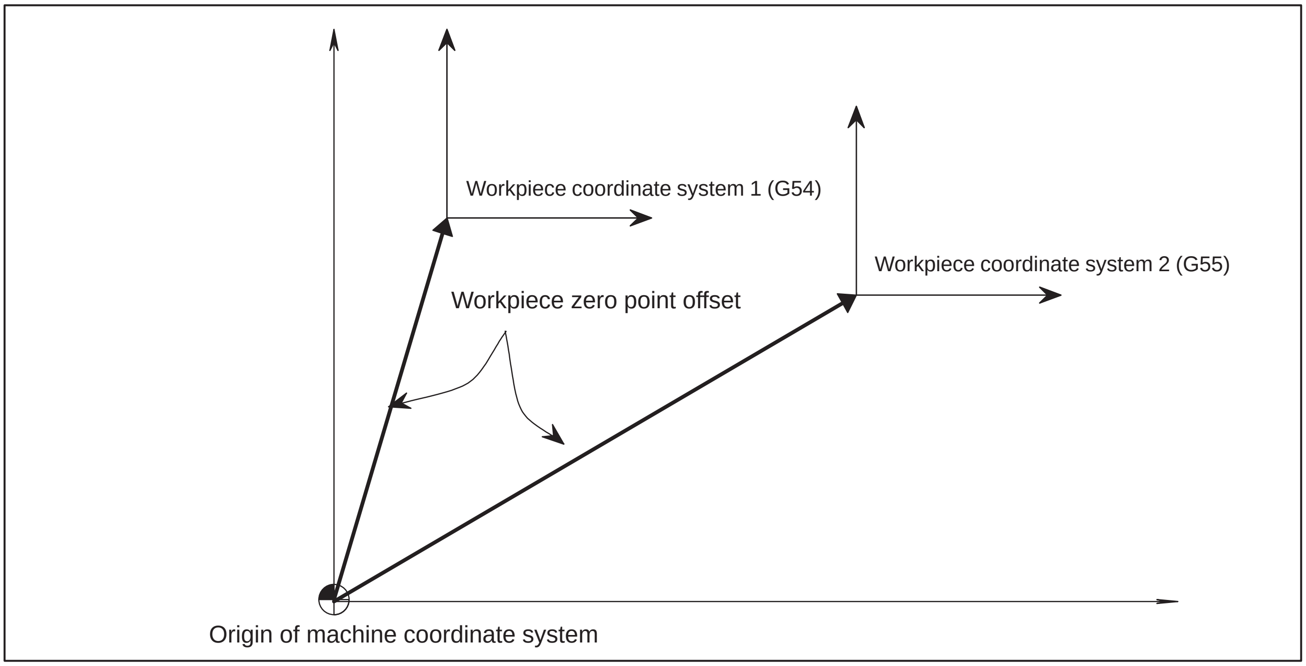
[Data type] Two–word axis

[Unit of data]

Input increment	IS–A	IS–B	IS–C	Unit
Linear axis (input in mm)	0.01	0.001	0.0001	mm
Linear axis (input in inches)	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

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



1250	Coordinate value of the reference position used when automatic coordinate system setting is performed

[Data type] Two–word axis

[Unit of data]

Input increment	IS–A	IS–B	IS–C	Unit
Linear axis (input in mm)	0.01	0.001	0.0001	mm
Linear axis (input in inches)	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically.

1251	Coordinate value of the reference position used when automatic coordinate system setting is performed with inch input

[Data type] Two–word axis

Input increment	IS–A	IS–B	IS–C	Unit
Linear axis (input in inches)	0.001	0.0001	0.00001	inch

[Valid data range] –99999999 to 99999999

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically when input is performed in inches.

Note 1 This parameter is valid when ZPI in parameter 1201 is set to 1.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.8.2	WORK COORDINATE SYSTEM
OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.8.2	WORK COORDINATE SYSTEM

1.5.3 Rotary Axis Roll Over

General

The roll–over function prevents coordinates for the rotation axis from overflowing. The roll–over function is enabled by setting bit 0 (ROAx) of parameter 1008 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 bit 1 (RABx) of parameter No. 1008 is set to 0. Displayed values for relative coordinates are also rounded by the angle corresponding to one rotation when bit 2 (RRLx) of parameter No. 1008 is set to 1.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

Note 1 When this parameter is changed, turn off the power before continuing operation.

[Data type] Bit axis

ROTx, ROSx Setting linear or rotation axis

ROSx	ROTx	Description
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (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 and relative coordinate values are rounded or not rounded by parameter No. 1008 #0 and #2. (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	0	Setting is invalid (unused)
1	1	Rotation axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values is linear axis type. (Is not rounded in 0 to 360°) Absolute coordinate values and relative coordinate values are rounded or not rounded by parameter No. 1008 #0 and #2. (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)

	#7	#6	#5	#4	#3	#2	#1	#0
1008						RRLx	RABx	ROAx

[Data type] Bit axis

ROAx The roll–over function of a rotation axis is
0 : Invalid
1 : Valid

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

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.

Note 2 Assign the amount of the shift per one rotation in parameter No. 1260.

1260	Move amount per rotation of rotary axis

Note 1 When this parameter is changed, turn off the power before continuing operation.

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Unit of data	0.01	0.001	0.0001	deg
Standard setting value	36000	360000	3600000	

Set move amount per rotation of rotation axis.

Note

Note 1 This function cannot be used together with the indexing function of the index table (M series).

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.21.2	ROTARY AXIS ROLL–OVER
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.20.2	ROTARY AXIS ROLL–OVER

1.6 SIMPLE SYNCHRONOUS CONTROL

General

A movement along an axis can be executed simply by executing a move command specified for that axis or by synchronizing the movement with another axis. Either of these two types can be selected by means of a signal sent from the machine.

• Simple synchronous control for the M series and T series

The M series and T series support different simple synchronization control functions. One of the greatest differences is that:

<T series> The function can synchronize only automatic operations. It cannot synchronize manual operations.

<M series> The function can synchronize both automatic and manual operations.

The following functions are provided only for the M series:

Synchronization error check function
Synchronization compensation function

• Synchronization error check function

The function monitors the difference between the machine coordinates on the master and slave axes. If the function detects a difference greater than or equal to a preset value, it stops the machine. This function constantly monitors the difference. Even if the synchronization control signal is erroneously set to 0, thus disabling synchronization control, the function can issue an alarm, stop the machine, and thus prevent damage.

If the detected difference is greater than or equal to the maximum error set in parameter 8314, servo alarm 407 is output.

Notes
1. Before using the synchronization error check function, set identical values for the reference positions of the master and slave axes.
2. If the synchronization error check function is not used, set parameter 8314 to 0.
3. To clear the alarm, first increase the maximum synchronization error set in parameter 8314, then press the reset key. Next, perform handle operations or other manual operations so that the machine coordinates agree. Then, restore the original value in parameter 8314.
4. If an alarm is detected during a synchronous operation, set the signals indicating that a synchronous operation is in progress (G138, G140) to off, then follow the procedure for clearing an alarm.

- **Synchronization compensation function**

If the agreement between the positions of the master and slave axes is lost when the system power is turned off, the function compensates for the difference between them. After performing a follow–up at power on, the function sends compensation pulses to the slave axis to adjust its position such that it agrees with that of the master axis. This function is enabled only when the slave axis of synchronization control supports the absolute–position detection function.

Notes
1. To use the synchronization compensation function, set the SOF bit, bit 1 of parameter 8301, to 1.
2. The synchronization compensation function is enabled after reference position returns have been performed. The function is not executed if the parameter is set before reference position returns are performed.
3. The synchronization deviation is processed as a position error on the slave axis while at rest. The position error is displayed as diagnostic data 300, in the same units as used to detect the error. If the error exceeds the value set in parameter 8315, servo alarm 410 is triggered. The alarm can be cleared by pressing the reset key. As the position error for the slave axis remains even after the alarm is cleared, however, the positions must be adjusted.
4. The synchronization compensation function is also enabled when emergency stop is canceled.
5. The synchronization compensation function is not executed when the servo alarm is eliminated.

The simple synchronization control functions are described separately for the T series and M series in the following explanations.

Signal

<T series and M series>

Signals to select the slave axis for simple synchronous control SYNC1 to SYNC8 <G138>

[Classification] Input signal

[Function] Synchronization control is performed for memory or MDI operation. The signal is provided for each controlled axis. The number at the end of the signal name represents the number of the controlled axis.

SYNC 1

1\. ... The first axis becomes the slave axis for synchronization control.
2\. ... The second axis becomes the slave axis for synchronization control.
3\. ... The third axis becomes the slave axis for synchronization control.
:
:

[Operation] When the signal is set to 1, the control unit operates as described below:

– During memory or MDI operation, the control unit supplies the move command, specified for the master axis, to both the master and slave axes of synchronization control.

The master axis is specified with a parameter.

<M series>

Signals for selecting the manual feed axis for simple synchronous control SYNCJ1 to SYNCJ8 <G140>

[Classification] Input signal

[Function] Synchronization control is performed in jog, handle, or incremental feed mode.
The signal is provided for each controlled axis. The number at the end of the signal name represents the number of the controlled axis.

SYNCJ 1

1\. ... The first axis becomes the slave axis for synchronization control.
2\. ... The second axis becomes the slave axis for synchronization control.
3\. ... The third axis becomes the slave axis for synchronization control.
:
:

[Operation] When the signal is set to 1, the control unit operates as described below:

– In jog, handle, or incremental feed mode, the control unit supplies the move command, specified for the master axis, to both the master and slave axes of synchronization control.

The master axis is specified with a parameter.

Signal address

T series

	#7	#6	#5	#4	#3	#2	#1	#0
G138	SYNC8	SYNC7	SYNC6	SYNC5	SYNC4	SYNC3	SYNC2	SYNC1

M series

	#7	#6	#5	#4	#3	#2	#1	#0
G138	SYNC8	SYNC7	SYNC6	SYNC5	SYNC4	SYNC3	SYNC2	SYNC1
G140	SYNCJ8	SYNCJ7	SYNCJ6	SYNCJ5	SYNCJ4	SYNCJ3	SYNCJ2	SYNCJ1

Parameter

T series

8311	Axis number of master axis in synchronous control

[Data type] Byte axis

[Valid data range] 0 – 7

Select a master axis and slave axis for simple synchronous control. Set a master axis number for the axis used as a slave axis. If the value of this parameter is 0, the first axis is the master axis. In this case, when the synchronous control select signal G138 is set to 1, operation starts with the 1st axis being the master axis.

Units digit in the parameter for the first axis
→ Set the master axis number for the first axis.

Tens digit in the parameter for the first axis
→ Set the master axis number for the second axis.

Units digit in the parameter for the second axis
→ Set the master axis number for the third axis.

Tens digit in the parameter for the second axis
→ Set the master axis number for the fourth axis.

Units digit in the parameter for the third axis
→ Set the master axis number for the fifth axis.

Tens digit in the parameter for the third axis
→ Set the master axis number for the sixth axis.

Units digit in the parameter for the fourth axis
→ Set the master axis number for the seventh axis.

Tens digit in the parameter for the fourth axis
→ Set the master axis number for the eighth axis.

Number	Tens digit	Units digit
First Second Third Fourth	Second axis Fourth axis Sixth axis Eighth axis	First axis Third axis Fifth axis Seventh axis

The axis number settings are: 0 for the first axis, 1 for the second axis, 2 for the third axis, and so on.

Example) To set the 3rd axis to teh master axis and the 4th axis to the slave axis, set as follows:

No. 8311
1st axis 00
2nd axis 20
3rd axis 00
4th axis 00

8312	Enabling/disabling mirror image in synchronous control

[Data type] Byte axis

[Valid data range] –127 – +128

This parameter sets the mirror image function. When 100 or a greater value is set with this parameter, the mirror image function is applied to synchronous control. Set this parameter to the slave axis.

Example: To establish reversed synchronization when using the third axis as the master axis and the fourth axis as the slave axis, set parameter No. 8311 and parameter No. 8312 as follows:

Parameter No. 8312 (first axis) = 0
Parameter No. 8312 (second axis) = 0
Parameter No. 8312 (third axis) = 0
Parameter No. 8312 (fourth axis) = 100

M series

	#7	#6	#5	#4	#3	#2	#1	#0
8301	SOR							

[Data type] Bit

SOF The synchronization funciton is:
0 : Not used.
1 : Used.

8311	Axis number of master axis in synchronous control

[Data type] Byte axis

[Valid data range] 0 – 7

Select a master axis and slave axis for simple synchronous control. Set a master axis number for the slave axis side.

Example: When using the first axis (X–axis) as the master axis, and the third axis (Z–axis) as the slave 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

Note 1 Specifying the third axis (Z–axis) as the master axis, and the first axis (X–axis) as the slave axis is not allowed. The master axis number must always be smaller than the slave axis number.

8313	Limit of the difference between the amount of positioning deviation of the master and slave axes

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

8314	Allowable error in synchronization error check

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 0 to 32767

This parameter sets, in the detection unit, the allowable error when a synchronization error check is made. The mechanical coordinates of the master axis and slave axis are monitored. When a synchronization error equal to or greater than the value set in this parameter is detected, servo alarm No. 407 is issued, and the machine is stopped. Set this parameter to the master axis. When 0 is set with this parameter, no synchronization error check is performed.

8315	Maximum compensation value for synchronization

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 0 to 32767

This parameter sets the maximum compensation value for synchronization. When a compensation value greater than the value set in this parameter is used, servo alarm No. 410 is issued.

Alarm and message

T series

Number	Message	Description
213	ILLEGAL COMMAND IN SYNCHRO–MODE	A move command was specified for the slave axis of synchronization control.
214	ILLEGAL COMMAND IN SYNCHRO–MODE	A command for coordinate system setting or shift–type tool compensation was executed during synchronization control. Correct the program.

M series

Number	Message	Description
213	ILLEGAL COMMAND IN SYNCHRO–MODE	One of the following errors occurred during synchronous operation (simple synchronization control): (1) The program contains a move command for the slave axis. (2) A command for jog feed, manual handle feed, or incremental feed was issued for the slave axis. (3) After power on, the command for automatic reference position return was specified before a manual reference position return had been performed. (4) The difference in position error between the master and slave axes exceeded the value set in parameter 8313.

Servo alarm

Number	Message	Description
407	SERVO ALARM: EXCESS ERROR	The difference in position error between the master and slave axes exceeded the value set with the parameter.

Diagnostic data

Number	Message	Description
540	SYNCHRO ERROR	The data represents the difference in position error between the master and slave axes during synchronization control.

Note

Note 1 When a manual reference position return is executed, identical movements are performed along the master and slave axes until deceleration commences. Subsequently, grids are detected separately.

Note 2 Pitch error compensation and backlash compensation are executed separately for the master and slave axes.

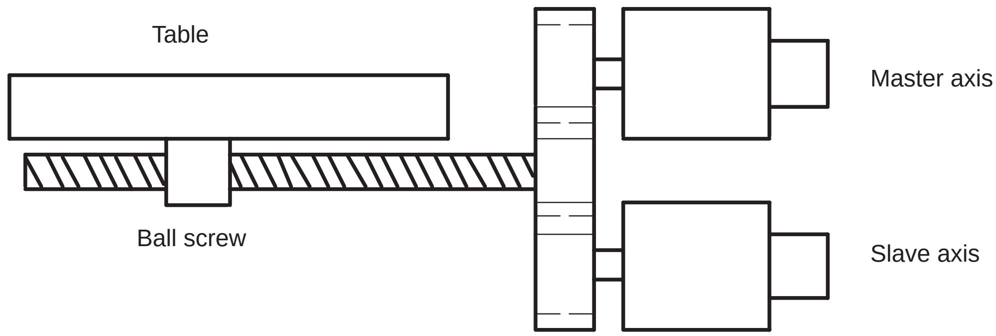
Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.21.1	Simple synchronous control
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.20.3	Simple synchronous control

1.7 TANDEM CONTROL (M SERIES)

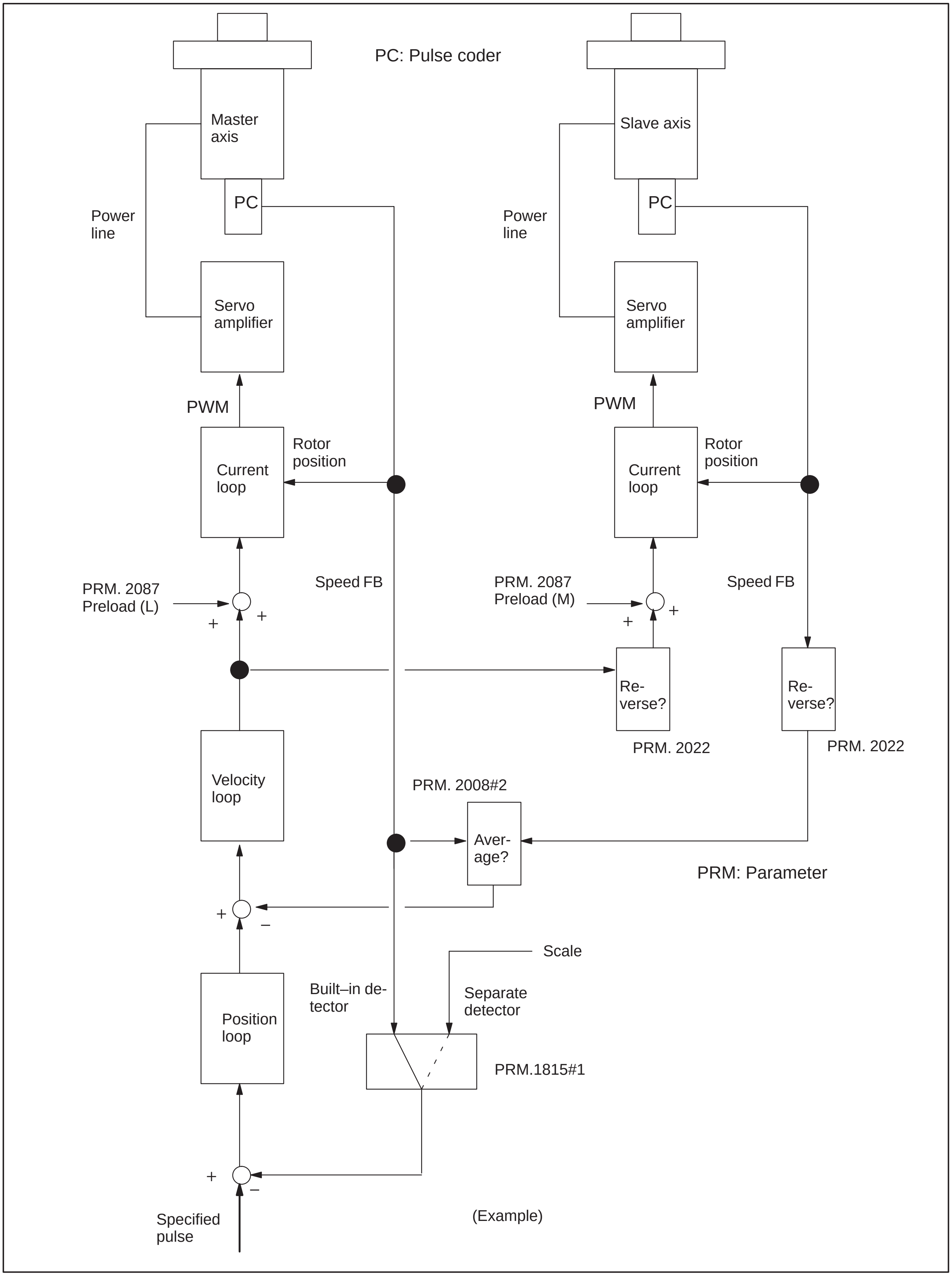
General

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.



Block Diagram of Tandem Control

Explanations

- **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.

(2) In terms of controlled axes, the pair of axes is handled as two separate axes. In terms of CNC–controlled axes (command axes), the pair of axes is handled as a single axis (master axis). Specify the number of CNC–controlled axes with parameter 1010, excluding the slave axis of tandem control. The slave axis must be handled as if it were controlled only by the PMC.

(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) If tandem control is performed for two or more pairs of axes, assign servo axis numbers to the master and slave axes in identical order.

(6) Specify a unique axis name for the slave axis.

(7) The slave axis is handled as a controlled axis. Set the NDPx bit (bit 0 of parameter 3115) to 1 to suppress the position display.

The following sample axis configuration is for a machine with six axes X, Y, Z, A, B (PMC axis), and C. The X–axis and Y–axis are the master axes of tandem control.

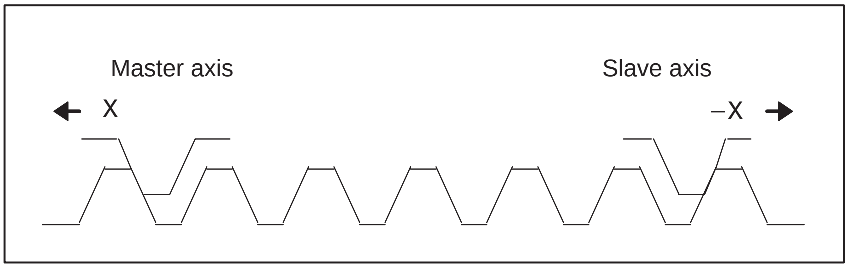
Number of controlled axes (optional parameter): 6

Number of CNC controlled axes (parameter 1010): 3

Axis number	Axis name	Servo axis number (PRM. 1023)	
1	X	3	CNC axis (master axis of tandem control)
2	Y	1	CNC axis (master axis of tandem control)
3	Z	6	CNC axis
4	A	4	Slave axis of tandem control (master axis: X–axis)
5	B	5	PMC axis
6	C	2	Slave axis of tandem control (master axis: Y–axis)

- **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:



Notes
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 very 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 FANUC AC SERVO MOTOR α series PARAMETER MANUAL (B–65150E) or the ”FANUC AC SERVO AMPLIFIER Maintenance Manual (B–65005E).”

- **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.

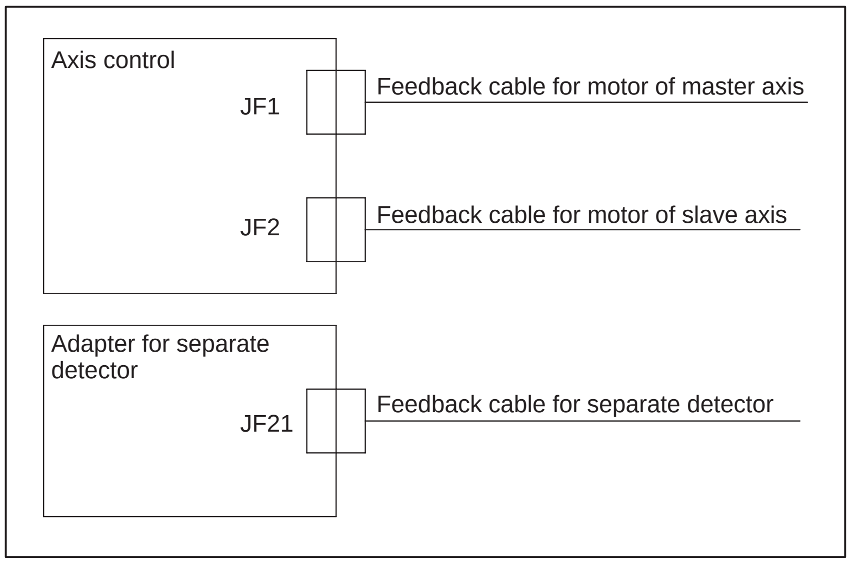
ii) Overtravel limit signal
Connect the signal so that 1 is always output as the overtravel limit signal for the slave axis.

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.

> Note
> 1 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.

  Parameter 2022: Direction of rotation of the motor

  Parameter 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.

  Bit 1 of parameter 1815: Separate pulse coder

  Bits 6 to 4 of parameter 1816: Detection multiplier (DMR)

  Parameter 2024: Number of position detection feedback pulses (PPLS)

  Parameter 1821: Capacity of an optional reference counter

  Parameter 2084: Numerator of flexible feed gear ratio

  Parameter 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 to only 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 feedrate
2008#2	Velocity feedback average function

Parameters that should be set different values between 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
1815#1	Separate type pulse coder
1816#6 to #4	Detection multiply (DMR)
1821	Arbitrary reference counter capacity
2024	Position detection feedback pulses (PPLS)
2084	Numerator of flexible feed gear ratio
2085	Denominator of flexible feed gear ratio

Parameters that should be set the same values to the master and slave axes

Parameter No.	Meaning of parameters
1006#7	Least command increment (0.001mm)
1005#0	Movement before reference position return
1005#1	Dogless reference position setting
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
1310#0	Soft OT2
1310#1	Soft OT3
1320	1st stroke limit of plus side
1321	1st stroke limit of minus side
1322	2nd stroke limit of plus side
1323	2nd stroke limit of minus side
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 feed exponential acceleration/deceleration
1623	FL of feed exponential acceleration/deceleration
1624	Time constant of manual continuous exponential acceleration/deceleration
1625	FL of manual continuous exponential acceleration/deceleration
1626	Time constant of exponential acceleration/deceleration during thread cutting cycle
1627	FL of exponential acceleration/deceleration during thread cutting cycle
1820	Command multiply (CMR)
18XX	Digital servo parameters
20XX	Digital servo parameters

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

[Data type] Bit axis (set to each axis)
Set both master and slave axes.

TAN Tandem control is
1 : ignored
0 : valid

	#7	#6	#5	#4	#3	#2	#1	#0
2008						VFBAVE		

[Data type] Bit axis (set to each axis)
Set only to the master axes.

VFBAVE Velocity feedback average function
1 : invalid
0 : valid

2087	Preload of each axis (Tcmd offset)

[Data type] Word axis

[Unit of data] (Preamplifier limit) /7282

[Valid data range] –1821 to 1821

An offset is added to a torque command to reduce backlash.
Set a slightly large value than friction torque.
As a reference set a value one–third the rated torque.

(Example) To set a torque of 3A in the opposite direction under amplifier limit of 40A:
3 / (40 / 7282) = 546
Master side = 546
Slave side = –546

2021	Load inertia

[Data type] Word axis

Set the same value to the master and slave axes.

[Unit of data] (All load inertia) / (Motor inertia) × 256 / 2

2022	Direction of rotation of motor

[Data type] Word axis

Set the direction of motor rotation.
If the rotation directions of master and slave axes are opposite, set them by this parameter.

Alarm and message

Number	Message	Description
417	SERVO ALARM: n–TH AXIS– PARAMETER INCORRECT	Illegal values are set for parameter 1010, 1023, or 1817 when tandem control is performed.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.21.4	Tandem control

1.8 SYNCHRONOUS CONTROL

General

This function enables synchronous control, in which an axis can be synchronized with another axis.

An axis can be moved in synchronization with another axis. This is done by issuing a move command for one axis (synchronous master axis) to another axis so that both axes behave in the same way. When this function is used in conjunction with the parking function, which keeps an axis at a rest by ignoring a move command for it, the following operations can be performed.

(1) Two axes move in synchronization. (Both master and slave axes move.)

(2) One axis moves according to a move command originally issued to another axis, but the latter is kept at a stop. (The master parks, but the salve moves.)

(3) The coordinate values for one axis are updated according to the amount of movement specified for another axis, but the former is kept at a rest. (The master moves, but the slave parks.)

Notes

1 The term synchronous control used here only refers to an operation in which issuing the same move command to two different servo systems at one time. Note that synchronous control does not involve out–of–synchronization compensation, in which the deviation between two servo motors is constantly checked and one of the servo motors is subjected to compensation to reduce the deviation.

2 The term parking means to issue no move command to a servo system. No coordinate values are updated during parking. However appropriate parameter setting can change absolute or relative coordinate systems.

3 If synchronous control is terminated during automatic operation, a move command or coordinate system setting may not be specified for the synchronous slave axis in the current block and one block that follows it (or two blocks for tool–tip radius compensation).

Automatic setting of workpiece coordinate system

- **Explanation**

When synchronous control is started for 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: For example, when synchronous control is used to move an axis differently from the way originally specified, 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 used to represent 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. This function does not work for the slave axis.

- **Setting and command**

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 No. 8163 (SPMx) to “1”. (Master axis parameter only)
Set parameter No. 8165 with the coordinates of the master axis when both master and slave axes are at the reference position.

(2) To resume the ordinary workpiece coordinate system automatically when synchronous control is terminated
Set parameter No. 8163 (SPSx) to “1”. (Master axis parameter only)
Set parameter No. 1250 with the workpiece coordinates of the master axis when the master axis is at the reference position.

This synchronous control can be specified using the G0138 signal similarly to the ordinary synchronous control. When the signal is raised to start synchronous control, the workpiece coordinate system for the master axis is automatically set up. Likewise, when the signal is dropped to terminate synchronous control, the original workpiece coordinate system for the master axis is set automatically.

- **Workpiece coordinate value 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 = 0
− :Master axis parameter SCDx = 1
(2) Master axis parameter SCMx = 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)

- **Others**

• 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.
• The same least command and input increments must be applied to both master and slave axes.
• The tool offset is taken into consideration when the coordinate system is set up. So, the coordinate system is set up normally even when tool geometry compensation is applied.
• If synchronous control that involves automatic coordinate system setting is started or terminated during automatic operation, a move command or other commands that reference the current position of an axis may not be specified in the current block and one block that follows it (or two blocks for tool–tip radius compensation), except for an M code in which the current block is not buffered; for the M code, a move command can be executed in the block next to the block that involves synchronous control.

Signal

Synchronous control axis selection signals SYNC1 to SYNC8 <G138>

[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 PK8 <G122>

[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.

Synchronous control under way signals SYN1O to SYN8O <F118>

[Classification] Output signal

[Function] These signals indicate each axis is being subjected to synchronous control.

[Operation] These signals become "1" under the following condition:

- The corresponding axis is under synchronous control.

These signals become "0" under the following condition:

- The corresponding axis is not under synchronous control.

Note 1 Whether each axis is under synchronous control does not always match whether the corresponding selection signal (synchronous 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 synchronous control, it is terminated automatically. Before attempting to perform synchronous control, always check the state of these signals.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0138					SYNC4	SYNC3	SYNC2	SYNC1

	#7	#6	#5	#4	#3	#2	#1	#0
G0122					PK4	PK3	PK2	PK1

	#7	#6	#5	#4	#3	#2	#1	#0
F0118					SYN4O	SYN3O	SYN2O	SYN1O

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8160	NRS							

[Data type] Bit

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						PKUx		SMRx

[Data type] Bit axis

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.)

PKUx In the parking sate,
0 : The absolute, relative, and machine coordinates are not updated.
1 : The absolute and relative coordinates are updated. The machine coordinates are not updated.

	#7	#6	#5	#4	#3	#2	#1	#0
8163				SCDx	SCMx	SPSx	SPMx	

Note 1 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.

[Data type] Bit axis

SPMx When synchronous control is started, automatic workpiece coordinate system setting for the master axis is
0 : Not performed.
1 : Performed.

Note 1 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.

SPSx When synchronous control terminates, automatic workpiece coordinate system setting for the master axis is:
0 : Not performed.
1 : Performed.

Note 1 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.

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.

SCDx The positive (+) directions of the master axis and slave axis in the coordinate system in synchronous control are:
0 : Identical.
1 : Opposite.

8180	Master axis with which an axis is synchronized under synchronous control

[Data type] Byte axis

[Valid data range] 201, 202, 203, ... to 200 plus the maximum number of control axes

This parameter specifies the number of the master axis with which an 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 axes, one master axis has two or more slave axes.

In the parameter of a slave axis, specify 200 plus the number of the master axis with which the slave axis is to be synchronized.
Setting: 201 to 208
The value specified here must not exceed 200 plus the maximum number of control axes.

(Example 1) Synchronizing the Y–axis with the Z–axis
Parameter No. 8180x 0
Parameter No. 8180z 0
Parameter No. 8180c 0
Parameter No. 8180y 202

8185	Workpiece coordinates on each axis at the reference position

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

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 SPMx of bit 1 parameter No. 8163 is set to 1. Set this parameter for the master axis.

Alarm and message

Number	Message	Description
225	Axis recomposition error	This alarm occurs under either of the following conditions. (It is detected when synchronous control is specified.) (1) There is an error in axis number parameter setting. (2) There is an error in a control command.
226	A move command was issued to a synchronous axis.	A move command was issued to an axis to be synchronized with another axis.

Note

Cautions for synchronous control

Note 1 When synchronous control is started or terminated, the target axes must be at a stop.

Note 2 More than one axis can be placed in synchronous state at the same time. However, an axis cannot be synchronized with more than one axis simultaneously.

Note 3 All axes subjected to synchronous control must have the same least command increment, detection unit, and diameter/radius specification. Otherwise, the amount of movement will differ from one axis to another.

Note 4 During synchronous control, do not change the parameters related to synchronous control.

Note 5 Before starting synchronous control after an emergency stop, servo–off, or servo alarm is released, be sure to make a return to the reference position and set up the necessary coordinate system.

Note 6 Before starting synchronous control, make sure that for the target axes, a reference position return after power–on has been made and a reference position has been set up according to the absolute pulse coder.

Note 7 Acceleration/deceleration control, pitch error compensation, backlash compensation, and stored stroke limit check are carried out regardless of synchronous control.

Note 8 Synchronous control and simplified synchronous control cannot be specified simultaneously.

Note 9 A move command should not be issued to a synchronous slave axis during synchronous control.

Note 10 The same acceleration/deceleration time constants and servo parameters should be used for the axes subjected to synchronous control as much as possible.

Note 11 The workpiece coordinate system of a synchronous slave is not affected by the synchronous master axis operations that affect the workpiece coordinate system but do not cause the machines to move, such as workpiece coordinate system set/shift and geometry offset commands.

Note 12 If a wear offset command or tool–tip radius compensation is performed for a synchronous master axis, the travel path of the slave axis is shifted by the offset, but the offset is not set (no offset vector is created).

Restrictions imposed during synchronous control

Function	During synchronous control
Acceleration/deceleration control	The same type of acceleration/deceleration control is performed for the synchronous axes, but different time constants are used.
Post–interpolation linear acceleration/deceleration	Not carried out for any axes during synchronous control.
Feedrate clamping	The axes are clamped at the feedrate of the master axis.
Reference position return	A reference position return is possible for the master axis unless it is in a parking state. In a parking state, only automatic reference position return (G28) is possible for the master axis (*1).
Reference position return check	Possible (*2)
PMC axis control	Possible for other than synchronous slave axes.
Polar coordinate interpolation and cylindrical interpolation	Possible
Handle interrupt	Performed regardless of synchronous control
Axial mirror image	Each signal originally belonging to a particular axis is effective for that axis.
Machine lock	Each signal originally belonging to a particular axis is effective for that axis.
Interlock	The signals for the master axis are effective for the slave axes.
Override	The signals for the master axis are effective for the slave axes.
External deceleration	The signals for the master axis are effective for the slave axes.
Skip function	Ineffective for slave axes.
Automatic tool compensation	Ineffective for slave axes.
Tool setter	Ineffective for slave axes.
Follow–up	Impossible during synchronous control.
Program restart	Impossible for a program involving synchronous control.
Cs axis	Synchronous control is impossible.

Notes

1 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 their reference position (the slave axis only moves in synchronization with the reference position return of the master axis) except for an 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 out for the slave 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. A return to the second (third or fourth) reference position by the G30 command works in the same way as G28.

2 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 axis 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 to see whether the lowest–numbered slave axis is at its reference position after completion of positioning.

Reading the coordinate values during synchronous control

The following list summarizes how positional information such as custom macro system variables and current coordinates from the PMC window are read during synchronous control.

Positional information	During synchronous control
Absolute coordinate	Readable
Machine coordinate	Readable
End of each block	Readable only for the master axis
Skip signal position	Readable only for the master axis

Terminating synchronous control

Synchronous control is terminated not only when the corresponding synchronization signal becomes off but also when one of the following conditions occurs.

(1) Emergency stop
(2) Reset
(3) Servo alarm
(4) Servo off
(5) Overtravel
(6) Alarm related to synchronous control
(7) P/S000 alarm

The above conditions terminate synchronous control for all axes.

Reference item

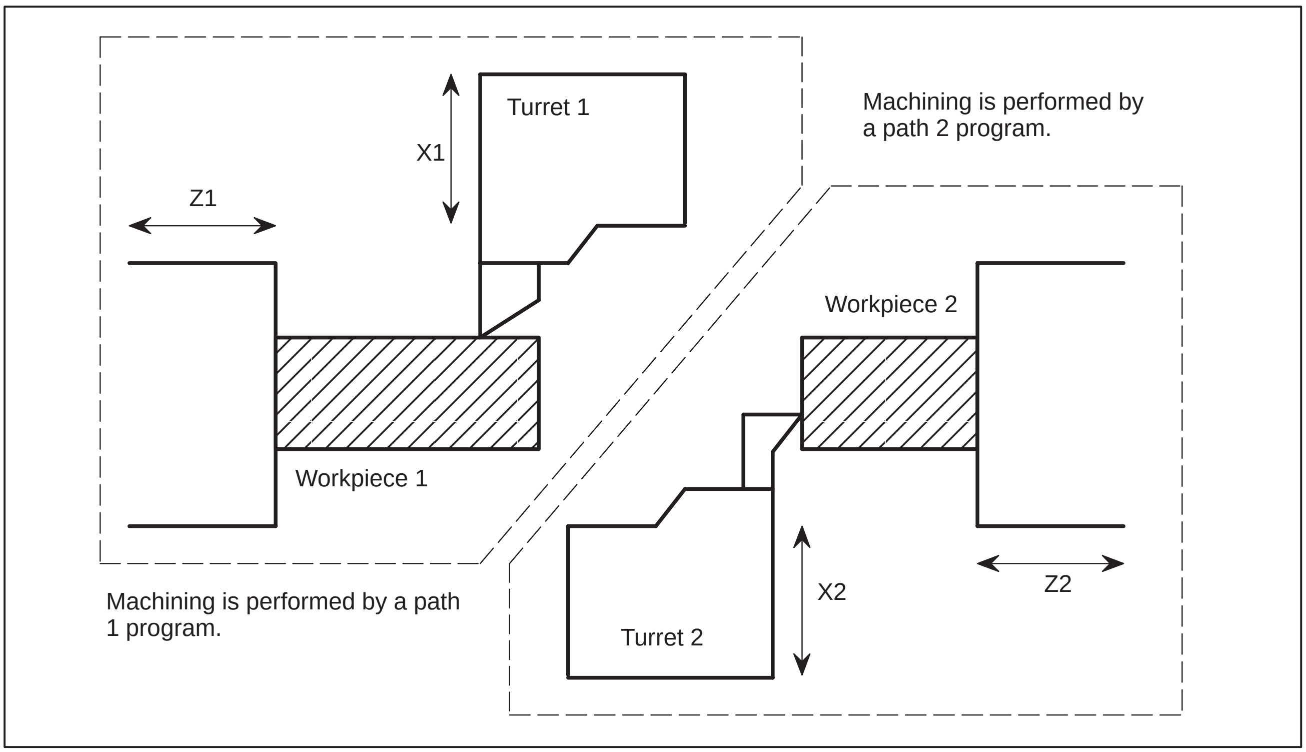
OPERATOR'S MANUAL(For Lathe) (B–62444E)	II.20.4	Synchronous control

1.9 SYNCHRONOUS CONTROL AND COMPOSITE CONTROL (T SERIES (TWO–PATH CONTROL))

1.9.1 Overview

The T series CNC has two independent control paths. For example, it can be used to control two turrets of a multiple–turret lathe independently. 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**

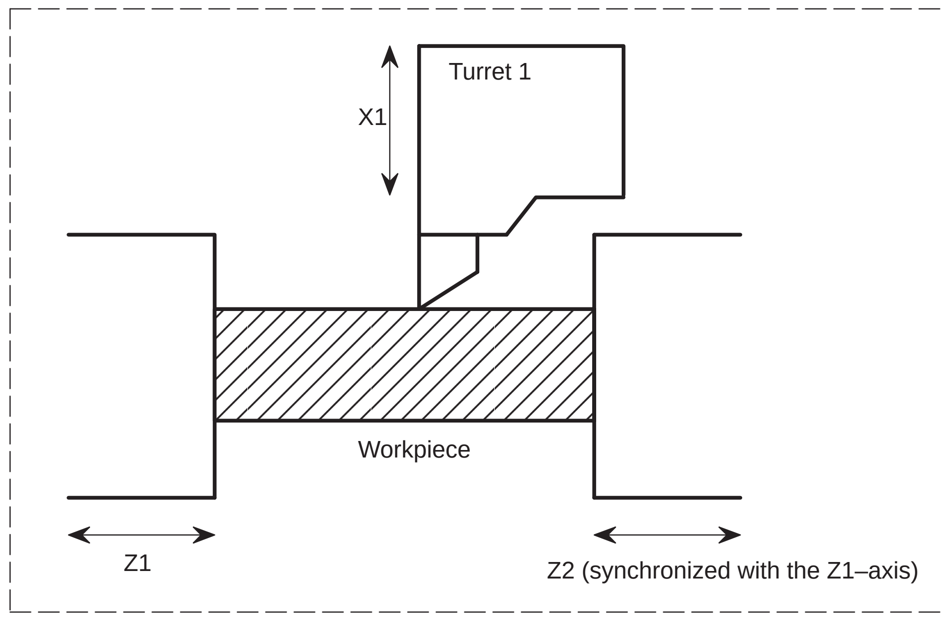


This function enables synchronous control between paths or within a path, composite control between paths, and superimposed control between paths, as explained below.

(1) Synchronous control

- Synchronization of an axis in one path with an axis in the other path

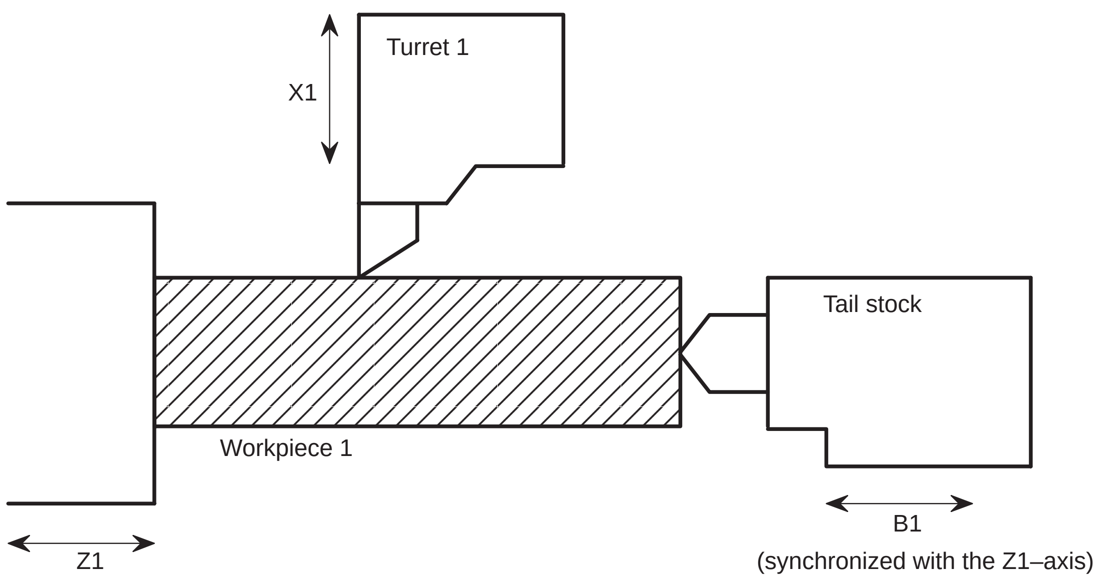
(Example) Synchronization of the Z1–axis with the Z2–axis



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 with the B1–axis

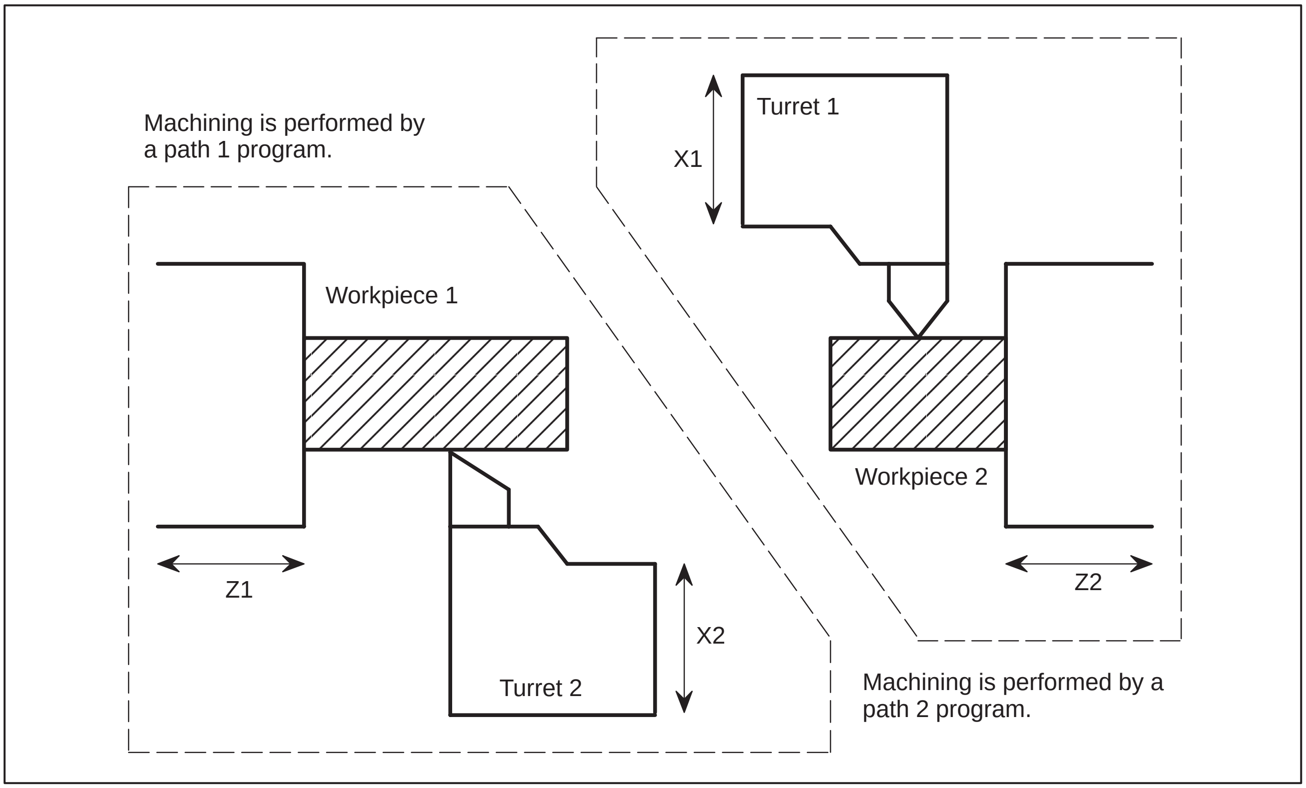


(2) Composite control

- 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

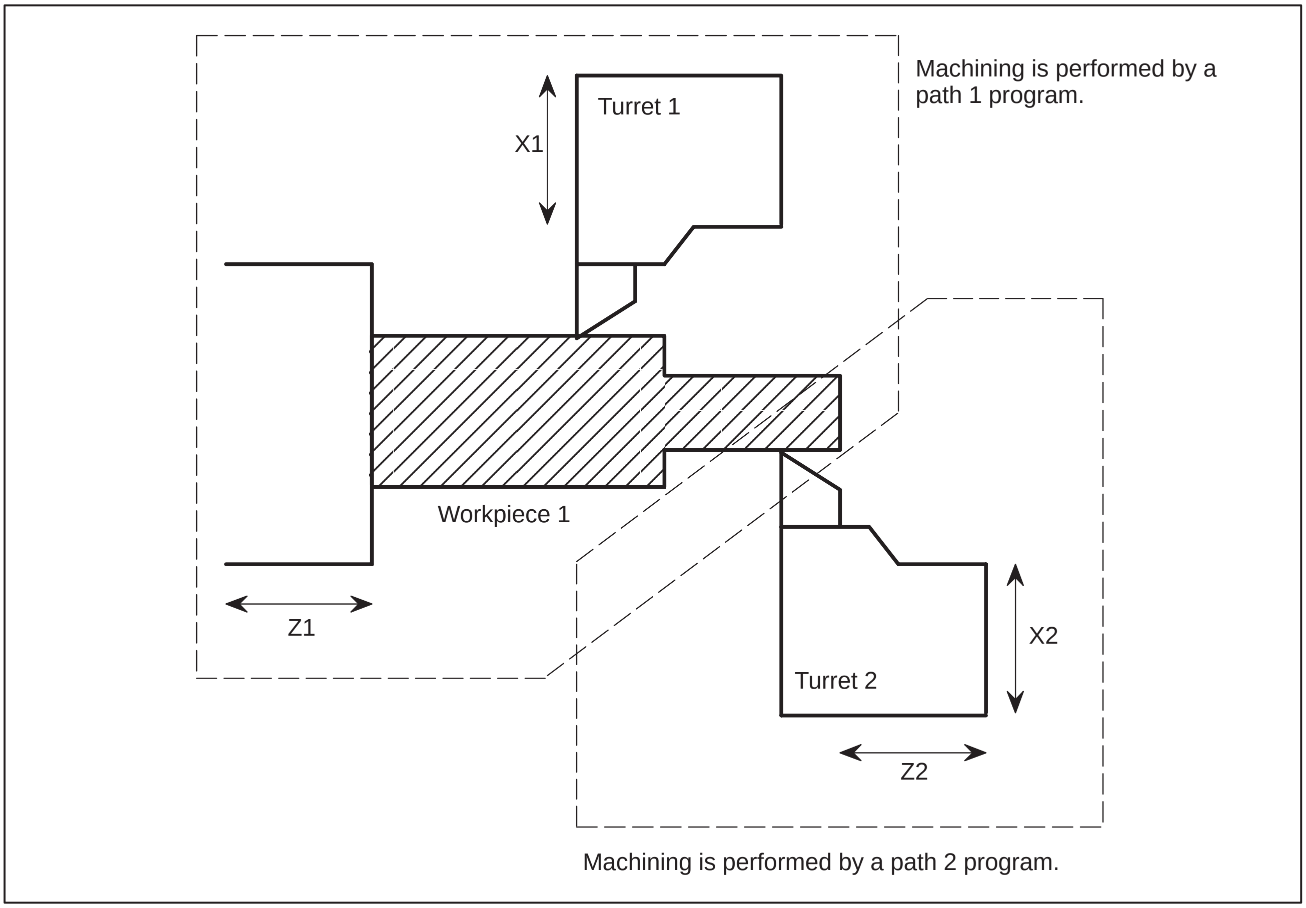
→ Control both X2– and Z1–axes by commands in a path 1 program
Control both X1– and Z2–axes by commands in a path 2 program



(3) Superimposed control

- Superimposing move commands for an axis in one path on an axis in the other path

(Example) Superimposing the movement of the Z1–axis on the Z2–axis



1.9.2 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 (bit 0 of parameter No. 8162) 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 (bit 2 of parameter No. 8162).

1.9.2.1 Setting

Parameter No. 8180 specifies which axis is to be synchronized with which axis.

(Example)
To synchronize the Z1–axis with the Z2–axis:
Parameter No. 8180z of path 1 = 2
To synchronize the Y2–axis with the X1–axis:
Parameter No. 8180y of path 2 = 1
To synchronize the Y1–axis with the X1–axis:
Parameter No. 8180y of path 1 = 201

1.9.2.2 Programming

Use M codes for wait, beginning, and terminating synchronization in a machining program in the stated order. It is also possible to begin and terminate synchronous control without using M codes.

1.9.2.3 Signal operation

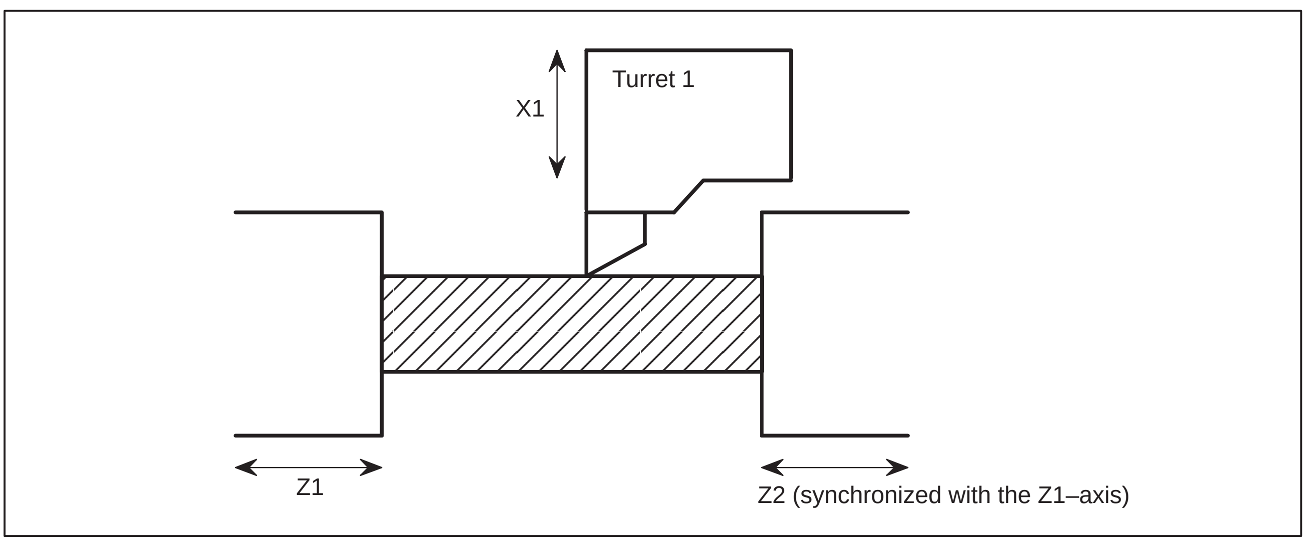
When synchronization begins or ends (when an M code is issued, for example), the synchronous control axis selection signals SYNC1 to SYNC7 for the slave axis (from the PMC to the CNC) are changed from “0” to “1” (to begin synchronization) or from “1” to “0” (to terminate synchronization). To place an axis in a parking state, a parking signal PK1 to PK7 is set to “1” for the target axis.

1.9.2.4 Examples of applications

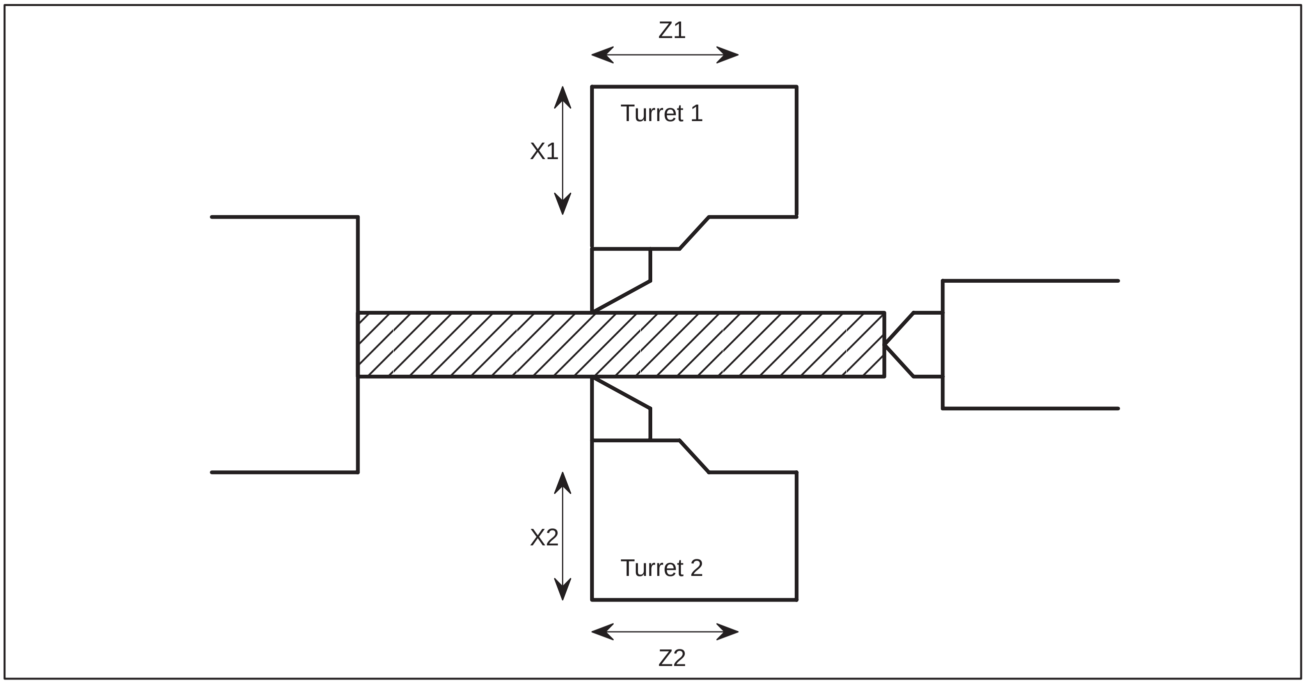
The following operations can be performed by using the synchronization functions together with the parking function, which causes move commands for an axis to be ignored and keeps the axis at a rest.

(1) Moving an axis in one path in synchronization with an axis in the other path (Both master and slave axes move.)

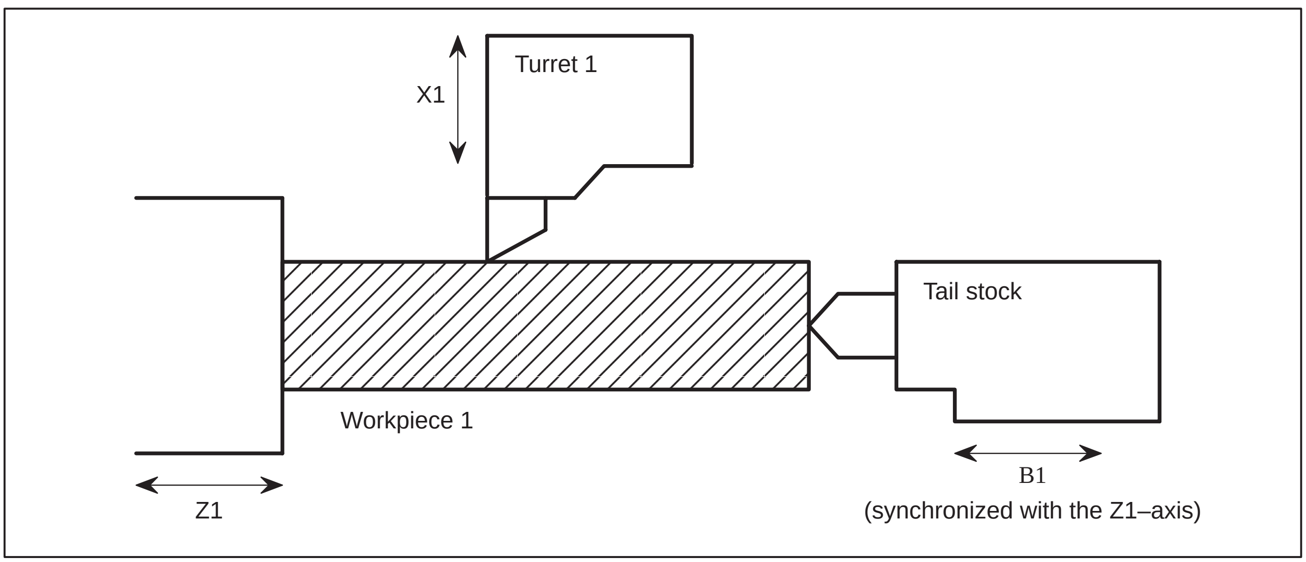
(Example 1) Synchronizing the Z2–axis with the Z1–axis (machining with both ends of a workpiece chucked)



(Example 2) Synchronizing the X2– and Z2–axes with the X1– and Z1–axes (balanced cutting)



(Example 3) Synchronizing the B1–axis (tail stock axis) with the Z1–axis



(2) Moving an axis in one path using move commands for an axis in the other path (The master axis parks and the slave axis moves.)

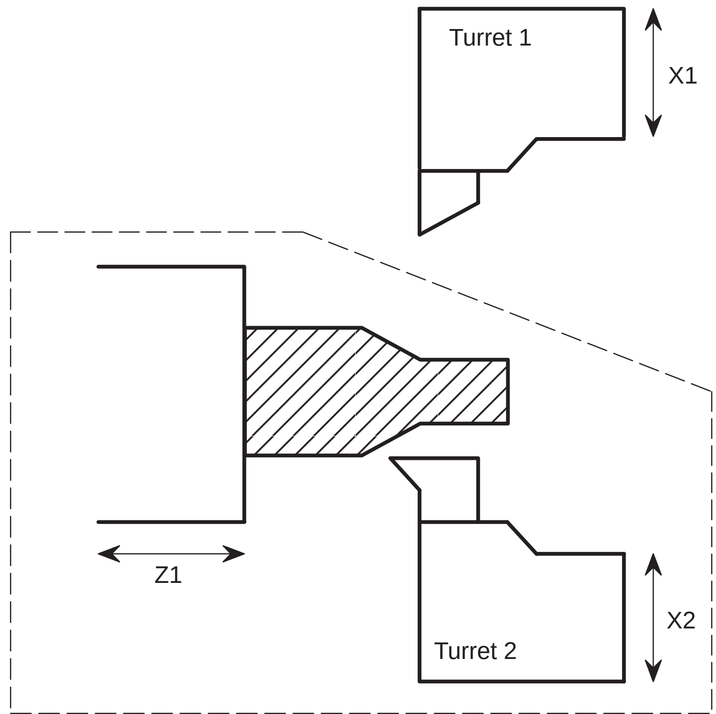
(3) Updating the coordinates of an axis in one path by the amount of movement for an axis in the other path (The master axis moves and the slave axis parks.)

Using method (2) or (3) can control one motor from both paths.

(Example 4) Sharing one motor with the Z1– and Z2–axes (assuming that the motor is linked to the Z1–axis)

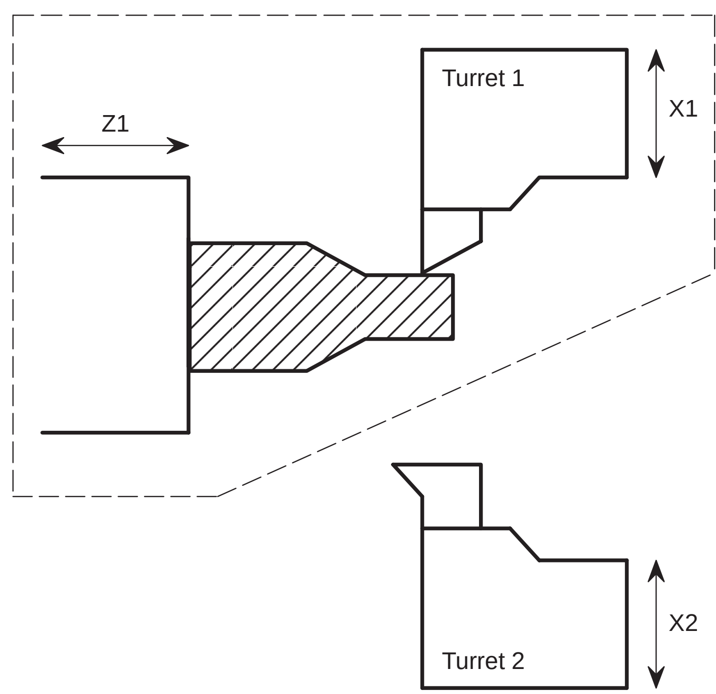
- **Master axis parking**

The X2– and Z1–axes are moved by commands in a path 2 program (by synchronizing the Z1–axis with the Z2–axis). At this point, the Z2–axis is kept in a parking state. The coordinates of both Z1– and Z2–axes are updated.



- **Slave axis parking**

The X1– and Z1–axes are moved by commands in a path 1 program (by synchronizing the Z2–axis with the Z1–axis). At this point, the Z2–axis is kept in a parking state. For the Z2–axis, only the coordinates are updated.



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.

1.9.2.5 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) except for an 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 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.

1.9.2.6 Out–of–synchronization detection

The term synchronous control used here only refers to an operation in which the same move command is issued to two different servo systems at one time. Note that synchronous control does not involve out–of–synchronization compensation, in which the positional deviation between two servo motors is constantly checked and one of the servo motors is subjected to compensation to reduce the deviation. However, using parameter SERx (bit 1 of parameter No. 8162) can specify detection of out–of–synchronization.
If out–of–synchronization is detected, synchronous control is immediately terminated, turning off the servo ready signal.

Note
1 Out–of–synchronization detection is not applied to synchronous control within one path.

1.9.2.7 Move command after switching between independent control and synchronous control

If synchronous control is terminated during automatic operation, do not issue a move command or re–set a coordinate system for the synchronous slave axis in the current block and one or two (during tool–tip radius compensation) subsequent blocks. This restriction is intended to reflect the coordinates changed during synchronous control in the preprocessing for the subsequent blocks.

(Example) Terminating synchronous control of the Z–axis (slave axis) in block N200

N190 ..... ;
N200 M55 ; (This M code terminates synchronous control.)
N210 ..... ;
N220 ..... ;
N230 ..... ;

In this example, not only block N200 but also block N210 (and N220 during tool–tip radius compensation) cannot issue a move command to the Z–axis. However, if the M55 code does not involve buffering, it can be issued in block N210 to move the Z–axis or update its coordinates. For other than the Z–axis, block N210 can issue move commands. These restrictions do not apply to the synchronous master axis.

1.9.2.8 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. This function does not work for the slave axis.

- **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 (bit 1 of parameter No. 8163) 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 (parameter No. 8163) to “1”.
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 signal (SYNC1 to SYNC7) similarly to the ordinary synchronous control. When the signal is raised to start synchronous control, a workpiece coordinate system for the master axis is automatically set up. When the signal is 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 (bit 4 of parameter No. 8163) = 0
–: Master axis parameter SCDx (bit 3 of parameter No. 8163) = 1

(2) Master axis parameter SCMx (bit 3 of parameter No. 8163) = 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)

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.

Note 2 The same least command and input increments must apply to both master and slave axes.

Note 3 The tool offset is taken into account when the coordinate system is set up. So, the coordinate system is set up normally even when tool geometry compensation is applied.

Note 4 If synchronous control that involves automatic coordinate system setting is started or terminated during automatic operation, a move command or other commands that reference the current position of an axis may not be specified in the current block and one or two (during tool–tip radius compensation) blocks, except when the M code in the current block does not involve buffering.

1.9.3 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.

1.9.3.1 Setting

Parameter No. 8183 of path 2 specifies the axes between which commands are to be interchanged.

(Example) Between the X1– and X2–axes:
Parameter No. 8183x of path 2 = 1
Between the Y1– and X2–axes:
Parameter No. 8183x of path 2 = 4
Between The X1– and Y2–axes:
Parameter No. 8183y of path 2 = 1

To set up coordinate systems automatically when composite control begins or ends, set parameters MPMx and MPSx (bits 4 and 5 of parameter No. 8162) to ”1”, and specify the positional relationship between the coordinate systems in parameter No. 8184.

1.9.3.2 Programming

Use M codes for wait, beginning, and terminating composite control in a machining program in the stated order. It is also possible to begin and terminate composite control without using M codes.

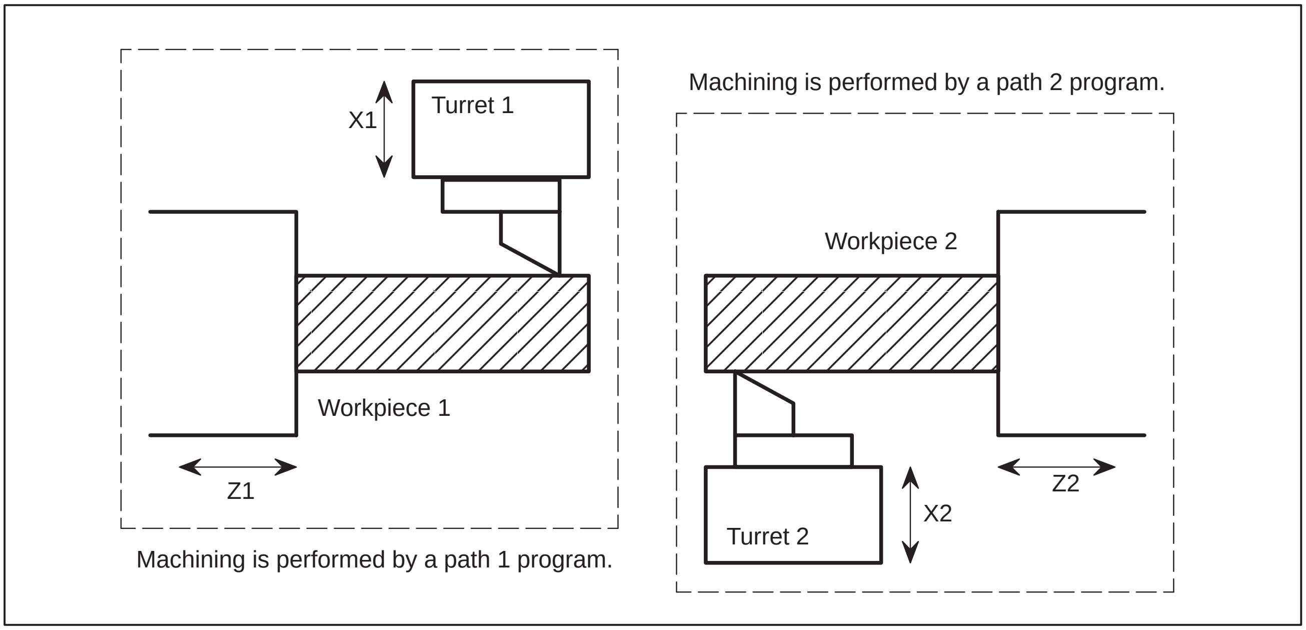
1.9.3.3 Signal operation

When composite control begins or ends (when an M code is issued), the composite control axis selection signals MIX1 to MIX7 for the target axis in path 1 (from the PMC to the CNC) are changed from “0” to “1” (to begin composite control) or from “1” to “0” (to end composite control).

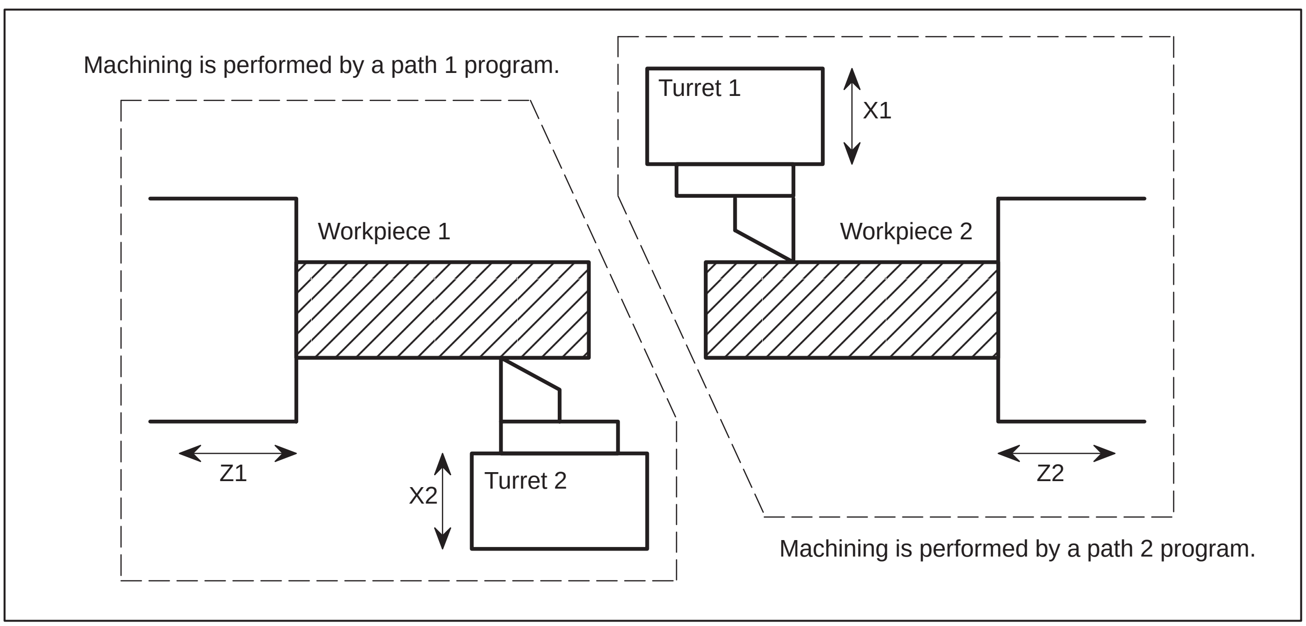
1.9.3.4 Examples of applications

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 indicate 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 indicate the position of turret 1.

1.9.3.5 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 following signals. (See Section 9.4.2 for details.)

- Spindle command selection signal SLSPA <G063#2> and SPSPB <G063#3>
- Spindle feedback selection signal SLPCA <G064#2> and SLPCB <G064#3>

1.9.3.6 Tool offset during composite control

A preset offset or tool–tip radius compensation is not changed when the control mode is switched between independent control and composite control. It is necessary to reset the offset using a T code after the control mode is switched.

1.9.3.7 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. A manual reference position return is not allowed.

1.9.3.8 Move commands after the control mode is switched between independent control and composite control

If the control mode is switched between independent control and composite control during automatic operation, do not issue a move command or re–set a coordinate system for the switched axis in the current block and one or two (during tool–tip radius compensation) subsequent blocks. This restriction is intended to reflect the coordinates changed due to coordinate system setting during control mode switching in the preprocessing for the subsequent blocks.

(Example) Starting composite control to switch between the X1– and X2–axes in block N200

```
N190 ..... ;
N200 M55 ;   (This M code starts composite control.)
N210 ..... ;
N220 ..... ;
N230 ..... ;
```

In this example, not only block N200 but also block N210 (and N220 during tool–tip radius compensation) cannot issue a move command to the X–axis. However, if the M55 code does not involve buffering, it can be issued in block N210 to move the X–axis or update its coordinates. For other than the X–axis, block N210 can issue move commands.

1.9.4 Superimposed Control

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 (bit 3 of parameter No. 8162) can reverse the direction in which the master and slave axes move.

1.9.4.1 Setting

Parameter No. 8186 specifies between which axes move commands are to be superimposed.

(Example) To superimpose the amount of movement of the Z2–axis to that of the Z1–axis: Parameter No. 8186z of path 1 = 2
To superimpose the amount of movement of the X1–axis to that of the Y2–axis: Parameter No. 8186y of path 2 = 1

1.9.4.2 Programming

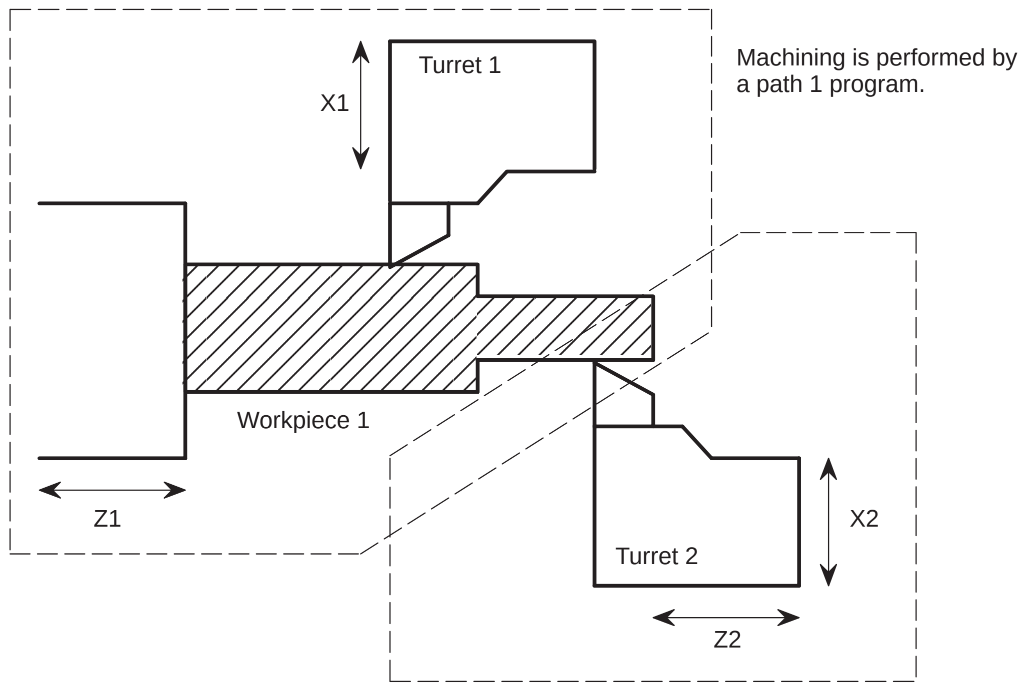
Use M codes for wait, beginning, and terminating superimposed control in a machining program in the stated order. It is also possible to begin and terminate superimposed control without using M codes.

1.9.4.3 Signal operation

When superimposed control begins or ends (when an M code is issued), the superimposed control axis selection signals OVLS1 to OVLS7 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).

1.9.4.4 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 on that of the Z2–axis.



1.9.4.5 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 more larger than a normal speed (such as rapid traverse speed specified in a parameter). To solve this problem, it is necessary to set feedrates that are used only during superimposed control. The feedrates and time constants that are used only during superimposed control include:

- Rapid traverse rate: Parameter No. 8190
- Rapid traverse override F0 rate: parameter No. 8191
- Rapid traverse linear acceleration/deceleration time constant: Parameter No. 8192
- Maximum cutting feedrate: Parameter No. 8193

(• Manual rapid traverse rate:
Parameter No. 8190 or 1424 whichever is smaller)

These special parameters are used for both master and slave axes during superimposed control. Appropriate values should be specified with the resulting feedrate taken into account. When superimposed control begins or ends during automatic operation, it is impossible to switch the maximum cutting feedrate in the current block and the next block. If an M code that does not involve buffering is used to direct superimposed control to begin or end, the maximum cutting feedrate is switched in a block next to the current block. The rates other than the maximum cutting feedrate are switched immediately when superimposed control begins or ends.

1.9.4.6 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.
- When superimposed control is terminated during automatic operation, move commands and coordinate re–setting can be executed for the slave axis immediately. Unlike synchronous control, superimposed control does not inhibit move commands in two or three blocks including the current block.
- A reference position return cannot be specified for the salve axis under superimposed control.

1.9.5 Signal

Synchronous control axis selection signals SYNC1 to SYNC7 <G138#0 to G138#6>

[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 PK7 <G122#0 to G122#6>

[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 selection signal MIX1 to MIX7 <G128#0 to G128#6>

[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 1 These signals are available only for path 1.

Superimposed control axis selection signals OVLS1 to OVLS7 <G190#0 to G190#6>

[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 SYN7O <F118#0 to F118#6>

[Classification] Output signal

[Function] These signals indicate each axis is being subjected to synchronous/composite/superimposed control.

[Output condition] 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.

Note 1 Whether each axis is under synchronous, composite, or superimposed control does not always match whether the corresponding selection signal (synchronous control axis selection, composite control axis selection, or 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 these types of control, they are terminated automatically. Before attempting to perform these types of control, always check the state of these signals.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G122		PK7	PK6	PK5	PK4	PK3	PK2	PK1

	#7	#6	#5	#4	#3	#2	#1	#0
G128		MIX7	MIX6	MIX5	MIX4	MIX3	MIX2	MIX1

	#7	#6	#5	#4	#3	#2	#1	#0
G138		SYNC7	SYNC6	SYNC5	SYNC4	SYNC3	SYNC2	SYNC1

	#7	#6	#5	#4	#3	#2	#1	#0
G190		OVLS7	OVLS6	OVLS5	OVLS4	OVLS3	OVLS2	OVLS1

	#7	#6	#5	#4	#3	#2	#1	#0
F118		SYN7O	SYN6O	SYN5O	SYN4O	SYN3O	SYN2O	SYN1O

1.9.6 Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8160	NRS	SPE				ZSI	XSI	MXC

[Data type] Bit

MXC During mixed control of the X– or Z–axis, measurement direct input function B for tool compensation performs calculation based on:
0 : Machine coordinates for the path being controlled
1 : Machine coordinates for another path subject to mixed control

Note 1 This parameter is valid for setting tool compensation values for the X– or Z axis and setting shift of the workpiece coordinate system for the Z–axis.

Note 2 This parameter cannot be used when mixed control is applied to paths for which different minimum command increments (metric or inch) are specified.

XSI When MXC = 1, the machine coordinates along the X–axis for the other path subject to mixed control are fetched:
0 : With the sign as is
1 : With the sign inverted

ZSI When MXC = 1, machine coordinates along the Z–axis for the other path subject to mixed control are fetched:
0 : With the sign as is
1 : With the sign inverted

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.

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							CZM	NMR

[Data type] Bit

NMR When an axis subject to mixed control is placed in servo–off state:
0 : Mixed control is stopped.
1 : Mixed control is not stopped, provided bit 0 (FUP) of parameter No. 1819 is set to 1 to disable follow–up for the axis.

Note 1 Mixed control is not stopped only when bit 0 (FUP) of parameter No. 1819 is set to 1. If follow–up is disabled with the follow–up signal (*FLWU <G007 bit 5> =1), mixed control is stopped.

CZM When two Cs contour axes are subject to mixed control, the function for mixing zero point return commands for Cs contour axes is:
0 : Not used
1 : Used

	#7	#6	#5	#4	#3	#2	#1	#0
8162	MUMx	MCDx	MPSx	MPMx	OMRx	PKUx	SERx	SMRx

[Data type] Bit axis

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.)

SERx The synchronization deviation is:
0 : Not detected.
1 : Detected.

Note 1 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.

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.

OMRx Superimposed mirror–image control is:
0 : Not applied. (The superimposed pulse is simply added.)
1 : Applied. (The inverted superimposed pulse is added.)

MPMx When composite control is started, the workpiece coordinate system is:
0 : Not set automatically.
1 : Set automatically.

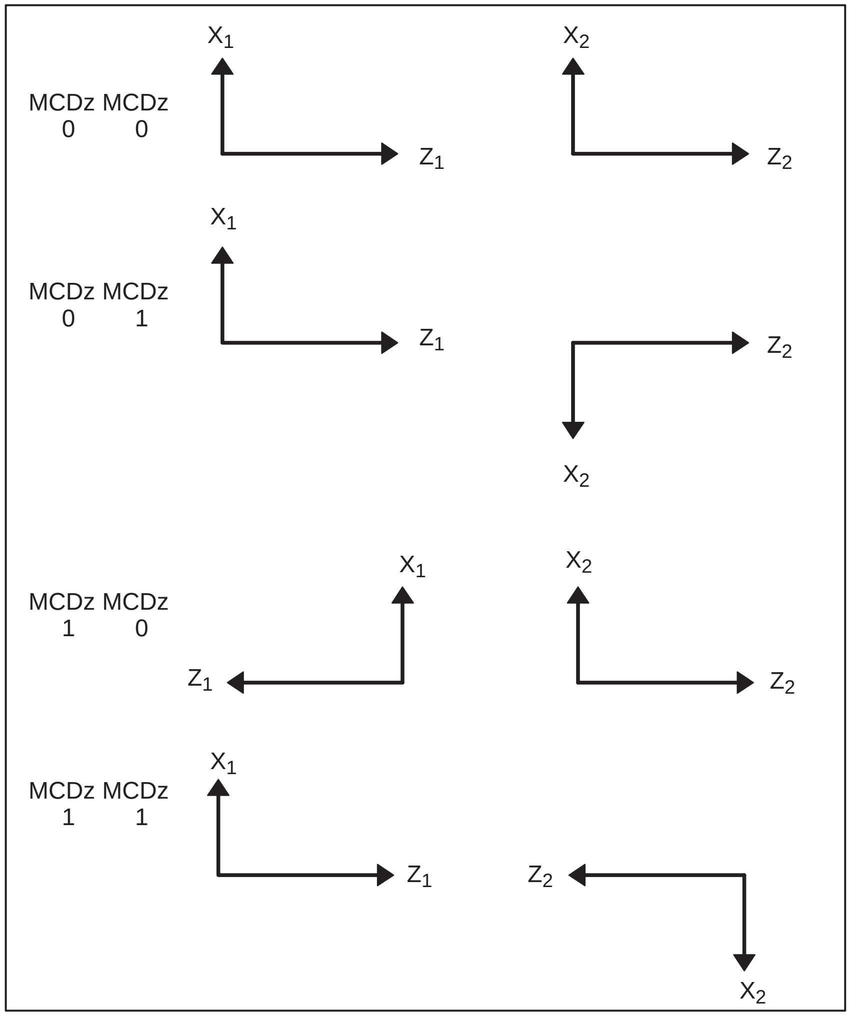
Note 1 When the workpiece coordinate system is automatically set at the start of composite control, it is calculated from the following: Current machine coordinates and the workpiece coordinates at the reference point of each axis (parameter No. 8184).

MPSx When composite control is terminated, the workpiece coordinate system is:
0 : Not set automatically.
1 : Set automatically.

Note 1 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)

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. (The axes of paths 1 and 2 move in the same direction.)
1 : In opposite directions. Mirror–image composite control is applied. (The axes of paths 1 and 2 move in opposite directions.)

This parameter determines the direction in which an axis moves. The parameter is also used to automatically set the coordinate system when composite control is started or terminated.



MUMx In mixed control, a move command for the axis:
0 : Can be specified.
1 : Cannot be specified.

Note 1 Upon the execution of a move command along an axis for which MUMx is set to 1 during mixed control, alarm P/S 226 is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
8163	NUMx			SCDx	SCMx	SPSx	SPMx	MDXx

Note 1 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.

[Data type] Bit axis

MDXx In mixed control, the current position (absolute/relative coordinates) display indicates:
0 : Coordinates in the local system.
1 : Coordinates in the other system under mixed control.

SPMx When synchronous control is started, automatic workpiece coordinate system setting for the master axis is
0 : Not Performed.
1 : Performed.

Note 1 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.

SPSx When synchronous control terminates, automatic workpiece coordinate system setting for the master axis is:
0 : Not performed.
1 : Performed.

Note 1 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.

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.

SCDx The positive (+) directions of the master axis and slave axis in the coordinate system in synchronous control are:
0 : Identical.
1 : Opposite.

NUMx When neither synchronous control nor mixed control is applied, a move command for the axis is:
0 : Not disabled.
1 : Disabled.

Note 1 If a move command is specified for an axis with NUMx set to 1 when neither synchronous control nor mixed control is applied, P/S alarm No. 226 is issued.

8180	Master axis with which an axis is synchronized under synchronous control

[Data type] Byte axis

[Valid data range] 1, 2, 3, ... to the maximum number of control axes, or 201, 202, 203, ... to 200 plus the maximum number of control axes

This parameter specifies the number of the master axis with which an 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 axes, one master axis has two or more slave axes.

- Exercising synchronous control between two paths
  In the parameter of a slave axis, specify the axis number of the master axis with which the salve axis is to be synchronized.
  Setting: 1 to 8
  The value specified here must not exceed the maximum number of control axes.

(Example 1) Synchronizing the Z_2–axis with the Z_1–axis

Path 1	Path 2
Parameter No. 8180x 0	Parameter No. 8180x 0
Parameter No. 8180z 0	Parameter No. 8180z 2
Parameter No. 8180c 0	
Parameter No. 8180y 0	

- Exercising synchronous control in a path
  In the parameter of a slave axis , specify 200 plus the number of the master axis with which the slave axis is to be synchronized.
  Setting: 201 to 208
  The value specified here must not exceed 200 plus the maximum number of control axes.

(Example 1) Synchronizing the Y_1–axis with the Z_1–axis

Path 1	Path 2
Parameter No. 8180x 0	Parameter No. 8180x 0
Parameter No. 8180z 0	Parameter No. 8180z 0
Parameter No. 8180c 0	
Parameter No. 8180y 202	

8181	Synchronization error limit of each axis (Synchronous or composite control)

[Data type] Two–word axis

[Unit of data] Unit of detection

[Valid data range] 0 to 32767

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.

8182	Display of the synchronization error of an axis (synchronous or composite control)

[Data type] Two–word axis

[Unit of data] Unit of detection

[Valid data range] 0 or more

When the synchronization deviation is detected (SERx of Bit #1 parameter No. 8162 is set to 1), this parameter specifies the difference between the positioning deviation of the slave axis and that of the master axis. (The value is used for diagnosis.) The deviation is displayed on the slave side
The parameter is only of display. It should not be set.
The difference between the positioning deviation is:

(Positioning deviation of the master axis) ± (Positioning deviation of the slave axis)

Plus for a mirror–image synchronization command
Minus for a simple synchronization command

8183	Axis under composite control in path 1 corresponding to an axis of path 2

[Data type] Byte axis

[Valid data range] 1, 2, 3, ... to the maximum number of control axes

This parameter specifies an axis of path 1 to be placed under composite control with each axis of path 2. The value specified here must not exceed the maximum number of axes that can be used in path 1. When zero is specified, control of the axis is not replaced under composite control. An identical number can be specified in two or more axes, but composite control cannot be exercised for all of tem at a time.

Note 1 Specify this parameter only for path 2.

(Example 1) Exercising composite control to replace the X_1–axis with the X_2–axis

Path 1	Path 2
Parameter No. 8183x 0	Parameter No. 8183x 1
Parameter No. 8183z 0	Parameter No. 8183z 0
Parameter No. 8183c 0	
Parameter No. 8183y 0	

(Example 2) Exercising composite control to replace the Y_1–axis with the X_2–axis

Path 1	Path 2
Parameter No. 8183x 0	Parameter No. 8183x 4
Parameter No. 8183z 0	Parameter No. 8183z 0
Parameter No. 8183c 0	
Parameter No. 8183y 0	

8184	Coordinates of the reference point of an axis on the coordinate system of another axis under composite control

[Data type] Two–word axis

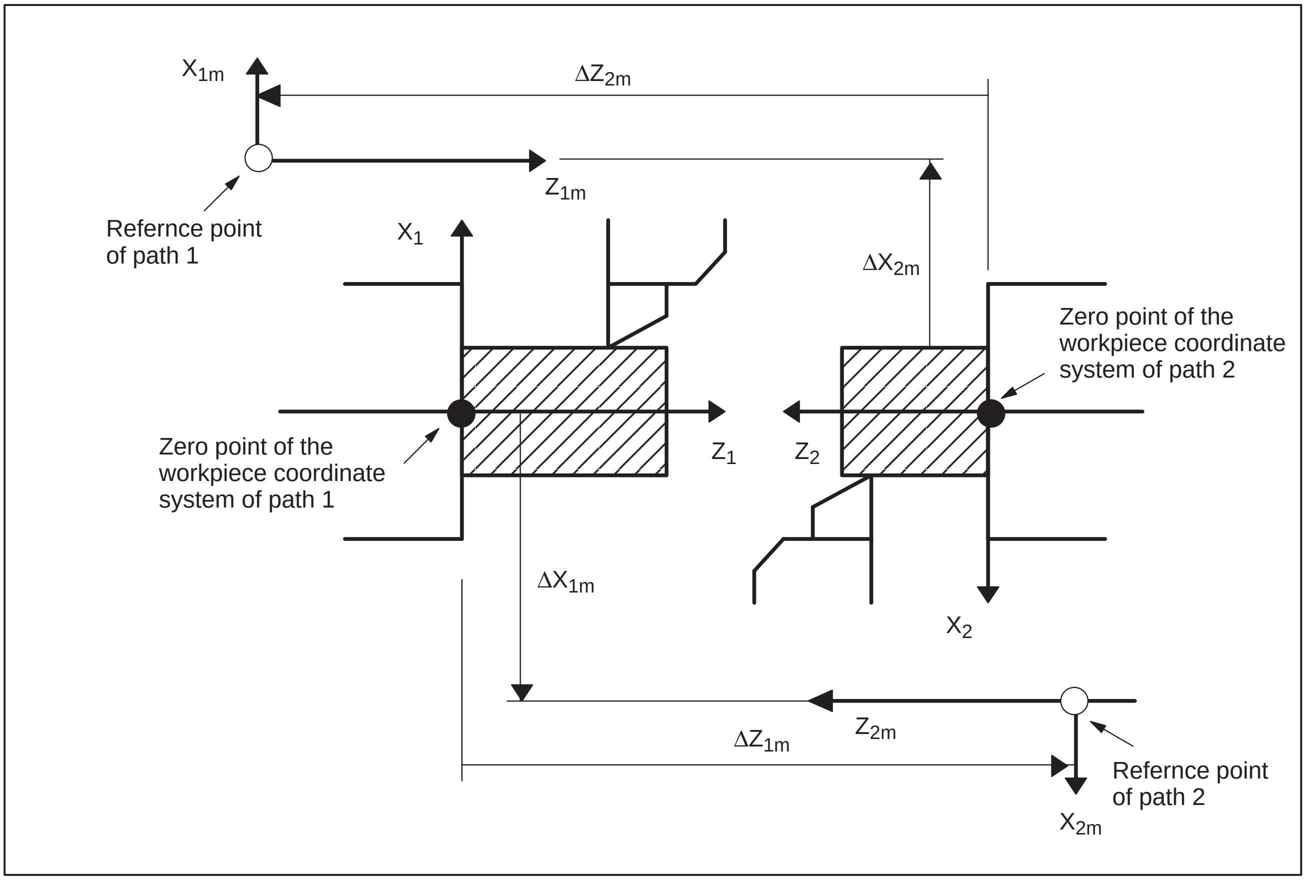
[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 0 to $\pm$99999999

This parameter specifies the coordinates of the reference point of an axis on the coordinate system of another axis under composite control.
The parameter is validated when MPMx of bit 4 parameter No. 8162 is set to 1.

(Example) Exercising composite control to replace the X_1–axis with the X_2–axis



(ΔX_{1m}, ΔZ_{1m}) are the coordinates of the reference point of path 2 on the workpiece coordinate system of path 1. (ΔX_{2m}, ΔZ_{2m}) are the coordinates of the reference point of path 1 on the workpiece coordinate system of path 2.

ΔX_{1m} is specified for the X–axis of path 1 and ΔX_{2m} for the X–axis of path 2.

If bit 4 of parameter No. 8162 MPMx is set to 1 when composite control is started, the workpiece coordinate system satisfying the following conditions is specified:

X_1 = (Value specified for the X–axis of path 1)± (Machine coordinates of X_2)

Plus when parameter No. 8162#6 MCDx of path 1 is set to 0
Minus when parameter No. 8162#6 MCDx of path 1 is set to 1

X_2 =(Value specified for the X–axis of path 2) ± (Machine coordinates of X_1)

Plus when parameter No. 8162#6 MCDx of path 2 is set to 0
Minus when parameter No. 8162#6 MCDx 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:
X_1 = Parameter No. 1250 of path 1 + Machine coordinate of X_1
X_2 = Parameter No. 1250 of path 2 + Machine coordinate of X_2

8185	Workpiece coordinates on each axis at the reference position

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

This parameter specifies the reference position coordinates along the slave axes, according to the workpiece coordinate system for the master axis, when the tool is positioned to the reference position along the master axis. This parameter is enabled when SPMx of bit 1 parameter No. 8163 is set to 1. Set this parameter for the master axis.

8186	Master axis under superimposed control

[Data type] Byte axis

[Valid data range] 1, 2, 3, ... to number of control axes

This parameter specifies the axis number of the master axis under superimposed control.
When zero is specified , the axis does not become a slave axis under superimposed control and the move pulse of another axis is not super imposed.

8190	Rapid traverse rate of an axis under superimposed control

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 to 240000	30 to 100000
Inch machine	0.1 inch/min	30 to 96000	30 to 48000
Rotaion axis	1 deg/min	30 to 240000	30 to 100000

Set a rapid traverse rate for each of the axes when the rapid traverse override of the axes (master and slave axes) under superimposed control is 100%.

8191	F0 velocity of rapid traverse override of an axis under superimposed control

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotaion axis	1 deg/min	6 to 150000	6 to 12000

This parameter specifies the F0 velocity of rapid traverse override for each of the axes (master and slave axes) under superimposed control.

8192	Linear acceleration/deceleration time constant in rapid traverse of an axis under superimposed control

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

This parameter specifies the linear acceleration/deceleration time constant in rapid traverse for each of the axes (master and slave axes) under superimposed control.

8193	Maximum cutting feedrate under superimposed control

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 to 240000	30 to 100000
Inch machine	0.1 inch/min	30 to 76000	30 to 48000
Rotaion axis	1 deg/min	30 to 240000	30 to 100000

This parameter specifies the maximum cutting feedrate under superimposed control.

8194	Maximum cutting feedrate of an axis under superimposed control

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 240000	6 to 100000
Inch machine	0.1 inch/min	6 to 96000	6 to 48000
Rotaion axis	1 deg/min	6 to 240000	6 to 100000

This parameter specifies the maximum cutting feedrate for an axis under superimposed control.

1.9.7 Alarms and messages

If one of the alarms listed below occurs, it terminates synchronous, composite, and superimposed control for all axes.

1.9.7.1 P/S alarms

Number	Message	Description
225	Synchronous or composite control error	This alarm occurs under either of the following conditions (detected when synchronous, composite, or superimposed control is terminated). (1) There is an error in an axis number parameter (parameter No. 1023). (2) An invalid control command is issued. If this alarm occurs when synchronous, composite, or superimposed control is terminated, place the machine in an emergency stop state before resetting the alarm.
226	A move command was issued to a synchronous axis.	When an axis is in a synchronization mode, a move command was issued to that axis. (Only during synchronous control) During composite control, a move command was issued to an axis for which parameter MUMx (bit 7 of parameter No. 8162) is “1”. (Only during composite control) In a control mode other than synchronous or composite control, a move command was issued to an axis for which parameter NUMx (bit 7 of parameter No. 8163) was “1”.
229	Synchronization cannot be maintained.	This alarm occurs under either of the following conditions. (1) Synchronous or composite control cannot be maintained because of system overload. (2) Synchronous or composite control cannot be maintained because of a hardware failure. (This alarm does occur during normal use.)
000	Turn the power off.	This message is issued if superimposed control is suspended because of an alarm that occurs when the axis is moving. Turn the power of the CNC off, then on gain.

1.9.7.2 Servo alarms

Number	Message	Description
407	Servo alarm: Excessive error	A positional deviation for a synchronous axis exceeded the specified value. (Only during synchronous control).

1.9.8 Cautions

1.9.8.1 Items common to synchronous, composite, and superimposed control

(1) When synchronous, composite, or superimposed control begins or ends, the target axes must be at a stop.

(2) More than one axis can be subjected to synchronous, composite, or superimposed control. On the other hand, an axis cannot be synchronized with more than one axis simultaneously. Moreover, an axis under composite control cannot be synchronized with another axis or cannot doubly be subjected to composite control.

(3) All axes subjected to synchronous, composite, or superimposed control must have the same least command, detection increment, and diameter/radius specification. Otherwise, the amount of movement will differ from one axis to another.

(4) Synchronous, composite, or superimposed control cannot be performed between a linear axis and a rotation axis.

(5) When an axis is under synchronous, composite, or superimposed control, do not change the parameters related to that axis.

(6) Before starting synchronous, composite, superimposed control, make sure that for the target axis, a reference position return after power–on has been made or a reference position has been set up according to the absolute pulse coder.

(7) Before starting synchronous, composite, or superimposed control after an emergency stop, servo–off, or servo alarm is released, be sure to make a return to the reference position and set up the necessary coordinate system.

(8) Acceleration/deceleration control, pitch error compensation, backlash compensation, and stored stroke limit check are carried out regardless of synchronous or composite control. During superimposed control, these operations except acceleration/deceleration are performed on the position where superimposed pulses have been added.

(9) Synchronous, composite, or superimposed control cannot be specified simultaneously with simplified synchronous control. Synchronous control within one path provides the same functions as simplified synchronous control.

1.9.8.2 Items related only to synchronous control

(1) A move command should not be issued to a synchronous slave axis during synchronous control.

(2) The same acceleration/deceleration time constants and servo parameters should be used for axes subjected to synchronous control as much as possible. If there is a large difference in a set value between the axes, a deviation will occur in the actual movement of the machine.

(3) The workpiece coordinate system of a synchronous slave axis is not affected by the synchronous master axis operations that affect workpiece coordinate systems but do not cause the machine to move, such as workpiece coordinate system set/shift and geometry offset commands.

(4) If a wear offset command or tool–tip radius compensation is performed for the synchronous master axis, the travel path of the slave axis is shifted by the offset, but the shift is not set as an offset (no offset vector is created).

1.9.8.3 Restrictions imposed during synchronous, composite, and superimposed control

Function	During synchronous control	During composite control	During superimposed control
Acceleration/deceleration control	The acceleration/deceleration control for the master axis is performed also for the synchronous slave axes, but different time constants are used.	The acceleration/deceleration control originally specified for one path is used also for the other path, but different time constants are used (*3).	The move pulses that are effective after acceleration/deceleration for the superimposed control master axis are added to those for the slave axes.
Post–interpolation linear acceleration/deceleration	Possible	Possible	Possible
Feedrate clamping	The axes are clamped at the feedrate of the master axis.	The axes in both paths are clamped at the feedrate originally specified for one path.	The axes are clamped to the feedrate specified for superimposed control.
Reference position return	A reference position return is possible for the master axis unless it is in a parking state. If the master axis is in a parking state, only automatic reference position return (G28) is possible for the master axis. (⇒ Section 1.9.2.5)	A reference position return is possible for axes not under composite control. For axes under composite control, only an automatic reference position return (G28) is possible.	Impossible for superimposed control slave axes.
Second–, third–, or fourth–reference position return	Possible (⇒ 1.9.2.5).	Possible	Impossible for superimposed control slave axes.

Function	During synchronous control	During composite control	During superimposed control
Reference position return check	Possible (⇒ 1.9.2.5).	Possible	Impossible for superimposed control slave axes.
PMC axis control	Possible for other than synchronous slave axes.	Possible	Possible
Polar coordinate interpolation and cylindrical interpolation	Possible	Switching between independent control and composite control should be carried out during cancel mode.	Possible
Handle interrupt	Performed regardless of synchronous control.	Possible for axes having nothing to do with composite control.	Performed regardless of superimposed control (*5).
Mirror image	Each signal originally belonging to a particular axis is effective for that axis (*1).	Signals originally specified for one path are effective for the other path (*4).	Signals for the master axis are effective as superimposed control pulses (*5).
Machine lock	Each signal originally belonging to a particular axis is effective for that axis (*1).	Signals originally specified for one path are effective for the other path (*4).	Signals for the master axis are effective as superimposed control pulses (*5).
Interlock	The signals for the synchronous master axis are effective for the synchronous slave axes (*2).	Signals originally specified for one path are effective for the other path (*4).	Signals for the master axis are effective as superimposed control pulses (*5).
Override	The signals for the synchronous master axis are effective for the synchronous slave axes (*2).	Signals originally specified for one path are effective for the other path (*4).	Signals for the master axis are effective as superimposed control pulses (*5)
External deceleration	The signals for the synchronous master axis are effective for the synchronous slave axes (*2).	Signals originally specified for one path are effective for the other path (*4).	Signals for the master axis are effective as superimposed control pulses (*5)
Skip function	Impossible for slave axes.	Possible for axes having nothing to do with composite control.	Possible.
Automatic tool compensation	Impossible for slave axes.	Possible for axes having nothing to do with composite control.	Impossible for superimposed control slave axes.
Direct tool compensation measurement input B	Impossible for slave axes.	Possible (*7)	Impossible for superimposed control slave axes.
Follow–up	Impossible during synchronous control.	Impossible during composite control.	Impossible during superimposed control.
Program restart	Impossible for a program involving synchronous control.	Impossible for a program involving composite control.	Impossible for a program involving superimposed control.
Cs contour control	Synchronous control is possible (*6).	Composite control is possible (*6).	Superimposed control is possible (*6).
Spindle positioning	Synchronous control is impossible.	Composite control is impossible.	Superimposed control is impossible.

(*1) Processed after synchronization pulses are sent to the slave axes.

(*2) After it is processed on the master side, synchronization pulses are sent.

(*3) Composite control pulses and acceleration/deceleration type are sent. The time constant for the slave axis is used.

(*4) Composite control pulses are sent after processed on the master side.

(*5) Performed normally for move commands originally intended to the master or slave axes, but not performed on the slave side for superimposed control pulses received from the master axis.

(*6) Restricted to a combination of Cs axes. Necessary signal operations and orientation should be performed for each axis separately. Also specify parameter CZM (bit 1 of parameter No. 8161).

(*7) Specify parameters MXC, XSI, and ZSI (bits 0, 1, and 2 of parameter No. 8160).

1.9.8.4 Reading the coordinates during synchronous, composite, or superimposed control

The following list summarizes how positional information such as custom macro system variables and current coordinates from the PMC window are read during synchronous, composite, or superimposed control.

Positional information	During synchronous control	During composite control	During superimposed control
Absolute coordinate	Readable	Readable (*1)	Readable (*2)
Machine coordinate	Readable	Readable	Readable
End of each block	Readable only for the master axis	Readable (*1)	Readable (*2)
Skip signal position	Readable only for the master axis	Unreadable	Readable (*2)

(*1) The coordinates are represented in the coordinate system that is effective during composite control. Their relationship with the machine coordinate system differs from the relationship that exists during independent control.

(*2) No superimposed control pulse is added.

1.9.8.5 Terminating synchronous, composite, or superimposed control

Synchronous, composite, or superimposed control is terminated not only when the corresponding synchronization signal becomes off but also when the following conditions occur.

(1) Emergency stop

(2) Reset

(3) Servo alarm

(4) Servo off (*1)

(5) Overtravel

(6) Alarm related to synchronous, composite, or superimposed control

(7) P/S000 alarm

If one of the above conditions occurs for either path, it terminates synchronous control, composite, and superimposed control for all axes. If one of the above conditions occurs for one path during synchronous, composite, or superimposed control, the other path is placed in a feed hold state (during automatic operation) or interlock state (during manual operation).

(*1) Setting parameter NMR (bit 0 of parameter No. 8161) specifies that synchronous, composite, or superimposed control be not terminated even when an axis under composite control enters a servo–off state. (If an axis under synchronous or superimposed control enters a servo–off state, synchronous, composite, and superimposed control is terminated.

1.9.8.6 Status output signals for an axis under synchronous, composite, or superimposed control

Status output signal	During synchronous control	During composite control	During superimposed control
Axis moving signal MVn F0102/F1102 (See Section 1.2.5.)	• The master axis moving signal becomes ”1” when the master or slave axis is moving. • The slave axis moving signal is always ”0” (*1).	• The moving signal for an axis to which a move command is originally issued becomes ”1”. The moving signal for the axis that is actually moving does not become ”1” (*1).	• The master axis moving signal works as usual. • The slave axis moving signal reflects the state of movement due to a command for the slave axis rather than superimposed control pulses.
Axis movement direction signal MVDn F0106/F1106 (See Section 1.2.5.)	• The master axis movement direction signal indicates the direction in which the master axis is moving. • The slave axis movement direction signal indicates the direction of movement after synchronous control mirror image processing.	• The axis movement direction signal indicates the actual movement direction (that is, direction after composite control mirror image processing).	• The master axis movement direction signal indicates the direction in which the master axis is moving. • The slave axis movement direction signal indicates the direction of movement after superimposed control pulses are added.
Axis in position signal INPn F0104/F1104 (See Section 7.2.6.1.)	• The master axis in position signal becomes “1” when both master and slave axes are in position. • The slave axis in position signal is always “1”.	• The in position signal for an axis in a path for which a move command is issued reflects the state of the axis that is driven by that move command.	• The master axis in position signal works as usual. • The slave axis in position signal is always “1”.

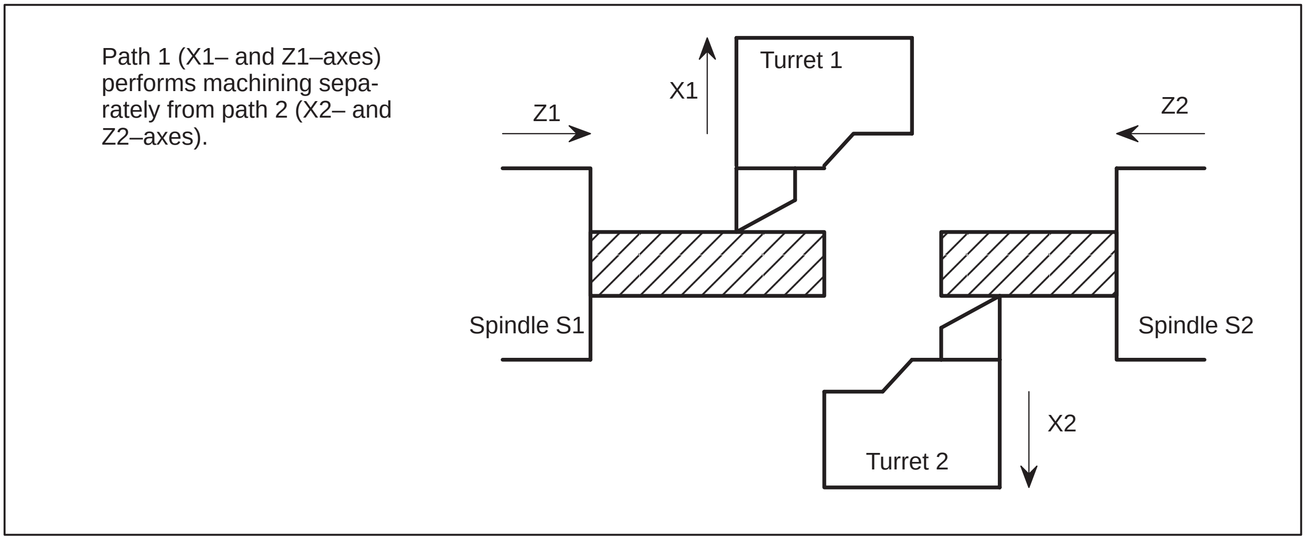
(*1) A positional deviation check does not depend on the state of this signal. If move command pulses have been output to a motor (either master or slave), parameter No. 1828 is used as a limit. Otherwise, parameter No. 1829 is used.

1.9.9 Examples of Applications

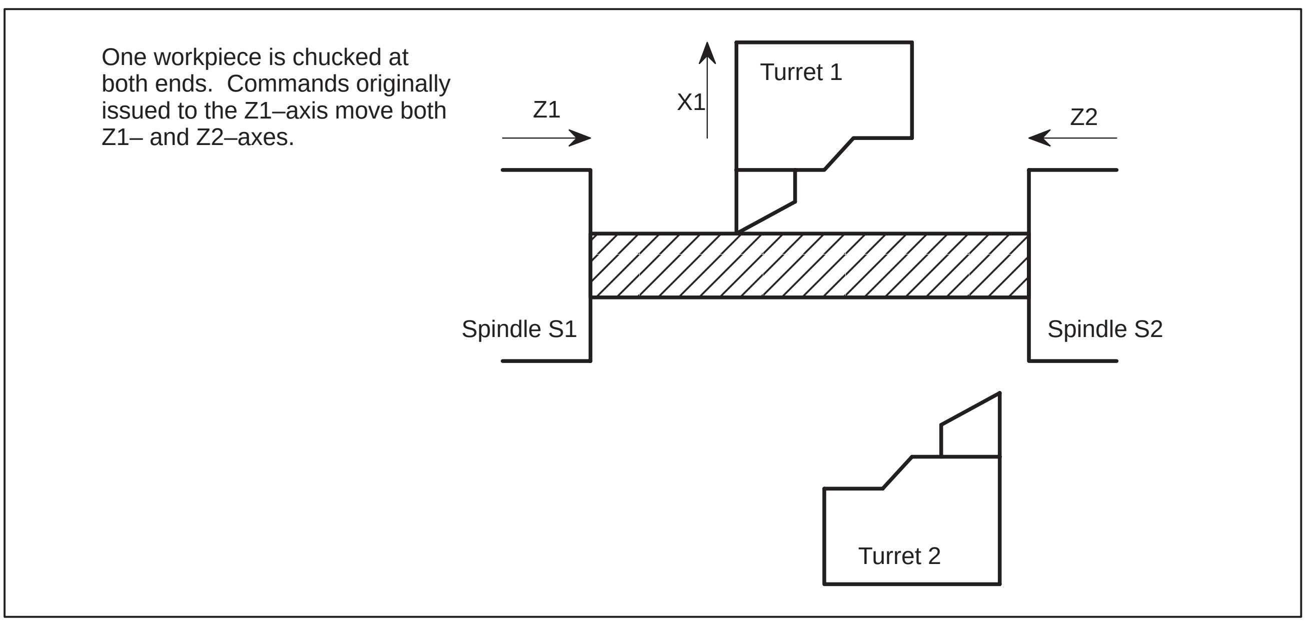
1.9.9.1 Independent control and synchronous control of the Z1– and Z2–axes

(1) Machine configuration

(a) Independent control



(b) Synchronous control of the Z1– and Z2–axes



(2) Parameter setting

- To synchronize the Z2–axis with the Z1–axis, set parameter No. 8180z of path 2 to “2”.

- To apply mirror–image synchronization (because initially the positive direction of one axis is opposite to that of the other axis), set SMRz (bit 0 of parameter No. 8162) of path 2 to "1".
- To detect out–of–synchronization (because both axes should move by the same amount), set SERz (bit 1 of parameter No. 8162) to "1". Set a value from 100 to 1000 as a limit to out–of–synchronization in parameter No. 8181z of path 2 (this limit varies from one machine to another).
- A difference in the positional deviation between the Z1– and Z2–axes is indicated in parameter No. 8182z during synchronization.

(3) Signal operation

- Set signal G1138#1 SYNC2 to "1" when the Z1– and Z2–axes start moving in synchronization.
- Reset signal G1138#1 SYNC2 to "0" when synchronization is terminated.
- Also reset signal G1138#1 SYNC2 to "0" if an emergency stop, NC reset, or alarm occurs.
- Keep signals other than G1138#1 SYNC2 reset to "0".

(4) Sample program

Path 1	Path 2	
N1000 ········	N2000 ········	Machining under control independent of the other path
N1010 Z80. ;	N2010 Z150. ;	Moves the workpiece and chuck to the specified position.
N1020 M200 ;	N2020 M200 ;	Waits for completion of movement.
N1030 M61 ;		Clamps the workpiece and begins synchronization
N1040 M3 S800 ;		Turns the spindle in normal direction.
N1050 Z- 25. ;		Moves the Z1–axis.
N1060 ········		Machining with the X1– and Z1–axes
N1070 M62 ;		Terminates synchronization and unclamps the workpiece.
N1080 M201 ;	N2080 M201 ;	Waits for synchronization to be terminated.
N1090 ;	N2090 ;	Dummy block (performing no move command)
N1100 ········	N2100 ·······	Machining under control independent of the other path

In this example, assume that M61 clamps the workpiece and sets signal G1138#1 SYNC2 to “1” and that M62 resets signal G1138#1 SYNC2 to “0” and unclamps the workpiece.

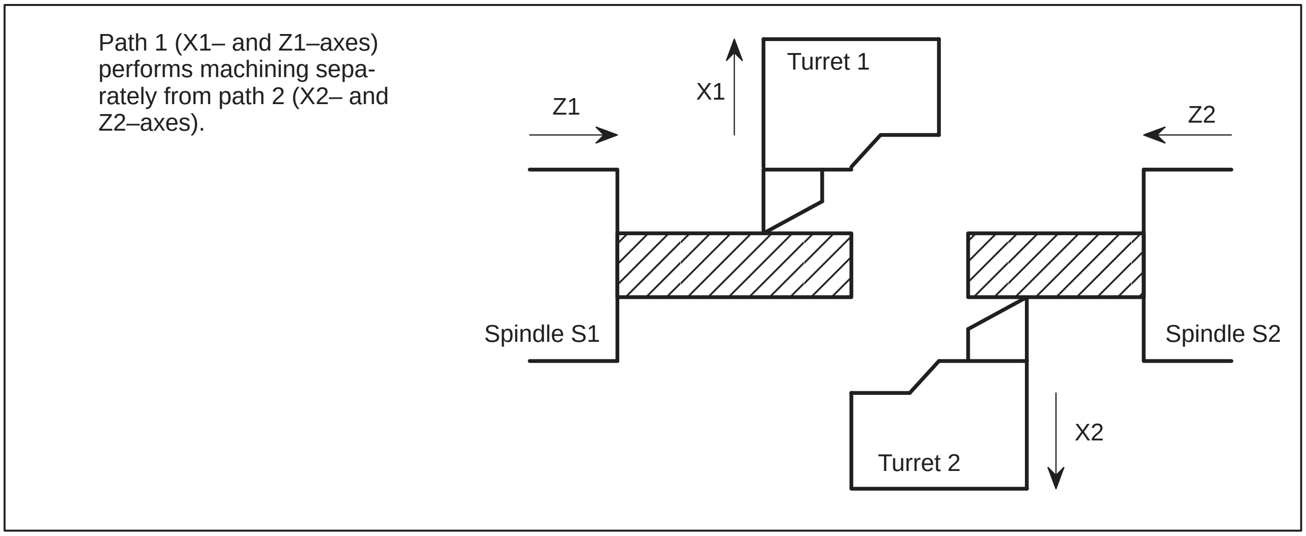
(5) Cautions

- It is necessary to make the speed of spindle S1 equal that of spindle S2. For example, issue spindle commands of path 1 to both S1 and S2.

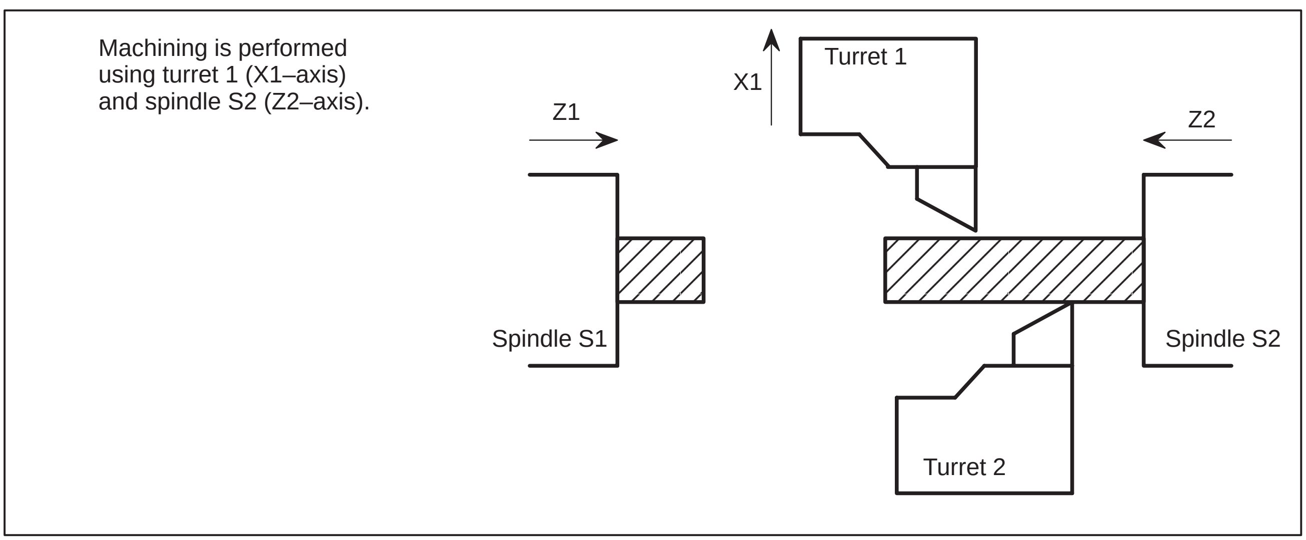
1.9.9.2 Independent control and interpolation for the X1– and Z2–axes

(1) Machine configuration

(a) Independent control



(b) Interpolation for the X1– and Z2–axes



Interpolation for the X1– and Z2–axes can be carried out by either of the following two methods.

1. The path 2 program directs the X2– and Z2–axes, synchronizes the X1–axis with the X2–axis, and causes the X2–axis to park. The path 1 program issues no move command.
2. Composite control is performed in which move commands are switched between the X1–axis in one path and the X2–axis in the other path. Path 1 does not issue move commands.

The following sections describe a case in which synchronous control is used and a case in which composite control is used, separately.

1.9.9.2.1 Using synchronous control

(1) Parameter setting

- To synchronize the X1–axis with the X2–axis, set parameter No. 8180x of path 1 to “1”.
- Do not specify mirror image, because for both X1– and X2–axes, the direction in which they go away from the workpiece center is defined as positive.
- Do not specify out–of–synchronization detection for the X2–axis because it is caused to park.
- Parameter No. 8182x indicates a difference in the positional deviation between the X2– and X1–axes during synchronous control.

(2) Signal operation

- Set signals G0138#0 SYNC1 and G1122#0 PK1 to “1” when synchronous control begins for the X2– and X1–axes.
- Reset signals G0138#0 SYNC1 and G1122#0 PK1 to “0” when synchronization is terminated.
- Also reset signals G0138#0 SYNC1 and G1122#0 PK1 to “0” if an emergency stop, NC reset, or alarm occurs.
- Keep signals other than G0138#0 SYNC1 or G1122#0 PK1 reset to “0”.

(3) Sample program

Path 1	Path 2	
N1000	N2000	Machining under control independent of the other path
N1010 Z0 ;	N2010 Z20. ;	Moves the workpiece to the specified position.
N1020 X120. ;	N2020 X120. ;	Moves each X–axis to their start position for synchronization (X1 = X2)
N1030 M200 ;	N2030 M200 ;	Waits for completion of movement.
	N2040 M55 ;	Synchronizes the X2– and X1–axes and causes the X2–axis to park.

	N2050 T0212 ;	Specifies an offset for turret 1.
	N2060 S1000 M4 ;	Reverses the spindle.
	N2070 G0 X30. Z55. ;	Performs machining using the X1– and Z2–axes.
	N2080 G1 F0. 2 W- 15. ;	
	N2090 ·······	
	N2100 M56 ;	Terminates synchronization and parking.
N1110 M201 ;	N2110 M201 ;	Waits for synchronization to be terminated.
N1120 ;	N2120 ;	Dummy block (performing no move command)
N1130 ······	N2130 ········	Machining under control independent of the other path

In this example, assume that M55 begins control of turret 1 in path 2 and that M56 terminates control of turret 1 in path 2.

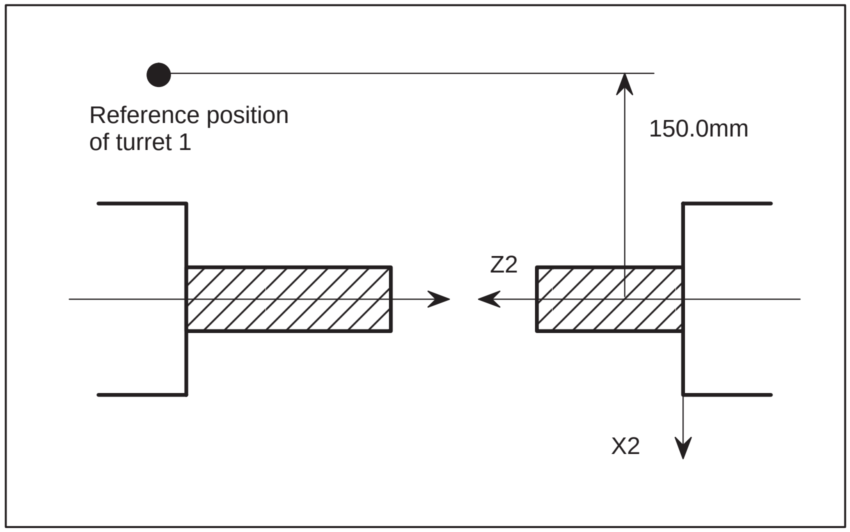
(4) Cautions

- When the X–axis is under synchronous control, path 1 cannot issue a move command to the X1–axis, but can move the Z1–axis.

1.9.9.2.2 Using composite control

(1) Parameter setting

- To specify composite control in which commands for the X1–axis are interchanged with those of the X2–axis, set parameter No. 8183x of path 2 to “1”.
- Set MCDx (bit 6 of parameter No. 8162) of path 2 to “1”, because the direction of the X1–axis is opposite to that of the X2–axis.
- To cause the position of turret 1 to be specified automatically in the workpiece coordinate system in path 2 when composite control begins, set MPSx (bit 5 of parameter No. 8162) to “1”.
- To cause the position of turret 1 to be specified automatically in the workpiece coordinate system in path 1 when composite control ends, set MPMx (bit 4 of parameter No. 8162) to “1”.
- Assuming that the X–coordinate of the reference position of turret 1 in the workpiece coordinate system in path 2 is –150.0 mm as shown below, set “–150000” in parameter No. 8184x of path 2 for automatic coordinate system setting.



(2) Signal operation

- Set signal G0128#0 MIX1 to “1” when composite control begins for the X2– and X1–axes.
- Reset signal G0128#0 MIX1 to “0” when composite control ends.
- Also reset G0128#0 MIX1 to “0” when an emergency stop, NC reset, or alarm occurs.
- Keep signals other than G0128#0 MIX1 reset to “0”.

(3) Sample program

Path 1	Path 2	
N1000 ·········	N2000 ·········	Machining under control independent of the other path
N1010 Z0 ;	N2010 Z20. ;	Moves each workpiece to the specified position.
	N2020 X120. ;	Moves the X2–axis to a position where no interference occurs.
N1030 M200 ;	N2030 M200 ;	Waits for completion of movement.
	N2040 M55 ;	Begins composite control of the X2– and X1–axes (the position of turret 1 is set up as workpiece coordinates in path 2.)
	N2050 ;	Dummy block (performing no move command)
	N2060 T0212 ;	Specifies an offset for turret 1.
	N2070 S1000 M4 ; !	Performs machining using the X1– and Z2–axes.
	N2080 G0 U10. W- 20. ;	
	N2090 G1 F0. 2 W- 15. ;	
	N2100 ········	

	N2110 M56 ;	Terminates composite control (the position of turret 1 is set up as workpiece coordinates in path 1.)
N1120 M201;	N2120 M201 ;	Waits for composite control to be terminated.
N1130 ;	N2130 ;	Dummy block (performing no move command)
N1140 ········	N2140 ·······	Machining under control independent of the other path

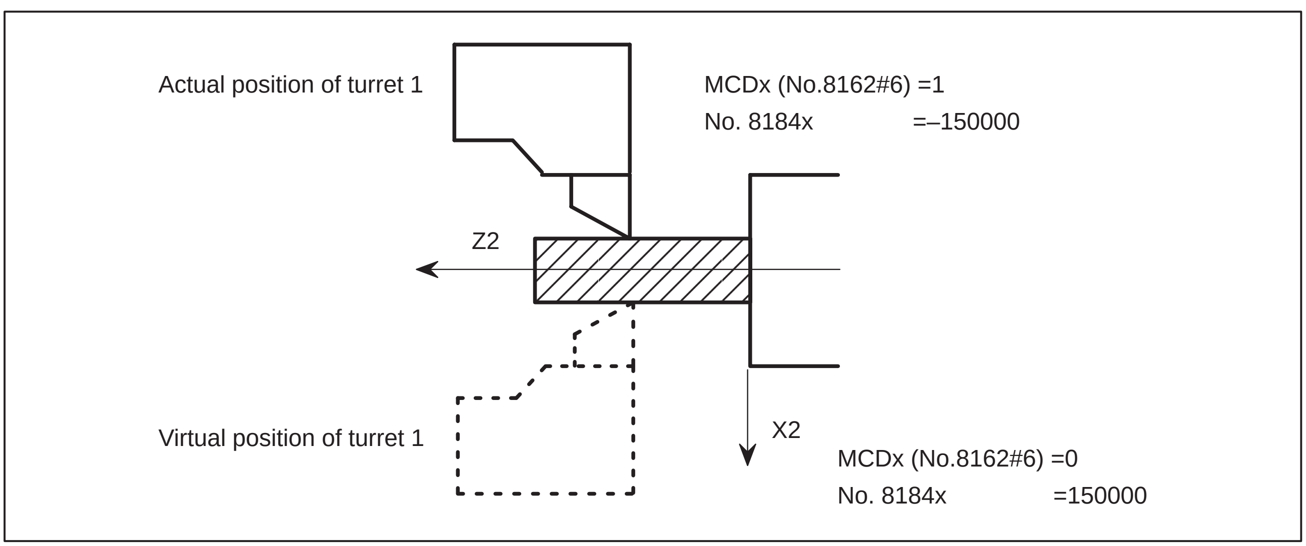
In this example, assume that M55 begins control of turret 1 by a path 2 program and that M56 terminates control of turret 1 by a path 2 program.

(4) Cautions

- It is not always necessary to cause a coordinate system to be set up automatically when composite control begins or ends. If automatic coordinate system setting is not specified, an appropriate coordinate system is set by program.
- When the X–axis is under composite control, the X2–axis can be moved in path 1 using move commands for the X–axis.
- The above parameter setting specifies that turret 1 is located on the negative side of the X–coordinate in the workpiece coordinate system of path 2. So, for example, to move turret 1 toward the center of the workpiece, specify U+10, and to move it away from the center, specify U–10 (note the sign is a minus). If this is inconvenient, set the following parameters as follows:

  Bit 6 of parameter No. 8162 (MCDx) = 0
  Parameter No. 8184x = 150000

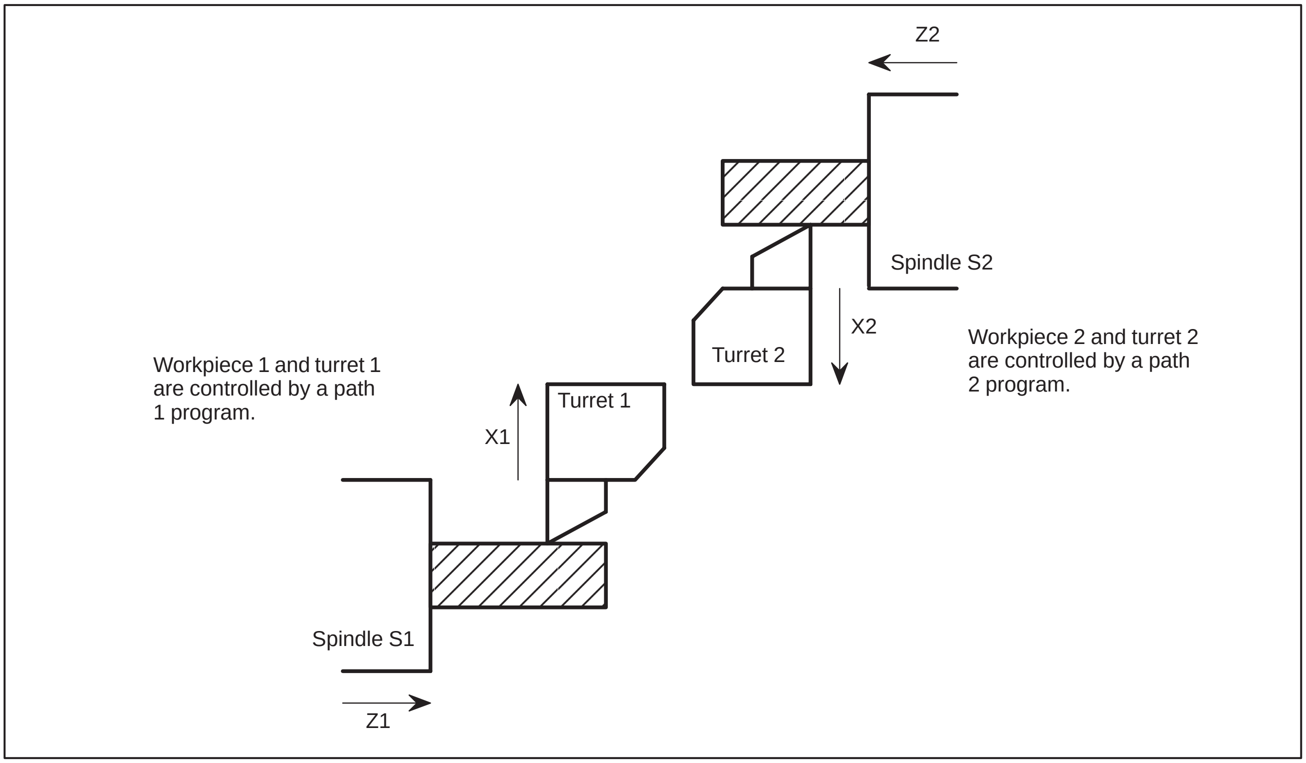
  This parameter setting specifies that turret 1 be located virtually on the positive side of the X–coordinate.



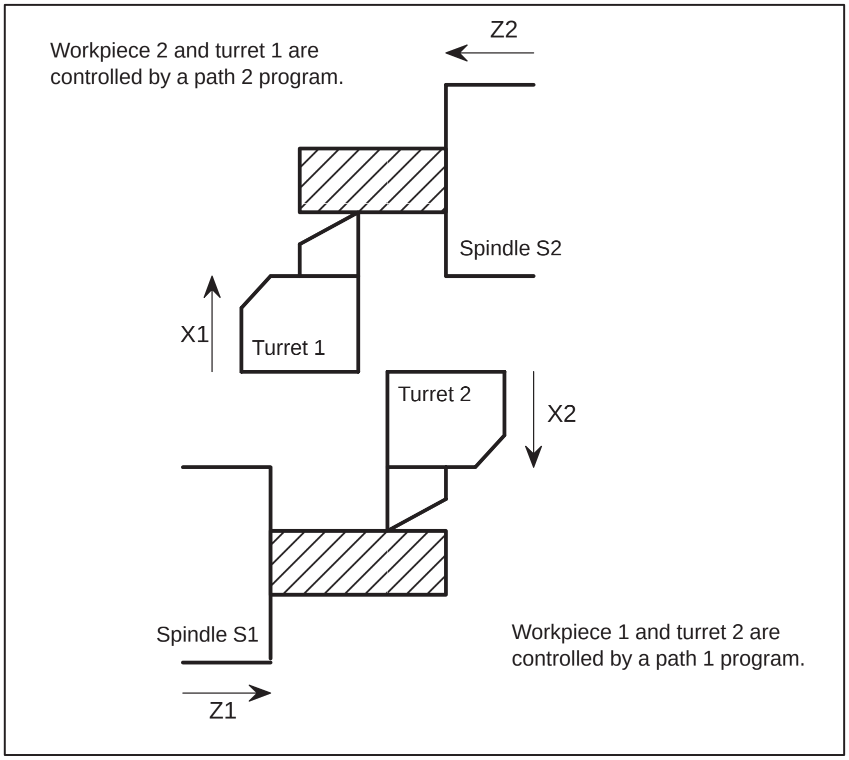
1.9.9.3 Independent control and interpolation between the X1– and Z2–axes and between the X2– and Z1–axes

(1) Machine configuration

(a) Independent control

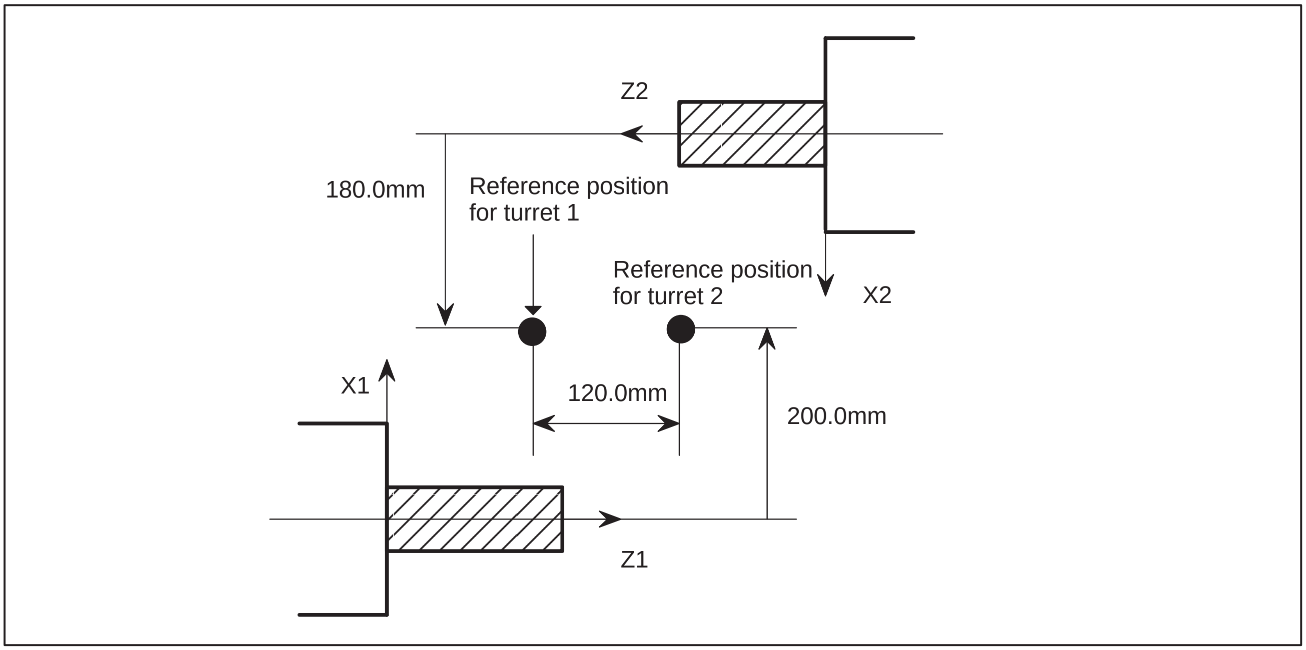


(b) Interpolation between the X1– and Z2–axes and between the X2– and Z1–axes



(2) Parameter setting

- To specify composite control in which commands for the X1–axis are interchanged with those for the X2–axis, set parameter No. 8183x of path 2 to “1”.
- Set MCDx (bit 6 of parameter No. 8162) of paths 1 and 2 to “1”, because the direction of the X1–axis is opposite to that of the X2–axis.
- To cause the position of a turret in one path to be specified automatically in the workpiece coordinate system of the other path when composite control begins, set MPMx (bit 4 of parameter No. 8162) to “1”.
- To cause the position of a turret in each path to be specified automatically in the workpiece coordinate system of that path when composite control ends, set MPSx (bit 5 of parameter No. 8162) to “1”.
- Assuming that the relationships between the workpiece coordinates and reference position of each path are as shown below, set “200000” in parameter No. 8184x of path 1 and “180000” in parameter No. 8184x of path 2 for automatic coordinate system setting.



(3) Signal operation

- Set signal G0128#0 MIX1 to “1” when composite control begins for the X2– and X1–axes.
- Reset signal G0128#0 MIX1 to “0” when composite control ends.
- Also reset G0128#0 MIX1 to “0” when an emergency stop, NC reset, or alarm occurs.
- Keep signals other than G0128#0 MIX1 reset to “0”.

(4) Sample program

Path 1	Path 2	
N1000	N2000	Machining under control independent of the other path
N1010 M350 ;	N2010 M350 ;	Waits for composite control to begin.
	N2020 M55 ;	Begins composite control for the X1– and X2–axes.
N1030 M351 ;	N2030 M351 ;	Composite control has begun.
N1040 ;	N2040 ;	Dummy block (performing no move command)

N1050 T0313	N2050 T0212 ;	Selects a tool for composite control and sets the offset.
N1060 G50 W120. ;	N2060 G50 W120. ;	Shifts the Z–axis workpiece coordinate system.
N1070 S1000 M4 ;	N2070 S1500 M4 ;	Performs machining under composite control.
N1080 G0 X20. Z15. ;	N2080 G0 X15. Z30.;	
N1090 G1 F0. 5 W- 8. ;	N2090 G1 F0. 1 W- 5. ;	
N1100	N2100	
N1110 M360 ;	N2110 M360 ;	Waits for composite control to be terminated.
	N2120 M56 ;	Terminates composite control.
N1130 M361 ;	N2130 M361 ;	Composite control has ended.
N1140 ;	N2140 ;	Dummy block (performing no move command)
N1150 G50 W- 120. ;	N2150 G50 W- 120. ;	Shifts the Z–axis workpiece coordinate system.
N1160	N2160	Machining under control independent of the other path

In this example, assume that M55 begins composite control (sets signal G0128#0 MIX1 to "1") and that M56 terminates composite control (resets signal G0128#0 MIX1 to "0").

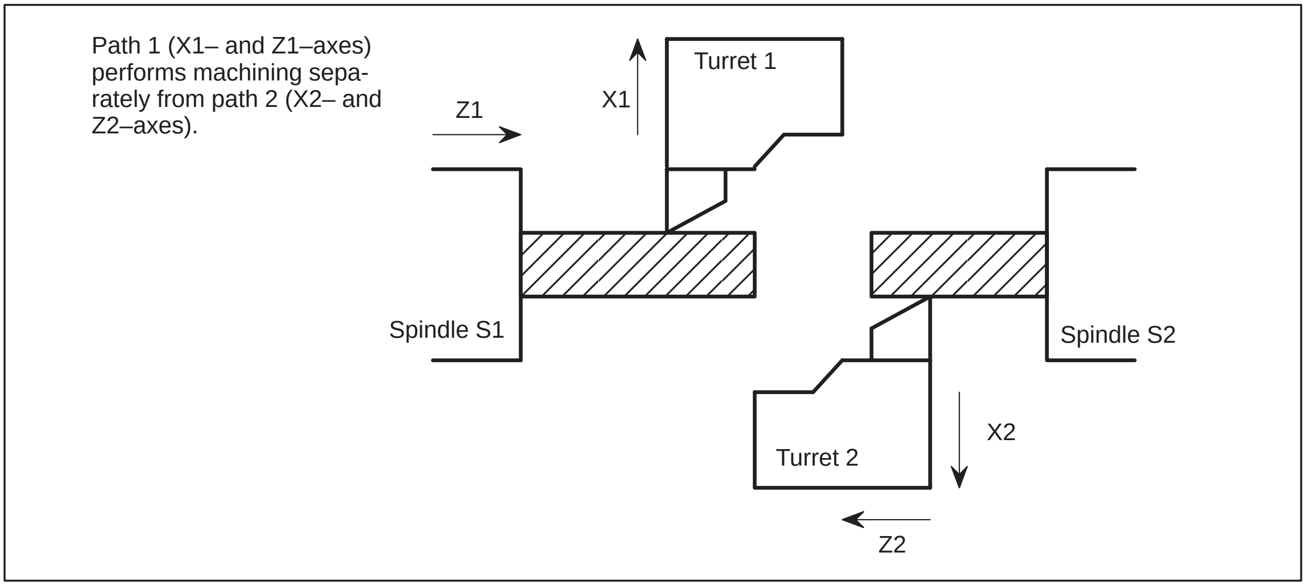
(5) Cautions

- It is not always necessary to cause a coordinate system to be set up automatically when composite control begins or ends. If automatic coordinate system setting is not specified, an appropriate coordinate system is set by program.

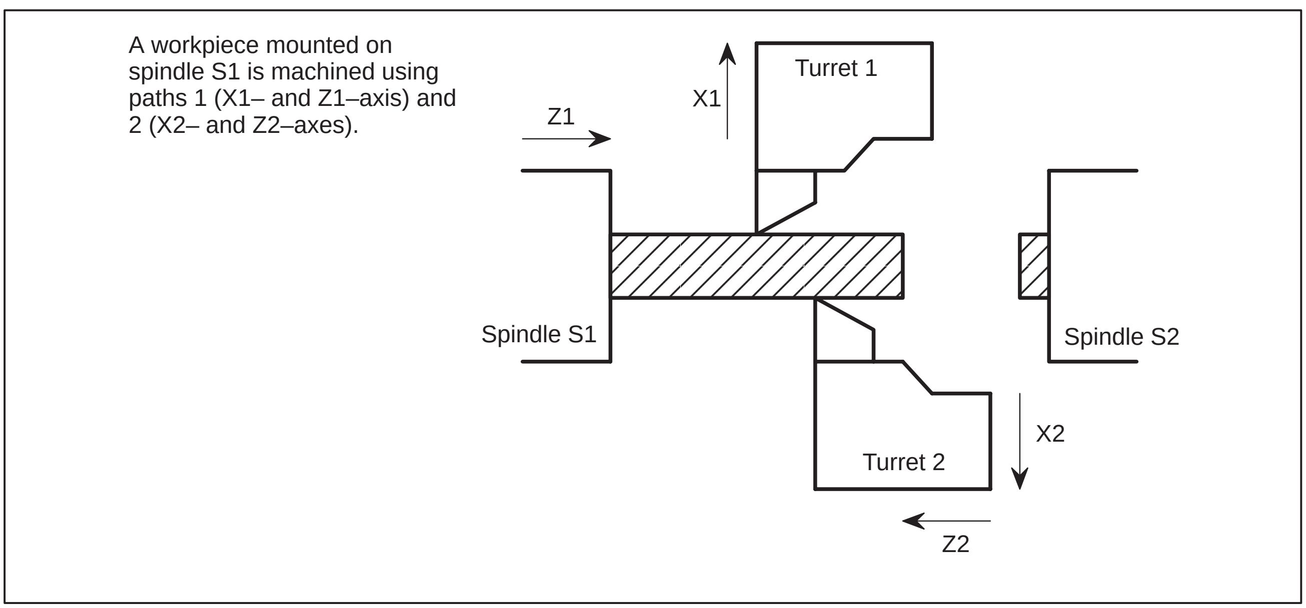
1.9.9.4 Independent control and superimposed control for the Z1– and Z2–axes

(1) Machine configuration

(a) Independent control



(b) Superimposed control for the Z1– and Z2–axes



(2) Parameter setting

- To superimpose the move commands for the Z1–axis on the Z2–axis, set parameter No. 8186z of path 2 to “2”.
- To apply mirror–imaged superimposed control (because the positive directions of the Z1– and Z2–axes do not match), set parameter No. 8162 (OMRz) of path 2 to “1”.

- Set the feedrate along each Z–axis for superimposed control in parameter Nos. 8190z, 8191z, and 8193 of both paths. Each value to be set must be about half the one for independent control.
- Set the rapid traverse time constant for each Z–axis under superimposed control in parameter No. 8192z of both paths. Each value to be set must be 1 to 2 times the one for independent control.

(3) Signal setting

- Set signal G1190#1 OVLS2 to “1” when superimposed control begins for the Z1– and Z2–axes.
- Reset signal G1190#1 OVLS2 to “0” when superimposed control ends.
- Also reset G1190#1 OVLS2 to “0” when an emergency stop, NC reset, or alarm occurs.
- Keep signals other than G1190#1 OVLS2 reset to “0”.

(4) Sample program

Path 1	Path 2	
N1000 ········	N2000 ········	Machining under control independent of the other path
N1010 M300 ;	N2010 M300 ;	Waits for superimposed control to begin.
N1020 M55 ;		Begins superimposed control in which commands for the Z1–axis are superimposed on those for the Z2–axis.
N1030 M301 ;	N2030 M301 ;	Superimposed control has begun.
	N2040 T0414 ;	Selects a tool for superimposed control and sets the offset.
N1050 S1000 M3 ;		Performs machining with turrets 1 and 2.
N1060 G0 X20. Z15. ;	N2060 G0 X18. Z120. ;	
N1070 G1 F0. 5 W- 8. ;	N2070 G1 F0. 1 W5. ;	
N1080 ······	N2080 ······	
N1090 M302 ;	N2090 M302 ;	Waits for superimposed control to end.
N1100 M56 ;		Terminates superimposed control.
N1110 M303 ;	N2110 M303 ;	Superimposed control has ended.

N1120 ·······	N2120 ········	Machining under control independent of the other path

(5) Cautions

- The speed of spindle S1 (feedback pulses from the position coder) is specified for both paths 1 and 2.
- When using constant surface speed control, be careful about which path has the spindle command that is effective for spindle S1.

1.9.9.5 Miscellaneous

- Synchronous control and composite control were described so far. In reality, however, it is possible to perform more than one set of synchronous control and/or composite control selectively or simultaneously. For this purpose, specify all necessary parameters and select which synchronous control or composite control to be performed using the appropriate signals. However, be careful not to perform more than one set of synchronous control or composite control for one axis at one time.
- Usually, it is possible to specify only one pair of axes for synchronous control and one pair for composite control. If it is necessary to specify more than one pair, specify so in a parameter with a program, using the programmable parameter input function (G10). This must be done when the related axes are not under synchronous or composite control.

(Example) Changing the parameter so that the Z2–axis is synchronized with the Y1–axis

(To set parameter No. 8180z of path 2 to “4”, run the following program in path 2.)

N0200 ········	
N0210 G10 L50 ;	Begins parameter setting.
N0220 N8180 P2 R4 ;	Sets parameter No. 8180z to “4”.
N0230 G11 ;	Terminates parameter setting.
N0240 ········	

The blocks with G10 to G11 must be run when the Z2– or Y1–axis is not under synchronous or composite control.

1.9.10 Troubleshooting

1.Synchronous, composite, or superimposed control cannot be started, but no alarm is issued.

(1) The synchronous or composite control option has not been specified.

⇒ The synchronous and composite control must be specified.

(2) The G0128, G0138, G0190, G1128, G1138, or G1190 signal has not risen.

⇒ Synchronous, composite, or superimposed control begins on the positive–going edge of the G0128, G0138, G0190, G1128, G1138, or G1190 signal. If synchronous, composite, or superimposed control ends because of a reset or alarm, merely releasing the reset or alarm cannot restart synchronous, composite, or superimposed control. It is also necessary to raise the signal.

(3) The axis number of an axis to be subjected to synchronous, composite, or superimposed control has not been specified in a parameter.

⇒ To use synchronous control, specify the axis number of the target master axis in parameter No. 8180. To use composite control, specify the axis number of the target axis in parameter No. 8183 of path 2. To use superimposed control, specify the axis number of the target master axis in parameter No. 8186.

(4) Synchronous, composite, or superimposed control cannot be started when the NC unit is under one of the following conditions.

- Emergency stop
- Reset
- Servo alarm
- P/S000 alarm
- Alarm related to synchronous, composite, or superimposed control

In addition, synchronous, composite, or superimposed control cannot be started when the NC unit is under one of the following conditions.

- Servo–off
- Overtravel

2. The P/S225 alarm occurs when a signal for synchronous, composite, or superimposed control arises.

   (1) An attempt was made to perform synchronous, composite, or superimposed control for an axis that was already under synchronous, composite, or superimposed control.

   ⇒ It is impossible to place an axis under more than one combination of synchronous, composite, and/or superimposed control simultaneously. However, a synchronous master axis can be the master of more than one synchronous slave axis, and other slave axes can be added under the same synchronous control.

   (2) The axis number specified in a parameter is greater than the number of controllable axes.

   ⇒ The axis number of a synchronous master axis, an axis under composite control, or the master axis under superimposed control in one path must not be greater than the number of controllable axes in the other path (or in the same path if synchronous control is performed within one path).

   (3) An axis to be placed in synchronization is already moving.

   ⇒ When synchronous, composite, or superimposed control begins, the target axis must be at a stop. An axis being at a stop means that the speed that is effective after acceleration/deceleration is zero.

3. The P/S225 alarm occurs when synchronous, composite, or superimposed control ends.

   (1) An axis to be released from synchronization is moving.

   ⇒ When synchronous, composite, or superimposed control ends, the target axis must be at a stop. An axis being at a stop means that the speed that is effective after acceleration/deceleration is zero. When terminating synchronization, make sure that the axis moving signal F0102/F1102 is “0”.

Note

1 Before the P/S225 alarm that occurs when synchronous, composite, or superimposed control is terminated can be reset, it is necessary to place the machine in an emergency stop state.

4. The P/S226 alarm occurs during synchronous or composite control.

   (1) A move command was issued to a synchronous slave axis.

   ⇒ A move command (either automatic or manual) cannot be used for a synchronous slave axis.

(2) A move command was issued to an axis under composite control for which parameter MUMx (bit 7 of parameter No. 8162) is "1".

⇒ No move command (either automatic or manual) can be issued to an axis under composite control for which parameter MUMx (bit 7 of parameter No. 8162) is "1".

5. Servo alarm No. 407 occurs during synchronous control.

(1) There is an excessive difference in the positional deviation between the synchronous master and slave axes.

⇒ Alarm SV407 can occur only when a check is being made for synchronous error. This alarm occurs typically when there is a large difference in acceleration/deceleration constants or servo parameters between the synchronous master and slave axes or when the actual machine movement is incorrect for any reason (such as incorrect synchronization).

6. The machine position deviates during synchronous control.

(1) The acceleration/deceleration constants or servo parameters (such as loop gain) do not match between the master and slave axes.

⇒ During synchronous control, acceleration/deceleration and servo control are performed for master and slave axes separately. (Instead, move commands are placed in synchronization.) Acceleration/deceleration time constants or servo characteristics may vary between the master and slave axes. In such cases, the actual machine movement does not match between axes.

7. The amount of movement is incorrect during synchronous, composite, or superimposed control.

(1) The diameter/radius specification or inch/metric input setting does match between the master and slave axes.

⇒ Synchronous, composite, superimposed control does not make conversion on diameter/radius or inch/metric input specifications between the master and slave axes. The least command input must match between the master and slave axes.

8. An axis does not move to a specified position after synchronous or composite control switching.

   (1) A move command was issued within two blocks after synchronous or composite control.

   ⇒ The coordinate system in the CNC must be re–set at synchronous or composite control switching. No move command can be issued to an axis subjected to synchronous or composite control during automatic operation within two (or three for tool–tip radius compensation) blocks (including the current one) after synchronous or composite control switching. However, this restriction does not apply when the current block is an M code that does not buffer the next block or when the target axis is a synchronous master axis.

9. Synchronous, composite, or superimposed control was terminated when the G0128, G0138, G0190, G1128, G1138, or G1190 did not drop.

   ⇒ Synchronous, composite, or superimposed control is terminated automatically, if one of the following conditions occurs in either path.

   - Emergency stop
   - Reset
   - Servo alarm
   - P/S000 alarm
   - Alarm related to synchronous, composite, or superimposed control

   In addition, synchronous, composite, or superimposed control is terminated for all axes, if one of the following conditions occurs for any axis under synchronous, composite, or superimposed control.

   - Servo–off
   - Overtravel

2 PREPARATIONS FOR OPERATION

2.1 EMERGENCY STOP

General

If you press Emergency Stop button on the machine operator's panel, the machine movement stops in a moment.

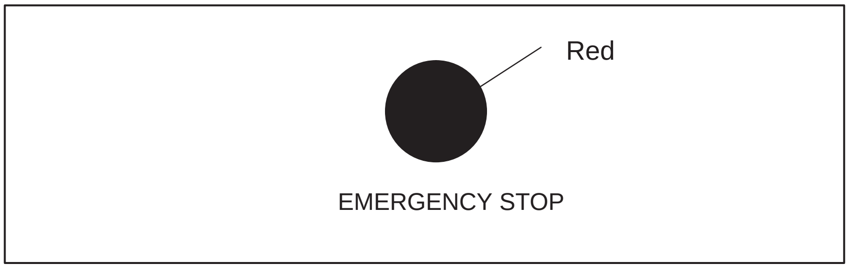


Fig. 2.1 (a) EMERGENCY STOP

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 *ESP<X008#4,G008#4>

[Classification] Input signal

[Function] Outputting 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 software limit 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) below).

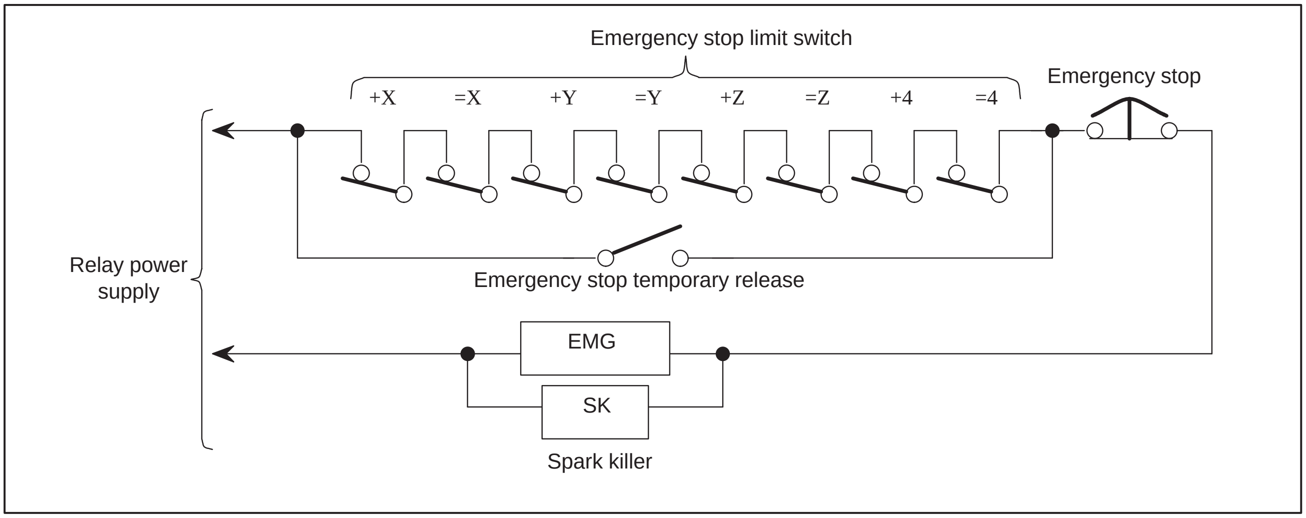


Fig. 2.1 (b) Connection of Emergency Stop Limit Switch

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 Descriptions.”

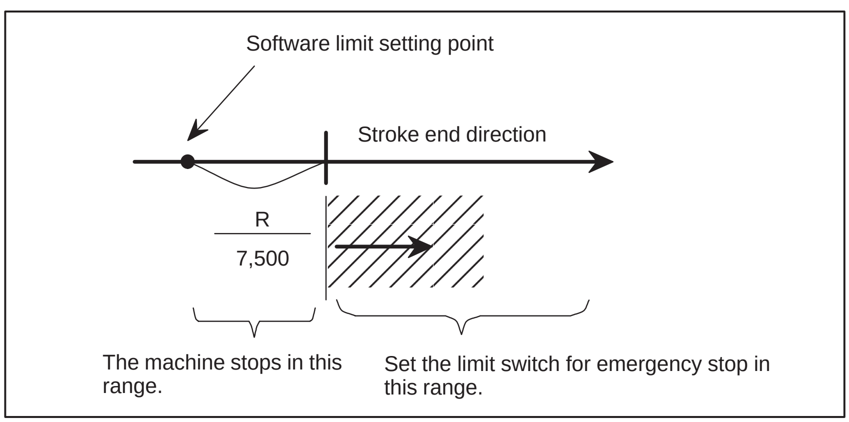
Note 1 Software limit setting point and operating point of limit switch for emergency stop

The stop point by the software limit goes beyond the setting point by as much as the following distance.

$$\frac{R}{7,500} \text{ (mm)}$$

R: Rapid traverse rate (mm/min)

The actual stopping point may exceed the position set by a parameter (Nos.1320 and 1321) by as much as R/7500 (mm). Set the limit switch for emergency stop including the allowance for the above value.



	#7	#6	#5	#4	#3	#2	#1	#0
X008				*ESP				

	#7	#6	#5	#4	#3	#2	#1	#0
G008				*ESP				

Reference item

AC SERVO MOTOR series DESCRIPTIONS	B–65002E
AC SERVO MOTOR α series DESCRIPTIONS	B–65142E

a

2.2 CNC READY SIGNAL

General

When the CNC is turned on and becomes ready for operation, the CNC ready signal is set to 1.

Signal

CNC Ready Signal MA<F001#7>

[Classification] Output signal

[Function] The CNC ready signal reports that the CNC is ready.

[Output condition] 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.

	#7	#6	#5	#4	#3	#2	#1	#0
F001	MA							

Servo Ready Signal SA <F000#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 sent and apply the brake when this signal is not sent.
Time chart of this signal is as follows:

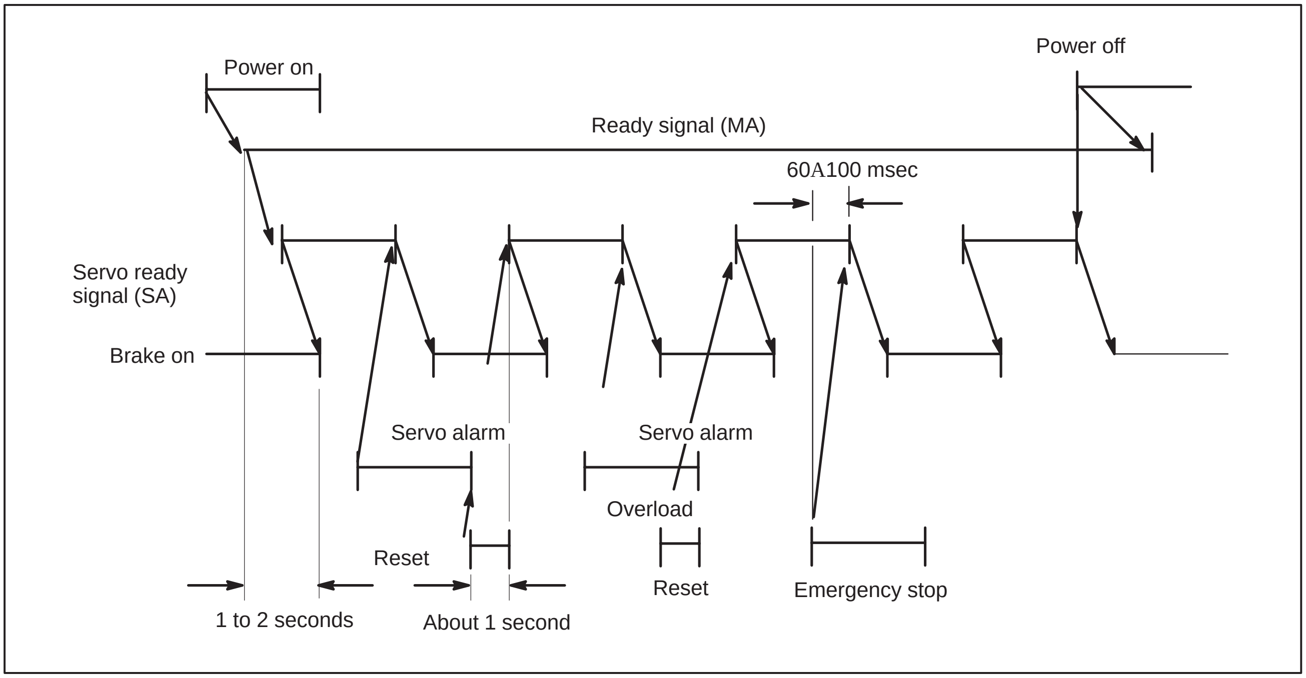


Fig.2.2 Time Chart for Servo Ready Signal

	#7	#6	#5	#4	#3	#2	#1	#0
F000		SA						

2.3 OVERTRAVEL CHECK

2.3.1 Overtravel Signal

General

When the tool tries to move beyond the stroke end set by the machine tool limit switch, the tool decelerates and stops because of working the limit switch and an OVER TRAVEL is displayed.

Signal

Overtravel signal
*+L1 to *+L8<G114>
*+L1 to *–L8<G116>

[Classification] Input signal

[Function] Indicates that the control axis has reached its stroke limit. There are signals for every direction in every control axis. The +/– in the signal name indicates the direction and the number corresponds to the control axis.

+ L 1

1 No. 1 axis is at stroke limit.
2 No. 2 axis is at stroke limit.
3 No. 3 axis is at stroke limit.
: :
8 No. 8 axis is at stroke limit.

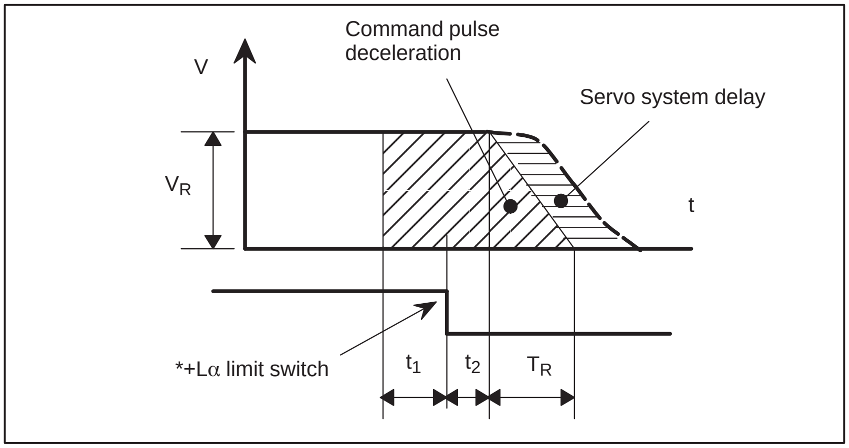
+ Has moved to and reached a + direction.
– Has moved to and reached a – direction.

[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 movement 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



$$L_1=V_R(t_1+t_2+\frac{T_R}{2}+T_S)\cdot\frac{1}{60000}\ \text{[mm or inch]}$$

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 *+Lα signal turn off (ms))

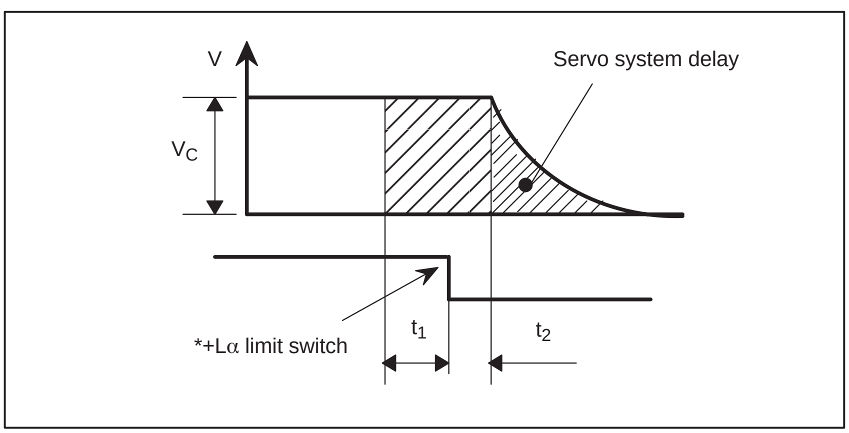
t_2: Receiver delay time 30ms

T_R: Rapid traverse acceleration/deceleration time constant (ms)

T_S: Servo system time constant (ms)

Note
Servo system time constant TS is 33 msec when the servo unit is adjusted to the standard setting.

ii) Cutting feed



$$L_2=V_C(t_1+t_2+\frac{T_R}{2}+T_S)\cdot\frac{1}{60000}\ \text{[mm or inch]}$$

L_2:Deceleration distance

Vc: Maximum feedrate (mm/min or inch/min)

t_1, t_2, Ts: Same as (i).

- **Releasing overtravel**

Press the reset button to reset the alarm after moving the tool to the safety direction by manual operation. For details on operation, refer to the operator's manual of the machine tool builder.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G114	*+L8	*+L7	*+L6	*+L5	*+L4	*+L3	*+L2	*+L1

G116	*–L8	*–L7	*–L6	*–L5	*–L4	*–L3	*–L2	*–L1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3004			OTH					

[Data type] Bit

OTH The overtravel limit signal is:
0 : Checked
1 : Not checked

Note 1 For safety, usually set 0 to check the overtravel limit signal.

Alarm and message

Number	Message	Description
506	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side hardware OT.
507	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side hardware OT.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.6.2	Overtravel
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.6.2	Overtravel

2.3.2 Stored Stroke Limit 1

General

When the tool exceeds a stored stroke limit, 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.

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.

Signal

Stored stroke limit select signal EXLM <G007#6>

[Classification] Input signal

[Function] Selects stroke limit 1–I (parameter Nos. 1320 and 1321) or stroke limit 1–II (parameter Nos. 1326 and 1327).

[Operation] When this signal is set to 1, the control unit operates as follows:
– Checks stroke limit 1 on the basis of parameter Nos. 1326 and 1327, instead of parameter Nos, 1320 and 1321.

Stroke limit external setting signals +LM1 to +LM8 <G110> and –LM1 to –LM8 <G112> (M series)

[Classification] Input signal

[Function] Change the values of the parameters governing the software limit (1320 and 1321).

[Operation] When these signals are set to 1, the control unit operates as follows:
– Change the software limits, set with parameter Nos. 1320 and 1321, to the machine coordinates when the signals are input.

Stroke limit release signal RLSOT <G007#7> (M series)

[Classification] Input signal

[Function] Selects whether the software limits are checked.

[Operation] When this signal is set to 1, the control unit operates as follows:
– Does not check the software limits.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G007	RLSOT	EXLM						

	#7	#6	#5	#4	#3	#2	#1	#0
G110	+LM8	+LM7	+LM6	+LM5	+LM4	+LM3	+LM2	+LM1

	#7	#6	#5	#4	#3	#2	#1	#0
G112	–LM8	–LM7	–LM6	–LM5	–LM4	–LM3	–LM2	–LM1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA	LZR				LMS		
	BFA	LZR				LMS		

[Data type] Bit

LMS The EXLM signal for switching stored stroke limit 1
0: Disabled
1: Enabled

LZR Checking of stored stroke limit 1 during the time from power–on to the manual position reference return
0: The stroke limit 1 is checked.
1: The stroke limit 1 is not checked

BFA When a command that exceeds a stored stroke limit is issued
0: An alarm is generated after the stroke limit is exceeded.
1: An alarm is generated before the stroke limit is exceeded.

1320	Coordinate value I of stored stroke limit 1 in the positive direction on each axis
1321	Coordinate value I of stored stroke limit 1 in the negative direction each axis

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] – 99999999 to 99999999

The coordinate values of stored stroke limits 1 in the positive and negative directions are set for each axis in the machine coordinate system. The outside area of the two limits set in the parameters is inhibited.

Note 1 For axes with diameter specification, a diameter value must be set.

Note 2 When the parameters are set as follows, the stroke becomes infinite:
parameter 1320 < parameter 1321

For movement along the axis for which infinite stroke is set, only incremental commands are available. If an absolute command is issued for this axis, the absolute register may overflow, and normal movement will not result.

1326	Coordinate value II of stored stroke limit 1 in the positive direction on each axis
1327	Coordinate value II of stored stroke limit 1 in the negative direction each axis

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] – 99999999 to 99999999

Set the coordinate values of stored stroke limits 1 in the positive and negative directions for each axis in the machine coordinate system.

When stroke limit switching signal EXLM is ON, stroke limits are checked with parameters 1326 and 1327, not with parameters 1320 and 1321. The area outside that set by parameters 1326 and 1327 is inhibited.

Note 1 The EXLM signal is enabled only when LMS, #2 of parameter 1300, is set to 1.

Alarm and message

Number	Message	Description
500	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side stored stroke limit I.
501	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side stored stroke limit I.

Note

Note 1 In setting a forbidden area, if two points to be set are the same, all area is forbidden in limit 1.

Note 2 Parameter LZR (bit 6 of No. 1300) selects whether each limit becomes effective after the power is turned on and manual reference position return or automatic reference position return by G28 has been performed or immediately after the power is turned on. (Only for M series)

Note 3 For the 2–path control, set a forbidden area for each tool post.

Note 4 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.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.6.3	Stroke check
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.6.3	Stroke check

2.3.3
Stored Stroke Limit 2, 3

General

Three areas which the tool cannot enter can be specified with stored stroke limit 1, stored stroke limit 2,and stored stroke limit 3.
(Stored stroke limit 3 is only for T series)

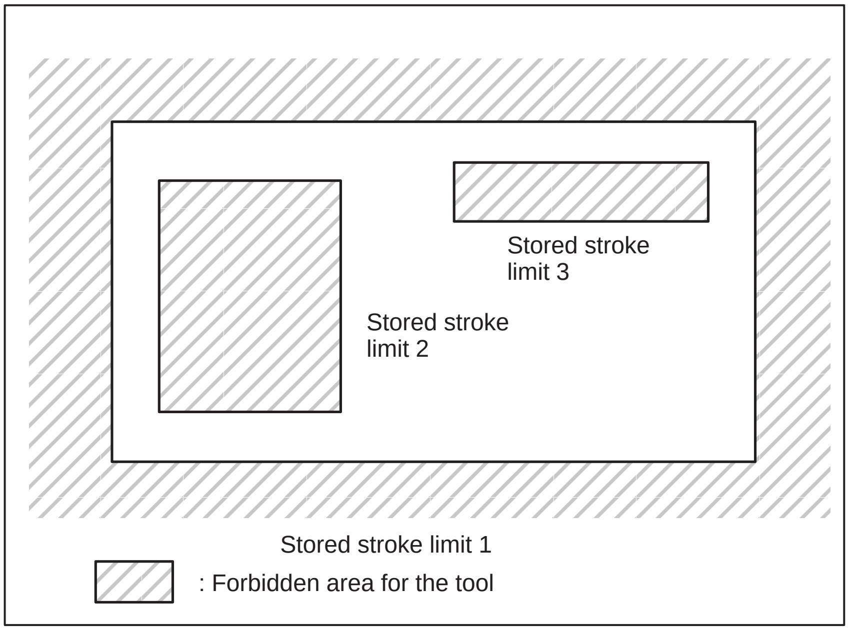


Fig. 2.3.3 (a) Stroke check (T series)

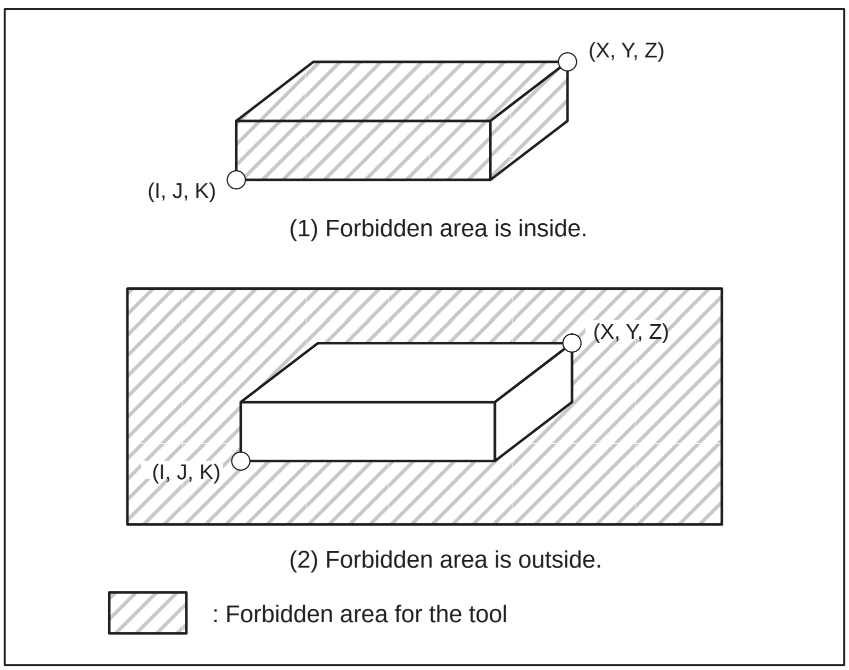


Fig. 2.3.3 (b) Stroke check (M series)

When the tool exceeds a stored stroke limit, 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.

Stored stroke limit 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:

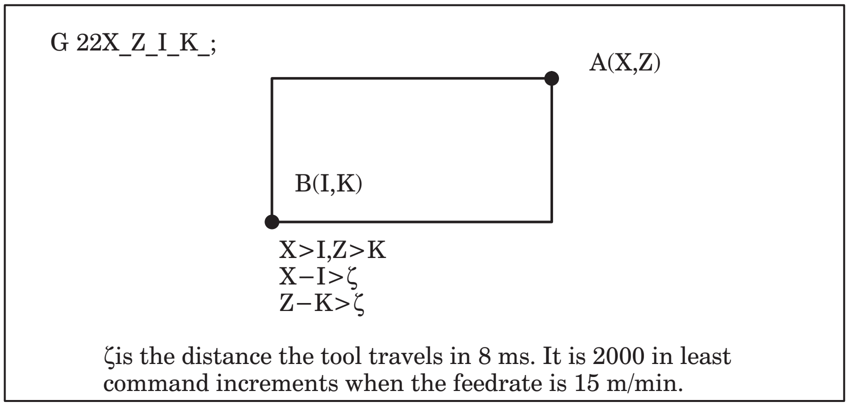


Fig. 2.3.3 (c) Creating or changing the forbidden area using a program (T series)

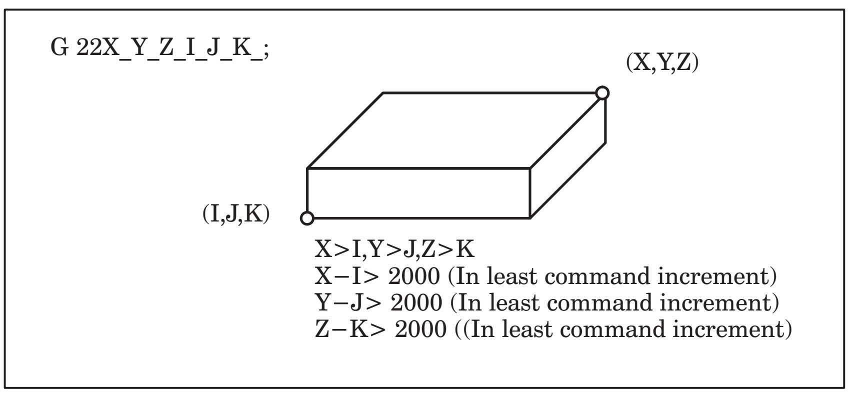


Fig. 2.3.3 (d) Creating or changing the forbidden area using a program (M series)

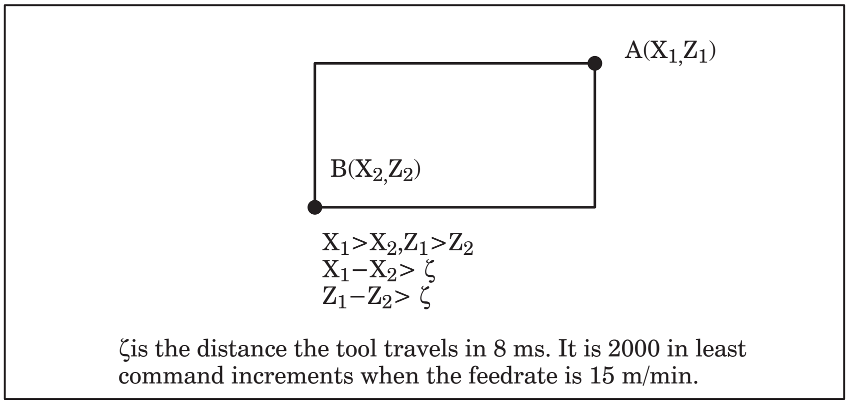


Fig. 2.3.3 (e) Creating or changing the forbidden area using a parameters (T series)

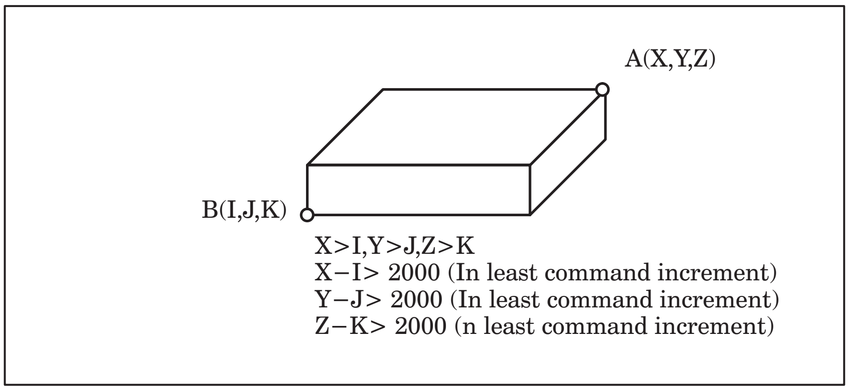


Fig. 2.3.3 (f) Creating or changing the forbidden area using a parameters (M series)

In limit 2, even if you mistake the order of the coordinate value of the two points, a rectangular, with the two points being the apexes, will be set as the area.
When you set the forbidden area X_1,Z_1,X_2,and Z_2 through parameters (Nos. 1322, 1323), the data should be specified by the distance from the reference position in the least command increment (output increment).
If set the forbidden area XZIK by a G22 command, specify the data by the distance from the reference position in the least input increment (input increment). The programmed data are then converted into the numerical values in the least command increment, and the values are set as the parameters.

- **Stored stroke limit 3**

Set the boundary with parameters Nos. 1324 and 1325. The area inside the boundary becomes the forbidden area (Only for T series).

- **Checkpoint for the forbidden area**

The parameter setting or programmed value (XZIK) depends on which part of the tool or tool holder is checked for entering the forbidden area. Confirm the checking position (the top of the tool or the tool chuck) before programming the forbidden area.
If point C (The top of the tool) is checked in Fig. 2.3.3 (g), the distance “c” should be set as the data for the stored stroke limit function. If point D (The tool chuck) is checked, the distance “d” must be set.

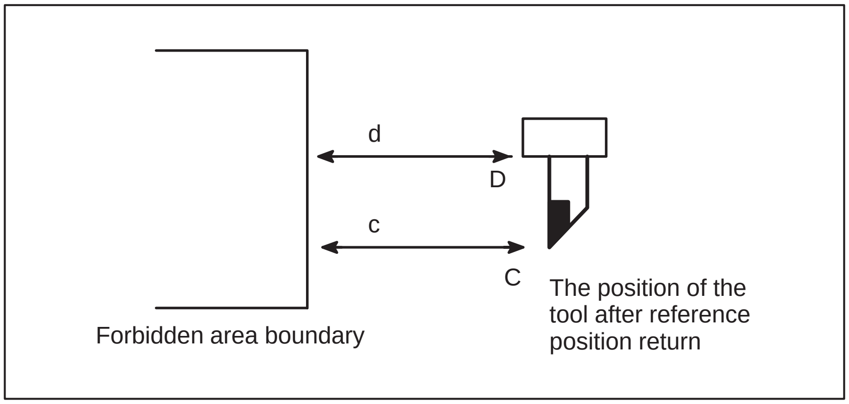


Fig. 2.3.3 (g) Setting the forbidden area (T series)

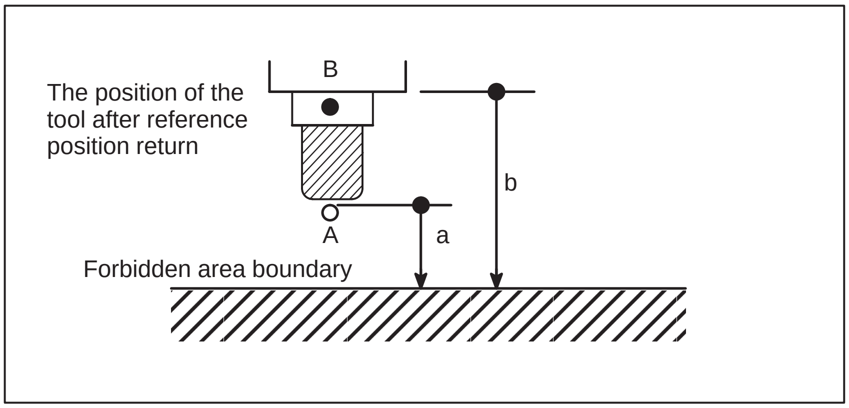


Fig. 2.3.3 (h) Setting the forbidden area (M series)

- **Forbidden area over–lapping**

Area can be set in piles.

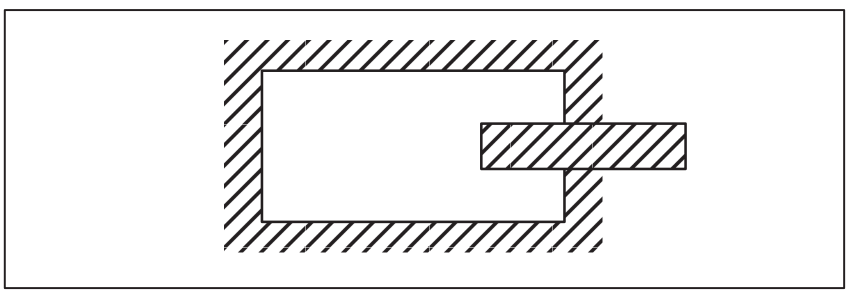


Fig. 2.3.3 (i) Setting the forbidden area overlapping (T series)

Unnecessary limits should be set beyond the machine stroke.

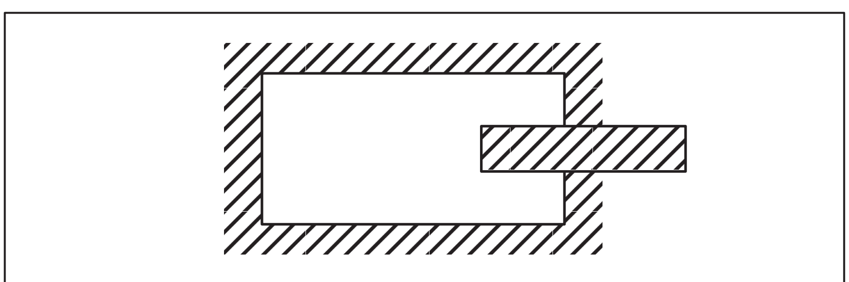


Fig. 2.3.3 (j) Setting the forbidden area overlapping (M series)

- **Effective time for a forbidden area**

Parameter LZR (bit 6 of No. 1300) selects whether each limit becomes effective after the power is turned on and manual reference position return or automatic reference position return by G28 has been performed or immediately after the power is turned on (Only for M series).
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 limit 2).

- **Releasing the alarms**

When the tool has become unmovable in the forbidden area, push the emergency stop button to release the forbidden condition and move the tool out of the forbidden area in the G23 mode; then, if the setting is wrong, correct it and perform the reference position return again.

- **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.

- **Creating the forbidden area for the 2–path control**

For the 2–path control, set a forbidden area for each tool post.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA	LZR						OUT
	BFA	LZR						OUT

[Data type] Bit

OUT The area inside or outside of the stored stroke limit 2 is set as an inhibition area.
0: Inside
1: Outside

LZR Checking of stored stroke limit 1 during the time from power–on to the manual position reference return
0: The stroke limit 1 is checked.
1: The stroke limit 1 is not checked

BFA When a command that exceeds a stored stroke limit is issued
0: An alarm is generated after the stroke limit is exceeded.
1: An alarm is generated before the stroke limit is exceeded.

	#7	#6	#5	#4	#3	#2	#1	#0
1310							OT3x	OT2x
								OT2x

[Data type] Bit axis

OT2x Whether stored stroke limit 2 is checked for each axis is set.
0: Stored stroke limit 2 is not checked.
1: Stored stroke limit 2 is checked.

OT3x Whether stored stroke limit 3 is checked for each axis is set.
0: Stored stroke limit 3 is not checked.
1: Stored stroke limit 3 is checked.

1322	Coordinate value of stored stroke limit 2 in the positive direction on each axis
1323	Coordinate value of stored stroke limit 2 in the negative direction on each axis

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] – 99999999 to 99999999

Set the coordinate values of stored stroke limits 2 in the positive and negative directions for each axis in the machine coordinate system. OUT, #0 of parameter 1300, sets either the area outside or the area inside specified by two limits as the inhibition area.

Note 1 For axes with diameter specification, a diameter value must be set.

1324	Coordinate value of stored stored limit 3 in the positive direction on each axis
1325	Coordinate value of stored stroke limit 3 in the negatice direction on each axis

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] – 99999999 to 99999999

Set the coordinate values of stored stroke limits 3 in the positive and negative directions for each axis in the machine coordinate system. The area inside the limits set in the parameters is forbidden.

Alarm and message

Number	Message	Description
502	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side stored stroke limit II. (Parameter No.1322)
503	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side stored stroke limit II. (Parameter No.1323)
504	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side stored stroke limit III. (Parameter No.1324)
505	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side stored stroke limit III. (Parameter No.1325)

Note 1 Over travel alarms No. 504 and No. 505 are provided only with the T series.

Note

Note 1 In setting a forbidden area, if the two points to be set are the same, the area is as follows:
(1)When the forbidden area is limit 1, all areas are forbidden areas.
(2)When the forbidden area is limit 2 or limit 3, all areas are movable areas.

Note 2 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.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.6.3	Stroke check
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.6.3	Stroke check

2.3.4 Chuck/Tailstock Barrier (T series)

General

The chuck/tailstock barrier function prevents damage to the machine by checking whether the tool tip interferes either the chuck or tailstock.
Specify an area into which the tool may not enter (entry–inhibition area). This is done using the special setting screen, according to the shapes of the chuck and tailstock. 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 cleared from the area only by retracting it in the direction opposite to that in which the tool entered the area.

Signal

Tailstock barrier select signal *TSB <G060#7>

[Classification] Input signal

[Function] Enables or disables the tailstock barrier.

[Operation] When this signal is set to 1, the control unit operates as follows:

– Disables the tailstock barrier, even when the G22 command (stored stroke limit on) is specified in the program.

G code	*TSB	Tailstock barrier	Chuck barrier
G22	0	Enabled	Enabled
	1	Disabled	Enabled
G23	0	Disabled	Disabled
	1	Disabled	Disabled

When the G23 command (stored stroke limit off) is specified, the tailstock barrier is disabled regardless of the *TSB signal. When the G22 command (stored stroke limit on) is specified, the tailstock can be disabled by setting the signal to 1.

This signal is used to select whether the tailstock area is forbidden, when machining a workpiece by attaching and detaching the tailstock to and from the workpiece according to M commands.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G060	*TSB							

Parameter

● Profile of a chuck

1330	Profile of a chuck

[Data type] Byte

[Valid data range] 0 or 1

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)
1332	Dimensions of the claw of a chuck (W)
1333	Dimensions of the part of a claw at which a workpiece is held (L1)
1334	Dimensions of the part of a claw at which a workpiece is held (W1)

[Data type] Two–word

[Unit of data]

Increment system	IS–B	IS–C	Unit
Metric input	0.001	0.0001	mm
Inch input	0.0001	0.00001	inch

[Valid data range] – 99999999 to 99999999

1335	X coordinate of a chuck (CX)
1336	ZX coordinate of a chuck (CZ)

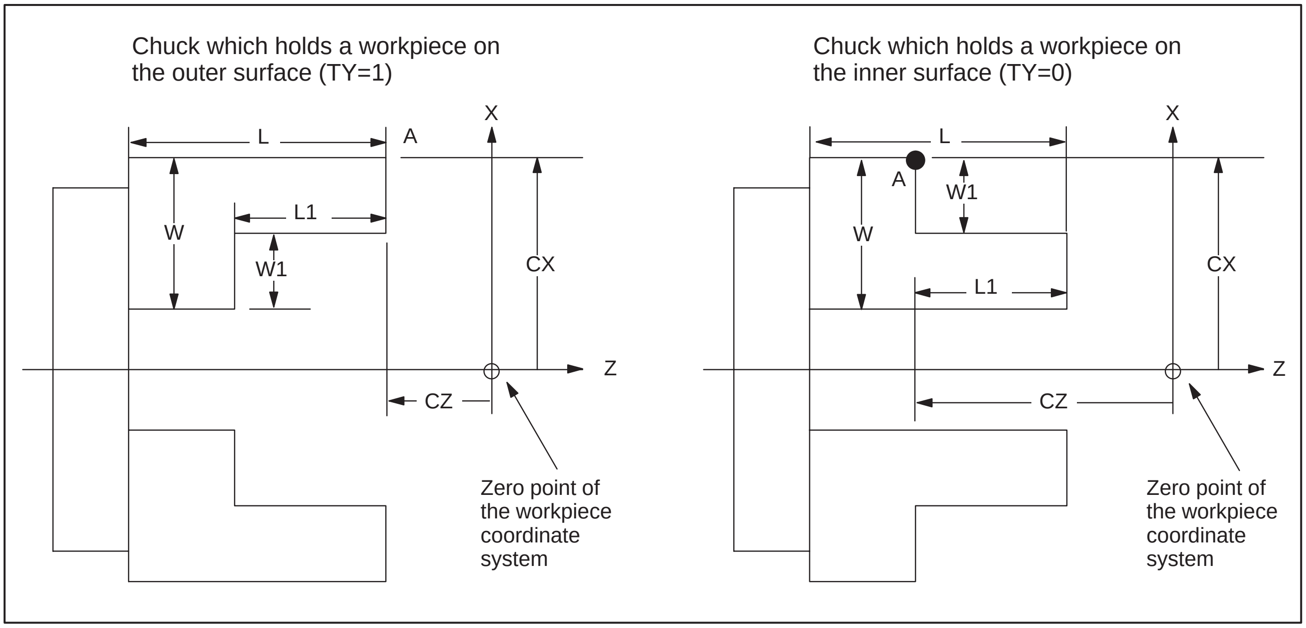
[Data type] Two–word

[Unit of data]

Increment system	IS–B	IS–C	Unit
Metric input	0.001	0.0001	mm
Inch input	0.0001	0.00001	inch

[Valid data range] – 99999999 to 99999999

Specify the profile of a chuck.



Symbol	Description
TY	Profile of a chuck (0: Chuck which holds a workpiece on the inner surface, 1: Chuck which holds a workpiece on the outer surface)
CX	X coordinate of a chuck
CZ	Z coordinate of a chuck
L	Dimensions of the claw of a chuck
W	Dimensions of the claw of a chuck (radius input)
L1	Dimensions of the part of a claw at which a workpiece is held
W1	Dimensions of the part of a claw at which a workpiece is held (radius input)

TY: Specifies the profile of a chuck. When TY is set to 0, the chuck holding a workpiece on the inner surface is specified. When TY is set to 1, the chuck holding a workpiece on the outer surface is specified. The profile of the chuck is assumed to be symmetrical with respect to the z–axis.

CX, and CZ: Specify the position (point A) of a chuck with the coordinates of the workpiece coordinate system. In this case, do not use the coordinates of the machine coordinate system.

Note 1 Specifying the coordinates with a diameter or radius depends on whether the corresponding axis conforms to diameter or radius specification . When the axis conforms to diameter specification, specify the coordinates with a diameter.

L, L1, W, and W1: Define the profile of a chuck.

Note 1 Always specify W and W1 with radiuses. Specify L and L1 with radiuses when the Z–axis conforms to radius specification.

1341	Length of a tailstock (L)

1342	Diameter of a tail stock (D)

1343	Length of a tailstock (L1)

1344	Diameter of a tail stock (D1)

1345	Length of a tailstock (L2)

1346	Diameter of a tail stock (D2)

1347	Diameter of the hole of a tailstock (D3)

[Data type] Two–word

Increment system	IS–B	IS–C	Unit
Millimeter machine	0.001	0.0001	mm
Inch machine	0.0001	0.00001	inch

[Valid data range] 0 to 99999999

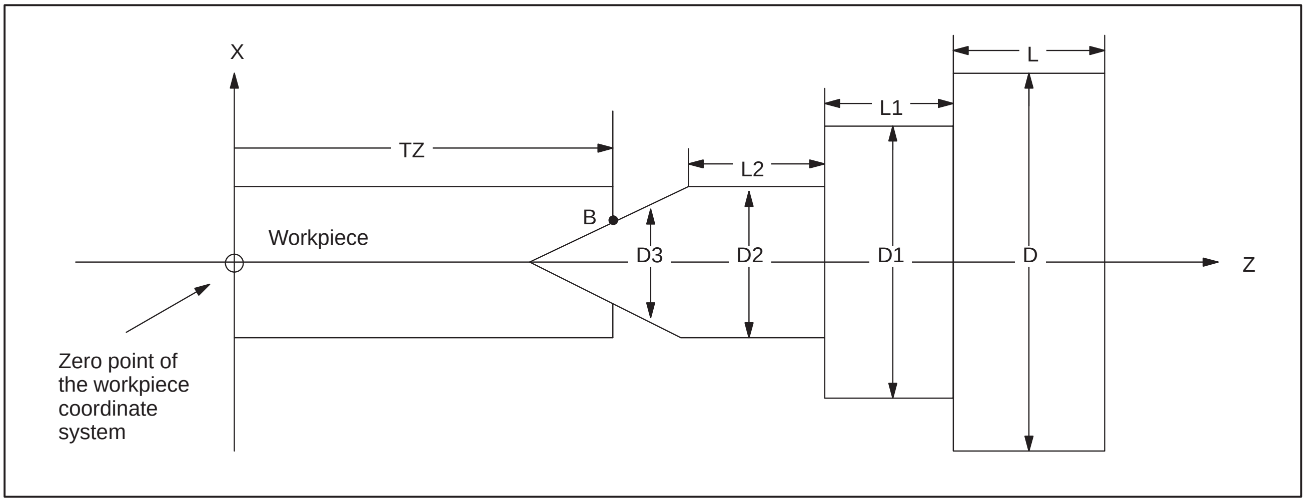
1348	Z coordinate of a tailstock (TZ)

[Data type] Two–word

Increment system	IS–B	IS–C	Unit
Millimeter machine	0.001	0.0001	mm
Inch machine	0.0001	0.00001	inch

[Valid data range] –99999999 to 99999999

Specify the profile of a tailstock.



Symbol	Description
TZ	Z–axis coordinate of a tailstock
L	Length of a tailstock
D	Diameter of a tailstock (diameter input)
L1	Length of a tailstock (1)
D1	Diameter of a tailstock (1) (diameter input)
L2	Length of a tailstock (2)
D2	Diameter of a tailstock (2) (diameter input)
D3	Diameter of the hole of a tailstock (diameter input)

TZ: Specifies the position (point B) of a tailstock with the Z–axis coordinate of the workpiece coordinate system. In this case, do not use the coordinate of the machine coordinate system. The profile of a tailstock is assumed to be symmetrical with respect to the Z–axis.

Note 1 Specifying the position of a tailstock with a radius or diameter depends on whether the Z–axis conforms to radius or diameter specification.

L, L1, L2, D, D1, D2, and D3:
Define the profile of a tailstock.

Note 1 Always specify D, D1, D2, and D3 with diameters. Specify L, L1, and L2 with radiuses if the Z–axis conforms to radius specification.

Alarm and message

Number	Message	Description
502	OVER TRAVEL : +X	The tool has entered the forbidden area when moving in the positive direction along the X–axis.
	OVER TRAVEL : +Z	The tool has entered the forbidden area when moving in the positive direction along the Z–axis.
503	OVER TRAVEL : –X	The tool has entered the forbidden area when moving in the negative direction along the X–axis.
	OVER TRAVEL : –Z	The tool has entered the forbidden area when moving in the negative direction along the Z–axis.

Note

Note 1 The area may not be forbidden if the setting for forbidding the area is invalid, as follows:

1 In the setting of the chuck shape, the jaw length (parameter No. 1331) is less than the grasp length (parameter No. 1333) or the jaw width (parameter No. 1332) is less than the grasp width (parameter No. 1334).

2 In the setting of the tailstock shape, the tailstock diameter (parameter No. 1346) is less than the hole diameter (parameter No. 1347).

3 The chuck and tailstock overlap.

Note 2 When the options for stored stroke limit 2 and chuck/tailstock barrier are used at the same time, the chuck/tailstock barrier is valid but stored stroke limit 2 is ignored.

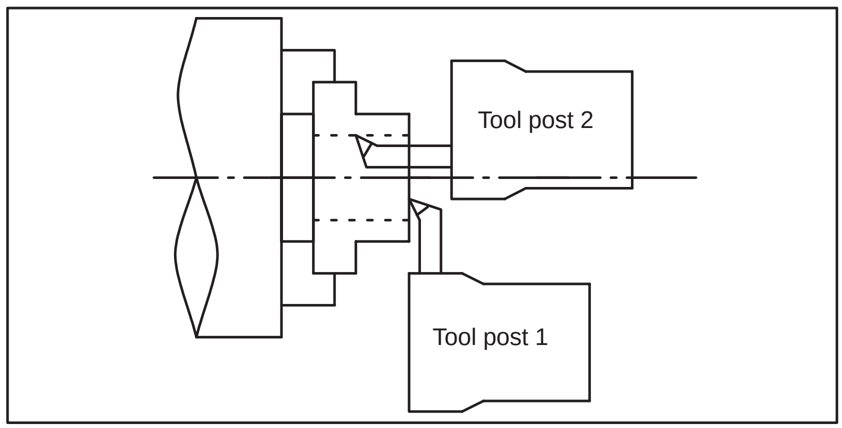
Reference item

OPERATOR’S MANUAL (For Lathe) (B–62444E)	III.6.4	CHUCK AND TAILSTOCK BARRIERS

2.3.5 Tool Post Interference Check (T series, Two–path Control)

General

When two tool posts machine the same workpiece simultaneously, the tool posts can approach each other very closely. If the two 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 “tool post interference check” is available which can decelerate and stop the two tool posts before the tool posts interfere with each other due to an incorrect command.



The contours of the two tool posts are checked to determine whether or not an interference occurs.

Signal

Tool post interference check signal TICHK <F064#6>

[Classification] Output signal

[Function] Indicates whether the tool post interference check function is being performed.

[Output condition] This signal goes “1” when:

(i) All requisites for the tool post interference check function are satisfied.

This signal goes “0” when:

(i) Any of the requisites for the tool post interference check function are not satisfied.

Note 1 For details of the requisites for the tool post interference check function, refer to the operator’s manual of FANUC Series 16–TB (B–62444E).

Tool post interference alarm signal TIALM <F064#7>

[Classification] Output signal

[Function] Indicates that the tool post interference alarm is activated.

[Output condition] This signal goes “1” when:

(i) The control unit judges that the two tool posts will interfere with each other during the execution of the tool post interference check function.

This signal goes “0” when:

(i) The control unit judges that the two tool posts will not interfere with each other during the execution of the tool post interference function.

(ii) When the tool post interference check function is not being performed (i.e., the TICHK signal is “0”).

Note 1 During the execution of the interference check function, if the control unit judges that the two tool posts will interfere with each other, it stops both tool posts by slowing them down, and then enters the alarm state. The CNC then sets the TIALM signal “1” to indicate that an interference alarm has occurred.

Note 2 If the interference alarm is activated, switch the operation mode to the manual mode, manually withdraw the tool posts to where they do not interfere each other, then release the alarm status by resetting the control unit.
As the result of manually withdrawing the tool posts, the TIALM signal goes “0” when the control unit judges that the tool posts are separated enough not to interfere with each other any more. When manually withdrawing the interfering tool posts, the TIALM signal is effective in identifying how far the tool posts must be separated from each other. This is because it is easy for the operator to check at which point the signal goes “0”.

Note 3 If an interference alarm occurs, the axis being moved and its moving direction are stored, and the axis cannot be moved in the stored direction until the alarm is released by resetting the control unit. This prevents the axis from interfering any further by prohibiting movement in the direction that caused the interference.

Signal address

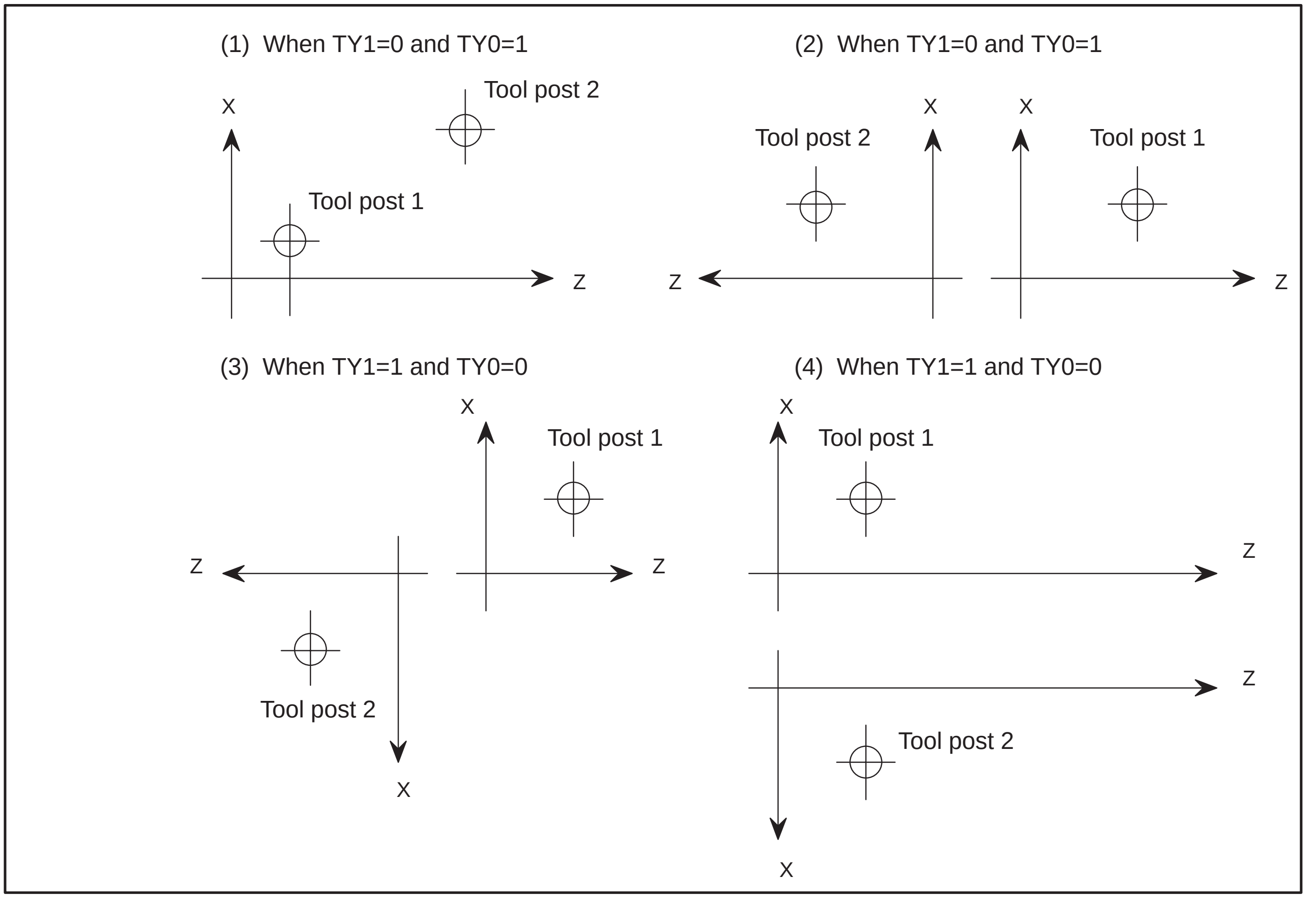
	#7	#6	#5	#4	#3	#2	#1	#0
F064	TIALM	TICHK						

Parameter

8140	#7	#6	#5	#4	#3	#2	#1	#0
			ZCL	IFE	IFM	ITO	TY1	TY0

[Data type] Bit

TY0, TY1 This parameter specifies the relationship between the coordinate systems of the two tool posts.



ITO When offset number 0 is specified by the T code,

0: Checking interference between tool posts is stopped until an offset number other than 0 is specified by the next T code.

1: Checking interference between tool posts is continued according to the previously specified offset number.

IFM Specifies whether interference between tool posts is checked in the manual operation mode.

0: Not checked

1: Checked

IFE Specifies whether interference between tool posts is checked.

0: Checked

1: Not checked

ZCL Specifies whether interference along the Z axis is checked while checking interference between tool posts.

0: Checked

1: Not checked (Only interference along the X axis is checked.)

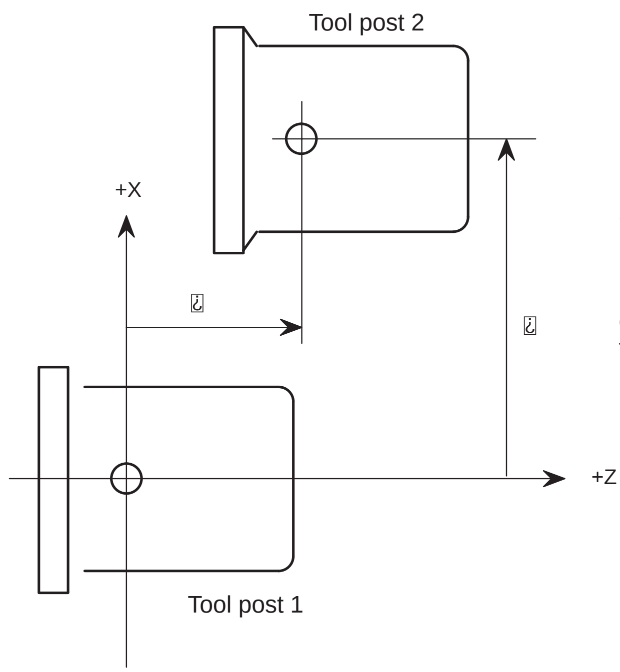
8151	Distance along the X axis between the reference positions of tool posts 1 and 2
8152	Distance along the Z axis between the reference positions of tool posts 1 and 2

[Data type] Two–word

[Unit of data]

Increment system	IS–B	IS–C	Unit
Metric input	0.001	0.0001	mm
Inch input	0.0001	0.00001	inch

[Valid data range] 0 to 99999999



Specify the X and Z values (ε, ζ) of the reference position of tool post 2 in parameters 8151 and 8152, respectively, in the Z–X plane coordinate system. However, note that the reference position of tool post 1 is plotted at the zero point.

Note 1 After the parameter values are changed, perform manual reference position return for individual tool posts. Otherwise, data on the positional relationship between the tool posts stored in memory will not be updated to the new parameter values.

Alarm and message

Number	Message	Description
169	ILLEGAL TOOL GEOMETRY DATA	Incorrect tool figure data in interference check.
508	INTERFERENCE : +X INTERFERENCE : +Z	An interference alarm has generated when X or Z axis is moving in the positive direction.
509	INTERFERENCE : –X INTERFERENCE : –Z	An interference alarm has generated when X or Z axis is moving in the negative direction.

Note

Note 1 When an alarm is raised, the CNC system and machine system stop with some delay in time. So an actual stop position can be closer to the other tool post beyond an interference forbidden position specified using tool shape data. So, for safety, tool shape data a little larger than the actual shape should be set. The extra distance, L, required for this purpose is calculated from a rapid traverse feedrate as follows

$$L = (\text{Rapid traverse rate}) \times \frac{1}{7500}$$

For example, when a rapid traverse feedrate of 15 m/min is used, L=2mm.

Note 2 When parameters or tool shape data (contact forbidden area) are set for the interference check function, check that the interference forbidden area is correctly set by moving the tool posts to foul each other in several directions in manual mode (interference check enabled with a parameter).

Reference item

OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.21.3	Tool post interference check

2.4 ALARM SIGNAL

General

When an alarm is triggered in the CNC, the alarm is indicated on the CRT 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<F001#0>

[Classification] Output signal

[Function] The alarm signal reports that the CNC is in an alarm state.

There are the following alarms. The following alarms are issued:

(a) TH alarm
(b) TV alarm
(c) P/S alarm
(d) Overtravel alarm
(e) Overheat alarm
(f) Servo alarm

[Output condition] 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<F001#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 condition] 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
F001						BAL		AL

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3111	NPA							

[Data type] Bit

NPA Action taken when an alarm is generated or when an operator message is entered

0 : The display shifts to the alarm or message screen.

1 : The display does not shift to the alarm or message screen.

2.5 START LOCK/ INTERLOCK

General

This signal disables machine movement along axes. When this signal is input during movement along axes, the tool movement is decelerated, then stopped.

Signal

Start lock signal STLK<G007#1>(T series)

[Classification] Input signal

[Function] This signal disables machine movement along axes in automatic operation (memory or MDI operation).

[Operation] When the STLK signal turns to "1", the axis movement is decelerated and stopped.
In automatic operation, blocks containing M, S, or T commands are executed consecutively until a block containing an axis move command is encountered; the system then stops and is placed into the automatic operation mode (STL is "1", SPL is "0"). When the STLK signal turns to "0", operation restarts. (Figs. 2.5 (a), (b)).

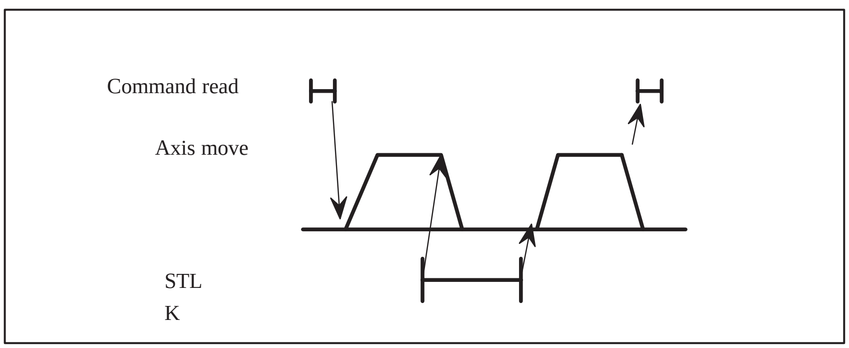


Fig. 2.5 (a) Block containing only axis move command

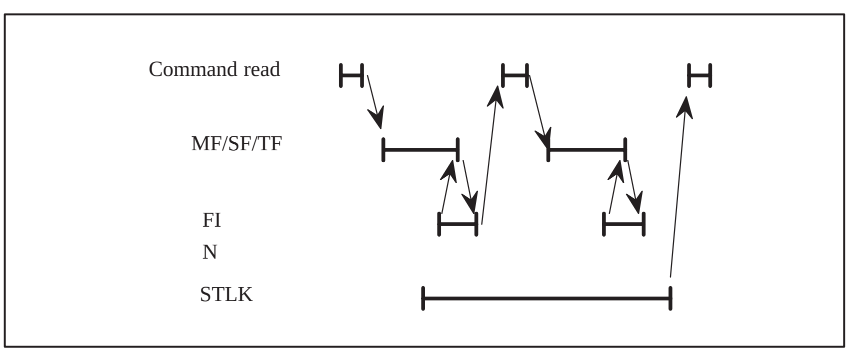


Fig. 2.5 (b) Block containing auxiliary functions only

Interlock signal
*IT<G008#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 ”0”, the axis movement is decelerated and stopped. In automatic operation, blocks containing M, S, or T commands are executed consecutively until a block containing an axis move command is encountered; the system then stops and is placed into the automatic operation mode (cycle start lamp signal STL is ”1”, feed hold lamp signal SPL is ”0”). When the *IT signal turns to ”1”, operation resumes (Figs. 2.5.2(c), (d)).

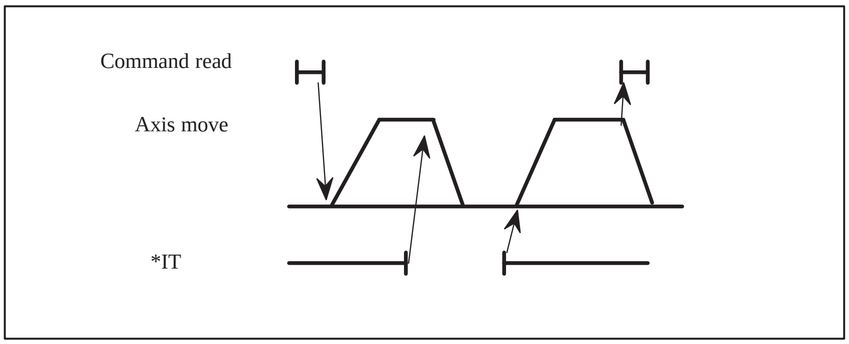


Fig. 2.5 (c) Block containing only axis move command (manual and automatic operation)

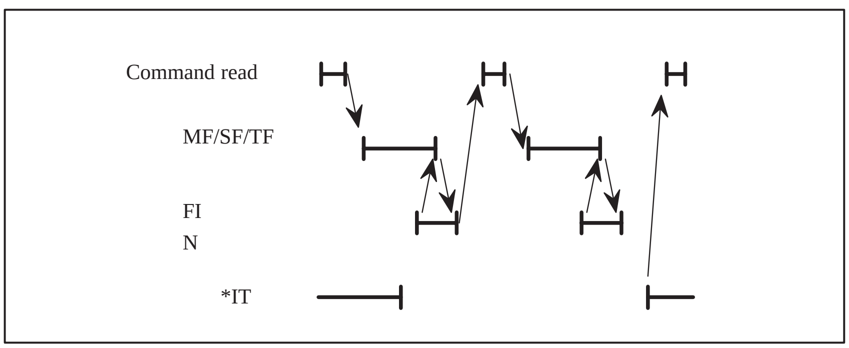


Fig. 2.5 (d) Block containing auxiliary functions only (automatic operation)

Note 1 The overtravel amount of the motor after turning *IT to “0” is represented by the following formula.

$$Q_{max} = F_m \cdot \frac{1}{60} \times \left(\frac{T_c}{1000} + \frac{T_s}{1000} + \frac{A}{1000} \right)$$

Where

Q_{max} : Overtravel quantity (mm or inch)

F_m : Feedrate (mm/min or inch/min)

T_c : Cutting time constant (ms)

T_s : Servo time constant (Ts = 33ms normally)

A : Processing time of CNC

A = 50ms with standard interlock signal
A = 16ms with high speed interlock signal

Interlock signal for each axis *IT1A*IT8<G130>

[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 1

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 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), all axes movements are prevented.

If a moving axis is interlocked, all axes stop moving after being decelerated, and they start moving again when it is released from being interlocked.

This function is effective during dry run.

Interlock signal for each axis and direction (M series) +MIT1, –MIT1, +MIT2, –MIT2, +MIT3, –MIT3, +MIT4, –MIT4 <G132#0A#3, G134#0A#3> (T series) +MIT1, –MIT1, +MIT2, –MIT2 <X004#2AX004#5>

[Classification] Input signal

[Function] This function allows a directional interlock for each axis. Together with the high-speed interlock, it is possible to release the interlock of an axis only in the direction in which no axis/directional interlock is applied.

[Operation] When the axis/directional interlock signal becomes “1”, CNC applies interlock only in the corresponding axial direction. However, during automatic operation, all axes will stop.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G007							STLK	

	#7	#6	#5	#4	#3	#2	#1	#0
G008								*IT

	#7	#6	#5	#4	#3	#2	#1	#0
G130	*IT8	*IT7	*IT6	*IT5	*IT4	*IT3	*IT2	*IT1

	#7	#6	#5	#4	#3	#2	#1	#0
G132					+MIT4	+MIT3	+MIT2	+MIT1

	#7	#6	#5	#4	#3	#2	#1	#0
G134					–MIT4	–MIT3	–MIT2	–MIT1

	#7	#6	#5	#4	#3	#2	#1	#0
X004			–MIT2	+MIT2	–MIT1	+MIT1		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3003					DIT	ITX		ITL

[Data type] Bit

ITL Interlock signal
0 : Enabled
1 : Disabled

ITX Interlock signals for each axis
0 : Enabled
1 : Disabled

DIT Interlock for each axis direction
0 : Enabled
1 : Disabled

Note

Note 1 The interlock signal for each axis and direction (T system) is supported regardless of whether the tool compensation measurement value direct input B function is provided.

2.6 MODE SELECTION

General

The mode select signal is a code signal consisting of the three bits MD1, MD2, and MD4. The seven modes -- memory edit (EDIT), memory operation (MEM), manual data input (MDI), manual handle/incremental feed (HANDLE/INC), manual continuous feed (JOG), TEACH IN JOG, and in addition, DNC operation mode can be selected by combining the (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 REF signal.
The currently selected operation mode can be posted by outputting the operation mode check signal.

Signal

Mode selection signal MDI, MD2, MD4 <G043#0A#2> DNCI <G043#5> ZRN <G043#7>

[Classification] Input signal

[Operation] As shown in the following table, the mode select signal is a grey code (a code in which only one bit is different than that of the adjacent mode). To prevent faulty mode switching, use an overcrossing contact rotary switch so that only one bit changes from that of the adjacent mode.
”Faulty mode switching” means” for example:

When the mode is switched to the EDIT mode during memory operation, the CNC enters the single block state and the operation stops at the end of the executing block.

For this mode switching, only MD2 should change from 0 to 1. If a transient signal status change occurs in a signal other than MD2 during mode switching, however, another mode (manual continuous feed mode, for example) is set between automatic operation mode and memory edit mode. When manual continuous feed mode is set while the CNC is in automatic status, the CNC immediately stops memory operation. As a result, although the operator intends to switch the mode to the memory edit mode, the CNC is, instead, placed in the feed hold state.

	Mode	Signal status				
		MD4	MD2	MD1	DNCI	ZRN
1	Memory edit (EDIT)	0	1	1	0	0
2	Memory operation (MEN)	0	0	1	0	0
3	Manual data input (MDI)	0	0	0	0	0
4	Manual handle/incremental feed (HANDLE/INC)	1	0	0	0	0
5	Manual continuous feed (JOG)	1	0	1	0	0
6	TEACH IN HANDLE	1	1	1	0	0
7	TEACH IN JOG	1	1	0	0	0
8	DNC operation (RMT)	0	0	1	1	0
9	Manual reference position return (REF)	1	0	1	0	1

Operation mode check signal
MMDI, MMEM , MRMT, MEDT, MH, MINC, MJ, MREF, MTCHIN
<F003, F004#6>

[Classification] Output signal

[Function] The currently selected operation mode is output.

[Operation] The following lists the relationship between the mode selection signals and check signals:

Mode	Input signal					Output signal
	MD1	MD2	MD4	REF	DNCI	
Manual data input (MDI)	0	0	0	0	0	MMDI<F003#3>
Memory operation (MEM)	1	0	0	0	0	MMEM<F003#5>
DNC operation(RMT)	1	0	0	0	1	MRMT<F003#4>
Memory edition(EDIT)	1	1	0	0	0	MEDT<F003#6>
Manual handle feed / Incremental feed (HANDLE/INC)	0	0	1	0	0	MH<F003#1> MINC<F003#0>
Manual continuous feed(JOG)	1	0	1	0	0	MJ<F003#2>
Manual reference position return	1	0	1	1	0	MREF<F004#6>
TEACH IN JOG	0	1	1	0	0	MTCHIN<F003#7>, MJ<F003#2>
TEACH IN HANDLE	1	1	1	0	0	MTCHIN<F003#7>, MH<F003#1>

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G043	ZRN		DNCI			MD4	MD2	MD1
F003	MTCHIN	MEDT	MMEM	MRMT	MMDI	MJ	MH	MINC
F004		MREF						

Note

Precautions on modes and mode switching

1) In the MDI mode, the STL signal turns to “0” and the CNC stops at the end of execution of the commands input from the CRT/MDI panel, but the SPL signal does not turn to “1”. Therefore, another command can be input from the manual data input unit under this state.
2) When the manual handle function is not provided, incremental feed is enabled in the HANDLE/INC mode. When the manual handle function is provided, only manual handle feed is enabled in that mode.
3) Manual operation in TEACH IN JOG and TEACH IN HANDLE mode.

a) When parameter JHD no.7100#0 is set to ”0” so that jog operation and handle feed operation are performed with separate modes:

In TEACH IN JOG mode, jog operation can be done.

In TEACH IN HANDLE mode, handle feed can be done when optional manual handle feed function is provided, and incremental feed can be done when handle feed function is not provided.

b) When parameter JHD no.7100#0 is set to “1” so that jog operation and handle feed operation are performed with the same mode:

In TEACH IN JOG mode, handle feed and jog feed can be done when optional manual handle feed function is provided, but jog feed only when it is not included.

In TEACH IN HANDLE mode, handle feed and jog feed can be done when optional manual handle feed function is provided, but incremental feed only when it is not included.

The program can be edited in both TEACH IN JOG and TEACH IN HANDLE modes.

4) When the CNC is operating in the MEM mode and is switched to MDI, the unit automatically switches to single block operation, then enters the MDI mode at the end of the executing block. The STL signal turns to “0” at this time, but the SPL signal does not turn to “1” (see Fig. 2.6 (a)).

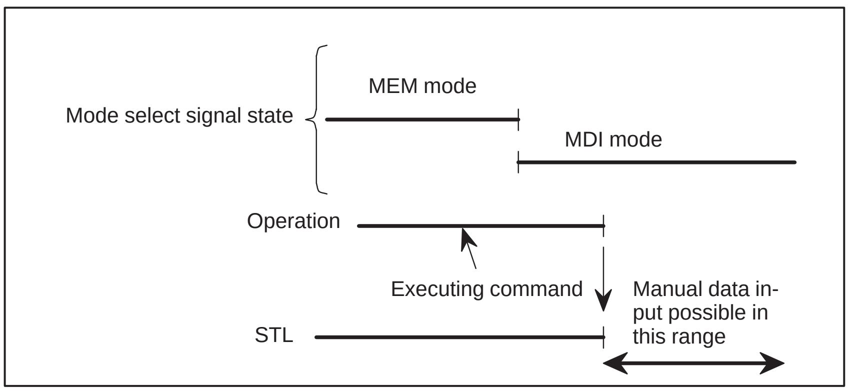


Fig. 2.6 (a)

5) When the CNC is operating in the MDI mode and is switched to MEM mode, the executing command is finished before the CNC switches to the MEM mode (Fig. 2.6 (b)).

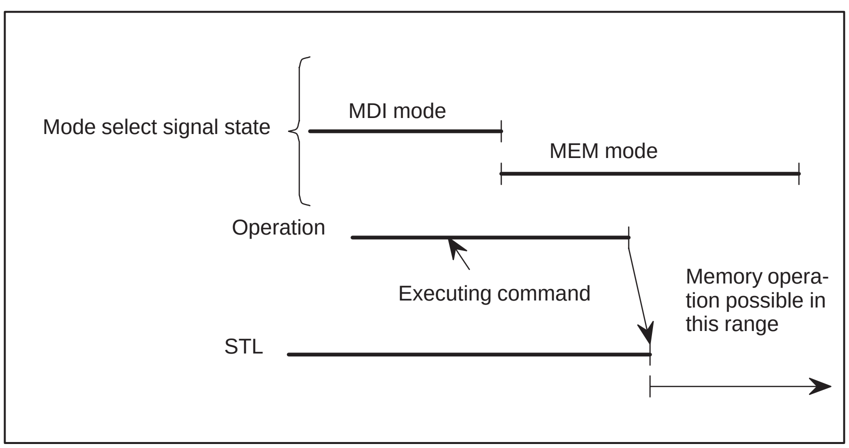


Fig. 2.6 (b)

6) When the HANDLE/INC or TEACH IN HANDLE mode is selected while the CNC is operating in the MEM 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 or TEACH IN HANDLE mode. Manual handle feed or incremental feed by axis direction select signal is possible under this state. Since the MEM mode or MDI mode commands are held, operation can be restarted by the cycle start signal by selecting the MDI or MEM mode. However, if operation was stopped by switching to the HANDLE/INC or TEACH IN HANDLE mode during manual data input or automatic operation, it can be restarted only by the mode in use before the operation was stopped (Fig. 2.6 (c)).

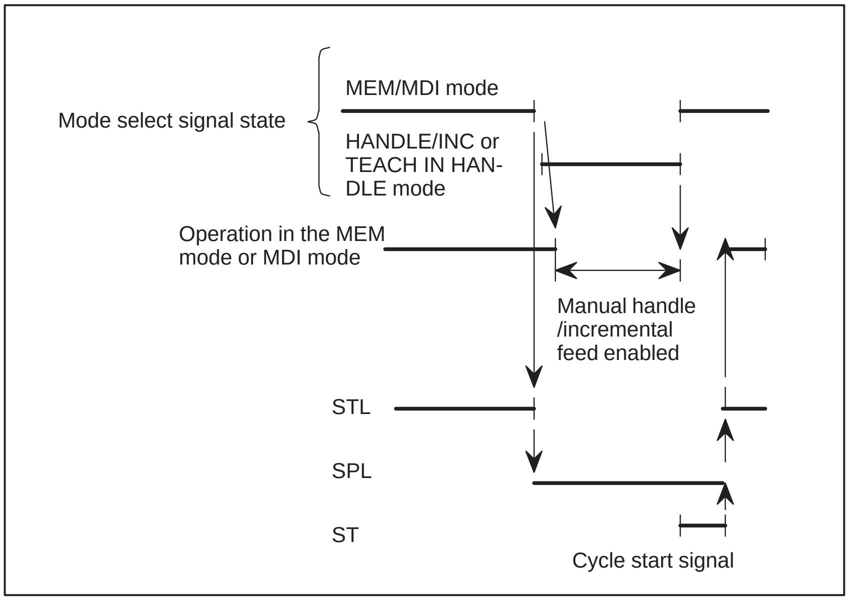


Fig. 2.6 (c)

7) When the JOG or TEACH IN JOG mode is selected during MEM 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 or TEACH IN JOG mode. Manual feed by feed axis direction select signal is possible under this state. Operation can be restarted by returning to the original state, as described for HANDLE/STEP or TEACH IN HANDLE mode . When the mode is switched to the JOG or TEACH IN JOG mode during manual handle feed or step feed operation, the CNC ignores the manual handle feed or step feed command and manual jog feed becomes effective. If a feed axis direction select signal turns to “1” before the JOG or TEACH IN 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”.
It is possible to perform handle feed in TEACH IN JOG mode by parameter THD no.7100#1. For details, refer to item (3) (Fig. 2.6 (d), (e)).

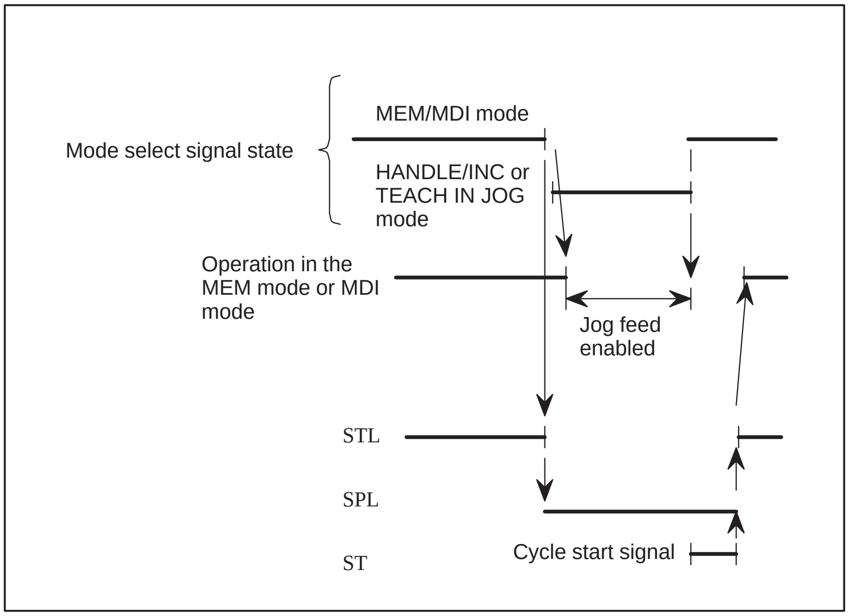


Fig. 2.6 (d)

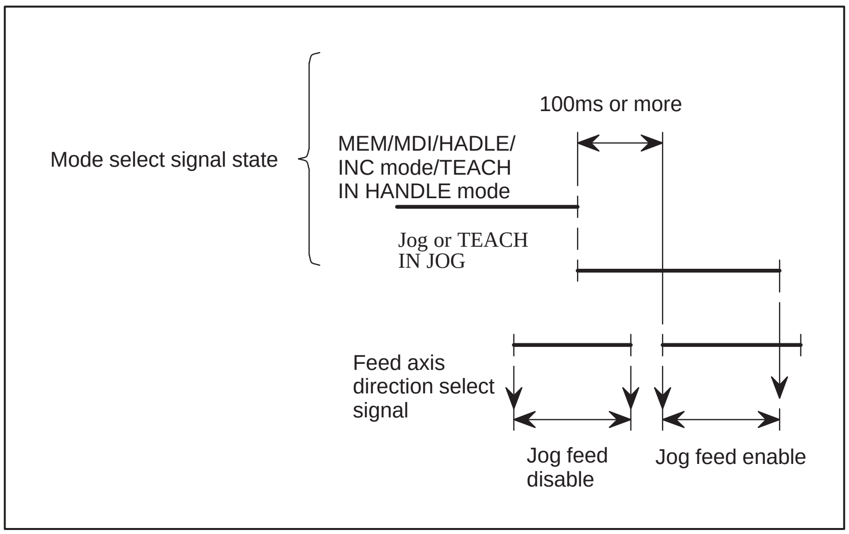


Fig. 2.6 (e)

8) The mode switching operation is summarized in the time chart below (Fig. 2.6 (f)).

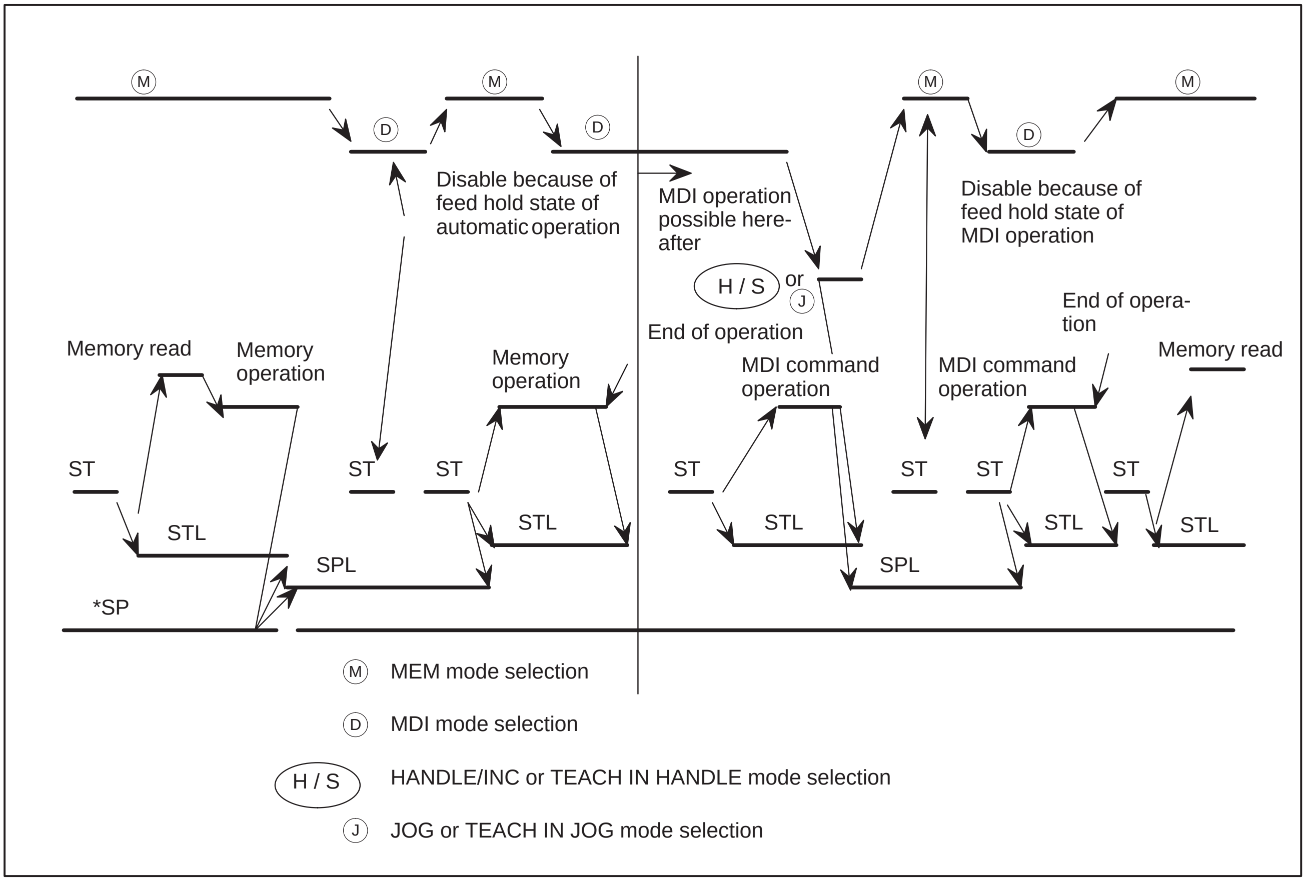


Fig. 2.6 (f) Mode signal time chart

Reference item

CONNECTION MANUAL (This manual)	4.1	Manual Reference Position Return

2.7 PATH SELECTION/ DISPLAY OF OPTIONAL PATH NAMES

General

Path selection specifies whether operations performed using the CRT/MDI panel are for path 1 or path 2.
The operations, as used here, include displaying and setting data items (such as tool compensation values), inputting command programs in the MDI mode, and editing machining programs in program memory.
Additionally, names of each path can be changed by parameter.

Signal

Path selection signal (Tool post selection signal) HEAD<G063#0>

[Classification] Input signal

[Function] Selects whether the CRT/MDI panel is used for path 1 or path 2.

[Operation] When this signal turned to “1”, operations performed using the CRT/MDI panel are for path 2.
When this signal turned to “0”, operations performed using the CRT/MDI panel are for path 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G063								HEAD

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8100							IAL	RST

[Data type] Bit

RST Reset key on the CRT/MDI panel
0 : Effective for both paths
1 : Effective for the path selected by the path select signal

IAL When an alarm is raised in one tool post in the automatic operation mode,
0 : The other path enters the feed hold state and stops.
1 : The other path continues operation without stopping.

Parameters for display of optional path names

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)

[Data type] Byte type

Specify a path name with codes (two–path control).
Any character strings consisting of alphanumeric characters and symbols (up to seven characters) can be displayed as path names on the CRT screen, instead of HEAD1 and HEAD2 for T series, and instead of PATH1 and PATH2 for M series.

Note 1 This parameter is dedicated to the two–path control.

Note 2 Specify these parameters for each series.

Note 3 For characters and codes, see the correspondence table in 2.1.15 software operator's panel.

Note 4 When codes are 0, HEAD1 and HEAD2 for T series and PATH1 or PATH2 for M series are displayed.

2.8 STATUS OUTPUT SIGNAL

General

The table below lists the status output signals for notifying 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	2.4
Battery alarm signal	BAL	2.4
Reset signal	RST	5.2
Rewinding signal	RWD	5.2
Tapping signal	TAP	11.7
Moving signal	MV1 – MV8	1.2.5
Moving direction signals	MVD1 – MVD8	1.2.5
In–position signals	INP1 – INP8	7.2.6.1
Rapid traversing signal	RPDO	2.8 (the section you are reading)
Cutting feed signal	CUT	2.8 (the section you are reading)
Thread cutting signal	THRD	6.4.1
Constant surface speed signal	CSS	9.5
Inch input signal	INCH	11.4

Signal

Rapid traversing signal RPDO <F002#1>

[Classification] Output signal

[Function] This signal indicates that a move command is executed at rapid traverse.

[Output condition] “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 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.

Note 2 Once rapid traverse has been selected, this signal remains “1”, including during a stop, until another feedrate has been selected and movement is started.

Cutting feed signal CUT <F002#6>

[Classification] Output signal

[Function] Notifies that cutting feed is being performed by automatic operation.

[Output condition] 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 canned cycle)

Note 1 This signal is output even when the feedrate override is 0%, or during interlock.

Note 2 This signal is not output in the feed hold state.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F002		CUT					RPDO	

2.9 VRDY OFF ALARM IGNORE SIGNAL

General

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<G066#0>

[Classification] Input signal

[Function] Disables the detection of servo alarm No. 401, 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 No. 401, 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 SAKx, bit 6 of parameter No. 1804.

Each–Axis VRDY_OFF Alarm Ignore Signal IGVRY1 – IGVRY8 <G192>

[Classification] Input signal

[Function] Disables the detection of servo alarm No. 401, 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 No. 401, VRDY OFF, even when the servo amplifier ready signal for the corresponding axis goes off. The servo ready signal SA remains set to 1. When each axis VRDY OFF alarm ignore signals for all axes are set to 1, the control unit sets the SA signal to 0. The SA signal, however, can remain set to 1 depending on the setting of SAKx, bit 6 of parameter No. 1804.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G066								IGNVRY
G192	IGVRY8	IGVRY7	IGVRY6	IGVRY5	IGVRY4	IGVRY3	IGVRY2	IGVRY1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1804		SAK						

[Data type] Bit

SAK When the VRDY OFF alarm ignore signal IGNVRY is 1, or when the VRDY OFF alarm ignore signals for all controlled axes IGVRY1 to IGVRY8 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
401	SERVO ALARM: n–TH AXIS VRDY OFF	The n–th axis (axis 1–8) servo amplifier READY signal (DRDY) went off.

Note

Note 1 While the VRDY OFF alarm ignore signal is set to 1, and a servo alarm other than alarm No. 401 occurs, the control unit detects the alarm.

Note 2 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.

Note 3 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.

2.10 ABNORMAL LOAD DETECTION

General

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 a load torque on the motors and sends it as an estimated load torque to the PMC via the CNC. If the detected load is abnormally great compared with the value specified in the parameter, the function stops the servo motor as early as possible or reverses the motor by an appropriate value specified in a parameter, in order to minimize a possible damage to the machine. (The function to reverse motors is effective only for servo motors.)

The abnormal load detection function is further divided as follows:

(1) Estimated load torque output function

The CNC is always calculating the estimated load torque for the motor (excluding acceleration/deceleration torque). The estimated load torque output function enables the PMC to read the calculated torque using the window function.

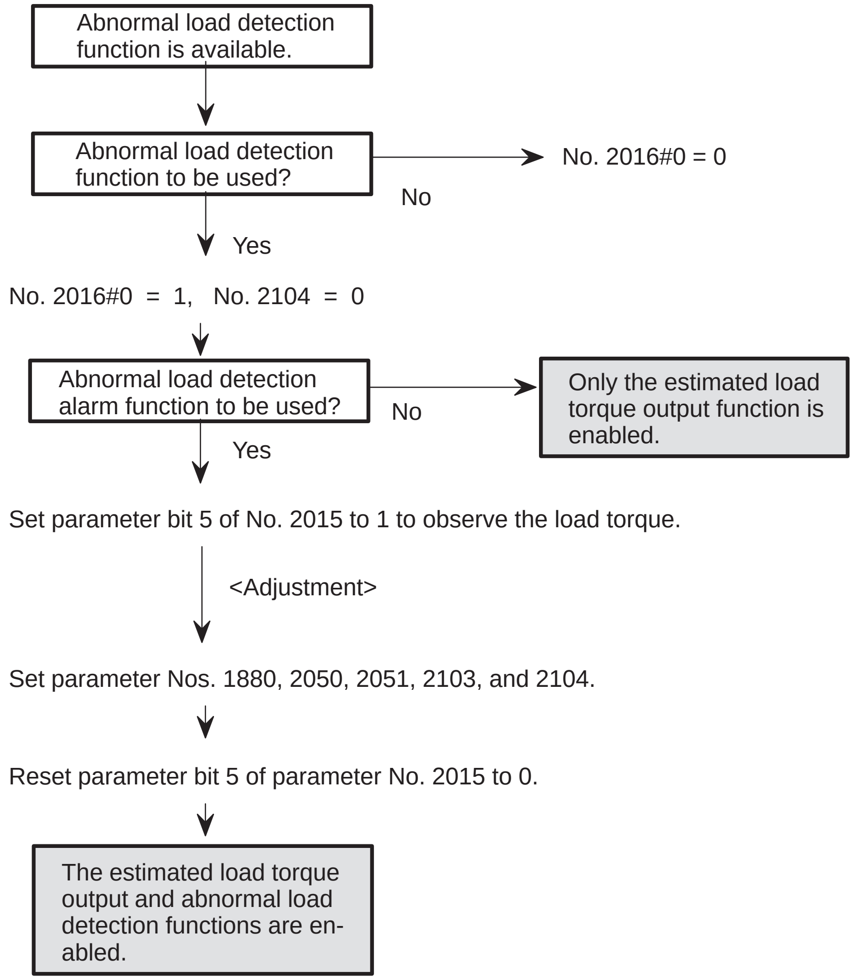
(2) Abnormal load detection alarm function

This function stops motors or reverses them by an amount specified in a parameter, causing the CNC to output an alarm, if the load torque obtained by the estimated load torque output function is greater than the value specified in a parameter. (The function to reverse motors is effective only for servo motors.)

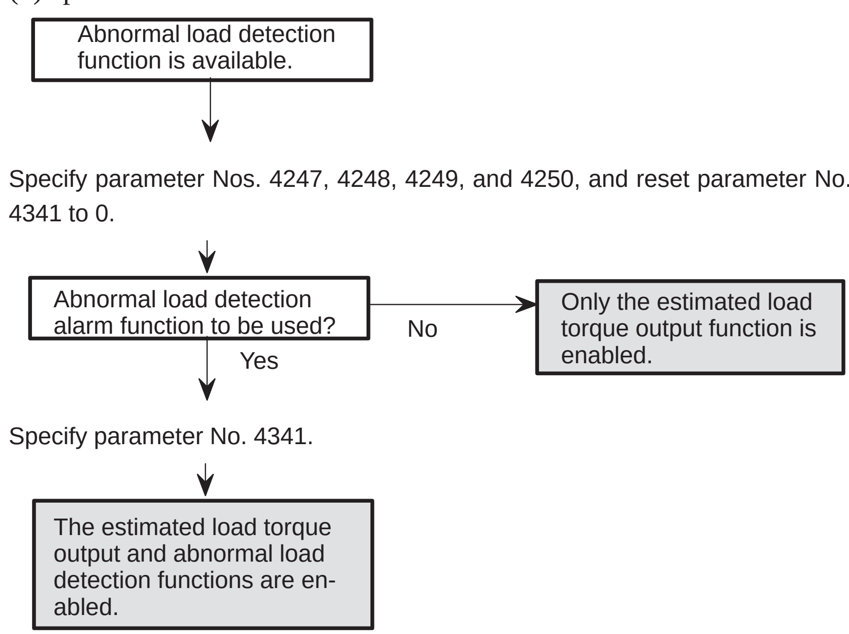
- **Parameter setting**

The following flowcharts explain how to specify parameters for the abnormal load detection function.

(1) Servo axis



(2) Spindle



Signal

Servo axis abnormal load detected signal ABTQSV <F090#0>

[Classification] Output signal

[Function] Informs the PMC that an abnormal load was detected on a servo axis.

[Output condition] This signal becomes "1" if:

- An abnormal load is detected for a servo axis, Cs axis, spindle positioning axis, or spindle axis during rigid tapping.

First–spindle abnormal load detected signal ABTSP1 <F090#1>

[Classification] Output signal

[Function] Informs the PMC that an abnormal load was detected on the first axis.

[Output condition] This signal becomes "1" if:

- An abnormal load is detected for the first spindle under speed control.

Second–spindle abnormal load detected signal ABTSP2 <F090#2>

[Classification] Output signal

[Function] Informs the PMC that an abnormal load was detected on the second axis.

[Output condition] This signal becomes "1" if:

- An abnormal load is detected for the second spindle under speed control.

The following list summarizes the alarms and signals output by each function.

	Signal output		Alarm	
	ABTQSV	ABTSP1/ ABTSP2	409	754/764
Servo axis	○	—	○	—
Cs contour control	○	—	○	—
Spindle positioning axis	○	—	○	—
Rigid tapping	○	—	—	○
Spindle axis for speed control	—	○	—	○

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F0090						ABTSP2	ABTSP1	ABTQSV

Parameter

(1) Parameter common to servo axes and spindles

1880	Timer for abnormal load detection alarm

[Data type] Word

[Unit of data] msec

[Valid data range] 0 to 32767
(If 0 is set, 200 ms is assumed.)

This parameter specifies the interval between the detection of an abnormal load and the issue of a servo alarm. When the set value is not a multiple of eight, it is rounded up to the nearest multiple of eight.

(Example) When 30 is set, the system assumes 32 ms.

(2) Servo axis parameters

	#7	#6	#5	#4	#3	#2	#1	#0
2015			TDOUT					

[Data type] Bit axis

TDOUT 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								ABNTDT

[Data type] Bit axis

ABNTDT Output of the estimated load torque for each axis
0 : Disabled
1 : Enabled

This parameter must be specified when using the estimated load torque output function or abnormal load detection alarm function.

2050	Velocity control observer

[Data type] Word axis

[Valid data range] 0 to 32767

[Setting value] 3559

When using the velocity loop observer (by setting bit 2 of parameter No. 2003 to 1), set 956 in this parameter.

2051	Velocity control observer

[Data type] Word axis

[Valid data range] 0 to 32767

[Setting value] 3329

When using the velocity loop observer (by setting bit 2 of parameter No. 2003 to 1), set 510 in this parameter.

2103	Retraction distance upon the detection of an abnormal load

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter specifies the amount by which the tool is retracted, by reversing the motor, if an abnormal load 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, upon the detection of an abnormal load 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 abnormal load causes the tool to e retracted in the reverse direction by an amount A, then stop , if the specified feedrate is equal to or greater than the value listed below, for each detection unit.

Detection unit	Feedrate
1μ 0.1μ	A / 8 mm/ min A / 80 mm/ min

When this parameter is set to 0, the motor stops immediately upon the detection of abnormal load.

2104	Threshold for abnormal load detection alarm

[Data type] Word axis

[Unit of data] Torque command unit (Refer to the digital servo operator's manual for details.)

[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 abnormal load detection alarm is issued.

Monitor the load torque by setting bit 5 of parameter No. 2015 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.

(3) Spindle parameters

	#7	#6	#5	#4	#3	#2	#1	#0
4015							SPLDMT	

[Data type] Bit axis

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

[Data type] Word axis

[Unit of data] 1 msec

[Valid data range] 0 to 8192

[Standard setting] Depends of the motor model.

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.

4248	Spindle load torque monitor constant

[Data type] Word axis

[Valid data range] 0 to 32767

[Standard setting] Depends of the motor model.

This constant is determined from the maximum output torque and inertia of the motor. It is used for observer processing.

4249	Observer gain 1 for spindle load torque monitor

[Data type] Word axis

[Valid data range] 0 to 32767

[Standard setting] 500

4250	Observer gain 2 for spindle load torque monitor

[Data type] Word axis

[Valid data range] 0 to 32767

[Standard setting] 500

4341	Threshold for abnormal load detection alarm

[Data type] Word axis

[Unit of data] 0.01 %

[Valid data range] 0 to 10000

This parameter specifies the threshold load torque at which an abnormal load 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 abnormal load detection alarm is issued for the spindle.

Alarm and message

(1) Servo axis

Number	Message	Description
409	Servo alarm: Abnormal load detected on axis n	An abnormal load was detected on a servo motor, or on a spindle motor during Cs mode. To release the alarm, use RESET.

(2) Spindle

Number	Message	Description
754	Abnormal load detected on the first spindle	An abnormal load was detected on the first spindle motor. To release the alarm, use RESET.
764	Abnormal load detected on the second spindle	An abnormal load was detected on the second spindle motor. To release the alarm, use RESET.

PMC window function

- **Reading the load torque data**

The load torque data can be read at the PMC using its window function.

(1) Servo axis

[Input data structure]

Top address +0	(Function code) 211	
2	(Completion code) — (Not to be set)	
4	(Data length) — (Not to be set)	
6	(Data number) 0	
8	(Data attribute) M (M = 1 to n or –1)	(M = 1 to n: Specifies separately for each axis whether data for it is to be read; n is an axis number. M = –1: Specifies that data for all axes are to be read.
10	(Data area) — (Not to be set)	
42		

[Completion code]

0: The load torque data was read normally.

4: Incorrect data was specified as a data attribute, that is a value other than –1 or 1 to n (number of axes) was specified. Alternatively, a value greater than the number of controllable axes was specified.

[Output data structure]

Top address +	Contents	Description of value
0	(Function code) 211	
2	(Completion code) ? (Refer to the above description about the completion code.)	
4	(Data length) L (L = 2*n, where n is the number of specified axes)	
6	(Data number) 0	
8	(Data attribute) M (M: Data at input)	
10	Load torque for the specified axis (2 bytes)	Signed binary format (negative value in two's complement)

Or if there are four controllable axes

Top address +	Contents	Description of value
10	Load torque for the first axis (2 bytes)	Signed binary format (negative value in two's complement)
12	Load torque for the second axis (2 bytes)	
14	Load torque for the third axis (2 bytes)	
16	Load torque for the fourth axis (2 bytes)	

(2) Spindle

[Input data structure]

Top address +0	(Function code) 211
2	(Completion code) — (Not to be set)
4	(Data length) — (Not to be set)
6	(Data number) 1
8	(Data attribute) M (M = 1 to n or –1)
10	(Data area) — (Not to be set)
42	

(M = 1 to n: Specifies separately for each axis whether data for it is to be read; n is an axis number.

M = –1: Specifies that data for all axes are to be read.

[Completion code]

0: The load torque data was read normally.

4: Incorrect data was specified as a data attribute, that is a value other than –1 or 1 to n (number of axes) was specified. Alternatively, a value greater than controllable axes was specified.

[Output data structure]

Address	Content	Description of value
Top address + 0	(Function code) 211	
2	(Completion code) ? (Refer to the above description about the completion code.)	
4	(Data length) L (L = 2*n, where n is the number of specified axes)	
6	(Data number) 1	
8	(Data attribute) M (M: Data at input)	
10	Load torque for the specified axis (2 bytes)	Signed binary format (negative value in two's complement)

Or if there are two controllable axes

Address	Content	Description of value
10	Load torque for the first axis (2 bytes)	Signed binary format (negative value in two's complement)
12	Load torque for the second axis (2 bytes)	

3 MANUAL OPERATION

3.1 JOG FEED/ INCREMENTAL FEED

General

- **Jog feed**

In the jog mode, turning a feed axis and direction selection signal to “1” on the machine operator’s panel continuously moves the tool along the selected axis in the selected direction.
Manual operation is allowed for one axis at a time. 3 axes can be selected at a time by parameter JAX (No.1002#0).

- **Incremental feed**

In the incremental feed mode, turning a feed axis and direction selection signal 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. Each step can be 10, 100, or 1000 times the least input increment.

The jog feedrate is specified in a parameter (No.1423)
The jog feedrate can be adjusted with the jog feedrate override dial.
With the rapid traverse selection switch the tool can be moved at the rapid traverse rate regardless of the jog feedrate override signal.

Signal

The following signals determine the way in which 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 – *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 select signal MP1 an d MP2.
For the signals selecting the mode, see Section 2.6, “Mode Selection Signals.” For the manual handle feed selection signals, MP1 and MP2 of selection of the move amount, see 3.2 “Manual handle feed.” For rapid traverse override signals ROV1 and ROV2, see Section 7.1.7.1, “Feedrate Override Signals.”
Other signals are described below.

Feed Axis and Direction Selection Signal +J1 – +J8<G100> –J1 – –J8<G102>

[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.

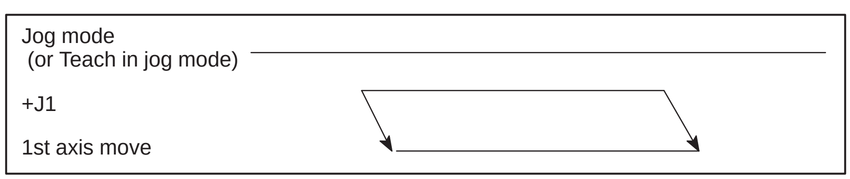
+ J 1

1 First axis is fed
2 Second axis is fed
3 Third axis is fed
: :
8 Eighth axis is fed

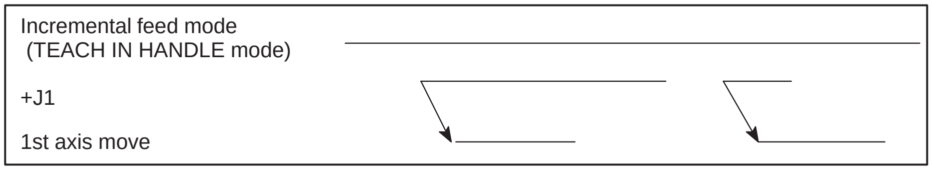
+ Feed in positive direction
– Feed in negative direction

[Operation] When the signal is high, 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 signal is “1”.

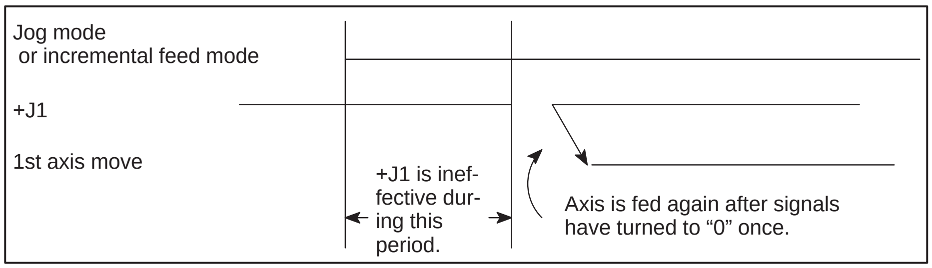


- In incremental feed, the control unit feeds the specified axis by the step distance which is specified by the manual handle feed move distance selection signal MP1, MP2. Then the control unit stops it. Even if the signal is set to “0” while the axis is being fed, the control unit does not stop feeding it.
To feed the axis again, set the signal to “0”, then set it to “1” again.

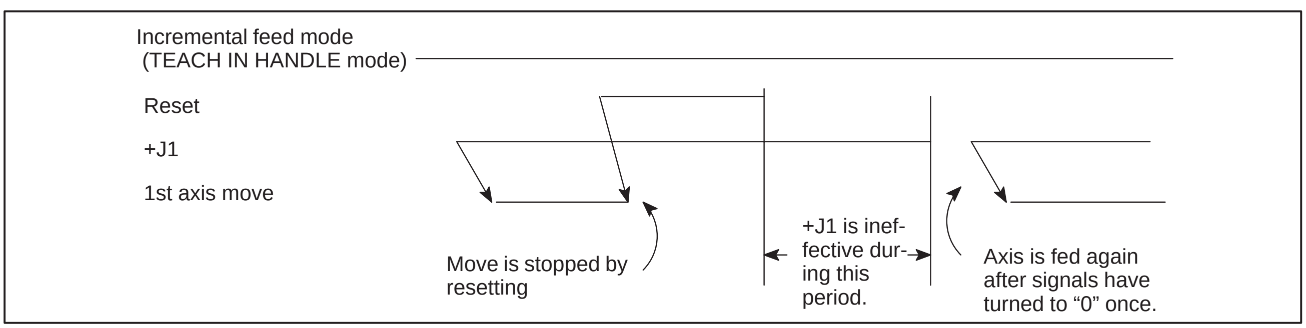


Note 1 If both the positive direction and negative direction signals of the same axis are simultaneously set to "1", neither the positive direction nor the negative direction is selected. The control unit assumes that both these signals are set to "0".

Note 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.



Note 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 cannot be validated by releasing the reset state. After the reset state is released, set these signals to "0", then set them to "1" again.



Manual Feedrate Override Signal *JV0 – *JV15<G011>

[Classification] Input signal

[Function] Selects a feedrate in jog feed or incremental feed. These signals are sixteen binary code signals, which correspond to the override values as follows:

$$\text{Override value (\%)} = 0.01\% \times \sum_{i=0}^{15} |2^{i} \times Vi|$$

where
Vi = 0 when the *JVi signal is “1”
Vi = 1 when the *JVi signal 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 – *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 the parameter (no. 1423) is overridden by the value specified by the JVi signal.

Note 1 The JVi signals also serve as the override signals during dry run in automatic operation mode.

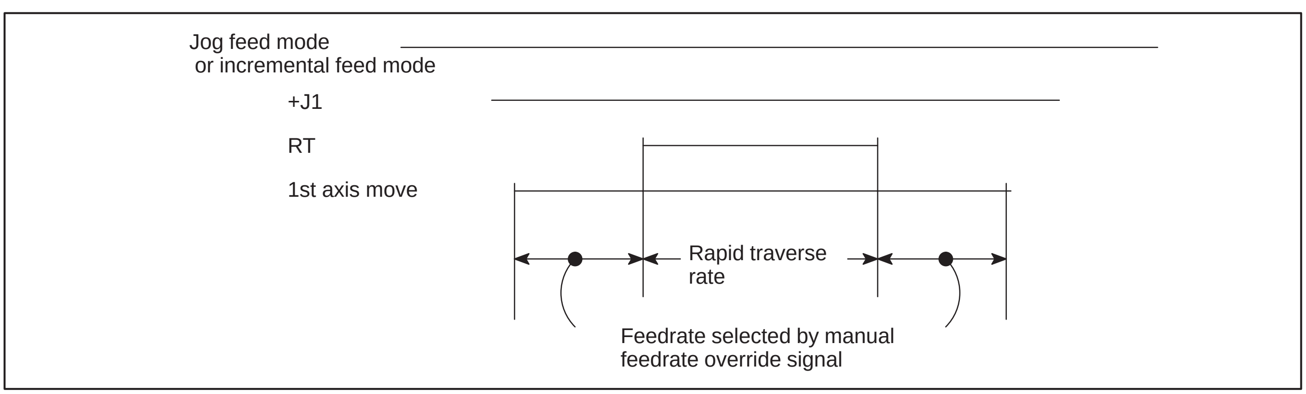
Manual rapid traverse selection signal RT<G019#7>

[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”.



Note 1 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. A parameter RPD (No. 1401#0) can be specified so that the rapid traverse is validated before the reference position return is completed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G010	*JV7	*JV6	*JV5	*JV4	*JV3	*JV2	*JV1	*JV0

	#7	#6	#5	#4	#3	#2	#1	#0
G011	*JV15	*JV14	*JV13	*JV12	*JV11	*JV10	*JV9	*JV8

	#7	#6	#5	#4	#3	#2	#1	#0
G019	RT							

	#7	#6	#5	#4	#3	#2	#1	#0
G100	+J8	+J7	+J6	+J5	+J4	+J3	+J2	+J1

	#7	#6	#5	#4	#3	#2	#1	#0
G102	–J8	–J7	–J6	–J5	–J4	–J3	–J2	–J1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002								JAX

[Data type] Bit

JAX Number of axes controlled simultaneously in jog 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

[Data type] Bit

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				

[Data type] Bit

JRV Manual continuous feed (jog feed)
0 : Jog feed is performed at feed per minute.
1 : Jog feed is performed at feed per rotation.

Note 1 Specify a feedrate in parameter No. 1423.

1423	Feedrate in manual continuous feed (jog feed) for each axis

[Data type] Word axis

(1) In M series, or in T series when JRV, bit 4 of parameter No. 1402, is set to 0 (feed per minute), specify a jog feedrate at feed per minute with an override of 100%.

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range
Millimeter machine	1 mm/min	6 – 32767
Inch machine	0.1 inch/min	
Rotation axis	1 deg/min	

(2) When JRV, bit 4 of parameter No. 1402, is set to 1 (feed per revolution) in T series, specify a jog feedrate (feed per revolution) under an override of 100%.

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range
Millimeter machine	0.01 mm/rev	0 – 32767
Inch machine	0.001 inch/rev	
Rotation axis	0.01 deg/rev	

1424	Manual rapid traverse rate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 240000	30 – 100000
Inch machine	0.1 inch/min	30 – 96000	30 – 48000
Rotation axis	1 deg/min	30 – 240000	30 – 100000

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 is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx				

[Data type] Bit axis

JGLx Acceleration/deceleration in manual continuous feed (jog feed)

0: Exponential acceleration/deceleration is applied.

1: Linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation is applied (depending on which is used in cutting feed).

1624	Time constant of exponential acceleration/deceleration or bell–shaped acceleration/deceleration or linear acceleration/deceleration after interpolation, in jog feed for each axis.

[Data type] Word axis

[Unit of data] 1 msec

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration, bell–shaped acceleration/deceleration or linear acceleration/deceleration after interpolation in jog feed for each axis.

1625	FL rate of exponential acceleration/deceleration in jog feed for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotation axis	1 deg/min	6 – 15000	6 – 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in jog feed for each axis.

3016	Clamp value of jog feedrate override

[Data type] Two–word

[Unit of data] 0.01%

[Valid data range] 0 to 65534

This parameter specifies the clamped value for manual feedrate override. When the entered manual feedrate override is greater than the value specified by this parameter, it is clamped to this value.

Note 1 Manual feedrate override is input using the manual feedrate override signal (*JV0 to *JV15) of the interface between the PMC and the CNC.

Note

Note 1 Time constant and method of automatic acceleration/ deceleration for manual rapid traverse are the same as G00 in programmed command.

Note 2 For incremental feeding along an axis under diameter programming, the tool moves in units of the diameter.

Note 3 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

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.3.2	JOG FEED
	III.3.3	INCREMENTAL FEED
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.3.2	JOG FEED
	III.3.3	INCREMENTAL FEED

3.2 MANUAL HANDLE FEED

General

In the manual handle feed mode, the tool can be minutely moved by rotating the manual pulse generator. Select the axis along which the tool is to be moved with the handle feed axis selection signal.
The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment. Or the distance the tool is moved when the manual pulse generator is rotated by one graduation can be magnified by 10 times or by one of the two magnifications specified by parameters (No. 7113 and 7114).
The handle magnifications can be selected by the manual handle feed move distance selection signal.

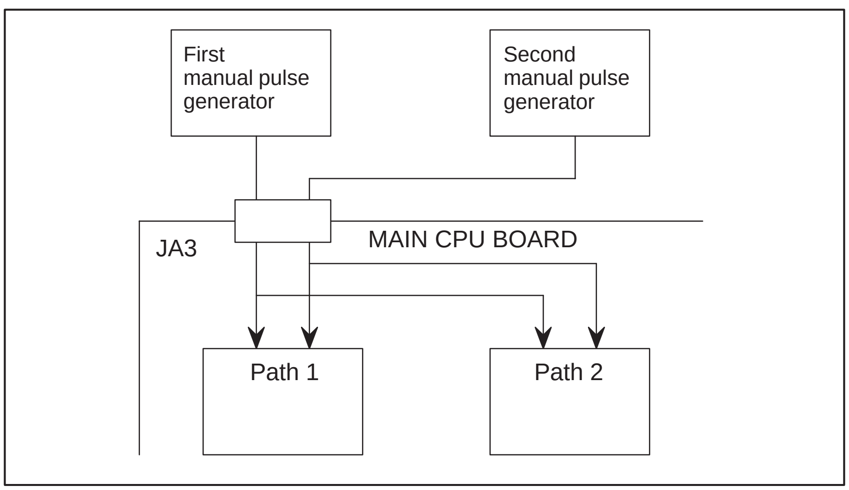
The number of manual pulse generators available depends on the type of an option used as listed below.

(M series)
– Control with one manual handle: Up to one generator
– Control with two or three manual handles: Up to three generators

(T series)
– Control with one manual handle: Up to one generator
– Control with two manual handles: Up to two generators

Two–path control

Which manual pulse generator moves which axis of which path depends on the setting of manual handle feed axis select signals for each path. For each path, eight bits are reserved as manual handle feed axis select signals.



- **Availability of manual pulse generator in Jog mode**

Parameter JHD (bit 0 of No. 7100) enables or disables the manual pulse generator in the JOG mode.
When the parameter JHD(bit 0 of No. 7100) is set 1,both manual handle feed and incremental feed are enabled.

- **Availability of manual pulse generator in TEACH IN JOG mode**

Parameter THD (bit 1 of No. 7100) enables or disables the manual pulse generator in the TEACH IN JOG mode.

- **A command to the MPG exceeding rapid traverse rate**

Parameter HPF (bit 4 of No. 7100) specifies as follows:

The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are ignored.(The distance the tool is moved may not match the graduations on the manual pulse generator.)

The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are not ignored but accumulated in the CNC. (No longer rotating the handle does not immediately stop the tool. The tool is moved by the pulses accumulated in the CNC before it stops.)

- **Movement direction of an axis to the rotation of MPG**

Parameter HNGx (No. 7102#0) switches the direction in which the tool moves along an axis, corresponding to the direction in which the handle of the manual pulse generator is rotated.

Signal

Manual Handle Feed Axis Selection Signals

- **(M series)**
  HS1A – HS1D
  <G018#0 – #3>
  HS2A – HS2D
  <G018#4 – #7>
  HS3A – HS3D
  <G019#0 – #3>

- **(T series)**
  HS1A – HS1D
  <G018#0 – #3>
  HS2A – HS2D
  <G018#4 – #7>

- **(Two–path control)**
  HS1A$^{\#1}$ – HS1D$^{\#1}$
  <G018#0 – #3>
  HS2A$^{\#1}$ – HS2D$^{\#1}$
  <G018#4 – #7>
  HS3A$^{\#1}$–HA3D$^{\#1}$ (M series)
  <G019#0–#3>
  HS1A$^{\#2}$ – HS1D$^{\#2}$
  <G1018#0 – #3>
  HS2A$^{\#2}$ – HS2D$^{\#2}$
  <G1018#4 – #7>
  HS3A$^{\#3}$–HS3D$^{\#2}$ (M series)
  <G1019#0–#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 three generators can be used.) (For two–path, these signals are provided for each manual pulse generator and each path.) The number in the signal name indicates the number of the manual pulse generator to be used.

HS 1 A

1 First manual pulse generator
2 Second manual pulse generator
3 Third manual pulse generator (M series)

Code signals A, B, C, and D correspond to the feed axes as listed in the following table:

Manual handle feed axis selection				**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
0	1	1	0	6th axis
0	1	1	1	7th axis
1	0	0	0	8th axis

<Two–path control>

Manual handle feed axis selection				**Feed axis**
HSnD#1	HSnC#1	HSnB#1	HSnA#1	
0	0	0	0	No selection (no axis is used for path 1)
0	0	0	1	1st axis of path 1
0	0	1	0	2nd axis of path 1
0	0	1	1	3rd axis of path 1
0	1	0	0	4th axis of path 1
0	1	0	1	5th axis of path 1
0	1	1	0	6th axis of path 1
0	1	1	1	7th axis of path 1

Manual handle feed axis selection				**Feed axis**
HSnD#2	HSnC#2	HSnB#2	HSnA#2	
0	0	0	0	No selection (no axis is used for path 2)
0	0	0	1	1st axis of path 2
0	0	1	0	2nd axis of path 2
0	0	1	1	3rd axis of path 2
0	1	0	0	4th axis of path 2
0	1	0	1	5th axis of path 2
0	1	1	0	6th axis of path 2
0	1	1	1	7th axis of path 2

Manual Handle Feed Amount Selection Signal MP1, MP2<G019#4, 5> (Incremental Feed Signal)

[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.

Travel distance select signal for manual handle feed		Distance traveled		
MP2	**MP1**	**Manual handle feed**	**Manual handle interrupt**	**Incremental feed**
0	0	Least input increment×1	Least command increment×1	Least input increment×1
0	1	Least input increment×10	Least command increment×10	Least input increment×10
1	0	Least input increment×m*1	Least command increment×m*1	Least input increment×100
1	1	Least input increment×n*1	Least command increment×n*1	Least input increment×1000

*1 Scale factors m and n are specified using parameter Nos. 7113 and 7114.

Note 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.

Note 2 For an axis under diameter programming, the tool moves by the diameter value.

Note 3 See Section 3.3, "Manual Handle Interrupt" for manual handle interrupts, and Section 3.1, "Jog Feed/Incremental Feed" for incremental feed.

Signal address

<For 1–path control>

	#7	#6	#5	#4	#3	#2	#1	#0
G018	HS2D	HS2C	HS2B	HS2A	HS1D	HS1C	HS1B	HS1A

	#7	#6	#5	#4	#3	#2	#1	#0
G019			MP2	MP1	HS3D	HS3C	HS3B	HS3A

<For 2–path control>

	#7	#6	#5	#4	#3	#2	#1	#0
G018	HS2D #1	HS2C #1	HS2B #1	HS2A #1	HS1D #1	HS1C #1	HS1B #1	HS1A #1

	#7	#6	#5	#4	#3	#2	#1	#0
G019			MP2#1	MP1#1	HS3D#1	HS3C#1	HS3B#1	HS3A#1

	#7	#6	#5	#4	#3	#2	#1	#0
G1018	HS2D #2	HS2C #2	HS2B #2	HS2A #2	HS1D #2	HS1C #2	HS1B #2	HS1A #2

	#7	#6	#5	#4	#3	#2	#1	#0
G1019			MP2#2	MP1#2	HS3D#2	HS3C#2	HS3B#2	HS3A#2

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7100				HPF			THD	JHD

[Data type] Bit

JHD Manual handle feed in JOG mode or incremental feed in the manual handle feed
0: Invalid
1: Valid

THD Manual pulse generator in TEACH IN JOG mode
0: Invalid
1: Valid

HPF When a manual handle feed exceeding the rapid traverse rate is issued,
0: The rate is clamped at the rapid traverse rate, and the handle pulses corresponding to the excess are ignored. (The graduations of the manual pulse generator may not agree with the distance the machine has traveled.)
1: The rate is clamped at the rapid traverse rate, and the handle pulses corresponding to the excess are not ignored, but stored in the CNC. (If the rotation of the manual pulse generator is stopped, the machine moves by the distance corresponding to the pulses preserved in the CNC, then stops.)

	#7	#6	#5	#4	#3	#2	#1	#0
7101								IOL

[Data type] Bit

IOL Selects a manual pulse generator interface to be used during manual handle feed.
0: Manual pulse generator interface on the main CPU board
1: Manual pulse generator interface provided in the machine operator's panel interface for I/O–Link

	#7	#6	#5	#4	#3	#2	#1	#0
7102								HNGx

[Data type] Bit axis

HNGx Axis movement direction for rotation direction of manual pulse generator

0: Same in direction
1: Reverse in direction

7110	Number of manual pulse generators used

[Data type] Byte

[Valid data range] 1, 2, or 3

This parameter sets the number of manual pulse generators.

7113	Manual handle feed magnification m

[Data type] Word

[Unit of data] One time

[Valid data range] 1 to 127

This parameter sets the magnification when manual handle feed movement selection signal MP2 is on.

7114	Manual handle feed magnification n

[Data type] Word

[Unit of data] One time

[Valid data range] 1 to 1000

This parameter sets the magnification when manual handle feed movement selection signals MP1 and MP2 are "1".

Note

Note 1 Rotate the manual pulse generator at a rate of five rotations per second or lower.

Note 2 Rotating the handle quickly with a large magnification such as x100 moves the tool too fast or 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. The feedrate is clamped at the rapid traverse rate.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.3.4	MANUAL HANDLE FEED
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.3.4	MANUAL HANDLE FEED

3.3 MANUAL HANDLE INTERRUPTION

General

Rotating the manual pulse generator during automatic operation can increase the distance traveled by the amount corresponding to the handle feed. The axis to which the handle interrupt is applied is selected using the manual handle interrupt axis select signal.
The minimum travel distance per graduation is the least command increment. The minimum travel distance can be increased by tenfold or by two scale factors (parameter Nos. 7113 and 7114). Each scale factor can be selected using the manual handle travel distance select signal (Section 3.2, "Manual Handle Feed").

Signal

Manual Handle Interrupt Axis Selection Signal

- **(M series)**
  HS1IA – HS1ID
  <G041#0 – #3>
  HS2IA – HS2ID
  <G041#4 – #7>
  HS3IA – HS3ID
  <G042#0 – #3>

- **(T series)**
  HS1IA – HS1ID
  <G041#0 – #3>
  HS2IA – HS2ID
  <G041#4 – #7>

- **(Two–path control)**
  HS1IA$^{\#1}$ – HS1ID$^{\#1}$
  <G041#0 – #3>
  HS2IA$^{\#1}$ – HS2ID$^{\#1}$
  <G041#4 – #7>
  HS3IA$^{\#1}$–HS3ID$^{\#1}$ (M series)
  <G042#0–#3>
  HS1IA$^{\#2}$ – HS1ID$^{\#2}$
  <G1041#0 – #3>
  HS2IA$^{\#2}$ – HS2ID$^{\#2}$
  <G1041#4 – #7>
  HS3IA$^{\#2}$–HS3ID$^{\#2}$ (M series)
  <G1042#0–#3>

[Classification] Input signal

[Function] These signals select an axis to which the manual handle interrupt is applied. There are three sets of signals, each corresponding to a manual pulse generator (up to three). Each set consists of four code signals A, B, C, and D. (For the T series (two–path control), each manual pulse generator has one set of signals for each tool post.) The number in each signal name corresponds to the number (position) of the manual pulse generator.

HS <u>1</u> IA

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 (M series)

The correspondence between the code signals and the selected feed axis is similar to the correspondence with the manual handle feed axis select signals. See Section 3.2, "Manual Handle Feed."

Signal address

<For 1–path control >

	#7	#6	#5	#4	#3	#2	#1	#0
G041	HS2ID	HS2IC	HS2IB	HS2IA	HS1ID	HS1IC	HS1IB	HS1IA
G042					HS3ID	HS3IC	HS3IB	HS3IA

<For 2–path control>

	#7	#6	#5	#4	#3	#2	#1	#0
G041	HS2ID #1	HS2IC #1	HS2IB #1	HS2IA #1	HS1ID #1	HS1IC #1	HS1IB #1	HS1IA #1
G042					HS3ID	HS3IC	HS3IB	HS3IA
G1041	HS2ID #2	HS2IC #2	HS2IB #2	HS2IA #2	HS1ID #2	HS1IC #2	HS1IB #2	HS1IA #2
G1042					HS3ID	HS3IC	HS3IB	HS3IA

Note

Note 1 The travel distance by handle interruption is determined according to the amount by which the manual pulse generator is turned and the handle feed magnification (x1, x10, xM, xN).
Since this movement is not accelerated or decelerated, it is very dangerous to use a large magnification value for handle interruption.

Note 2 No handle interrupt can be used in a mode in which manual handle feed is enabled (for example, manual handle feed mode and TEACH IN HANDLE mode).

Note 3 Handle interruption is disabled when the machine is locked or interlocked.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	III.4.6	MANUAL HANDLE INTERRUPTION
OPERATOR’S MANUAL (For Lathe) (B–62444E)	III.4.6	MANUAL HANDLE INTERRUPTION

3.4 TOOL AXIS DIRECTION HANDLE FEED FUNCTION (M SYSTEM)

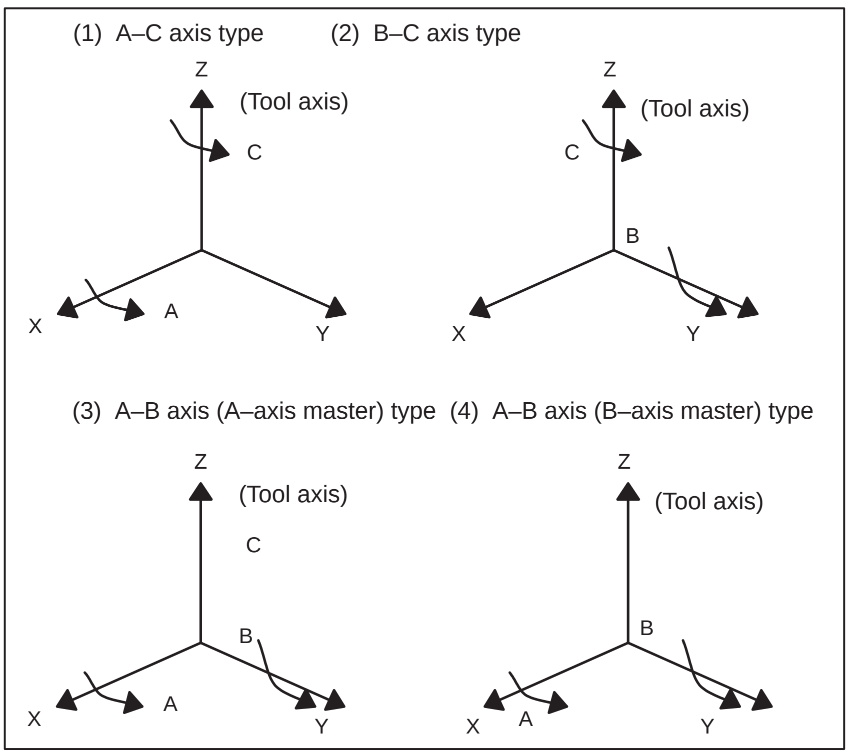
General

By using the tool axis direction handle feed function, the tool can be moved in the axis direction of the tool by an amount equal to the manual pulse generator rotation.

Tool axis direction handle feed is enabled when the following four conditions are satisfied:

(1) Handle mode is selected.
(2) The tool axis direction handle feed mode signal is 1.
(3) In parameter No. 7121, the axis number for the first manual pulse generator is set as the tool axis direction handle feed mode axis.
(4) A manual handle feed axis is selected for the axis set in parameter No. 7121.

Assume that the rotation axes for basic axes X, Y, and Z are A, B, and C, respectively. Assume also that the Z–axis represents the tool axis in the machine coordinate system. Then, depending on the axis configuration of the machine, four tool axis directions are available. Specify the desired type with parameter No. 7120.



Output pulse (Hp) distribution by the manual pulse generator to the X–axis, Y–axis, and Z–axis for the four types is expressed below.

(1) A–C axis type

$Xp = Hp \times \sin(a) \times \sin(c)$
$Yp = -Hp \times \sin(a) \times \sin(c)$
$Zp = Hp \times \cos(a)$

(2) B–C axis type
$Xp = Hp \times \sin(b) \times \cos(c)$
$Yp = Hp \times \sin(b) \times \sin(c)$
$Zp = Hp \times \cos(b)$

(3) A–B axis type (A–axis master)
$Xp = Hp \times \sin(b)$
$Yp = -Hp \times \cos(b) \times \sin(a)$
$Zp = Hp \times \cos(b) \times \cos(a)$

(4) A–B axis type (B–axis master)
$Xp = Hp \times \cos(a) \times \sin(b)$
$Yp = -Hp \times \sin(a)$
$Zp = Hp \times \cos(a) \times \cos(b)$

In the above expressions, a, b, and c represent the positions (angles) of the A–axis, B–axis, and C–axis relative to the machine zero point; those values that are present when tool axis direction handle feed mode is set, or when a reset occurs, are used. To change the feed direction, reenter tool axis direction handle feed mode, or press the reset key.

Signal

Tool axis direction handle feed mode signal ALNGH <G023#7>

[Classification] Input signal

[Function] This signal selects tool axis direction handle feed mode. When the following conditions are all satisfied, tool axis direction handle feed mode is set:

1. This signal is 1.
2. The value of the manual handle feed axis selection signal for the first manual pulse generator matches the value set in parameter No. 7121.
3. Handle mode is set.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G023	ALNGH							

Parameter

• Settings for tool axis direction handle feed

	#7	#6	#5	#4	#3	#2	#1	#0
7104						CXC		TLX

[Data type] Bit

TLX When the tool axis direction handle feed function is used, this parameter selects a tool axis direction when the rotation axes for the three basic axes in the basic coordinate system are positioned to the machine zero point:
0 : Z–axis direction
1 : X–axis direction

CXC Tool axis direction handle feed is performed with:
0 : 5–axis machine.
1 : 4–axis machine.

• Axis configuration for using the tool axis direction handle feed function

7120	Axis configuration for using the tool axis direction handle feed function

[Data type] Byte

[Valid data range] 1 to 4

When using the tool axis direction handle feed function, suppose that the rotation axes for the three basic axes X, Y, and Z in the basic coordinate system are axes A, B, and C, respectively. Suppose also that the Z–axis represents the tool axis direction when the rotation axes are positioned to the machine zero point. Then, depending on the axis configuration of the machine, four types are available. For a 4–axis machine, types (1) and (2) are available.

(1) A–C axis type
(2) B–C axis type
(3) A–B axis (A–axis master) type
(4) A–B axis (B–axis master) type

This parameter selects a type. Values of 1 to 4 are assigned to these types, in order, from top to bottom. When the X–axis represents the tool axis direction, the above types are changed to B–A axis type, C–A axis type, B–C axis (B–axis master) type, and B–C axis (C–axis master) type.

• Axis selection in the tool axis direction handle feed mode

7121	Axis selection in tool axis direction handle feed mode

[Data type] Byte

[Valid data range] 1 to number of controlled axes

This parameter sets an axis number for the manual handle feed axis selection signal for the first manual pulse generator to enable tool axis direction handle feed mode. When the value set in this parameter matches the value of the manual handle feed axis selection signal, tool axis direction handle feed mode is enabled.

Alarm and message

No.	Message	Meaning
5015	* No message appears.	In tool axis direction handle feed mode, a specified rotation axis does not exist.

Note

Note 1 The basic axes X, Y, and Z are determined by parameter No. 1022 (plane selection). The rotation axes A, B, and C are determined by parameter No. 1020 (axis name).

Note 2 If one of the two axes specified by a type set depending on the axis configuration does not exist, alarm P/S 5015 is issued.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.3.6	Tool axis direction handle feed

4 REFERENCE POSITION ESTABLISHMENT

4.1 MANUAL REFERENCE POSITION RETURN

General

The tool is moved in the direction specified in parameter ZMI (bit 5 of No. 1006) for each axis by turning the feed axis and direction select signal to “1” in the manual reference position return mode, and is returned to the reference position.

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.

• Automatic setting of coordinate system

Bit 0 of parameter 1201 (ZPR) can be set to automatically determine the coordinate system at manual reference position return. Parameter 1250 can be set to determine the workpiece coordinate system by assigning, upon the completion of reference position return, the value set in a parameter to a reference point on the tool holder or the tip position of the reference tool.

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 signal for reference position return	*DEC1, *DEC2, *DEC3, ...
Completion signal for reference position return	ZP1, ZP2, ZP3, ...
Reference position establishment signal	ZRF1, ZRF2, ZRF3, ...

Basic Procedure for Reference Position Return

(1) Select the JOG mode or TEACH IN JOG mode, and set the manual reference position return selection signal ZRN to “1”.

(2) Feed a target axis toward the reference position by making an appropriate feed axis and direction selection signal (+J1, –J1, +J2, –J2,…) “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 turned on, making the deceleration signal (*DEC1, *DEC2, *DEC3,…) for reference position deceleration “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).

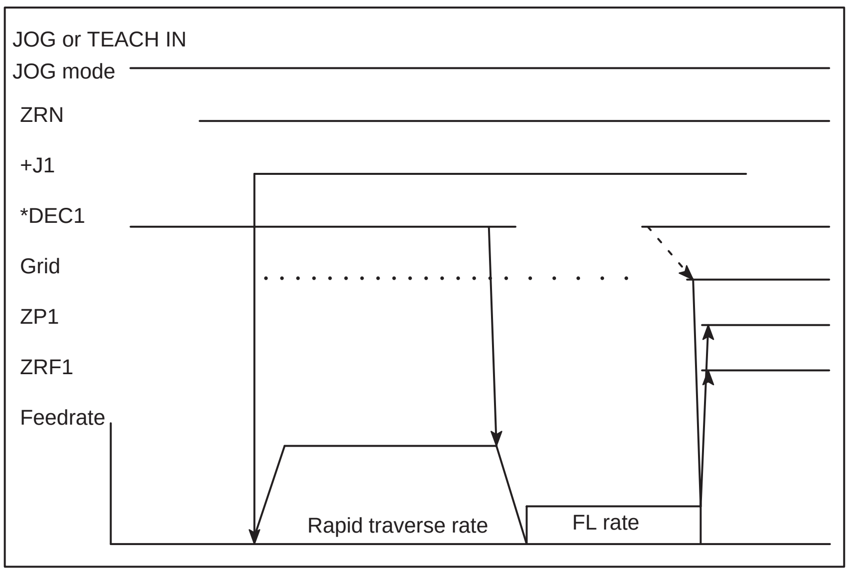
(5) When the deceleration signal turns to “1” again after the limit switch for deceleration is passed, the tool is fed with the feedrate unchanged, then the tool stops at the first grid point (electric grid point).

(6) Upon confirmation that the current position is in 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”.

Step (2) and subsequent steps are performed independently for each axis. The number of simultaneously controlled axes is usually one, but it becomes three by parameter JAX (No. 1002#0).

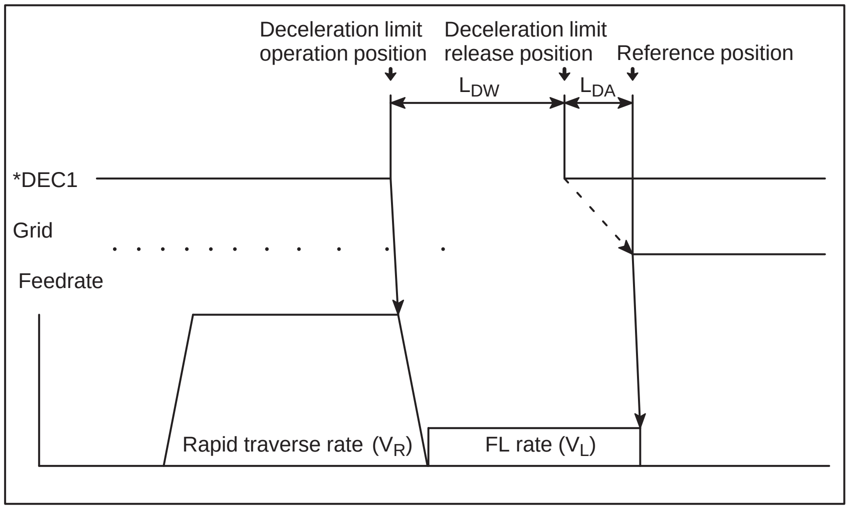
If the feed axis direction selection signal (+J1, –J1, +J2, –J2,…) turns to “0” between step (2) and (5) , the tool is stopped at once, and reference position return is assumed to be canceled. If the signal turn to “1” again, operation resumes from step (3) (rapid traverse).

The timing charts for the basic procedures are given below.



Installation conditions for deceleration limit switch

When installing the deceleration limit switch for manual reference position return, ensure that following conditions are satisfied:



- L_{DW}: Deceleration dog width (mm or inch)

$$L_{DW} > \frac{V_R(\frac{T_R}{2} + 30 + T_S) + 4V_L \times T_S}{60 \times 1000}$$

V_R: Rapid traverse (mm/min or inch/min)
T_R: Rapid traverse time constant (ms)
T_S: Servo time constant (ms)
V_L: FL speed for reference position return (mm/min or inch /min)

- L_{DA}: Distance between deceleration limit switch released position and reference position

L_{DA}: 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 manual reference position return when the reference position has not yet been established, 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 this time, the tool must cross the grid line corresponding to a one–rotation signal from the position detector.

The servo position error is calculated from the following formula:

Servo position error amount$=\frac{F \times 1000}{60} \times \frac{1}{G} \times \frac{1}{U}$

F: Feedrate
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:

Servo position error $= \frac{6000 \times 1000}{60} \times \frac{1}{30} \times \frac{1}{1}$

$= 3,333$

By reversing the formula above, the following formula gives the feedrate F needed to set the servo position error to 128 when the servo loop gain G is 30 s^{-1} and the detection unit U is 1 μm:

$$F = \frac{128 \times 60}{1000} \times 30$$

$$= 230 \text{ [mm/min]}$$

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 manual reference position return.

Grid shift

The grid can be shifted by the distance set in parameter 1850, thus shifting the reference position. The grid shift to be set in the parameter must not exceed the reference counter capacity (parameter No. 1821) (grid interval).

Signal

Manual reference position return selection signal (ZRN) <G043#7>

[Classification] Input signal

[Function] This signal selects manual reference position return. Manual reference position return is a kind of jog feed. Therefor, 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 becomes 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, manual reference position return selection check signal MREF turns to “1”.

Note 1 If the ZRN status changes from “0” to “1” or “1” to “0” during jog feed, the feedrate is decelerated to 0. Then, to make 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 <F004#5>

[Classification] Output signal

[Function] This signal reports that manual reference position return has been selected once.

[Output condition] This signal turns to “1” when:

· Manual reference position return has been selected.

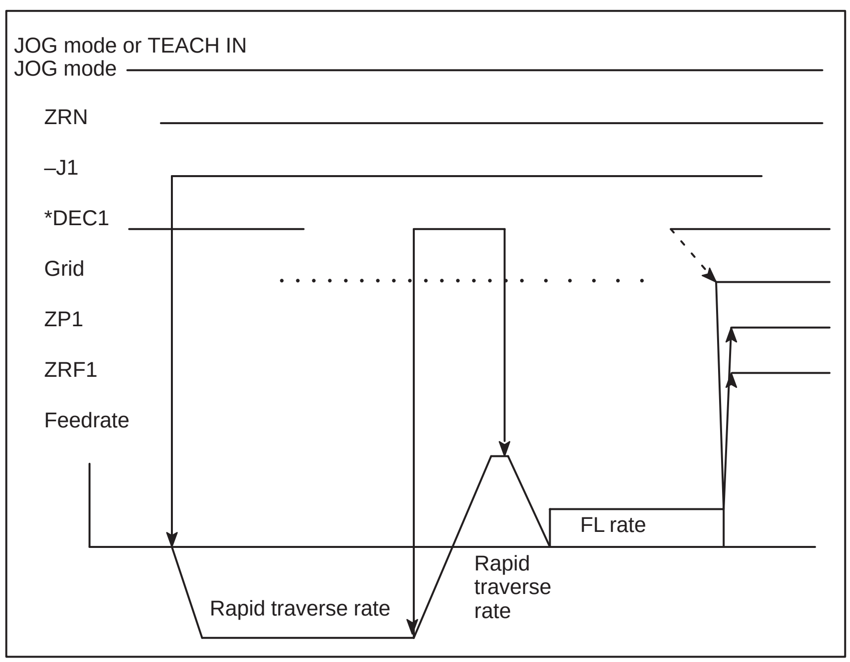
The signal turns to “0” when:

· The selection of manual reference position return has terminated.

Feed Axis and Direction Selection Signal

For details about this signal, see 3.1.2, “Feed Axis and Direction Selection Signal”. Here, only notes on use of reference position return are given.

Note 1 The direction of reference position return is predetermined for each axis by parameter ZMI (No. 1006#5). If the tool is fed in the opposite direction to the predetermined direction in manual reference position return, the deceleration signal for reference position return turns to “0”, and the tool is returned to the point at which the deceleration signal turns to “1” again (that is, the point where the deceleration limit switch would be encountered if the tool were fed in the predetermined direction). Then reference position return is performed automatically in the predetermined direction.



Note 2 When reference position return is selected, an axis whose reference position return end signal is already “1” or an axis whose reference position return end signal was set “1” upon completion of reference position return is locked, and movement along that axis is disabled while the reference position return selection signal (ZRN) is “1”. To perform movement along such an axis, ZRN must be set “0”, and the feed axis and direction selection signal must be set “0” then set “1” again.

Reference position return deceleration signals *DEC1 to *DEC8 <X009>

[Classification] Input signal

[Function] These signals decelerate the feedrate for manual reference position return so that the reference position is approached at a low feedrate.
The deceleration signals are provided for axes in a one–to–one correspondence. A number appended to a deceleration signal represents a controlled axis number.

*DEC 1

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 the operation of the control unit in response to the deceleration signal, see the description of the basic procedure for manual reference position return.

Reference position return end signals ZP1 to ZP8 <F094>

[Classification] Output signal

[Function] These signals report that the tool is at the refernce position on a controlled axis.
These siganls are provided for axes in a one–to–one corespondence.
A number appended to a signal represents a controlled axis number.

ZP 1

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 condition] These signals turn to “1” when:

- Manual reference position returns is completed, and the current position is in the in–position area.
- Automatic reference position return (G28) is completed, and the current position is in the in–position area.
- Reference position return check (G27) is completed, and the current 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 ZRF1AZRF8<F120>

[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 name indicates the number of the controlled axis.

ZRF 1

1 : 1st–axis reference position establishment signal
2 : 2nd–axis reference position establishment signal
3 : 3rd–axis reference position establishment signal
:
:

[Output condition] 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

The signals are set to 0 in the following case:

- When the reference position is lost

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X009	*DEC8	*DEC7	*DEC6	*DEC5	*DEC4	*DEC3	*DEC2	*DEC1

	#7	#6	#5	#4	#3	#2	#1	#0
G043	ZRN							

	#7	#6	#5	#4	#3	#2	#1	#0
F004			MREF					

	#7	#6	#5	#4	#3	#2	#1	#0
F094	ZP8	ZP7	ZP6	ZP5	ZP4	ZP3	ZP2	ZP1

	#7	#6	#5	#4	#3	#2	#1	#0
F120	ZRF8	ZRF7	ZRF6	ZRF5	ZRF4	ZRF3	ZRF2	ZRF1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002								JAX

[Data type] Bit

JAX Number of axes controlled simultaneously in JOG feed, manual rapid traverse and manual reference position return
0 : 1 axis
1 : 3 axes

	#7	#6	#5	#4	#3	#2	#1	#0
1005								ZRNx

[Data type] Bit axis

ZRNx When a command specifying the movement except for G28 is issued in automatic operation (MEM, RMT, or MDI) when a return to the reference position has not been performed since the power was turned on
0 : An alarm is generated. (P/S alarm 224).
1 : An alarm is not generated.

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

Note 1 When this parameter is changed, turn off the power before continuing operation.

[Data type] Bit axis

ZMIx The direction of reference position return and the direction of initial backlash at power–on
0 : Positive direction
1 : Negative direction

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

[Data type] Bit

ZPR Automatic setting of a coordinate system when the manual reference position return is performed
0 : Not set automatically
1 : Set automatically

ZPI Coordinates at the reference position when a coordinate system is set automatically
0 : Value set in parameter No. 1250 is used.
1 : For input in mm, the value set in parameter 1250 is used, or for input in inches, the value set in parameter No. 1251 is used.

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.

1240	Coordinate value of the reference position on each axis in the machine coordinate system

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

Set the coordinate values of the reference positions in the machine coordinate system.

1250	Coordinate value of the reference position on each axis used for setting a coordinate system automatically

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Linear axis (input in mm)	0.01	0.001	0.0001	mm
Linear axis (input in inches)	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically.

1251	Coordinate value of the reference position on each axis used for setting a coordinate system automatically when input is performed in inches

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Linear axis (input in inches)	0.001	0.0001	0.00001	inch

[Valid data range] –99999999 to 99999999

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically when input is performed in inches.

Note 1 This parameter is valid when ZPI in parameter 1201#1 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1300								
		LZR						

[Data type] Bit

LZR Checking of stored stroke limit 1 during the time from power–on to the manual reference position return

0: The stroke limit 1 is checked.

1: The stroke limit 1 is not checked

[Data type] Bit

JZR The manual reference position return at JOG feedrate

0 : Not performed

1 : Performed

	#7	#6	#5	#4	#3	#2	#1	#0
1401						JZR		

1425	FL rate of the reference position return for each axis

[Data type] Word axis

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set feedrate (FL rate) after deceleration when the reference position return is performed for each axis.

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

[Data type] Bit

OZR When manual reference position return is attempted in the halt state during automatic operation (feed hold stop state) under any of the conditions listed below:

0: Manual reference position return is not performed, with P/S alarm No. 091.

1: Manual reference position return is performed without an alarm occurring.

< Conditions >

- When there is a remaining distance to travel.
- When an auxiliary function (miscellaneous function, spindle–speed function, tool function) is being executed.
- When a cycle such as a dwell cycle or canned cycle is being executed.

1821	Reference counter size for each axis

[Data type] Two–word

[Valid data range] 0 to 99999999

Set the size of the reference counter.

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 = $\dfrac{\text{grid interval}}{\text{detection unit}}$

Grid interval = the amount of travel per rotation of the pulse coder

Note 1 When this parameter has been set, the power must be turned off before operation is continued.

[Unit of data]

[Valid data range]

1836	Servo error amount where reference position return is possible

[Data type] Byte axis

[Unit of data] Detection unit

[Valid data range] 0 to 127

This parameter sets a servo error used to enable reference position return in manual reference position return.

In general, set this parameter to 0. (When 0 is set, 128 is assumed as the default.)

Note 1 When bit 0 of parameter No. 2000 is set to 1, a value ten times greater than the value set in this parameter is used to make the check.

Example: When the value 10 is set in this parameter, and bit 0 of parameter No. 2000 is set to 1, reference position return operation is enabled when a servo error of 100 or more occurs.

1850	Grid shift for each axis

[Data type] Two–word

[Unit of data] Detection unit

[Valid data range] –99999999 to 99999999

A grid shift is set for each axis.

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.

Note 1 When this parameter has been set, the power must be turned off before operation is continued.

	#7	#6	#5	#4	#3	#2	#1	#0
3003			DEC					

[Data type] Bit

DEC Deceleration signal (*DEC1 to *DEC8) for manual reference position return

0 : Deceleration is applied when the signal is 0.

1 : Deceleration is applied when the signal is 1.

Alarm and Message

Number	Message	Description
090	REFERENCE RETURN INCOMPLETE	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. Check the program contents.
091	REFERENCE RETURN INCOMPLETE	Manual reference position return cannot be performed in the feed hold state. Perform a manual reference position return in the automatic operation stop state or reset state.
224	RETURN TO REFERENCE POINT	Not returned to reference point before cycle start. (Only when parameter No. #0=0). Do reference position return.

Reference Item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.3.1	MANUAL REFERENCE POSITION RETURN
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.3.1	MANUAL REFERENCE POSITION RETURN

4.2 SETTING THE REFERENCE POSITION WITHOUT DOGS

General

This function moves the tool near around the reference position set for each axis in the manual continuous feed mode. Then it sets the reference position in the reference position return mode without the deceleration signal for reference position return by turning 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 switch for deceleration for reference position return.

Also, 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.

Basic Procedure for Setting the Reference Position Without Dogs

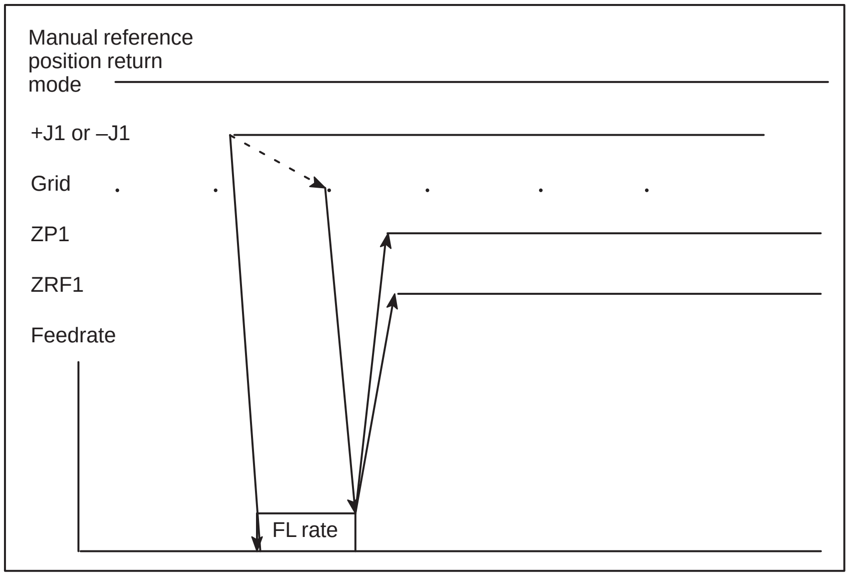
(1) Feed the tool, along the axis for which the reference position is to be set, by manual continuous feed in the reference position return direction. Stop the tool near the reference position, but do not exceed the reference position.

(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.

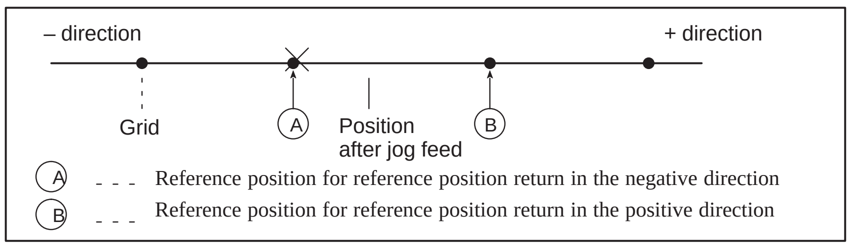
(3) The CNC positions the tool to the nearest grid line (based on one–rotation signals from the position detector) in the reference position return direction specified with bit 5 (ZMIx) of parameter No. 1006. The point to which the tool is thus positioned becomes the reference position.

(4) The CNC checks that the tool is positioned to within the in–position area, then sets the completion signal for reference position return and the reference position establishment signal 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 dogs, 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 explains how to calculate the servo position error.

Grid shift

To shift the reference position, the grid can be shifted by the distance set in parameter No. 1859. The grid shift to be set in the parameter must not exceed the reference counter capacity (parameter No. 1821).

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 with the feed axis and direction selection signal. The completion signal for reference position return is then set to 1.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002							DLZ	JAX

[Data type] Bit

JAX Number of axes controlled simultaneously in manual continuous feed, manual rapid traverse and manual reference position return
0 : 1 axis
1 : 3 axes

DLZ Function setting the reference position without dog
0 : Disabled
1 : Enabled

Note 1 This function can be specified for each axis by DLZx, bit 1 of parameter No. 1005.

	#7	#6	#5	#4	#3	#2	#1	#0
1005							DLZx	ZRNx

[Data type] Bit axis

ZRNx When a command specifying the movement except for G28 is issued in automatic operation (MEM, RMT, or MDI) and when a return to the reference position has not been performed since the power was turned on
0 : An alarm is generated (P/S alarm 224).
1 : An alarm is not generated.

DLZx Function for setting the reference position without dogs
0 : Disabled
1 : Enabled

Note 1 When DLZ of parameter No. 1002 is 0, DLZx is enabled. When DLZ of parameter No. 1002 is 1, DLZx is disabled, and the function for setting the reference position without dogs is enabled for all axes.

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

Note 1 When this parameter is changed, turn off the power before continuing operation.

[Data type] Bit axis

ZMIx The direction of reference position return and the direction of initial backlash at power–on
0 : Positive direction
1 : Negative direction

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

[Data type] Bit

ZPR Automatic setting of a coordinate system when the manual reference position return is performed
0 : Not set automatically
1 : Set automatically

ZPI Coordinates at the reference position when a coordinate system is set automatically
0 : Value set in parameter No. 1250 is used.
1 : For input in mm, the value set in parameter 1250 is used, or for input in inches, the value set in parameter No. 1251 is used.

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.

1240	Coordinate value of the reference position on each axis in the machine coordinate system

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

Set the coordinate values of the reference positions in the machine coordinate system.

1250	Coordinate value of the reference position used when automatic coordinate system setting is performed

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Linear axis (input in mm)	0.01	0.001	0.0001	mm
Linear axis (input in inches)	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically.

1251	Coordinate value of the reference position on each axis used for setting a coordinate system automatically when input is performed in inches

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Linear axis (input in inches)	0.001	0.0001	0.00001	inch

[Valid data range] –99999999 to 99999999

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically when input is performed in inches.

Note 1 This parameter is valid when ZPI in parameter 1201 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1300		LZR						

[Data type] Bit

LZR Checking of stored stroke limit 1 during the time from power–on to the manual position reference return

0: The stroke limit 1 is checked.

1: The stroke limit 1 is not checked

1425	FL rate of the reference position return for each axis

[Data type] Word axis

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set feedrate (FL rate) after deceleration when the reference position return is performed for each axis.

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

[Data type] Bit

OZR When manual reference position return is attempted in the halt state during automatic operation (feed hold stop state) under any of the conditions listed below:

0: Manual reference position return is not performed, with P/S alarm No. 091.

1: Manual reference position return is performed without an alarm occurring.

< Conditions >

- When there is a remaining distance to travel.
- When a auxiliary function (miscellaneous function, spindle–speed function, tool function) is being executed.
- When a cycle such as a dwell cycle or canned cycle is being executed.

1821	Reference counter size for each axis

[Data type] Two–word

[Valid data range] 0 to 99999999

Set the size of the reference counter.

As the size of the reference counter, specify the grid interval for the reference position return in the grid method.

$$\text{Size of the reference counter} = \frac{\text{grid interval}}{\text{detection unit}}$$

Grid interval = the amount of travel per rotation of the pulse coder

Note 1 When this parameter has been set, the power must be turned off before operation is continued.

1836	Servo error amount where reference position return is possible

[Data type] Byte axis

[Unit of data] Detection unit

[Valid data range] 0 to 127

This parameter sets a servo error used to enable reference position return in manual reference position return.

In general, set this parameter to 0. (When 0 is set, 128 is assumed as the default.)

Note 1 When bit 0 of parameter No. 2000 is set to 1, a value ten times greater than the value set in this parameter is used to make the check.

Example: When the value 10 is set in this parameter, and bit 0 of parameter No. 2000 is set to 1, reference position return operation is enabled when a servo error of 100 or more occurs.

1850	Grid shift for each axis

[Data type] Two–word

[Unit of data] Detection unit

[Valid data range] –99999999 to 99999999

A grid shift is set for each axis.

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.

Note 1 When this parameter has been set, the power must be turned off before operation is continued.

Alarm and Message

Number	Message	Description
090	REFERENCE RETURN INCOMPLETE	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. Check the program contents.
091	REFERENCE RETURN INCOMPLETE	Manual reference position return cannot be performed in the feed hold state. Perform a manual reference position return in the automatic operation stop state or reset state.
224	RETURN TO REFERENCE POINT	Not returned to reference position before cycle start. (Only when parameter No. 1005#0=0). Do reference position return.

Note

Note 1 Alarm No. 090 is issued when G28 is specified if the reference position has not yet be established.

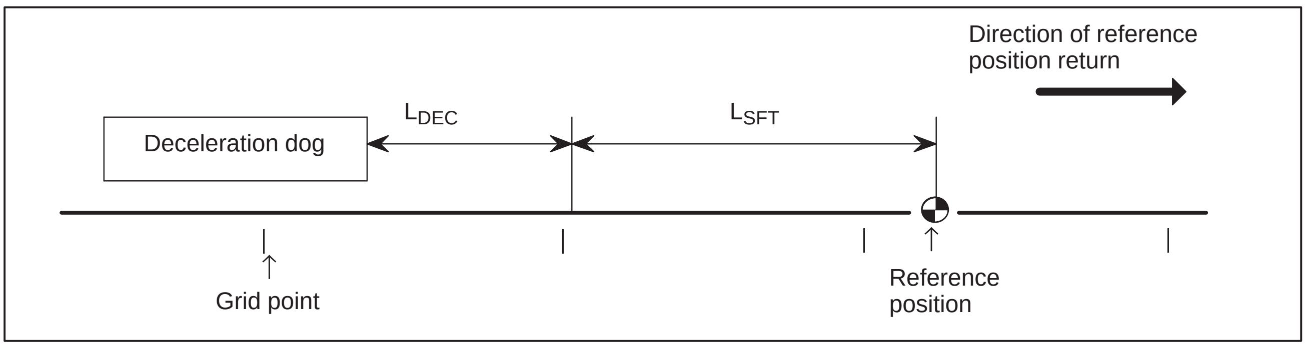
4.3 REFERENCE POSITION SHIFT

General

When reference position return is performed using a grid method, the reference position can be shifted by a parameter–set distance without having to move the deceleration dog.

This function is enabled by setting bit 2 of parameter No. 1002 (SFD) to 1. When distance L_{SFT}, shown below, is set in parameter No. 1850, the reference position can be shifted.

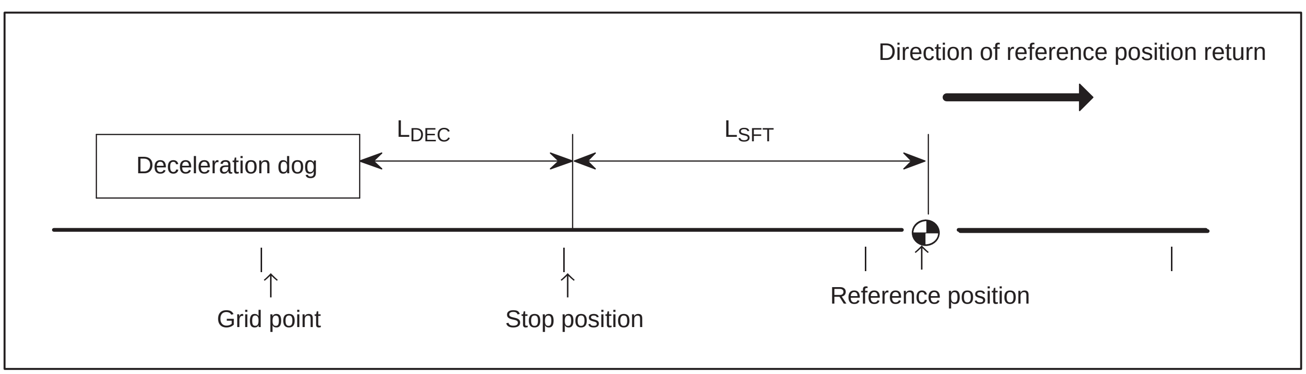
Distance L_{DEC}', shown below, for the axis along which reference position return was last made is indicated on the diagnostic screen (No. 0302).



L_{SFT}: Reference position shift amount
L_{DEC}: Distance from the position where the deceleration dog is turned off to the first grid point (grid point when the shift amount is 0)

● How to adjust the reference position

(1) Set the SFD bit (bit 2 of parameter No. 1002) to 1, and set the reference position shift amount to 0. Then, perform reference position return.

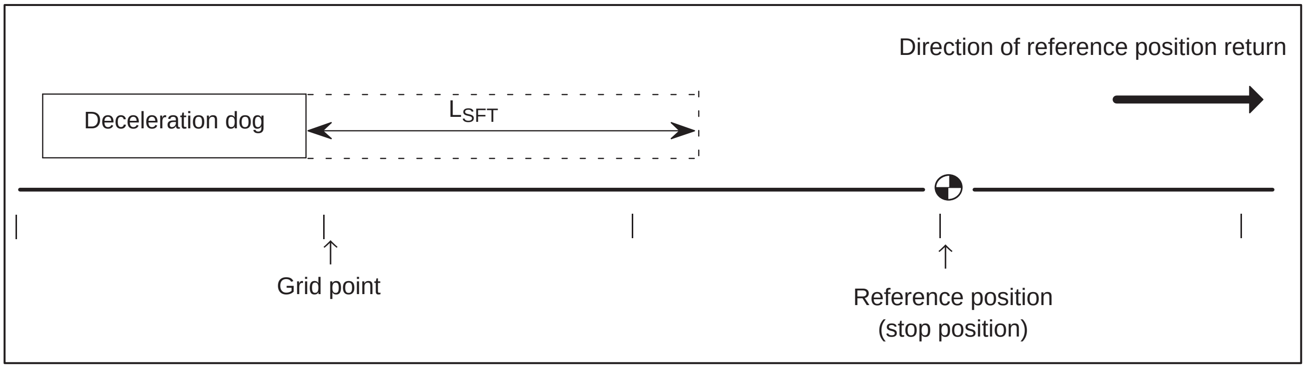


After the deceleration dog is turned off, the tool stops when the first grid point is reached. Distance L_{DEC} is indicated on the diagnostic screen (No. 0302).

(2) Determine the distance L_{SFT} (reference position shift amount) from the stop position to the reference position, and set it in parameter No. 1850.

This completes the adjustment of the reference position.

(3) Perform reference position return again. Then, the tool stops when it reaches the reference position.



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002						SFD		

[Data type] Bit

SFD The function for shifting the reference position is
0 : Not used
1 : Used

1850	Reference position shift for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] –99999999 to 99999999

A reference position shift is set for each axis.

Note 1 When this parameter has been set, the power must be turned off before operation is continued.

Note 2 When bit 2 of parameter No. 1002, SFD, is set to 0, this parameter is used for reference position shift.

Alarm and Message

- **Diagnostic display**

0302	Distance from the position where the deceleration dog is turned off to the first grid point

[Data type] Two–word axis

[Unit of data] 0.001 mm (metric output), 0.0001 inch (inch output)

[Valid data range] –99999999 to 99999999

Note

Note 1 The reference position can be shifted only in the direction of reference position return.

Note 2 When the SFD bit (bit 2 of parameter No. 1002) is 0, only the distance from the position where the deceleration dog is turned off to the first grid point (the grid point after grid shift) is indicated.

4.4 REFERENCE POSITION RETURN

General

The G28 command positions the tool to the reference position, via the specified intermediate point, along the specified axis, then sets the completion signal for reference position return (see Section 4.1) 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.

Parameter

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

Note 1 When this parameter is changed, turn off the power before continuing operation.

[Data type] Bit axis

ZMIx The direction of reference position return and the direction of initial backlash at power–on
0 : Positive direction
1 : Negative direction

1240	Coordinate value of the reference position on each axis in the machine coordinate system

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

Set the coordinate values of the reference positions in the machine coordinate system.

Alarm and Message

Number	Message	Description
405	SERVO ALARM: (ZERO POINT RETURN FAULT)	Position control system fault. Due to an NC or servo system fault in the reference position return, there is the possibility that reference position return could not be executed correctly. Try again from the manual reference position return.

Note

Note 1 The tool is moved from the intermediate point in a sequence similar to manual reference position return, if the G28 command is issued in the following cases:

· When the reference position has not yet been established

· When the input increment (millimeter/inch) is changed at a position other than the reference position

In these cases, the tool leaves the intermediate point in the reference position return direction specified with bit 5 (ZMIx) of parameter No. 1006. The intermediate point must therefore be specified at a position from which reference position return is possible.

Note 2 If the G28 command is issued in the machine lock status, the completion signal for reference position return is not set to 1.

Note 3 If millimeter input is selected for an inch–system machine, the completion signal for reference position return may be set to 1, even when the programmed tool position deviates from the 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

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.6	REFERENCE POSITION
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.6	REFERENCE POSITION

4.5 2ND REFERENCE POSITION RETURN/3RD, 4TH REFERENCE POSITION RETURN

General

The G30 command positions the tool to the 2nd, 3rd, or 4th reference position, via the specified intermediate point, along the specified axis. Then, it sets the completion signal for 2nd, 3rd, or 4th reference position return to 1.

The 2nd, 3rd, or 4th reference position must be set in parameter No. 1241, 1242, or 1243 with coordinates in the machine coordinate system, before issuing the G30 command.

The tool moves to the intermediate point or 2nd, 3rd, or 4th reference position at the rapid traverse rate.

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 ZP21 to ZP 28 <F096> Third reference position return end signals ZP31 to ZP38 <F098> Fourth reference position return end signals ZP41 to ZP48 <F100>

[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, respectively. These signals are provided for axes in a one–to –one correspondence. 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.

ZP 2 1

1 : Return end signal for the first axis
2 : Return end signal for the second axis
3 : Return end signal for the third axis
:
:
2 : Second reference position return
3 : Third reference position return
4 : Fourth reference position return

[Output condition] 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
F096	ZP28	ZP27	ZP26	ZP25	ZP24	ZP23	ZP22	ZP21

	#7	#6	#5	#4	#3	#2	#1	#0
F098	ZP38	ZP37	ZP36	ZP35	ZP34	ZP33	ZP32	ZP31

	#7	#6	#5	#4	#3	#2	#1	#0
F100	ZP48	ZP47	ZP46	ZP45	ZP44	ZP43	ZP42	ZP41

Parameter

1241	Coordinate value of the second reference position on each axis in the machine coordinate system
1242	Coordinate value of the third reference position on each axis in the machine coordinate system
1243	Coordinate value of the fourth reference position on each axis in the machine coordinate system

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

Set the coordinate values of the reference positions in the machine coordinate system.

Alarm and Message

Number	Message	Description
046	ILLEGAL REFERENCE RETURN COMMAND	Other than P2, P3 and P4 are commanded for 2nd, 3rd and 4th reference position return command. Correct program.

Note

Note 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.

Note 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

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.6	REFERENCE POSITION
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.6	REFERENCE POSITION

4.6 FLOATING REFERENCE POSITION RETURN

General

It is possible to return the tool to the floating reference position by commanding the G30.1.
The floating reference position is located on the machine and can be a reference position of some sort of machine operation. It is not always a fixed position but may vary in some cases. The floating reference position can be set using the soft keys of MDI and can be memorized even if the power is turned off.
Generally, the position where the tools can be replace on machining center or milling machine is a set position on the machinery. The tools cannot be replaced at any position. Normally the tool change position is at any of the No. 1 to No. 4 reference position. The tool can be restored to these positions easily by G28 or G30 command. However, depending on the machine, the tools can be replaced at any position as long as it does not contact the workpiece.
In lathes, the tool can generally be changed at any position unless it touches the workpiece or tailstock.
For machinery such as these, in order to reduce the cycle time, it is advantageous to replace tools at a position as close as possible to the workpiece. For this purpose, change position should be changed for each workpiece and this feature can be easily realized by this function. Namely, the tool change position which is suitable for workpieces can be memorized as the floating reference position and it is possible to return the tool to the tool change position easily by commanding the G30.1.
When the G30.1 is commanded, the axis commanded goes to the specified intermediate position with rapid traverse at first and then goes to the floating reference position from the intermediate point with rapid traverse. The positioning to the intermediate position or to the floating reference position is performed at rapid traverse for each axis (non–linear positioning). The floating reference position return completion signal turns to “1” after completing the floating reference position return.

Signal

Floating reference position return end signal FRP1 to FRP8 <F116>

[Classification] Output signal

[Function] Notify the system that the tool is at the floating reference position on a controlled axis.

A floating reference position return end signal is provided for each axis. The number appended to each signal name indicates the number of the controlled axis.

FRP 1
1 : Floating reference position return end signal of 1st axis
2 : Floating reference position return end signal of 2nd axis
3 : Floating reference position return end signal of 3rd axis
:
:

[Output condition] The signals are set to 1 in the following case:

· When the tool is positioned to within the in–position area after floating reference position return (G30.1)

These signals are set to "0" when:

· The tool is moved from the floating reference position

· An emergency stop is applied.

· A servo alarm is raised.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F116	FRP8	FRP7	FRP6	FRP5	FRP4	FRP3	FRP2	FRP1

Parameter

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

[Data type] Bit

FPC When the floating reference position is specified using soft keys on the current position display screen
0 : The value of the displayed relative position is not preset. (In other words, the value does not change.)
1 : The value of the displayed relative position is preset to 0.

1244	Coodinates of the floating reference positon for each axis

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

This parameter specifies the coordinates of the floating reference position for each axis. The parameter is automatically set when the floating reference position is specified using soft keys on the current position display screen.

Reference Item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.7	FLOATING REFERENCE POSITION RETURN (G30.1)
	III.11.1.7	Setting the Floating Reference Position
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.7	FLOATING REFERENCE POSITION RETURN (G30.1)
	III.11.1.7	Setting the Floating Reference Position

5 AUTOMATIC OPERATION

5.1 CYCLE START/FEED HOLD

General

- **Start of automatic operation (cycle start)**

When automatic operation start signal ST is set to 1 then 0 in either memory (MEM) mode 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 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 starting
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 handle feed (HND), incremental feed (INC), or jog feed (JOG)

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 manual data input (MDI) 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 ”Signals” 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.

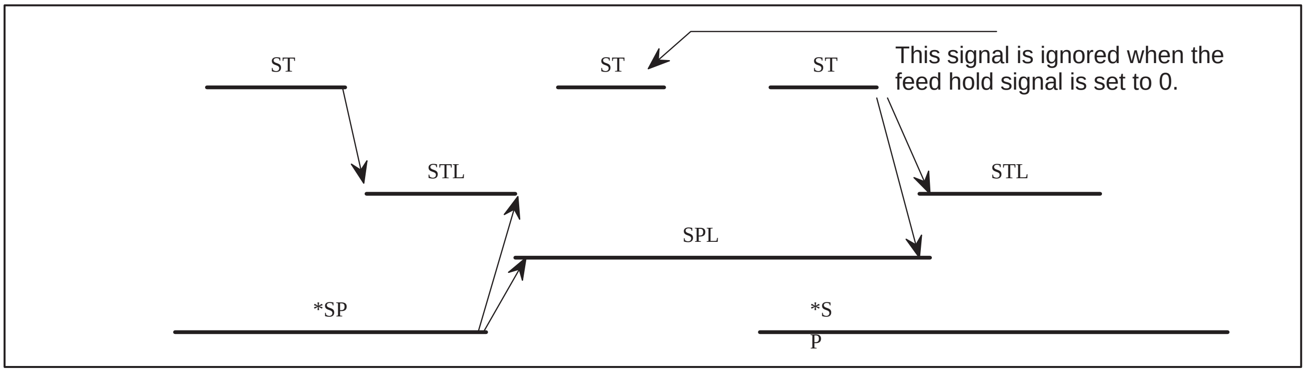


Fig. 5.1 Time Chart for Automatic Operation

When signal *SP is set to 0 during the execution of a block containing only the M, S, or T 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.
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 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 cannec 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.

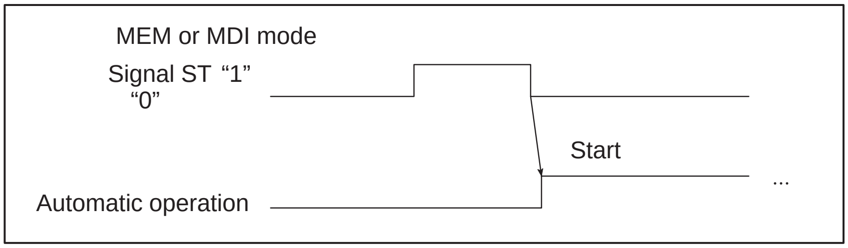
Signal

Cycle start signal ST <G007#2>

[Classification] Input signal

[Function] Starts automatic operation.

[Operation] When signal ST is set to 1 then 0 in memory (MEM) mode or manual data input (MDI) mode, the CNC enters the cycle start state and starts operation.

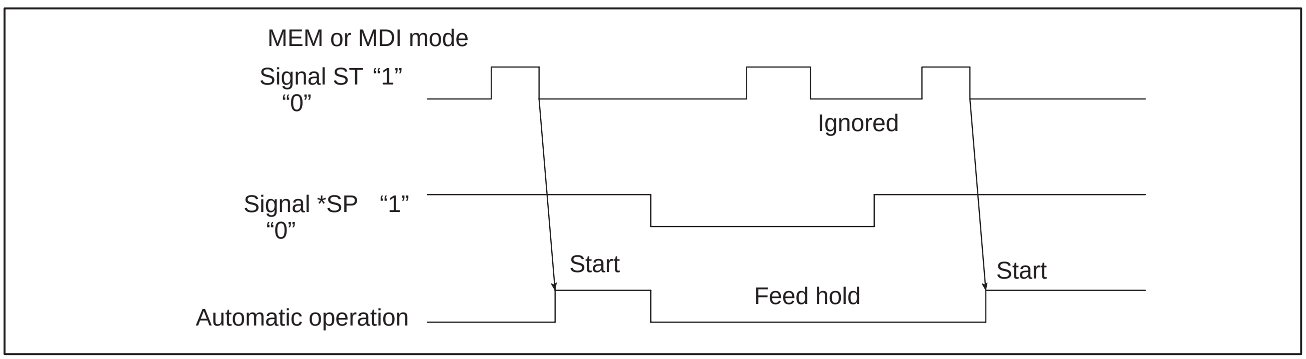


Feed hold signal *SP <G008#5>

[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.



Automatic operation signal OP <F000#7>

[Classification] Output signal

[Function] Notifies the PMC that automatic operation is in progress.

[Output condition] This signal is set to 1 or 0, according to the state of the CNC, as listed in Table 5.1.

Cycle start lamp signal STL <F000#5>

[Classification] Output signal

[Function] Notifies the PMC that automatic operation start is entered.

[Output condition] This signal is set to 1 or 0, according to the state of the CNC, as listed in Table 5.1.

Feed hold lamp signal SPL <F000#4>

[Classification] Output signal

[Function] Notifies the PMC that feed hold state is entered.

[Output condition] This signal is set to 1 or 0, according to the state of the CNC, as listed in Table 5.1.

Table 5.1 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

- Cycle start state
  The CNC is executing memory operation or manual data input operation commands.
- Feed hold state
  The CNC is not executing memory operation nor manual data input operation commands while the commands to be executed remain.
- Automatic operation stop state
  Memory operation or manual data input operation has been completed and stopped.
- Reset state
  The automatic operation has been forcibly terminated.

Note 1 If the sequence number search is performed through MDI panel during Memory mode (MEM), the signal op turns to “1”.

- **Signal address**

	#7	#6	#5	#4	#3	#2	#1	#0
G007						ST		
G008			*SP					
F000	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 000 to 015).

Detailed information on the automatic operation stop or feed hold state can also be displayed (diagnosis numbers 020 to 025).

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.4.1	MEMORY OPERATION
	III.4.2	MDI OPERATION
	III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.4.1	MEMORY OPERATION
	III.4.2	MDI OPERATION
	III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN

5.2 RESET AND REWIND

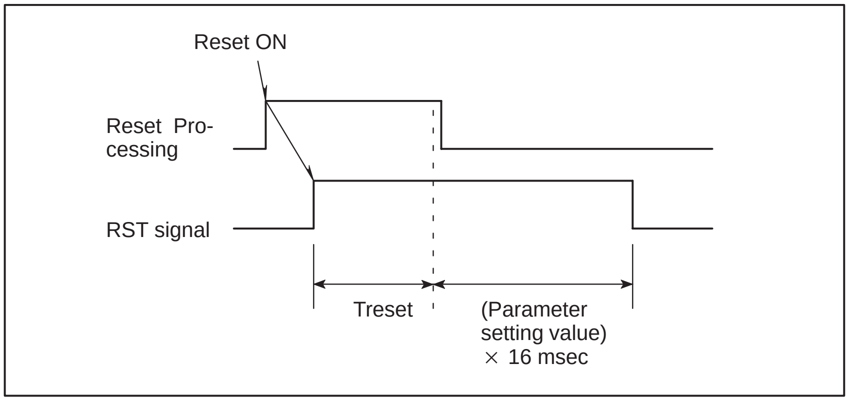
General

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.



Note) 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, or T function, signal MF, SF, or TF is set to 0 within 100 ms.

Tool movement along the controlled axis is also decelerated and stopped(*1) in jog feed (JOG), manual handle feed (HND), and incremental feed (INC) mode.

*1 When the emergency stop signal (*ESP) is set to 0, the tool is stopped by an emergency stop.

Bit 6 (CLR) of parameter No. 3402 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 (B–62444E or B–62454E), 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
  Whether programs created in MDI mode are erased or stored

- Bit 6 (CCV) of parameter No. 6001
  Whether custom macro variables #100 to #149 (#499) are cleared or stored
- Bit 7 (CLV) of parameter No. 6001
  Whether custom macro local variables #1 to #33 are cleared or stored

● Reset & Rewind

When the reset & rewind signal (RRW) is set to 1, reset is performed and the following rewinding operation is also performed.

1. When the DNC operation selection signal (DNC1) is 1 in automatic operation mode, and a portable tape reader is connected as the current input/output device, the tape reader is rewound.

   While the tape reader is being rewound, the rewinding–in–progress signal (RWD) is output. This signal goes 0 when the tape reader has been rewound.
2. In cases other than case 1, the head of the selected main program is searched for. Setting RWM, bit 2 of parameter no. 3001, determines whether the rewinding–in–progress signal is output.

   When RWM is set to 1:

   The rewinding–in–progress signal is output. It is set to 1, then set to 0 after about 100 ms. Since searching for the main program in memory takes little time, when the rewinding–in–progress signal (RWD) is set to 0, the main program has already been searched for.

Signal

External reset signal ERS<G008#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<G008#6>

[Classification] Input signal

[Function] CNC is reset and a program under an automatic operation is rewound.

[Operation] As described in the item, “RESET AND REWIND”.

Resetting signal RST <F001#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 condition] 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 <F000#0>

[Classification] Output signal

[Function] Notifies the PMC that the CNC is being rewound.

[Output condition] As described in the item, "RESET AND REWIND".

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G008	ERS	RRW						
F000								RWD
F001							RST	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3001						RWM		

[Data type] Bit

RWM RWD signal indicating that rewinding is in progress

0: Output only when the tape reader is being rewound by the reset and rewind signal RRW

1: Output when the tape reader is being rewound or a program in memory is being rewound by the reset and rewind signal RRW

3017	Output time of reset signal RST

[Data type] Byte

[Unit of data] 16 ms

[Valid data range] 0 to 255

To extend the output time of reset signal RST, the time to be added is specified in this parameter.
RST signal output time = time required for reset + parameter value × 16 ms

	#7	#6	#5	#4	#3	#2	#1	#0
3203	MCL							

[Data type] Bit

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						

[Data type] Bit

CLR Reset button on the CRT/MDI panel, external reset signal, reset and rewind signal, and emergency stop signal
0: Cause reset state.
1: Cause clear state.

	#7	#6	#5	#4	#3	#2	#1	#0
6001	CLV	CCV						

[Data type] Bit

CCV Custom macro's common variables Nos. 100 through 149
0: Cleared to "vacant" by reset
1: Not cleared by reset

CLV Custom macro's local variables Nos. 1 through 33
0: Cleared to "vacant" by reset
1: Not cleared by reset

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	APPEN-DIX E	STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET
OPERATOR'S MANUAL (For Lathe) (B–62444E)	APPEN-DIX E	STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET

5.3 TESTING A PROGRAM

Before machining is started, the automatic running check can be executed. It checks whether the created program can operate the machine as desired. This check can be accomplished by running the machine actually or viewing the position display change without running the machine.

5.3.1 Machine Lock

General

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 MLK8 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 <G044#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 MLK8 <G108>

[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 8th) in manual or automatic operation.

All–axis machine lock check signal MMLK <F004#1>

[Classification] Output signal

[Function] Notifies the PMC of the state of the all–axis machine lock signal.

[Output condition] 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
G044							MLK	
G108	MLK8	MLK7	MLK6	MLK5	MLK4	MLK3	MLK2	MLK1
F004							MMLK	

Note

Note 1 Automatic operation in the machine lock state (M, S, and T 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, and T commands are also performed normally.

Note 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 ZP4) are not output.
The reference position return check command (G27) is ignored in the machine lock state.

Note 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

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.5.1	MACHINE LOCK AND AUXILIARY FUNCTION LOCK
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.5.1	MACHINE LOCK AND AUXILIARY FUNCTION LOCK

5.3.2 Dry Run

General

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.

*1 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	Feed
1	Rapid traverse rate	Max. cutting feedrate
0	Dry run speed × JV,or rapid traverse rate*1	Dry run feedrate × JV

Max. cutting feedrate Setting by parameter No.1422
Rapid traverse rate Setting by parameter No.1420
Dry run feedrate Setting by parameter No.1410
JV Manual feedrate override

*1:Dry run feedrate x JV when parameter RDR (bit 6 of No. 1401) is 1. Rapid traverse rate when parameter RDR is 0.

Signal

Dry run signal DRN <G046#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.

Note 1 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 <F002#7>

[Classification] Output signal

[Function] Notifies the PMC of the state of the dry run signal.

[Output condition] 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
G046	DRN							
F002	MDRN							

Parameter

[Data type] Bit

TDR Dry run during threading or tapping (tapping cycle G74 or G84; rigid tapping)
0 : Enabled
1 : Disabled

RDR Dry run for rapid traverse command
0 : Disabled
1 : Enabled

1410	Dry run rate

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800

Set the dry run rate when the manual feedrate is overridden by 100%.

1420	Rapid traverse rate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 240000	6 – 100000
Inch machine	0.1 inch/min	30 – 96000	6 – 48000
Rotation axis	1 deg/min	30 – 240000	30 – 100000

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1422	Maximum cutting feedrate for all axes

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

Specify the maximum cutting feedrate.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

Note 1 To specify the maximum cutting feedrate for each axis, use parameter No. 1430 instead.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.5.4	Dry run
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.5.4	Dry run

5.3.3 Single Block

General

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 bit 5 (SBM) of parameter No. 6000, as follows:

SBM = 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.

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 (HND), incremental feed (INC), or jog feed (JOG), by using the mode select signals (MD1, MD2, and MD4).

Signal

Single block signal SBK <G046#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 <F004#3>

[Classification] Output signal

[Function] Notifies the PMC of the state of the single block signal.

[Output condition] 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
G046							SBK	
F004					MSBK			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6000			SBM					

[Data type] Bit

SBM Custom macro statement
0: Not stop the single block
1: Stops the single block

Note

Note 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.

Note 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

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.5.5	Single block
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.5.5	Single block

5.4 MANUAL ABSOLUTE ON/OFF

General

This function selects whether the movement of the tool with manual operation (such as jog feed and manual handle feed) is counted for calculating the current position in the workpiece coordinate system. A check signal is also output to indicate whether the manual absolute function in the CNC is turned on or off.

When manual absolute turns on (manual absolute signal *ABSM =0)

When manual operation interrupts during automatic operation:

i) At the end of the block where manual operation interrupts, the tool position moves in parallel by the manual move amount, regardless of the absolute or incremental command.

ii) In subsequent blocks, the parallel-moved tool position remains unchanged until an absolute command block appears. Therefore, if all blocks are programmed by incremental commands, the tool keeps the parallel-moved position until machining ends.

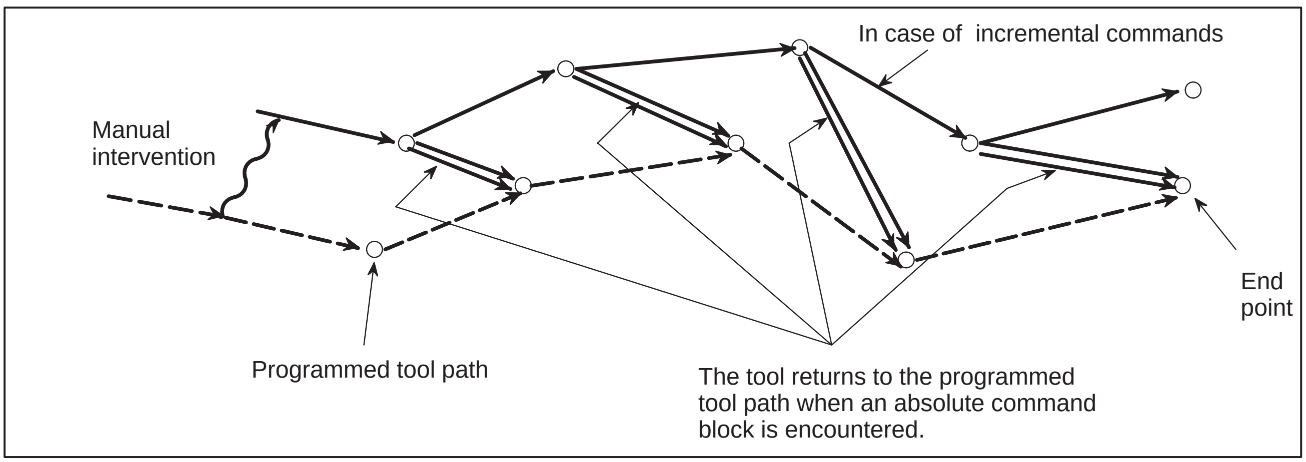


Fig. 5.4 (a) Manual absolute ON

Note 1 If the machining end position has shifted by the manual move amount because all blocks are programmed by incremental commands only, the present value is displayed shifted by the manual move amount.

When manual absolute turns off (manual absolute signal *ASM=1)

The manual move amount is not counted to the present position on the workpiece coordinate system. The present value display on the CRT includes the manual move amount. The display is reset to the initial value (before manual operation) when the control is reset, or when operation in the MEM or MDI mode is started after the manual operation.

During automatic operation, if manual intervention of a block interrupts, the tool position moves in parallel by the manual move amount, regardless of the absolute or incremental command at the end point of that block, as well as at the end point of subsequent blocks.

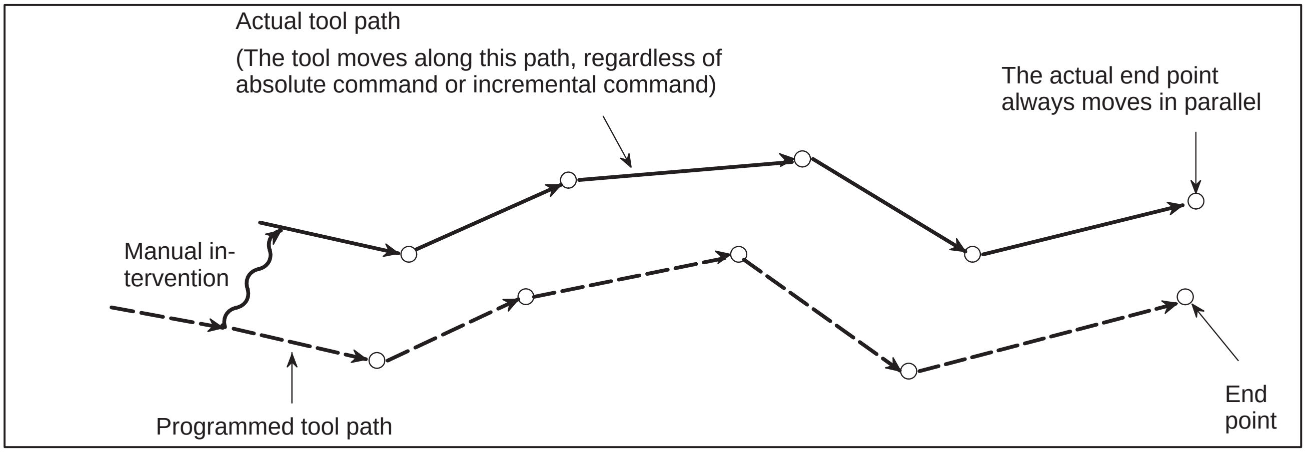


Fig. 5.4 (b)   Manual absolute OFF

The present value display at the finish of the operation shows an end point value on the program as if manual intervention had not been executed. However, the tool position moves in parallel.

Signal

Manual absolute signal *ABSM <G006#2>

[Classification] Input signal

[Function] Turns the manual absolute function on or off.

[Operation] When this signal is set to 1, the control unit operates as follows:
– Turns off the manual absolute function.

When this signal is set to 0, the control unit operates as follows:
– Turns on the manual absolute function.

Manual absolute check signal MABSM <F004#2>

[Classification] Output signal

[Function] Notifies the PMC of the state of the manual absolute signal.

[Output condition] This signal is set to 1 in the following case:
– When the manual absolute signal *ABSM is set to 1

This signal is set to 0 in the following case:
– When manual absolute signal *ABSM is set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G006						*ABSM		
F004						MABSM		

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.3.5	Manual absolute ON/OFF
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.3.5	Manual absolute ON/OFF

5.5 OPTIONAL BLOCK SKIP/ADDITION OF OPTIONAL BLOCK SKIP

General

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. Y200. ;

Input signal	Code specified at teh head of a block
BDT1	/ or /1 (Note 1)
BDT2	/2
BDT3	/3
BDT4	/4
BDT5	/5
BDT6	/6
BDT7	/7
BDT8	/8
BDT9	/9

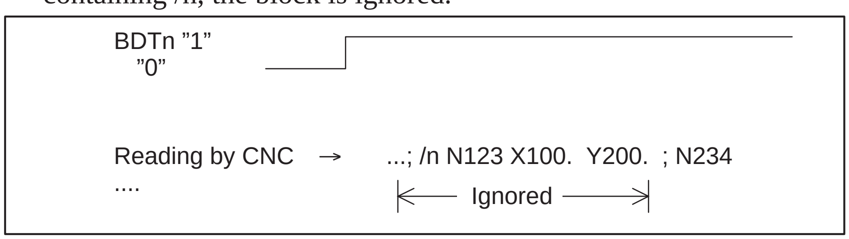
Note 1 Number 1 for /1 can be omitted. However, when two or more optional block skip switches are used in one block, number 1 for /1 cannot be omitted.

(Example)

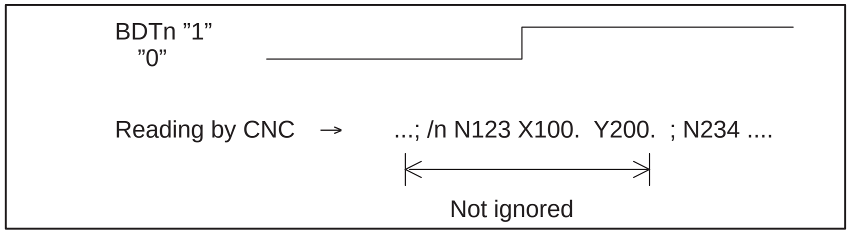
//3 N123 X100. Y200. ; — Invalid
/1 /3 N123 X100. Y200. ;— Valid

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:

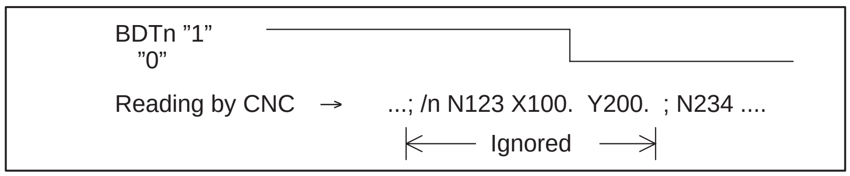
1. When BDTn is set to 1 before the CNC starts reading a block containing /n, the block is ignored.



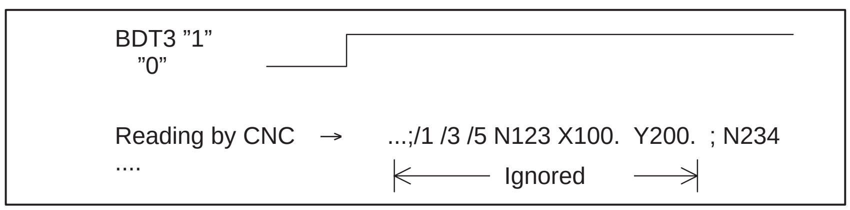
2. When BDTn is set to 1 while the CNC is reading a block containing /n, the block is not ignored.



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 <G044#0> BDT2 to BDT9 <G045>

[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 <F004#0> MBDT2 to MBDT9 <F005>

[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 condition] 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
G044								BDT1
G045	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2
F004								MBDT1
F005	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.

Note 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.

Note 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

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.13.2	Program section configuration
OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.13.2	Program section configuration

5.6 SEQUENCE NUMBER COMPARISON AND STOP

General

During program execution, this function causes a single block stop right after a block with a specified sequence number is executed.

To use this function, first specify the program number (1 to 9999) of a program that contains a sequence number where operation is to be stopped and the sequence number on the setting data screen:
With this setting, a single block stop occurs after the execution of the block with the specified sequence number during automatic operation.

Setting data

Setting data

– SEQUENCE STOP (PROGRAM NO.)
  Specify the program number (1 to 9999) of a program to which a sequence to be stopped belongs.
– SEQUENCE STOP (SEQUENCE NO.)
  Specify the sequence number (1 to 99999) of a sequence to be stopped.

Note

Note 1 After the specified sequence number is found during the execution of the program, the sequence number set for sequence number compensation and stop is decremented by one. When the power is turned on, the setting of the sequence number is 0.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.11.4.4	Sequence Number Comparison and Stop
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.11.4.8	Sequence Number Comparison and Stop

5.7 PROGRAM RESTART

General

A program may be restarted at a block by specifying the sequence number of the block, after automatic operation is stopped because of a broken tool or for holidays. This function can also be used as a high–speed program check function.

There are two types of restart methods.

P type: Restart after a tool is broken down
Q type: Restart after holidays

Signal

Program restart signal SRN<G006#0>

[Classification] Input signal

[Function] Selects program restart.

[Operation] When the program restart signal is set to logical 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 logical 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<F002#4>

[Classification] Output signal

[Function] Indicates the program is being restarted.

[Output condition] The program restart under way signal becomes logical 1 when:

– The program restart signal is set to logical 0 after the CRT screen changes to the program restart screen.

The signal is reset to logical 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
G006								SRN
G002				SRNMV				

Parameter

7310	Movement sequence to program restart position

Setting entry is accepted.

[Data type] Byte

[Valid data range] 1 to no. of controlled axes

This parameter sets the axis sequence when the machine moves to the restart point by dry run after a program is restarted.

[Example]
The machine moves to the restart point in the order of the fourth, first, second, and third axes one at a time when the first axis = 2, the second axis = 3, the third axis = 4, and the fourth axis = 1 are set.

Alarm and message

Number	Message	Description
094	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.
095	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.)
096	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.)
097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be specified when the program is restarted. (After power ON, after emergency stop or P / S alarm 94 to 97 were reset, no automatic operation was performed.) Perform automatic operation.

Number	Message	Description
098	G28 FOUND IN SE-QUENCE 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.
099	MDI EXEC NOT AL-LOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI.

Note

Note 1 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.

·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 specified for an intermediate block for a multiple repetitive canned cycle

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.4.3	PROGRAM RESTART
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.4.3	PROGRAM RESTART

5.8 TOOL RETRACTION AND RETURN (M series)

General

The tool can be retracted from a workpiece to replace the tool, if damaged during machining, or to check the status of machining. Then, the tool can be returned to restart machining efficiently.

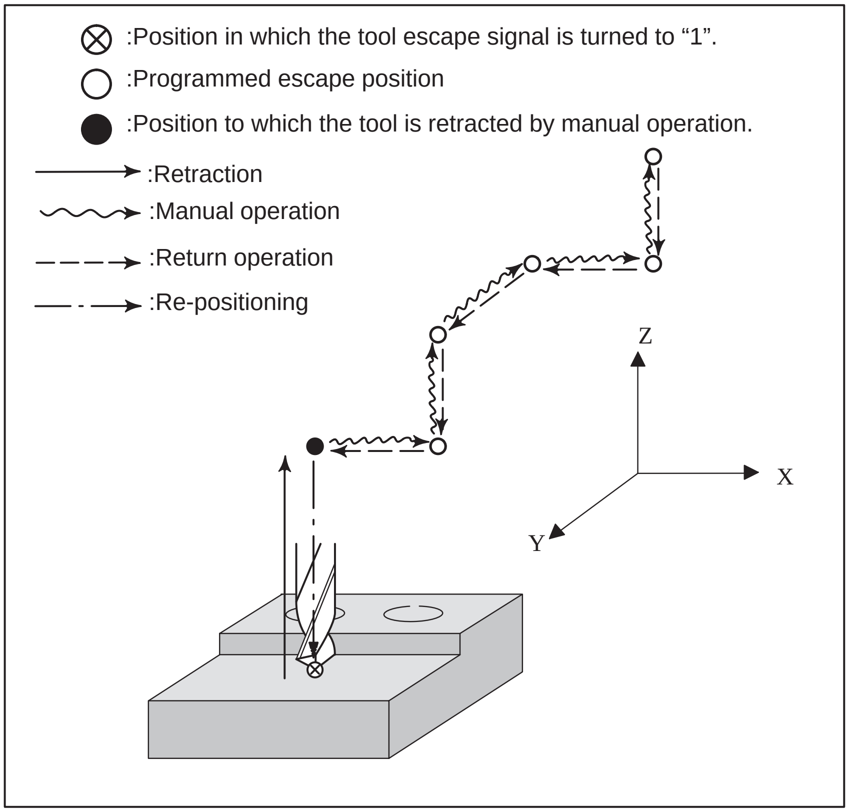
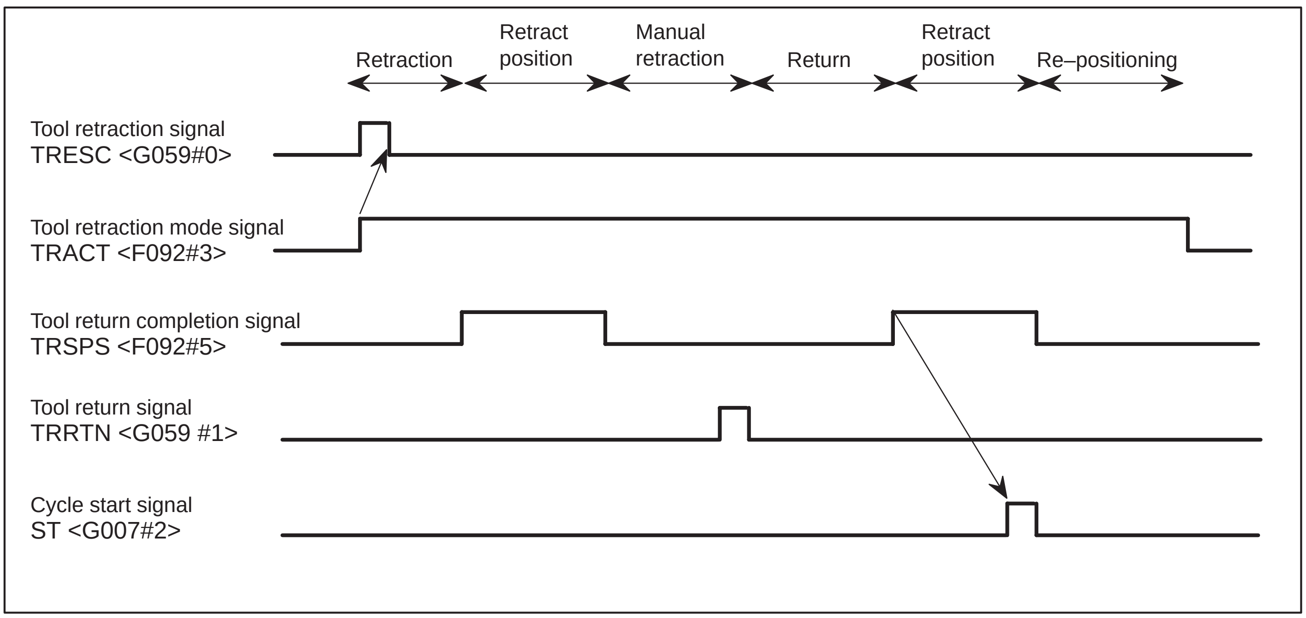


FIg. 5.8 Path of tool retraction and return for M series

Basic procedure for tool retraction and return

- When tool retraction signal TRESC input from the PMC turned to “1” while automatic operation is started, stopped, or held , the tool is retracted by the distance specified in the program. The position to which the tool retracts is called the retraction position. When TRESC turned to “1” while automatic operation is started, if a block is being executed, the tool is retracted after the block is suspended. When retraction ends, the machine enters the automatic operation hold state. The clearance and direction can be specified by the program. If these values are not specified by the program, the tool is not retracted.
  When TRESC turned to “1”, the machine enters the tool retraction mode, and tool retraction mode signal TRACT turned to “1” to notify the PMC that the machine has entered the tool retraction mode.

- In the manual mode, when it is necessary to replace the tool or measure workpieces, the tool can be moved manually, such as by manual continuous feed, or manual handle feed. This operation is called manual retraction. The path along which the tool retracts is automatically stored in the control unit. The number of paths which can be stored, however, is limited.
- Set automatic operation mode again. Set the tool return signal TRRTN to 1, then to 0. Then, the control unit traces back the path of the manually moved tool to automatically return the tool to the retract position. This operation is called return. When the tool has been returned to the retraction position, tool retraction completion signal TRSPS is turned to “1”.
- When the cycle is started at the retraction position, the tool is first returned to the position where TRESC was turned to “1”. This operation is called repositioning. When repositioning completes, TRACT is turned to “0” to notify the PMC of the end of the tool retraction mode. Then the operation differs according to the status of automatic operation when the machine entered the tool retraction mode.
  - ❑ When the machine enters the tool retraction mode while automatic operation is started, the automatic operation that has been held is resumed after the tool is repositioned.
  - ❑ When the machine enters the tool retraction mode while automatic operation is stopped or held, the machine returns to the same mode after the tool is repositioned. If the cycle is restarted, automatic operation is resumed.



Signal

Tool retraction signal TRESC<G059#0>

[Classification] Input signal

[Function] Tool retraction mode is selected.

[Operation] When this signal is turned to 1, the control unit retracts the tool by a pre–programmed distance.

Tool retraction mode signal TRACT<F092#3>

[Classification] Output signal

[Function] This signal reports that tool retraction mode is set. When the control unit is reset while the signal is 1, the signal is turned to 0, and tool retraction mode is canceled.

[Output condition] The signal is turned to 1 when:
- Tool retraction mode is selected.

The signal is turned to 0 when:
- Tool retraction mode is not selected.

Tool return signal TRRTN<G059#1>

[Classification] Input signal

[Function] In tool retraction mode, a tool that has been retracted manually along an axis is returned to the retract position along the same axis.

[Operation] When this signal is turned to 1, the control unit traces back the path of the manually moved tool to automatically return the tool to the retract position.

Tool return completion signal TRSPS<F092#5>

[Classification] Output signal

[Function] This signal reports that the tool is in the retract position in tool retraction mode. When this signal is 0, re–positioning cannot be performed by pressing the cycle start button.

[Output condition] The signal is set to 1 when:
- Retraction has been completed.
- The tool has been returned to the retract position.

The signal is set to 0 when:
- The tool is not in the retract position in tool retraction mode.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G059							TRRTN	TRESC

	#7	#6	#5	#4	#3	#2	#1	#0
F092			TRSPS		TRACT			

Note

Note 1 The retraction axes and retraction distances specified with G10.6 need to be changed in appropriate blocks depending on the figure to be machined. An incorrectly specified retraction distance may damage a workpiece, the machine, or the tool. So, be very careful when specifying a retraction distance.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III. 4.8	Tool Retraction and Return

5.9 EXACT STOP/EXACT STOP MODE/ TAPPING MODE/ CUTTING MODE (M SERIES)

General

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 inposition check is performed per block thereafter. This G61 is valid till G62 (automatic corner override), G63 (tapping mode), or G64 (cutting mode), is commanded.

• Tapping Mode (G63)

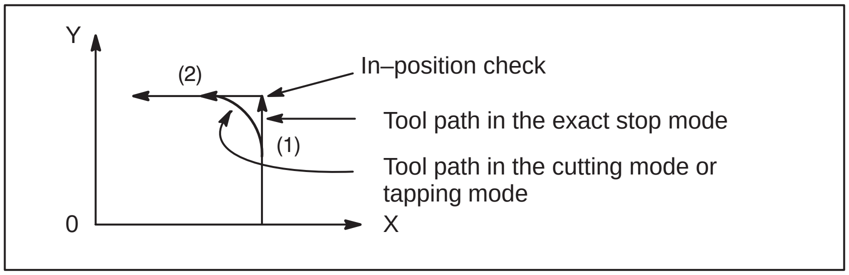
When G63 is commanded, feed rate override is ignored (always regarded as 100%), and feed hold also becomes invalid. Cutting feed does not decelerate at the end of block to transfer to the next block. This G63 is valid till G61 (exact stop mode), G62 (automatic corner override), or G64 (cutting mode) 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 till 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 7.2.6.1 and 7.2.6.2 for details.

(Example) Tool paths from block (1) to block (2)



Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.5.4.1	Exact Stop (G09, G61) Cutting Mode (G64) Tapping Mode (G63)

5.10 BALANCE CUT (2–PATH CONTROL FOR T SYSTEM)

General

When a thin workpiece is to be machined as shown in fig. NO TAG, a precision machining can be achieved by machining each side of the workpiece with a tool simultaneously;this function can prevent the workpiece from distortion that results when only one side is machined at a time. When both sides are machined at the same time, the movement of one tool must synchronize with that of the other tool. Otherwise, the workpiece may vibrate, resulting in poor machining. With this function, the movement of one tool post can easily synchronize with that of the other tool post.

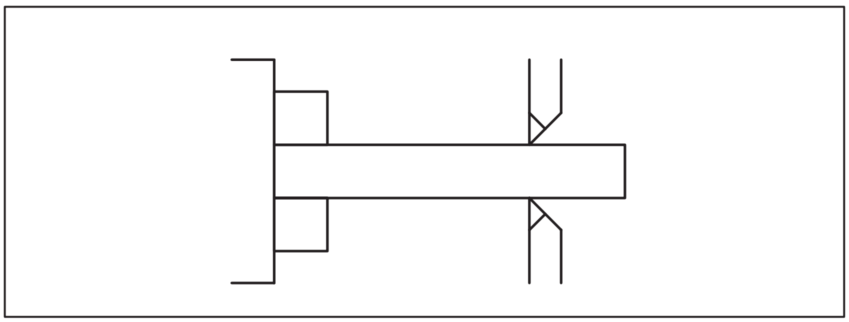


Fig. NO TAG Balance cut

Alarm and message

No.	Message	Contents
163	COMMAND G68/G69 INDEPENDENTLY (2–system control)	G68 and G69 are not independently commanded in balance cut. Correct program.

Note

Note 1 Time delay before the pulse distribution of both tool posts is started is 2 ms or shorter.

Note 2 In the balance cut mode, synchronization is established at the start of a move block, so movement may momentarily stop.

Note 3 If feed hold operation is performed during balance cutting using both tool posts, balance cut processing is not performed at restart time, it is performed when the next move command is specified for both tool posts.

Note 4 Balance cutting is not performed in dry run or machine lock state.

Note 5 When rapid traverse operation is specified, balance cut processing is not performed.

Note 6 A workpiece for which thread cutting has been performed in the balance cut mode cannot be subjected to thread cutting in the cancel mode. Thread cutting starts at a different position.

Note 7 The cancel mode (G69) is set by a reset.

Note 8 When the option ”mirror image for double turrets” is selected, the balance cut function cannot be used.

Note 9 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 data for both tool posts, such as the travel distance and feedrate, must be the same.

Reference item

OPERATOR’S MANUAL (For Lathe) (B–62444E)	II. 21.4	Balance Cut (G68, G69)

5.11 DNC OPERATION

General

By activating automatic operation during the DNC operation mode (RMT), it is possible to perform machining (DNC operation) while a program is being read in via the reader/puncher interface, or remote buffer.
If the floppy cassette directory display option is available, it is possible to select files (programs) saved in an external input/output unit of a floppy format (Handy File, Floppy Cassettes, or FA card) and specify (schedule) the sequence and frequency of execution for automatic operation.
To use the DNC operation function, it is necessary to set the parameters related to the reader/punch interface, and remote buffer in advance.

Signal

DNC operation select signal DNCI<G043#5>

[Classification] Input signal

[Function] Selects the DNC operation mode (RMT).
To select the DNC operation mode (RMT), it is necessary to select the memory mode (MEM) and set the DNC operation select signal to logical 1 simultaneously.

[Operation] When the DNC operation select signal becomes logical 1, the control unit operates as follows:

– If the memory mode (MEM) has not been selected, the signal is ignored, and nothing happens.
– If the memory mode (MEM) has been selected, the DNC operation mode (RMT) is selected, and DNC operation becomes possible. In this case, the DNC operation selection confirm signal MRMT becomes logical 1.

DNC operation selection confirm signal MRMT<F003#4>

[Classification] Output signal

[Function] Indicates that the DNC operation mode (RMT) has been selected.

[Output condition] The DNC operation selection confirm signal becomes logical 1 when:
– The DNC operation mode (RMT) is selected.

The DNC operation selection confirm signal becomes logical 0 when:
– The DNC operation mode (RMT) is deselected.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G043			DNCI					
F003				MRMT				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0100			ND3					

Setting entry is accepted.

[Data type] Bit

ND3 In DNC operation, a program is:
0 : Read block by block. (A DC3 code is output for each block.)
1 : Read continuously until the buffer becomes full. (A DC3 code is output when the buffer becomes full.)

Note 1 In general, reading is performed more efficiently when ND3=1. This specification reduces the number of buffering interruptions caused by reading of a series of blocks specifying short movements. This reduces the cycle time.

Alarm and message

Number	Message	Description
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
210	CAN NOT COMAND M198/M199	M198 and M199 are executed in the schedule operation. Or M198 is executed in the DNC operation.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	III.4.4	SCHEDULING FUNCTION
OPERATOR’S MANUAL (For Lathe) (B–62444E)	III.4.4	SCHEDULING FUNCTION
CONNECTION MANUAL	13.1	READER/PUNCHER INTERFACE
	13.2	REMOTE BUFFER

5.12 MANUAL INTERVENTION AND RETURN

General

If the tool movement along the axes is stopped by a feed hold during automatic operation, then restarted after manual intervention such as tool exchange, the tool moves back to the point of intervention before automatic operation is resumed.
This function is easy to operate because unlike the program restart function or the tool retract and restore function, it is unnecessary to operate switches on the operator's panel or MDI keys.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7001								MIN

[Data type] Bit

MIN The manual intervention and return function is:
0 : Disabled.
1 : Enabled.

Note

Note 1 If you do not make manual intervention correctly according to the direction of machining and the shape of the workpiece, the machine and tool may be broken down. Use sufficient care.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.4.10	MANUAL INTERVENTION AND RETURN
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.4.8	MANUAL INTERVENTION AND RETURN

6 INTERPOLATION FUNCTION

6.1 POSITIONING

General

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 path for positioning is determined in the same manner as linear interpolation (G01). The tool moves at an appropriate speed so that positioning can be performed in the shortest time without the speed exceeding the rapid traverse rate for each axis.
- 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	

[Data type] Bit

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

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 240000	30 – 100000
Inch machine	0.1 inch/min	30 – 96000	30 – 48000
Rotaion axis	1 deg/min	30 – 240000	30 – 100000

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

Note

The rapid traverse rate cannot be specified in the address F.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.4.1	POSITIONING (G00)
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.4.1	POSITIONING (G00)

6.2 LINEAR INTERPOLATION

General

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$\alpha\underline{\alpha}\beta\underline{\beta}\gamma\underline{\gamma}\zeta\underline{\zeta}$ $F\underline{f}$;

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

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

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

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

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

The feedrate of the rotary axis is commanded in the unit of deg/min (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 axisβ (such as A, B, or C) are linearly interpolated, the feed rate is that in which the tangential feed rate in the αandβ 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 fromula, then the β –axis feedrate unit is changed to deg/min.

A calculation example is as follows.

(Example)

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

$$\frac{\sqrt{20^2 + 40^2}}{300} \doteq 0.14907 \text{ (min)}$$

The feed rate for the C axis is

$$\frac{40}{0.14907} \doteq 268.3 \text{ deg/min}$$

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

Parameter

1441	Cutting feedrate when the power is turned on

Setting entry is acceptable.

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 32767	6 – 32767
Inch machine	0.01 inch/min	6 – 32767	6 – 32767

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 in the NC command data.

1422	Maximium cutting feedrate for all axes

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

Specify the maximum cutting feedrate.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

Note 1 To specify the maximum cutting feedrate for each axis, use parameter No. 1430 instead.

1430	Maximum cutting feedrate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000
Rotaion axis	1 deg/min	6 – 240000	6 – 100000

Specify the maximum cutting feedrate for each axis.

A feedrate for each axis is clamped in cutting feed so that it does not exceed the maximum feedrate specified for each axis.

Note 1 This parameter is effective only in linear and circular interpolation. In polar coordinate, cylindrical, and involute interpolation, the maximum feedrate for all axes specified in parameter No. 1422 is effective.

Note 2 If the setting for each axis is 0, the maximum feedrate specified in parameter No. 1422 is applied to all axes and the feedrate is clamped at the maximum feedrate.

	#7	#6	#5	#4	#3	#2	#1	#0
3402								G01

[Data type] Bit

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

No.	Message	Description
011	NO FEEDRATE COMMAN-DED	Cutting feedrate was not commanded or the feedrate was inadequate. Modify the program.

Reference item

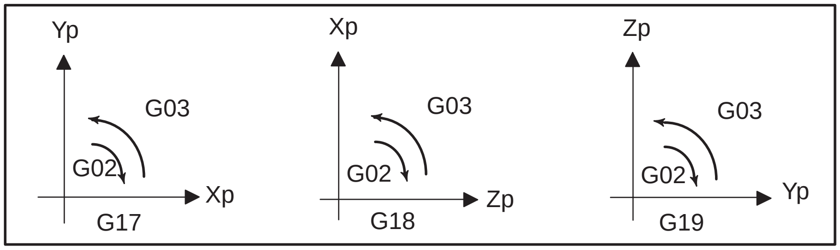
OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.4.2	LINEAR INTERPOLATION (G01)
OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.42.	LINEAR INTERPOLATION (G01)

6.3 CIRCULAR INTERPOLATION

General

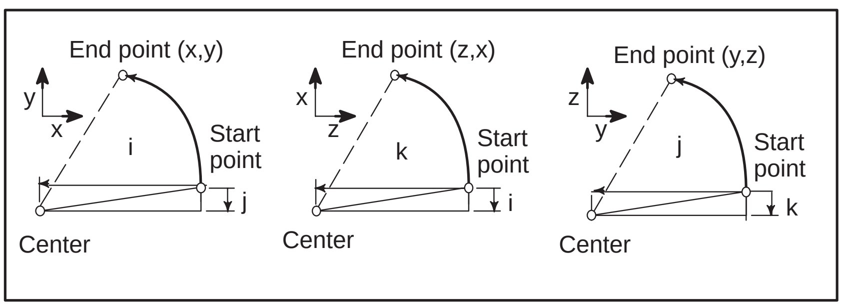
The command below can move a tool along a circular arc in the defined plane.

“Clockwise”(G02) and “counterclockwise”(G03) on the X_pY_p plane (Z_pX_p plane or Y_pZ_p plane) are defined when the X_pY_p plane is viewed in the positive–to–negative direction of the Z_p axis (Y_p axis or X_p 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.



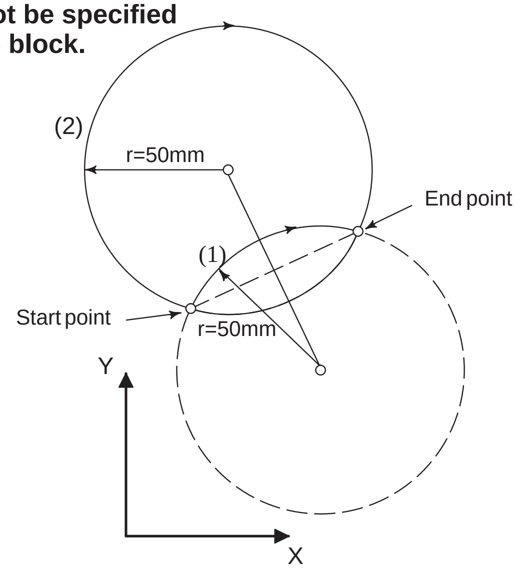
I0,J0, and K0 can be omitted. When X_p, Y_p , and Z_p are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360° arc (circle) is specified.
G02I; Command for a circle
If the difference between the radius at the start point and that at the end point exceeds the value in a parameter (No.3410), an alarm (No.020) 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.

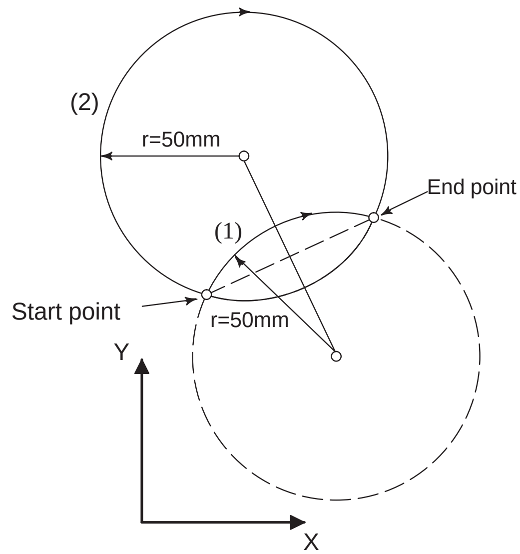
For T series, an arc with a sector angle of 180° or wider cannot be specified (Alarm 023).
For M series, specify an arc more than 180° with a negative radius value commanded.

If Xp, Yp, and Zp are all omitted, if the end point is located at the same position as the start point and when R is used, an arc of 0°is programmed.
G02Rr ; (The tool does not move.)

(Example) (T series)
For arc (1) **(less than 180°)**
G02 W60.0 U10.0 R50.0 F300.0 ;
For arc (2) **(greater than 180°)**
An arc with a sector angle of 180° or wider cannot be specified within a single block.



(Example) (M series)
For arc (1)**(less than 180°)**
G91 G02 X60.0 Y20.0 <u>R50.0</u> F300.0 ;
For arc (2)**(greater than 180°)**
G91 G02 X_P60.0 Y_P20.0 <u>R–50.0</u> 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.
The error between the specified feedrate and the actual tool feedrate is ±2% or less. However, this feedrate is measured along the arc after the cutter compensation (M series) or tool nose radius compensation (T series) is applied.

Parameter

1022	Setting of each axis in the basic coordinate system

Note 1 When this parameter is set, power must be turned off before operation is continued.

[Data type] Byte axis

To determine the following planes used for circular interpolation, cutter compensation C (for the M series), tool nose radius compensation (for the T series), etc., each control axis is set to one of the basic three axes X, Y, and Z, or an axis parallel to the X, Y, or Z axis.

G17: Plane Xp–Yp

G18: Plane Zp–Xp

G19: Plane Yp–Zp

Only one axis can be set for each of the three basic axes X, Y, and Z, but two or more parallel axes can be set.

Set value	Meaning
0	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

	#7	#6	#5	#4	#3	#2	#1	#0
3402								
						G19	G18	

[Data type] Bit

G18 and G19 Plane selected when power is turned on or when the control is cleared

G19	G18	G17, G18 or G19 mode
0	0	G17 mode (plane XY)
0	1	G18 mode (plane ZX)
1	0	G19 mode (plane YZ)

3410	Tolerance of arc radius

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1 to 99999999

When a circular interpolation command (G02, G03) is executed, the tolerance for the radius between the start point and the end point is set. If the difference of radii between the start point and the end point exceeds the tolerance set here, a P/S alarm No. 20 is informed.

Note 1 When the set value is 0, the difference of radii is not checked.

Alarm and message

Number	Message	Description
011	NO FEEDRATE COMMANDED	Cutting feedrate was not commanded or the feedrate was inadequate. Modify the program.
020	OVER TOLERANCE OF RADIUS	In circular interpolation (G02 or G03), difference of the distance between the start point and the center of an arc and that between the end point and the center of the arc exceeded the value specified in parameter No. 3410.
021	ILLEGAL PLANE AXIS COMMANDED	An axis not included in the selected plane (by using G17, G18, G19) was commanded in circular interpolation. Modify the program.
023	ILLEGAL RADIUS COMMAND (T series)	In circular interpolation by radius designation, negative value was commanded for address R. Modify the program.
025	CANNOT COMMAND F0 IN G02/G03 (M series)	F0 (rapid traverse) was instructed by F1 –digit command in circular interpolation. Modify the program.
028	ILLEGAL PLANE SELECT	In the plane selection command, two or more axes in the same direction are commanded. Modify the program.

Note

Note 1 The U, V and Waxes (parallel with the basic axis) can be used with G–code system B and C.

Note 2 If I, J, K, and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.

Note 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 (No.028)is displayed.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.4.3	CIRCULAR INTERPOLATION (G02,G03)
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.4.3	CIRCULAR INTERPOLATION (G02,G03)

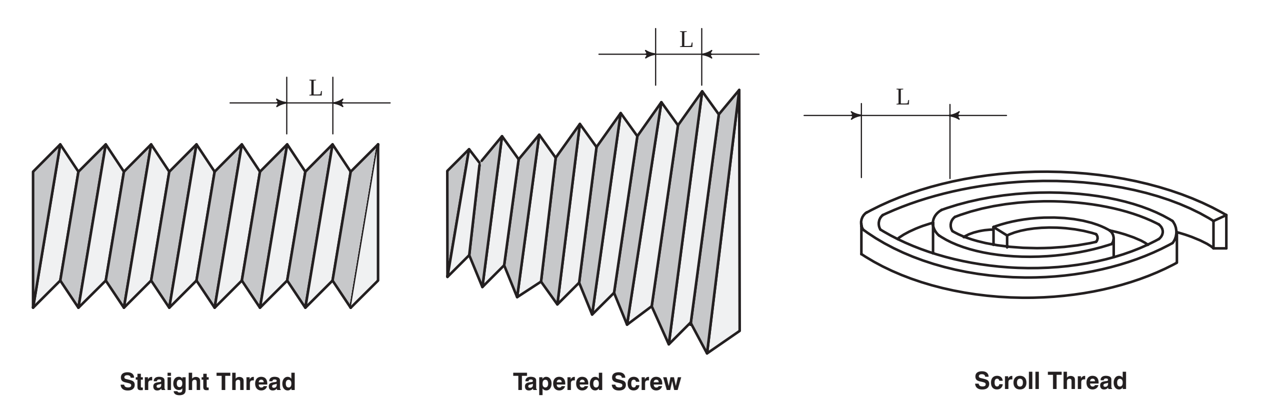
6.4 THREAD CUTTING

6.4.1 Thread Cutting

General

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

Thread cutting signal THRD<F002#3>

[Function] This signal indicates that thread cutting is in progress.

[Output condition] This signal turns to “1” in the following cases:
- Thread cutting mode in progress
- Thread cutting cycle for turning

This signal turns to “0” in the following case.
- Neither thread cutting mode nor thread cutting are in progress.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F002					THRD			

Parameter

Enabling/disabling dry run operation during threading

	#7	#6	#5	#4	#3	#2	#1	#0
1401			TDR					
			TDR					

[Data type] Bit

TDR Dry run during threading or tapping (tapping cycle G74 or G84, rigid tapping)
0 : Enabled
1 : Disabled

Checking the spindle speed arrival signal before starting threading

	#7	#6	#5	#4	#3	#2	#1	#0
3708							SAT	SAR
								SAR

[Data type] Bit

SAR: The spindle speed arrival signal is:
0 : Not checked
1 : Checked

SAT: Check of the spindle speed arrival signal at the start of executing the thread cutting block
0 : The signal is checked only when SAR, #0 of parameter 3708, is set.
1 : The signal is always checked irrespective of whether SAR is set.

Note 1 When thread cutting blocks are consecutive, the spindle speed arrival signal is not checked for the second and subsequent thread cutting blocks.

Setting the time constant for the threading cycle

1626	Time constant of exponetial acceleration/deceleration in the thread cutting cycle for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in the thread cutting cycle (G76, G78 (G92 in G code system A)) for each axis.

Setting the FL feedrate for the thread cutting cycle

1627	FL rate of exponential acceleration /deceleration in the thread cutting cycle for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in the thread cutting cycle (G76, G78 (G92 in G code system A)) for each axis.

Setting the chamfering distance for the thread cutting cycle

5130	Chamfering distance in the thread cutting cycles G76 and G92

[Data type] Byte

[Unit of data] 0.1 pitch

[Valid data range] 0 to 127

This parameter sets the chamfering in the thread cutting cycles G76 and G92.

Setting the minimum depth of cut for the multiple repetitive canned cycle G76

5140	Minimium depth of cut in the multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 99999999

This parameter sets the minimum depth of cut in the multiple repetitive canned cycle G76.

Setting the finishing allowance for the multiple repetitive canned cycle G76

5141	Finishing allowance in the multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1 to 99999999

This parameter sets the finishing allowance in the multiple repetitive canned cycle G76.

Setting the repetition count of finishing for the multiple repetitive canned cycle G76

5142	Repetition count of final finishing in the multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data] Cycle

[Valid data range] 1 to 99999999

This parameter sets the repetition count in the multiple repetitive canned cycle G76.

Setting the tool angle for the multiple repetitive canned cycle G76

5143	Tool nose angle in the multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data] Degree

[Valid data range] 0 to 120 (When FS15 TAPE FORMAT is used)
0, 29, 30, 55, 60, 80 (When FS15 TAPE FORMAT is not used)

This parameter sets the tool nose angle in the multiple repetitive canned cycle G76.

Note

- **Notes applicable to both the M and T series**

Notes

1. Feedrate override is ignored during thread cutting, 100% being assumed.
2. During threading, spindle override is ignored, 100% being assumed.
3. It is very dangerous to stop feeding the thread cutter without stopping the spindle. This will suddenly increase the cutting depth. Thus, the feed hold function is ineffective while thread cutting. If the feed hold button is pressed during thread cutting, the tool will stop after a block not specifying thread cutting is executed as if the SINGLE BLOCK button were pushed. However, the feed hold lamp (SPL lamp) lights when the FEED HOLD button on the machine control panel is pushed. Then, when the tool stops, the lamp is turned off (Single Block stop status).
4. 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.
5. When thread cutting is executed in the single block status, the tool stops after execution of the first block not specifying thread cutting.
6. When the previous block was a thread cutting block, cutting will start immediately without waiting for detection of the 1–turn signal even if the present block is a thread cutting block.
7. When a dry run operation is performed the dry run rate becomes the longitudinal axis feedrate.

- **Notes applicable to the T series only**

Notes

The thread cutting retract function is supported only for the threading cycle.

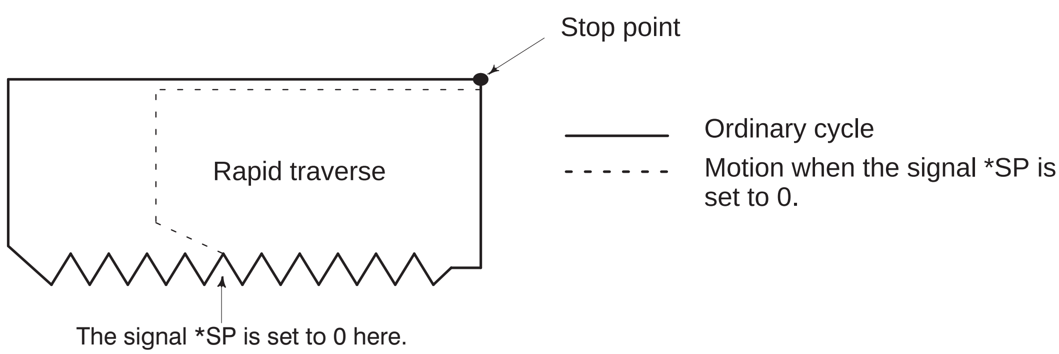
Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.4.10	CONSTANT LEAD THREAD CUTTING
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.4.7 II.4.8 II.4.9 II.4.10 II.14.1.2 II.14.2.7	CONSTANT LEAD THREAD CUTTING VARIABLE LEAD THREAD CUTTING CONTINUOUS THREAD CUTTING MULTIPLE THREAD CUTTING Thread Cutting Cycle Multiple Thread Cutting Cycle

6.4.2 Thread Cutting Cycle Retract (T series)

General

When the automatic operation stop signal *SP <G008#5> is set to 0 during threading in a threading cycle, the tool immediately retracts while performing chamfering, then returns to the start point of the current cycle, first along the X–axis, then along the Z–axis.



Parameter

- **Setting to enable the override function during thread cutting cycle retraction**

	#7	#6	#5	#4	#3	#2	#1	#0
1403	RTV							

[Data type] Bit

RTV Override while the tool is retracting in threading
0 : Override is effective.
1 : Override is not effective.

- **Setting a chamfering distance in thread cutting cycle retraction**

5130	Chamfering distance in thread cutting cycles G76 and G92

[Data type] Byte

[Unit of data] 0.1 pitch

[Valid data range] 0 to 127

This parameter sets the chamfering in thread cutting cycles G76 and G92.

Note

Note 1 While the tool is retracting, automatic operation stop signal *SP <G008#5> is ignored.

Note 2 The chamfering distance for retraction is determined by the setting of parameter No. 5130.

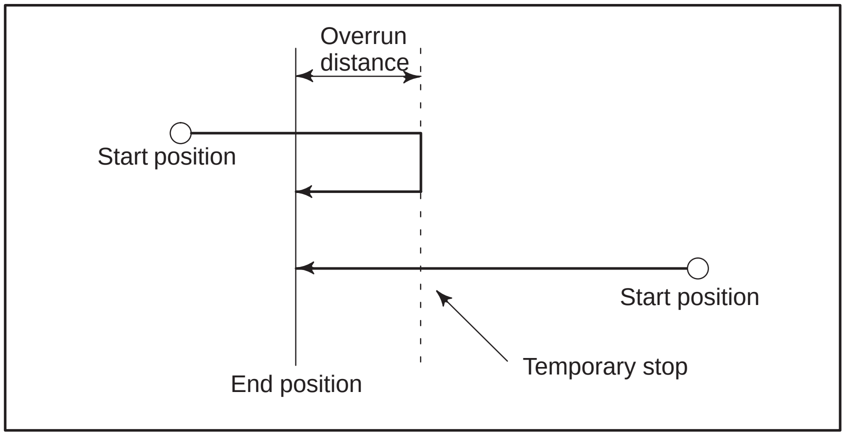
Reference item

OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.14.1.2	Thread Cutting Cycle
	II.14.2.7	Multiple Thread Cutting Cycle

6.5 SINGLE DIRECTION POSITIONING (M SERIES)

General

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



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.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5431								MDL

[Data type] Bit

MDL Specifies whether the G code for single direction positioning (G60) is included in one–shot G codes (00 group) or modal G codes (01 group)
0: One–shot G codes (00 group)
1: Modal G codes (01 group)

5440	
	Positioning direction and overrun distance in uni–directional positioning for each axis

[Data type] Word axis

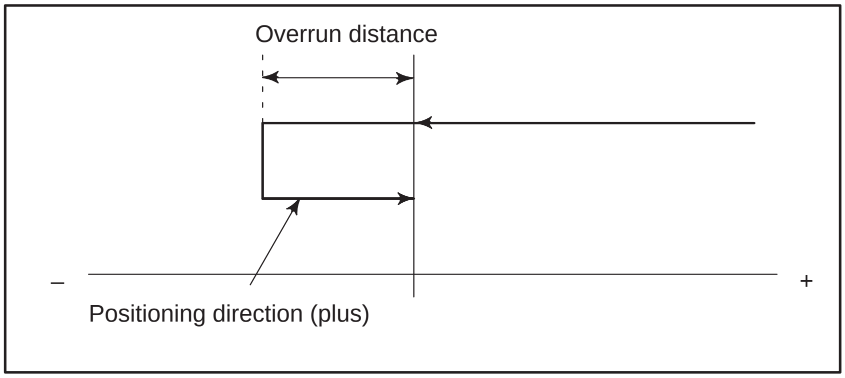
[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –16383 to +16383

This parameter sets the positioning direction and overrun distance in uni–directional positioning (G60) for each axis. The positioning direction is specified using a setting data sign, and the overrun distance using a value set here.

Approach > 0: The positioning direction is positive (+).
Approach < 0: The positioning direction is negative (−).
Approach = 0: Uni–directional positioning is not performed.



Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.4.2	Single direction positioning

6.6 HELICAL INTERPOLATION

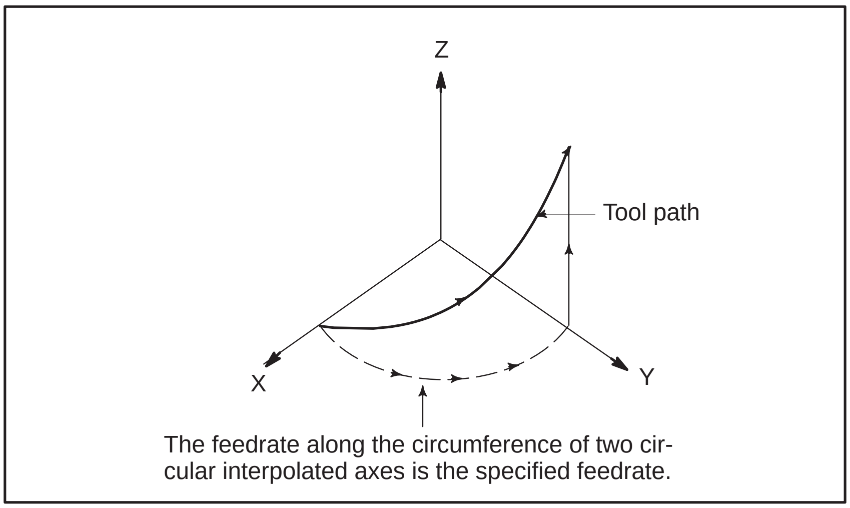
General

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

The command method is to simply add one or two move command axes which is not circular interpolation axes. An F command specifies a feedrate along a circular arc. Therefore, the feedrate of the linear axis is as follows:

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

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



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1404								HFC

HFC The feedrate for helical interpolation is:

0 : Clamped so that the feedrates along an arc and linear axis do not exceed the maximum cutting feedrate specified by parameter.

1 : Clamped so that the composite feedrate along an arc and linear axis does not exceed the maximum cutting feedrate specified by parameter.

When HFC is 1, and two linear axes exist, the combined feedrate for the four axes (two axes (arc) + two axes (straight line)) is clamped so that it does not exceed the maximum cutting feedrate.

Note 1 Parameters used for clamping

When HFC is 0

No. 1430:
Maximum cutting feedrate for each axis

Since the cutting feedrate for the arc is clamped to the above parameter value, the feedrate along the linear axis is clamped to the smaller parameter value.

Example: No. 1430
X 1000

Y 1200

Z 1400
G17 G03 X0. Y100. R100. Z1000. F5000;

The feedrate along the linear axis is clamped to 1000.

No. 1422:
Maximum cutting feedrate (common to all axes)

If parameter No. 1430 is set to 0, the feedrate is clamped to the value set in this parameter.

When HFC is 1

No. 1422:
Maximum cutting feedrate (common to all axes)

The cutting feedrate is clamped to the value set in this parameter. The value set with parameter No. 1430 is ignored.

Alarm and Message

If more than two axes are specified together with the two axes for circular interpolation in a block specifying a helical interpolation operation, alarm No. 232 is issued.

No.	Message	Description
0232	TOO MANY HELICAL AXIS COMMANDS	Three or more axes are specified as helical axes.

Reference Item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.4.5	Helical Interpolation
OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.4.4	Helical Interpolation

6.7 INVOLUTE INTERPOLATION (M SERIES)

General

With the involute interpolation function, an involute curve can be machined. Cutter compensation C is also possible. The use of involute interpolation eliminates the need to use short lines or arcs to approximate an involute curve. Pulse distribution is no longer interrupted by the high–speed operation of small blocks. As a result, smooth, high–speed operation is possible. In addition, part programs can be created more easily, and the required paper tape can be shortened.

Parameter

5610	Limit of initial permissible error during involute interpolation

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 99999999

This parameter sets the allowable limit of deviation between an involute curve passing through a start point and an involute curve passing through an end point for an involute interpolation command.

Alarm and Message

No.	Message	Description
P/S 241	END POINT, I, J, K, AND R ARE MISSING	The end point of an involute curve, I, J, or K is not specified.
P/S 242	SPECIFICATION ERROR (INVOLUTE)	An illegal value is specified for involute interpolation. (1)The specified start point or end point is located inside the base circle. (2)Zero is specified for I, J, K, or R. (3)The start point or end point is located more than 100 turns from the beginning of the involute curve.
P/S 243	END POINT NOT ON INVOLUTE CURVE	The end point is not on the involute curve that passes through the start point, and is beyond the range specified with in parameter No. 5610.

Reference Item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.4.8	Involute Interpolation

6.8 POLAR COORDINATE INTERPOLATION

General

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.

G12.1 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane (Fig. 6.8 (a)). Polar coordinate interpolation is performed on this plane.

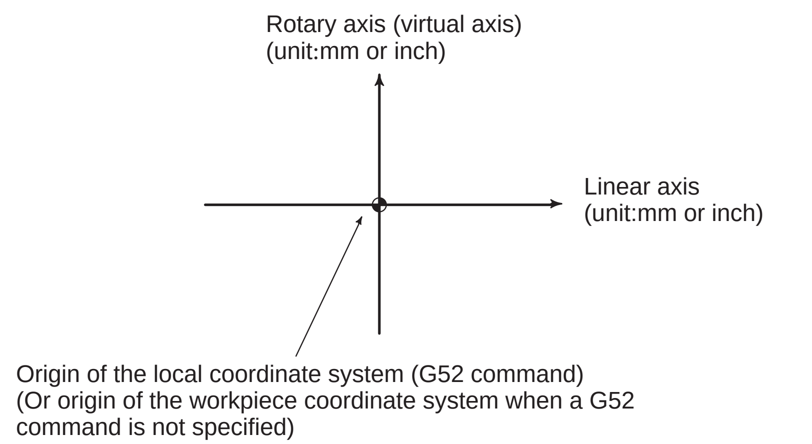


Fig. 6.8 (a) Polar coordinate interpolation plane

When the power is turned on or the system is reset, polar coordinate interpolation is canceled (G13.1).
The linear and rotation axes for polar coordinate interpolation must be set in parameters (No. 5460 and 5461) beforehand.

Parameter

1422	Maximum cutting feedrate for all axes

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

Specify the maximum cutting feedrate.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

Note 1 To specify the maximum cutting feedrate for each axis, use parameter No. 1430 instead.

5460	Axis (linear axis) specification for polar coordinate interpolation
5461	Axis (rotary axis) specification for polar coordinate interpolarion

[Data type] Byte

[Valid data range] 1, 2, 3, ... control axes count

These parameters set control axis numbers of linear and rotary axes to execute polar interpolation.

5462	Maximum cutting feedrate during polar coordinate interpolation

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0, 6 – 240000	0, 6 – 100000
Inch machine	0.1 inch/min	0, 6 – 96000	0, 6 – 48000
Rotation axis	1 deg/min	0, 6 – 240000	0, 6 – 100000

This parameter sets the upper limit of the cutting feedrate that is effective during polar coordinate interpolation. If a feedrate greater than the maximum feedrate is specified during polar coordinate interpolation, it is clamped to the feedrate specified by the parameter. When the setting is 0, the feedrate during polar coordinate interpolation is clamped to the maximum cutting feedrate usually specified with parameter 1422.

Alarm and Message

No.	Message	Description
145	ILLEGAL CONDITIONS IN POLAR COORDINATE INTERPOLATION	The conditions are incorrect when the polar coordinate interpolation starts or it is canceled. 1) In modes other than G40, G12.1/G13.1 was specified. 2) An error is found in the plane selection. Parameters No. 5460 and No. 5461 are incorrectly specified. Modify the value of program or parameter.

Reference Item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.4.6	Polar Coordinate Interpolation
OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.4.5	Polar Coordinate Interpolation

6.9 CYLINDRICAL INTERPOLATION

General

The amount of travel of a rotary axis specified by an angle is once internally converted to a distance of a linear axis along the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back to the amount of travel of the rotary axis.
The cylindrical interpolation function allows the side of a cylinder to be developed for programming. So programs such as a program for cylindrical cam grooving can be created very easily.

Use parameter No. 1022 to specify whether the rotation axis is the X–, Y–, or Z–axis, or an axis parallel to one of these axes.
Only one rotation axis can be set for cylindrical interpolation.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

Note 1 When this parameter is changed, turn off the power before continuing operation.

[Data type] Bit axis

ROTx, ROSx Setting linear or rotation axis

ROSx	ROTx	Description
0	0	Linear axis · Inch/metric conversion is done. · All coordinate values are linear axis type. (Not rounded in 0 to 360°) · Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotation axis (A type) · Inch/metric conversion is not done. · Machine coordinate values are rounded in 0 to 360° Absolute coordinate values and relative coordinate values are rounded or not rounded by parameter No. 1008#0 and #2. · Stored pitch error compensation is the rotation type (Refer to parameter No. 3624) · Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	0	Setting is invalid (unused)
1	1	Rotation axis (B type) · Inch/metric conversion is not done. · Machine coordinate values is linear axis type (Is not rounded in 0 to 360°). Absolute coordinate values and relative coordinate values are rounded or not rounded by parameter No. 1008#0 and #2. · Stored pitch error compensation is linear axis type (Refer to parameter No. 3624). · Cannot be used with the rotation axis roll over function and the index table indexing function (M series).

1022	Setting of each axis in the basic coordinate system

[Data type] Byte axis

To determine the following planes used for circular interpolation, cutter compensation C (for the M series), tool nose radius compensation (for the T series), etc., each control axis is set to one of the basic three axes X, Y, and Z, or an axis parallel to the X, Y, or Z axis.

G17: Plane Xp–Yp

G18: Plane Zp–Xp

G19: Plane Yp–Zp

Only one axis can be set for each of the three basic axes X, Y, and Z, but two or more parallel axes can be set.

Set value	Meaning
0	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

Alarm and Message

Number	Message	Description
175	ILLEGAL G107 COMMAND	Conditions when performing cylindrical interpolation start or cancel not correct. To change the mode to the cylindrical interpolation mode, specify the command in a format of “G07.1 rotation–axis name radius of cylinder.”
176	IMPROPER G–CODE IN G107	Any of the following G codes which cannot be specified in the cylindrical interpolation mode was specified. 1) G codes for positioning, such as G28, G76, G81 – G89, including the codes specifying the rapid traverse cycle 2) G codes for setting a coordinate system: G50, G52 3) G code for selecting coordinate system: G53 G54–G59 Modify the program.

Reference Item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.4.7	Cylindrical Interpolation
OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.4.6	Cylindrical Interpolation

6.10 POLYGONAL TURNING (T SERIES)

Polygonal turning means machining a polygonal figure by rotating the workpiece and tool at a certain ratio.

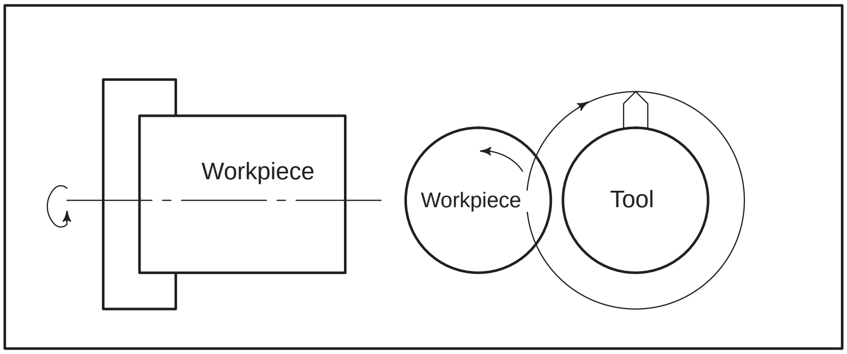


Fig. 6.10 (a) Polygonal turning

By changing conditions which are rotation ratio of workpiece and tool and number of cutters,the machining figure can be changed to a square or hexagon. The machining time can be reduced as compared with polygonal figure machining using C and X axes of the polar coordinate. The machined figure however, is not exactly polygonal. Generally, polygonal turning is used for the heads of square and/or hexagon bolts or hexagon nuts.

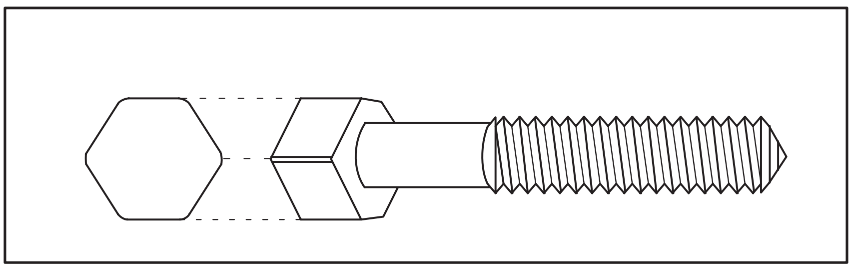


Fig. 6.10 (b) Hexagon bolt

This function controls the workpiece (spindle) and tool (rotation tool axis) so that the relationship between the spindle speed and tool speed is maintained at a constant ratio specified in a command given to the CNC.

(For the principle of polygonal turning, refer to Section 20.1, Part II of the “Operator’s Manual (For Lathe).”)

Either of the following can be selected as the tool rotation axis:

· CNC controlled axis (servo axis)

· Second spindle (with two serial spindles connected)

In the following descriptions, the term polygonal turning refers to a turning operation in which a servo axis is used as the tool rotation axis (See Section 6.10.1.).

The term polygonal turning with two spindle refers to a turning operation in which the second spindle is used as the tool rotation axis (See Section 6.10.2.).

6.10.1 Polygonal Turning

General

One of the axes (servo axes) controlled by the CNC is assigned as a tool rotation axis. Either serial spindle or analog spindle can be used as a workpiece axis (spindle).

Polygonal turning using a servo axis is detailed in the operator's manual (for lathe).

This section focuses on supplementary information and examples for the connection.

• Spindle connection

A position coder must be mounted on the spindle. However, polygonal turning requires no additional changes to the spindle connection (See Section 9.3.).

Polygonal 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 polygonal turning except for some parameters.

When the machine is not in the polygonal 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. Read the operator's manual (for lathe) and the following examples.

• Examples of parameter setting

· The following descriptions exemplify typical parameter setting for polygonal turning using a serial pulse coder (with a million pulse capability).

→ The parameter setting described here is not a must for polygonal turning.

→ Specify typical values for parameters unless otherwise stated.

- Tool rotation axis setting

  This example uses the CNC's fourth axis (connected as the Y–axis) as a rotation tool axis for polygonal 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.

- Servo parameter setting

  Set the servo parameters as listed below:

  CMR = 1
  DMR = 36/1000

(With the above setting, the reference counter capacity is 36000.)

Parameter No. 1820 = 2 (CMR)
Parameter No. 1821 = 36000 (reference counter capacity)
Parameter No. 2084 = 36 (DMR numerator)
Parameter No. 2085 = 1000 (DMR denominator)

For the other servo parameters, specify typical values.

- Parameter setting for polygonal turning

The least command increment, detection unit, the angle to rotate through per rotation for the polygon axis are as follows:

$$\text{Least command increment} = \frac{L \times CMR}{Q \times DMR}$$

$$\text{Detection unit} = \frac{\text{least command increment}}{DMR} = \frac{L}{Q \times DMR}$$

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 polygon axis. It is determined regardless of what is specified in parameter No. 1004 (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:

$$\text{Least command increment} = \frac{360 \times 1}{1000000 \times \frac{36}{1000}} = 0.01 \text{ (degrees)}$$

Detection unit = 0.01 (degrees)

$$\text{Angle to rotate through per tool axis rotation} = \frac{360}{0.01} = 36000$$ (degrees)

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 2000 rpm, and the servo motor is directly connected to the servo motor:

Upper limit to the tool rotation axis speed = 2000 × 1 = 2000 (rpm)

This means the parameters must be set as follows:

No. 7620 = 36000 (angle to rotate through per tool axis rotation)
No. 7621 = 2000 (upper limit to tool rotation axis speed)

- Feedrate parameter setting

Because the least command increment is 0.01 degrees, the input unit for the feedrate is 10 degrees/min.

To obtain a rapid traverse speed of 2000 rpm, for example, specify as follows:

No. 1420 = 72000 (= 2000 $\times \frac{360}{10}$)

Also specify other feedrates in 10 degrees/min units.

- Commands from the NC program

When the machine is not performing polygonal turning, the machining program can issue move commands to the polygon axis.

Such commands can be issued in the same way as for ordinary axes. However, be careful about the angle to rotate through and feedrate.

Assuming the polygon axis is the Y–axis, the polygon axis rotates through 0.03 degrees by the following command:

V3;

Likewise, the polygon axis rotates through 10.00 degrees by the following command:

V1.0;

The feedrate unit is also increased by tenfold.

The current position of the polygon in the machine coordinate system is normalized according to the value specified by parameter No. 7620.

Typical values range from 0.000 to 35.999.

Signal

Polygon synchronization under way signal PSYN <F063#7>

[Classification] Output signal

[Function] Informs the PMC that the machine is in the polygon turning mode.

[Output condition] The polygon synchronization signal is set to logical “1” by the polygon turning mode command (G51.2) and stays at “1” during the polygonal 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 polygonal turning mode.

Note 1 This signal uses the same address for both polygonal turning (using the servo axis) and polygonal turning with two spindles.

· Other signals (related to the tool rotation axis)

→ 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 polygonal turning mode.

For these signals, read the note in operator’s manual (for lathe).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F063	PSYN							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7600	PLZ							

[Data type] Bit

PLZ Synchronous axis using G28 command

0: Returns to the reference position in the same sequence as the manual reference position return.

1: Returns to the reference position by positioning at a rapid traverse. The synchronous axis returns to the reference position in the same sequence as the manual reference position return when no return–to–reference position is performed after the power is turned on.

7610	Control axis number of tool rotation axis for polygon turning

[Data type] Byte

[Valid data range] 1, 2, 3, . . . number of control axes

This parameter sets the control axis number of a rotation tool axis used for polygon turning.

7620	Movement of tool rotation axis per revolution

[Data type] Two–word

Increment system	IS–A	IS–B	IS–C	Unit
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 1 to 9999999

This parameter sets the movement of a tool rotation axis per revolution.

7621	Maximum allowable speed for the tool rotation axis (polygon synchronization axis)

[Data type] Word

[Unit of data] rpm

[Valid data range] For polygonal turning using servo motors:

$$\frac{0 \text{ to } 1.2 \times 10^8}{\text{set value of the parameter No. 7620}}$$

This parameter sets the upper–limit rotation speed of a tool rotation axis. The rotation speed of the tool rotation axis is clamped by the set upper–limit rotation speed during polygon turning. The spindle and tool rotation axis go out of synchronization when the rotation speed is clamped (P/S alarm No. 5018).

Alarm and message

Number	Message	Description
217	DUPLICATE G251 (COMMANDS)	G51.2 (or G251) is further commanded in the polygonal turning mode. Modify the program.
218	NOT FOUND P/Q COMMAND IN G251	P or Q is not commanded in the G51.2 (or the G251) block, or the command value is out of the range. Modify the program.
219	COMMAND G250/G251 INDEPENDENTLY	G51.2 (or G251) and G50.2 (or G250) are not independent blocks.
220	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.
221	ILLEGAL COMMAND IN SYNCHR–MODE	Polygon machining synchronous operation and Cs contouring control or balance cutting are executed at a time. Modify the program.

Note

Note 1 Before issuing a G51.2, rotate the spindle. If it is not rotating when the G51.2 is issued, the program stops to wait for a one–rotation signal from the position coder on the spindle. This does not apply to a dry run.

Note 2 A reset releases the polygonal turning mode.

Note 3 Machine a workpiece at the same spindle speed until finish machining for the workpiece.

Reference item

OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.20.1	POLYGONAL TURNING

6.10.2 Polygonal Turning with Two Spindles

General

In the polygonal turning with two spindles, the first spindle is used as a workpiece rotation axis (master axis). The second spindle is used as a tool rotation axis (polygon synchronization axis). Spindle rotation control is applied to both spindles with a constant ratio.

The polygonal 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 this function, it is also possible to specify the phase difference between the master and polygon synchronization axes.

Moreover, polygonal turning works with the first and second spindles on each tool post in a two–path lathe application. However, polygonal turning does not work with spindles on different tool posts.

• Command format

The CNC command format for polygonal turning with two spindles is described below. For the CNC command format for polygonal turning (see Section 6.10.1), refer to the operator's manual (for lathe). The two formats are almost identical. The differences are in that the polygonal turning with two spindles can specify a phase command (R) and re–issue commands.

• Mode command and command value change

◆ G51.2 P_Q_R_;

This command starts the polygon synchronization mode or changes the values specified for the polygon synchronization mode.

P: Master axis (first spindle) rotation ratio

Range of command value: Integer 1 to 999

(The direction in which the master axis rotates depends on the commands (such as M03 and M04) issued under ordinary spindle control.)

Q: Polygon synchronization axis (second spindle) rotation ratio

Range of command value: Integers 1 to 999 and –1 to –999

(The direction in which the polygon synchronization axis rotates depends on the algebraic sign of the Q value, except when bit 1 (GDRC) of parameter No. 7603 = 1, in which case the polygon synchronization axis rotates in the same direction as the first spindle. In this case, Q cannot take a negative value.)

R: Relative phase difference between the master and polygon synchronization axes

The range of command value and the increment system are the same as for the rotation axis. However, the angle to rotate through is in 360/4096 degrees units.

(R is omissible. If it is not specified at all, the phase difference is assumed to be 0. If bit 5 (PCOF) of parameter No. 7602 = 1 to disable phase control, the R command is ignored, but no alarm condition is assumed.)

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.

The S command issued to the first spindle during polygon synchronization mode specifies that the second spindle be used as a polygon synchronization axis and rotates at a speed of S × Q/P with a phase difference of R.

- **Release command**

◆ G50.2

This command releases the polygon synchronization mode. This mode is released also when:

(1) Reset
(That mode is not released by bit 0 (RPLM) of parameter No. 7603.)

(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<G0071#1> and *ESPB<G0075#1>.

(4) P/S alarm 218, 219, or 221 occurs

- **Cautions for using commands**

G51.2 and G50.2 must be issued separately from other commands.

In a G51.2 issued to enter the polygon synchronization mode, R is omissible, but P and Q are required.

After a G51.2 is issued to enter the polygon synchronization mode, changing modal values of P, Q, and R requires another G51.2. In this case, R can be specified separately from P and Q. However, P and Q must be specified together even if only one of them is changed.

- **Spindle operation during the spindle–spindle polygon synchronization mode**

When a G51.2 is issued to start the spindle–spindle polygon synchronization mode, the speed of the spindle (polygon synchronization axis) is changed to Q/P times the speed of the first spindle (master axis) to achieve a speed ratio of P:Q, and phase adjustment is performed.

(If no S command is issued to the first spindle after a G51.2, the previous S command remains effective.)

If the spindle is in an acceleration, deceleration, or phase adjustment state, synchronization at a rotation ratio of P:Q is not guaranteed. So, it is necessary to control SAR<G0029#4> by checking the speed arrival signal PSAR<F0063#2> for polygonal turning with two spindles or to allow sufficient time in the program.

The method to specify the spindle speed during the polygon synchronization mode is the same as for ordinary modes. However, each time an S command is issued to the first spindle during polygon synchronization mode, or a rotation ratio P:Q or phase value R command is re–specified during the polygon synchronization mode, phase adjustment is performed after speed control. In addition, speed commands (such as under multi-spindle control or spindle output control for the second spindle by the PMC) for the second spindle become ineffective.

If the specified polygon synchronization axis speed (S × Q/P for the first spindle at S rpm) exceeds the clamp speed specified in parameter No. 7621, the polygon synchronization axis speed is clamped, and P/S alarm No. 5018 is issued.

Each time the spindle speed command for the first spindle changes or P and Q are re–specified in a G51.2, the clamp speed is checked to determine whether to issue P/S alarm No. 5018.
Note that a reset can clear the alarm with the speed clamped.
(A rotation speed ratio of P:Q cannot be maintained with the speed clamped. Bit 2 (QCL) of DGN No. 471 indicates whether the speed is clamped.)

- **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 signals in Section 6.10.2).

- **Example of polygonal turning with two spindles**

This example of polygonal turning with two spindles produces a square using single–edged cutting tools (for roughing and finishing).

.		
M○□; T○○ΔΔ ;	Step 1.	Mount a roughing tool on the polygon synchronization axis (second spindle as tool rotation axis).
G00 X100. Z20. M03 S1000 ; . .	Step 2.	Rotate the workpiece (with the first spindle as the master axis at 1000 rpm)
G51. 2 P1 Q2 ;	Step 3.	Start rotating the tool. Energize the second spindle in response to the polygon synchronization under way signal using the PMC ladder. After accelerating the second spindle as the polygon synchronization axis to 2000 rpm, perform phase adjustment (Execute R0 to omit an R value.). By checking PSYC<F0063#7>, the PMC ladder can detect when the polygon synchronization mode is entered. During the polygon synchronization mode, the PMC ladder controls SAR<G0029#4> based on the speed arrival signal PSAR<F0063#2> during the polygon synchronization mode.
G01 X80. F10. ; . .	Step 4.	Starts cutting along the X–axis after SAR<G0029#4> becomes logical 1 in signal control at step 3.
G04 P4000 ; .	Step 5.	Polygonal turning (roughing 1)
G00 X100. ; .	Step 6.	Retract the tool along the X–axis.
G51. 2 R180 ; .	Step 7.	Change the phase by 180 degrees.
[Repeat steps 4, 5, and 6.] .	Step 8.	Polygonal turning (roughing 2)
G50. 2 ; MΔ□ ; TΔΔ □□ ; .	Step 9.	Release the polygon synchronization mode. Change to a finishing tool.

G51. 2 P1 Q2 ; S2000; . .	Step 10.	Change the spindle speed for finishing (master axis at 2000 rpm and polygon synchronization axis at 4000 rpm with a phase difference of 0).
[Repeat steps 4, 5, and 6.] .	Step 11.	Polygonal turning (finishing 1)
[Repeat step 7.] .	Step 12.	Change the phase by 180 degrees.
[Repeat steps 4, 5, and 6.] . .	Step 13.	Polygonal turning (finishing 2)
G50. 2 ; . . .	Step 14.	Release the polygon synchronization mode. The polygon synchronization axis (second spindle as tool rotation axis) stops. The first spindle rotates at a speed specified by an S command.
M05; . .	Step 15.	The first spindle stops (end).

- **Diagnosis display (DGN)**

For polygonal turning with two spindles, the following information is displayed on the diagnosis display screen.

Polygonal turning with two spindles Indication of information about the polygon synchronization mode

DGN	#7	#6	#5	#4	#3	#2	#1	#0
470	SC0	LGE		SCF			PST	SPL

SPL Spindle–spindle polygon synchronization under way

PST Spindle–spindle polygon synchronization mode being activated

#2 Spindle–spindle polygon synchronization mode released

#3 Spindle speed being changed during spindle–spindle polygon synchronization mode

SCF Spindle speed changed during spindle–spindle polygon synchronization mode

#5 → Not used

LGE The loop gain is different between the spindles during spindle–spindle polygon synchronization mode.

SC0 Actual speed command is 0 during spindle–spindle polygon synchronization mode.

Note
- The normal state during spindle–spindle polygonal turning is: SPL = 1, SCF = 1, #1 = 0, #2 = 0, and #3 = 0
- If only PST 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.
- 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 first and second spindles.

· DGN 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 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<G0029#6> and SOV0–SOV7<G0030> and the signal related to multi–spindle control <G0027>, 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 <G0032, G0033>, and it is 0.

If SC0 = 1, the spindle speed becomes 0 and bit 0 of DGN No. 471 becomes 1. In this case, the polygon synchronization rotation ratio is impractical, but P/S alarm No. 5018 does not occurs, because it is regarded as the result of the command.

Polygonal turning with two spindles Indication of causes for P/S alarms 5018 and 218

	#7	#6	#5	#4	#3	#2	#1	#0
471	NPQ	PQE		NSP	SUO	QCL	PCL	

#0 to #3 → Causes for P/S alarm No. 5018
P/S alarm No. 5018 is cleared by a reset, but the indication of its causes remains until the causes are cleared or the polygon synchronization mode is released.

#4 to #7 → Causes for P/S alarm No. 218
When P/S alarm No. 218 occurs, the polygon synchronization mode is released, but the indication of its causes remains until the alarm is cleared by a reset.

#0 The specified speed is too low during spindle–spindle polygon synchronization mode. (The unit of speed calculated internally becomes 0.)

PCL The first spindle (polygon synchronization master axis) is clamped.

QCL The second spindle (polygon synchronization axis) is clamped.

SUO The specified speed is too high during the spindle–spindle polygon synchronization mode. (It is clamped to the upper limit calculated internally.)

NSP A spindle necessary for control is not connected. (For example, there is not a serial spindle or the second spindle.)

#5 When bit 1 (QDRC) 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.

Note 1

· #0 becomes 1 also when the specified spindle speed is 0 (DGN 470#7 = 1). In this case, however, P/S alarm No. 5018 is not issued (because the command is 0). When DGN 470#7 = 0 and DGN 471#0 = 1, P/S alarm No. 5018 occurs. Normally this does not occur with speed at which the spindle can rotate.

· PCL indicates that the master axis has received a command with a speed that is higher than the value specified by the maximum first spindle speed parameters (No. 3741 to 3744) and is clamped to that speed. PCL will not become 1 as long as the first spindle is connected correctly.

· QCL becomes 1, when the second spindle (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 occurs, if a result of (speed specified for the first spindle)/(value specified at P) is higher than 59998. In other words, the first spindle must rotate at a speed lower than 59998 rpm assuming P = 1.

Indication of values specified during the spindle–spindle polygon synchronization mode

474	Rotation ratio for the master axis during the spindle–spindle polygon synchronization mode (P command value)

This indication is the current rotation ratio (P command value) of the master axis (first spindle) during the spindle–spindle polygon synchronization mode.

475	Rotation ratio for the polygon synchronization axis during the spindle–spindle polygon synchronization mode (Q command value)

This indication is the current rotation ratio (Q command value) of the polygon synchronization axis (second spindle) during the spindle–spindle polygon synchronization mode.

476	Phase difference between the two spindles under spindle–spindle polygon synchronization control (R command value)

This indication is the current phase value (R command value) specified during the spindle–spindle polygon synchronization mode. (The unit of measurement is the least increment system for the rotation axis of the machine.)

However, if bit 5 (RDGN) 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 spindle–spindle polygon synchronization mode

477	Actual master axis speed (rpm) during the spindle–spindle polygon synchronization mode

This indication is the actual speed of the master axis (first spindle) during the spindle–spindle polygon synchronization mode.

478	Actual polygon synchronization axis speed (rpm) during the spindle–spindle polygon synchronization mode

This indication is the actual speed of the polygon synchronization axis (second spindle) during the spindle–spindle polygon synchronization mode.

Note 1 The indications of DGN Nos. 477 and 478 vary because of no sampling being performed. Consider these DGN values only guidelines.

Signal

Polygon synchronization under way signal PSYN<F063#7>

[Classification] Output signal

[Function] Informs the PMC that the system is in the polygon synchronization mode.

[Output condition] 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 1 The same address is used for this signal in both polygonal turning (using the servo axis) and the polygonal turning with two spindles.

Polygon spindle stop signal *PLSST<G038#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 polygonal turning mode with two spindles.

“0” = polygon spindle stop
“1” = polygon spindle operable

During the polygonal turning mode with two spindles, the spindles are controlled with a positional loop set up. When issuing a spindle stop command (like M05) to deenergize the spindle, it is necessary to specify S = 0 using *SSTP<G00296#6>. Otherwise the stop command remains effective even after the spindle is deenergized. This error accumulates and causes a dangerous behavior of the spindle when it is energized again. This signal is intended to inhibit distribution of the S command to the spindle if it cannot be set to 0 while the spindle is deenergized. In such a case, the signal should be used in step with the energizing state of the polygon spindle.

Spindle polygonal speed arrival signal PSAR<F063#2>

[Classification] Output signal

[Function] Informs the PMC that the spindle has reached its constant–speed for polygon synchronization during polygonal turning with two spindles.

[Output condition] During polygonal turning 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

Details → During the polygon control 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 polygonal 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.

Master axis not arrival signal PSE1<F063#0> Polygon synchronization axis not arrival signal PSE2<F063#1>

[Classification] Output signal

[Function] Informs the PMC whether the actual speed of each spindle has reached the specified speed during polygonal turning mode with two spindles.

[Output condition] During polygonal turning 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)

Details → During the polygon control mode, this signal becomes logical “1” when the speed of master axis (first spindle) and polygon synchronization axis (second spindle) does not reach the acceptable level specified in parameter No. 7631.

- **PMC sequence**

·When a G51.2 is issued to put the system in the polygon synchronization mode, the polygon synchronization under way signal PSYN<F063#7> becomes on.

Set up a PMC sequence for the polygon synchronization mode by monitoring this signal with a PMC ladder.

There are two control methods, (A) and (B), to control energizing of the spindle. First select (A) or (B) and creates a PMC sequence according to the selected method.

- **Method A**

(A)Energize the first and second spindles automatically during the G51.2 mode.
Basically, do not discontinue energizing during this mode.

→ In this case, keep bit 7 (PST) of parameter No. 7603 = 0.

In the PMC sequence, detect when the polygon synchronization under way signal PSYN<F063#7> changes from 0 to 1, then energize the first and second spindles.

The NC stops at the G51.2 command block which puts the system in the polygon synchronization mode, and remains there until the spindle reaches the polygon synchronization speed.

Also, keep the spindle energized, for example, by preventing it from receiving a spindle stop command (like M05) for ordinary spindle control while PSYN<F063#7> = 1. Basically, deenergize the spindle when the polygon synchronization under way signal PSYN<F063#7> changes from 1 to 0.

- **Method B**

(B)Control the energizing of the spindle using M codes even during the G51.2 mode. Alternatively, deenergize the spindle even during the G51.2

→ In this case, keep bit 7 (PST) of parameter No. 7603 = 1.

This parameter setting enables use of the spindle stop signal *PLSST<G038#0> during the polygon synchronization mode. It also makes the G51.2 command block stop waiting for the spindle to reach the polygon synchronization speed.

In the PMC sequence, while the polygon synchronization under way signal PSYN<F063#7> = 1, set *PLSST<G038#0> to 1 after confirming both first spindle and second spindles are energized.

If either spindle has been deenergized, reset *PLSST<G038#0> to 0.

When the polygon synchronization under way signal PSYN <F063#7> changes from 1 to 0, basically keep *PLSST <G038#0> at 0. (This is intended to keep *PLSST <G038#0> from becoming 1 in a deenergized state when the polygon synchronization mode is entered again.)

When *PLSST <G038#0> changes from 0 to 1 during the polygon synchronization mode, the spindle is accelerated from a stop state to the specified speed and placed under phase control, even if the S command has not been changed.

- **Sequence common to methods (A) and (B)**

Regardless of whether the method you use is (A) or (B), set up the PMC sequence as follows:

- Do not use the SFR/SRV signal to switch the rotation direction of the first spindle. Instead, fix the energizing method of the spindle at SFR and change the polarity of the command. (To change the polarity of the command, issue M03/M04 with bit 7 (TCW) of parameter No. 3706 = 1, or control SGN<G033#5> with SSIN<G033#6> = 1.)

  Also fix the energizing method of the second spindle at SFR.
- To check whether the spindle has reached its constant–speed, control SAR<G029#4> using PSAR<F063#2>.

  PSAR<F063#2> can be used to check whether both spindles have reached the conditions specified in parameter Nos. 7631 and 7632 after completion of phase control. Checking the speed arrival signal for each spindle (SARA<F045#3> and SARB<F049#3>) cannot guarantee proper cutting start conditions, because phase control may occur afterward.

  The specification (parameter No. 3740) of time allowed before the spindle constant–speed reached signal is checked remains valid until after execution of the G51.2 command.

  If you are not using SAR<G029#4> as the cutting feed start condition, start cutting after allowing time using the program (like G04) for both spindles to reach their constant speeds, when the polygon synchronization mode is entered and each time the S command is changed during the polygon synchronization mode.
- It is impossible to rotate the second spindle separately from the first spindle and to use the spindle orientation function (ORCMA<G070#6>, ORCMB<G074#6>) during polygon synchronization mode with two spindles. Basically, do not perform gear change, tool change, or workpiece change during the polygon synchronization mode. Have the PMC reject such commands and output a message prompting to release the two–spindle polygon synchronization mode, as required.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G038								*PLSST

	#7	#6	#5	#4	#3	#2	#1	#0
F063	PSYN					PSAR	PSE2	PSE1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7602			COF	HST	HSL	HDR	SNG	MNG

[Data type] Bit

MNG The rotational direction of the master axis (first spindle) in the spindle–spindle polygon turning mode is:
0: Not reversed.
1: Reversed.

SNG The rotational direction of the polygon synchronization axis (second spindle) in the spindle–spindle polygon turning mode is:
0: Not reversed.
1: Reversed.

HDR When phase control is exercised in spindle–spindle polygon turning mode (COF = 0), the phase shift direction is:
0: Not reversed for phase synchronization.
1: Reversed for phase synchronization.

Note 1 Use MNG, SNG, and HDR when the specified rotational direction of the master axis or polygon synchronization axis, or the specified phase shift direction is to be reversed in spindle–spindle polygon turning mode.

HSL When phase control is exercised in spindle–spindle polygon turning mode (COF = 0), this parameter selects the spindle that is subject to a phase shift operation for phase synchronization:
0: The polygon synchronization axis (second spindle) is selected.
1: The master axis (first spindle) is selected.

HST When phase control is applied in spindle–spindle polygon turning mode (COF = 0), and spindle–spindle polygon turning mode is specified:
0: Spindle–spindle polygon turning mode is entered with the current spindle speed maintained.
1: Spindle–spindle polygon turning mode is entered after the spindle is stopped.

Note 1 This parameter can be used, for example, when single–rotation signal detection cannot be guaranteed at an arbitrary feedrate because a separate detector is installed to detect the spindle single–rotation signal, as when a built–in spindle is used. (When bit 7 of parameter No. 4016 for the serial spindle is set to 1, together with this parameter, a single–rotation signal detection position in spindle–spindle polygon turning mode is guaranteed.)

COF In spindle–spindle polygon turning mode, phase control is:
0: Used.
1: Not used.

Note 1 When the use of phase control is not selected, the steady state is reached in a shorter time because phase synchronization control is not applied. Once steady rotation is achieved, however, polygonal turning must be completed without changing the steady state. (If the rotation is stopped, or the rotational speed altered, polygonal turning is disabled because of the inevitable phase shift.)

Setting this parameter to 1 does not issue an alarm on the R command (phase command) in the same block as the G51.2. It is only ignored.

	#7	#6	#5	#4	#3	#2	#1	#0
7603	PST		RDG				QDR	RPL

[Data type] Bit

RPL Upon reset, spindle–spindle polygon turning mode is:
0: Released.
1: Not released.

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 − is specified for Q, P/S alarm No. 218 is issued.)

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 1 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.

PST The polygon spindle stop signal *PLSST (bit 0 of G038) is:
0: Not used.
1: Used.

7621	Maximum allowable speed for the tool rotation axis (polygon synchronization axis)

[Data type] Word

[Unit of data] rpm

[Valid data range] For polygon turning with two spindles:
Set a value between 0 and 32767, but which does not exceed the maximum allowable speed, as determined by the performance of the second spindle and other mechanical factors.

This parameter sets the maximum allowable speed of the tool rotation axis (polygon synchronization axis).
If the speed of the tool rotation axis (polygon synchronization axis) exceeds the specified maximum allowable speed during polygon turning, the speed is clamped at the maximum allowable speed. When the speed is clamped at a maximum allowable speed, however, synchronization between the spindle and tool rotation axis (polygon synchronization axis) is lost. And, when the speed is clamped, P/S alarm No. 5018 is issued.

7631	Allowable spindle speed deviation level in spindle–spindle polygon turning

[Data type] Byte

[Unit of data] rpm

[Valid data range] 0 to 255

[Standard setting value] 1 to 10

This parameter sets the allowable level of deviation between the actual speed and specified speed of each spindle in spindle–spindle polygon turning. The value set with this parameter is used for both the master axis and polygon synchronization axis.

.

7632	Steady state confirmation time duration in spindle polygon turning

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 32767

This parameter sets the duration required to confirm that both spindles have reached their specified speeds in spindle–spindle polygon turning. 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 (bit 2 of F0063) is set to 1.

Alarm and message

Number	Message	Description
218	NOT FOUND P/Q COMMAND IN G251	The G51.2 block does not contain P or Q, or a specified value is invalid. The causes of this alarm are detailed in DGN No. 471. (See below.) DGN No. 471#7 NPQ → When P and Q are not specified at all, R is specified. Alternatively, none of P, Q, and R has been specified. DGN No. 471#6 PQE → P or Q is out of the valid data range. Alternatively, P and Q are not specified as a pair. DGN No. 471#5 → A negative value is specified at Q when bit 1 (QDRC) of parameter No. 7603 = 1. DGN No. 471#4 NSP → There is no spindle necessary for control. (The spindle is not a serial spindle, or there is not the second spindle, etc.)
219	COMMAND G250/G251 INDEPENDENTLY	G51.2/G251 and G50.1/G250 are specified together with other commands in the same block. Correct the program.
221	ILLEGAL COMMAND IN SYNCHRO	An attempt was made to perform polygon synchronization operation together with Cs contouring control or balance cutting. Alternatively the program issued a command for spindle–spindle polygon synchronization mode, when the spindle is under spindle synchronization control, Cs contouring control, spindle positioning control, or rigid tapping control, etc. Correct the program.
5018	POLYGON SPINDLE SPEED ERROR	The specified rotation ratio cannot be maintained during the G51.2 mode, because the speed of the spindle or polygon synchronization axis exceeds the clamping value or is too low. The causes of this alarm are detailed in DGN No. 471. (See below.) DGN No. 471#3 SUO → The specified speed is too high. DGN No. 471#2 QCL → The polygon synchronization axis (second spindle) is clamped. DGN No. 471#1 PCL → The master axis (first spindle) is clamped. DGN No. 471#0 → The specified speed is too low.

(Remark)

· In a properly connected machine, P/S alarm No. 5018 does not basically occur for other than a reason that the polygon synchronization axis is clamped. (See descriptions of DGN for details.)

· To the contrary to P/S alarm No. 221, P/S alarm No. 194 occurs if another NC control spindle function is specified during the two–spindle polygon synchronization mode.

Note

Note 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.

Note 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 (PHST) of parameter No. 7602 and bit 7 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 spindle–spindle polygonal turning mode is entered, thus guaranteeing a correct phase relationship during spindle–spindle polygon synchronization mode.

Note 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).
Specify the same serial spindle loop gain for both spindles.
If the same serial spindle loop gain is not used for both spindles, polygonal turning may not be accurate.
If an attempt is made to perform spindle–spindle polygonal turning, DGN 470#6 LGE becomes 1. (No alarm is issued.)

Note 4 Before using the polygon synchronization mode, place both first and second spindles in the spindle control mode. The polygon synchronization mode cannot be used if they are already in other modes (Cs contouring control mode or spindle orientation mode, etc.).

Note 5 During the polygon synchronization mode, the speed of the second spindle cannot be controlled independently of the speed of the first spindle.
During the polygon synchronization mode, the spindle orientation function (ORCMA<G070#6>, ORCMB<G074#6) cannot be used for either the first spindle or second spindle. Therefore, gear, tool or workpiece change is basically unusable during the polygon synchronization mode.

Note 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)

Note 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. (The master axis speed may be up to 2 rpm lower than specified.)

Note 8 During the polygon synchronization mode, phase control is performed in the least command increment of 36/4096 = 0.08789...(degrees) in reference to the one–rotation signal for each spindle. Actually, a command value out of a range from 0 to 359.999... is meaningless because the relative phase is controlled within one rotation of each spindle. However, this function does not limit the R command value and the CNC converts the R command value to a value below 360 degrees.

Note 9 The G51.2 command during conversational function is equivalent to the G50.2 command. (The system does not enter the polygon synchronization mode. If it is already in the polygon synchronization mode, release it using a G51.2 command.)

Reference item

OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.20.1	POLYGONAL TURNING

6.11 NORMAL DIRECTION CONTROL (M SERIES)

General

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. 6.11).

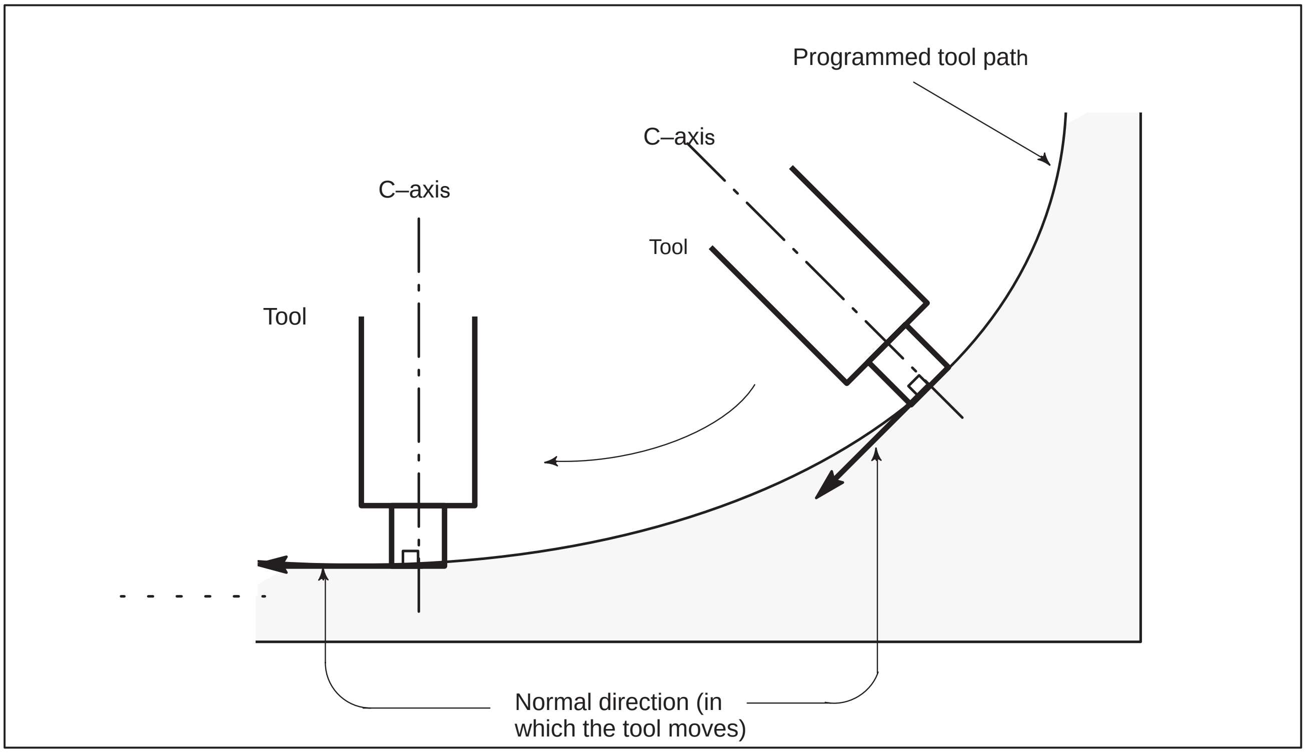


Fig. 6.11 Sample Movement of the tool

Movement of the tool inserted at the beginning of each block is executed at the feedrate set in parameter 5481. If dry run mode is on at that time, the dry run feedrate is applied. If the tool is to be moved along the X–and Y–axes in rapid traverse (G00) mode, the rapid traverse rate is applied.

If the federate of the C axis exceeds the maximum cutting feedrate of the C axis specified to parameter No. 1422, the federate of each of the other axes is clamped to keep the federate 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

Note 1 When this parameter is changed, turn off the power before continuing operation.

[Data type] Bit axis

[Valid data range] ROTx, ROSx Setting linear or rotation axis

ROSx	ROTx	Description
0	0	Linear axis · Inch/metric conversion is done. · All coordinate values are linear axis type. (Not rounded in 0 to 360°) · Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotation axis (A type) · Inch/metric conversion is not done. · Machine coordinate values are rounded in 0 to 360° Absolute coordinate values and relative coordinate values are rounded or not rounded by parameter No. 1008#0 and #2. · Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) · Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	0	Setting is invalid (unused)
1	1	Rotation axis (B type) · Inch/metric conversion is not done. · Machine coordinate values is linear axis type (is not rounded in 0 to 360°). · Absolute coordinate values and relative coordinate values are rounded or not rounded by parameter No. 1008#0 and #2. · Stored pitch error compensation is linear axis type (Refer to parameter No. 3624). · Cannot be used with the rotation axes roll over function and the index table indexing function (M series).

Note 1 The rotation axis must be set to the normal direction control axis.

5480	Number of the axis for controlling the normal direction

[Data type] Byte

[Valid data range] 1 to the maximum control axis number

This parameter sets the control axis number of the axis for controlling the normal direction.

5481	Rotation feedrate of normal direction control axis

[Data type] Word

[Unit of data] 1 deg/min

[Valid data range] 1 to 15000

This parameter sets the feedrate of a normal direction control axis that is inserted at the start point of a block during normal direction control.

5482	Limit value that ignores the rotation insertion of normal direction control axis

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 1 to 99999999

The rotation block of a normal direction control axis is not inserted when the rotation insertion angle calculated during normal direction control does not exceed this setting value. The ignored rotation angle is added to the next rotation insertion angle. The block insertion is then judged.

Note 1 No rotation block is inserted when 360 or more degrees are set.

Note 2 If 180 or more degrees are set, a rotation block is inserted only when the circular interpolation is 180 or more degrees.

5483	Limit value of movement that is executed at the normal direction angle of a preceding block

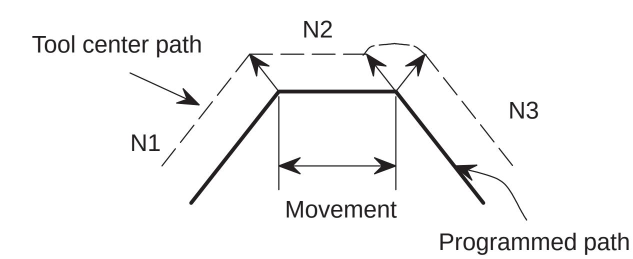
[Data type] Two–word

[Unit of data]

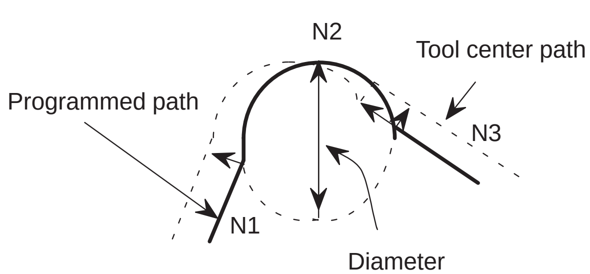
Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1 to 99999999

This parameter sets the limit value of movement at the normal direction angle of a preceding block.



For straight line
Block N2 is machined with the tool being normal to block N1 when the movement of N2 in the figure on the left does not exceed the set value.



For arc
Arc N2 is machined with the tool being normal to block N1 when the arc diameter of N2 in the figure on the left does not exceed the setting value. A normal direction axis is not controlled to move in the normal direction according to the arc movement.

1422	Maximum cutting feedrate for all axes

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

Specify the maximum cutting feedrate.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

Note 1 To specify the maximum cutting feedrate for each axis, use parameter No. 1430 instead.

Note

The helical interpolation option is required to use this function. Helical interpolation cannot be specified in the normal direction control mode.

Reference Item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.15.12	Normal Direction Control

7 FEEDRATE CONTROL/ACCELERATION AND DECELERATION CONTROL

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 switch on the machine operator's panel.

7.1.1 Rapid Traverse Rate

General

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 machine tool builder (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 switch on the machine operator's panel:F0, 25, 50, 100%
F0: Allows a fixed feedrate to be set for each axis by parameter No. 1421.

Signal

Rapid traversing signal RPDO <F002#1>

[Function] This signal indicates that a move command is executed at rapid traverse.

[Output condition] "1" indicates that 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 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.

Note 2 Once rapid traverse has been selected, 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
F002							RPDO	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401		RDR		RFO			LRP	

[Data type] Bit

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.

RFO When cutting feedrate override is 0% during rapid traverse,
0 : The machine tool does not stop moving.
1 : The machine tool stops moving.

RDR Dry run for rapid traverse command
0 : Disabled
1 : Enabled

1420	Rapid traverse rate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 240000	30 – 100000
Inch machine	0.1 inch/min	30 – 96000	30 – 48000
Rotaion axis	1 deg/min	30 – 240000	30 – 100000

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1424	Manual rapid traverse rate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 240000	30 – 100000
Inch machine	0.1 inch/min	30 – 96000	30 – 48000
Rotaion axis	1 deg/min	30 – 240000	30 – 100000

Set the rate of manual rapid traverse for each axis when the rapid traverse override is 100% for each axis.

Note 1 If 0 is set, the rate set in parameter 1420 is assumed.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.5.2	Rapid traverse
OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.5.2	Rapid traverse

7.1.2 Cutting Feedrate Clamp

General

A common upper limit can be set on the cutting feedrate along each axis with parameter No. 1422. If an actual cutting feedrate (with an override applied) exceeds a specified upper limit, it is clamped to the upper limit.

Parameter

1422	Maximum cutting feedrate for all axes

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

Specify the maximum cutting feedrate.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

Note 1 To specify the maximum cutting feedrate for each axis, use parameter No. 1430 instead.

1430	Maximum cutting feedrate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000
Rotaion axis	1 deg/min	6 – 240000	6 – 100000

Specify the maximum cutting feedrate for each axis.

A feedrate for each axis is clamped in cutting feed so that it does not exceed the maximum feedrate specified for each axis.

Note 1 This parameter is effective only in linear and circular interpolation. In polar coordinate, cylindrical, and involute interpolation, the maximum feedrate for all axes specified in parameter No. 1422 is effective.

Note 2 If the setting for each axis is 0, the maximum feedrate specified in parameter No. 1422 is applied to all axes and the feedrate is clamped at the maximum feedrate.

Alarm and message

Number	Message	Description
5009	PARAMETER ZERO (DRY RUN)	The maximum feedrate (parameter No. 1422) or the feedrate in dry run (parameter No. 1410) is set to 0 in the HPCC model.
5011	PARAMETER ZERO(CUT MAX)	The maximum cutting feedrate (parameter No. 1422)is set to 0 in the HPCC mode.

Note 1 For HPCC mode, refer to 7.1.14 HIGH PRECISION CONTOUR CONTROL.

Note

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 during the steady state:

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.5.3	Cutting Feed
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.5.3	Cutting Feed

7.1.3 Feed per Minute

General

Override can be applied to a rapid traverse rate or cutting feedrate using the switch on the machine operator's panel.

- **Feed per minute (G94)**

After specifying G94 (G98 for T series) (in the feed per minute mode), the amount of feed of the tool per minute is to be directly specified by setting a number after F. 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 T series) (feed per revolution) is specified. At power–on, the feed per minute mode (feed per revolution mode for T series) is set.
An override from 0% to 254% (in 1% steps) can be applied to feed per minute with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

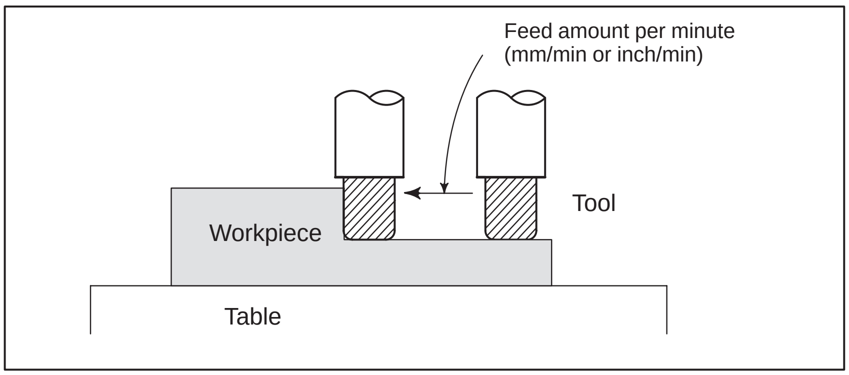


Fig. 7.1.3 Feed per minute

Note
No override can be used for any commands such as for threading.

Format

For M series

G94;	G code for feed per minute (Group 05)
F_;	Feed rate (mm/min or inch/min)

For T series

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
1403								MIF

[Data type] Bit

MIF Cutting feedrates at feed per minute is specified by F commands

0 : In units of 1 mm/min for millimeter machines or 0.01 inches/min for inch machines.

1 : In unit of 0.001 mm/min for millimeter machines or 0.00001 inches/min for inch machines.

Note 1 M series are not equipped with this parameter. Cutting feedrates are specified by F commands in units of 0.001 mm/min for millimeter machines or 0.00001 inches/min for inch machines.

	#7	#6	#5	#4	#3	#2	#1	#0
3401							FCD	

[Data type] Bit

FCD When an F command and a G command (G98, G99) for feed per minute or feed per rotation are specified in the same block, and the G command (G98, G99) is specified after the F command, the F command is:

0 : Assumed to be specified in the mode (G98 or G99) when the F command is specified

1 : Assumed to be specified in the mode of the G command (G98 or G99) of the same block

Note 1 When FCD = 1:

If the block containing a G command (G98, G99) does not include an F command, the last F command specified is assumed to be specified in the G command mode of the block.

Example 1: N1 G99 ;

N2 Faaaa G98 ; - Faaaa is assumed to be specified in the G98 mode.

N3 Fbbbb ; - Fbbbb is assumed to be specified in the G98 mode.

N4 G99 ; - Fbbbb is assumed to be specified in the G99 mode.

Note 2 In G code system B or C, G98 and G99 function are specified in G94 and G95.

Alarm and message

Number	Message	Description
011	NO FEEDRATE COM-MANDED	Feedrate was not commanded to a cutting feed or the feedrate was inadequate. Modify the program.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.5.3	Cutting feed
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.5.3	Cutting feed

7.1.4 Feed per Revolution/ Manual Feed per Revolution

General

- **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 from 0% to 254% (in 1% steps) can be applied to feed per revolution with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

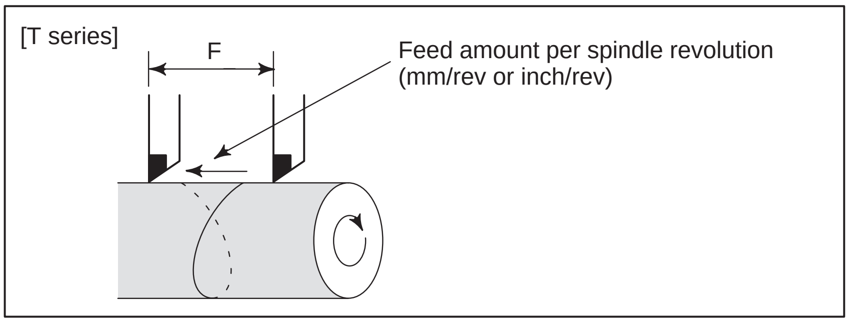


Fig. 7.1.4 Feed per revolution

- **Manual feed per revolution**

Jog feedrate can be specified by feed per revolution.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1402				JRV				

[Data type] Bit

JRV Manual continuos feed (jog feed)
0 : Jog feed is performed at feed per minute.
1 : Jog feed is performed at feed per rotation.

Note 1 Specify a feedrate in parameter No. 1423.

1423	Feedrate in manual continuous feed (jog feed) for each axis

[Data type] Word axis

When JRV, bit 4 of parameter No. 1402, is set to 1 (feed per revolution) in T series, specify a feedrate in jog feed (feed per revolution) with an override of 100% applied to the jog feedrate.

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range
Millimeter machine	0.01 mm/rev	0 A 32767
Inch machine	0.001 inch/rev	
Rotaion axis	0.01 deg/rev	

Note

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

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.5.3	Cutting feed
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.5.3	Cutting feed

7.1.5 F1-digit Feed (M series)

General

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.

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:

$$\Delta F = \frac{Fmax}{100X}$$

Fmax : feedrate upper limit for F1–F4 set by parameter 1460, or feedrate upper limit for F5–F9 set by parameter 1461

X : any value of 1–127 set by parameter 1450

The feedrate set or altered is kept even while the power is off. The current feedrate is displayed on the CRT screen.

Signal

F1-digit feed select signal F1D <G016#7>

[Classification] Input signal

[Function] Increases or decreases F1-digit 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 F1–digit speed can be increased/decreased using the manual pulse generator.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G016	F1D							

Parameter

1450	Change of feedrate for one graduation on the manual pulse generator during F1 digit feed

[Data type] Byte

[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 F1-digit feed.

$$\Delta F = \frac{Fmaxi}{100n}$$

(where, i=1 or 2)

In the above equation, set n. That is, the number of revolutions of the manual pulse generator, required to reach feedrate Fmaxi is obtained. Fmaxi refers to the upper limit of the feedrate for an F1-digit feed command, and set it in parameter 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 digit command F1
1452	Feedrate for F1 digit command F2
1453	Feedrate for F1 digit command F3
1454	Feedrate for F1 digit command F4
1455	Feedrate for F1 digit command F5
1456	Feedrate for F1 digit command F6
1457	Feedrate for F1 digit command F7
1458	Feedrate for F1 digit command F8
1459	Feedrate for F1 digit command F9

Input for setting is enabled.

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	0.1 mm/min	6 – 150000	6 – 120000
Inch machine	0.01 inch/min	6 – 60000	6 – 48000
Rotaion axis	0.1 deg/min	6 – 150000	6 – 120000

Set Feedrates for F1-digit feed commands F1 to F9.

When an F1-digit feed command is executed, as the feedrate is changed by turning the manual pulse generator, these parameter values also change accordingly.

1460	Upper limit of feedrate for the F1–digit feed command (F1 to F4)

1461	Upper limit of feedrate for the F1-digit feed command (F5 to F9)

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the upper limit of feedrate for the F1-digit 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 F1-digit feed command F1 to F4 is executed, the upper limit is that set in parameter 1460. If an F1-digit command F5 to F9 is executed, the upper limit is that set in parameter 1461.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.5.3	Cutting feed

7.1.6 Feedrate Inverse Time Specification (M series)

General

Feedrate of the tool can be specified by the move distance of the block and inverse time (FRN).

- **Linear interpolation (G01)**

$$FRN = \frac{1}{\text{Time (min)}} = \frac{\text{Speed}}{\text{Distance}}$$

Speed: mm/min (metric input)
inch/min (inch input)
Distance: mm (metric input)
inch (inch input)

- **Circular interpolation (G02, G03)**

$$FRN = \frac{1}{\text{Time (min)}} = \frac{\text{Speed}}{\text{Circle radius}}$$

Speed: mm/min (metric input)
inch/min (inch input)
Circle radius:
mm (metric input)
inch (inch input)

> Notes
> 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
011	NO FEEDRATE COMMANDED	Feedrate was not commanded to a cutting feed, F0 was specified or the feedrate calculated (less than 0.001 mm/min, for metric input or less than 0.00001 inch for inch input) becomes less than an allowable range.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.5.3	Cutting feed

7.1.7 Override

7.1.7.1 Rapid Traverse Override

General

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).
Also, 1% rapid traverse override select signal allows rapid traverse override every 1% in the range of 0 to 100%.

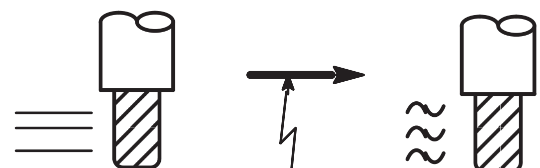


Fig.NO TAG Rapid traverse override

- **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).

- **Fo rate**

For Fo 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 selection signal**

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, signals ROV1 and ROV2 are ignored and override is applied to the rapid traverse rate using seven signals *HROV6 to *HROV0. The binary value corresponding to the signals *HROV6 to *HROV0 indicates an override value applied.

- **PMC axis control**

These 1% step rapid traverse override 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 <G014#0, #1>

[Classification] Input signal

[Function] These signals override the rapid traverse rate

[Operation] These code signals correspond to the rates as follows:

Rapid traverse override		Override value
ROV2	ROV1	
0	0	100 %
0	1	50 %
1	0	25 %
1	1	Fo %

Fo: Set in parameter No. 1421

1% step rapid traverse override selection signal HROV <G096 #7>

[Classification] Input signal

[Function] Selects whether the rapid traverse override signals or the 1% step rapid traverse override signals are enabled.

[Operation] When HROV is 1, signals *HROV0 to *HROV6 are effective and rapid traverse override with signals ROV1 and ROV2 is ignored.

When HROV is 0, signals *HROV0 to *HROV6 are ineffective, and rapid traverse override with signals ROV1 and ROV2 is effective.

1% step rapid traverse override signals *HROV0 to *HROV6 <G096 #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 override applied 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.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G014							ROV2	ROV1

	#7	#6	#5	#4	#3	#2	#1	#0
G096	HROV	*HROV6	*HROV5	*HROV4	*HROV3	*HROV2	*HROV1	*HROV0

Parameter

1421	F0 rate of rapid traverse override for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the F0 rate of the rapid traverse override for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
8001						OVE		

[Data type] Bit

OVE Dry run and override signals during axis control by the PMC
0 : Use the same signals as CNC
(1) Feedrate override signal *FV0 to *FV7
(2) Override cancel signal OVC
(3) Rapid traverse override signals ROV1 and ROV2
(4) Dry run signal DRN
(5) Rapid traverse selection signal RT
1 : Use dedicated axis control signals by the PMC.
(1) Feedrate override signal *FV0E to *FV7E
(2) Override cancel signal OVCE
(3) Rapid traverse override signals ROV1E and ROV2E
(4) Dry run signal DRNE
(5) Rapid traverse selection signal RTE

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.5.3	Rapid traverse override
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.5.3	Rapid traverse override

7.1.7.2 Feedrate Override

General

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.

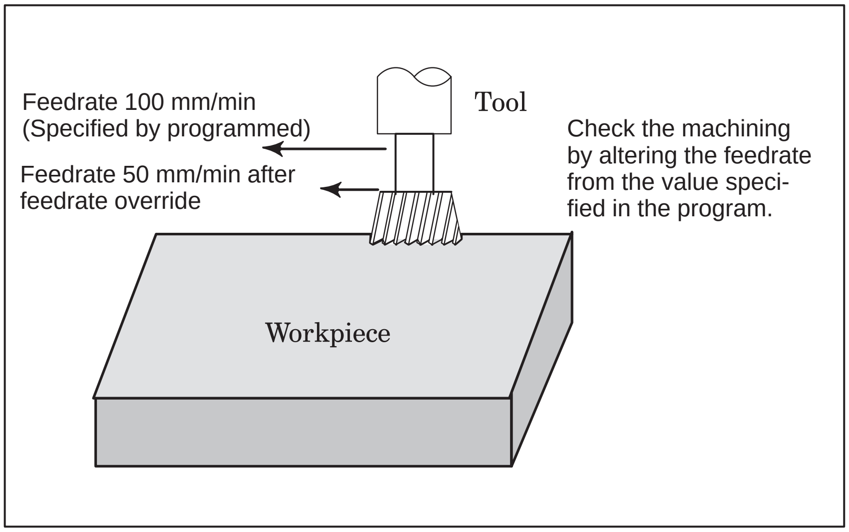


Fig. NO TAG Feedrate override

Signal

Feedrate Override signal *FV0 to *FV7 <G012>

[Classification] Input signal

[Function] These signals override the cutting feedrate. Eight binary code signals correspond to override values as follows:

$$\text{Override value} = \sum_{i=0}^{7} (2^i \times Vi) \%$$

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 speed specified in cutting feed in automatic operation mode 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 is “1”.
·During cutting in tap cycle of canned cycle;
·Tapping mode (63); or
·Thread cutting is in progress.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G012	*FV7	*FV6	*FV5	*FV4	*FV3	*FV2	*FV1	*FV0

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401				RFO				

[Data type] Bit

RFO When cutting feedrate override is 0% during rapid traverse,
0 : The machine tool does not stop moving.
1 : The machine tool stops moving.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.5.3	Cutting feed
OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.5.3	Cutting feed

7.1.7.3 Second feedrate override

General

These signals override the cutting feedrate after the cutting feedrate has been overridden by first override *FV0 to *FV7.

Signal

Second feedrate override signal *AFV0 to *AFV7 <G013>

These eight binary code signals correspond to the override values as follows.

$$\text{Override value} = \sum_{i=0}^{7} (2^i \times Vi)$$

Vi=0 when *AFVi is “1” and
Vi=1 when *AFVi is “0”

These signals have the following weight.

*AFV0 : 1%	*AFV1 : 2%
*AFV2 : 4%	*AFV3 : 8%
*AFV4 : 16%	*AFV5 : 32%
*AFV6 : 64%	*AFV7 : 128 %

If all signals are “0” or “1”, the override is regarded as 0%. The override is selectable in steps over a range of 0 to 254%.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G013	*AFV7	*AFV6	*AFV5	*AFV4	*AFV3	*AFV2	*AFV1	*AFV0

7.1.7.4 Override Cancel

General

The override cancel signal fixes the feedrate override to 100%.

Signal

Override cancel signal OVC <G006#4>

[Classification] Input signal

[Function] Feedrate override is fixed to 100%.

[Operation] When the signal is “1”, the CNC operates as follows:

- Feedrate override is fixed to 100% irrespective of feedrate override signal.
- Rapid traverse override and spindle speed override are not affected.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G006				OVC				

7.1.8 Automatic Corner Override (M series)

General

- **Inner corner automatic override**

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).
2,$\leqq \theta \leqq \theta p \leqq$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.

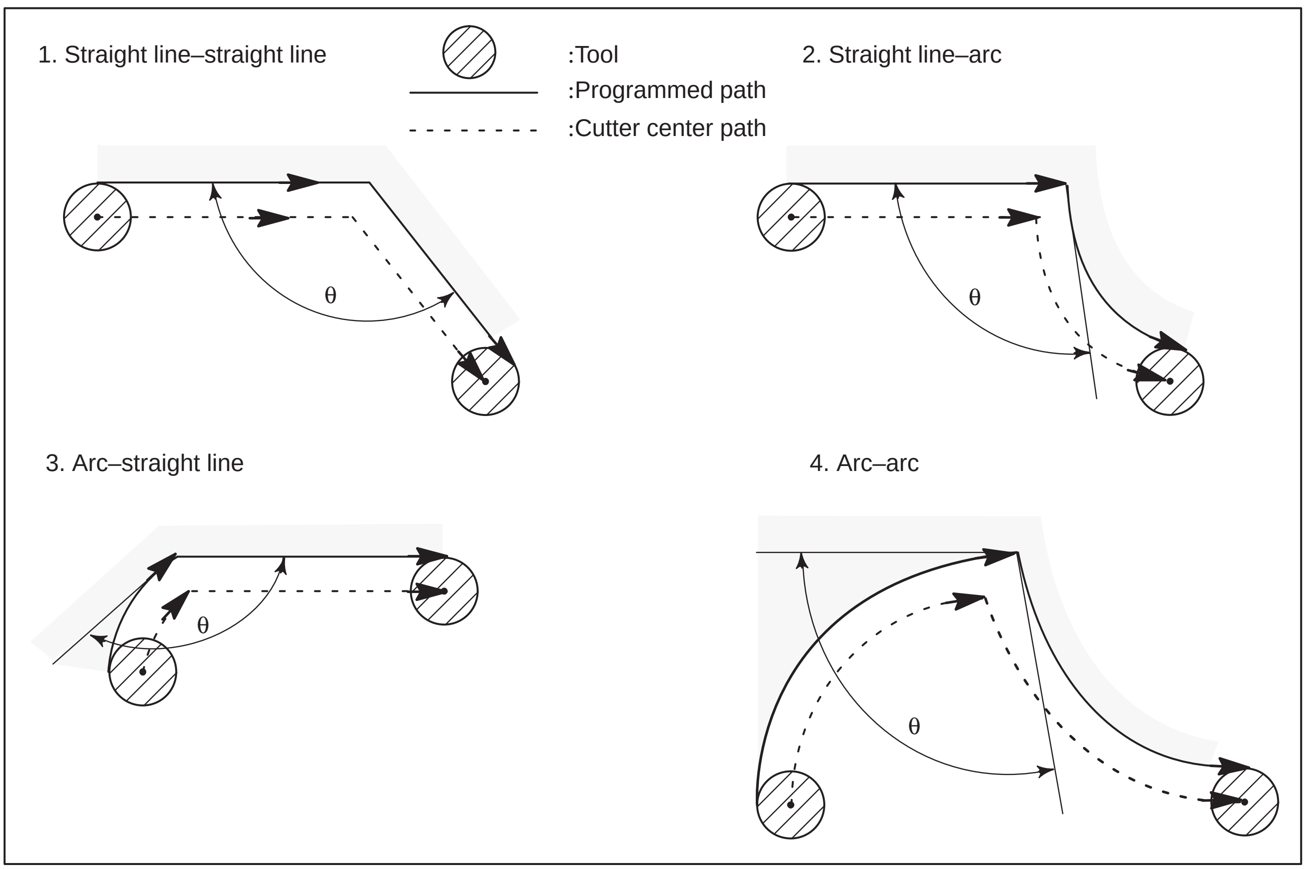


Fig. 7.1.8 (a) Inner corner

Note 1 When the block before a corner is a start–up block, or the block after a corner includes G41 or G42, the feedrate is not overridden. The feedrate override function is disabled when the offset value is 0.

- **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)

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. 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 radius

It is also valid for the dry run and the F1–digit feed command.

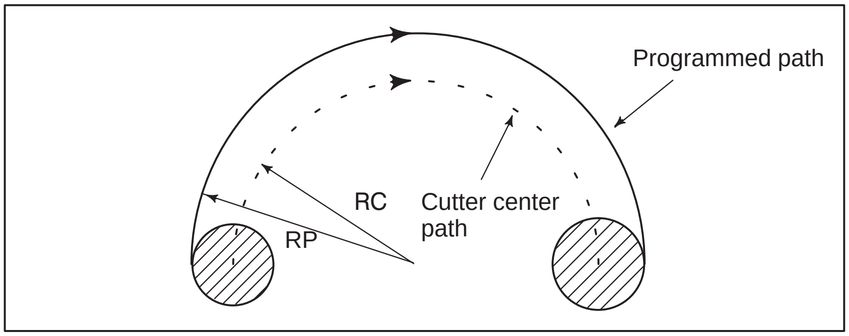


Fig. 7.1.8 (b) Internal circular cutting feedrate change

If Rc is much smaller than Rp, $Rc/Rp \doteq 0$; the tool stops. A minimum deceleration ratio (MDR) is to be specified with parameter No. 1710. When $Rc/Rp \leqq MDR$, the feedrate of the tool is $(F \times MDR)$.

Note 2 When internal circular cutting must be performed together with automatic override for inner corners, the feedrate of the tool is as follows:

$$F \times \frac{Rc}{Rp} \times \text{(inner corner override)} \times \text{(feedrate override)}$$

Parameter

1710	Minimum deceleration ratio (MDR) of the inner circular cutting rate in automatic corner override

[Data type] Byte

[Unit of data] %

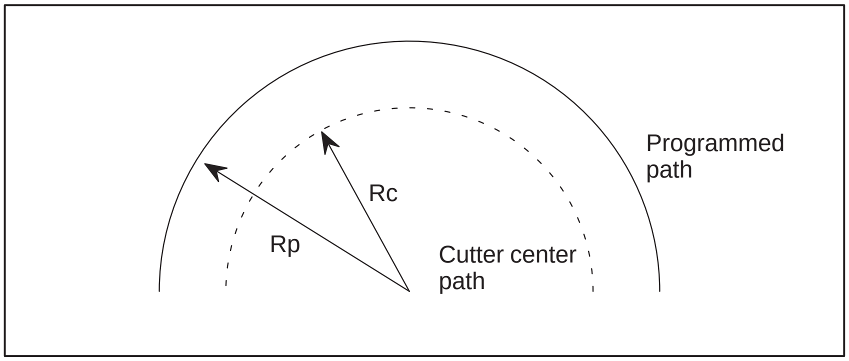
[Valid data range] 1 to 100

Set the minimum deceleration ratio (MDR) in changing the inner circular cutting feedrate by automatic corner override.

In circular cutting with an inward offset, the actual feedrate for a specified feedrate (F) becomes as follows:

$$F \times \frac{Rc}{Rp} \times \left(\begin{array}{l} \text{Rc: Radius of the path of the cutter's center} \\ \text{Rp: Programmed radius} \end{array} \right)$$

As the actual feedrate becomes the value obtained from the above equation, the specified rate F can be achieved on the program path.



If Rc is too samall in comparison with Rp so that $\frac{Rc}{Rp} \doteq 0$, the cutter will stop. To prevent this, the minimum deceleration ratio (MDR) is set.
When $\frac{Rc}{Rp} \doteq 0$,
the actural rate becomes as follows:
F × (MDR)

1711	Angle (θp) to recognize the inner corner in automatic override

[Data type] Byte

[Unit of data] Degree

[Valid data range] 1 to 179 (standard value = 91)

Set the angle to recognize the inner corner when automatic corner override is performed for the inner corner

1712	Amount of automatic override for an inner corner

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 100 (standard value = 50)

Set inner corner automatic override value when automatic corner override is performed.

1713	Distance Le from the starting point in inner corner automatic override

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Input in mm	1	0.1	0.01	mm
Input in inches	0.1	0.01	0.001	inch

[Valid data range] 0 to 3999

Set distance Le from the starting point in an inner comer for automatic corner override.

[Data type] Word

1714	Distance Ls up to the ending point in inner corner automatic override

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Input in mm	1	0.1	0.01	mm
Input in inches	0.1	0.01	0.001	inch

[Valid data range] 0 to 3999

Set distance Ls up to the end point in an inner corner for automatic corner override.

If $\theta \leqq \theta p$, the inside of a comer is recognized. (θ is set in parameter 1711.)

When an inner corner is recognized, the feedrate is overridden in the range of Le in the block immediately before the intersection of the corner and Ls in the next block following the intersection.

Ls and Le are each a straight line connecting the intersection of the corner and a given point on the path of the cutter's center.

Ls and Le are set in parameters 1713 and 1714.

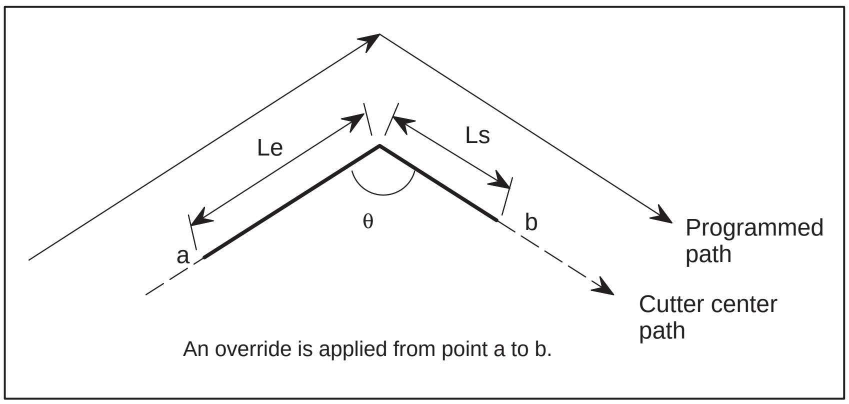


Fig. 7.1.8 (c) Distance Le and Ls in the automatic corner override at an inner corner

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.5.4.2	Automatic Override for Inner Corners
	II.5.4.3	Internal Circular Cutting Feedrate Change

7.1.9 External Deceleration

General

These signals decelerate the feedrate of the control axes down to the speed which has been set by parameter No. 1426 and 1427.

Signal

External deceleration signal *+ED1 to *+ED8<G118> *–ED1 to *–ED8<G120>

[Classification] Input signal

[Function] These signals are used to apply deceleration; provided for each direction of each control axis; +/– indicates the direction, while the signal number corresponds to the number of the controlled axis.

* + ED ×

1 … The 1st axis is decelerated.

2 … The 2nd axis is decelerated.

3 … The 3rd axis is decelerated.

\+ … 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 decelerate to stop in the specified direction.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G118	*+ED8	*+ED7	*+ED6	*+ED5	*+ED4	*+ED3	*+ED2	*+ED1

	#7	#6	#5	#4	#3	#2	#1	#0
G120	*–ED8	*–ED7	*–ED6	*–ED5	*–ED4	*–ED3	*–ED2	*–ED1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1005			EDMx	EDPx				

[Data type] Bit axis

EDPx External deceleration signal in the positive direction for each axis
0 : Valid only for rapid traverse
1 : Valid for rapid traverse and cutting feed

EDMx External deceleration signal in the negative direction for each axis
0 : Valid only for rapid traverse
1 : Valid for rapid traverse and cutting feed

1426	External deceleration rate of cutting feed

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800

Set the external deceleration rate of cutting feed.

1427	External deceleration rate of rapid traverse for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

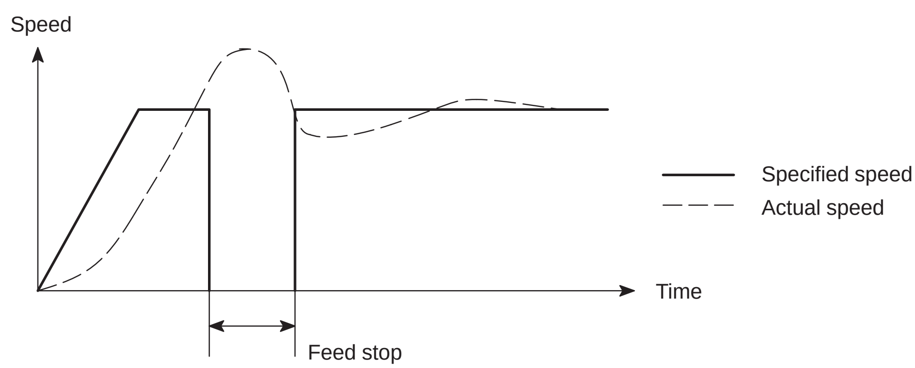
Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the external deceleration rate of rapid traverse for each axis.

7.1.10
Feed Stop Function

General

During axis motion, the feed stop function checks a position deviation amount at all times. When the amount exceeds the “feed stop position deviation amount” set by the parameter (No. 1832), the function suspends pulse distribution and acceleration/deceleration control during such a period of time, and terminates the move command for the positioning control circuit. Thus the function can minimize an overshoot that may occur with a large servo motor in rapid traverse acceleration operation.



Parameter

1832	Feed stop positioning deviation for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Set the feed stop positioning deviation for each axis.

If the positioning deviation exceeds the feed stop positioning deviation during movement, pulse distribution and acceleration/deceleration control are stopped temporarily. When the positioning deviation drops to the feed stop positioning deviation or below, pulse distribution and acceleration/deceleration control are resumed.

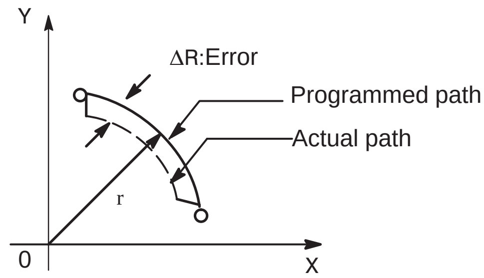
The feed stop function is used to reduce overshoot in acceleration/deceleration mainly by large servo motor.

Generally, set the middle value between the positioning deviation limit during movement and the positioning deviation at rapid traverse as the feed stop positioning deviation.

7.1.11 Feedrate Clamping By Arc Radius

General

When an arc is cut at a high speed in circular interpolation, a radial error exists between the actual tool path and the programmed arc. An approximation of this error can be obtained from the following expression:



$$\Delta R = \frac{1}{2}(T_1^2 + T_2^2)\frac{V^2}{R}$$

ΔR:Maximum radial error (mm)
V:Feedrate (mm/s)
R:Arc radius (mm)
T_1:Time constant (s) for exponential acceleration/deceleration of cutting feed
T_2:Time constant of the servo motor (s)

When actual machining is performed, radius r of the arc to be machined and permissible error Δr are given. Then, maximum allowable feedrate v (mm/min) is determined from the above expression.

The function for clamping the feedrate by the arc radius automatically clamps the feedrate of arc cutting to the value set in a parameter. This function is effective when the specified feedrate may cause the radial error for an arc with a programmed radius to exceed the permissible degree of error.

Parameter

1730	Maximum feedrate for arc radius R

[Data type] Word

[Unit of data]

[Valid data range]

.Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	8 – 15000	0 – 12000
Inch machine	0.1 inch/min	8 – 6000	0 – 4800

Set a maximum feedrate for the arc radius set in parameter No. 1731. Set this parameter when the arc radius–based feedrate clamping function is enabled.

1731	Arc radius value corresponding to a maximum feedrate

[Data type] Two–word

[Unit of data]

Unit	IS–A	IS–B	IS–C	Unit
Linear axis (millimeter machine)	0.01	0.001	0.0001	mm
Linear axis (inch machine)	0.001	0.0001	0.00001	inch

[Valid data range] 1000 to 99999999

Set the arc radius corresponding to the maximum feedrate set in parameter No. 1730. Set this parameter when the arc radius–based feedrate clamping function is enabled.

1732	Minimum value (RV min) for arc radius–based feedrate clamp

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0 – 15000	0 – 12000
Inch machine	0.1 inch/min	0 – 6000	0 – 4800

The arc radius–based feedrate clamping function reduces the maximum feedrate as the arc radius decreases. When the specified maximum feedrate is not greater than RV min (minimum value for arc radius–based feedrate clamping), RV min is used as the maximum feedrate.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.20.1	Feedrate clamp by circle radius

7.1.12 Automatic Corner Deceleration (M series)

General

This function automatically controls the feedrate during corner machining according to the angle of a corner made by machining blocks or according to the feedrate difference for each axis.

This function is enabled when G64 (machining) mode is selected and deceleration of the first of two consecutive cutting feed blocks is executed.

Feedrate control can be performed according to the angle of a corner made by machining blocks or according to the feedrate difference for each axis. The desired method is selected by specifying the corresponding value in the CSD bit (bit 4 of parameter No. 1602).

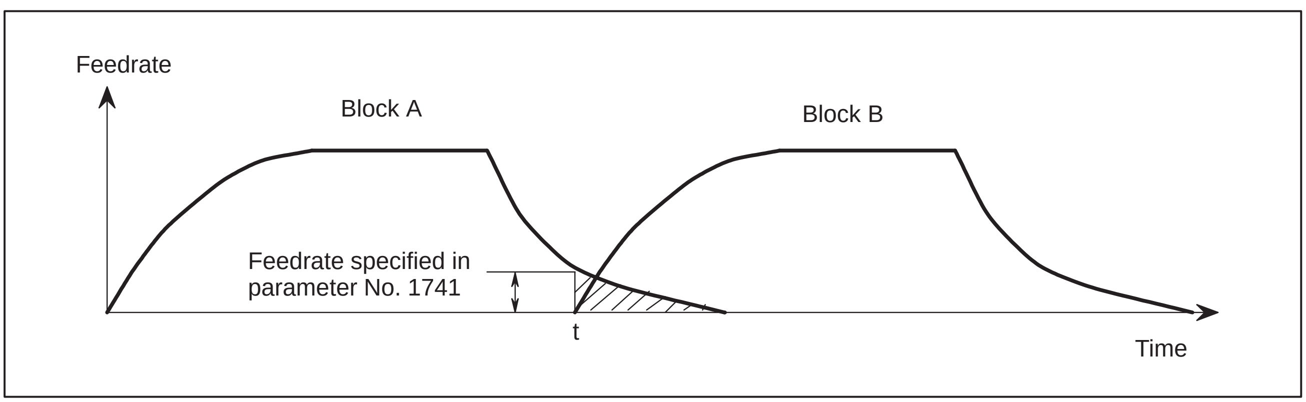
Feedrate control according to corner angle

• Overview

If the angle made by blocks A and B is smaller than that specified in parameter No. 1740 (for the selected plane), and if the feedrates along the first and second axes on that plane are lower than that specified in parameter No. 1741, the system executes block B, assuming that no pulses are accumulated.

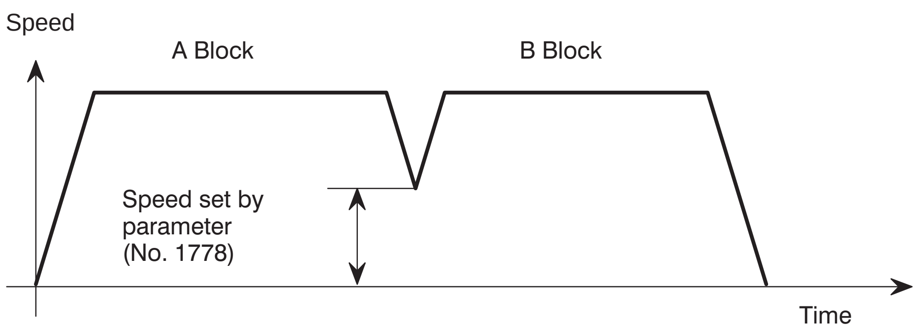
The figure shows the relationship between feedrate and time when a corner angle is smaller than the angle specified in the parameter.

At time t, some accumulated pulses remain, as indicated by the shaded part. The system, however, starts the next block because the feedrate of the automatic acceleration/deceleration circuit is lower than that specified in the parameter.



· **When linear acceleration/deceleration before interpolation for cutting feed is enabled**

If the angle made by blocks A and B is smaller than that specified in parameter No. 1740 (for the selected plane), and if the feedrates programmed for blocks A and B are higher than the value set in parameter No. 1778, the feedrate is reduced to the value specified in the parameter in block A. In block B, the feedrate is increased to the programmed feedrate. The rate of acceleration depends on the parameter for linear acceleration/deceleration before interpolation for cutting feed.



● Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601		ACD						

[Data type] Bit

ACD Function for automatically reducing the feedrate at corners (automatic corner override function)
0 : The function is not used.
1 : The function is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1602				CSD				

[Data type] Bit

CSD In the function for automatically reducing a feedrate at corners,
0 : Angles are used for controlling the feedrate.
1 : Differences in feedrates are used for controlling the feedrate.

1740	Critical angle subtended by two blocks for automatic corner deceleration

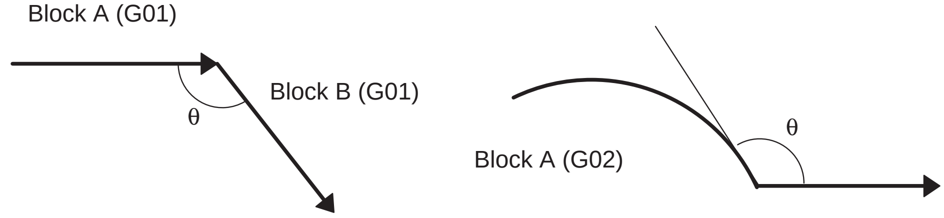
[Data type] Two–word

[Unit of data] 0.001 deg

[Valid data range] 0 to 180000

Set a critical angle to be subtended by two blocks for corner deceleration when the angle–based automatic corner deceleration function is used.

The angle subtended by two blocks is defined as θ in the examples shown below.



1741	Feedrate for assuming the termination of automatic corner deceleration (for acceleration/deceleration after interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the feedrate for assuming the termination of deceleration in automatic corner deceleration.

1778	Minimum speed for the automatic corner deceleration function (for linear acceleration/deceleration before interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set a speed at which the number of buffered pulses in deceleration is assumed to be 0 when linear acceleration/deceleration before interpolation is used.

Notes

Note 1 The angle of the machining tool path is compared with that specified in parameter No. 1740 only for the selected plane.
The actual feedrate and that specified in parameter No. 1741 are compared only for the first and second axes of the selected plane. Even if simultaneous movement is performed along three or more axes, the feedrates of only the first and second axes are compared for the selected plane.

Note 2 The roundness of a corner is determined by the angle and feedrate specified in parameter Nos. 1740 and 1741, respectively. If a sharp corner is always required, set a feedrate of zero and an angle of 180000 (180 degrees).

Note 3 If a G09 (exact stop) command is executed, an exact stop is performed, irrespective of the angle and feedrate specified in parameter Nos.1740 and 1741.

Note 4 This function is disabled in single block and dry run mode.

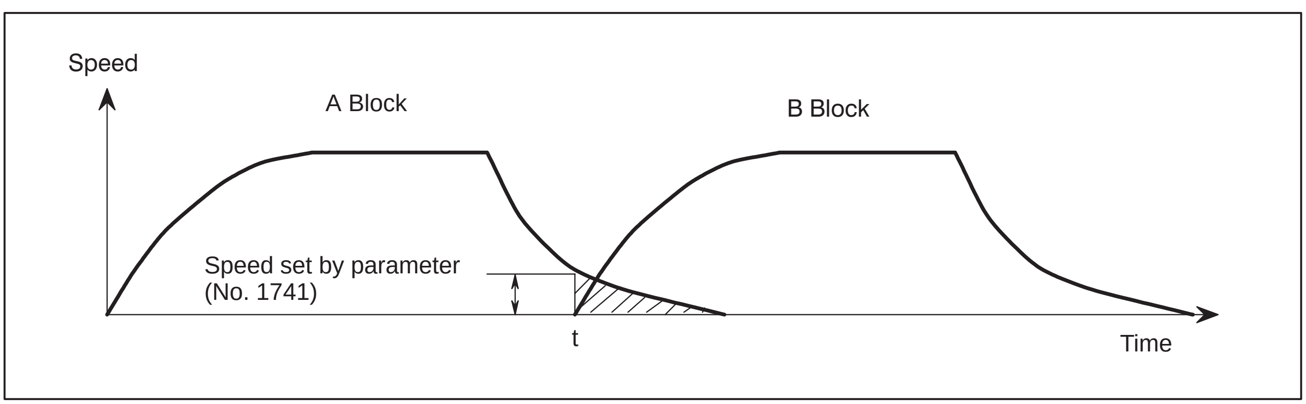
Feedrate control according to the feedrate difference for each axis

Overview

If the difference between the programmed feedrates at the end of block A and at the beginning of block B for each axis exceeds the value specified in parameter No. 1781, and if the feedrates for all axes are lower than that specified in parameter No. 1741, the system executes block B, assuming that no pulses are accumulated.

The figure shows the relationship between the feedrate and time when the feedrate difference for each axis exceeds the value specified in parameter No. 1781.

At time t, some accumulated pulses remain, as indicated by the shaded section. The system, however, starts the next block because the feedrate of the automatic acceleration/deceleration circuit is lower than that specified in parameter No. 1741.



· **When linear acceleration/deceleration before interpolation for cutting feed is enabled**

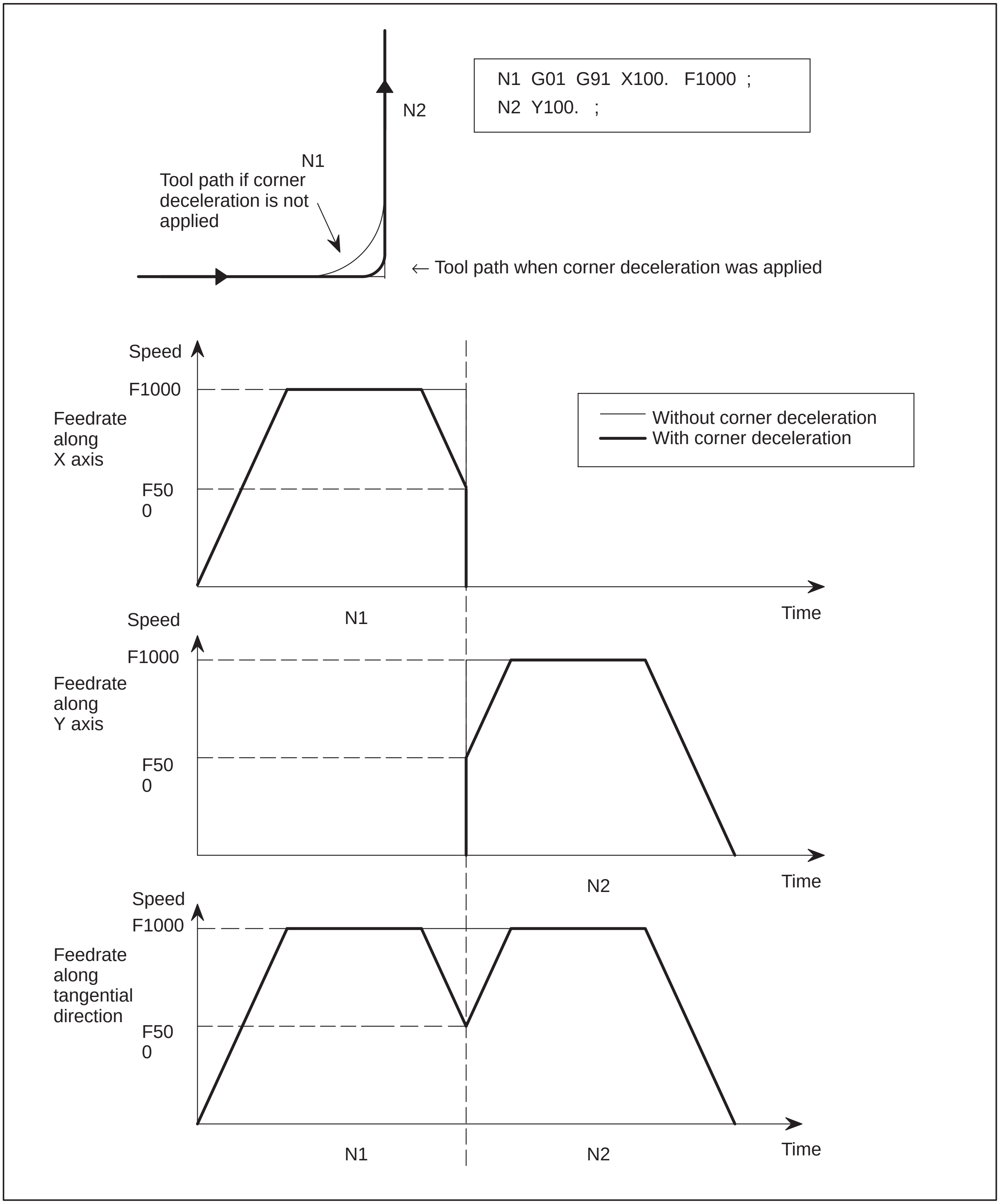
If the difference between the feedrates of blocks A and B for each axis exceeds the value specified in parameter No. 1780, the feedrate at the corner is calculated from the difference for each axis, as shown below. The feedrate is reduced to the calculated value in block A.

The feedrate change for each axis (Vc[X], Vc[Y], ...), caused by the movement at programmed feedrate F, is compared with Vmax specified in parameter No. 1780. If an feedrate change exceeding Vmax is detected, the target feedrate after deceleration Fc is calculated, using maximum comparison value Rmax.

$$R = \frac{Vc}{Vmax}$$

$$Fc = \frac{F}{Rmax}$$

If, for example, the direction of movement is changed from the X–axis to the Y–axis, that is through 90 degrees, and if the programmed feedrate is 1000 mm/min and the permissible feedrate difference specified in parameter No. 1780 is 500 mm/min, the deceleration shown below is performed:



Different permissible feedrate differences can be specified for different axes. If a value is specified in parameter No. 1783, the permissible feedrate difference for each axis becomes valid. Deceleration at a corner is calculated for the axis for which the permissible feedrate difference is exceeded with the highest ratio of actual feedrate difference to permissible feedrate difference.

● Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601		ACD						

[Data type] Bit

ACD Function for automatically reducing the feedrate at corners (automatic corner override function)
0 : The function is not used.
1 : The function is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1602				CSD				

[Data type] Bit

CSD In the function for automatically reducing a feedrate at corners,
0 : Angles are used for controlling the feedrate.
1 : Differences in feedrates are used for controlling the feedrate.

1741	Feedrate for assuming the termination of automatic corner deceleration (for acceleration/deceleration after interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the feedrate for assuming the termination of deceleration in automatic corner deceleration.

1780	Allowable speed difference for the speed difference–based automatic corner deceleration function (for linear acceleration/deceleration before interpolation)

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800

Set the speed difference for the speed difference–based automatic corner deceleration function when linear acceleration/deceleration before interpolation is used.

1781	Allowable speed difference for the speed difference–based automatic corner deceleration function (linear acceleration/deceleration after interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set a speed difference for the speed difference–based automatic corner deceleration function when linear acceleration/deceleration after interpolation is used.

- **Note**

Note 1 Even during dry run or external deceleration, the feedrate difference is checked according to the F command in the program.

Note 2 If the G09 (exact stop) command is executed, an exact stop is performed, irrespective of the values specified for the parameters.

Note 3 This function is invalid for the feed per rotation command, F1–digit feed command, and rigid tapping command, as well as in single block mode.

Note 4 If the override is changed during operation, the feedrate difference cannot be checked correctly.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.5.4.4	Automatic corner deceleration

7.1.13 Look–Ahead Control

General

This function is designed for high–speed precise machining. With this function, the delay due to acceleration/deceleration and the delay in the servo system which increase as the feedrate becomes higher can be suppressed.
The tool can then follow specified values accurately and errors in the machining profile can be reduced.
This function becomes effective when look–ahead control mode is entered by G08P1 command.

● Available functions

In look–ahead control mode, the following functions are available:

(1) Linear acceleration/deceleration before interpolation for cutting feed
(2) Automatic corner deceleration function
(3) Block overlap function

For details on the above functions, see the descriptions of the functions.

Signal

Look–ahead control mode signal G08MD <F066#0>

[Classification] Output signal

[Function] Informs that is in the look–ahead control mode.

[Output condition] The signal is “1” in the following case:
·In the look–ahead control mode
The signal is “0” in the following case:
·It is not the look–ahead control mode

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F066								G08MD

Parameter

1431	Maximum cutting feedrate for all axes in the look–ahead control mode

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0 – 240000	0 – 100000
Inch machine	0.1 inch/min	0 – 96000	0 – 48000
Rotaion axis	1 deg/min	0 – 240000	0 – 100000

Specify the maximum cutting feedrate for all axes in the look–ahead control mode.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

Note 1 To specify the maximum cutting feedrate for each axis, use parameter No. 1432 instead.

Note 2 In a mode other than the look–ahead mode, the maximum cutting feedrate specified in parameter No. 1422 or No. 1430 is applied and the feedrate is clamped at the maximum feedrate.

1432	Maximum cutting feedrate for each axis in the look–ahead control mode

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0 – 240000	0 – 100000
Inch machine	0.1 inch/min	0 – 96000	0 – 48000
Rotaion axis	1 deg/min	0 – 240000	0 – 100000

Specify the maximum cutting feedrate for each axis in the look–ahead control mode.

A feedrate for each axis is clamped during cutting feed so that it does not exceed the maximum cutting feedrate specified for each axis.

Note 1 This parameter is effective only in linear and circular interpolation. In polar coordinate, cylindrical, and involute interpolation, the maximum feedrate for all axes specified in parameter No. 1431 is effective.

Note 2 If a setting for each axis is 0, the maximum feedrate specified in parameter No. 1431 is applied to all axes and the feedrate is clamped at the maximum feedrate.

Note 3 In a mode other than the look–ahead mode, the maximum cutting feedrate specified in parameter No. 1422 or No. 1430 is applied and the feedrate is clamped at the maximum feedrate.

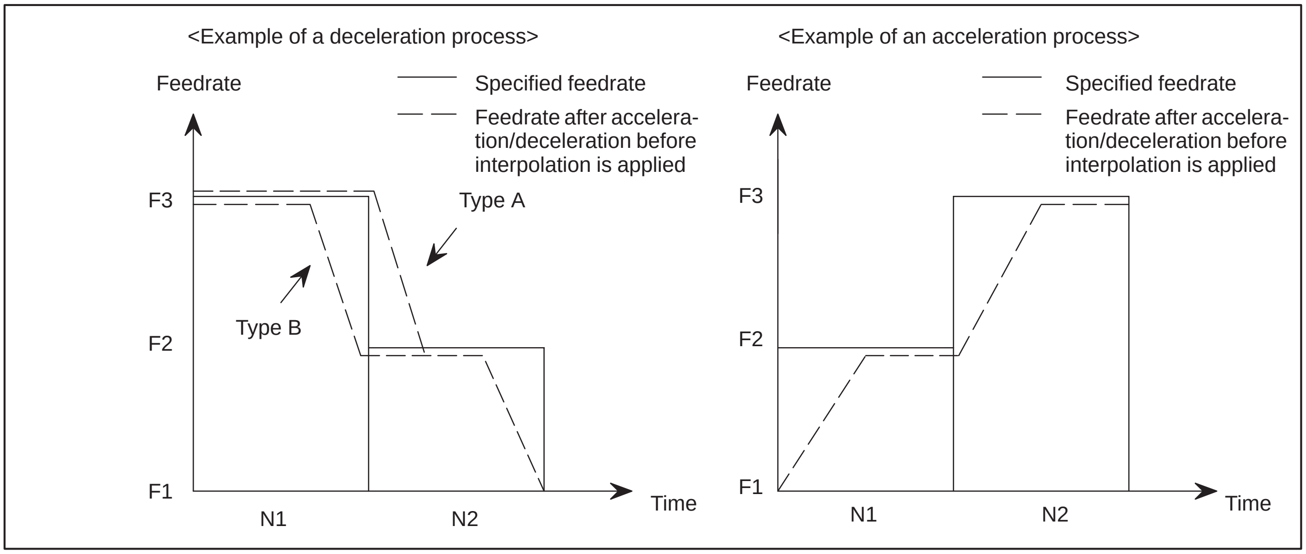
	#7	#6	#5	#4	#3	#2	#1	#0
1602		LS2		CSD				FWB

[Data type] Bit

FWB Cutting feed acceleration/deceleration before interpolation
0 : Type A of acceleration/deceleration before interpolation is used.
1 : Type B of acceleration/deceleration before interpolation is used.

Type A: When a feedrate is to be changed by a command, acceleration/deceleration starts after the program enters the block in which the command is specified.

Type B: When a feedrate is to be changed by a command, deceleration starts and terminates at the block before the block in which the command is specified.



CSD In the function for automatically reducing a feedrate at corners,
0 : Angles are used for controlling the feedrate.
1 : Differences in feedrates are used for controlling the feedrate.

LS2 Acceleration/deceleration after interpolation for cutting feed in the look–ahead control mode
0 : Exponential acceleration/deceleration
1 : Linear acceleration/deceleration. (The function for linear acceleration/deceleration after interpolation for cutting feed is required.)

1762	Exponential acceleration/deceleration time constant for cutting feed in the look–ahead control mode

[Data type] Word axis

[Unit of data] 1 ms

[Valid data range] 0 to 4000

Set an exponential acceleration/deceleration time constant for cutting feed in the look–ahead control mode.

1763	Minimum speed in exponential acceleration/deceleration for cutting feed in the look–ahead control mode

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set a minimum speed (FL) in exponential acceleration/deceleration for cutting feed in the look–ahead control mode.

1768	Time constant for linear acceleration/deceleration during cutting feed in the look–ahead control mode.

[Data type] Word

[Unit of data] ms

[Valid data range] 8 to 512

This parameter sets a time constant for linear acceleration/deceleration for cutting feed in the look–ahead control mote.

Note 1 The function for linear acceleration/deceleration after interpolation for cutting feed is required.

1770	Parameter 1 for setting an acceleration for linear acceleration/deceleration before interpolation in the look–ahead control mode (maximum machining speed during linear acceleration/deceleration before interpolation)

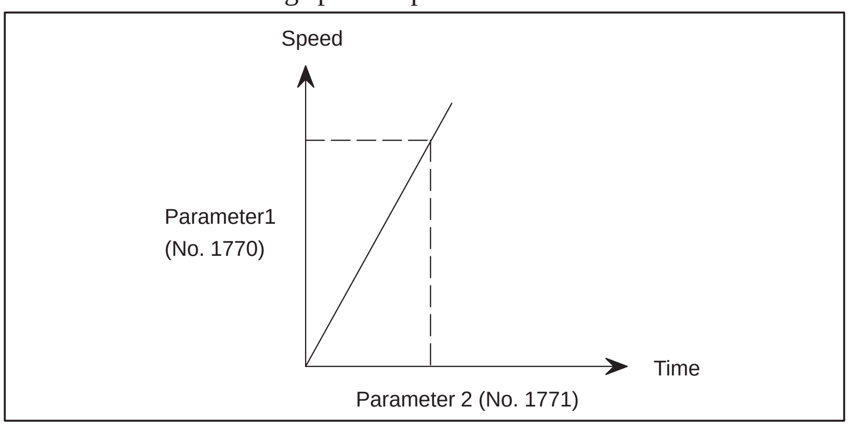
[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

This parameter is used to set an acceleration for linear acceleration/deceleration before interpolation in the look–ahead control mode.
In this parameter, set the maximum machining speed during linear acceleration/deceleration before interpolation. Set the time used to reach the maximum machining speed in parameter No. 1771.



Note 1 When 0 is set in parameter No. 1770 or parameter No. 1771, linear acceleration/deceleration before interpolation is disabled.

1771	Parameter 2 for setting an acceleration for linear acceleration/deceleration before interpolation in the look–ahead control mode (time used to reach the maximum machining speed during linear acceleration/deceleration before interpolation)

[Data type] Word

[Unit of data] 1 ms

[Valid data range] 0 to 4000

This parameter is used to set an acceleration for linear acceleration/deceleration before interpolation in the look–ahead control mode. In this parameter, set the time (time constant) used to reach the speed set in parameter No. 1770.

Note 1 When 0 is set in parameter No. 1770 or parameter No. 1771, linear acceleration/deceleration before interpolation is disabled.

Note 2 In parameter Nos. 1770 and 1771, set values that satisfy the following:
Parameter No. 1770/Parameter No. 1771 ≥ 5

1777	Minimum speed for the automatic corner deceleration function (for the look–ahead control)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set a speed at which the number of buffered pulses in deceleration is assumed to be 0 when linear acceleration/deceleration before interpolation is used.

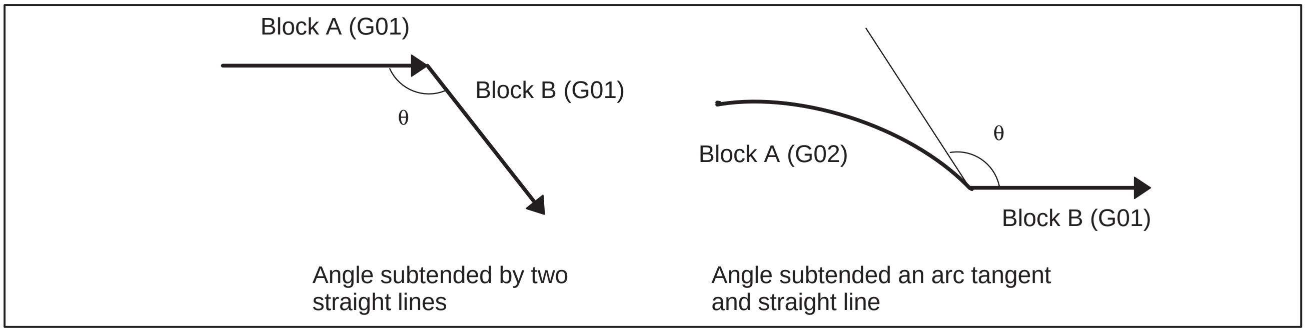
1779	Critical angle subtended by two blocks for automatic corner deceleration (for the look–ahead control)

[Data type] Two–word

[Unit of data] 0.001 deg

[Valid data range] 0 to 180000

Set a critical angle to be subtended by two blocks for corner deceleration when the angle–based automatic corner deceleration function is used. The angle subtended by two blocks is defined as θ in the examples shown below.



1780	Allowable speed difference for the speed difference based corner deceleration function (for linear acceleration/deceleration before interpolation)

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800

Set the speed difference for the speed difference based automatic corner deceleration function when linear acceleration/deceleration before interpolation is used.

1783	Allowable speed difference for the speed difference based corner deceleration function (for linear acceleration /deceleration before interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

A separate allowable feedrate difference can be set for each axis. Among the axes that exceed the specified allowable feedrate difference, the axis with the greatest ratio of the actual feedrate difference to the allowable feedrate difference is used as the reference to calculate the reduced feedrate at the corner.

1784	Speed when overtravel alarm has generated during acceleration/deceleration before interpolation

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800

Deceleration is started beforehand to reach the feedrate set in the parameter when an overtravel alarm is issued (when a limit is reached) during linear acceleration/deceleration before interpolation. By using this parameter, the overrun distance that occurs when an overtravel alarm is output can be reduced.

Note 1 When 0 is set in this parameter, the control described above is not exercised.

Note 2 Use type–B linear acceleration/deceleration before interpolation (by setting bit 0 (FWB) of parameter No. 1602 to 1).

Note 3 The control described above is applicable only to stored stroke limit 1.

Parameters for look–ahead control mode and normal mode

· **Parameters for the cutting feed acceleration/deceleration before interpolation**

Parameter description	Parameter No.	
	Normal mode	Look–ahead control mode
Acceleration/deceleration type (A type/B type)	FWB (1602#0)	←
Acceleration (Parameter 1)	1630	1770
Acceleration (Parameter 2)	1631	1771
Speed when overtravel alarm has generated	1784	←

· **Parameters for automatic corner deceleration**

Parameter description	Parameter No.	
	Normal mode	Look–ahead control mode
Automatic corner deceleration according to the corner angle or the speed difference	CSD (1602#4)	←
Minimum speed (according to the corner angle)	1778	1777
Critical angle (according to the corner angle)	1740	1779
Allowable speed difference for all axes (according to speed difference)	1780	←
Allowable speed difference for each axis (according to speed difference)	1783	←

Alarm and message

Number	Message	Description
109	P/S ALARM	A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.

Note

In the look–ahead control mode, the functions listed below cannot be specified. To specify these functions, cancel the look–ahead control mode, specify the desired function, then set look–ahead control mode again.

·Rigid tapping function
·Cs contour axis control function
·Feed per rotation
·F1–digit feed
·C–axis normal direction control function
·Polar coordinate interpolation function
·Cylindrical interpolation function
·Involute interpolation function
·Exponential interpolation
·Three–dimensional coordinate conversion
·Retrace function
·Normal direction control
·Polar coordinate command
·Index table indexing
·Tool withdrawal and return
·Threading and synchronous feed
·High–speed cycle machining
·Handle interrupt
·Program restart
·Simplified synchronization control
·Feed stop
·High–speed skip function
·Constant surface speed control
·Interrupt type custom macro
·Small–diameter peck drilling cycle
·High–speed remote buffer A/B
·Automatic tool length measurement
·Skip cutting
·G28 (low–speed reference position return)

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.20.2	LOOK–AHEAD CONTROL (G08)

7.1.14 High–precision Contour Control by RISC (M series)

General

Some machining errors are due to the CNC. Such errors include machining errors caused by acceleration/deceleration after interpolation. To eliminate these errors, the following functions are performed at high speed by an RISC processor. These functions are called high–precision contour control functions.

- Function for multiple–block look–ahead acceleration/deceleration before interpolation. This function eliminates machining errors due to acceleration/deceleration.
- Automatic feedrate control function which enables smooth acceleration/ deceleration by considering changes in the figure and speed and allowable acceleration for the machine. This is performed by reading multiple blocks in advance.

Furthermore, smoother acceleration/deceleration is achieved, enabling the feed–forward factor to be increased. This feature also reduces follow–up error in the servo system.

Name	Function
No. of controlled axes	1 axis to 8 axes
No. of simultaneously controlled axes	Up to max. controlled axes
Axis names	Any of A, B, C, U, V, W, X, Y, Z
Increment system	0.01, 0.001, 0.0001 mm 0.001, 0.0001, 0.00001 inch
Max. programming dimensions	$\pm$8 digits
Positioning	Yes (Available with parameter MSU (No. 8403#1)=1)
Linear interpolation	Yes
Multi–quadrant circular interpolation	Yes
Feed per minute	Yes
Feedrate clamp	Yes
Feedrate override	0 – 254%, Every 1%
2nd feedrate override	0 – 254%, Every 1% ☆
Workpiece coordinate system	Yes (Unchangeable in G05P10000 mode) ☆
Absolute/incremental command	Combined use possible in the block
Sequence number	5 digits
Tape code	EIA, ISO

Name	Function
Tape format	Word address format
Control in/out	Yes
Optional block skip	Yes
Circle radius R specification	Yes
Automatic operation	Memory operation, Tape operation
Method of tape operation	RS–232–C, RS–422, DNC1, and remote buffer
Manual absolute on/off	Yes (FS15 type)
Cycle start, Feed hold	Yes
Dry run	Yes
Feedrate override under dry run	0 – 655.34%, Every 0.01%
Single block	Yes
Inch/metric conversion	Yes (Unchangeable in G05P10000 mode)
Multi–buffer	Yes
Cutter compensation C	Yes
Interlock (all axes)	Yes
Machine lock	Yes
Subprogram call (M98, M198)	Yes (Usable with parameter MSU (No. 8403#1)=1)
Auxiliary function	Yes (Usable with parameter MSU (No. 8403#1)=1)

☆ (Option concerned required)

- **Data that can be specified**

G00 : Positioning (Note 1)
G01 : Linear interpolation
G02 : Circular interpolation (CW)
G03 : Circular interpolation (CCW)
G17 : Plane selection (XpYp plane)
where, Xp is the X–axis or its parallel axis;
G18 : Plane selection (ZpXp plane)
where, Yp is the Y–axis or its parallel axis;
G19 : Plane selection (YpZp plane)
where, Zp is the Z–axis or its parallel axis.
G38 : Cutter compensation C with vector held
G39 : Cutter compensation C corner arc
G40 : Cutter compensation C cancel
G41 : Cutter compensation C, left
G42 : Cutter compensation C, right
G90 : Absolute command
G91 : Incremental command
Dxxx : D code
Fxxxxx : F code
Nxxxxx : Sequence number
G05P10000 : Setting the HPCC mode

G05P0	:	Canceling the HPCC mode
I, J, K, R	:	I, J, K, and R specified for circular interpolation
( )	:	Control in / out
/n	:	Optional block skip
Mxxxx	:	Miscellaneous function (Note 1)
Sxxxx	:	Spindle speed function (Note 1)
Txxxx	:	Tool function (Note 1)
Bxxxx	:	Second miscellaneous function (Note 1)

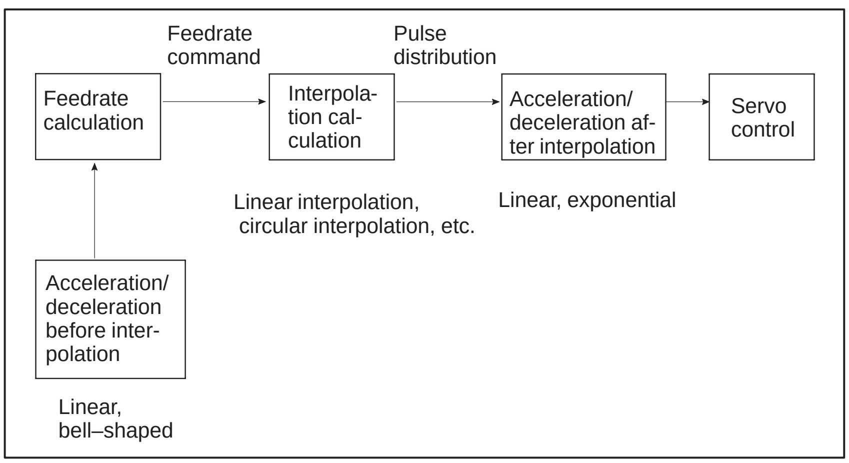
Note 1 Before G00, miscellaneous function, spindle speed function, tool function and second miscellaneous function can be specified in high–precision contour control mode (HPCC mode), bit 1 (MSU) of parameter No. 8403 must be set to 1. Otherwise, an alarm will be issued.

7.1.14.1 Look–ahead acceleration/deceleration before interpolation

When feed per minute is specified, this function reads several tens of blocks ahead to perform acceleration/deceleration before interpolation, that is, to apply acceleration/deceleration to the specified feedrate.

When acceleration/deceleration after interpolation is used, acceleration/deceleration is applied to the interpolated data. Consequently, the interpolated data is changed by acceleration/deceleration. When acceleration/deceleration before interpolation is used, however, acceleration/deceleration is applied to the feedrate data before interpolation. Consequently, the interpolated data is not changed by acceleration/deceleration.

Accordingly, interpolation data ensures that machining follows a specified line or curve at all times, thus eliminating the machining profile errors that result from delays in acceleration/deceleration.

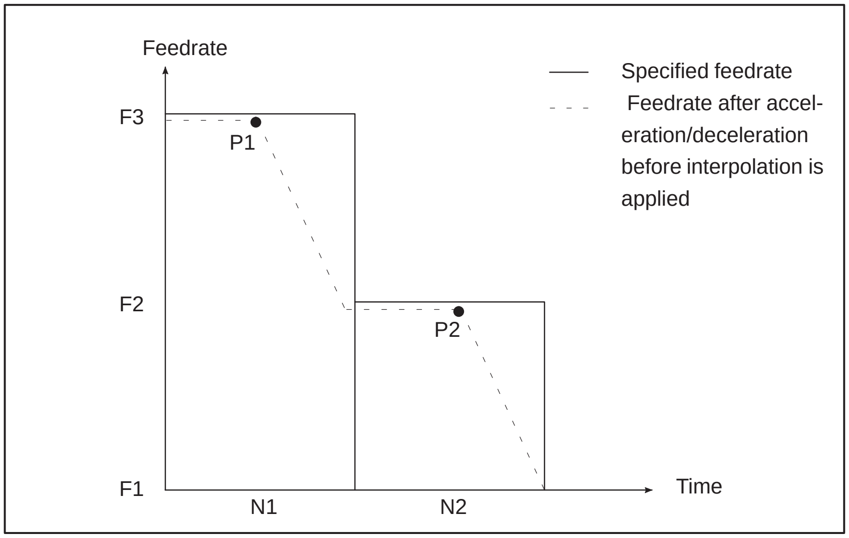


If a feedrate change along any axis is greater than the value set in a parameter for the joint (corner) between two successive blocks, a feedrate is calculated so that the difference in the feedrates does not exceed the specified value. The feedrate is automatically reduced to this calculated value at the corner.

Before this function can be used, specify parameter Nos. 8400 and 8401 for determining an acceleration for acceleration/deceleration before interpolation.

Example of deceleration

To ensure that the feedrate specified for a block is reached when the block is executed, deceleration is started in the previous block.

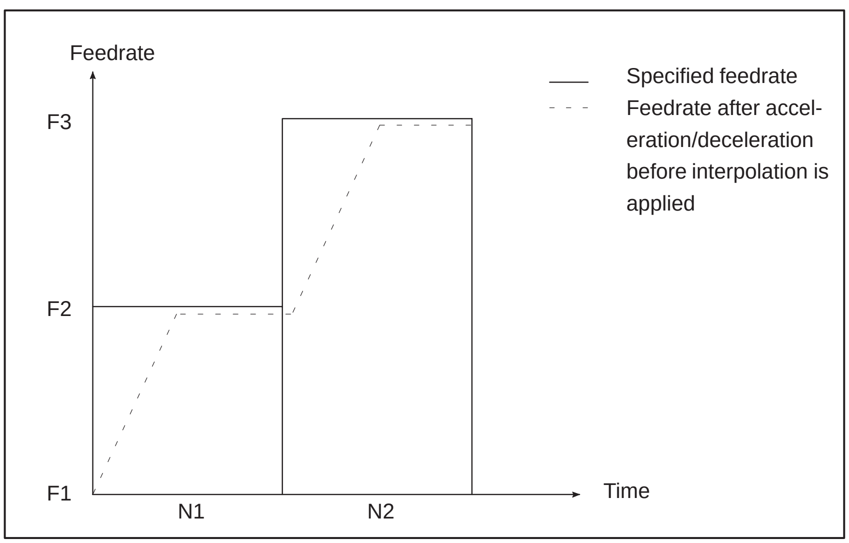


To reduce feedrate F3 to feedrate F2, deceleration must be started at P1. To reduce feedrate F2 to feedrate F1, deceleration must be started at P2.

The tool can be decelerated over several blocks, because several tens of blocks are read in advance.

Example of acceleration

Acceleration is started to reach the specified feedrate for a block when the block is executed.



Look–ahead bell–shaped acceleration/deceleration before interpolation

To use this function, set bit 7 (BADO) and bit 1 (NWBL) of parameter No. 8402 to 1, and also set the following parameters:

Parameter No. 8400: Parameter 1 for setting the acceleration used for acceleration/deceleration before interpolation
Parameter No. 8401: Parameter 2 for setting the acceleration used for acceleration/deceleration before interpolation

Parameter No. 8402, bit 5 (DST) = 1, bit 4 (BLK) = 0
Parameter No. 8416: Time needed to reach maximum allowable acceleration

For details, see the description of the parameters.

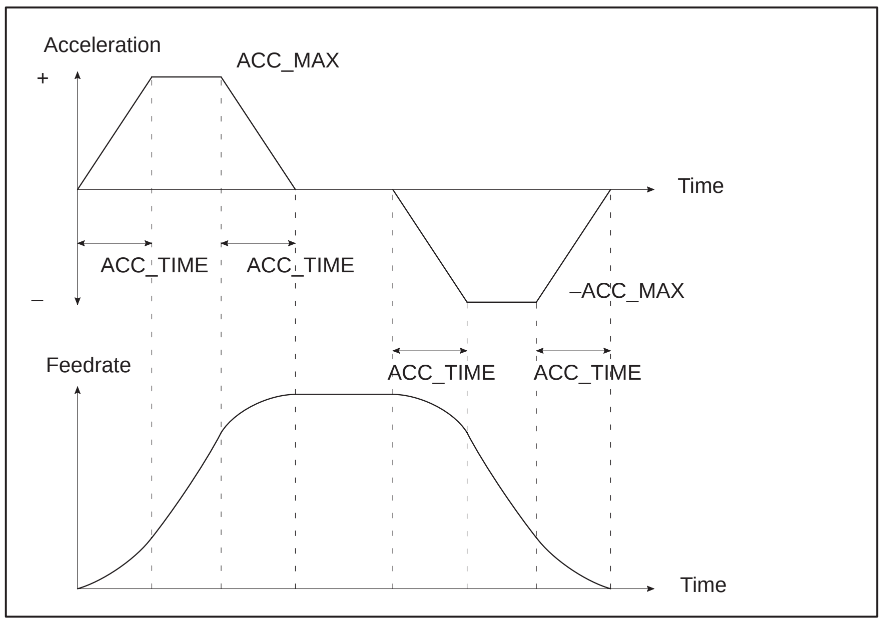
Description

Look–ahead bell–shaped acceleration/deceleration before interpolation controls acceleration as described below.

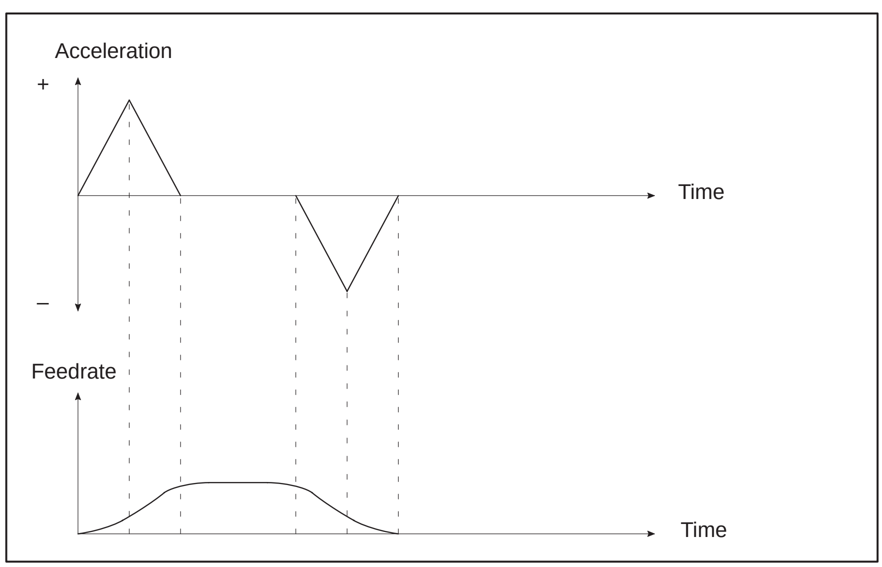
$$\text{Maximum acceleration ACC\_MAX} = \frac{\text{Setting of parameter No. 8400 [mm/min, inch/min]}}{\text{Setting of parameter No. 8401 [ms]}}$$

Time needed to reach maximum acceleration: ACC_TIME = Setting in parameter No. 8416 [ms]

- When maximum acceleration is reached

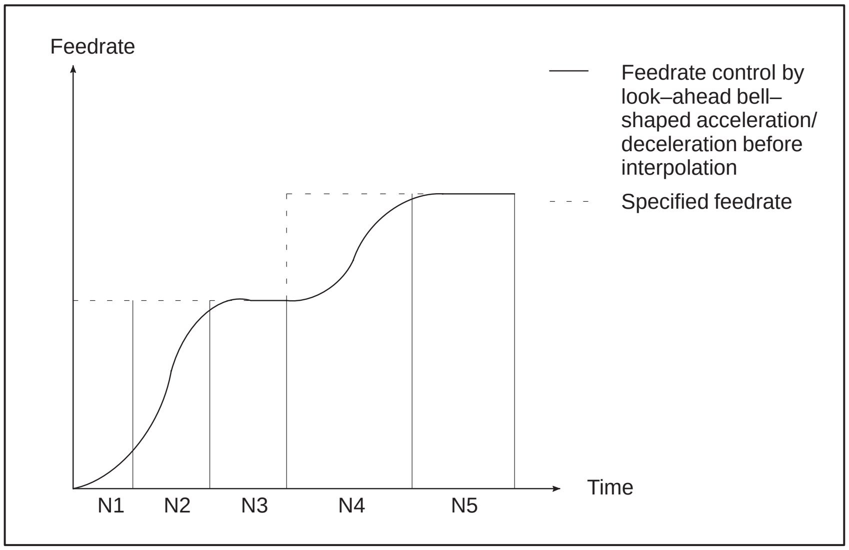


- When maximum acceleration is not reached



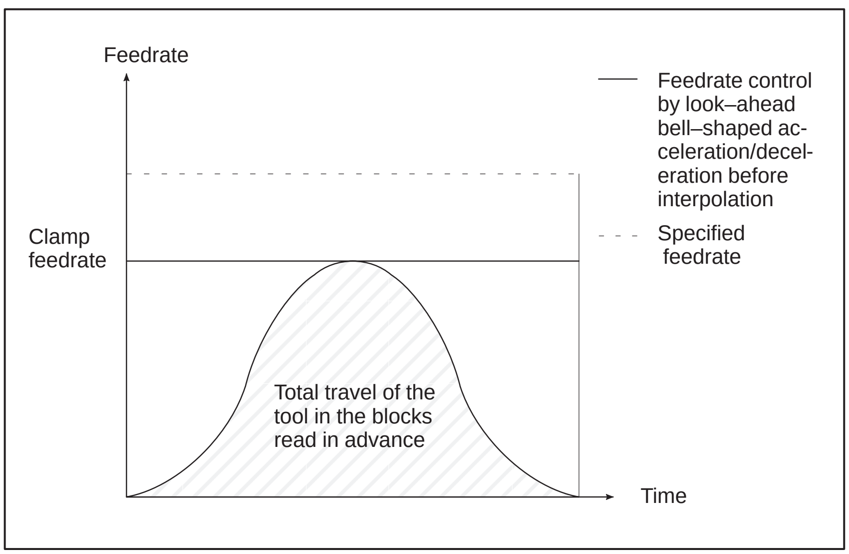
Acceleration

The tool is accelerated to a specified feedrate, starting at the beginning of a block.
The tool can be accelerated over multiple blocks.



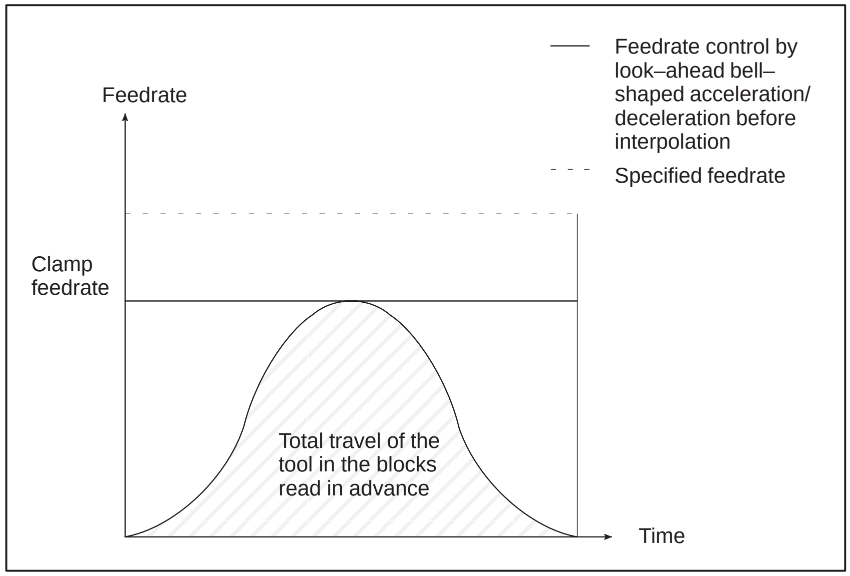
Feedrate clamping based on the total travel of the tool in look–ahead blocks

When the distance required to decelerate the tool from a specified feedrate is less than the total travel of the tool in the blocks read in advance, the feedrate is automatically clamped to a feedrate from which the tool can be decelerated to a feedrate of zero.

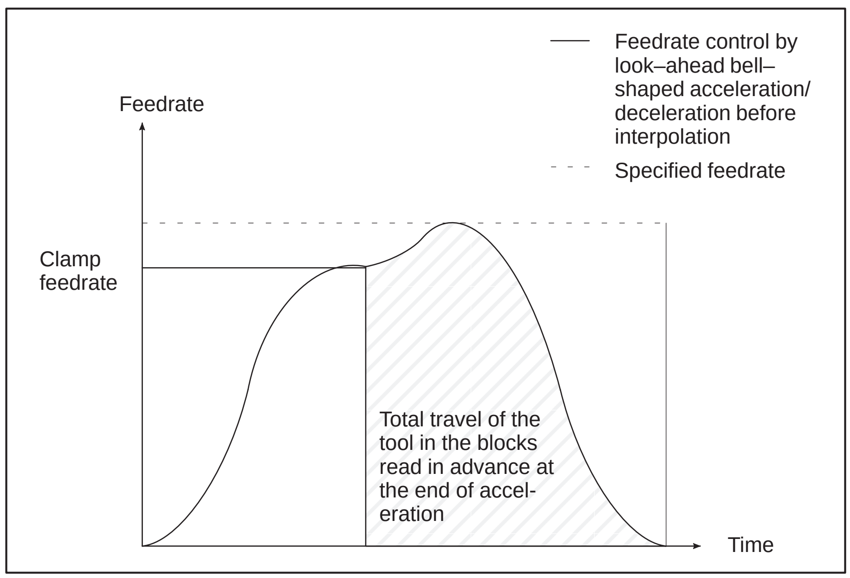


When several blocks, each specifying a short travel, are specified in succession, the following situation can occur:
The total travel of the tool in the blocks read in advance at the start of acceleration is less than the distance required to decelerate the tool from a specified feedrate, but the total travel of the tool in the blocks read in advance at the end of acceleration is greater than the distance required to decelerate the tool from a specified feedrate.
In such a case, the tool is accelerated once and clamped to the feedrate obtained based on the total travel of the tool in the blocks read in advance. Then, the tool is accelerated to a specified target feedrate.

- At the start of acceleration



- At the end of acceleration

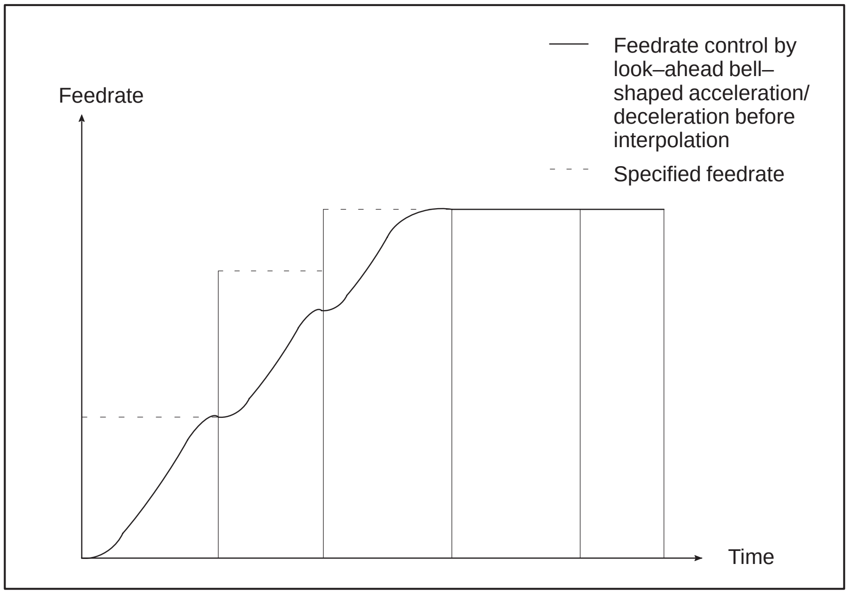


Feedrate command and feedrate

If an F command is changed by, for example, another F command, the corner deceleration function, or the automatic feedrate determination function, look–ahead bell–shaped acceleration/deceleration before interpolation treats the changed feedrate as a new target feedrate, and restarts acceleration/deceleration.

Whenever an F command is changed, bell–shaped acceleration/deceleration is performed.

Bell–shaped acceleration/deceleration is performed each time a different feedrate command is specified, for example, in a program containing successive blocks, each specifying a short travel.



When the feed hold function is used during acceleration

When the feed hold function is used during acceleration, control is performed as described below.

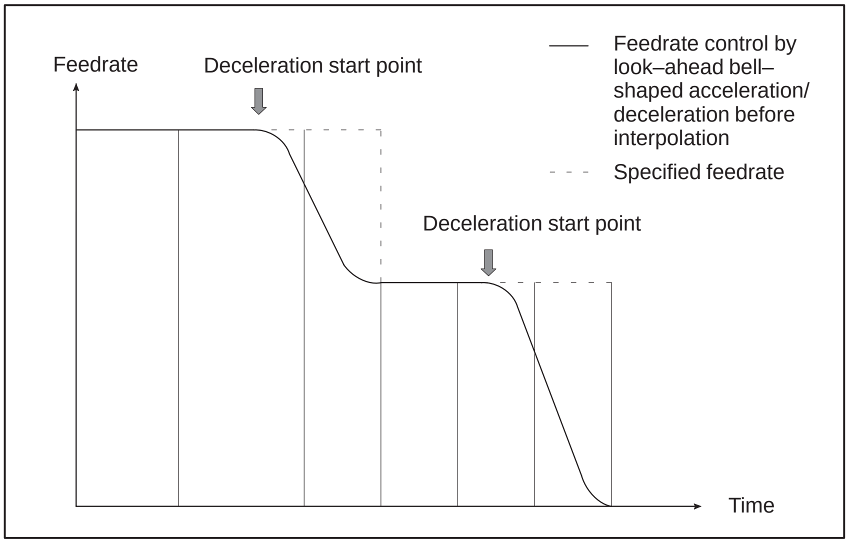
- While applying constant or increasing acceleration

  Starting at the point where the feed hold function is specified, the acceleration is gradually reduced to 0. Then, the feedrate for the tool is gradually reduced to 0. Thus, the feed hold function does not always immediately reduce the feedrate of the tool; it instead may sometimes increase the feedrate for a brief instant before reducing the feedrate.
- While applying decreasing acceleration

  First, the acceleration is gradually reduced to 0. Then, the feedrate is gradually reduced to 0.

Deceleration

The tool is decelerated to the feedrate specified for a block, starting at the previous block.
The tool can be decelerated over multiple blocks.

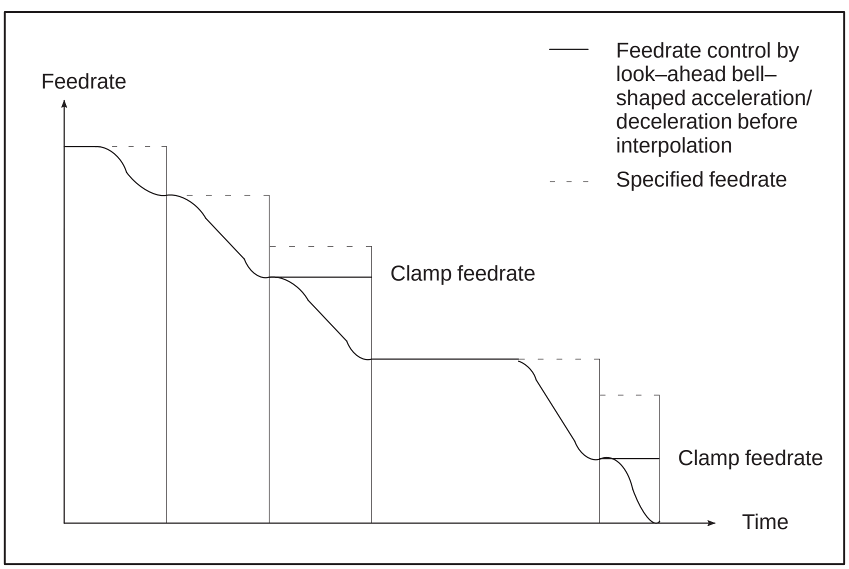


Feedrate command and deceleration

If an F command is changed by, for example, another F command, the corner deceleration function, or the automatic feedrate determination function, look–ahead bell–shaped acceleration/deceleration before interpolation treats the changed feedrate as a new target feedrate, and restarts acceleration/deceleration.

Whenever an F command is changed, bell–shaped acceleration/deceleration is performed.

When the distance required to decelerate the tool from a specified feedrate is longer than the total travel of the tool in the blocks read in advance, the feedrate is automatically clamped, as in the case of acceleration.



Deceleration based on tool travel

The deceleration of the tool is started when the total travel of the tool in the blocks read in advance is less than the distance required to decelerate the tool from the current feedrate.

When the total travel of the tool in the blocks read in advance increases at the end of deceleration, the tool is accelerated.

When blocks specifying a short travel are specified in succession, the tool may be decelerated, then accelerated, then decelerated, and so on, resulting in an unstable feedrate. In such a situation, specify a smaller feedrate.

Feed hold during deceleration

When the feed hold function is used during deceleration, control is performed as described below.

- While applying constant or increasing deceleration

  The point where the deceleration starts being reduced to 0 is shifted from the usually used point (i.e., that used when feed hold is not applied) to ensure that the feedrate for the tool is gradually reduced to 0.

- While applying decreasing deceleration

  The deceleration is gradually reduced to 0, after which the feedrate is reduced to 0.

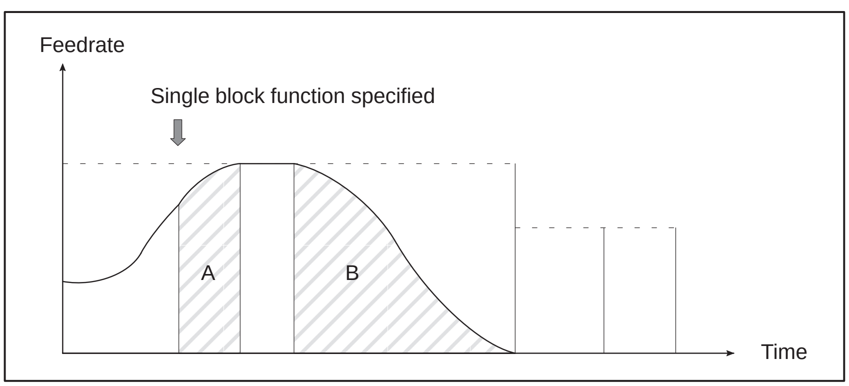
Single block function while look–ahead bell–shaped acceleration/deceleration before interpolation is used

When the single block function is specified while look–ahead bell–shaped acceleration/deceleration before interpolation is used, control is performed as described below.

While the tool is being accelerated or decelerated when the single block function is specified

(1) A + B ≦ Remaining travel for the tool in the block being executed when the single block function is specified

The tool is gradually decelerated so that the feedrate is 0 upon completion of the execution of the block that was being executed when the single block function was specified.



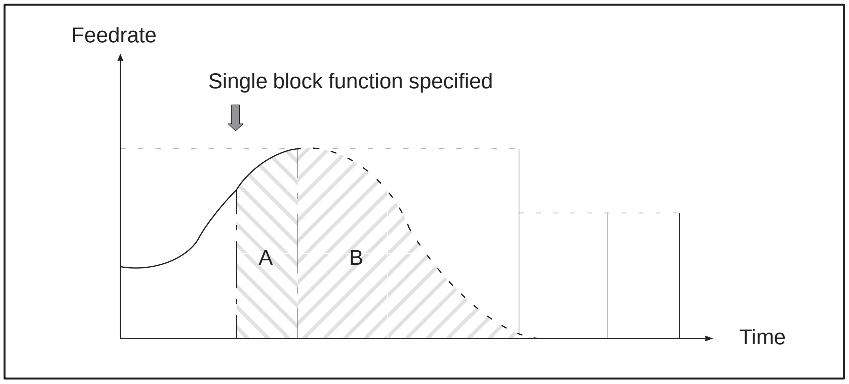
A: Distance traveled before the tool reaches the specified feedrate from the current acceleration/deceleration

B: Distance traveled before the feedrate falls to 0 from a feedrate to which no acceleration/deceleration is applied

(2) A + B > Remaining travel for the tool in the block being executed when the single block function is specified

The tool may be decelerated over multiple blocks until it stops.

How the tool is stopped is described later.



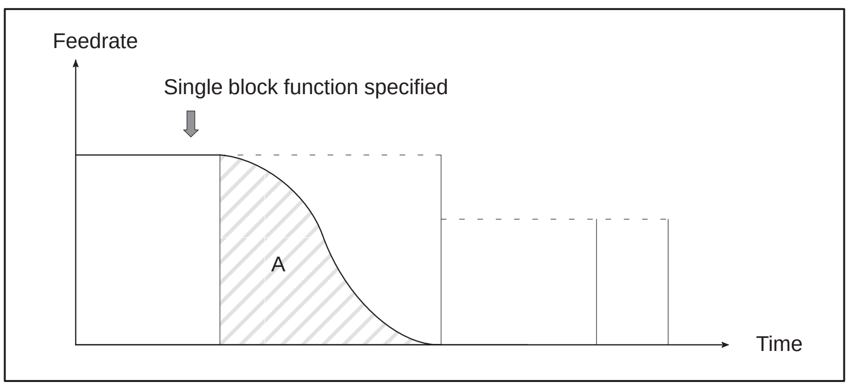
A: Distance traveled before the tool reaches the specified feedrate with the current acceleration/deceleration

B: Distance traveled until the feedrate falls to 0 from a feedrate to which no acceleration/deceleration is applied

While the tool is not being accelerated or decelerated when the single block function is specified

(1) A ≦ Remaining travel for the tool in the block being executed when the single block function is specified

The tool is gradually decelerated so that the feedrate is 0 upon completion of the execution of the block that was being executed when the single block function was specified.

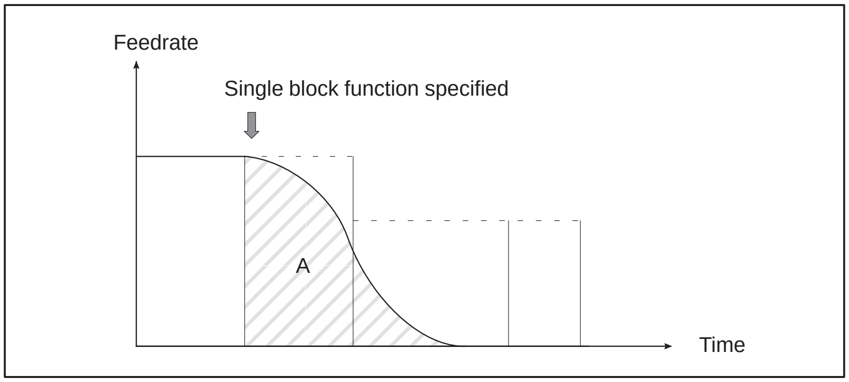


A: Distance traveled until the feedrate falls from the current feedrate value to 0

(2) A > Remaining travel of the tool in the block being executed when the single block function is specified

The tool may be decelerated over multiple blocks until it stops.

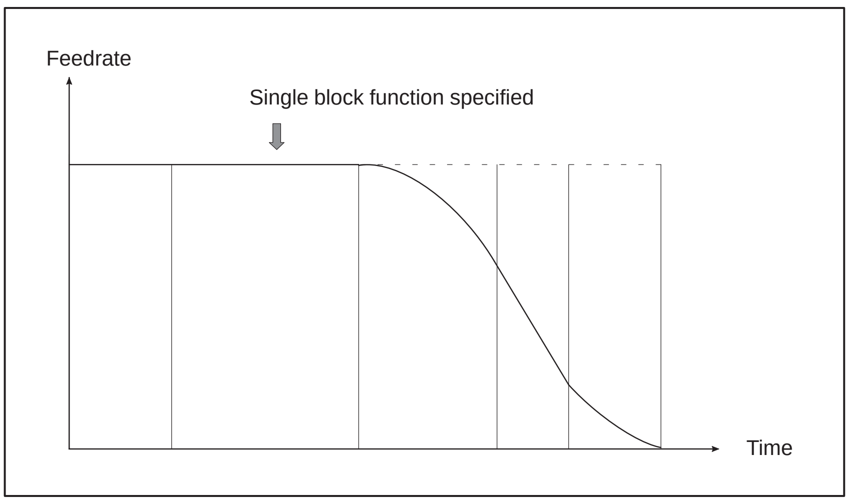
How the tool is stopped is described later.



A: Distance traveled until the feedrate falls from the current feedrate value to 0

How the tool is stopped when decelerated over multiple blocks

The tool is decelerated (or accelerated) over multiple blocks until the feedrate becomes 0.



Note 1 Depending on the stop point and remaining blocks, two or more acceleration/deceleration operations may be performed.

Note 2 When the single block function is specified, an acceleration/deceleration curve recalculation is required while the tool is moving along an axis. So, the tool is not always decelerated over the minimum number of blocks before stopping.

Dryrun/feedrateoverride

When a change in the specification of the dry run function or feedrate override function results in a change in the specified feedrate (feedrate change due to an external cause) while look–ahead bell–shaped acceleration/deceleration before interpolation is being used, control is performed as described below.

While the tool is being accelerated or decelerated when the specification of the dry run function or feedrate override function is changed

After the current acceleration/deceleration operation brings the tool to a specified feedrate and is terminated, the tool is accelerated or decelerated to the new target feedrate.

While the tool is not being accelerated or decelerated when the specification of the dry run function or feedrate override function is changed

The tool is accelerated or decelerated from the current feedrate to the specified feedrate.

Notes

Note 1 When the specification of the dry run function or feedrate override function is changed, the acceleration/deceleration curve must be recalculated while the tool is actually moving along an axis. For this reason, there will be a slight delay before a feedrate change is actually started after the specification of the dry run function or feedrate override function is changed.

Note 2 When the specification of the dry run function or feedrate override function is changed, the tool may be decelerated to below a specified feedrate and then accelerated, depending on the remaining amount of travel, current feedrate, and target feedrate.

7.1.14.2 Automatic feedrate control function

This function reads several tens of blocks ahead to exercise automatic feedrate control.

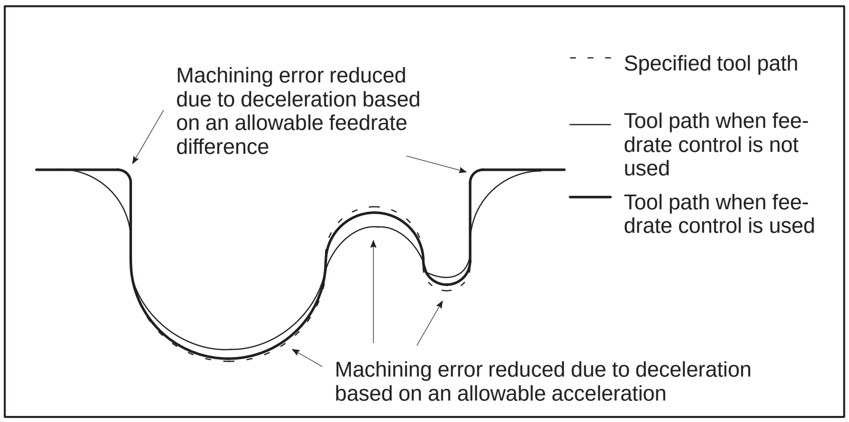
A feedrate is determined on the basis of the conditions listed below. If a specified feedrate exceeds a calculated feedrate, acceleration/deceleration before interpolation is used so that the calculated feedrate can be established.

(1) Feedrate change and specified allowable feedrate difference along each axis at a corner

(2) Anticipated acceleration and specified allowable acceleration along each axis

(3) Cutting load change anticipated from the direction of motion along the Z–axis

In automatic feedrate control mode, the feedrate is automatically reduced with acceleration/deceleration before interpolation to minimize the stress and strain applied to the machine.



To use this function, set bit 0 (USE) of parameter No. 8451 to 1, and set the following parameters:

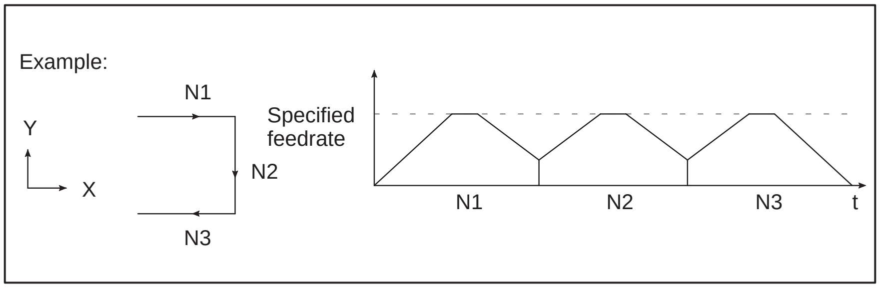
Parameter No. 8410: Allowable feedrate difference used for feedrate determination, based on a corner feedrate difference
Parameter No. 8475, bit 2 (BIP) = 1: Enables deceleration at a corner.
Parameter No. 8470: Parameter specifying an allowable acceleration for feedrate determination, based on acceleration
Parameter No. 8459, bit 1 (CTY) = 1, bit 0 (CDC) = 0
Parameter No. 8464: Initial feedrate for automatic feedrate control
Parameter No. 8465: Maximum allowable feedrate for automatic feedrate control

For details, see the description of each parameter.

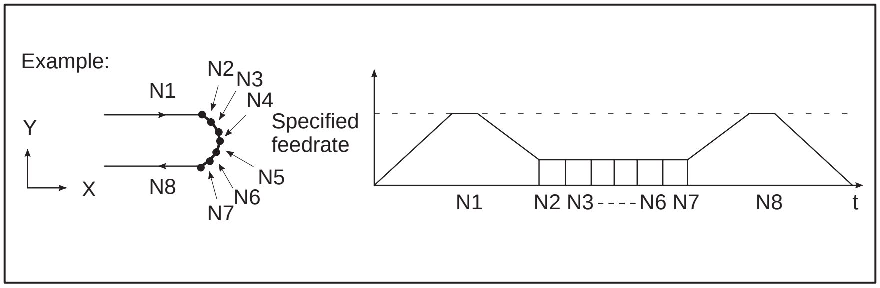
Feedrate control conditions

In automatic feedrate control mode, the feedrate for the tool is controlled as described below.

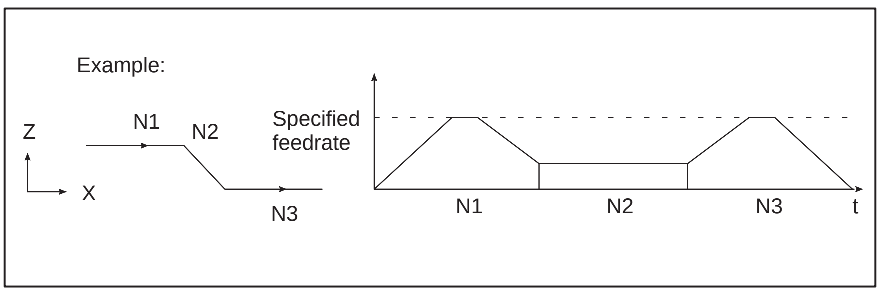
- ❑ The feedrate required at a corner is calculated from the specified feedrate difference at the corner along each axis, the tool being decelerated to the calculated feedrate at the corner.



❑ The feedrate required in a block is calculated from the specified acceleration along each axis at the start point and end point of the corner, the tool being decelerated so that the feedrate in the block does not exceed the calculated feedrate.



❑ The feedrate required in a block is calculated from the angle of downward movement along the Z–axis, the tool being decelerated so that the feedrate in the block does not exceed the calculated feedrate.



Example of feedrate determination based on a feedrate difference along each axis

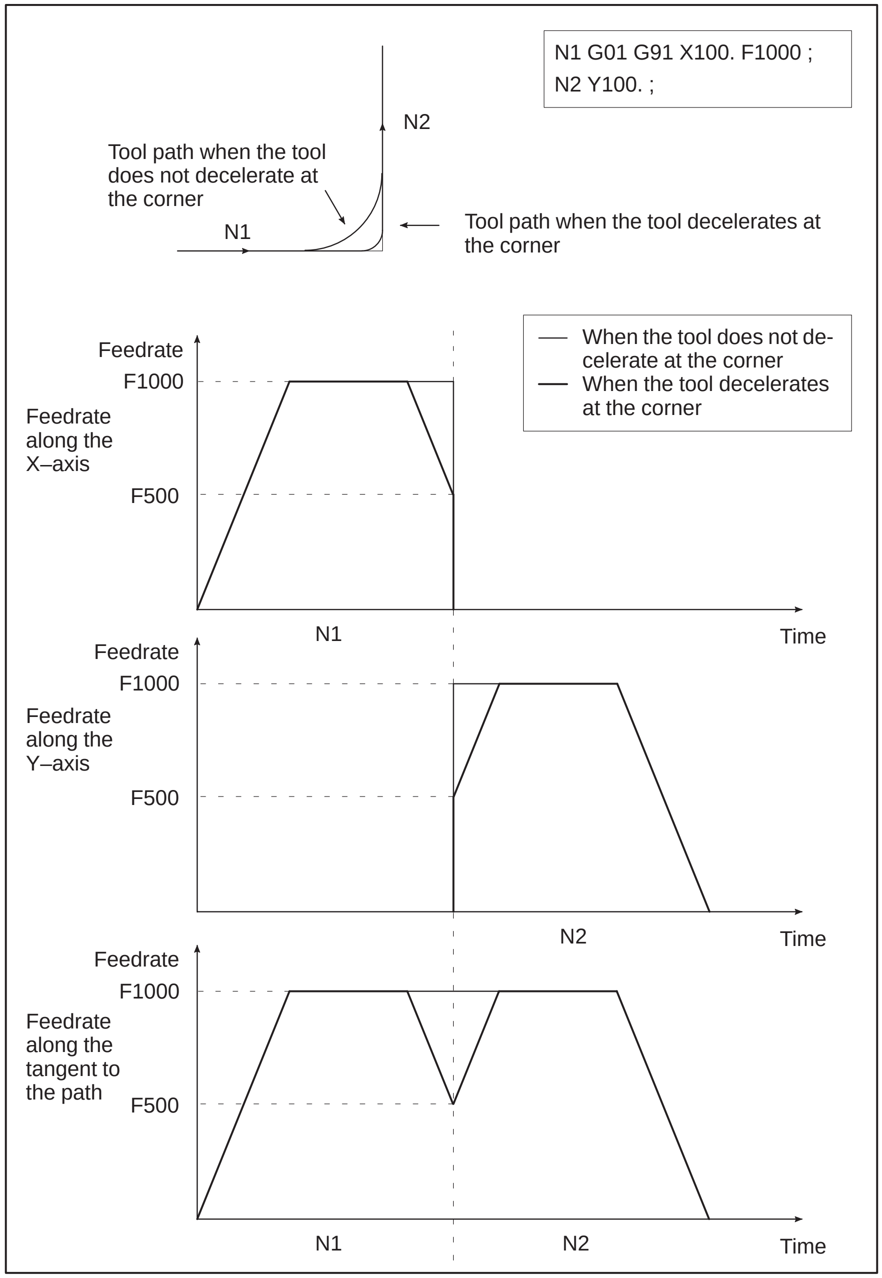
The feedrate required at a corner is calculated from the feedrate difference along each axis, as described below.

When the tool is to move at the specified feedrate F, a comparison is made between the feedrate change along each axis (Vc[X], Vc[Y], ...) and the value (Vmax) set in parameter No. 8410. If Vmax is exceeded by a feedrate change along any axis, the tool is decelerated at the corner to the required feedrate Fc:

$$Fc=F\times\frac{1}{Rmax}$$

where Rmax is the largest value of $R=\frac{Vc}{Vmax}$

Suppose that the specified feedrate for the tool is 1,000 mm/min, and that the direction of tool movement changes by 90 degrees (from along the X–axis to along the Y–axis). Suppose also that an allowable feedrate difference of 500 mm/min is set. Then, the tool will decelerate as shown below.



Example of feedrate determination based on acceleration along each axis

As shown below, when a curve is formed by very short successive line segments, there is no significant feedrate difference along each axis at each corner. Consequently, the tool need not be decelerated to compensate for feedrate differences. When taken as a whole, however, successive feedrate differences generate a large acceleration along each axis.

In this case, the tool must be decelerated to minimize the stress and strain imposed on the machine, as well as the machining error that may result from such excessive acceleration. The tool is decelerated to the feedrate at which the acceleration along each axis, found from the formula below, is equal to or less than a specified allowable acceleration.

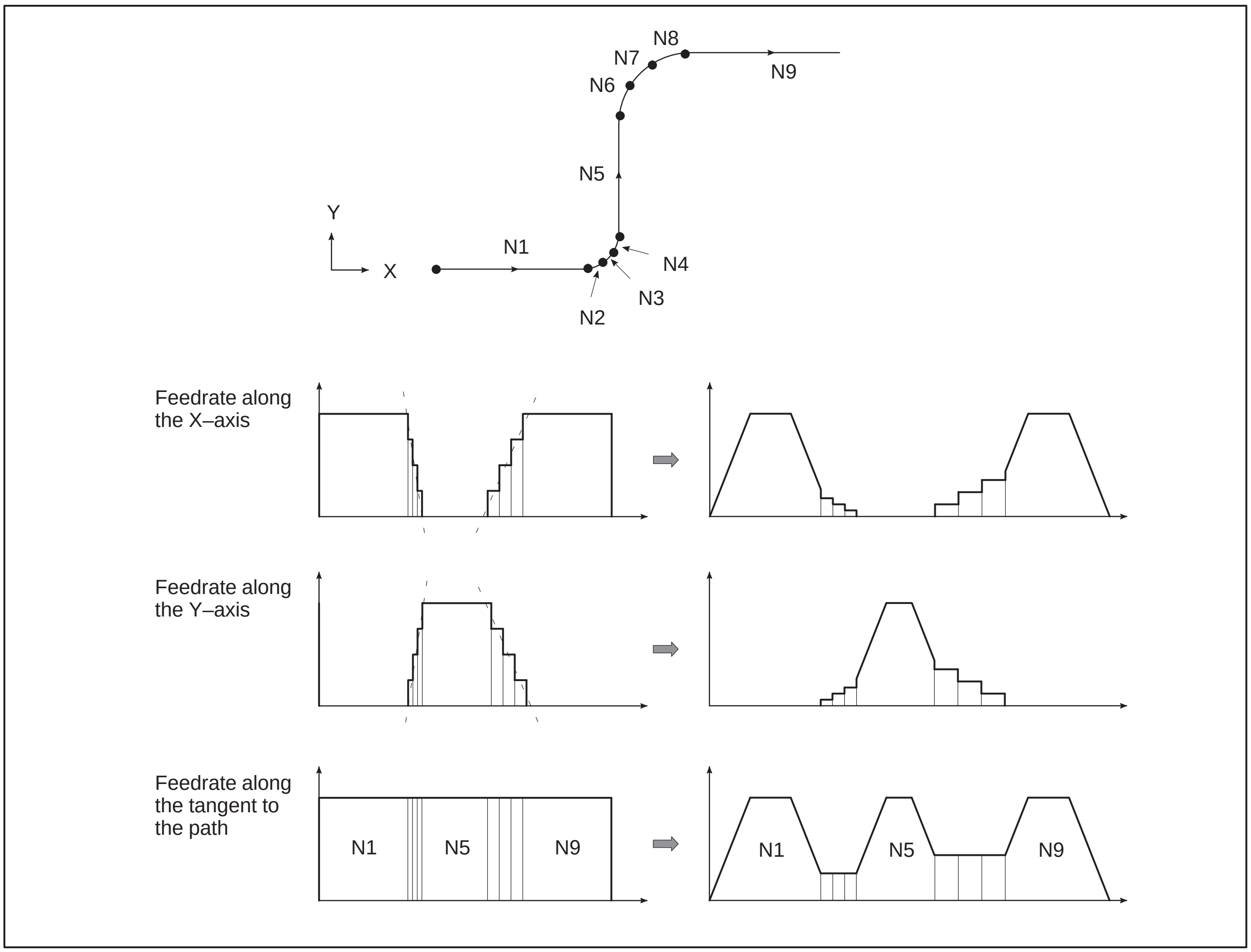
The allowable acceleration is determined from a maximum cutting feedrate (set in parameter No. 1432, No. 1430, or No. 1422) and the time needed to reach the maximum cutting feedrate (set in parameter No. 8470).

Acceleration along each axis =

$$\frac{\text{Feedrate difference along each axis at a corner}}{\max\left(\dfrac{\text{Travel in the previous block}}{F},\ \dfrac{\text{Travel in the next block}}{F}\right)}$$

The reduced feedrate required for each corner is calculated. The tool is decelerated to the decreased feedrate found at either the start point or the end point of each block, whichever is smaller.

Example: In the example shown below, the tool is accelerated too quickly from N2 to N4 and from N6 to N8 (as indicated by the dashed–line inclinations in the feedrate graphs) when automatic feedrate control is not used. So, the tool is decelerated.



Feedrate determination based on an allowable acceleration during circular interpolation

When a block specifies circular feed per minute and bit 3 (CIR) of parameter No. 8475 is set to 1, the feedrate of the tool is automatically determined so that the acceleration along each axis does not exceed an allowable acceleration.

The allowable acceleration is determined from the maximum cutting feedrate (set in parameter No. 1432, No. 1430, or No. 1422) and the time needed to reach the maximum cutting feedrate (set in parameter No. 8470).

During circular interpolation, the tool is controlled so that it always moves along the path at the specified feedrate. At this time, the total acceleration of the tool, consisting of the acceleration along each axis, is calculated as follows:

$$\text{Acceleration} = \frac{F^2}{R}$$

F: Feedrate
R: Arc radius

A feedrate is calculated, as shown below, so that the total acceleration does not exceed the smaller of the allowable accelerations along the two axes of circular interpolation. If a specified feedrate is greater than the calculated feedrate, the tool is decelerated to the calculated feedrate.

$$\frac{F^2}{R} = \min(\alpha_x, \alpha_y)$$

$$F = \sqrt{R \times \min(\alpha_x, \alpha_y)}$$

α_x, α_y : Allowable accelerations along X–axis and Y–axis

Example of feedrate determination based on cutting load

This function can be used when bit 4 (ZAG) of parameter No. 8451 is set to 1.

Cutting the workpiece with the end of the cutter (Fig. 7.1.14.2 (b)) incurs a greater resistance than when cutting the workpiece with the side of the cutter (Fig. 7.1.14.2 (a)). Therefore, for (Fig. 7.1.14.2 (b)), the tool must be decelerated. To calculate the required degree of feedrate deceleration, the automatic feedrate control function uses the angle of downward movement of the tool along the Z–axis.

When the tool is moving down along the Z–axis, the angle (θ) of downward movement formed by the XY plane and cutter path is as shown in the Fig. 7.1.14.2 (b). The angle of downward movement is divided into four areas, with an override value for each area specified in a parameter, as follows:
Area 2: Parameter No. 8456
Area 3: Parameter No. 8457
Area 4: Parameter No. 8458

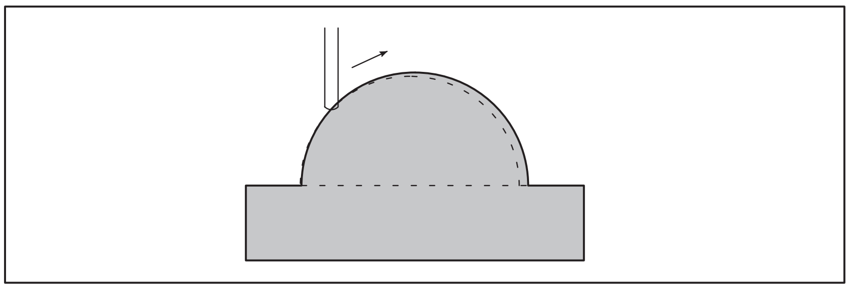


Fig. 7.1.14.2 (a) When the tool is moving up along the Z–axis

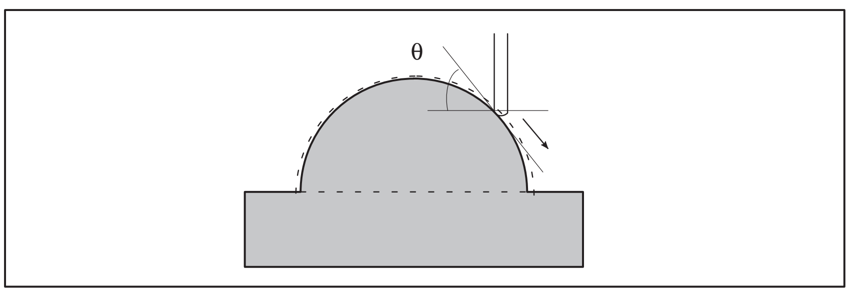
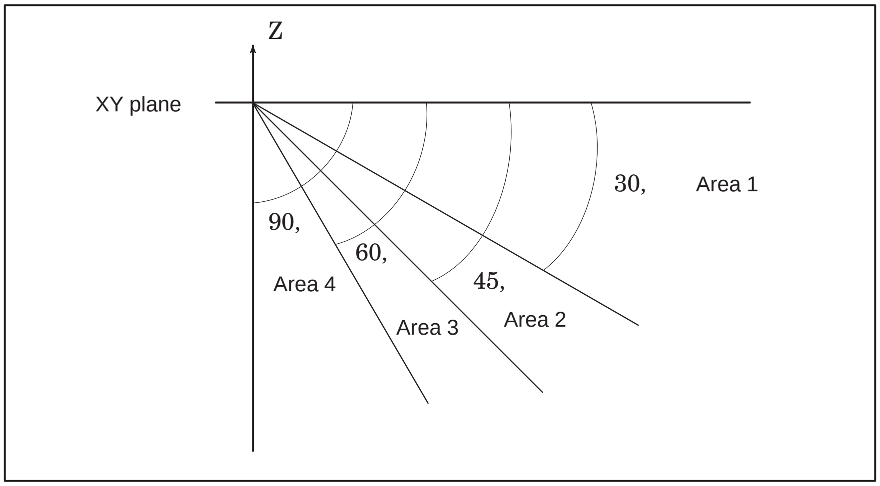


Fig. 7.1.14.2 (b) When the tool is moving down along the Z–axis

No override parameter is provided for area 1; the override value for area 1 is always 100%. A feedrate determined with a separate feedrate control function is multiplied by the override value specified for the area to which the angle θ of downward movement belongs.

Area 1: $0° \leq \theta < 30°$
Area 2: $30° \leq \theta < 45°$
Area 3: $45° \leq \theta < 60°$
Area 4: $60° \leq \theta \leq 90°$



Note 1 The feedrate determination function that is based on cutting load uses an NC command to determine the direction of movement along the Z–axis. This means that the direction of movement along the Z–axis cannot be found if the movement along the Z–axis is subject to manual intervention with manual absolute on/off function set to on, or if the mirror image function is used with the Z–axis. So, never use these functions when using feedrate determination based on cutting load.

Ignoring F code commands

In a block for which the automatic feedrate control function is enabled, the ignoring of all feed commands (F commands) can be specified by setting bit 7 (NOF) of parameter No. 8451. The feed commands are:

(1) Modal F command specified before a block for which the automatic feedrate control function is enabled

(2) Modal F command and F command specified in a block for which the automatic feedrate control function is enabled

Note, however, that specified F commands and modal F commands are stored in the CNC.

This means that in a block for which the automatic feedrate control function is disabled, a modal F command of (1) or (2) is used instead of a modal F command calculated by the automatic feedrate control function.

Other examples of feedrate determination conditions

If a calculated feedrate exceeds the maximum allowable feedrate for automatic feedrate control, specified in parameter No. 8465 or with an F command, the feedrate is clamped to the maximum allowable feedrate or F command, whichever is smaller.

7.1.14.3 Signal

HPCC mode signal MHPCC (F066, #6)

[Classification] Output signal

[Function] Indicates that the system is set to high–precision contour control mode (HPCC mode).

[Output condition] The signal is set to 1 if G05 P10000 (HPCC mode ON) is specified in a program. The signal is set to 0 if G05 P0 is specified in a program or if HPCC mode is canceled by a reset.

HPCC operation signal EXHPCC (F066, #7)

[Classification] Output signal

[Function] Indicates that the system is operating in high–precision contour control mode (HPCC operation is in progress).

[Output condition] The signal is set to 1 if G05 P10000 (HPCC mode ON) is specified in a program and if specifiable data of except G00, M, S, T, B is executed. The signal is set to 0 when:

(1) Automatic operation is halted.

(2) Automatic operation is stopped.

(3) Specifiable data of G00, M, S, T or B is executed.

(4) HPCC mode is canceled.

For the specifiable date, refer to 7.1.14.

Signal addresses

	#7	#6	#5	#4	#3	#2	#1	#0
F066	EXHPCC	MHPCC						

7.1.14.4 Parameter

Parameters of linear acceleration and deceleration before interpolation

8400	Parameter 1 for determining a linear acceleration/deceleration before interpolation

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	10 – 60000	1 – 6000
Inch machine	0.1 inch/min	10 – 60000	1 – 6000
Rotation axis	1 deg/min	10 – 60000	1 – 6000

This parameter determines a linear acceleration and deceleration before interpolation. Usually, set the maximum cutting speed (parameter No. 1422).

8401	Parameter 2 for determining a linear acceleration/deceleration before interpolation

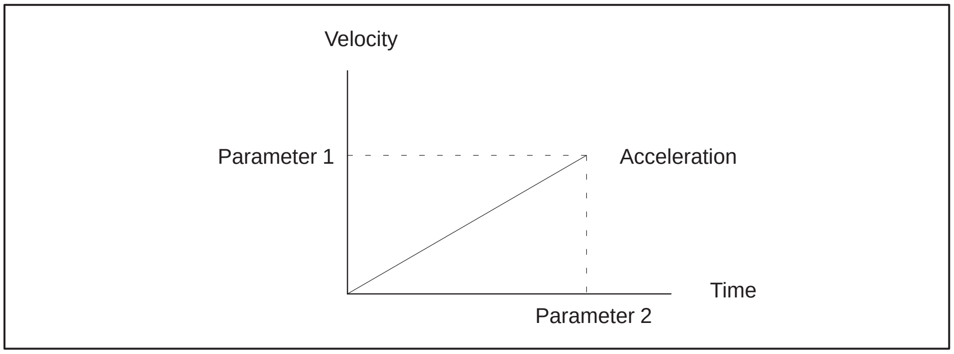
[Data type] Word

[Unit of data] 1 ms

[Valid data range] 0 to 4000

This parameter specifies the time required until the speed specified in parameter 1 is achieved.

Note 1 The function for linear acceleration/deceleration before interpolation is canceled when either parameter no. 8400 or 8401 is set to 0.



	#7	#6	#5	#4	#3	#2	#1	#0
8402	BADO		DST	BLK				NWBL

[Data type] Bit

BADO, NWBL Set the type of acceleration/deceleration before interpolation.

BADO	NWBL	Meaning
0	0	Acceleration/deceleration prior to interpolation is of linear type
1	1	Acceleration/deceleration prior to interpolation is of bell shape type

BLK Be sure to set to 0.

DST Be sure to set to 1.

Parameters of bell–shaped acceleration and deceleration prior to interpolation pre–read

	#7	#6	#5	#4	#3	#2	#1	#0
8402	BADO		DST	BLK				NWBL

[Data type] Bit

BADO, NWBL Set the type of acceleration/deceleration before interpolation.

BADO	NWBL	Meaning
0	0	Acceleration/deceleration prior to interpolation is of linear type
1	1	Acceleration/deceleration prior to interpolation is of bell shape type

BLK Be sure to set to 0.

DST Be sure to set to 1.

8400	Parameter 1 for determining a acceleration/deceleration before interpolation

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Millimeter machine	1 mm/min	10 – 60000	1 – 6000
Inch machine	0.1 inch/min	10 – 60000	1 – 6000
Rotation axis	1 deg/min	10 – 60000	1 – 6000

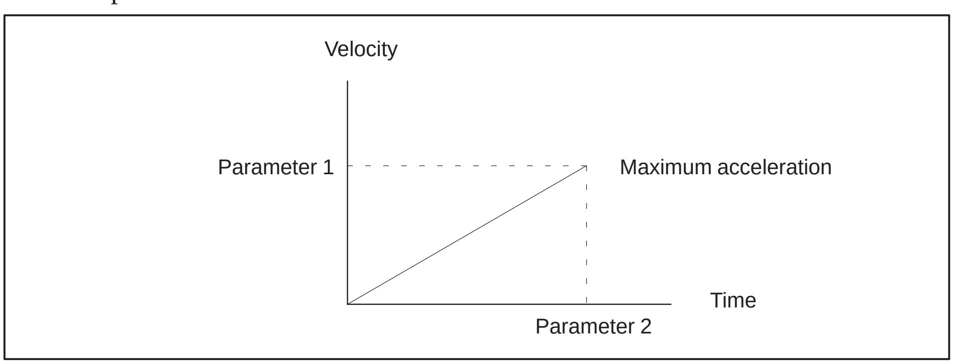
8401	Parameter 2 for determining a acceleration/deceleration before interpolation

[Data type] Word

[Unit of data] 1 ms

[Valid data range] 0 to 4000

This parameter specifies the time required until the speed specified in parameter 1 is achieved.



[Data type] Two–word

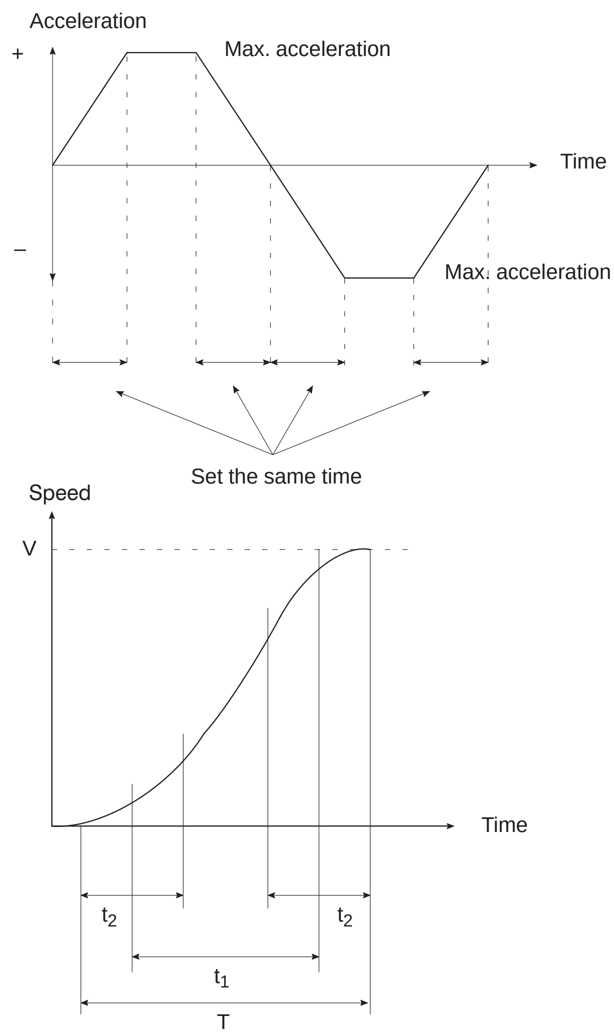
8416	The time required to the maximum acceleration in look–ahead bell–shaped acceleration/deceleration before interpolation

[Unit of data] mec

[Valid data range] 0 to 99999999

This parameter sets the time required to reach the maximum acceleration in look–ahead bell–shaped acceleration/deceleration before interpolation.

Also, this parameter is used for deceleration time taken from the maximum acceleration to zero.



t1: Time constant assumed in linear acceleration/deceleration (Parameter No. 8401)
t2: Time for corner rounding (parameter No. 8416)
V: Speed to set acceleration time t1 (Parameter No. 8400)
Usually set the max. cutting speed.

Total time	=T
Time of linear part	=T–2 * t_2
Time of curved part	=t_2

When target speed is different, total time also changes (constant acceleration).

Parameters of automatic velocity setting

8410	Allowable velocity difference in velocity determination considering the velocity difference at corners

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Millimeter machine	1 mm/min	10 – 60000	1 – 6000
Inch machine	0.1 inch/min	10 – 60000	1 – 6000
Rotation axis	1 deg/min	10 – 60000	1 – 6000

If zero is specified for all axes, the machine does not decelerate at corners.

When the function for determining the velocity considering the velocity difference at corners is used, the system calculates the feedrate whereby a change in the velocity element of each axis does not exceed this parameter value at the interface between blocks. Then the machine decelerates using acceleration/ deceleration before interpolation.

	#7	#6	#5	#4	#3	#2	#1	#0
8451	NOF			ZAG				USE

[Data type] Bit

USE Automatic velocity control is:
0 : Not applied.
1 : Applied.

ZAG The velocity is:
0: Not determined according to the angle at which the machine descends along the Z–axis.
1: Determined according to the angle at which the machine descends along the Z–axis.

NOF In a block where automatic velocity control is validated, the F command is:
0 : Validated.
1 : Ignored.

(Maximum speed of automatic feedrate control set by parameter No. 8465 is used for command speed in spite of F command)

8452	Range of velocity fluctuation to be ignored

[Data type] Byte

[Unit of data] %

[Valid data range] 0 to 100 (Standard setting: 10)

8456	Area–2 override

[Data type] Word

[Unit of data] %

[Valid data range] 1 to 100 (Standard setting: 80)

This parameter specifies an override in area 2 of velocity calculation considering the cutting load.

8457	Area–3 override

[Data type] Word

[Unit of data] %

[Valid data range] 1 to 100 (Standard setting: 70)

This parameter specifies an override in area 3 of velocity calculation considering the cutting load.

8458	Area–4 override

[Data type] Word

[Unit of data] %

[Valid data range] 1 to 100 (Standard setting: 60)

This parameter specifies an override in area 4 of velocity calculation considering the cutting load.

	#7	#6	#5	#4	#3	#2	#1	#0
8459							CTY	CDC

[Data type] Bit

CDC Be sure to set this value to 0.

CTY Be sure to set this value to 1.

8464	Initial feedrate for automatic feedrate control

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Millimeter machine	1 mm/min	0 – 600000	0 – 6000
Inch machine	0.1 inch/min	0 – 600000	0 – 6000
Rotation axis	1 deg/min	0 – 600000	0 – 6000

This parameter sets the initial feedrate for automatic feedrate control.

In automatic feedrate control, the initial feedrate set with this parameter is used at the beginning if no F command is specified in the program. Usually, set the maximum cutting feedrate (specified in parameter No. 1422).

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C

8465	maximum allowable feedrate for automatic feedrate control

Millimeter machine	1 mm/min	10 – 240000	1 – 100000
Inch machine	0.1 inch/min	10 – 96000	1 – 48000
Rotation axis	1 deg/min	10 – 240000	1 – 100000

This parameter sets the maximum allowable feedrate for automatic feedrate control. Usually, set the maximum allowable cutting feedrate (set in parameter No. 1422).

8470	Parameter for determining allowable acceleration in feedrate calculation considering acceleration

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 32767

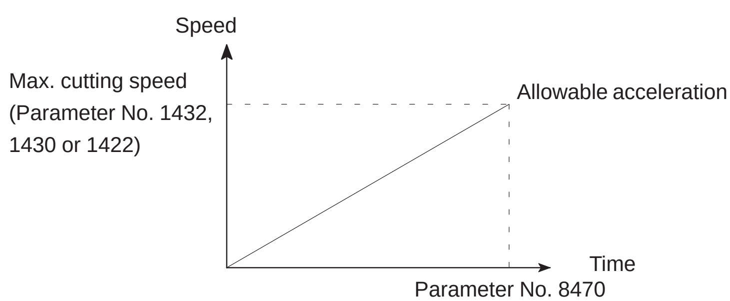
When the function for calculating the feedrate considering the acceleration is used under automatic feedrate control, this parameter is used to determine the allowable acceleration. The time required until the maximum cutting feedrate (parameter No. 1422) is reached must be specified here.

Allowable acceleration is determined from the maximum cutting feedrate and the value set in this parameter. Where, the maximum cutting feedrate is any of value set in parameter No. 1432, 1430 or 1422. Which parameter No. is used depends on the following conditions:

· When a value other than 0 is set to No. 1432, the value set to No. 1432 is used.

· When 0 is set to No. 1432 and a value other than 0 is set to No. 1430, the value set to No. 1430 is used.

· When 0 is set to No. 1432 and 1430, the value set to No. 1422 is used.



	#7	#6	#5	#4	#3	#2	#1	#0
8475					CIR	BIP		

[Data type] Bit

CIR The function of automatic feedrate control considering acceleration and deceleration during circular interpolation is:
0 : Not used.
1 : Used.

When 1 is set, parameter NO. 8470 for determining the allowable acceleration must be specified.

BIP The function of deceleration at corners is:
0 : Not used.
1 : Used. (Always set 1.)

Parameters of axis control

7510	Maximum number of axes in High Precision Contour Control

[Data type] Byte

[Valid data range] 1, 2, 3, … to the maximum number of control axes

This parameter specifies the maximum number of axes to controlled by High Precision Contour Control.
Example) Axis configuration is X, Y, Z, A, B, and C from the 1st axis in this order and to make HPCC valid to the 4th axis (A), set this parameter to 4. In this case, HPCC is also effective for the X, Y, Z axes.

X, Y, Z, A axes   Axes on which HPCC is valid
B, C axes . Axes on which HPCC is not valid.

[Data type] Bit

Set the interpolation frequency during the high precision contour control mode (HPCC mode).
Be sure to set the following values:

RI2	RI1	RI0
0	1	0

Parameters of acceleration/deceleration after interpolation

	#7	#6	#5	#4	#3	#2	#1	#0
1602		LS2						

[Data type] Bit

LS2 Acceleration/deceleration after interpolation for cutting feed in the look–ahead control mode is:
0 : Exponential acceleration/deceleration
1 : Linear acceleration/deceleration (The function for linear acceleration/ deceleration after interpolation for cutting feed is required.)

1768	Time constant for linear acceleration/deceleration during cutting feed in look–ahead control mode

[Data type] Word axis

[Unit of data] ms

[Valid data range] 8 to 512

Note 1 The function for linear acceleration/deceleration after interpolation is required.

Parameters of cutter compensation C

	#7	#6	#5	#4	#3	#2	#1	#0
5000								SBK

[Data type] Bit

SBK An internally created block for cutter compensation C:
0 : Does not cause a single block stop.
1 : Cause a single block stop.

	#7	#6	#5	#4	#3	#2	#1	#0
5003				BCK	ICK			

[Data type] Bit

ICK In HPCC mode, when cutter compensation C interference check is:
0 : Done
1 : Not done

BCK In HPCC mode, when cutter compensation C interference check determines that the programmed move direction differs from the offset move direction by between 90 and 270 degrees:
0 : An alarm is issued.
1 : No alarm is issued.

The other parameters

	#7	#6	#5	#4	#3	#2	#1	#0
8403	SG0				PLC2	PLC1	MSU	

[Data type] Bit

MSU When G00, or an M, S, T, or B code is specified in HPCC mode:
0 : An alarm is issued.
1 : The CNC executes the command.

PLC1 In HPCC mode, a stroke check before movement for stored stroke limit 1 is:
0 : Not performed.
1 : Performed.

PLC2 In HPCC mode, a stroke check before movement for the second stored stroke limit is:
0 : Not performed.
1 : Performed.

SG0 When G00 is specified in HPCC mode:

0 : The setting of bit 1 (MSU) of parameter No. 8403 is followed.
1 : The tool is moved along the axis at the feedrate set with parameter No. 8481, replacing the G00 command with the G01 command, regardless of the setting made for bit 1 (MSU) of parameter No. 8403. Refer to note 1 in description of parameter No. 8481.

8481	Rapid–traverse rate in HPCC mode

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Millimeter machine	1 mm/min	0 – 600000	0 – 60000
Inch machine	0.1 inch/min	0 – 600000	0 – 60000
Rotation axis	1 deg/min	0 – 600000	0 – 60000

When bit 7 (SG0) of parameter No. 8403 is set to 1, this parameter sets the rapid traverse rate in the HPCC mode.

Note 1 The G00 command is replaced with the G01 command before execution. So, even if a feedrate is specified for two axes, the rapid traverse rate set with this parameter is always used.

Example: If the following command is specified when a rapid traverse rate of 1000 mm/min is set F1000, rather than F1414, is used:

G00 X100.Y100.;

For details, refer to Notes on positioning (G00).

7.1.14.5 Alarm and Message

Number	Message	Description
PS5012	G05 P10000 ILLEGAL START UP	G05P10000 was specified in a mode from which HPCC mode cannot be entered.
PS5000	ILLEGAL COMMAND CODE(HPCC)	An invalid code was specified in HPCC mode.
PS5003	ILLEGAL PARAMETER (RISC)	Parameter setting is erroneous.
PS5004	RISC NOT READY	RISC processor board is not ready state.
PS5006	TOO MANY WORD IN ONE BLOCK	The number of words in a block exceeds allowable range (HPCC mode).
PS5013	HPCC : CRC OFS REMAIN AT CANCEL	G05P0 was specified in G41/G42 mode or a state in which an offset value remains.

7.1.14.6 Note

- **Acceleration/deceleration before interpolation in look–ahead blocks**

Note 1 If there is a series of very short blocks, for each of which the rate of acceleration/deceleration before interpolation is low, the actual feedrate may not reach the programmed feedrate.

- **Automatic feedrate control**

Note 1 If the upper limit for automatic feedrate control is set to 0 in parameter No. 8465, no feedrate exceeding 0 is permitted, such that the issue of an F command causes PS alarm 011 (FEED ZERO). To prevent this, specify a value other than zero in the parameter.

Note 2 If the override is changed while the automatic feedrate control function is enabled, the calculated clamp feedrate is overridden.

- **Notes on operation**

Note 1 A single–block stop cannot be made at the end of the G05P10000 block.

Note 2 External deceleration, the F1–digit command, and automatic corner override are disabled.

Note 3 In HPCC mode, the operation mode cannot be switched to MDI mode. Also, MDI operation is not permitted.

Note 4 In HPCC mode, axial interlocking (each axis, each direction) is inhibited (signals G130, G132, G134).

Note 5 In HPCC mode, the external mirror image (DI) signal, a mirror image determined by setting data, or axial machine lock must not be changed. Pocket calculator type decimal point input is disabled in HPCC mode (when bit 0 of parameter No. 3401 is set to 1).

Note 6 A program including G50 P10000; cannot be resumed.

Notes on positioning (G00)

When executing a G00 command when bit 7 of parameter No. 8403 (SG0) is set to 1, note the following:

Note 1 The G00 command is replaced with the G01 command upon being executed. Even if two axes are specified, movement is performed at the feedrate specified in parameter No. 8481.

Example)
If the following command is specified when parameter No. 8481 is set to 1000 mm/min, F1000 is executed instead of F1414:
G00 X100. Y100.;

Note 2 Because the G00 command is replaced with the G01 command upon being executed, rapid traverse override is disabled, the cutting feedrate override being enabled instead.

Note 3 Because the G00 command is replaced with the G01 command upon being executed, acceleration/deceleration after interpolation is done using the time constant of acceleration/deceleration after interpolation for cutting feed.

Note 4 Because the G00 command is replaced with the G01 command upon being executed, acceleration/deceleration before interpolation in RISC HPCC mode is enabled. (Both linear acceleration/deceleration and bell–shaped acceleration/deceleration are supported.)

Note 5 In–position check is not executed.

Note 6 Linear interpolation positioning is performed.

Reference Item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.20.4	High–precision contour control

7.2 ACCELERATION/ DECELERATION CONTROL

7.2.1 Automatic Acceleration/ Deceleration

· Rapid traverse linear acceleration/deceleration
· Cutting feed exponential acceleration/deceleration
· Jog feed exponential acceleration/deceleration

General

- **Automatic acceleration/ deceleration**

To prevent a mechanical shock, acceleration/deceleration is automatically applied when the tool starts and ends its movement (Fig. 7.2.1 (a)).

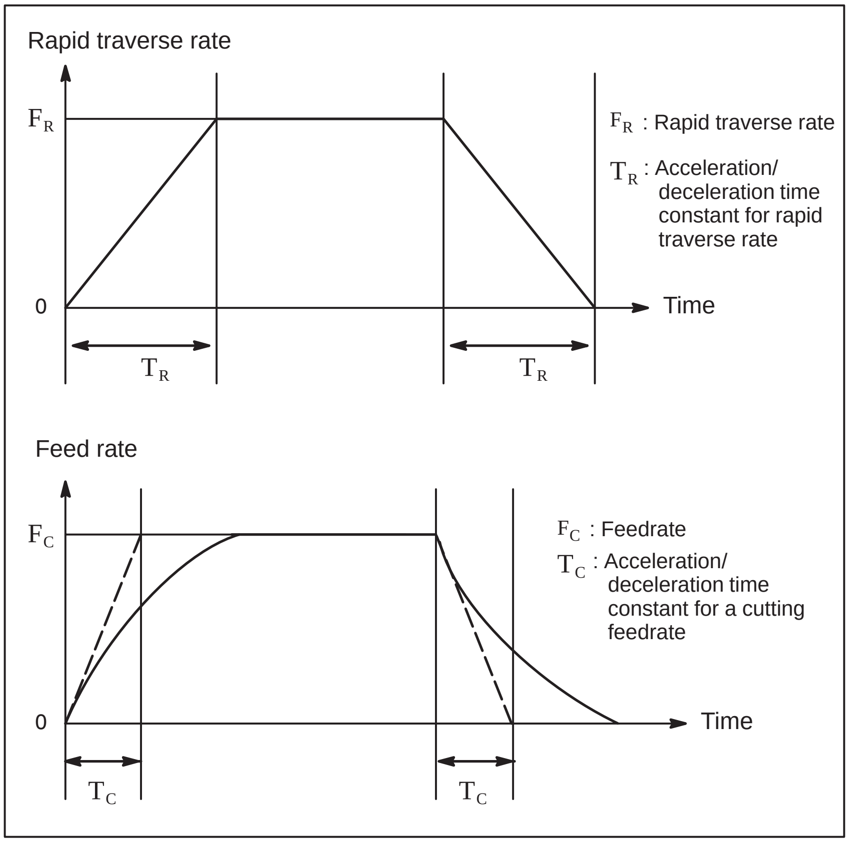


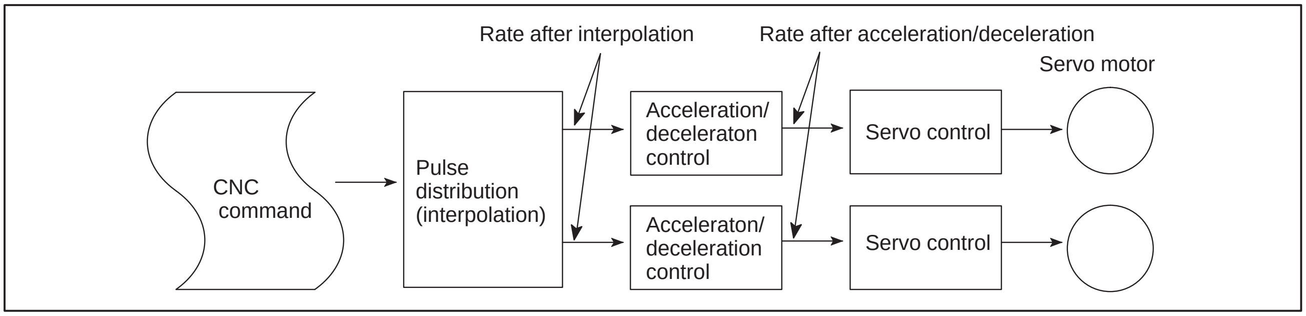
Fig. 7.2.1 (a) Automatic acceleration/deceleration (example)

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 change in speed is also smoothly done.
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 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 1624)



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx			CTBx	CTLx

[Data type] Bit axis

CTLx Acceleration/deceleration in cutting feed including feed in dry run
0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation is applied.

Note 1 If the optional function of linear acceleration/deceleration after interpolation in cutting feed is not provided, exponential acceleration/deceleration is used irrespective of this setting.

To use bell–shaped acceleration/deceleration after interpolation, set this parameter to 0 and select the acceleration/deceleration using CTBx, bit 1 of parameter No. 1610.

Parameter		Acceleration/deceleration
CTBx	CTLx	
0	0	Exponential acceleration/deceleration
0	1	Linear acceleration/deceleration after interpolation
1	0	Bell–shaped acceleration/deceleration after interpolation

CTBx Acceleration/deceleration in cutting feed including feed in dry run
0 : Exponential acceleration/deceleration or linear acceleration/deceleration after interpolation is applied (depending on the setting in CTLx, bit 0 of parameter No. 1610).
1 : Bell–shaped acceleration/deceleration after interpolation is applied.

Note 1 This parameter is effective only when the function of bell–shaped acceleration/deceleration after interpolation in cutting feed is provided. If the function is not provided, the setting in CTLx, bit 0 of parameter No. 1610, determines the type of acceleration/deceleration irrespective of the setting in this parameter.

JGLx Acceleration/deceleration in manual continuous feed (jog feed)

0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation is applied (depending on which is used for cutting feed).

1620	Time constant used for linear acceleration/deceleration or bell–shaped acceleration/deceleration in rapid traverse for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Specify a time constant used for acceleration/deceleration in rapid traverse. When the optional function of bell–shaped acceleration/deceleration in rapid traverse is provided, bell–shaped acceleration/deceleration is applied in rapid traverse. If the function is not provided, linear acceleration/deceleration is applied.

(1) When the function is provided, set this parameter to time constant T1 used in bell–shaped acceleration/deceleration in rapid traverse, and set parameter No. 1621 to time constant T2.

(2) When the function is not provided, specify a time constant used in linear acceleration/deceleration.

Note 1 When parameter No. 1621 (time constant T2 used for bell–shaped acceleration/deceleration in rapid traverse) is set to 0, linear acceleration/deceleration is applied in rapid traverse even if the function is provided. In this case, this parameter stands for a time constant used in linear acceleration/deceleration in rapid traverse.

1622	Time constant of exponential acceleration/deceleration or bell–shaped acceleration/deceleration after interpolation, in cutting feed for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in cutting feed or bell–shaped acceleration/deceleration after interpolation 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 in cutting feed for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in cutting feed for each axis. Except for special applications, this parameter must be set to 0 for all axes. If a value other than 0 is specified, proper straight lines and arcs cannot be obtained.

1624	Time constant of exponential acceleration/deceleration, bell–shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation, in jog feed for each axis.

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration, bell–shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation in jog feed for each axis.
The type of acceleration/deceleration is determined depending on the setting in parameter No. 1610.

1625	FL rate of exponential acceleration/deceleration in jog feed for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in jog feed for each axis.

1626	Time constant of exponential acceleration/deceleration in the thread cutting cycle for each axis

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in the thread cutting cycle (G76, G78 (G92 in G code system A)) for each axis.

1627	FL rate of exponential acceleration/deceleration in the thread cutting cycle for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in the thread cutting cycle (G76, G78 (G92 in G code system A)) for each axis.

1722	Rapid traverse feedrate reduction ratio for overlapping rapid traverse blocks

[Data type] Byte axis

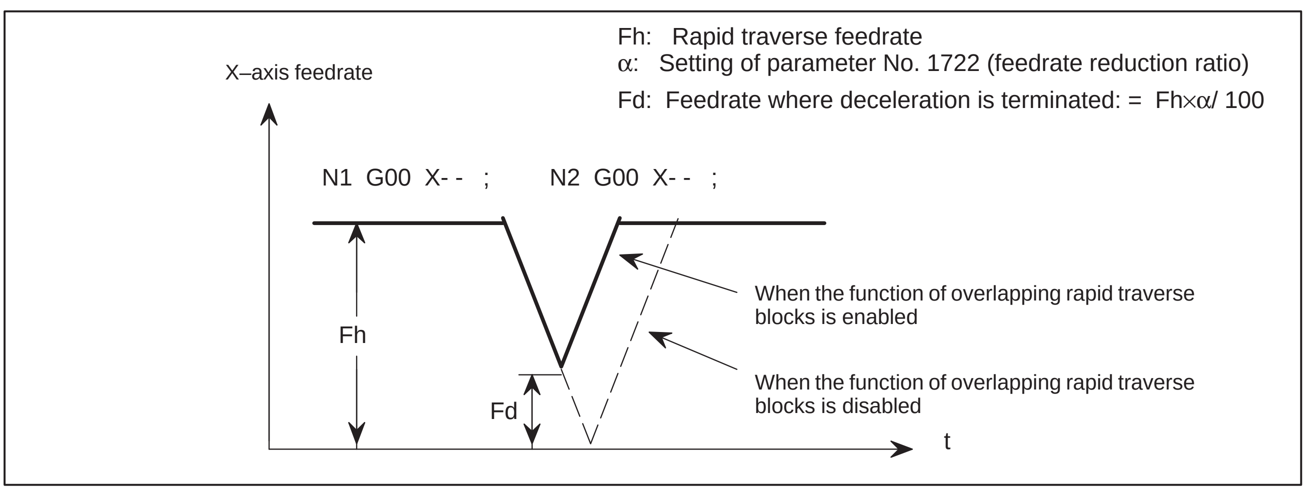
[Unit of data] %

[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 1 The parameter No. 1722 is effective when parameter No. 1601 #4 (RTO) is set to 1.

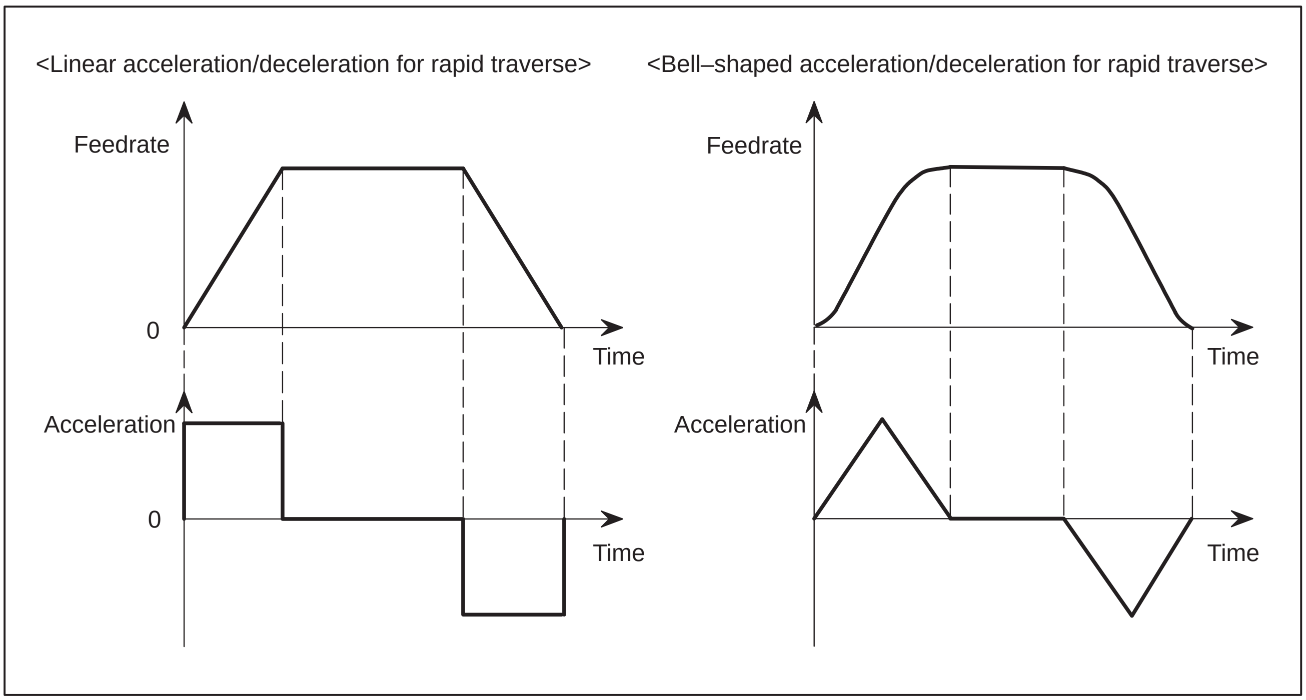
Example



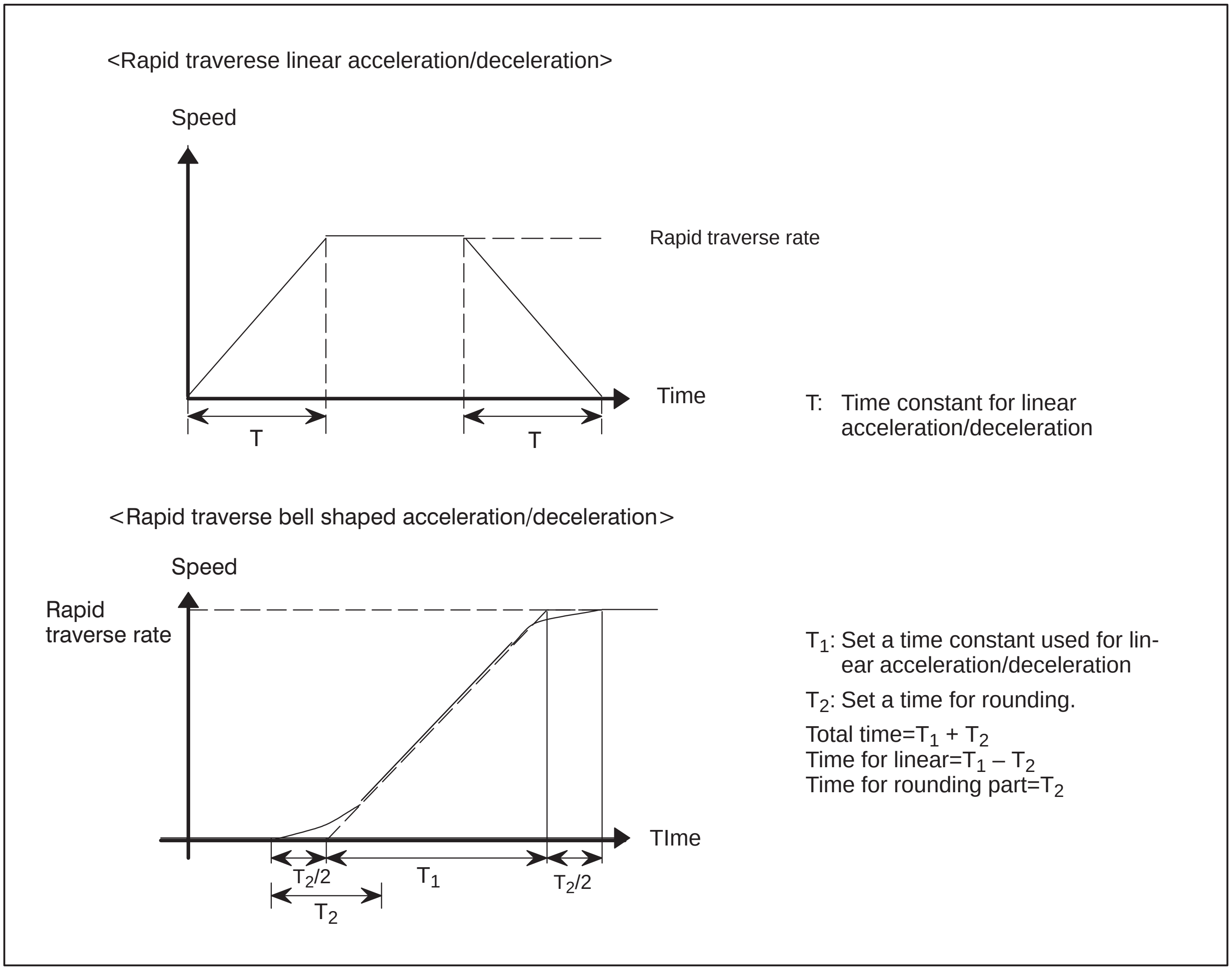
7.2.2 Bell–shaped Acceleration/ Deceleration for Rapid Traverse

General

Bell–shaped acceleration/deceleration for rapid traverse 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.



This function is enabled when the time constants for bell–shaped acceleration/deceleration for rapid traverse T1 and T2 are specified in parameter Nos. 1620 and 1621, respectively.



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 ms or 32 ms.

Parameter

1620	Time constant used in linear acceleration/deceleration or bell–shaped acceleration/deceleration in rapid traverse for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Specify a time constant used for acceleration/deceleration in rapid traverse. When the optional function of bell–shaped acceleration/deceleration in rapid traverse is provided, bell–shaped acceleration/deceleration is applied in rapid traverse. If the function is not provided, linear acceleration/deceleration is applied.

- When the function is provided, set this parameter to time constant T1 used in bell–shaped acceleration/deceleration in rapid traverse, and set parameter No. 1621 to time constant T2.

- When the function is not provided, specify a time constant used for linear acceleration/deceleration.

Note 1 When parameter No. 1621 (time constant T2 used for bell–shaped acceleration/deceleration in rapid traverse) is set to 0, linear acceleration/deceleration is applied in rapid traverse even if the function is provided. In this case, this parameter stands for a time constant used for linear acceleration/deceleration in rapid traverse.

1621	Time constant t T2 used for bell–shaped acceleration/deceleration in rapid traverse for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 512

Specify time constant T2 used for bell–shaped acceleration/deceleration in rapid traverse for each axis.

Note 1 This parameter is effective when the function of bell–shaped acceleration/deceleration in rapid traverse is provided. Set parameter No. 1620 to time constant T1 used for bell–shaped acceleration/deceleration in rapid traverse, and set this parameter to time constant T2.

For details of time constants T1 and T2, see the description of General of this section.

Note 2 When this parameter is set to 0, linear acceleration/deceleration is applied in rapid traverse. The setting in parameter No. 1620 is used as a time constant in linear acceleration/deceleration.

Reference item

CONNECTION MANUAL (This manual)	7.2.1	Automatic Acceleration/Deceleration

7.2.3 Linear Acceleration/ Deceleration after Interpolation for Cutting Feed

General

If linear acceleration/deceleration after interpolation for cutting feed is enabled (bit 0 of parameter No. 1610, CTL), acceleration/deceleration is performed as follows:

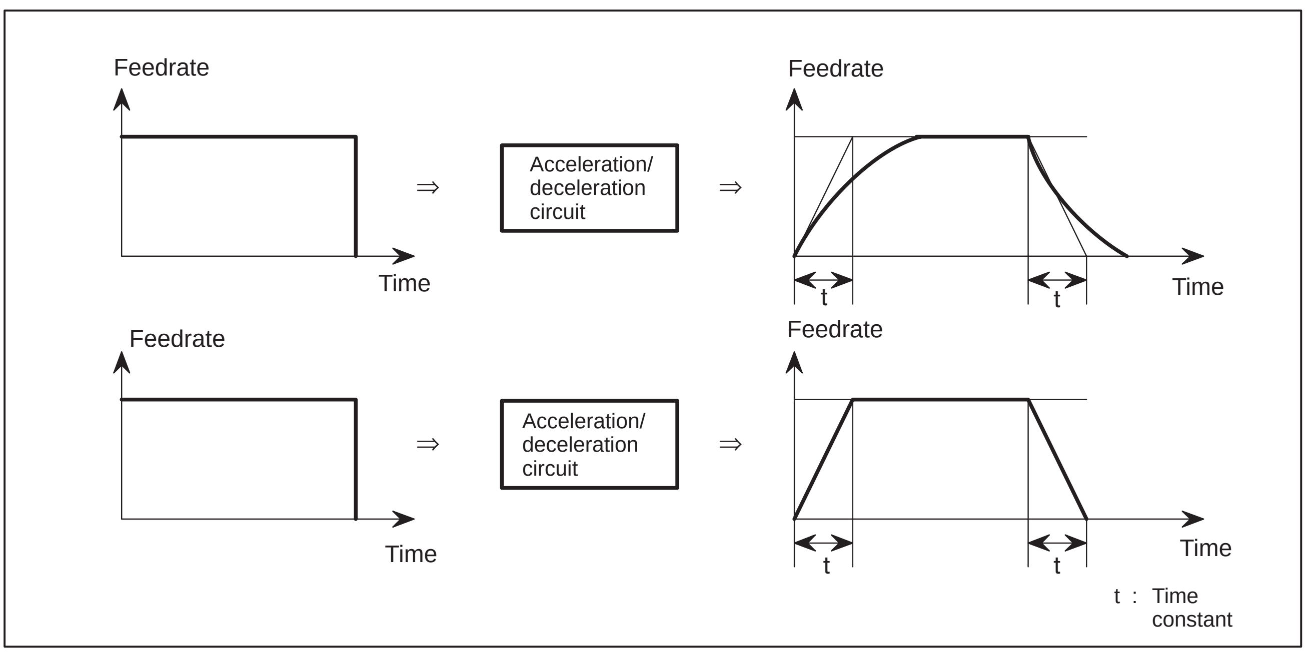
Cutting feed: Linear acceleration/deceleration (constant acceleration time)

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.



Linear acceleration/deceleration after interpolation for cutting feed is an optional function. This function is enabled when the CTL bit (bit 0 of parameter No. 1610) is specified. 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

[Data type] Bit axis

CTLx Acceleration/deceleration in cutting feed including feed in dry run
0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation is applied.

Parameter		Acceleration/deceleration
CTBx	CTLx	
0	0	Exponential acceleration/deceleration
0	1	Linear acceleration/deceleration after interpolation
1	0	Bell–shaped acceleration/deceleration after interpolation

JGLx Acceleration/deceleration in manual continuous feed (jog feed)
0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation is applied (depending on which is used in cutting feed).

1622	Time constant of exponential acceleration/deceleration, linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation, in cutting feed for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration, linear acceleration/deceleration after interpolation or bell–shaped 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.

[Data type] Word axis

1624	Time constant of exponential acceleration/deceleration, bell–shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation, in jog feed for each axis.

[Unit of data] ms

[Valid data range] 0 to 4000

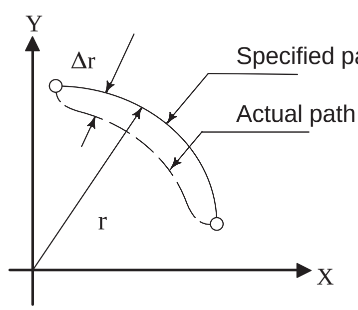
Set the time constant used for exponential acceleration/deceleration, bell shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation in jog feed for each axis.
The type of acceleration/deceleration is determined depending on the setting in parameter No. 1610.

Note

If the optional function for linear acceleration/deceleration after interpolation for cutting feed is not provided, exponential acceleration/deceleration is always selected, irrespective of the setting.

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 JGL bit (bit 4 of parameter No. 1610) is specified accordingly.

Note 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 (s)
T_2 :Time constant of servo loop (s)
(Reciprocal of position loop gain)

The maximum error in the radial direction (Δr) can be approximated by the following expressions:

$$\Delta r = \left(\frac{1}{2}T_1^2 + \frac{1}{2}T_2^2\right)\frac{v^2}{r}$$ ....Exponential acceleration/deceleration

$$\Delta r = \left(\frac{1}{24}T_1^2 + \frac{1}{2}T_2^2\right)\frac{v^2}{r}$$ ....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.

Note 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

General

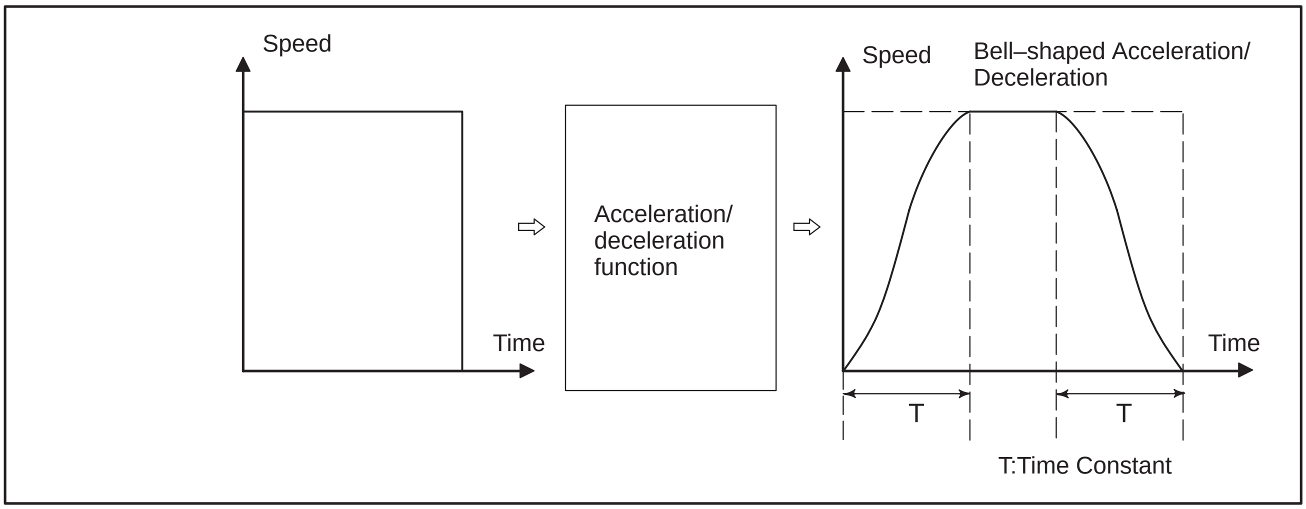
The bell–shaped acceleration/deceleration after interpolation for cutting feed provides smooth acceleration and deceleration to reduce stress and strain on the machine. If this function is enabled (bit 1 of parameter No. 1610, CTB), acceleration/deceleration is performed as follows:

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 interpolation for cutting feed 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 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			CTLBx	CTLx

[Data type] Bit axis

CTLx Acceleration/deceleration in cutting feed including feed in dry run
0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation is applied.

Note 1 If the optional function of linear acceleration/deceleration after interpolation in cutting feed is not provided, exponential acceleration/deceleration is used irrespective of this setting.
To use bell–shaped acceleration/deceleration after interpolation, set this parameter to 0 and select the acceleration/deceleration using CTBx, bit 1 of parameter No. 1610.

Parameter		Acceleration/deceleration
CTBx	CTLx	
0	0	Exponential acceleration/deceleration
0	1	Linear acceleration/deceleration after interpolation
1	0	Bell–shaped acceleration/deceleration after interpolation

CTBx Acceleration/deceleration in cutting feed including feed in dry run
0 : Exponential acceleration/deceleration or linear acceleration/deceleration after interpolation is applied (depending on the setting in CTLx, bit 0 of parameter No. 1610).
1 : Bell–shaped acceleration/deceleration after interpolation is applied.

Note 1 This parameter is effective only when the function of bell–shaped acceleration/deceleration after interpolation in cutting feed is provided. If the function is not provided, the setting in CTLx, bit 0 of parameter No. 1610, determines the type of acceleration/deceleration irrespective of the setting in this parameter.

JGLx Acceleration/deceleration in manual continuous feed (jog feed)
0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation is applied (depending on which is used in cutting feed).

1622	Time constant of exponential acceleration/deceleration, linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation, in cutting feed for each axis

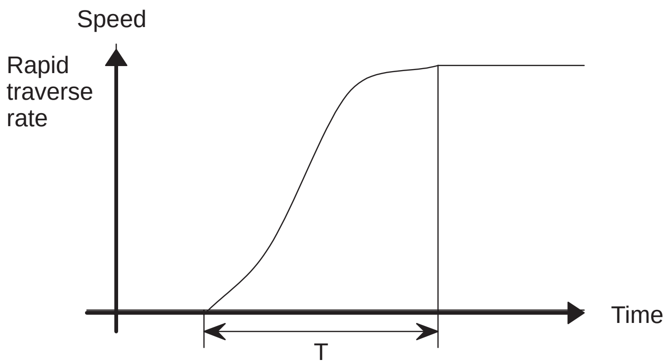
[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration, linear acceleration/deceleration after interpolation or bell–shaped 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.

<Bell–shaped acceleration/deceleration after interpolation for rapid traverse>



T : Total time. It is constant irrespective of feedrate. (Time constant is constant).

The curve corresponds to that T1 = T/2 and T2 = T/2 set in parameter No. 1620 and 1621.
No linear part exists.

1624	Time constant of exponential acceleration/deceleration, bell–shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation, in jog feed for each axis.

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration, bell–shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation in jog feed for each axis.
The type of acceleration/deceleration is determinded depending on the setting in parameter No. 1610.

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 JGL bit (bit 4 of parameter No. 1610) is specified accordingly.

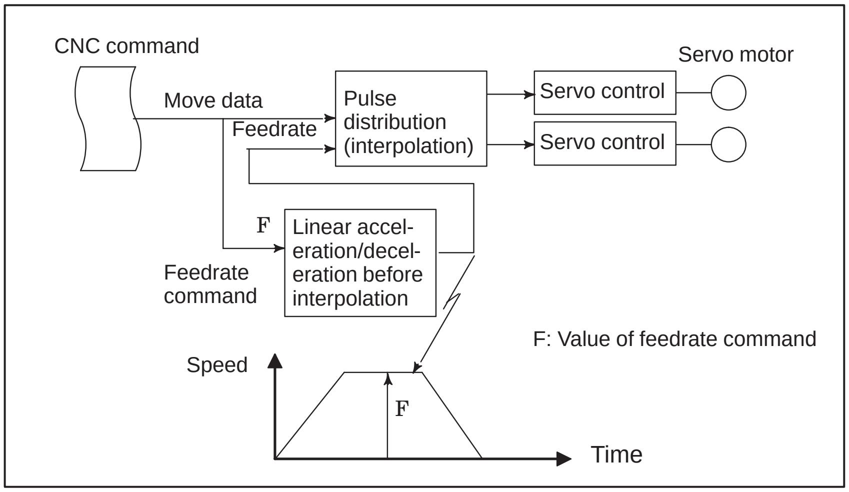
Note 2 In circular interpolation, the actual tool path after acceleration/deceleration deviates from the programmed arc in the radial direction. To overcome this radial deviation, see the note on linear acceleration/deceleration after interpolation for cutting feed in Subsection 7.2.3.

Note 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 Linear Acceleration/ Deceleration of Cutting Feed before Interpolation

General

A specified cutting feedrate can be linearly increased or decreased before interpolation. This function eliminates machining profile errors caused by the delay occurring in acceleration or deceleration. The time required for acceleration or deceleration by this function is significantly shorter than that by the function of exponential acceleration/deceleration.



The function of linear acceleration/deceleration before interpolation increases or decreases the feedrate specified in the tangential direction.

If the feedrate command is changed

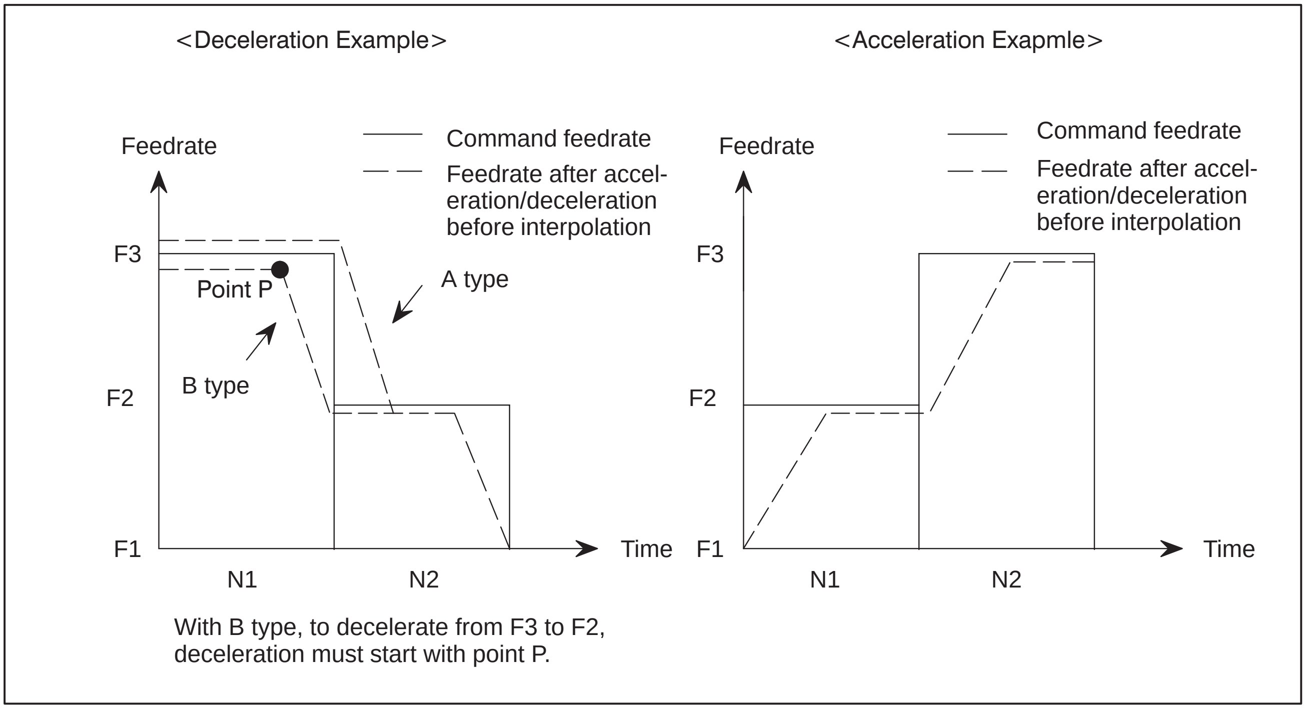
- Type A

Acceleration/deceleration is started in the block in which a new feedrate command is specified.

- Type B (Set the FWB bit (bit 0 of parameter No. 1602) to 1.)

Deceleration: Deceleration is started in a prior block such that deceleration is completed before the beginning of the block in which a new feedrate command is specified.

Acceleration: Acceleration is started in the block in which a new feedrate command is specified.



If an overtravel alarm occurs during linear acceleration/deceleration before interpolation, the movement is decelerated and stopped. As deceleration and stop are performed after the alarm occurs, the tool will overrun by an amount equal to the distance required for the deceleration. The actual overrun depends on the feedrate when the overtravel alarm occurs.

The distance can be minimized by starting deceleration in advance, such that the feedrate has fallen to the value specified in parameter No. 1784 when an overtravel alarm occurs. Because deceleration is executed such that the feedrate at the time an overtravel alarm occurs does not exceed the feedrate specified in the corresponding parameter, deceleration may be completed earlier. After deceleration is completed, the feedrate specified in the parameter is maintained.

Deceleration is performed when the following condition is satisfied:

Distance to stored stroke limit for each axis	<	Distance needed to reduce the current feedrate (tangential feedrate) to that specified in parameter No. 1784

The overrun is calculated as follows:

$$\text{Overrun} \leqq \frac{[\text{FIX}\ (\frac{F_{OT}}{F} \times \frac{T}{8}) + 1.5]^2 \times \frac{F}{T}}{1875}$$

F : Maximum cutting feedrate in linear acceleration/deceleration before interpolation (parameter No. 1630)

T : Time needed to attain the maximum cutting feedrate in linear acceleration/deceleration before interpolation (parameter No. 1631)

F_{OT}: Feedrate at the time an overtravel alarm occurs during linear acceleration/deceleration before interpolation (parameter No. 1784)

FIX: Round off at the unit of the increment system.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1602								FWB

[Data type] Bit

FWB Linear acceleration/deceleration of cutting feed before interpolation
0 : Type A of acceleration/deceleration before interpolation is used.
1 : Type B of acceleration/deceleration before interpolation is used.

1630	Parameter 1 for setting an acceleration for linear acceleration/deceleration before interpolation (maximum machining feedrate during linear acceleration/deceleration before interpolation)

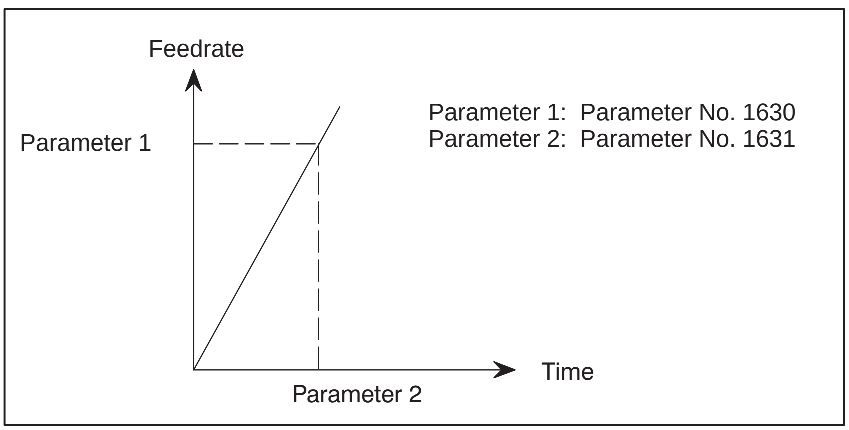
[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

This parameter is used to set an acceleration for linear acceleration/deceleration before interpolation. In this parameter, set a maximum machining feedrate during linear acceleration/deceleration before interpolation. In parameter No. 1631, set a time used to reach the maximum machining feedrate.



Note 1 When 0 is set in parameter No. 1630 or parameter No. 1631, linear acceleration/deceleration before interpolation is disabled.

Note 2 In the look–ahead control mode, parameter No. 1770 and parameter No. 1771 are valid.

[Data type] Word

[Unit of data] 1 ms

1631	Parameter 2 for setting an acceleration for linear acceleration/deceleration before interpolation (time used to reach the maximum machining feedrate during linear acceleration/deceleration before interpolation.)

[Valid data range] 0 to 4000

This parameter is used to set an acceleration for linear acceleration/deceleration before interpolation. In this parameter, set the time (time constant) used to reach the feedrate set in parameter No. 1630.

Note 1 When 0 is set in parameter No. 1630 or parameter No. 1631, linear acceleration/deceleration before interpolation is disabled.

Note 2 In parameter Nos. 1630 and 1631, set values that satisfy the following:

$$\frac{\text{Parameter No. 1630}}{\text{Parameter No. 1631}} \geqq 5$$

Note 3 In the look–ahead control mode, parameter No. 1770 and parameter No. 1771 are valid.

1784	Feedrate when overtravel alarm has generated during acceleration/deceleration before interpolation

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800

Deceleration is started beforehand to reach the feedrate set in the parameter when an overtravel alarm is issued (when a limit is reached) during linear acceleration/deceleration before interpolation. By using this parameter, the overrun distance that occurs when an overtravel alarm is output can be reduced.

Note 1 When 0 is set in this parameter, the control described above is not exercised.

Note 2 Use type–B linear acceleration/deceleration before interpolation (by setting bit 0 (FWB) of parameter No. 1602 to 1).

Note 3 The control described above is applicable only to stored stroke limit 1.

Note

Note 1 If a block without a move command is found during acceleration/deceleration before interpolation, the movement is decelerated and temporarily stopped in the previous block.

Note 2 If a one–shot G code is specified during acceleration/deceleration before interpolation, the movement is decelerated and temporarily stopped in the previous block.

Note 3 If an M, S, or T code is specified in a block containing a move command during acceleration/deceleration before interpolation, the movement is decelerated and temporarily stopped in that block.

Note 4 Even during acceleration/deceleration before interpolation, the block of G31 (skip function) is not subjected to acceleration/deceleration.

Note 5 If the machine lock signal (MLK1 to MLK8) for an axis is set on or off during acceleration/deceleration before interpolation, the axis for which machine lock is performed is not subjected to acceleration/deceleration.

Note 6 During acceleration/deceleration before interpolation, automatic corner override is enabled only when the internal circular cutting feedrate is changed.

Note 7 Even during acceleration/deceleration before interpolation, acceleration/deceleration after interpolation can be executed. So that acceleration/deceleration is executed only before interpolation, set the time constant for acceleration/deceleration after interpolation to zero.

Note 8 In acceleration/deceleration before interpolation of type B, deceleration is started if preprocessing for the next block has not been completed before the remaining distance of the current block becomes less than that needed to decelerate and stop the movement.

Note 9 If an F1–digit command is executed in the inch input system, avoid specifying a command for simultaneous movement on two axes, including a rotation axis during acceleration/deceleration before interpolation (M series).

Note 10 The error detect signal (SMZ) is invalid during acceleration/deceleration before interpolation (T series).

7.2.6
Corner Control

7.2.6.1
In–position check

General

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 1 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 INP8 <F104>

[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 number in the signal name corresponds to the control axis number.

I N P 1

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 condition] These signals turn to "1" in the following case :

· The acceleration/deceleration delay of the corresponding control axis is zero and the servo error is within the specified allowance.

These signals turn to "0" in the following cases :

· The acceleration/deceleration delay of the corresponding control axis is not zero.

· The servo error of the corresponding control axis exceeds the specified allowance.

	#7	#6	#5	#4	#3	#2	#1	#0
F104	INP8	INP7	INP6	INP5	INP4	INP3	INP2	INP1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601			NCI					

[Data type] Bit

NCI Inposition check at deceleration
0 : Performed
1 : Not performed

1826	In–position width for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

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.

Note

Note 1 The in–position signals may turn to “1” even during the movement if the axis is fed at very low speed.

Note 2 The in–position check function is enabled, at the interface between two cutting blocks, in the following cases:

M series	When the exact stop command (G09) or exact stop mode command (G61) is specified
T series	When the error detect signal is on

7.2.6.2 In–position check Independently of Feed/Rapid traverse

General

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 Subsection 7.2.6.1.

Parameter

1601	#7	#6	#5	#4	#3	#2	#1	#0
			NCI					

[Data type] Bit

NCI Inposition check at deceleration
0 : Performed
1 : Not performed

1801	#7	#6	#5	#4	#3	#2	#1	#0
			CIN	CCI				

[Data type] Bit

CCI The in–position area for cutting feed is:
0 : Set in parameter No. 1826 (same as for rapid traverse).
1 : Set in bit 5 (CIN) of parameter No. 1801.

CIN When bit 4 (CCI) of parameter No. 1801 = 1, the in–position area for cutting feed is:
0 : Use value in parameter No. 1827 if the next block is also for cutting feed, or use value in parameter No. 1826 if the next block is for rapid traverse.
1 : Use value in parameter No. 1827, regardless of the next block. (The setting of parameter No. 1826 is used for rapid traverse, and the setting of parameter No. 1827 is used for cutting feed.)

		Parameter CIN (No. 1801#5)			
		0		**1**	
Parameter CCI (No. 1801#4)	0	Rapid→Rapid Rapid→Feed Feed→Feed Feed→Rapid	No. 1826 No. 1826 No. 1826 No. 1826	Rapid→Rapid Rapid→Feed Feed→Feed Feed→Rapid	No. 1826 No. 1826 No. 1826 No. 1826
	1	Rapid→Rapid Rapid→Feed Feed→Feed Feed→Rapid	No. 1826 No.1826 No.1827 No. 1826	Rapid→Rapid Rapid→Feed Feed→Feed Feed→Rapid	No. 1826 No. 1826 No. 1827 No. 1827

1826	In–position width for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

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.

[Data type] Word

1827	In–position width in cutting feed for each axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set an in–position width for each axis in cutting feed. This parameter is valid when bit 4 (CCI) of parameter No. 1801=1.

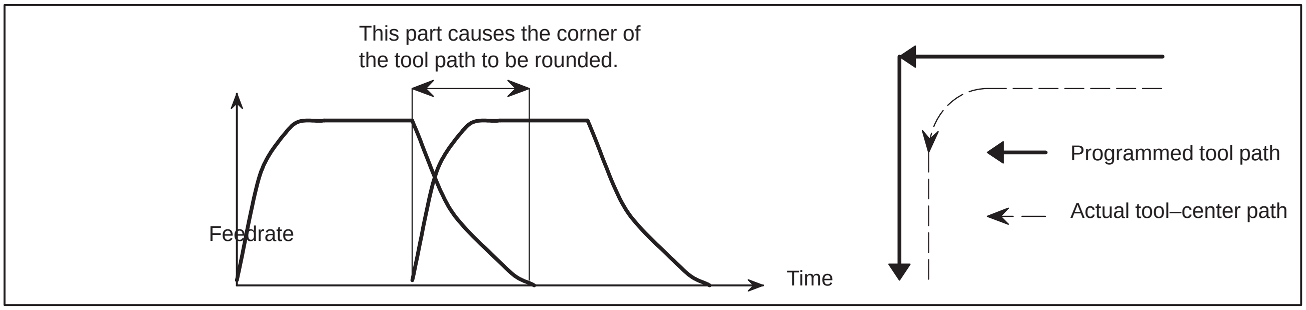
Note

Note 1 If the NCI bit (bit 5 of parameter No. 1601) 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.

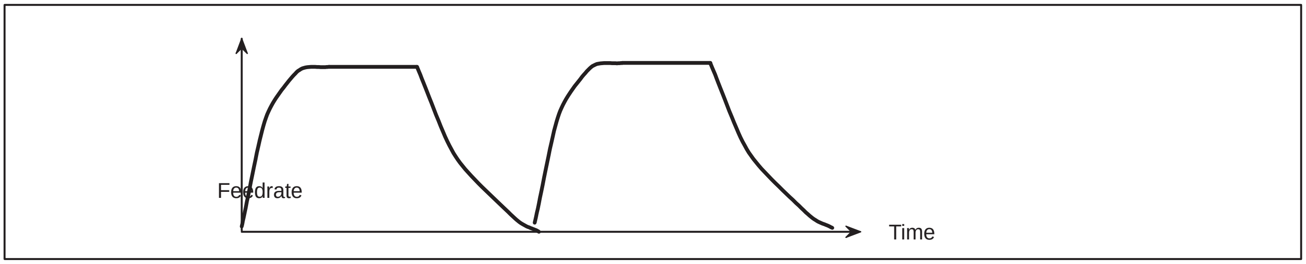
7.2.6.3 Error Detect (T series)

General

Generally, the CNC does not zero the feedrate at the interface of two blocks during cutting feed.
Because of this, a corner of a tool path may be rounded.



If the error detect signal is used, it is possible to specify that a block not be started until the acceleration/deceleration of the previous block has been completed.



Signal

Error detect signal SMZ <G053, #6>

[Classification] Input signal

[Function] Enables error detection.

[Operation] If the signal is set to 1, the control unit operates as follows:

- At the interface of two blocks during cutting feed, the control unit waits until the acceleration/deceleration of the first block has been completed. Only then does the control unit execute the next block.
  The setting of the SMZ signal determines whether, at the interface of two cutting blocks, the control unit waits until the acceleration/deceleration of the previous block has been completed.

Siganl address

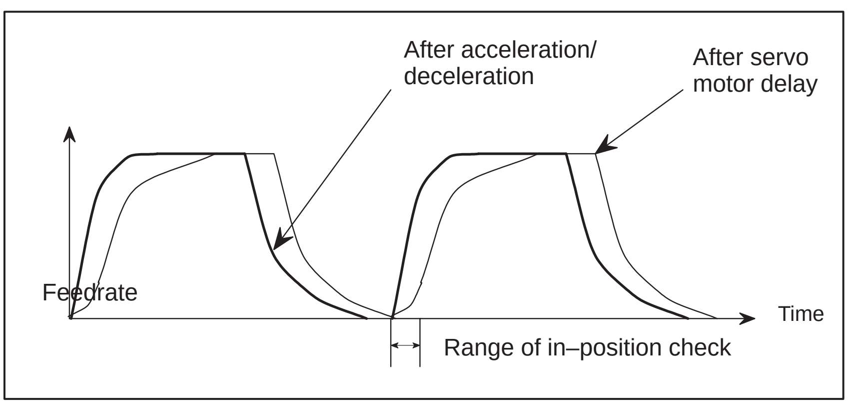
	#7	#6	#5	#4	#3	#2	#1	#0
G053		SMZ						

Note

Note 1 If the error detect signal is on, a cutting block is not executed until the acceleration/deceleration of the previous cutting block has been completed.

This function alone cannot prevent corner rounding due to delay caused by the servo motor, however.

To prevent corner rounding due to delay caused by the servo motor, use the in–position check function together with this function.



7.2.7 Feed Forward in Rapid Traverse

General

Feed–forward control can be performed even during rapid traverse. In this case, the servo position error is reduced, thus reducing the time required for positioning to within the specified range.

Parameter

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

[Data type] Bit

FFR Feed–forward control is enabled for
0 : Cutting feed only
1 : Cutting feed and rapid traverse

Reference item

For details of this function, refer to the ”FANUC AC SERVO AMPLIFIER Maintenance Manual (B–65005E)” or the ”FANUC CONTROL MOTOR α series Maintenance Manual (B–65165E).”

8 AUXILIARY FUNCTION

8.1 MISCELLANEOUS FUNCTION/2ND AUXILIARY FUNCTION

General

- **Miscellaneous Function (M code)**

When a numeral of up to 8 digits is specified following address M, code signal and a strobe signal are sent to the machine. The machine uses these signals to turn on or off its functions.
Usually, only one M code can be specified in one block. In some cases, however, up to three M codes can be specified for some types of machine tools (see 8.3 “Multiple M code per 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 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 more digits than the specified value are specified, an alarm occurs.

- **Command Format of 2nd Auxiliary Function (T series)**

- **Command range**

0 to ±99,999,999

- **Command method**

1.The decimal point can be used for input.

Command	Output value
B10.	10000
B10	10

2.It is possible to change over the scale factor of B output, 1000 or 1 when the decimal point input is omitted, using the parameter DPI (No.3401#0).

Command		Output value
When DPI is 1:	B1	1000
When DPI is 0:	B1	1

3.It is possible to change over the scale factor of B output 1000 or 10000 when the decimal point input is omitted in the inch input system, using the parameter AUX (No.3405#0). When DPI=1.

Command		Output value
When AUX is 1:	B1	10000
When AUX is 0:	B1	1000

Basic procedure

The following signals are used with these functions. (For details of the spindle–speed function and tool function, see Sections 9 and 10.)

Function	Program address	Output signal			Input signal
		Code signal	Strobe signal	Distribution end signal	Completion signal
Miscellaneous function	M	M00 to M31	MF	DEN	FIN
Spindle–speed function	S	S00 to S31	SF		
Tool function	T	T00 to T31	TF		
Secondary 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 miscellaneous function is described below. The procedures for the spindle–speed function, tool function, and secondary 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 more digits than the specified value are specified, an alarm occurs.

(2) Code signal M00 to M31 is sent. After period TMF, specified in parameter No. 3010 (standard value: 16 msec), strobe signal MF is set to 1. The code signal is the binary representation of the programmed value xxx.(*1) If the move, dwell, spindle–speed, or another function is specified at the same time as the miscellaneous function, the execution of the other function is started when the code signal of the miscellaneous function is sent.

(3) When the strobe signal is set to 1, the PMC reads the code signal and performs the corresponding operation.

(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, the PMC sets completion signal FIN to 1. The completion signal is used by the miscellaneous function, spindle–speed function, tool function, secondary 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 miscellaneous function.(*2)

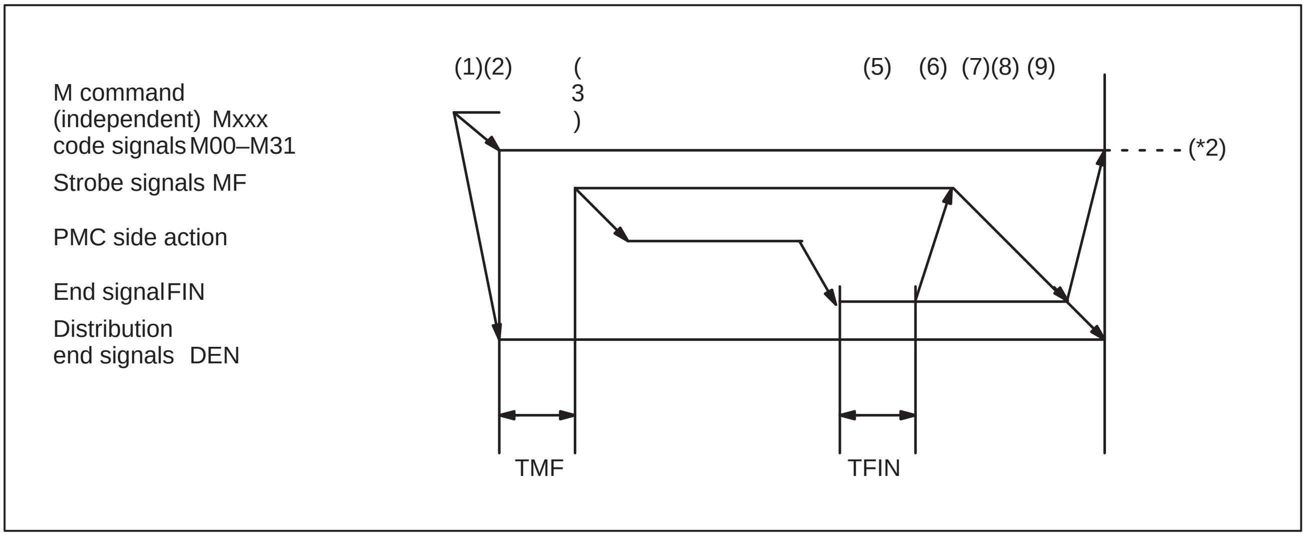
(9) Once all other commands in the same block have been completed, the CNC executes the next block.

*1 When the tool function is executed, the programmed tool number is sent as the code signal (T series).

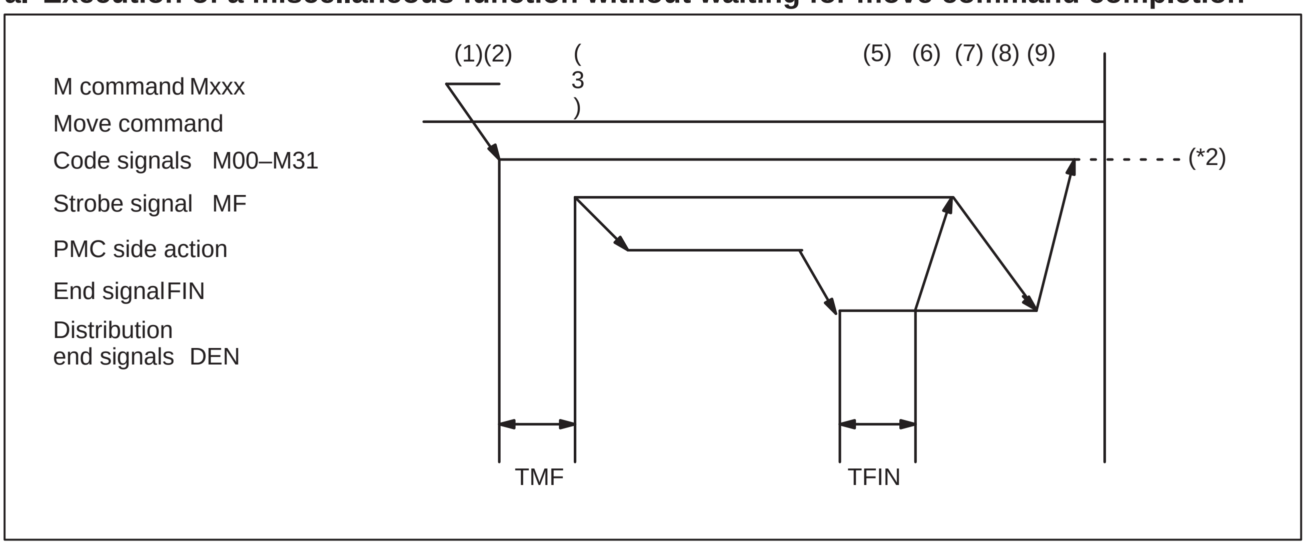
*2 When the spindle–speed function, tool function, or secondary miscellaneous function is executed, the code signal is maintained until a new code for the corresponding function is specified.

The timing diagram is shown below:

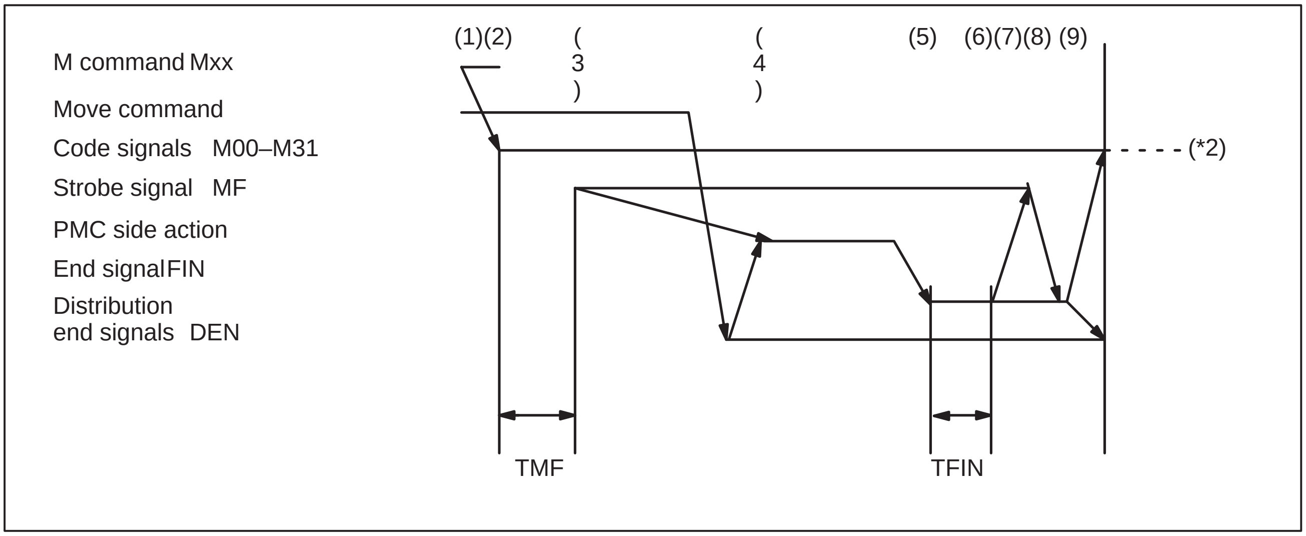
Example 1 Single miscellaneous function specified in a block



Example 2 Move command and miscellaneous function in the same block
a. Execution of a miscellaneous function without waiting for move command completion



b. Execution of a miscellaneous function after move command completion



Signal

Miscellaneous function code signals M00 to M31 <F010 to F013> Miscellaneous function strobe signal MF <F007#0>

[Classification] Output signal

[Function] These signals report the specification of miscellaneous functions.

[Output condition] For the output conditions and procedure, see the description of "Basic procedure" above.

Note 1 The following miscellaneous functions are only processed internally by the control unit; they are not subject to output even when specified:
- 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)

Note 2 Decode signals as well as the code signals and strobe signal are output for the miscellaneous functions listed below.
M00, M01, M02, M30

Decode M signals DM00<F009#7>, DM01 <F009#6>, DM02 <F009#5>, DM30 <F009#4>

[Classification] Output signal

[Function] These signals report particular miscellaneous functions are specified. The miscellaneous 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 condition] A decode M signal goes “1” when:

· The corresponding miscellaneous 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 miscellaneous 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

Spindle–speed code signals S00 to S31 <F022–F025> Spindle–speed strobe signal SF <F007#2>

[Classification] Output signal

[Function] These signals report that spindle speed functions have been specified.

[Output condition] For the output conditions and procedure, see the description of “Basic procedure” above.

Tool function code signals T00 to T31 <F026–F029> Tool function strobe signal TF <F007#3>

[Classification] Output signal

[Function] These signals report that tool functions have been specified.

[Output condition] For the output conditions and procedure, see the description of “Basic procedure” above.

Second auxiliary function code signals B00 to B31 <F030–F033> Second auxiliary function strobe signal BF <F007#4> (For lathes) <F007#7> (For machining center)

[Classification] Output signal

[Function] These signals report that second auxiliary functions have been specified.

[Output condition] For the output conditions and procedure, see the description of "Basic procedure" above.

End signal FIN <G004#3>

[Classification] Input signal

[Function] This signal reports the completion of a miscellaneous function, spindle–speed function, tool function, second 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.

Note 1 Only one end signal is used for all functions above. The end signal must go "1" after all functions are completed.

Distribution end signals DEN <F001#3>

[Classification] Output signal

[Function] These signals report that all commands (such as move commands and dwell) are completed except those miscellaneous functions, spindle–speed 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 condition] The DEN signal turns to "1" when:

· The completion of miscellaneous functions, spindle–speed functions, tool functions, and so forth is being awaited, 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 1 A parameter 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.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G004					FIN			
F001					DEN			
F007	BF			BF	TF	SF		MF
F009	DM00	DM01	DM02	DM30				
F010	M07	M06	M05	M04	M03	M02	M01	M00
F011	M15	M14	M13	M12	M11	M10	M09	M08
F012	M23	M22	M21	M20	M19	M18	M17	M16
F013	M31	M30	M29	M28	M27	M26	M25	M24
F022	S07	S06	S05	S04	S03	S02	S01	S00
F023	S15	S14	S13	S12	S11	S10	S09	S08
F024	S23	S22	S21	S20	S19	S18	S17	S16
F025	S31	S30	S29	S28	S27	S26	S25	S24
F026	T07	T06	T05	T04	T03	T02	T01	T00
F027	T15	T14	T13	T12	T11	T10	T09	T08
F028	T23	T22	T21	T20	T19	T18	T17	T16
F029	T31	T30	T29	T28	T27	T26	T25	T24
F030	B07	B06	B05	B04	B03	B02	B01	B00
F031	B15	B14	B13	B12	B11	B10	B09	B08
F032	B23	B22	B21	B20	B19	B18	B17	B16
F033	B31	B30	B29	B28	B27	B26	B25	B24

Parameter

3010	Time lag in strobe signals MF, SF, TF, and BF

[Data type] Word

[Unit of data] 1 ms

[Valid data range] 16 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.

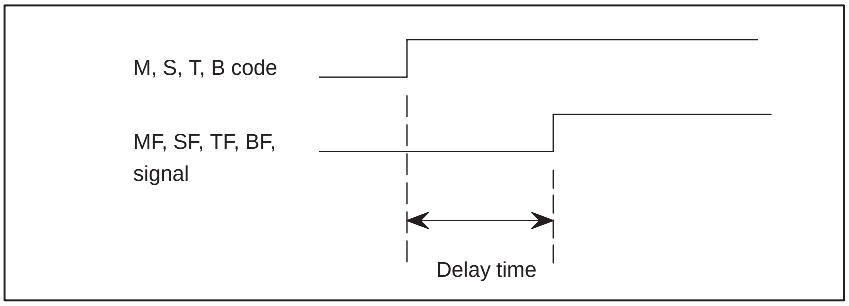


Fig. 8.1 (a) Delay Time of the strobe signal

Note 1 The time is counted in units of 8 ms. If the set value is not a multiple of eight, it is raised to the next multiple of eight.

Example: When 30 is set, 32 ms is assumed.
When 32 is set, 32 ms is assumed.
When 100 is set, 104 ms is assumed.

3011	Acceptable width of M, S, T, and B function completion signal (FIN)

[Data type] Word

[Unit of data] 1 ms

[Valid data range] 16 to 32767

Set the minimum signal width of the valid M, S, T, and B function completion signal (FIN).

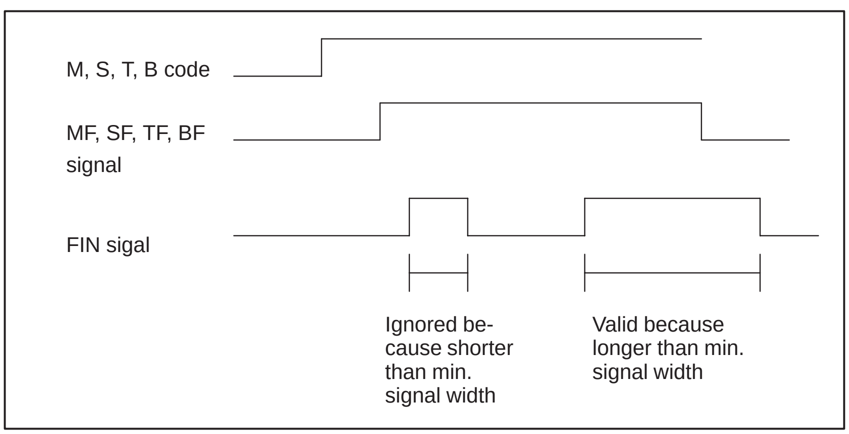


Fig. 8.1 (b) Valid Width of the FIN (M,S, T, and B Function Completion) Signal

Note 1 The time is counted in units of 8 ms. If the set value is not a multiple of eight, it is raised to the next multiple of eight.

Example: When 30 is set, 32 ms is assumed.

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
3033	Allowable number of digits for the B code

[Data type] Byte

[Valid data range] 1 to 8

Set the allowable numbers of digits for the M, S, T, and B codes.

Note 1 Up to 5 digits can be specified in the S code

	#7	#6	#5	#4	#3	#2	#1	#0
3401								DPI

[Data type] Bit

DPI When a decimal point is omitted in an address that can include a decimal point

0 : The least input increment is assumed.

1 : The unit of mm, inches, or second is assumed. (Pocket calculator type decimal point input)

	#7	#6	#5	#4	#3	#2	#1	#0
3404			M02	M30				

[Data type] Bit

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 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.)

M02 When M02 is specified in memory operation

0 : M02 is sent to the machine, and the head of the program is automatically searched for. So, when the end signal FIN is returned and a reset or reset and rewind operation is not performed, the program is executed, starting from the beginning.

1 : M02 is sent to the machine, but the head of the program is not searched for. (The head of the program is searched for by the reset and rewind signal.)

	#7	#6	#5	#4	#3	#2	#1	#0
3405								AUX

[Data type] Bit

AUX The least increment of the command of the second miscellaneous function specified with a decimal point

0 : Assumed to be 0.001

1 : Depending on the input increment. (For input in mm, 0.001 is assumed, or for input in inches, 0.0001 is assumed.)

3411	M code preventing buffering 1
3412	M code preventing buffering 2
3413	M code preventing buffering 3
⋮	⋮
3420	M code preventing buffering 10

[Data type] Byte

[Valid data range] 0 to 255

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	Minimum value 1 of M code preventing buffering
3422	Maximum value 1 of M code preventing buffering
3423	Minimum value 2 of M code preventing buffering
3424	Maximum value 2 of M code preventing buffering
3425	Minimum value 3 of M code preventing buffering
3426	Maximum value 3 of M code preventing buffering
3427	Minimum value 4 of M code preventing buffering
3428	Maximum value 4 of M code preventing buffering
3429	Minimum value 5 of M code preventing buffering
3430	Maximum value 5 of M code preventing buffering
3431	Minimum value 6 of M code preventing buffering
3432	Maximum value 6 of M code preventing buffering

[Data type] Word

[Valid data range] 0 to 65535

When a specified M code is within the range specified with parameter Nos. 3421 and 3422, 3433 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 · The specification of a minimum value that exceeds the specified maximum value is invalid.
· When there is only one data item, set the following: minimum value = maximum value.

Note

Note 1 When a move command and miscellaneous function are specified in the same block, the commands are executed in one of the following two ways:
i) Simultaneous execution of the move command and miscellaneous function commands.
ii) Executing miscellaneous function commands upon completion of move command execution.
The selection of either sequence depends on the sequence of PMC.

Note 2 When 2nd auxiliary function is equipped, axis address B cannot be used.

Note 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 (Nos. 3411 to 3420).

Note 4 For M00 and M01 only, miscellaneous function code signal, auxiliary function strobe signal, and M decode signals are sent; the control of program stop and optional stop shall be designed on the PMC side.

Note 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..

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.12.1	Miscellaneous function (M code)
	II.12.4	2nd Auxiliary function (B code)
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.12.1	Miscellaneous function (M code)
	II.12.4	2nd Auxiliary function (B code)

8.2 AUXILIARY FUNCTION LOCK

General

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 <G005#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 or specified for 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 miscellaneous 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 miscellaneous 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.

Note 1 Even when this signal is "1", spindle analog output or spindle serial output is executed.

Auxiliary function lock check signal MAFL <F004#4>

[Classification] Output signal

[Function] This signal reports the state of the auxiliary function lock signal AFL.

[Output condition] 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
G005		AFL						

	#7	#6	#5	#4	#3	#2	#1	#0
F004				MAFL				

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.5.1	Machine lock and auxiliary function lock
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.5.1	Machine lock and auxiliary function lock

8.3 MULTIPLE M COMMANDS IN A SINGLE BLOCK

General

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. 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 ; : : : : :

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.

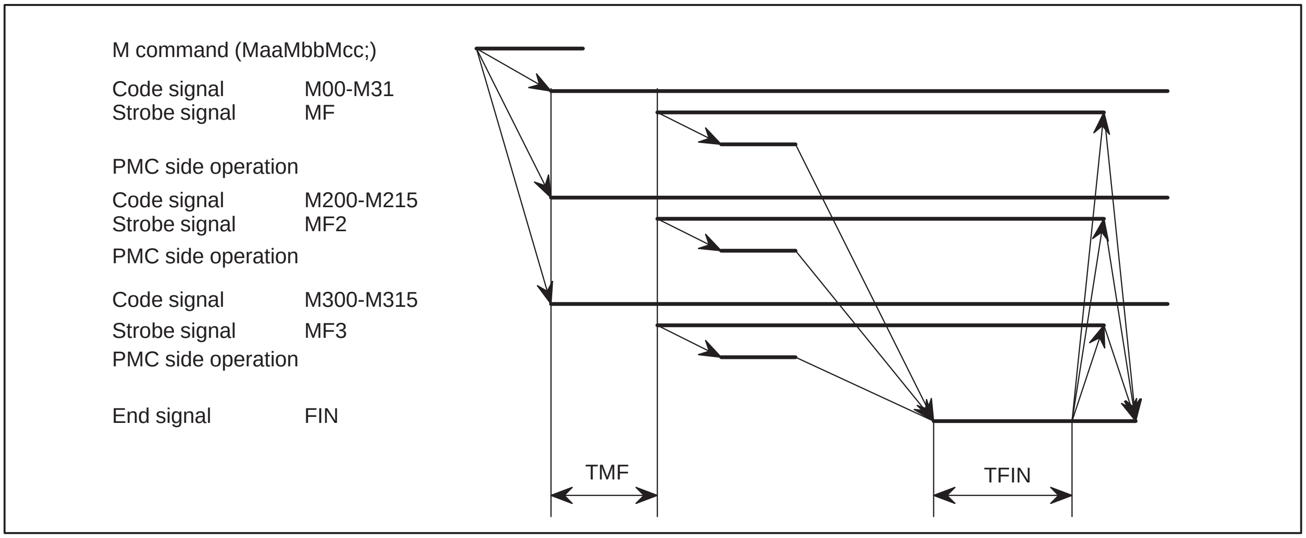
(3) On the PMC side, the code signals corresponding to the respective strobe signals are read when the strobe signals become "1", and the appropriate operations are performed.

(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 signal M200 to M215 <F014, F015> M300 to M315 <F016, F017> 2nd, 3rd M Function strobe signal MF2 <F008#4> MF3 <F008#5>

[Classification] Output signal

[Function] Indicates that second and third auxiliary functions have been issued. The output conditions and procedures are the same as that described in “Basic procedure”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F008			MF3	MF2				
F014	M207	M206	M205	M204	M203	M202	M201	M200
F015	M215	M214	M213	M212	M211	M210	M209	M208
F016	M307	M306	M305	M304	M303	M302	M301	M300
F017	M315	M314	M313	M312	M311	M310	M309	M308

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3404	M3B							

[Data type] Bit

M3B The number of M codes that can be specified in one block
0 : One
1 : Up to three

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 M00, M01, M02, M30, M98, M99, or M198 must not be specified together with another M code.

Note 3 Some M codes other than M00, M01, M02, M30, M98, M99, and M198 cannot be specified together with other M codes; each of those M codes must be specified in a single block.
Such M codes include these which direct the CNC to perform internal operations in addition to sending the M codes themselves to the machine. To be specified, such M codes are M codes for calling program numbers 9001 to 9009 and M codes for disabling advance reading (buffering) of subsequent blocks.
The M codes which can be specified in a single block must be those which the CNC send only the M code signals to the machine side.

Note 4 Only one M code can be specified in a block for MDI operation.

Note 5 The 1st M code can be up to 8 digits and 2nd, 3rd M codes can be the values up to 65535.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.12.2	Multiple M commands in a single block
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.12.2	Multiple M commands in a single block

8.4 HIGH–SPEED M/S/T/B INTERFACE

General

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 miscellaneous functions (M code commands) as an example. The same description applies to the spindle–speed function (S code), tool function (T code) and 2nd auxiliary function (B code).

Basic procedure

(1) Assume that the following program is given:
Mxx;
Myy;

(2) In response to an M command, the NC system sends out the code signals M00 to M31. After a time set by parameter (No. 3010) elapses, the NC 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 figure 1 below shows the timing chart of these signals with the new system. For comparison, Fig 2 shows the timing chart of the conventional system.

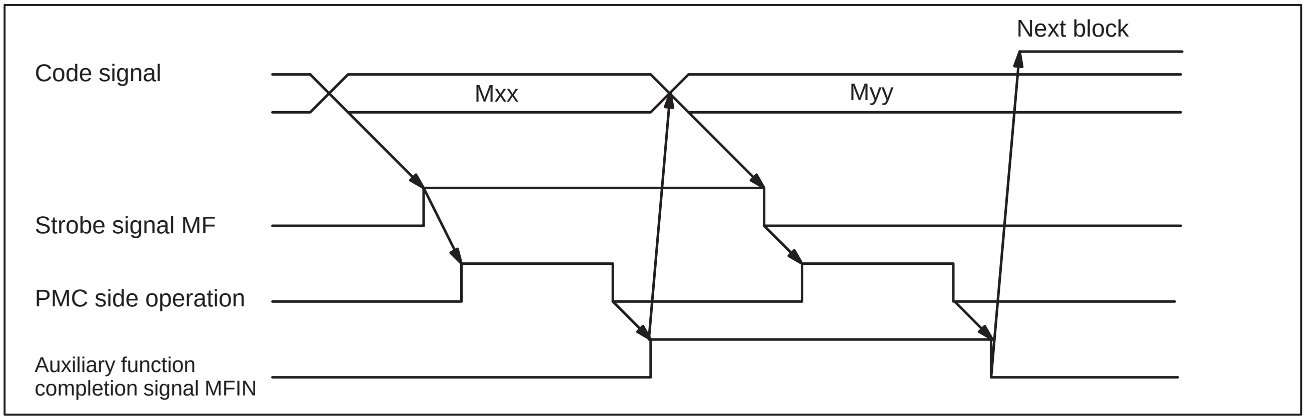


Fig. 8.4 (a) Timing Chart of the High–Speed System

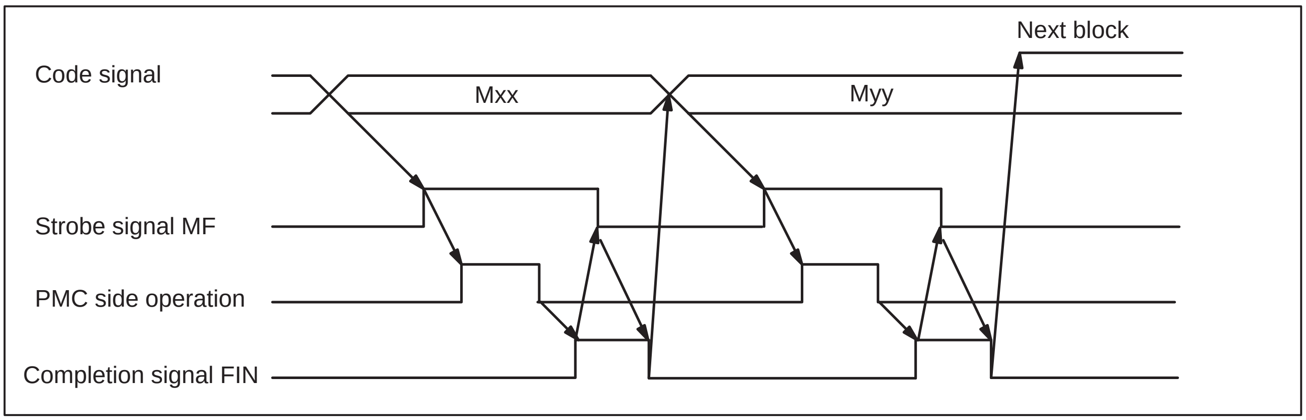


Fig. 8.4 (b) Timing Chart of the Usual System

A high–speed interface can also be used for multiple M commands issued for one block. This interface provides separate completion signals for each M code. 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 miscellaneous function (M series).

Signal

Miscellaneous function completion signal MFIN <G005#0>

[Classification] Input signal

[Function] Reports that the execution of a miscellaneous function using the high–speed M/S/T/B interface is completed.

[Operation] For the operation and aprocedure of the contol unit when this signal turns to “1” and “0”, see the description of “Basic procedure” above.

Spindle function completion signal SFIN <G005#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” and “0”, see the description of “Basic procedure” above.

Tool function completion signal TFIN <G005#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” and “0”, see the description of “Basic procedure” above.

2nd auxiliary function completion signal BFIN <G005#4> (T series) <G005#7> (M series)

[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” and “0”, see the description of “Basic procedure” above.

2nd, 3rd M function completion signals MFIN2, MFIN3 <G004#5>

[Classification] Input signal

[Function] Indicate that when the high–speed interface is used for multiple M commands per block, the second to 3rd M functions have been completed.

[Operation] See “Basic procedure” for how the control unit operates and what it performs when the signal turns to “1” and “0”.

External operation signal for high–speed interface (M series) EFD <F007#1>

[Classification] Output signal

[Function] Indicates that positioning for the external operation function has been completed for the high–speed M, S, T, or B interface, and that another external operation is required.

[Operation] Refer to the description of the output conditions and procedure described in “basic procedure.”

External operation function completion signal (M series) EFIN <G005#1>

[Classification] Input signal

[Function] Indicates that the external operation function has been completed for the high–speed M, S, T, or B interface.

[Operation] The “basic procedure” describes the procedure and operation of the control unit when the signal is set to 1 or 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G004			MFIN3	MFIN2				

	#7	#6	#5	#4	#3	#2	#1	#0
G005	BFIN			BFIN	TFIN	SFIN	EFIN	MFIN

	#7	#6	#5	#4	#3	#2	#1	#0
F007							EFD	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3001	MHI							

MHI Exchange of strobe and completion signals for the M, S, T, and B codes
0 : Normal
1 : High–speed

Note

Note 1 The strobe signals MF, SF, TF, and BF are "0" when the power is turned on.

Note 2 When the control unit is reset, MF, SF, TF, and BF are set to "0".

Reference item

CONNECTION MANUAL (This manual)	8.1	Miscellaneous function/2nd auxiliary function
	8.3	Multiple M commands in a block
	11.8	External operation function

8.5 WAITING M CODE (Two–path control)

General

Control based on M codes is used to cause one path to wait for the other during machining. By specifying an M code in a machining program for each path, the two paths can wait for each other at a specified block. When an M code for waiting is specified in a block for one path during automatic operation, the other path waits for the same M code to be specified before staring the execution of the next block.

A range of M codes used as M codes for waiting is to be set in the parameters (Nos. 8110 and 8111) beforehand.

Signal

No–wait signal NOWT <G063#1>

[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 M code for waiting 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 <F063#6>

[Classification] Output signal

[Function] Indicates that the CNC is waiting for the M code of either path 1 or 2.

[Operation] This signal is “1” as long as:

(i) 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:

(ii) Neither of the paths are waiting for the other.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G063							NOWT	

	#7	#6	#5	#4	#3	#2	#1	#0
F063		WATO #1						

Parameter

8110	Waiting M code range (minimum value)

[Data type] Two–word

[Valid data range] 0 and 100 to 99999999

This parameter specifies the minimum value of the waiting M code.

The waiting M code range is specified using parameter 8110 (minimum value) and parameter 8111 (maximum value).

(parameter 8110) ≤ (waiting M code) ≤ (parameter 8111)

Note 1 A value of 0 indicates that the waiting M code is not used.

●

8111	Waiting M code range (maximum value)

[Data type] Two–word

[Valid data range] 0 and 100 to 99999999

This parameter specifies the maximum value of the waiting M code.

Alarm and message

Number	Message	Description
160	MISMATCH WAITING M–CODE	Different M code is commanded in path 1 and 2 as waiting M code. Modify the program.

Note

Note 1 The waiting M code, unlike other M codes, does not issue code signal nor strobe signal.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.22.2	Waiting for path
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.21.2	Waiting for tool post

8.6 M CODE LOOP CHECK FUNCTION

General

This function checks whether combinations of M codes (up to three) specified in one block are correct.

The function has two purposes. One of the purposes is to alarm if an M code which must not be combined with any other M codes is combined with another. The other purpose is to alarm if an M code in a group is combined with another M code in the same group. These errors are reflected in P/S alarm No. 5016.

• Setting the group data

Pressing the SYSTEM key, then the [next menu] key several times causes the [M CODE] soft key to appear. Pressing this soft key displays the screen shown in Fig. 8.6 (a).

```
M CODE GROUP SETTING              O0000   N0000
NO.     MCG    NO.      MCG    NO.      MCG
0000    0  0010     0   0020       0
0001    0  0011     0   0021       0
0002    0  0012     0   0022       0
0003    0  0013     0   0023       0
0004    0  0014     0   0024       0
0005    0  0015     0   0025       0
0006    0  0016     0   0026       0
0007    0  0017     0   0027       0
0008    0  0018     0   0028       0
0009    0  0019     0   0029       0
 >

 MDI   ****   ***   ***        00 : 00 : 00
(        )(        )(M CODE)(        )(OPRT)
```

Fig. 8.6 (a)

Basically, item numbers correspond to M codes. However, there may be exceptions depending on parameter setting. (See descriptions of the relevant parameters.) The data specified on this screen remains until all parameters are cleared simultaneously. To go to the next page of the screen, use the [PAGE] key.

Pressing the [(OPRT)] soft key or the corresponding numeric key displays the soft keys shown in Fig. 8.6 (b). Searches for the desired item number and enter the data.

Semicolon “;” may be used for continuous data entry.

Data entry is enabled when PARAMETER WRITE ENABLE is set to “1” on the parameter setting screen.

```
 >
 MDI
       ****   ***   ***        00 : 00 : 00
(No. SRH)(        )(        )(        )( INPUT )
```

Fig. 8.6 (b)

For M codes which must be used separately from other M codes, always set their group number to “1”. Such M codes include M00, M01, M02, M30, M98, and M99. For M codes for which the CNC performs internal processing in addition to sending them to the machine, also set their group number to “1”. See Section 8.3 for details.

For M codes you do not need to check, leave them at an initial value of “0”.

The M codes can be set with a number from 0 to 127. Neither negative values nor fractions can be specified.

- **Parameter setting**

Basically, item numbers 0 to 99 correspond to M00 to M99. For item numbers 100 and higher, parameter Nos. 3441 to 3444 can specify the corresponding M codes.

The initial values for these parameters are “0”.

3441	First of the M codes assigned to item numbers 100 to 199
3442	First of the M codes assigned to item numbers 200 to 299
3443	First of the M codes assigned to item numbers 300 to 399
3444	First of the M codes assigned to item numbers 400 to 499

[Data type] Two–word

[Valid data range] 0 to 99999999

If a parameter is specified as “0”, it is set to a value specified to the previous parameter plus 100. For example, if parameter No. 3441 is specified as “0”, it is internally set to “100”. If parameter No. 3442 is specified as “0” under this condition, it is internally set to “200”.

Negative values are assumed to be “0”.

Basically, these parameters can be specified as any value. However, the following conditions should be observed to save memory space.

$X + 99 < Y$, $Y + 99 < Z$, and $Z + 99 < W$

where X = value specified for parameter No. 3441, Y = value specified for parameter No. 3442, Z = value specified for parameter No. 3443, and W = value specified for parameter No. 3444.

(Example of setting)

(i) No. 3441 = 0. No. 3442 = 0. No. 3443 = 0, No. 3444 = 0

In this case, these parameters are set to “100”, “200”, “300”, and “400” respectively. The item numbers correspond directly to the M codes. The group numbers for M500 and larger M codes are assumed to be “0” automatically.

(ii) No. 3441 = 200. No. 3442 = 0. No. 3443 = 500, No. 3444 = 800

In this case, parameter No. 3442 is set to “300”. Item numbers 100 to 299 correspond to M200 to M399, 300 to 399 correspond to M500 to M599, and 400 to 499 correspond to M800 to M899. The group numbers for M100 to M199, M400 to M499, M600 to M799, M900, and all M codes with higher numbers are automatically set to “0”.

(iii) No. 3441 = 234. No. 3442 = 345. No. 3443 = 456, No. 3444 = 567

In this case, item numbers 100 to 199 correspond to M234 to M333, 200 to 299 correspond to M345 to M444, 300 to 399 correspond to M456 to M555, and 400 to 499 correspond to M567 to M666. The group numbers for M100 to M233, M334 to M344, M446 to M455, M556 to M566, M667, and all M codes with higher numbers are automatically set to “0”.

The examples above meet the setting conditions. With these settings, up to 500 M codes can be set.

Examples that do not meet the setting conditions follow.

(iv) No. 3441 = 200. No. 3442 = 50. No. 3443 = 100, No. 3444 = 600

In this case, item numbers 0 to 99 correspond to M00 to M99, 100 to 199 correspond to M200 to M299, and 400 to 499 correspond to M600 to M699. Item numbers 200 to 399 are meaningless. With these settings, only up to 300 M codes can be set.

(v) No. 3441 = 50. No. 3442 = 100. No. 3443 = 150, No. 3444 = 200

In this case, the correspondence between the item numbers and M codes is set up as listed below. With these settings, up to 300 M codes can be set. The group number for M300 and all M codes with higher numbers are automatically set to “0”.

Item number	M code
0 – 49	0 – 49
50 – 99	Meaningless
100 –199	50 – 149
200 –249	Meaningless
250 –299	150 – 199
300 –349	Meaningless
350 –399	200 – 249
400 –449	Meaningless
450 –499	250 – 299

- **Input/output with floppy cassettes**
- **Input**

The file you want to read out should be set on the read station. First locate the file in the program screen in the EDIT mode. In this mode, display the M code group setting screen. See Section 8.2, Part III of the operator’s manual for how to locate the file.

On the soft key screen shown in Fig. 8.6 (b), pressing the next menu key several times displays the soft keys shown in Fig. 8.6 (c).

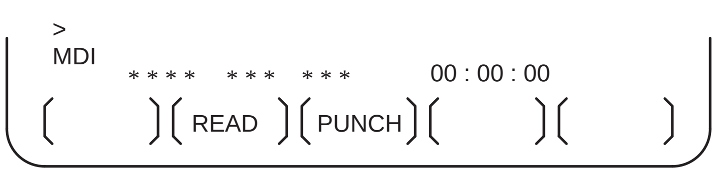


Fig. 8.6 (c)

Now pressing the [READ] key displays the soft keys shown in Fig. 8.6 (d).

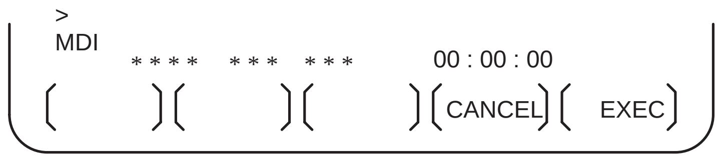


Fig. 8.6 (d)

To execute the read operation, just press the [EXEC] key.

- **Output**

Pressing the [PUNCH] key on the screen shown in Fig. 8.6 (c) displays the soft keys shown in Fig. 8.6 (d). To execute the punch operation, just press the [EXEC] key.

After this operation, you can confirm that the file M CODE GROUP is output, by searching through floppy cassette files. The output data has a format with 60000 added to the item number:

N60xxxPyyy (where xxx = 0 to 499, yyy = 0 to 127)

Parameter

3441	First of the M codes assigned to item numbers 100 to 199
3442	First of the M codes assigned to item numbers 200 to 299
3443	First of the M codes assigned to item numbers 300 to 399
3444	First of the M codes assigned to item numbers 400 to 499

[Data type] Two–word

[Valid data range] 0 to 99999999

Alarm and message

Number	Message	Description
5016	ILLEGAL COMBINATION OF M CODE	M codes which belonged to the same group were specified in a block. Alternatively, an M code which must be specified without other M codes in the block was specified in a block with other M codes.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.12.3	M CODE GROUP CHECK FUNCTION
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.12.3	M CODE GROUP CHECK FUNCTION

9 SPINDLE SPEED FUNCTION

9.1 SPINDLE SPEED FUNCTION (S CODE OUTPUT)

General

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 8.1.

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 codes.

Note

Note 1 When a move command and miscellaneous function are specified in the same block, the commands are executed in one of the following two ways:

i) Simultaneous execution of the move command and miscellaneous function commands.

ii) Executing miscellaneous function commands upon completion of move command execution.

The selection of either sequence depends on the PMC processing sequence.

Note 2 For S code output when the spindle serial output/analog spindle output is used, refer to 9.3.

9.2 SPINDLE SERIAL OUTPUT/SPINDLE ANALOG OUTPUT

9.2.1 General

There are two types of spindle motor control interfaces, spindle serial output and spindle analog output.

The spindle serial output interface can control two serial spindles. The spindle analog output interface can control one analog spindle.

The table below lists the relationships between the spindle control interfaces and the configuration of the spindle.

Spindle serial output	Spindle analog output	First spindle	Second spindle	Third spindle
○	○	First serial spindle The PC can be used.	Second serial spindle The PC can be used. (*)	Analog spindle The PC cannot be used.
○	×	First serial spindle The PC can be used.	Second serial spindle The PC can be used. (*)	=
×	○	Analog spindle The PC can be used.	=	=
×	×	See Section 9.1. ⇒ Controlled by the PMC using an external interface.		

· PC = position coder

· (*) The multispindle function is necessary to use the position coder of the second spindle (T series only).

· See section 15.4 or 9.10 for how to control the speed of the second and third spindles.

The table below lists the relationship between the spindles and functions.

○=Available ×=Unavailable

Function \ Spindle	Serial spindle		Analog spindle	
	First serial spindle	**Second serial spindle**	**When used as the first spindle (with no serial SP)**	**When used as the third spindle (with a serial SP)**
Thread cutting/feed per rotation (synchronous feed)	○	○ (*1)	○	×
Constant surface speed control	○	○ (*1)	○	×
Spindle speed fluctuation detection	○	○ (*1)	○	×
Actual spindle speed output (T series only)	○	○ (*1)	○	×
Spindle positioning (T series only)	○	×	○	×
Cs contour control	○	×	×	×
Multispindle (T series only) (*2)	○ (First spindle)	○ (Second spindle)	×	○ (Third spindle)
Rigid tapping	○	○ (*1)	○	×
Spindle synchronization	○ Master (*3)	○ Slave (*3)	×	×
Spindle control unit functions (*4), such as spindle orientation, spindle output switch, and other types of spindle switching	○	○	○	○
Polygon turning (T series only) (using the servo motor axis and spindle)	○	○ (*1)	○	×
Polygon turning between spindles (T series only) (using two spindles)	○ Master (*5)	○ Slave (*5)	×	×
Spindle output control using PMC	○	○	○	○

Notes

1. The multispindle function (T series only) is necessary. The function cannot be used for the first and second spindles simultaneously.
2. The multispindle function can control the speed of three spindles and switch the feedback signal between two position coders. It also can work without the second or third spindle.
3. For a two–path lathe application, the first spindle on tool post 1 is the master, and the first spindle on tool post 2 is the slave. The second spindle of either tool post cannot be used in spindle synchronization.
4. These functions belong to the spindle control unit. They cannot be used unless the spindle control unit supports those functions.
5. The master–slave relationship does change for the two–path lathe application. No spindle polygon turning is available for a combination of the spindle of tool post 1 and the spindle of tool post 2.

The signals and parameters for spindle speed control are common to both spindle serial output and spindle analog output. (See Section 9.3.)

The table below lists the differences related to direct control of the spindle control unit.

	Spindle control unit for spindle serial output interface	**Spindle control unit for spindle analog output interface**
Parameters for the spindle control unit	Specified as NC parameters (4000 to 4351/S1, S2) Used after being transferred to the spindle control unit	Directly specified for the spindle control unit
Control signal for the spindle control unit	Connected to the PMC via the NC G0070 to G0073 and F0045 to F0048: Addresses for the first spindle G0074 to G0077 and F0049 to F0052: Addresses for the second spindle	Connected to the PMC via an external contact
Spindle speed command interface	Digital data in a range from 0 to ± maximum spindle motor speed	Analog voltage from 0 to ±10 V (excluding portion for offset voltage adjustment)
Position coder interface	Connected to the NC via the spindle control unit	Connected directly to the NC

Signal

· Spindle control unit signals for the serial spindle

<G0070 to G0073> (input), <F0045 to F0048>
→ for the first serial spindle
<G0074 to G0077> (input), <F0049 to F0052>
→ for the second serial spindle

These addresses are on the CNC. Actually, however, they are input/output signals for the spindle control unit for the serial spindle.

For details of the signals belonging to these addresses, refer to the manuals for the serial spindle:

FANUC AC SPINDLE MOTOR series (Serial Interface) Descriptions (B–65042E)
FANUC AC SPINDLE SERVO UNIT Serial Interface S Series Maintenance Manual (B–65045E)
FANUC CONTROL MOTOR AMPLIFIER α series Descriptions (B–65162E)

Signal address

- **For 1st SERIAL SPINDLE**

	#7	#6	#5	#4	#3	#2	#1	#0
G070	MRDYA	ORCMA	SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
G071	RCHA	RSLA		SOCNA	MCFNA	SPSLA	*ESPA	ARSTA
G072								
G073								
F045	ORARA	TLMA	LDT2A	LDT1A	SARA	SDTA	SSTA	ALMA
F046					RCFNA	RCHPA	CFINA	CHPA
F047								
F048								

- **For 2ND SERIAL SPINDLE**

	#7	#6	#5	#4	#3	#2	#1	#0
G074	MRDYB	ORCMB	SFRB	SRVB	CTH1A	CTH2B	TLMHB	TLMLB
G075	RCHB	RSLB		SOCNB	MCFNB	SPSLB	*ESPB	ARSTB
G076								
G077								
F049	ORARB	TLMB	LDT2B	LDT1B	SARB	SDTB	SSTB	ALMB
F050					RCFNB	RCHPB	CFINB	CHPB
F051								
F052								

Parameter

- **Connection of serial spindle control unit**

	#7	#6	#5	#4	#3	#2	#1	#0
3701				SS2			ISI	

[Data type] Bit

ISI Specifieds whether the serial spindle interface is used.
0 : Used
1 : Not used

Note 1 This parameter is enabled only when the serial spindle interface option is provided. The parameter is used when the NC is started after serial spindle interface control is temporarily disabled during startup adjustment of the NC. This bit should normally set to be 0.

SS2 The number of connections in serial spindle control
0 : 1
1 : 2

Note 1 To connect two serial spindles, set jumper S1 on the 1st serial spindle control unit to B.

- **Parameters of serial spindle control unit**

No. 4000 – 4351: S1 → For 1st serieal spindle
S2 → For 2nd serial spindle

The above parameters are on the CNC, but actually they are used for the spindle control unit of serial spindle.
For details of these parameters, refer to the following manual:

FANUC AC SPINDLE MOTOR α series PARAMETER MANUAL (B–65160E)

FANUC AC SPINDLE SERVO UNIT Serial Interface S series Maintenance Manual (B–65045E)

Alarm and message

Number	Message	Description
749	S–SPINDLE LSI ERROR	A communication error occurred for the serial spindle. The cause may be noises, disconnection of an optical cable or the interruption of the power to the spindle amplifier. (Note) Unlike alarm No. 750, this alarm occurs when a serial communication alarm is detected after the spindle amplifier is normally activated.
750	SPINDLE SERIAL LINK START FAULT	This alarm is generated when the spindle control unit is not ready for starting correctly when the power is turned on in the system with the serial spindle. The four reasons can be considered as follows: 1) An improperly connected optic cable, or the spindle control unit's power is OFF. 2) When the NC power was turned on under alarm conditions other than SU–01 or AL–24 which are shown on the LED display of the spindle control unit. In this case, turn the spindle amplifier power off once and perform startup again. 3) Other reasons (improper combination of hardware) 4) The second spindle (when SP2, bit 4 of parameter No. 3701, is 1) is in one of the above conditions 1) to 3). Note) This alarm does not occur after the system including the spindle control unit is activated. See diagnostic display No. 409 for details.
751	FIRST SPINDLE ALARM DETECTION (AL–XX)	This alarm indicates the NC that an alarm is generated in the spindle control unit of the serial spindle. The alarm is displayed in form AL–XX (XX is a number). Refer to the manuals for serial spindle. The alarm number XX is the number indicated on the spindle amplifier. The CNC holds this number and displays on the screen.
752	FIRST SPINDLE MODE CHANGE FAULT	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contouring, 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.
761	SECOND SPINDLE ALARM DETECTION (AL–XX)	Refer to alarm No. 751. (For 2nd axis)
762	SECOND SPINDLE MODE CHANGE FAULT	Refer to alarm No. 752.(For 2nd axis)

DIAGNOSIS SCREEN

- **Information on spindle control**

	#7	#6	#5	#4	#3	#2	#1	#0
400				SAI	SS2	SSR	POS	SIC

SIC 0: No module is available for spindle serial output.
1: A module for spindle serial output is available.

POS 0: No module is available for spindle analog output.
1: A module for spindle analog output is available.

SSR 0: Spindle serial output is not used.
1: Spindle serial output is used.

SS2 0: The second spindle is not used with spindle serial output.
1: The second spindle is used with spindle serial output.

SAI 0: Spindle analog output is not used.
1: Spindle analog output is used.

401	Alarm condition for the serial spindle unit for the first spindle (AL–??)
402	Alarm condition for the serial spindle unit for the second spindle (AL–??)

- **Communication error on spindle serial output interface**

	#7	#6	#5	#4	#3	#2	#1	#0
408	SSA		SCA	CME	CER	SNE	FRE	CRE

CRE 1 : CRC error (warning)
FRE 1 : Framing error (warning)
SNE 1 : Mismatch between sending and receiving sections
CER 1 : Abnormal reception
CME 1 : No answer during auto scanning
SCA 1 : Communication error in the spindle amplifier
SSA 1 : System error in the spindle amplifier
(These errors are reflected in spindle alarm 749. They are caused by noise, disconnection, or instantaneous power interruption.)

- **Information related to the activation of the spindle serial output interface**

	#7	#6	#5	#4	#3	#2	#1	#0
409					SPE	S2E	S1E	SHE

SHE 1 : Abnormal operation in the serial spindle communication module of the CNC
S1E 1 : Abnormal operation on the first spindle during activation
S2E 1 : Abnormal operation on the second spindle during activation
SPE 1 : Serial spindle parameter not meeting activation conditions
(These errors are reflected in spindle alarm 750.)

- **Load and speed meter readings for the serial spindle**

410	First serial spindle: Load meter reading (%)
411	First serial spindle: Speed meter reading (rpm)
412	Second serial spindle: Load meter reading (%)
413	Second serial spindle: Speed meter reading (rpm)

To display the load and speed meter readings, the following parameters must be specified correctly.

Maximum motor speed: Parameter Nos. 4020 (main) and 4196 (sub)

Load meter reading at maximum output:
Parameter Nos. 4127 (main) and 4276 (sub)

Note 1 The spindle switch function is used for main/sub switching. Select main if the spindle switch function is not used.

- **Position error display during spindle synchronization**

414	Master spindle motion error during spindle synchronization
415	Slave spindle motion error during spindle synchronization
416	Absolute value of synchronization error during spindle synchronization

The display for parameter Nos. 414 to 416 are in pulse units (one pulse = 360/4096 degrees)

- **Position error display during spindle synchronization**

417	First serial spindle: Position coder feedback information
418	First serial spindle: Position error
419	Second serial spindle: Position coder feedback information
420	Second serial spindle: Position error

The above display data is the information obtained directly from the serial spindle control unit.

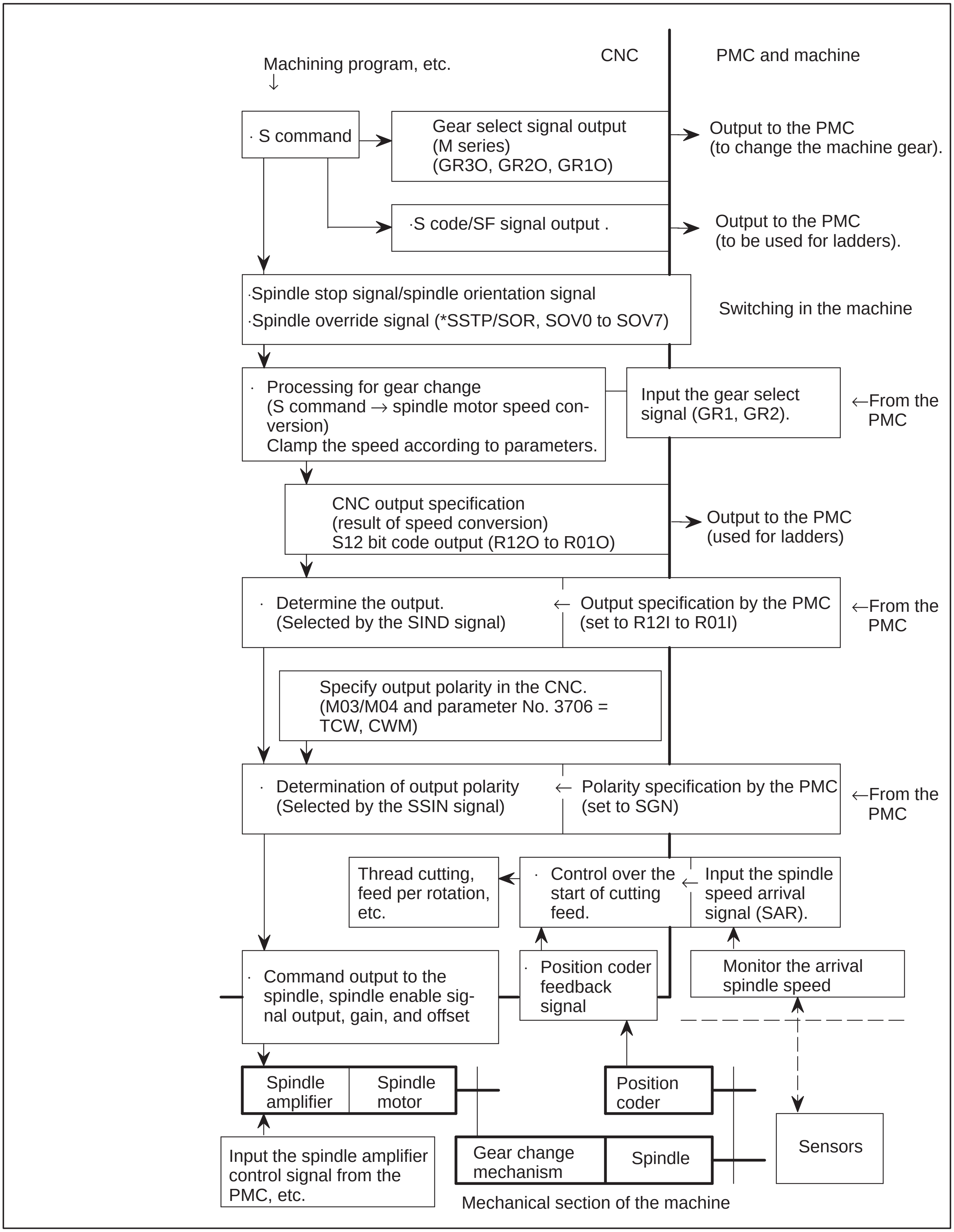
9.3 SPINDLE SPEED CONTROL

General

This section describes spindle speed control. It also explains the position coder and the spindle 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.

In the M series with bit 4 (GTT) of parameter No. 3706 = 0 without the constant surface speed control option, the CNC specifies the gear stage for the desired spindle speed to the PMC according to parameter Nos. 3741, 3742, and 3743, and the S command.

(GR3O, GR2O, GR1O <F034#2, #1, #0>)

- **S code/SF signal output**

With the spindle serial output or spindle analog output option, the spindle control function in the CNC converts the S command value to the output value for the spindle motor. Unlike a system without such options, the system with the options responds to the S command with the S code/SF signals as follows to enable gear change and constant surface speed control:

M series → Outputs the S code.
The SF signal is output only when the CNC directs the PMC to change the gear.

T series → Outputs neither S code nor SF signal.
(This is because the S code is not always the spindle speed when the constant surface speed control option is used.)

If you use the S code for processing in the PMC ladder, you must specify parameters related to parameter No. 3705.

- **Spindle stop signal (*SSTP)**

This signal sets the S command value in the CNC to 0. If the CNC has the spindle output specified (see descriptions on the SIND signal), this signal sets the speed command for the spindle to 0.

Even if the function of the spindle stop signal is not used, the signal must be set to logical 1 for the CNC to perform spindle speed control.

- **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.

In the M series, specifying bit 1 (GST) of parameter No. 3705 causes the spindle motor to rotate at 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 (G84, G74) for the M series
Thread cutting (G32, G92, G76) for the T series

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

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 (one of the four gear stages) being currently used by the machine is input in response to the gear select signal inputs (GR1, GR2).

The machine determines which gear to use.

The CNC outputs the spindle speed that corresponds to the gear stage input.

- **Selection of gear change system**

The M series system can use either M or T type.

M type ← Without constant surface speed control option, and bit 4 (GTT) of parameter No. 3706 = 0

T type ← With constant surface speed control, or bit 4 (GTT) of parameter No. 3706 = 1

The T series system can use only T type.

- **Details of M type (Output of GR10, GR20, GR30)**

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 (GR10 and GR20) or three-speed range (GR10, GR20, GR30), 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 (parameter SFA no. 3705#6).
The meaning of the gear signals is shown below:

	Gear 2–stage	Gear 3–state	Remarks
GR10	Low	Low	Low :Low Gear Middle :Middle Gear High :High Gear
GR20	High	Middle	
GR30		High	

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.

- M type gear change method A (Fig. 9.3 (a))

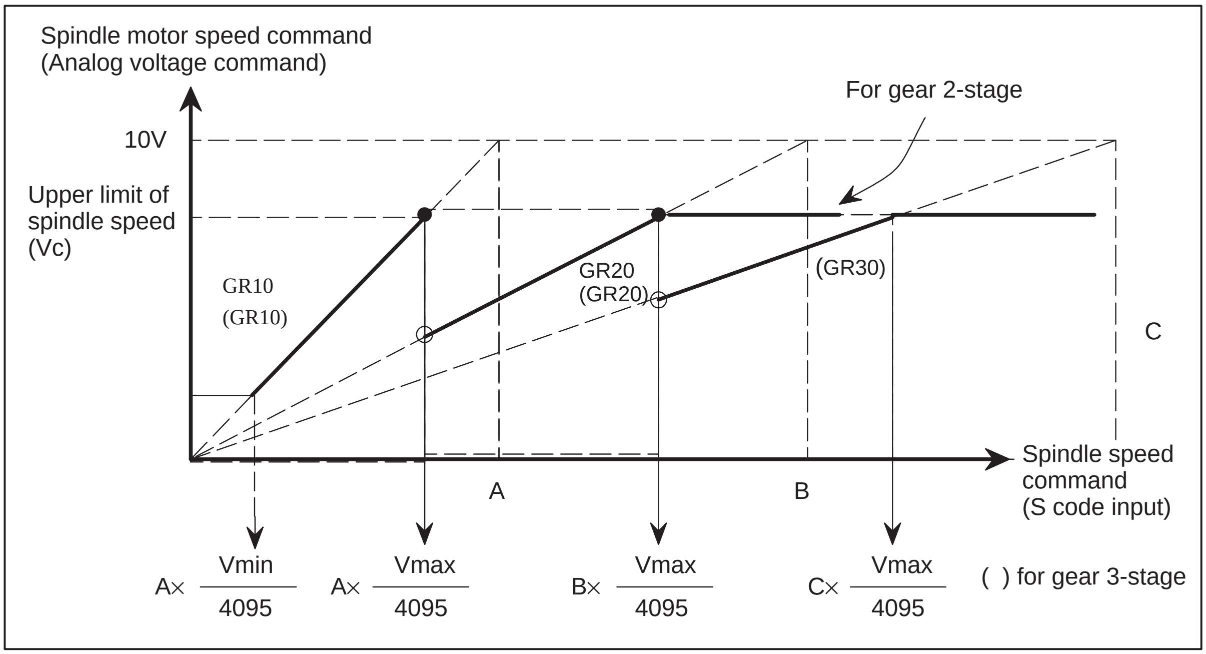


Fig. 9.3 (a) S code input and output voltage

Set the following values as parameters:

· Constant Vmax: for upper limit of spindle speed (parameter No. 3736)

$$\text{Vmax}=4095\times\frac{\text{Upper limit of spindle speed}}{\text{Spindle speed at command voltage 10V}}$$

· Constant Vmin; for lower limit of spindle speed (parameter No. 3735)

$$\text{Vmin}=4095\times\frac{\text{Lower limit of spindle speed}}{\text{Spindle speed at command voltage 10V}}$$

· Spindle speed A (rpm) ; at command voltage 10V and low gear (parameter no. 3741)

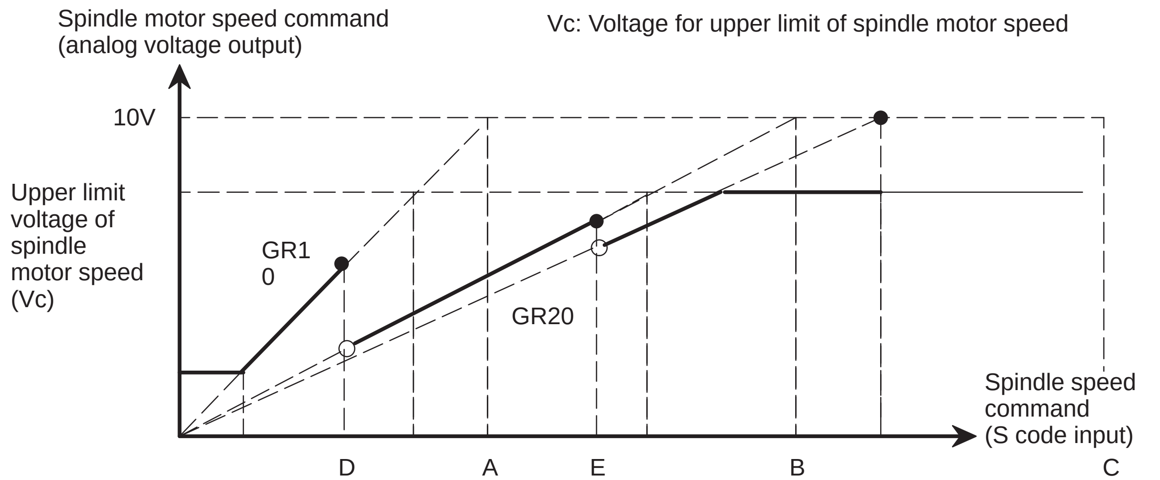
· Spindle speed B (rpm) ; at command voltage 10V and high gear (or middle-high gear) (parameter no. 3742)

· Spindle speed C (rpm) ; 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 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 (G30, G20, G10) are output to the spindle motor as shown in Fig. 9.3.1

- 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. 9.3 (b)).



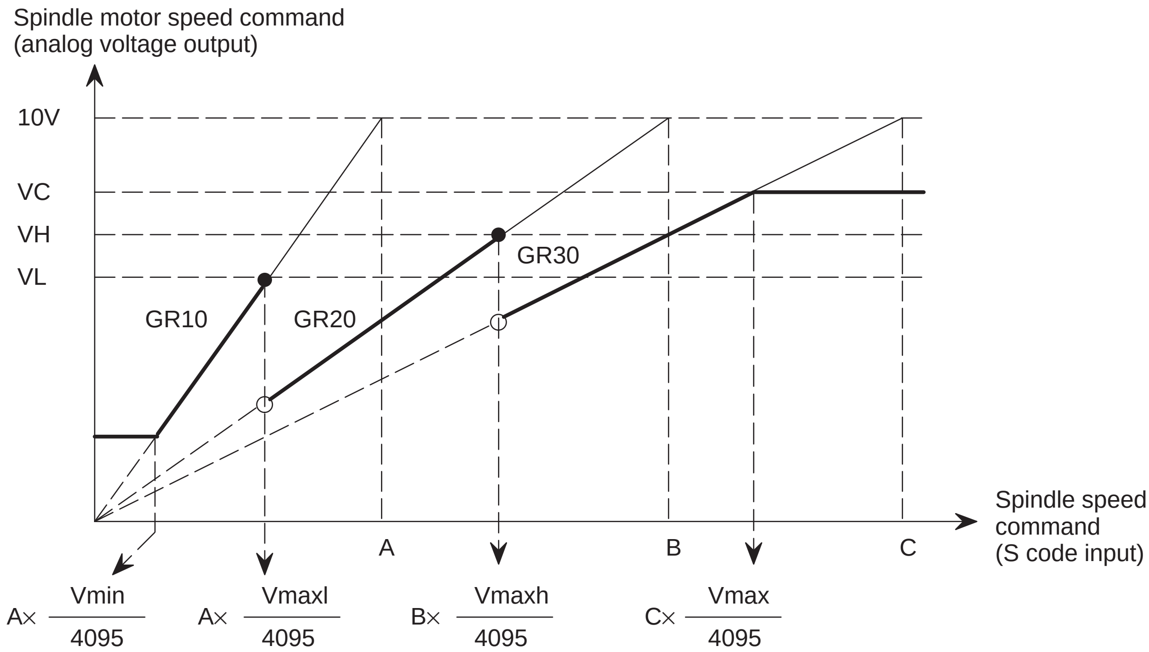
D : Gear shift point (parameter No. 3761) for low/high speed change (or low/medium speed change) (set by parameter as rpm value) during tapping.

E : Gear shift point (parameter No. 3762) (3-stage gear) for medium/high speed change (set by parameter as rpm value) during tapping.

Fig. 9.3 (b) S code input and output voltage (in tapping)

- M type gear change method B (M series) (Fig. 9.3 (c))

The speed (rpm) at which the low-speed and the high-speed gears are changed over can be set as a parameter (nos.3751, 3752) by setting parameter SGB no. 3705# 2. When a 3-step gear is used, it is possible to set the speeds (rpm) for switching low-speed and medium-speed gears, and medium-speed and high-speed gears, using parameters nos. 3751, 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-step gear)

VL: Voltage corresponding to the upper limit of output value to spindle motor at low-speed gear

Fig. 9.3 (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 (rpm)

$$\text{Vmax} = 4095\times\frac{\text{Upper limit of spindle motor speed (rpm)}}{\text{Spindle motor speed (rpm) when the command voltage is 10V}}$$

· Constant Vmin (Parameter No. 3735) related to the lower limit of spindle motor speed (rpm)

$$\text{Vmin} = 4095\times\frac{\text{Lower limit of spindle motor speed (rpm)}}{\text{Spindle motor speed (rpm) when the command voltage is 10V}}$$

· Constant Vmaxl (Parameter No. 3751) related to the upper limit of spindle motor speed (rpm) with low-speed gears

$$\text{Vmaxl} = 4095\times\frac{\text{Upper limit of spindle motor speed (rpm) with low-speed gears}}{\text{Spindle motor speed (rpm) when the command voltage is 10V}}$$

· Constant Vmaxh (Parameter No. 3752) related to the upper limit of spindle motor speed (rpm) with high-speed gears (medium-speed gear for 3-step gear)

$$\text{Vmaxh} = 4095\times\frac{\text{Upper limit of spindle motor speed (rpm) with high-speed gears (medium-speed gear for 3-step gear)}}{\text{Spindle motor speed (rpm) when the command voltage is 10V}}$$

· Spindle speed A (Parameter No.3741) (rpm) with low-speed gears when the command voltage is 10V

· Spindle speed B (Parameter No.3742) (rpm) with high-speed gears when the command voltage is 10V (medium-speed gear for 3-step)

· Spindle speed C (Parameter No.3743) (rpm) with high-speed gears when the command voltage is 10V (3-step gear)

Spindle motor speed commands (0 to 10V) and gear selecting signals (GR10, GR20, GR30) are issued on each S code command as shown in the figure:

Notes
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=1 (No. 3705#2)), when only one-step gear is used, the voltage corresponding to the upper limit value to the spindle motor is calculated using Vmaxl, and when 2-step gear is used, it is calculated according to Vmaxh. Therefore, when SGB is 1, set Vmaxl when only one-step gear is used, Vmaxl and Vmaxh when 2-step gear is used.

- **Time chart**

When S code is commanded, the I/O signal time chart is :

- When Gear select signal does not change

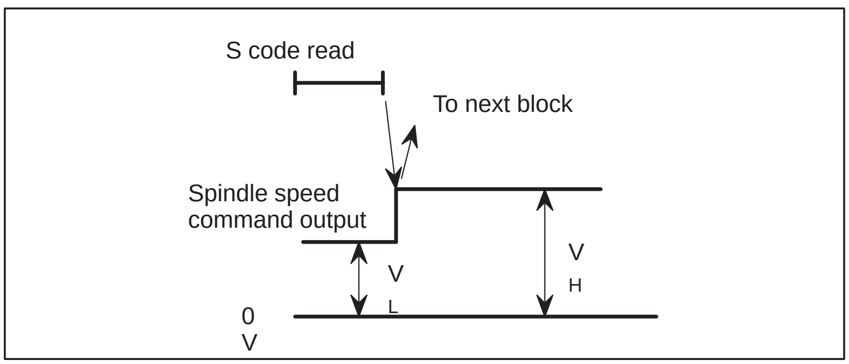


Fig. 9.3 (d) Time chart when gear select signal does not change

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.

- When Gear select signal change

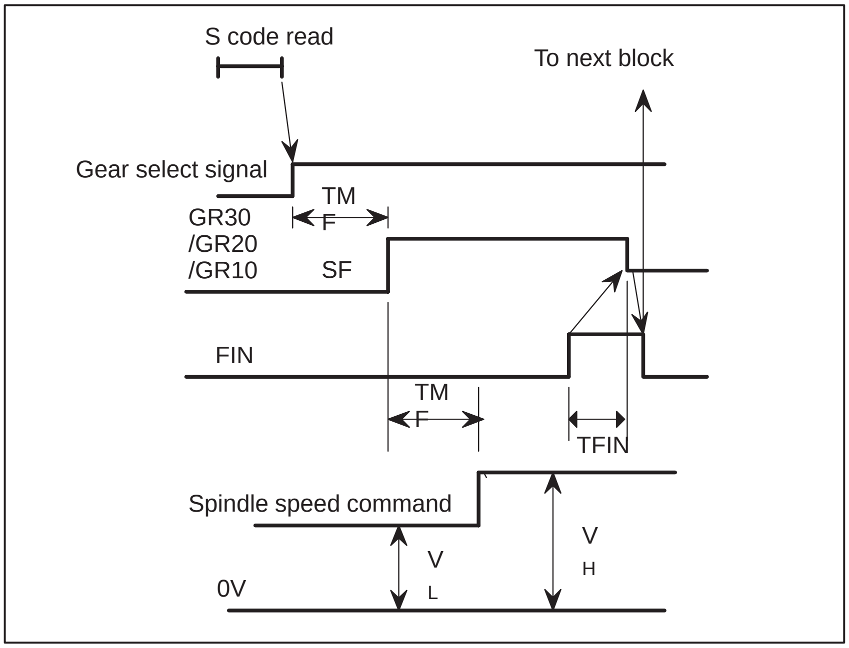


Fig. 9.3 (e) Time chart when gear select signal changes

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 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)**

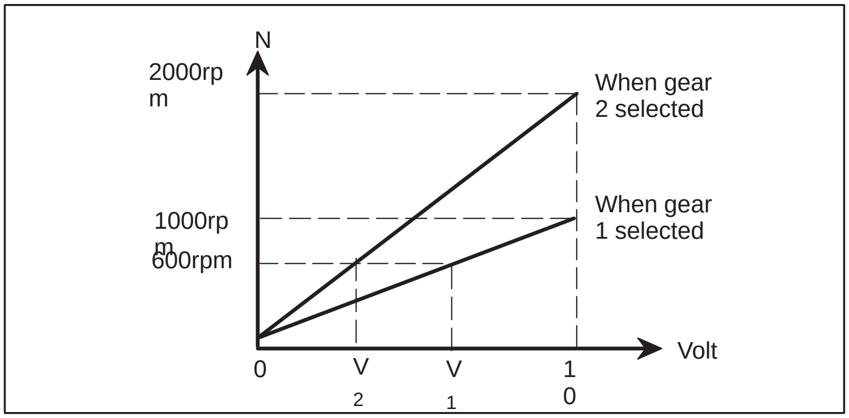
To perform the T type gear changing, the maximum spindle speed for each gear select signal issued from the PMC side must be set by parameter nos. 3741-3744.
The gear select signal is a 2 bit code signal (GR1, GR2). The relationship between the signal and gear number is :

GR1	GR2	Gear No.	Parameter No. for max. spindle speed
0	0	1	No. 3741
1	0	2	No. 3742
0	1	3	No. 3743
1	1	4	No. 3744

The following descriptions exemplify 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 rpm for the low speed gear (G1) and 2000 rpm for the high speed gear (G2), set these speeds by the parameter nos. 3741, 3742. In this case, the analog voltage has the linear relationship shown below.



When spindle speed S=600 is given, V_1 (for G1) or V_2 (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:

$$V= \frac{10N}{R}$$

R: Spindle speed at 10V output voltage

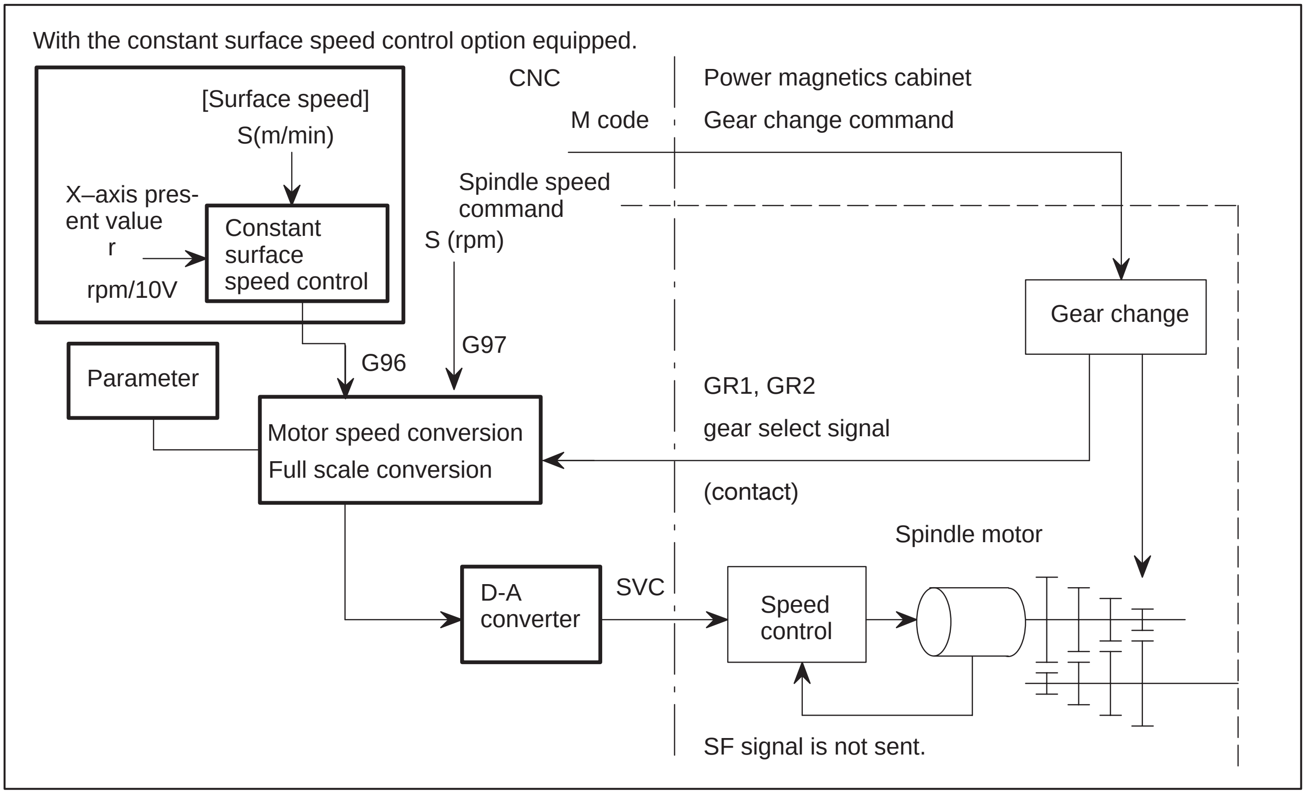
N: Spindle speed given by S5-digits

This is equivalent to the G97 mode for constant surface speed control.

See Section 9.5 for operations during the constant surface speed control mode (G96).

In addition, parameter No. 3772 (upper limit to the spindle speed) can specify speed clamping for all gear positions.

Reference→ Block Diagram for Analog Voltage Output



- **Determination of 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.

For either serial spindle or analog spindle control, the calculation results are output as the S12 bit code signal from 0 to 4095 to the PMC.

(R12O to R01O<F037#3 to F036#0>)

After the calculation results are received, the SIND signal <G033#7> determines which is to be used, the speed command output calculated by the CNC or the data specified in the PMC. Thus speed command output control for the spindle motor is determined. (See also Section 10.4.)

- **Determination of output polarity SSIN/SGN (Input)**

The speed command output to the spindle motor is determined as described above, but the actually used output polarity is determined by the CNC as follows:

· If bit 7 (TCW) of parameter No. 3706 = 0
→ Determined according to bit 6 (CWM) of parameter No. 3706

· If bit 7 (TCW) of parameter No. 3706 = 1
→ Determined according to bit 6 (CWM) of parameter No. 3706 and M03/M04 given to the CNC

After that, the SSIN signal <G033#6> determines which is to be used, the output polarity calculated by the CNC or the polarity specified in the PMC. In this way, the output polarity of the speed command output to the spindle motor is determined. (See also Section 10.4.)

Keep in mind the following: Even with bit 7 (TCW) of parameter No. 3706 = 1, the CNC cannot determine the output polarity if it has not issued M03/M04, and therefore, actual output does not work even if the speed command output has been specified.

- **Command output to spindle**

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

- **Requirement of output**

After power is switched on, a nonzero command is output to the spindle only when the following conditions are met: A nonzero spindle speed command is specified, and the output polarity is determined.

With bit 7 (TCW) of parameter No. 3706 = 1, no command output is sent to the spindle, because the output polarity is not determined until an M03/M04 is issued.

- **Requirement to stop output**

The command output to the spindle is reset to 0 when a command to specify so (such as SSTP = 0 or S0 command) is issued.

M05, emergency stop, or reset does not cause the CNC to reset the command output to the spindle to 0.

- **Spindle enable signal**

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.

- **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.

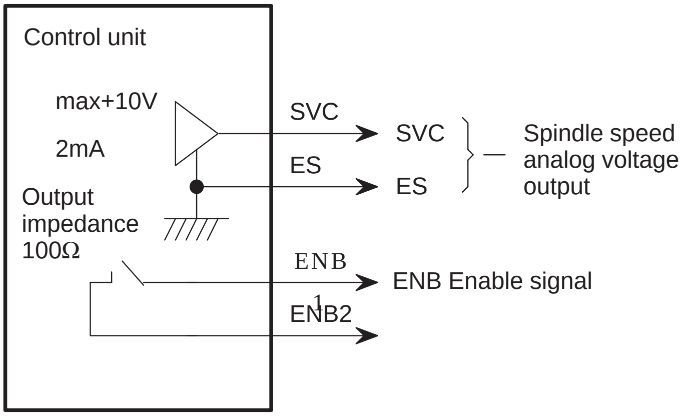
· Analog spindle as the first spindle
→ Gain adjustment data: Parameter No. 3730
Offset voltage compensation: Parameter No. 3731

· Analog spindle as the third spindle
→ Gain adjustment data: Parameter No. 3820
(valid for multispindle control)
Offset voltage compensation: Parameter No. 3821

- **Electrical specification of analog spindle interface**

The signals related to analog spindle interface are described below.

The ENB1 and ENB2 signals are turned on and off under the same condition as for the ENB signal <F001#4>. They can be used also for the serial spindle.



Name	Connector/Pin
SVC	JA8A/7
ES	JA8A/5
ENB1	JA8A/8
ENB2	JA8A/9

> **Note**
> 1 Since the output voltage is a weak signal, do not relay it through contacts.

- **Position coder feedback signal**

The position coder is necessary for thread cutting or feed per rotation. (For the M series, a software option must also be purchased.)

The position coder detects the actual spindle speed and the one–rotation signal (used to detect a fixed point on the spindle for thread cutting).

Ideally, the position coder should be connected directly to the spindle (with a gear ratio of 1:1). If it is necessary to use a gear, select a gear ratio from 1:1, 1:2, 1:4, and 1:8 that reduces the position coder speed.

When using a gear between the spindle and position coder, specify the gear ratio in bits 1 and 0 (PG2, PG1) of parameter No. 3706.

See Section 9.11 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 to cause 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 using the PC ladder regularly, 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. 06 (SPINDLE SPEED ARRIVAL CHECK) on the diagnosis screen is kept at 1 while this function is keeping the cutting feed block at a halt.

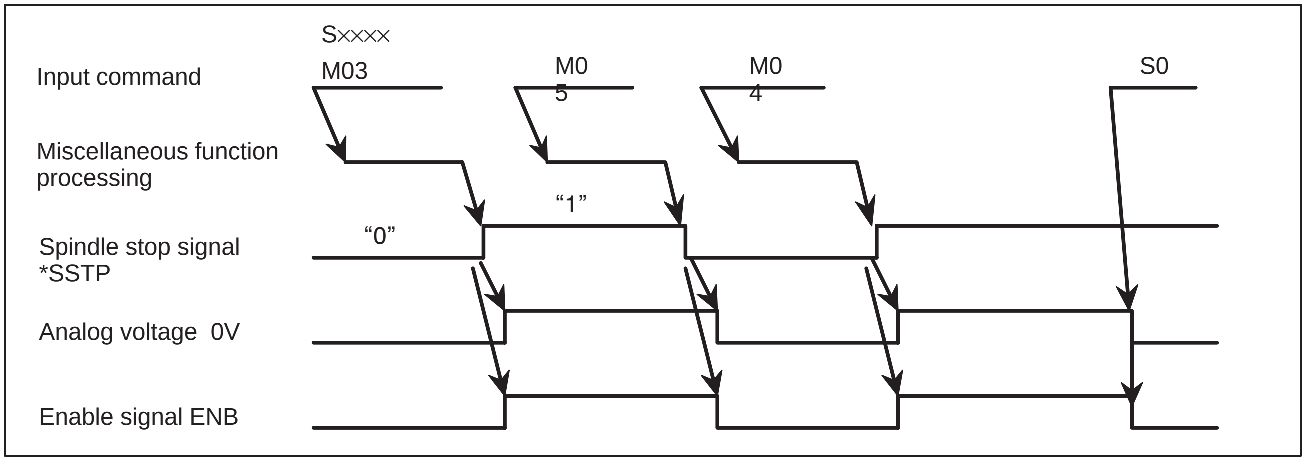
Signal

Spindle stop signal *SSTP<G029#6>

[Classification] Input signal

[Function] The command output to the spindle is held.

[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".

· M03, M04, M05 are not processed inside the CNC.

Spindle orientation signal SOR <G029#5>

[Classification] Input signal

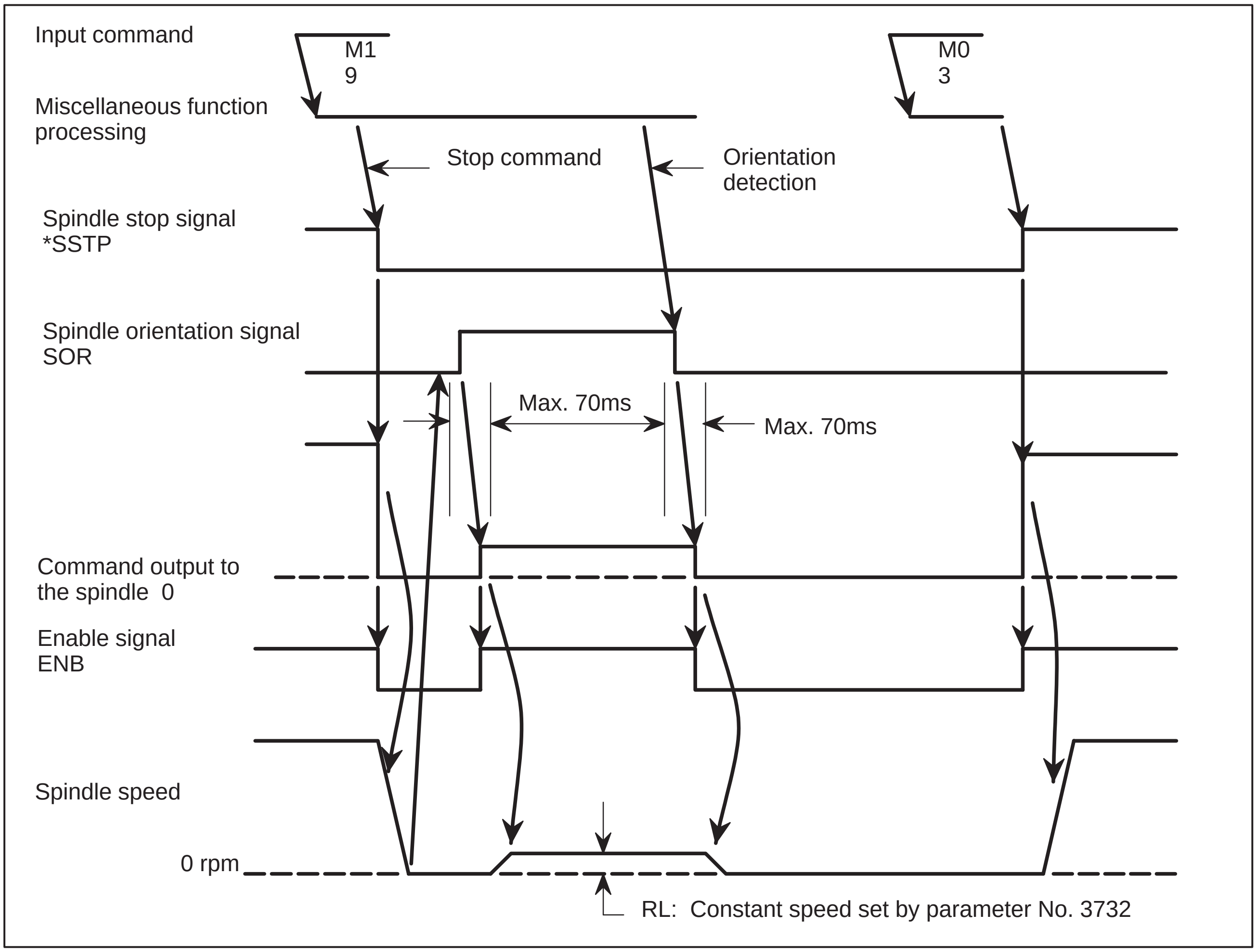
[Function] The spindle or the spindle motor is rotated at a constant speed.

[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".

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 signal SOV0 to SOV7 <G030>

[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 (G84, G74), M series
· Thread cutting (G32, G92, G78), T series

→ 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 <G029#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.

Note

1 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 gated with 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 <F001#4>

[Classification] Output signal

[Function] Informs absence or presence of spindle output command.

[Output condition] 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 (JA8A) 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 signal GR1O, GR2O, GR3O <F034#0 to #2>

[Classification] Output signal

[Function] The gear select signal specifies a gear stage to the PMC.

[Output condition] For details of this signal, see descriptions on the M type gear selection method in General.

Gear selection signal GR1, GR2 <G028#1, #2>

[Classification] Input signal

[Function] This signal informs the CNC of the gear stage currently selected.

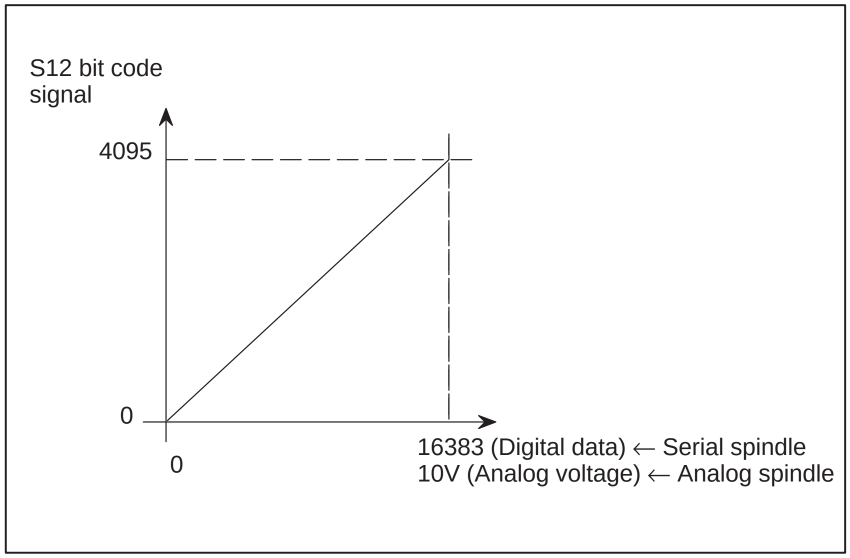
[Output condition] For details of this signal, see descriptions on the T type gear selection method in General.

S12–bit code signal R01O, R12O <F036#0 to F037#3>

[Classification] Output signal

[Function] This signal converts the spindle speed command value calculated by the CNC to code signals 0 to 4095.

[Output condition] The relationship between the spindle speed command value (calculated by the CNC) and the value output by this signal is as shown below.



This signal converts 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. Note that the conversion result is not the actual output value. (See Section 10.4.)

Other signals

Spindle speed function code signal S00 to S31 <F025 to F022> (Output) Spindle speed function strobe signal SF<F007#2> (Output)

See Sections 9.1 and 10.4 for these signals.

Spindle speed output control signal by PMC SIND<G033#7> (Input) R01I to R12I <G032#0 to G033#3> (Input) SSIN <G033#6> (Input) SGN <G033#5> (Input)

See Section 10.4 for these signals.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G027	CON		*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
G028						GR2	GR1	
G029		*SSTP	SOR	SAR				

G030	SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0
G032	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
G033	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
F001				ENB				
F007						SF		
F022	S07	S06	S05	S04	S03	S02	S01	S00
F023	S15	S14	S13	S12	S11	S10	S09	S08
F024	S23	S22	S21	S20	S19	S18	S17	S16
F025	S31	S30	S29	S28	S27	S26	S25	S24
F036	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F037					R12O	R11O	R10O	R09O

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS				ESF
		SFA	NSF		SGT	SGB	GST	ESF

[Data type] Bit

ESF When the spindle control function (S analog outpu or S serial output) is used, and the constant surface speed control function is used or bit 7 (GTT) of parameter No. 3705 is set to 1:
0 : S codes and SF are output for all S commands.
1 : S codes and SF are not output for an S command in constant surface speed control mode (G96 mode) or for an S command used to specify maximum spindle speed clamping (G50S––;).

Note 1 For the T series, this parametar is enabled when bit 4 (EVS) of parameter No. 3705 is set to 1.
For the M series, SF is not output:
(1) For an S command used to specify maximum spindle speed clamping (G92S––;) in constant surface speed control mode
(2) When bit 5 (NSF) of parameter No. 3705 is set to 1

GST: The SOR signal is used for:
0 : Spindle orientation
1 : Gear shift

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.)

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).

EVS When the spindle control function (S analog output or S serial output) is used, S codes and SF are:
0 : Not output for an S command.
1 : Output for an S command.

Note 1 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.

NSF: When an S code command is issued in constant surface speed control,
0 : SF is output.
1 : SF is not output.

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				PG2	PG1
	TCW	CWM	ORM	GTT			PG2	PG1

[Data type] Bit

PG2 and PG1: Gear ratio of spindle to position coder

Magnification	PG2	PG1
×1	0	0
×2	0	1
×4	1	0
×8	1	1

$$\text{Magnification} = \frac{\text{Spindle speed}}{\text{Number of position coder revolutions}}$$

GTT Selection of a spindle gear selection method
0: Type M
1 : Type T

Note 1 Type M:
The gear selection signal is not entered. In response to an S command, the CNC selects a gear according to the speed range of each gear specified beforehand in parameters. Then the CNC reports the selection of a gear by outputting the gear selection signal. The spindle speed corresponding to the gear selected by the gear selection signal is output.
Type T:
The gear selection signal is entered. The spindle speed corresponding to the gear selected by this signal is output.

Note 2 When the constant surface speed control option is selected, type T is selected, regardless of whether this parameter is specified.

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#3 SGT,
No. 3761, No. 3762, No. 3705#6 SFA, No. 3735, No. 3736
However, parameter No. 3744 is valid.

ORM Voltage polarity during spindle orientation
0 : Positive
1 : Negative

TCW, CWM 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
3709								SAM

[Data type] Bit

SAM The sampling frequency to obtain the average spindle speed
0 : 4 (Normally, set to 0.)
1 : 1

3730	Data used for adjusting the gain of the analog output of spindle speed

[Data type] Word

[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.

$$\text{Set value} = \frac{10\ (\text{V})}{\text{Measured data (V)}} \times 1000$$

(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 1 This parameter needs not to be set for serial spindles.

3731	Compensation value for the offset voltage of the analog output of the spindle speed

[Data type] Word

[Unit of data] Velo

[Valid data range] –1024 to+1024

Set compesation value for the offset voltage of the analog output of the spindle speed.
Set value =–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.

$$\text{Set value} = \frac{-8191 \times \text{Offset voltage (V)}}{12.5}$$

(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 1 This parameter need not to be set for serial spindles.

3732	The spindle speed during spindle orientation or the spindle motor speed during spindle gear shift

[Data type] Two–word

[Valid data range] 0 to 20000

Set the spindle speed during spindle orientation or the spindle motor speed during gear shift.

When GST, #1 of parameter 3705, is set to 0, set the spindle speed during spindle orientation in rpm.

When GST, #1 of parameter 3705, is set to 1, set the spindle motor speed during spindle gear shift calculated from the following formula.

$$\text{Set value} = \frac{\text{Spindle motor speed during spindle gear shift}}{\text{Maximum spindle motor speed}} \times 16383 \text{ (For a serial spindle)}$$

$$\text{Set value} = \frac{\text{Spindle motor speed during spindle gear shift}}{\text{Maximum spindle motor speed}} \times 4095 \text{ (For an analog spindle)}$$

3735	Minimum clamp speed of the spindle motor

[Data type] Word

[Valid data range] 0 to 4095

Set the minimum clamp speed of the spindle motor.

$$\text{Set value} = \frac{\text{Minimum clamp speed of the spindle motor}}{\text{Maximum spindle motor speed}} \times 4095$$

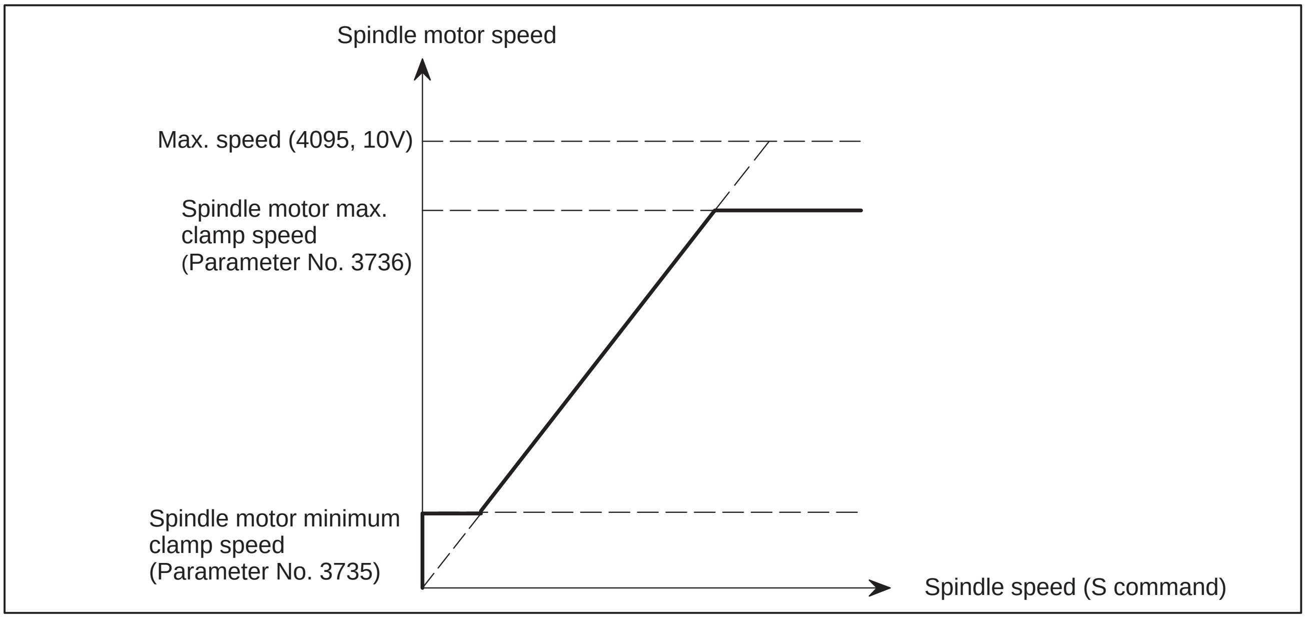
3736	Maximum clamp speed of the spindle motor

[Data type] Word

[Valid data range] 0 to 4095

Set the maximum clamp speed of the spindle motor.

$$\text{Set value} = \frac{\text{Maximum clamp speed of the spindle motor}}{\text{Maximum spindle motor speed}} \times 4095$$



3740	Time elapsed prior to checking the spindle speed arrival signal

[Data type] Byte

[Unit of data] msec

[Valid data range] 0 to 225

Set the time elapsed from the execution of the S function up to the checking of the spindle speed arrival signal.

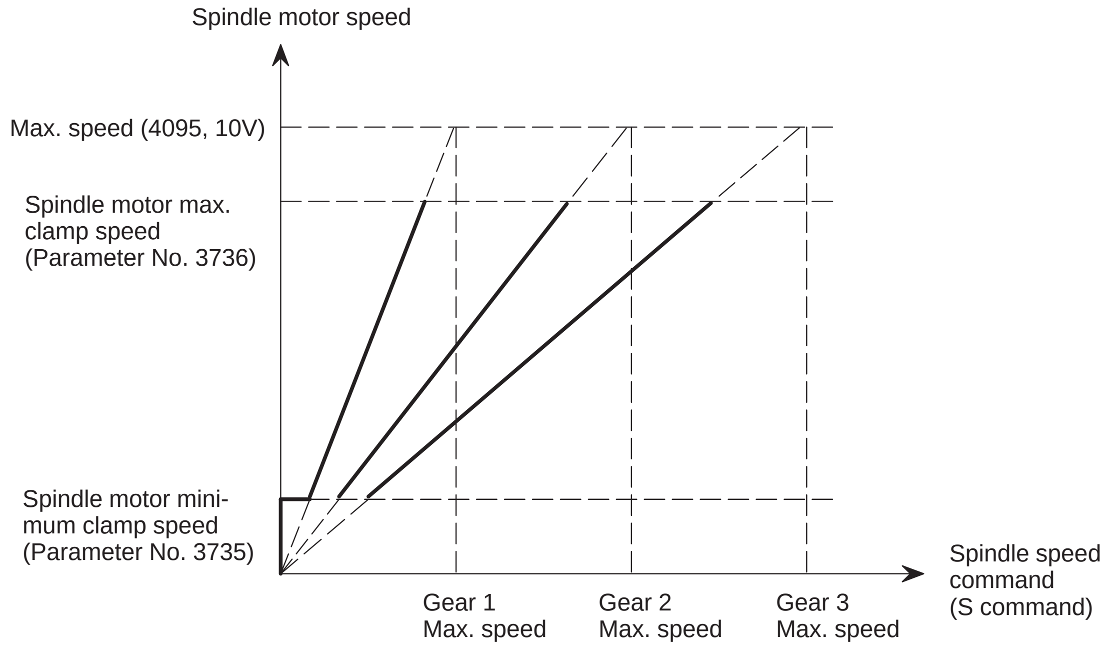
3741	Maximum spindle speed for gear 1
3742	Maximum spindle speed for gear 2
3743	Maximum spindle speed for gear 3
3744	Maximum spindle speed for gear 4

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

Set the maximum spindle speed corresponding to each gear.



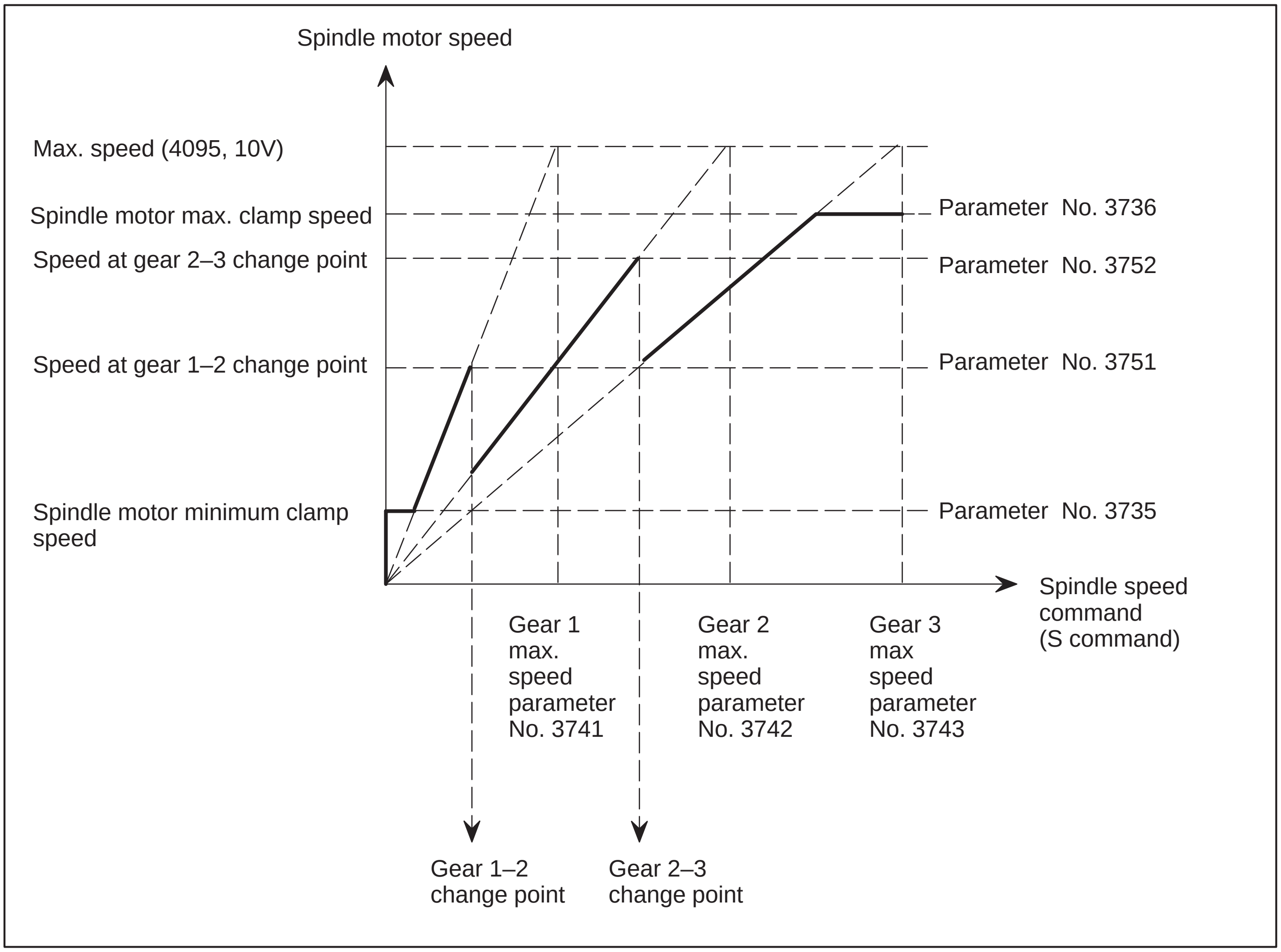
3751	Spindle motor speed when switching from gear 1 to gear 2
3752	Spindle motor speed when switching from gear 1 to gear 3

[Data type] Word

[Valid data range] 0 to 4095

For gear switching method B, set the spindle motor speed when the gears are switched.

$$\text{Set value} = \frac{\text{Spindle motor speed when the gears are switched}}{\text{Maximum spindle motor speed}} \times 4095$$



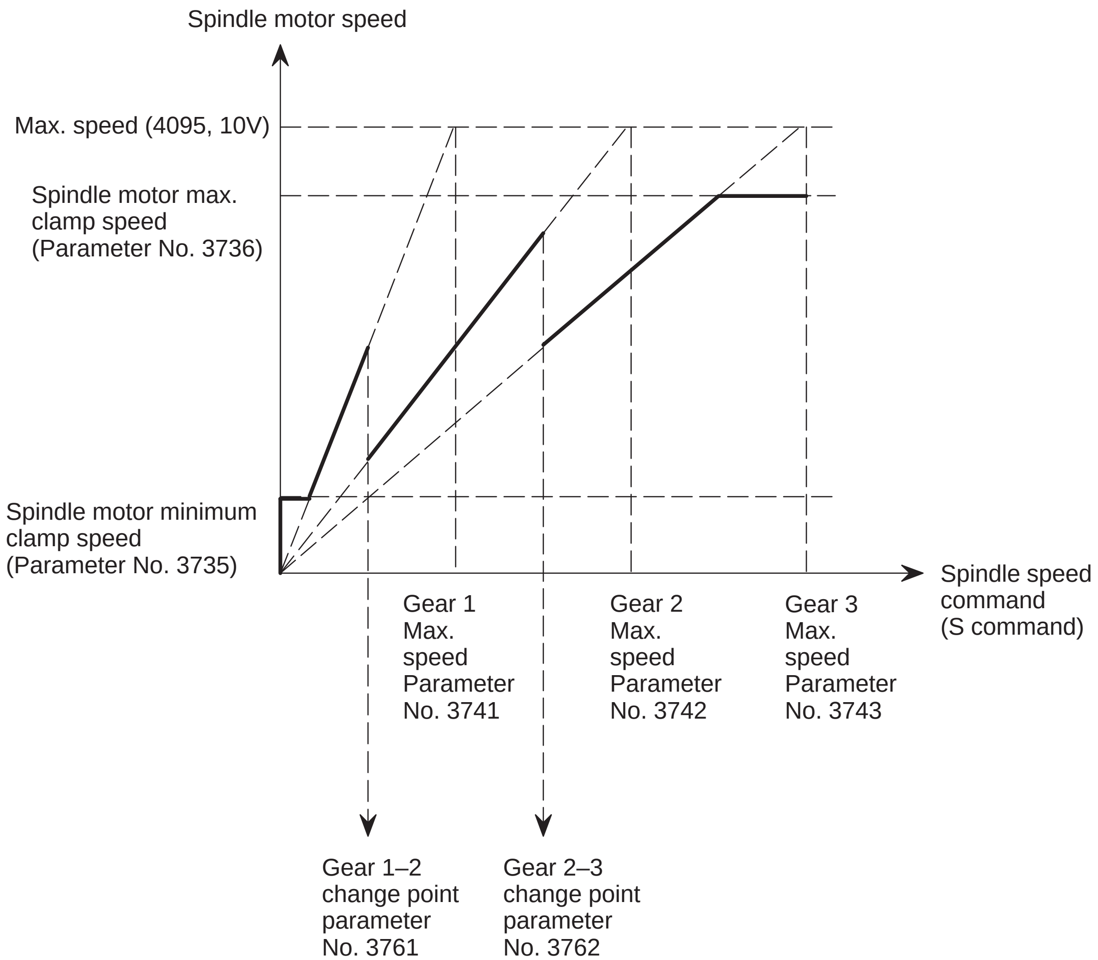
3761		Spindle speed when switching from gear 1 to gear 2 during tapping
3762		Spindle speed when switching from gear 2 to gear 3 during tapping

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

When method B is selected (SGT,#3 of parameter 3705, is set to 1) for the tapping cycle gear switching method, set the spindle speed when the gears are switched.



3772	Maximum spindle speed

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

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.

Note 1 In the M series, this parameter is valid when the constant surface speed control option is selected.

Note 2 When the constant surface speed control option is selected, the spindle speed is clamped at the maximum speed, regardless of whether the G96 mode or G97 mode is specified.

Note 3 When 0 is set in this parameter, the speed of the spindle is not clamped.

Note 4 When spindle speed command control is applied using the PMC, this parameter has no effect, and the spindle speed is not clamped.

Note 5 When the multi–spindle control option is selected (T series), set the maximum speed for each spindle in the following parameters:
Parameter No. 3772: Sets the maximum speed for the first spindle.
Parameter No. 3802: Sets the maximum speed for the second spindle.
Parameter No. 3822: Sets the maximum speed for the third spindle.

3821	Offset–voltage compensation value of the analog output of the third–spindle speed

[Data type] Word

[Unit of data] Velo

[Valid data range] –1024 to 1024

Set the offset–voltage compensation value of the analog output of the third–spindle speed.

Note 1 This parameter is used for controlling the multi–spindles.

Note

Note 1 This section mentioned a spindle speed control that should be prepared on the CNC side. But it is also necessary to design the signals to the spindle control unit.

Consult the manual of the spindle control unit used and take necessary actions on the spindle control unit.

9.4 SPINDLE SPEED CONTROL FOR TWO–PATH LATHE

General

In a two–path lathe application, the additional path section (path No. 2) can have the same spindle interface as a one–path lathe (see Section 9.2.).

Each spindle is controlled by a command issued by tool post 1 or 2. Which spindle is controlled by which tool post can be switched by signals.

This section describes the configuration of a two–path lathe application and how it is controlled.

In the following description, the term tool post 1 refers to path No. 1, and the term tool post 2 refers to path No. 2.

One–spindle control and two–spindle control

In a two–path lathe application, there are two selectable configurations, a configuration in which the entire system uses one spindle (one–spindle control) and a configuration in which each spindle is controlled separately (two–spindle control). Parameter No. 3703 (2SP) is used to select a configuration.

One–spindle control

The spindle interface for tool post 2 is not used.

- **Selection of the spindle command**

The spindle command select signal SLSPA<G063#2> (input) specifies the tool post whose spindle command is to be followed by the spindle.

- **Position coder feedback signal (serial spindle)**

When a serial spindle is used, the position coder feedback signal is supplied to both paths in the NC. Either tool post can be used for thread cutting and feed per rotation.

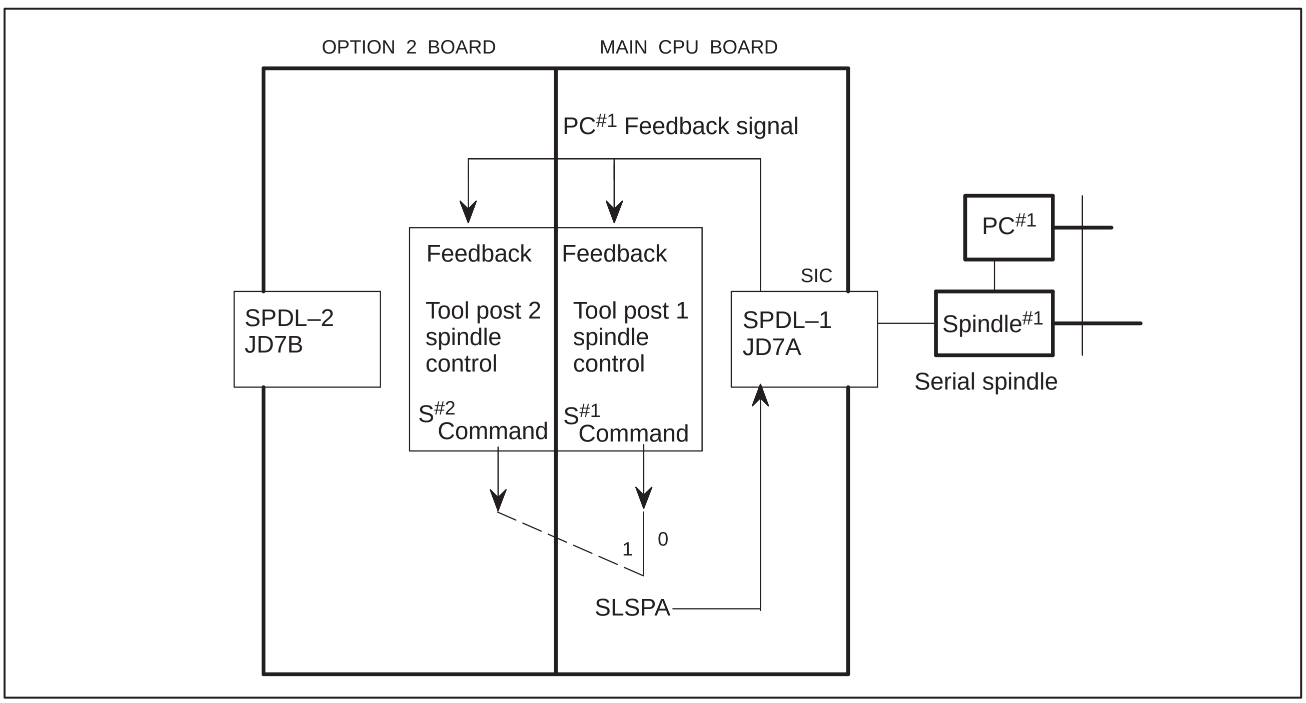


Fig. 9.4 (a) One spindle control (Serial spindle)

- **Position coder feedback signal (Analog)**

When an analog spindle is used, supplying the position coder feedback signal to the position coder interface of tool post 2 via an external distribution circuit makes it possible to use either tool post for thread cutting and feed per rotation.

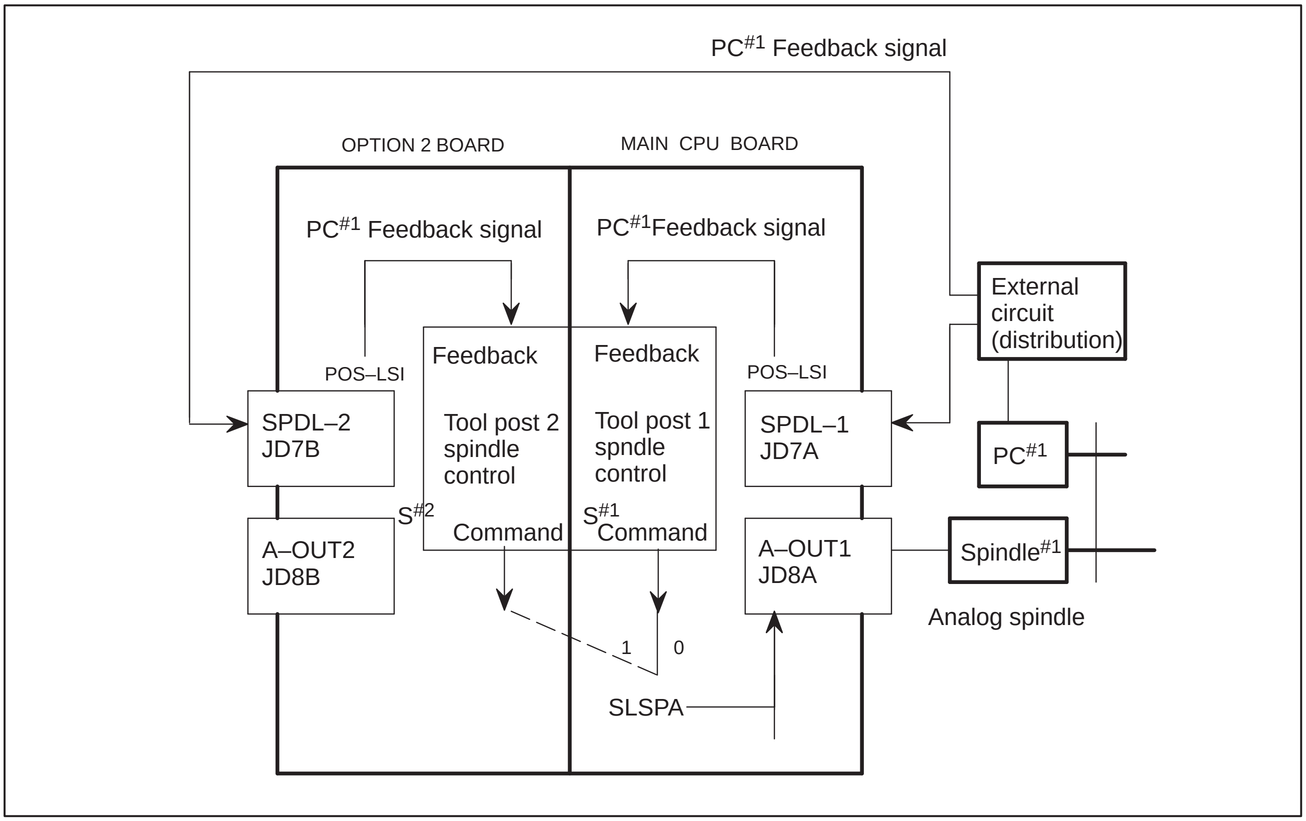


Fig. 9.4 (b) One spindle control (Analog spindle)

Two–spindle control

The spindle interface for either tool post is used.

- **Selection of spindle command**

The spindle command select signals SLSPA <G063#2> and SLSPB <G063#2> (input) specify the tool post whose spindle command is to be followed by each spindle.

- **Position coder feedback signal (When both are serial spindle)**

When the serial spindles are used on both tool posts, the position coder feedback signals SLPCA<G064#2> and SLPCB<G064#3> (input) direct each tool post to select which spindle's position coder feedback signal is used. Therefore, it is possible to use the spindle of the other system; for example, tool post 1 can perform thread cutting or feed per rotation using the spindle connected to tool post 2.

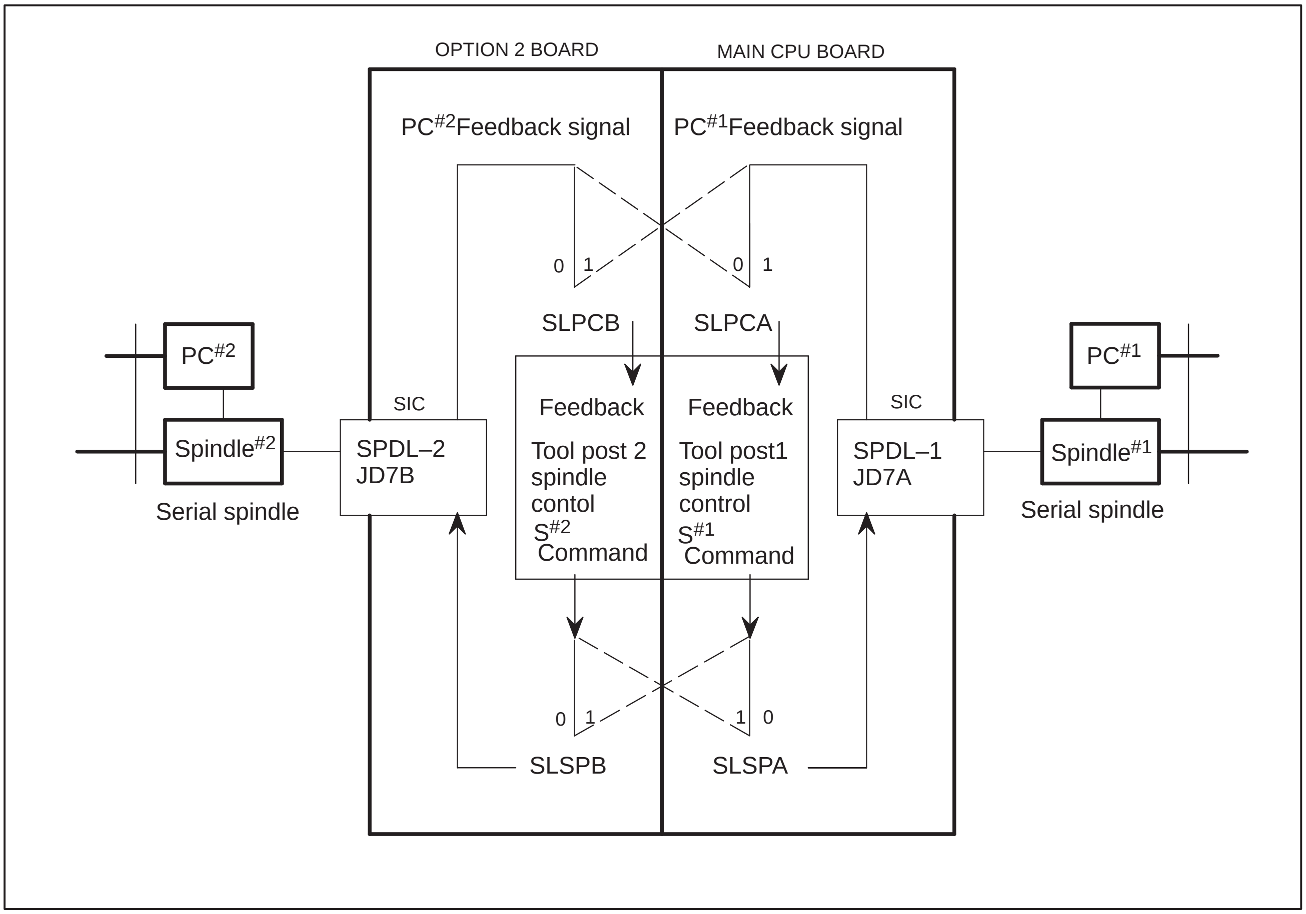


Fig. 9.4 (c) Two–spindle Control (Serial spindle)

- **Position coder feedback signal (when an analog spindle is used)**

If either tool post uses an analog spindle as the first spindle, the spindle feedback signals SLPCA<G064#2> and SLPCB<G064#3> (input) cannot cause the NC to select a position coder feedback signal.

If both tool posts use an analog spindle, switching the position coder feedback signal inputs to the NC using an external circuit makes it possible to use the spindle of the other system.

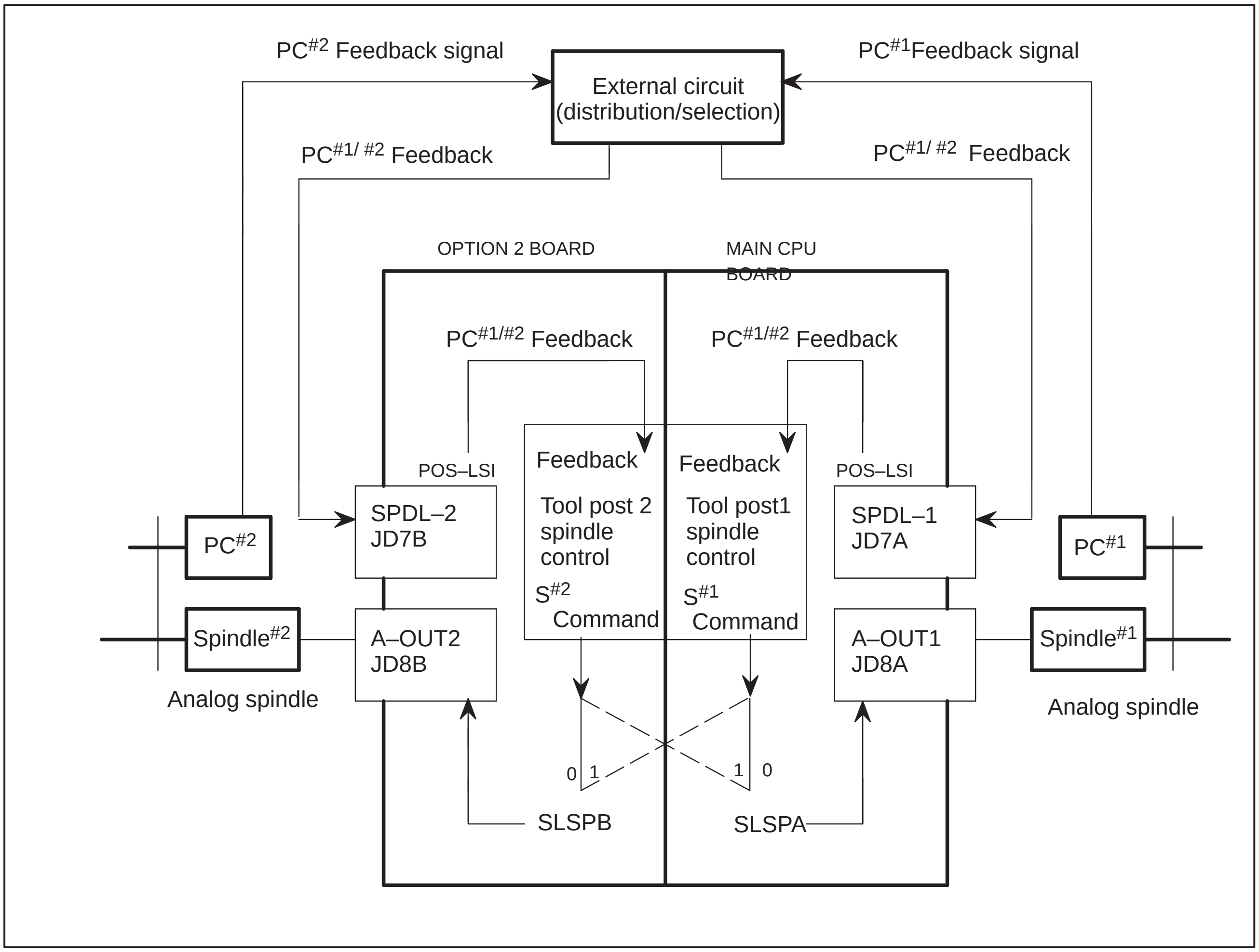


Fig. 9.4 (d) Two–spindle control (analog spindle)

2nd and 3rd spindles

If the first spindle is a serial spindle, the second and third spindles can also be used in a two–path lathe application. (See Section 9.2.)

In the following chart, all spindles are connected under two–spindle control.

Under one–spindle control, any spindle ($SP1^{\#2}$, $SP2^{\#2}$, $SP3^{\#2}$) of tool post 2 cannot be used.

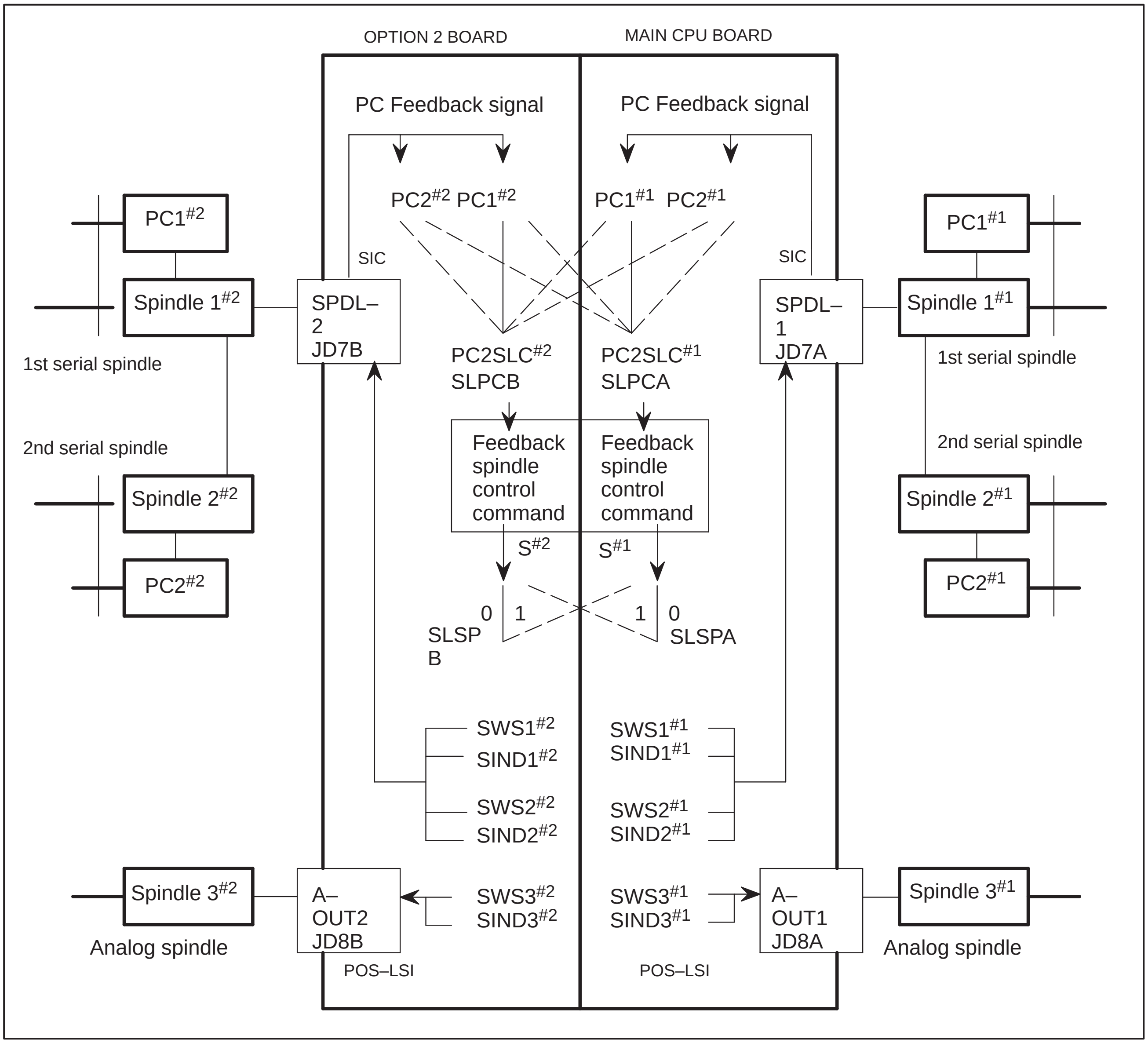


Fig. 9.4 (e) Spindle Configuration and Flow of Commands and Feedback Signals in an two–path lathe application (with All Spindles under Two–Spindle Control)

The second and third spindles should be controlled using the PMC or be under multi–spindle control. (See Section 15.4 or 9.19.)

If multispindle control is applied to both tool posts, the position coder feedback signal for the second spindle of each tool post also becomes usable.

See descriptions of bit 3 (PCS) of parameter No. 3706.

Options related to spindles

Optional functions for spindles are valid for both tool posts. However, you may want to use the optional functions for only one of the tool posts because of relationships with the interface and PMC ladder.

Parameters are available to disable the following functions for individual tool posts.

· Spindle serial output

· Spindle analog output

· Cs contour control

· Spindle positioning

· Multi–spindle control

Refer to parameter No. 3702.

Signal

Spindle command select signals SLSPA <G063#2>, SLSPB <G063#3>

[Classification] Input signal

[Function] Selects which tool post receives spindle command of which spindle.

SLSPA: Selects the spindle command for spindle connected to tool post 1.

SLSPB: Selects the spindle command for spindle connected to tool post 2.

The spindle command select signals are associated with the spindle commands as follows:

(1) In the 1-spindle control mode

Signal input SLSPA	Command to spindle
0	Spindle command of tool post 1
1	Spindle command of tool post 2

Note 1 SLSPB is ineffective.

(2) In the 2-spindle control mode

Signal input		Command to the spindle connected to tool post 1	Command to the spindle connected to tool post 2
SLPCA	SLPCB		
0	0	Spindle command of tool post 1	Spindle command of tool post 2
0	1	Spindle command of tool post 1	Spindle command of tool post 1
1	0	Spindle command of tool post 2	Spindle command of tool post 2
1	1	Spindle command of tool post 2	Spindle command of tool post 1

Spindle feedback select signals SLPCA <G064#2> SLPCB <G064#3>

[Classification] Input signal

[Function] Selects which spindle sends the feedback signal of the position coder to which tool post.

Note 1 This function is effective only in the 2-spindle control system in which both tool posts use serial spindle.

SLPCA: Selects the feedback signal for tool post 1.
SLPCB: Selects the feedback signal for tool post 2.

The spindle feedback select signals are associated with the feedback signals of the position coder as follows:

(1) In 1-spindle control mode
The feedback signal of the spindle connected to tool post 1 is always sent to both tool posts; the SLPCA and SLPCB signals have no meaning.

(2) In 2-spindle control mode

Signal input		Tool post 1	Tool post 2
SLPCA	SLPCB		
0	0	PC#1	PC#2
0	1	PC#1	PC#1
1	0	PC#2	PC#2
1	1	PC#2	PC#1

PC#1=Position coder feedback signal for the spindle connected to tool post 1.

PC#2=Position coder feedback signal for the spindle connected to tool post 2.

Note 1 The SLPCA and SLPCB signals are effective only in the 2-spindle control mode using two serial spindles. In the 2-spindle control mode using analog spindles, the feedback signal of spindle 1 is input to tool post 1, and the feedback signal of spindle 2 is input to tool post 2, regardless of the setting of the SLPCA and SLPCB signals.

Spindle command signal COSP <F064#5>

[Classification] Output signal

[Function] Indicates which tool post issued the latest spindle command.

[Output condition] The COSP signal turns to “1” when:

· Tool post 2 issues the spindle command.

The COSP signal turns to “0” when:

· Tool post 1 issues the spindle command, or neither of the tool posts issues the spindle command.

[Use] In 1-spindle control mode, if this signal is input to the SLSPA signal (spindle command select signal), the spindle speed specified by the latest spindle command can always be output to the spindle, regardless of whether it is from tool post 1 or 2.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G063					SLSPB	SLSPA		
G064					SLPCB	SLPCA		
F064			COSP					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3702	ECS	ESS	EAS	ESI			EMS	

[Data type] Bit

EMS Multi–spindle control function
0 : Used
1 : Not used

Note 1 If the multi–spindle control function is not required for one tool post in two–path control, specify this parameter for the tool post to which the multi–spindle control function need not be applied.

ESI The spindle positioning function is
0 : Used
1 : Not used

Note 1 This parameter is used when the spindle positioning option specified with two–path control, and the spindle positioning function is not required for either path. Set ESI to 1 for a system that does not require the spindle positioning function.

EAS For tool post 1 (or tool post 2), the S analog output function is:
0 : Used.
1 : Not used.

ESS For tool post 1 (or tool post 2), the S serial output function is:
0 : Used.
1 : Not used.

ECS For tool post 1 (or tool post 2), the Cs contour control function is:
0 : Used.
1 : Not used.

Note 1 Parameter EAS, ESS, and ECS are used for 16–TB 2–path control. These parameters are used to determine whether the optional function, S analog output function, S serial output function, and Cs contour control function, are used for each tool post.

	#7	#6	#5	#4	#3	#2	#1	#0
3703								2SP

[Data type] Bit

2SP Specifies whether one or two spindles are controlled (16–TB 2–path control).

0 : One spindle (two tool posts)

1 : Two spindle (two tool posts)

	#7	#6	#5	#4	#3	#2	#1	#0
3706					PCS			

[Data type] Bit

PCS When multi–spindle control is applied to two tool posts in two–path control, this parameter specifies whether a position coder feedback signal from the other tool post is selectable, regardless of the state of the PC2SLC signal (bit 7 of G028/bit 7 of G1028) of the other tool post:

0 : Not selectable.

1 : Selectable. (To select a position coder for the other tool post, the SLPCA signal (bit 2 of G064) and the SLPCB signal (bit 3 of G064) are used.)

Note 1 Multi–spindle control based on the same serial spindle must be applied to both tool posts.

· Selecting position coder feedback signals for both tool posts in a two–path lathe application under multi–spindle control.

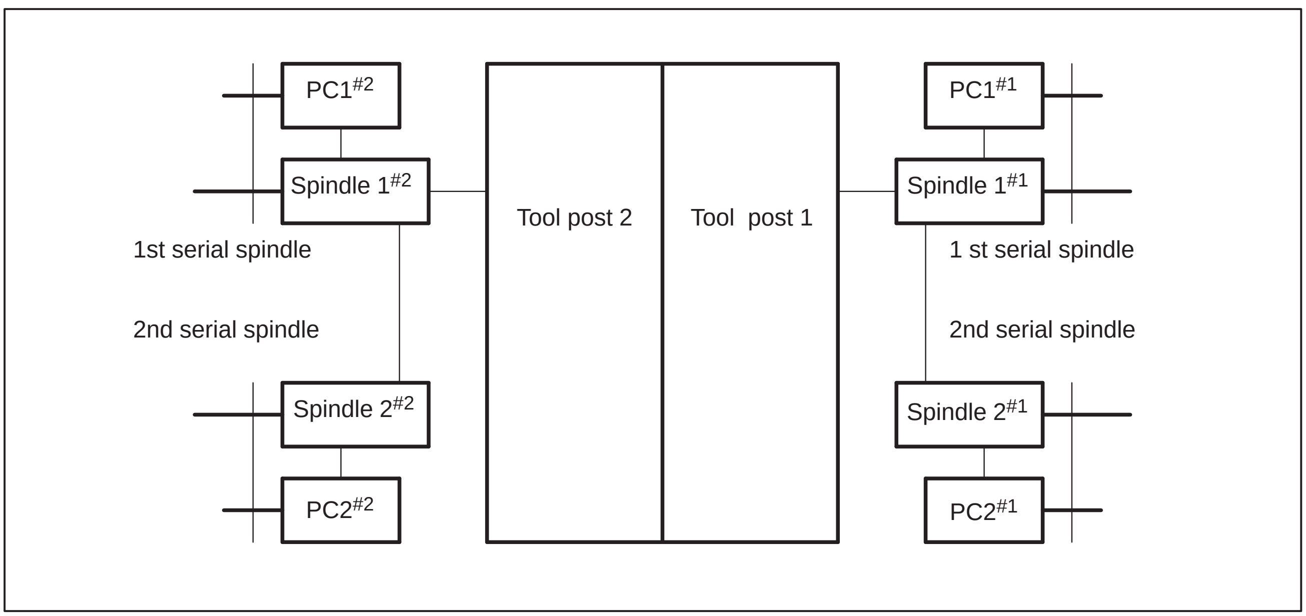


Table 9.4 lists the position coder feedback signals used for each tool post in the above configuration. These position coder feedback signals are selected according to the following:

· Bit 3 (PCS) of parameter No. 3706

· Spindle feedback select signals SLPCA <G064#2> and SLPCB <G064#3>

· Multi–spindle control
Second position coder select signals PC2SLC#1 <G028#7> and PC2SLC#2 <G1028#7>

When parameter No. 3706#3=1

Table 9.4 Selection of Position Coder Feedback Signal in Two–path Lathe
(— means position coder selection is indifferent on the tool post side)

	Position coder selected	SLPCA	SLPCB	PC2SLC$^{\#1}$	PC2SLC$^{\#2}$
Tool post 1	PC1$^{\#1}$	"0"	—	"0"	—
	PC2$^{\#1}$	"0"	—	"1"	—
	PC1$^{\#2}$	"1"	—	"0"	—
	PC2$^{\#2}$	"1"	—	"1"	—
Tool post 2	PC1$^{\#1}$	—	"1"	—	"0"
	PC2$^{\#1}$	—	"1"	—	"1"
	PC1$^{\#2}$	—	"0"	—	"0"
	PC2$^{\#2}$	—	"0"	—	"1"

When parameter No. 3706#3=0

	Position coder selected	SLPCA	SLPCB	PC2SLC$^{\#1}$	PC2SLC$^{\#2}$
Tool post 1	PC1$^{\#1}$	"0"	—	"0"	—
	PC2$^{\#1}$	"0"	—	"1"	—
	PC1$^{\#2}$	"1"	—	—	"0"
	PC2$^{\#2}$	"1"	—	—	"1"
Tool post 2	PC1$^{\#1}$	—	"1"	"0"	—
	PC2$^{\#1}$	—	"1"	"1"	—
	PC1$^{\#2}$	—	"0"	—	"0"
	PC2$^{\#2}$	—	"0"	—	"1"

Note

Note 1 The spindle commands include S code commands, maximum speed command (G50S__), M03, M04, M05, and constant surface speed control commands (G96 and G97)

Note 2 Signals to operate the spindle control unit are not affected by the spindle command select signals SLSPA<G063#2> or SLSPB<G063#3>. They may be processed in the PMC ladder, as required.

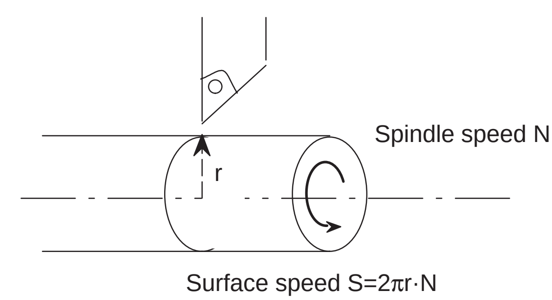
(Example: G070#5SFRA is always a forward rotation command for the first spindle control amplifier of tool post 1.)

Note 3 The machine tool builder should prepare an external circuit to distribute and select position coder feedback signals for the analog spindle.

9.5 CONSTANT SURFACE SPEED CONTROL

General

With the spindle serial output or analog output function, specifying the surface speed (m/min or feet/min) directly in an S command makes it possible to change the spindle output continuously so as to maintain a constant surface speed at a programmed point. (For the rapid traverse command, however, the surface speed for the end point is output at the beginning of rapid traverse.)



Whether or not constant surface speed control is performed is selected by G code.

G96: Constant surface speed control performed. S in the G96 mode is m/min or feet/min.

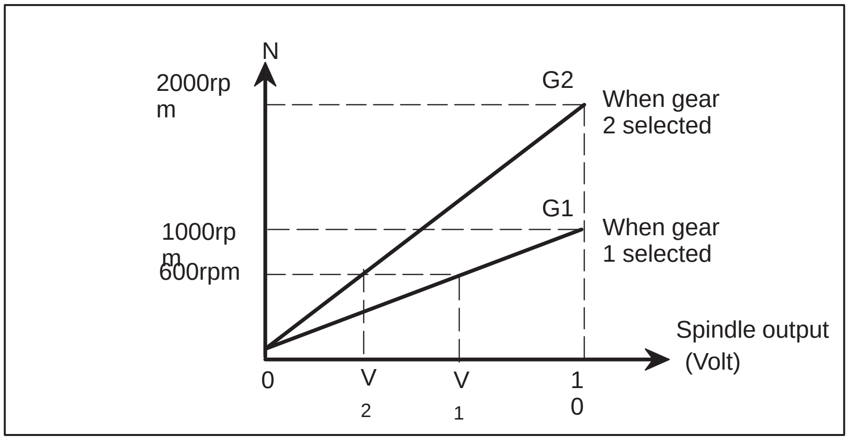
G97: Constant surface speed control not performed. S in the G97 mode is rev/min.

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 nos. 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

Example of Spindle Analog Output

Assume that gear switching is two stage switching. If the spindle speed with the output 10 V is 1000 rpm for the low speed gear (G1) and 2000 rpm for the high speed gear (G2), set these speeds to the parameter nos. 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 rpm ($S = 2\pi r N$). Therefore, V_1 (for G1) or V_2 (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

$$V = \frac{10S}{2\pi rR}$$

R: Spindle speed (rpm) 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

$$V = \frac{10N}{R}$$

R: Spindle speed at 10V output voltage (rpm)

N: Spindle speed given by S command (rpm)

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

$$D = \frac{4095S}{2\pi rR}$$

R: Spindle speed (rpm) 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

$$D= \frac{4095N}{R}$$

R: Spindle speed at maximum spindle motor speed (rpm)
N: Spindle speed given by S command (rpm)

Signal

Gear selection signal (Input) GR2, GR1 <G028#2, #1>

Refer to 9.3 “Spindle Control”.

Constant surface speed signal CSS <F002#2>

[Classification] Output signal

[Function] This signal indicates that constant surface speed control is in progress.

[Output condition] “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
F002						CSS		

Parameter

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

[Data type] Word

[Unit of data] rpm

3770	
	Axis as the calculation reference in constant surface speed control

[Data type] Byte

[Valid data range] 1, 2, 3, ..., number of control axes

Set the axis as the calculation reference in constant surface speed control.

3771	Minimum spindle speed in constant surface speed control mode (G96)

[Data type] Word

[Unit of data] rpm

[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

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

This parameter sets the maximum spindle speed.
When a command specifying a speed exceeding the maximum spindle speed is specified, or the spindle speed exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

Note 1 In the M series, this parameter is valid when the constant surface speed control option is selected.

Note 2 When the constant surface speed control option 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
190	ILLEGAL AXIS SELECT (M series)	In the constant surface speed control, the axis specification is wrong. (See parameter No. 3770.) The specified axis command (P) contains an illegal value. Correct the program.

Note

Note 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 specified speed.

Note 2 Simultaneous use of multi–spindle control (T series) enables constant surface speed control for spindles other than the first spindle. (See Section 9.10.)

Note 3 If the constant surface speed control function is provided for a machining center system, it affects gear change under normal spindle control. (See Section 9.3.)

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.10.3	CONSTANT SURFACE SPEED CONTROL (G96, G97)
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.10.3	CONSTANT SURFACE SPEED CONTROL (G96, G97)

9.6 SPINDLE SPEED FLUCTUATION DETECTION

General

With this function, an overheat alarm (No. 704) 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.

Detection of Spindle Speed Fluctuation

The function for detecting spindle speed fluctuation checks whether the actual speed varies for the specified speed or not. Sd or Sr, whichever is greater, is taken as the allowable fluctuation speed (Sm). An alarm 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).

Sd: The allowable constant variation width which is independent of the specified spindle speed (Sd is set with parameter 4913.)

Sr: The allowable variation width which is obtained by multiplying Sc (commanded spindle speed) by r (constant ratio). (r is set with parameter 4912.)

Sm: Sd 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:

· The actual spindle speed falls in a range of (Sc – Sq) to (Sc + Sq)

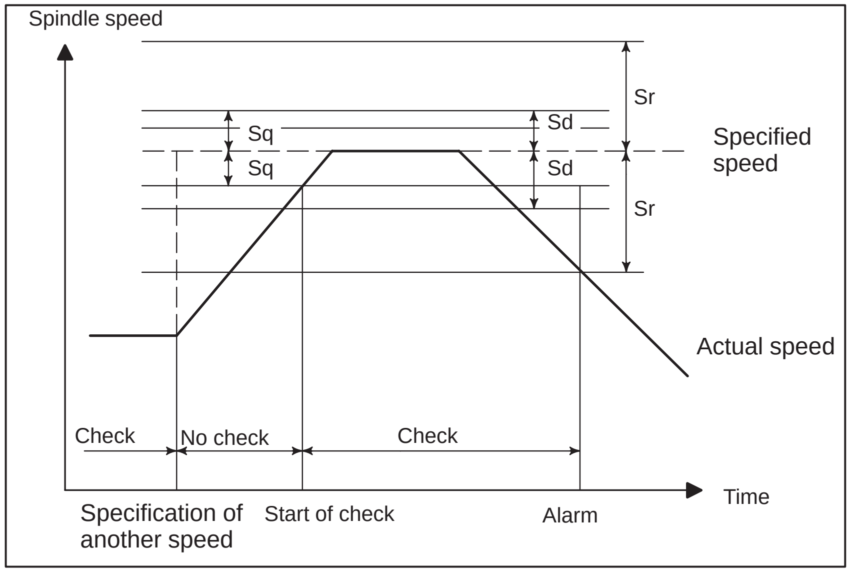
where Sq = Sc × q/100

q: Percent tolerance of the target spindle speed, specified in parameter No. 4911. If the actual spindle speed is in a range of the specified speed ± q, it is assumed that the actual speed has reached the specified speed.

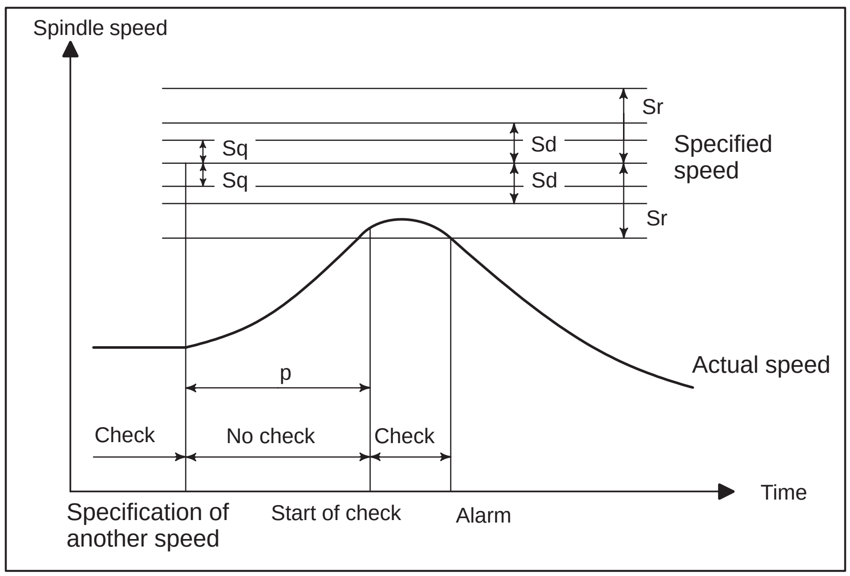
· When time p specified in parameter No. 4914 elapses after the specified speed Sc changes.

Parameter Nos. 4914, 4911, and 4912 can be rewritten also by program (G26 PpQqRr).

1. When an alarm is issued after a specified spindle speed is reached



2. When an alarm 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

Signal

Spindle fluctuation detection alarm signal SPAL <F035#0>

[Classification] Output signal

[Function] This signal indicates that the actual spindle speed is not within a tolerance to the specified speed.

[Output condition] 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 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
F035								SPAL

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3708				SVD				

[Data type] Bit

SVD When the SIND signal is on, the detection of spindle speed fluctuation is:
0 : Disabled
1 : Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
4900								FLR

[Data type] Bit

FLR When the spindle speed fluctuation detection function is used, the rates of allowance (q) and fluctuation (r) those are set in parameter No. 4911 and No. 4912, respectively are set in steps of:
0 : 1%
1 : 0.1%

4911	Ratio (q) of the fluctuation of spindle speed which is assumed to be the specified spindle speed

[Data type] Word

[Unit of data]

[Valid data range]

Unit of data	1%	0. 1% (T series)
Data range	1 – 100	1 – 1000

Note 1 Unit of data depends on parameter No. 4900#0 FLR (T series only)

Set the ratio (q) of the spindle speed which is assumed to be the specified spindle speed in the spindle speed fluctuation detection function.

4912	Spindle speed fluctuation ratio (r) for which no alarm is activated in the spindle speed fluctuation detection function

[Data type] Word

[Unit of data]

[Valid data range]

Unit of data	1%	0. 1% (T series)
Data range	1 – 100	1 – 1000

Note 1 Unit of data depends on parameter No. 4900#0 FLR (T series only).

Set the spindle speed fluctuation ratio (r) for which no alarm is activated in the spindle speed fluctuation detection function.

4913	Spindle speed fluctuation value (d) for which no alarm is activated in the spindle speed fluctuation detection function

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

Set the allowable fluctuation speed (Sd) for which no alarm is activated in the spindle speed fluctuation detection function.

4914	Time (p) elapsed from when the commanded spindle speed is changed to the start of spindle speed fluctuation detection

[Data type] Two–word

[Unit of data] ms

[Valid data range] 0 to 999999

Set the time elapsed from when the specified spindle speed is changed to the start of spindle speed fluctuation detection in the spindle speed fluctuation detection function. That is, the fluctuation in the spindle speed is not detected until the specified time elapses from when the specified spindle speed is changed.

Alarm and message

Number	Message	Description
704	OVERHEAT: SPINDLE	Spindle overheat in the spindle fluctuation detection (1) If the cutting load is heavy, relieve the cutting condition. (2) Check whether the cutting tool is share. (3) Another possible cause is a faulty spindle amp.

Note

Note 1 When an alarm is issued in automatic operation, a single block stop occurs.

Note 2 No check is made during spindle stop state (*SSTP = 0).

Note 3 An alarm is issued one second later if the actual spindle speed is found to be 0 rpm.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.10.4	SPINDLE SPEED FLUCTUATION DETECTION FUNCTION (G25, G26)
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.10.4	SPINDLE SPEED FLUCTUATION DETECTION FUNCTION (G25, G26)

9.7 ACTUAL SPINDLE SPEED OUTPUT

General

The PMC can read actual spindle speed.

Signal

Actual spindle speed signal AR0 to AR15 <F040, F041>

[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.

[Operation] Spindle speed = $\sum_{i=0}^{15}\{2^i \times V_i\}$rpm

where Vi = 0 when ARi is "0" and Vi = 1 when ARi is "1"

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F040	AR7	AR6	AR5	AR4	AR3	AR2	AR1	AR0
F041	AR15	AR14	AR13	AR12	AR11	AR10	AR09	AR08

Note

Note 1 The AR0 - AR15 signals are always output. Their values change every 64 msec.

Note 2 An absolute error of about 0.5 rpm exists as a measuring error.

9.8 SPINDLE POSITIONING (T series)

General

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 with ease because the position detector is a position coder.

Generally, the spindle positioning axes are clamped mechanically except when positioning is under way.

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. The value for the spindle speed is input from the spindle controller to the spindle amplifier.

When the optional spindle positioning function is activated, the spindle is moved to a defined angle, and the workpiece is repositioned at that angle. The specified move distance is input to the error counter, and the velocity command is issued for the spindle motor through the spindle amplifier. The position of the spindle is detected by the installed position coder (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. (This is called spindle orientation.)
- Position the spindle in the spindle positioning mode

  Position an optional angle using address C (H), and 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.

Also, relationship between M codes and these operations are set by parameters (refer to No. 4950#2 (ISZ), #7 (IMB)).

· Least command increment

$$\frac{360}{4096} \doteq 0.088 \text{ deg}$$

· Least input increment

0.001 deg

· Maximum command value

±9999.999 deg

Selecting a spindle positioning axis

Any axis in the control axis group can be used as the C axis (parameter no. 1020). Specify −1 as its servo axis number (parameter no. 1023).

Only one set of this setting can be used for each control path. The spindle subjected to spindle positioning is the first spindle.

Switching to spindle positioning mode (Orientation)

Orientation is required in advance if spindle positioning is first performed after the spindle motor is used as a normal spindle, or when spindle positioning is interrupted. The orientation stops the spindle in a constant position. The orientation position can be sifted in the range of ±180deg for analog spindle and in the range from 0 to 360° for serial spindle.

To specify orientation, use the M code whose parameter no. 4960.
The orientation direction is specified by using parameters ZMIx no. 1006 #5 for analog or RETURN no. 4000#5 for serial spindles.

• Orientation speed

The spindle moves at rapid traverse set by parameter No. 1420 until it reaches the orientation enable speed (shown below). After the spindle crosses that speed point, it performs orientation at the speed set by parameter no. 1425. When a serial spindle is used, orientation speed depends on the spindle.

· Orientation enable speed
RPD>9×(loop gain) KPPM
Loop gain: Parameter no. 4970 (unit: 1/sec)
Set rapid traverse speed at above value.

(Example)

When the loop gain parameter no. 4970 is set to 20 [1/sec], the orientation speed is:

RPD>9×20×1000×(360/4096)=15820 [deg /min]

The serial spindle stops at the orientation position as soon as the command is issued. The lower limit to the rapid traverse speed value does not need to be specified for the serial spindle to reach the orientation enable speed.

The analog spindle stops after the spindle speed is changed from rapid traverse to the FL speed. The rapid traverse speed lower limit must be specified for the analog spindle, or obtaining the orientation enable speed need not be specified for the serial spindle. However, it must be specified for the analog spindle.

• Program origin

The orientation position is regarded as a program origin. It is possible to modify the program origin using the coordinates system or automatic coordinates system settings (parameter ZPR no. 1201#0)

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 by M code

A 2-digit numerical value following the M address is used for the command. There are six positioning angle values (Mα to M(α + 5)), where α is set by parameter no. 4962. Indexing angle β also requires prior parameter setting data no. 4963. Rotation can be done in any direction, using parameter IDM data no. 4950#1.
Also, when extended specification is used (parameter No. 4950#6 ESI=1), max. 256 kinds of values (Mα to M(α+255)) can be specified.

M–code	Indexing angle	eg) when 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°

- **Optional angle positioning by C or H address**

Numerical value following either the C or H address is used to command the position. C and H addresses are commanded in G00 mode.

(Example) C-1000

H45000

(i) Minimum setting unit :

0.001deg

(ii) Maximum command value:

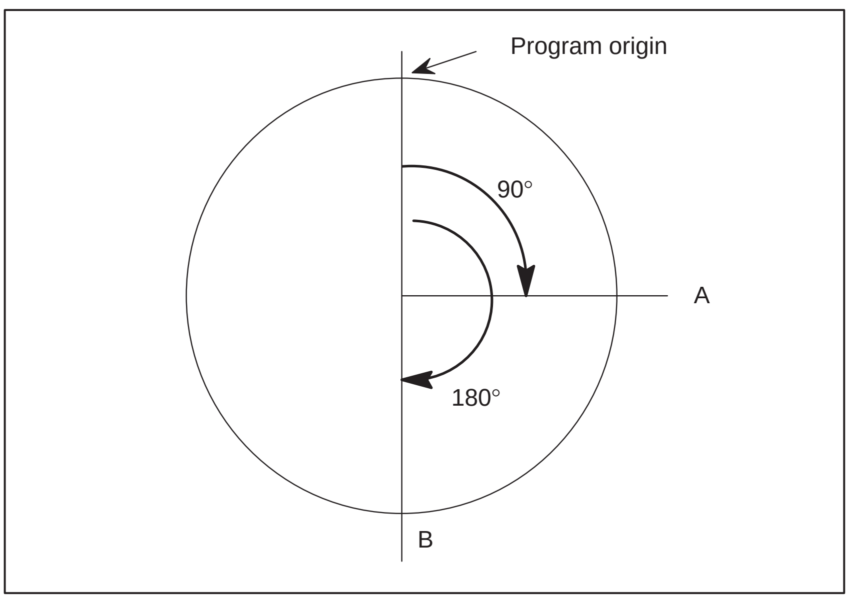
±9999.999 deg

(iii) Decimal point input: A numerical value with decimal point can be entered. The decimal point location is in "degrees", for instance:

C35.0=C35 degrees

- **Absolute and incremental commands**

The semi–fixed angle is always incremental. The optional angle positioning using address C or H differs as follows:



Command method		G code system A		G code system B, C	
		Address used	Command of A–B on the above illustration	Address used and G–code	Command of A–B on the above illustration
Absolute command	Direct the end point position by the distance from the program origin.	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, to which linear acceleration deceleration are applied. 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.

Signal

Spindle stop complete signal (SPSTP) <G028#6>

[Classification] Input signal

[Function] When this signal is 1, the CNC orients and positions the spindle.

Spindle unclamp signal SUCLP <F038#1>

[Classification] Output signal

[Function] This signal specifies that spindle mechanical clamping be released in a spindle positioning sequence.

When this signal is output, unclamp the spindle on the machine (release the brakes or extract the pin).

[Output condition] Refer to the sequence (time chart) in this section.

Spindle unclamp completion signal *SUCPF <G028#4>

[Classification] Input signal

[Function] This signal indicates that unclamping the spindle is complete in response to the spindle unclamp signal SUCLP.

Spindle clamp signal SCLP <F038#0>

[Classification] Output signal

[Function] This signal specifies 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 condition] Refer to the sequence (time chart) in this section.

Spindle clamp completion signal *SCPF <G028#5>

[Classification] Input signal

[Function] This signal indicates that clamping the spindle is complete in response to the spindle clamp signal SCLP.

Other signals

Gear selection signal GR1, GR2, <G028#1, #2>

Refer to 9.3 “Spindle Control.”

CTH1A, CTH2A <G07#3, #2>

Refer to the manual of serial spindle.

The spindle loop gain multiplier corresponding to the gear currently selected by this signal is used. When the serial spindle is used, input gear selection signals CTH1A and CTH2A, as well.

Relationship between the selected gear and spindle gear selection signal

Analog spindle		
GR2	GR1	Selected gear
0	0	1st gear
0	1	2nd gear
1	0	3rd gear
1	1	4th gear

Serial spindle		
CTH1A	CTH2A	Selected gear
0	0	HIGH
0	1	MEDIUM HIGH
1	0	MEDIUM LOW
1	1	LOW

Spindle orientation completion signal ZPx <F094>

[Classification] Output signal

[Function] This signal indicates that the spindle orientation for the spindle positioning has been completed.

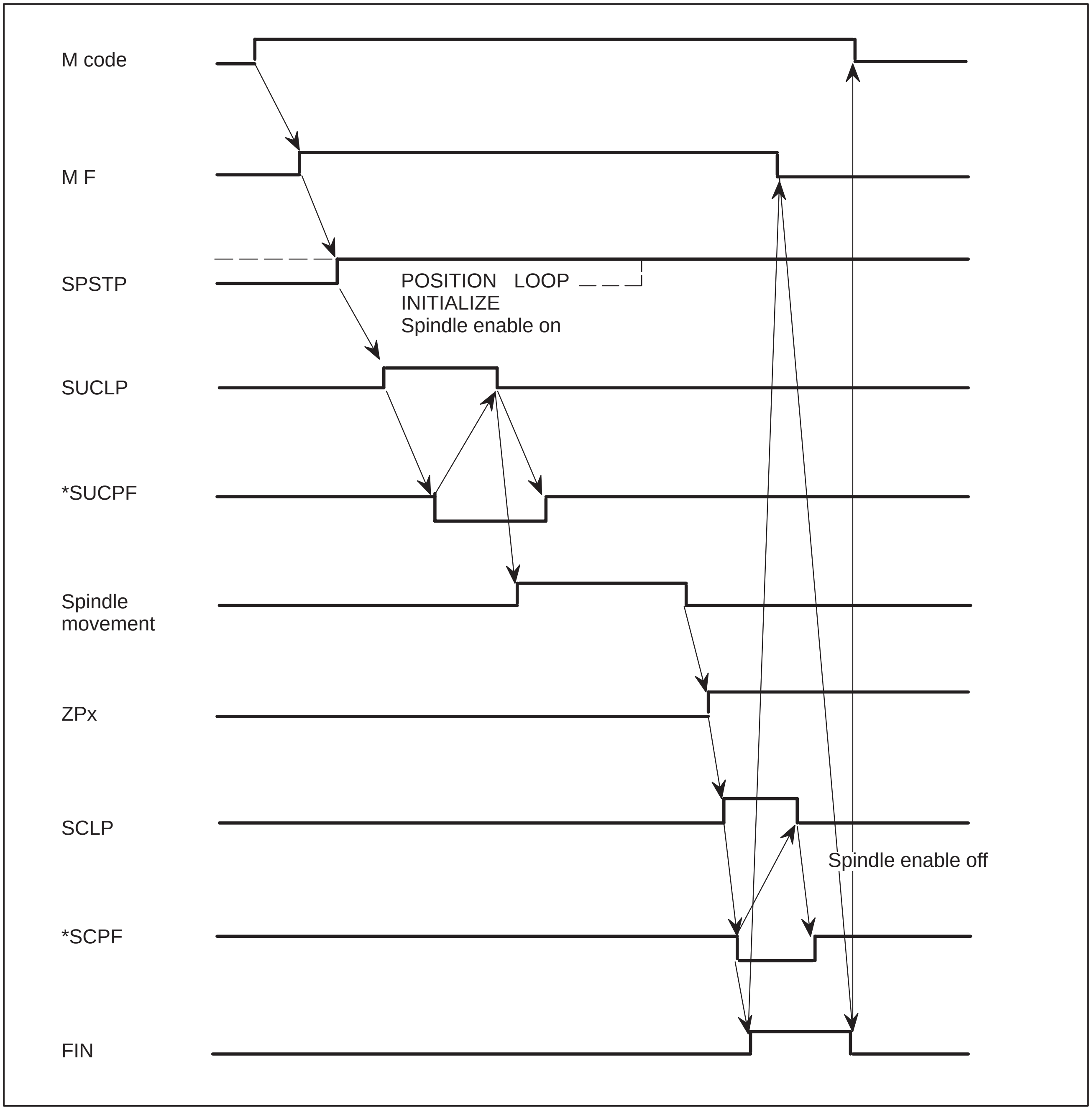
[Output condition] When spindle orientation is complete, this signal turns to 1. When spindle positioning is performed or cleared, it turns to 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G028		SPSTP	*SCPF	*SUCPF		GR2	GR1	
G070					CTH1A	CTH2A		
F038							SUCLP	SCLP
F094	ZP8	ZP7	ZP6	ZP5	ZP4	ZP3	ZP2	ZP1

Sequence (Time chart)

❑ Spindle Orientation



⇒ POSITION LOOP INITIALIZE is performed within the CNC.

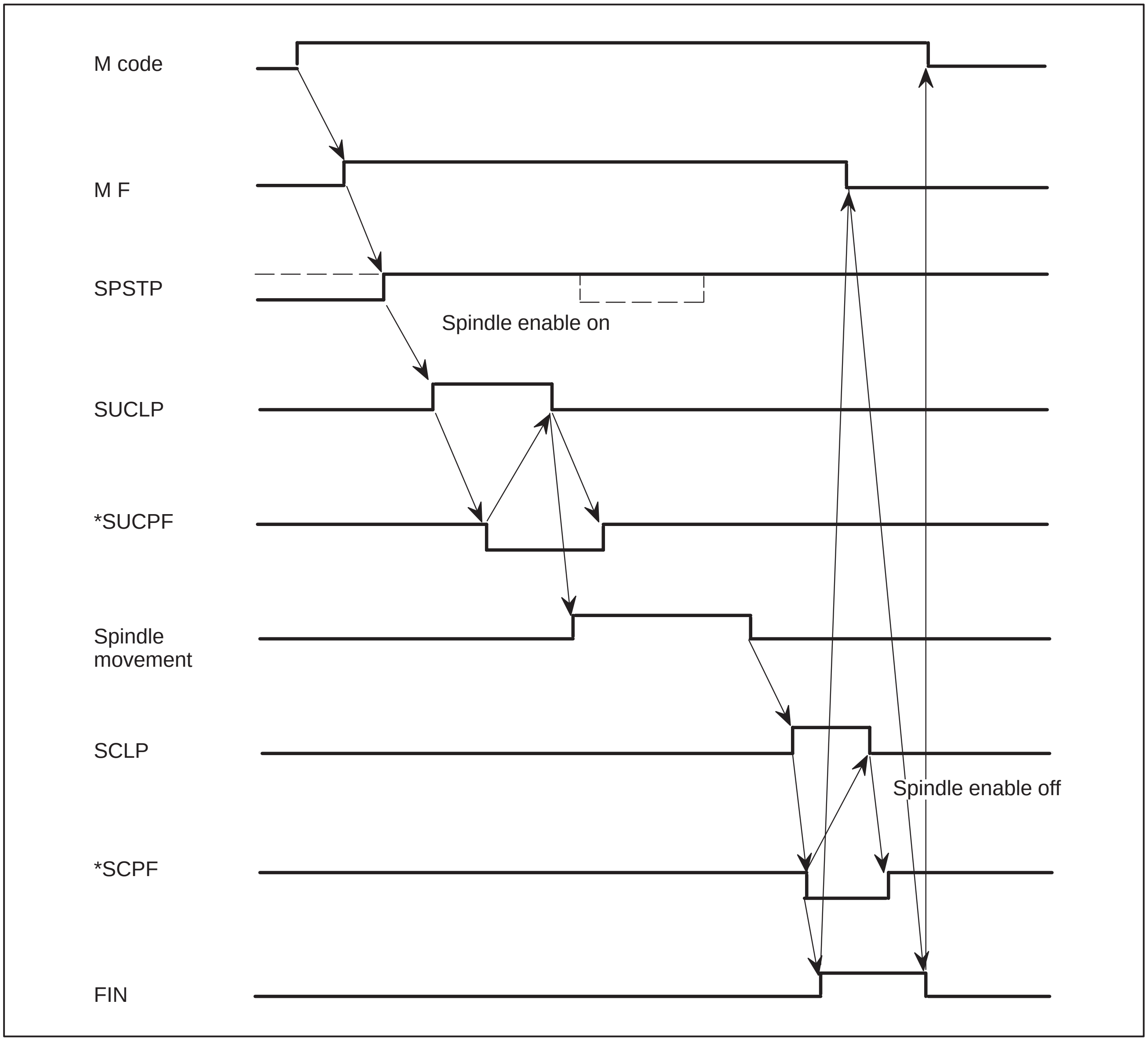
⇒ Spindle ENABLE ON/OFF specifies that the PMC ladder direct the spindle control unit to run or stop the spindle motor.

(Example) For serial spindles, the ladder should contain the following command or something like that:

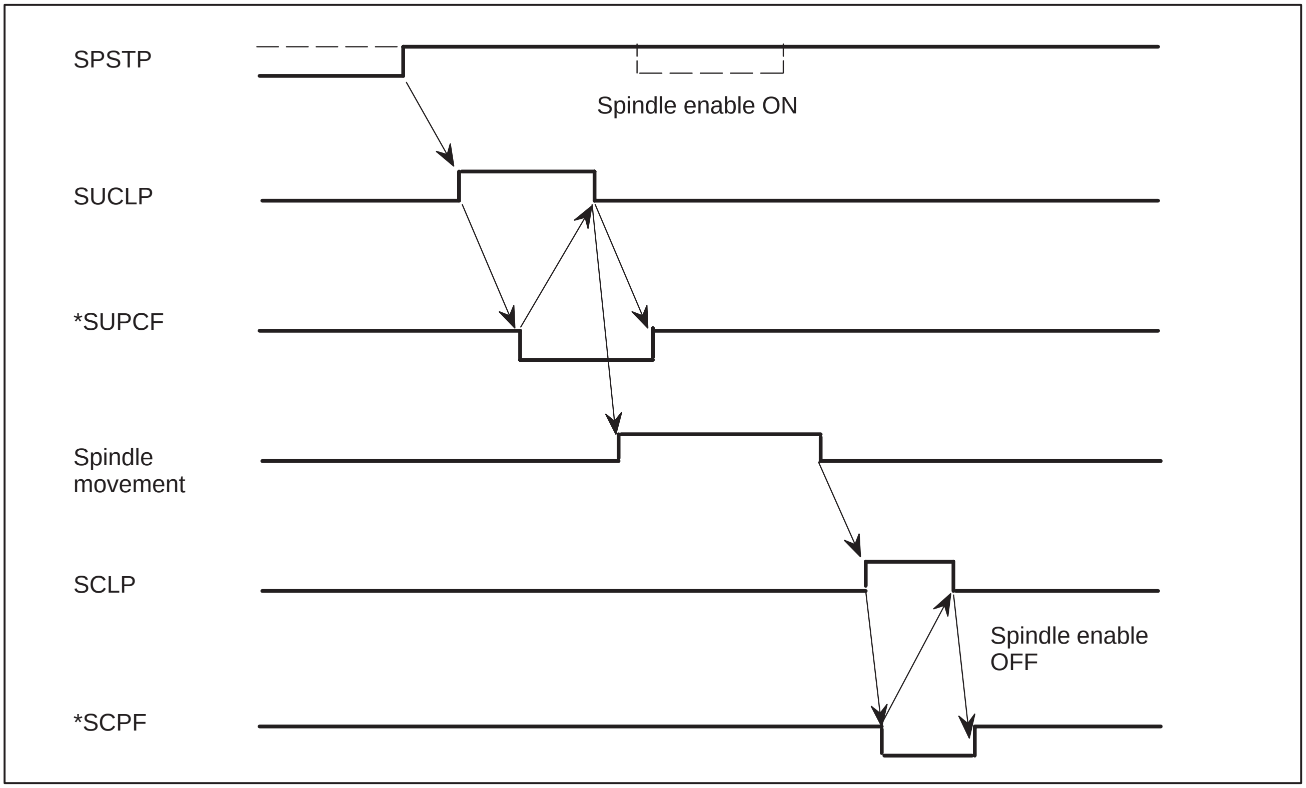
ENABLE ON, and SFRA<G070#5> ⇒ 1
ENABLE OFF, and SFRA<G070#5> ⇒ 0

For details, refer to the manual for the spindle control unit you actually use.

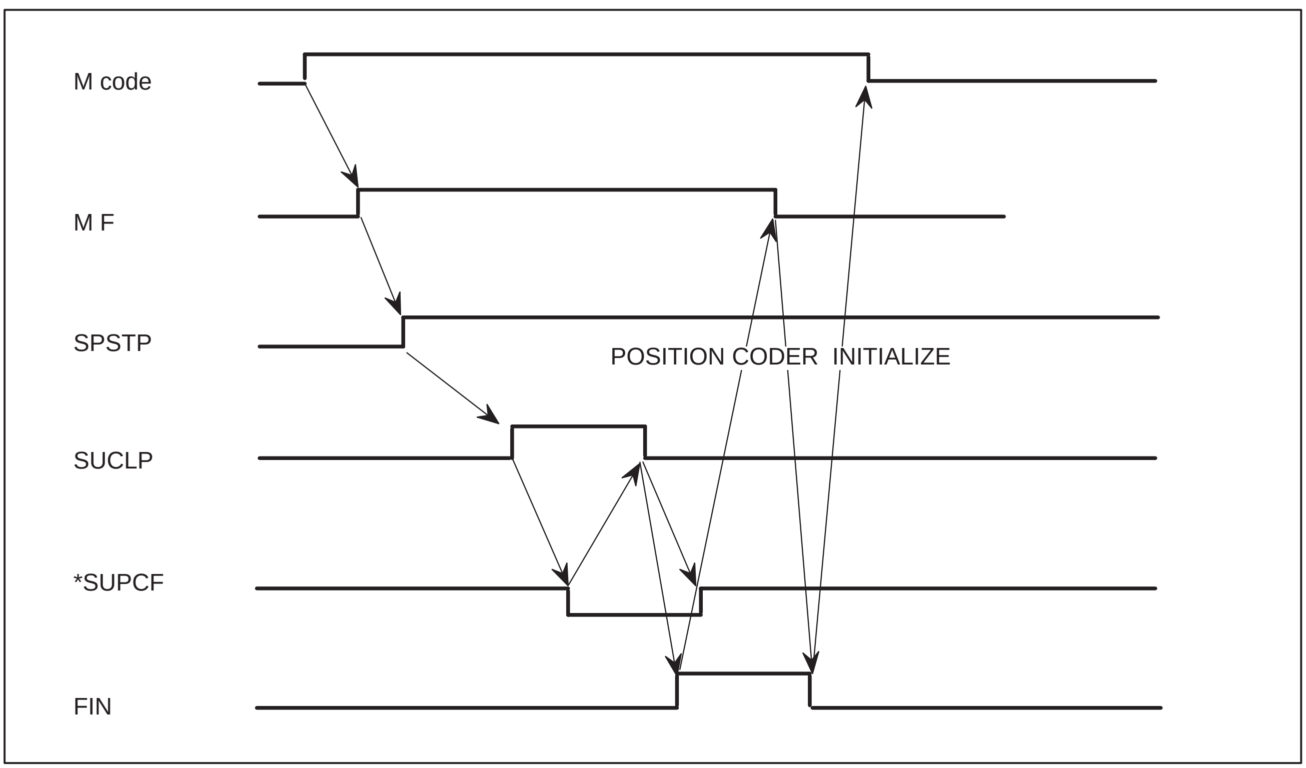
❑ Spindle Positioning by M code



❑ Spindle Positioning by Address C,H



❑ Spindle Positioning Reset



⇒ POSITION CODER INITIALIZE is performed only in the CNC.

Parameter

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

[Data type] Bit axis

ZMIx The direction of reference position return and the direction of initial backlash at power–on
0 : Positive direction
1 : Negative direction

1020	Name of the axis used for programming for each axis

[Data type] Byte axis

Set the name of the program axis for each control axis, with one of the values listed in the following table:

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

Note 1 In the T series, when G code system A is used, neither U, V, nor W can be used as an axis name. Only when G code system B or C is used, U, V, and W can be used as axis names.

Note 2 The same axis name cannot be assigned to more than one axis.

<u>The axis name of spindle positioning is C axis.</u>

1023	Number of the servo axis for each axis

[Data type] Byte axis

Set the servo axis for each control axis.

Generally, the same number shall be assigned to the control axis and the corresponding servo axis.

<u>Set –1 to the C axis when spindle positioning function is used.</u>

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

[Data type] Bit

ZPR Automatic setting of a coordinate system when the manual reference position return is performed
0 : Not set automatically
1 : Set automatically

1250	Coordinate value of the reference position used when automatic coordinate system setting is performed

[Data type] Two–word axis

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically.

For spindle positioning.

[Unit of data] 0.001 deg

[Valid data range] –99999999 to 99999999

1420	Rapid traverse rate for each axis

[Data type] Word axis

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

For spindle positioning.

[Unit of data] 10 deg/min

[Valid data range] 30 to 12000

1421	F0 rate of rapid traverse override for each axis

[Data type] Word axis

Set the F0 rate of the rapid traverse override for each axis.

For spindle positioning.

[Unit of data] deg/min

[Valid data range] 600 to 15000

1425	FL rate of the reference position return for each axis

[Data type] Word axis

Set feedrate (FL rate) after deceleration when the reference position return is performed for each axis.

For spindle positioning.

[Unit of data] deg/min

[Valid data range] 600 to 15000

Note 1 When serial spindle is used, this parameter becomes invalid.

1620	Time constant of rapid traverse linear acceleration/deceleration for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Set time constant of rapid traverse linear acceleration/deceleration for each axis.

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

[Data type] Bit axis

DM1x to DM3x Setting of detection multiply

Set this parameter to “111” (=4) for spindle positioning.

1820	Command multiply for each axis (CMR)

[Data type] Byte axis

- When command multiply is 1/2 to 1/27

$$\text{Set value}=\frac{1}{(\text{Command multiply})}+100 \quad [\text{Valid data range: 102 to 127}]$$

- When command multiply is 0.5 to 48

Set value = 2 × command multiply [Valid data range: 1 to 96]

Set this parameter to 2 for spindle positioning.

1821	Reference counter size for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range] 0 to 99999999

Set the size of the reference counter.

Set this parameter to 1000 for spindle positioning.

1826	In–position width for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set the in–position width for each axis.

1828	Positioning deviation limit for each axis in movement

[Data type] Two–word

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for each axis.

1829	Positioning deviation limit for each axis in the stopped state

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set the positioning deviation limit in the stopped state for each axis.

1850	Grid shift for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] 0 to ±99999999

Set a grid shift for each axis.

Note 1 Set this parameter when the analog spindle is used. When the serial spindle is used, set the value to No. 4073.

1851	Backlash compensating value for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] –9999 to +9999

Set the backlash compensation value for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
3405				CCR				

[Data type] Bit

CCR Addresses used for chamfering and corner rounding

0 : Address used for chamfering and corner rounding is I or K, not C. In direct drawing dimension programming, addresses ’C, ’R, and ’A (with comma) are used in stead of C, R, and A.

1 : Addresses used for chamfering, corner rounding, and direct drawing dimension programming are C, R, and A without comma. Thus, addresses A and C cannot be used as the names of axes.

Always set this parameter to “0” for spindle positioning.

	#7	#6	#5	#4	#3	#2	#1	#0
4000					RETRN			

[Data type] Bit

RETRN Reference position return direction of spindle.

0 : CCW (Counter clockwise)

1 : CW (Clockwise)

Note 1 The direction for spindle orientation (or reference position return) in spindle positioning using a serial spindle is determined by this parameter.

4044	Velocity loop proportion gain in servo mode (High gear)
4045	Velocity loop proportion gain in servo mode (Low gear)

[Data type] Word

[Unit of data]

[Valid data range] 0 to 32767

This parameter sets a velocity loop proportional gain in servo mode (spindle positioning, rigid tapping, etc.)

Note 1 Set this parameter when serial spindle is used.

[Data type] Word

[Unit of data]

4052	Velocity loop integral gain in servo mode (High gear)
4053	Velocity loop integral gain in servo mode (Low gear)

[Valid data range] 0 to 32767

This parameter sets a velocity loop integral gain in servo mode (spindle positioning, rigid tapping, etc.)

Note 1 Set this parameter when serial spindle is used.

4056	Gear ratio (HIGH)
4057	Gear ration (MEDIUM HIGH)
4058	Gear ratio (MEDIUM LOW)
4059	Gear ratio (LOW)

[Data type] Word

[Unit of data] Motor speed per spindle rotation $\times$ 100

[Valid data range] 0 to 32767

These parameters set the gear ration between the spindle and AC spindle motor.

Note 1 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

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 32767

This parameter sets a servo loop gain in servo mode.
(spindle positioning, rigid tapping, etc.)

Note 1 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) CTH1A, CTH1B.

	#7	#6	#5	#4	#3	#2	#1	#0
4950	IMB	ESI	TRNSP			ISZ	IDM	IOR

[Data type] Bit

IOR Resetting the system in the spindle positioning mode
0 : Does not releases the mode.
1 : Releases the mode

IDM The positioning direction for the spindle using a M code is

0 : The positive direction
1 : The negative direction

ISZ When an M code for spindle orientation is specified in spindle positioning:
0 : The spindle rotation mode is cleared and the mode is switched to the spindle positioning mode, and spindle orientation operation is performed.
1 : The spindle rotation mode is cleared and the mode is switched to the spindle positioning mode but spindle orientation operation is not performed.

TRNSP Direction of rotation of spindle positioning
0 : Normal
1 : Reverse

ESI Selection of a spindle positioning specification
0 : The conventional specificaion is used.
1 : The extended specificaion is used.

Note 1 The extended specification includes the following two extensions:

- With the conventional specification, the number of M codes for specifying a spindle positioning angle is always 6. With the extended specification, an arbitrary number of such M codes from 1 to 256 can be selected by parameter setting (See parameter No. 4964.)
- The maximum feedrate for spindle positioning (setting of parameter No. 1420) can be extended from 240000 to 269000 (in increments of 10 deg/min).

IMB When the spindle positioning function is used, half–fixed angle positioning based on M codes uses:
0 : Specification A
1 : Specification B

Note 1 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.

(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 specifiection A:
Operations (1) to (3) are specified using separate M codes.
(1)–Specified using M codes for performing spindle orientation. (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.)

4960	M code specifying the spindle orientation

[Data type] Word

[Unit of data] Integer

[Valid data range] 6 to 97

Set an M code to change the spindle rotating mode to the spindle positioning mode. Setting the M code performs the spindle orientation. Spindle positioning can be specified from the next block.

4961	M code releasing the spindle positioning mode

[Data type] Word

[Unit of data] Integer

[Valid data range] 6 to 97

Set the M code to release the spindle positioning mode and to change the mode to the spindle rotating mode.

4962	M code for specifying a spindle positioning angle

[Data type] Word

[Unit of data] Integer

[Valid data range] 6 to 92

Two methods are availablel for specifying spindle positioning. One method uses address C 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.

- When bit 6 (ESI) of parameter No. 4950=0

  Six M code from M α to M(α+5) are used for half–fixed angle positioning, when α is the value of this parameter.

- When bit 6(ESI) of parameter No. 4950=1

  Set the start M code in this parameter, and set the number of M codes in parameter No. 4964. Then β M codes from Mα to M(α+β–1) are used for half fixed angle positioning.

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 code	Positioning angle	Example: Positioning angle when θ = 30°
M (α+5)	6θ	180°
⋮	⋮	⋮
M (α+n)	(n+1)θ	

Note 1 θ represents the basic angular diplacement set in pamrameter No. 4963.

4963	M code for specifying a spindle positioning angle

[Data type] Word

[Unit of data] deg

[Valid data range] 1 to 60

This parameter sets a basic angular displacement used for half–fixed angle positioning using M codes.

4964	Number of M codes for specifying a spindle positioning angle

[Data type] Byte

[Unit of data] Integer

[Valid data range] 0, 1 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+5) are used for half–fixed angle positioning.

Note 1 This parameter is valid when bit 6 (ESI) of parameter NO. 4950=1.

Note 2 Make sure that M codes from Mα to M (α+β–1) do not duplicate other M codes.

Note 3 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.

4970	Servo loop gain of the spindle

[Data type] Word

[Unit of data] 0.01 sec^{-1}

[Valid data range] 1 to 9999

Set the servo loop gain of the spindle in the spindle positioning mode.

4971	Servo loop gain multiplier of the spindle for gear 1

[Data type] Word

4972	Servo loop gain multiplier of the spindle for gear 2
4973	Servo loop gain multiplier of the spindle for gear 3
4974	Servo loop gain multiplier of the spindle for gear 4

Set the servo loop gain multipliers of the spindle for gears 1 to 4.

The multipliers are used to convert the amount of the position deviation to the voltage used in the velocity command. Assign the data obtained from the following equation to the parameters.

Loop gain multiplier = 2048000 $\times$ E $\times$ A/L

where;

E: Voltage required to rotate the spindle motor at 1000 rpm in the velocity command

L: Rotation angle of the spindle per one motor rotation (normally 360)

A: Unit used for the detection (degree)

Example) Let E be 2.2 V, L be 360 degrees, and A be 0.088 degrees/pulse.

Loop gain multiplier = 2048000 $\times$ 2.2 $\times$ 0.088/360 = 1101

(Note) When the voltage specified for the spindle motor is 10 V at a spindle speed of 4500 rpm, E is regarded as 2.2 V.

Note 1 The above parameters No. 4970 to No. 4974 are for analog spindles

Alarm and message

Number	Message	Description
053	TOO MANY ADDRESS COMMANDS	In the chamfering and corner R commands, two or more of I, K and R are specified. Otherwise, the character after a comma(”,”) is not C or R in direct drawing dimensions programming. Or comma (“,”) was specified with parameter No. 3405#4=1. Modify the program.
056	NO END POINT & ANGLE IN CHF/CNR	Neither the end point nor angle is specified in the command for the block next to that for which only the angle is specified (A). In the chamfering or corner R command, I(K) is commanded for the X(Z) axis. Modify the program.
135	SPINDLE ORIENTATION PLEASE	Without any spindle orientation , an attept was made for spindle indexing. Perform spindle orientation.
136	C/H–CODE & MOVE CMD IN SAME BLK.	A move command of other axes was specified to the same block as spindle indexing addresses C, H. Modify the program.
137	M–CODE & MOVE CMD IN SAME BLK.	A move command of other axes was specified to the same block as M–code related to spindle indexing. Modify the program.
194	SPINDLE COMMAND IN SYNCHRO–MODE	A contour control mode, spindle positioning (Cs–axis control) mode, or rigid tapping mode was specified during the serial spindle synchronous control mode. Correct the program so that the serial spindle synchronous control mode is released in advance.
751	FIRST SPINDLE ALARM DETECTION (AL–XX)	This alarm indicates in the NC that an alarm is generated in the spindle unit of the system with the serial spindle. The alarm is displayed in form AL–XX (XX is a number). The alarm number XX is the number indicated on the spindle amplifier. The CNC holds this number and displays on the screen.
752	FIRST SPINDLE MODE CHANGE FAULT	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contouring, 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.

Note

Note 1 Direct spindle positioning with an independent block. X- and Y-axis positioning cannot be commanded to the sample block.

Note 2 Spindle positioning cannot be done by manual operation.

Note 3 Feed hold is invalid during spindle positioning.

Note 4 Spindle positioning stops when emergency stop is applied; restart with orientation operation.

Note 5 Dry run, machine lock, and auxiliary function lock are not available during spindle positioning.

Note 6 Automatic drift compensation is not effective for spindle positioning. To adjust the amount of drift compensation for each axis, set values manually and adjust the spindle amplifier to minimize the spindle motor rotation at a voltage of 0 V. (parameter no. 3731). Insufficient adjustment causes poor positioning accuracy. Drift compensation is not needed with a serial spindle.

Note 7 The spindle positioning function and the serial spindle Cs contour control function cannot be used together. If both functions are specified, positioning has priority.

Note 8 Specify parameter no. 4962 even if semi-fixed angle positioning is not used; otherwise M codes (M00 to M05) do not work.

Note 9 The machine coordinates for the spindle positioning axis are displayed in pulses units.

Reference item

OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.10.5	SPINDLE POSITIONING FUNCTION

9.9 Cs CONTOUR CONTROL

General

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.

• Increment system

Least input increment: 0.001 [deg]
Least command increment: 0.001 [deg]

• Maximum command value

+9999.999 [deg]

• Feedrate

Rapid traverse rate: 30 to 2400 [deg/min] (parameter no.1420)

Cutting feedrate (feed per minute):
1 to 15000 [deg/min] (for machines that use millimeters)
0.01 to 600 [deg/min] (for machines that use inches)

The spindle motor controlled by this function must be a serial interface spindle (Serial spindle).
Positioning and linear interpolation between the serial spindle and other servo axes can be specified.

Explanations

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

The axis used for Cs contour control must be set as an axis of the axes controlled by the CNC. Using parameter no. 1023, assign “–1” in the field corresponding to the chosen servo axis. Also set the spindle contour control axis as a rotation axis by setting ROTx (bit 0) of parameter nos. 1006 and 1022.

Only one set of this setting can be used for each control path. The spindle that operates under Cs contour control is a serial spindle as the first spindle.

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 option is provided, address B cannot be used for the name of the contour axis. For the T series machines, when either address A or C is used for the name of the contour axis, clear CCR (parameter no. 3405#4) to “0”.

Setting Axes that interpolate with Cs contour axis

Up to five servo axes can be specified for linear interpolation against the Cs contour control axis, by setting defined parameters :

- When no servo axis is used for interpolation, specify “0” in parameter nos. 3900, 3910, 3920, 3930, 3940.
- When one or more servo axes are used for interpolation, set the parameter for each as follows :

(1) Assign the axis number (1 to 8) to each of the servo axes used for interpolation in parameter nos. 39n0 (n=0, 1, 2, 3, or 4).

(2) Set the loop gain for each of the servo axes specified in (1) in parameter nos. 39n1, 39n2, 39n3, 39n4. 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 five, set “0” in remaining parameter nos. 39n0.

Switching spindle speed control/Cs contour control

❑ Switching from spindle speed control to Cs contour control

The serial spindle is put in the Cs contour control mode by setting the DI signal CON (G027#7) to “1”. When the mode is switched while the spindle is rotating, the spindle stops immediately to perform the change.

❑ Switching from Cs contour control to spindle speed control

Turning the DI signal CON (G027#7) to “0” puts the serial spindle in 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.

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

After the serial spindle enters the Cs contour control mode, the spindle returns to the reference position when G28 is specified. Under certain conditions, the G00 command returns the spindle to the reference position, depending upon the setting of parameter NRF no. 3700#1:

(i) G00 command

Returning to the reference position using the G00 command differs from using the G28 command or the manual method. The serial spindle can be positioned at any point using the G00 command, while the latter two methods always return the serial spindle to the reference position.

When parameter NRF no. 3700#1 is “0” and the serial spindle is put in the Cs contour control mode, if the G00 command is given before returning the spindle to the reference position, the serial spindle returns to the reference position and indexes it before moving to the commanded position. After positioning at the reference position, the reference position return completion signal (ZPn(F094)) turns to “1”. When the G00 command is issued after the serial spindle has returned to the reference position at least once, normal positioning operation is executed.

(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 P/S alarm 197 occurs during automatic operation.

In the spindle speed mode, inhibit manual operation of the Cs contour control axis using the PMC ladder.

Display of Position Error of Cs Contour Control Axis

DGN No.

418	Position deviation amount of 1st spindle

Position deviation amount of the position loop for the 1st 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. 300x) for an axis under Cs contour control.

Signal

Spindle contour control change signal CON <G027#7>

[Classification] Input signal

[Function] This signal specifies that with the Cs contour control function, the first serial spindle be switched between the spindle speed control and Cs contour control modes.

When this signal turns to ”1”, the spindle mode changes from speed control to Cs contour control.

If the spindle is moving at the time of the change, it stops immediately. Turning the signal to “0” changes the spindle mode from Cs contour control back to speed control.

Spindle contour control change completion signal FSCSL <F004#1>

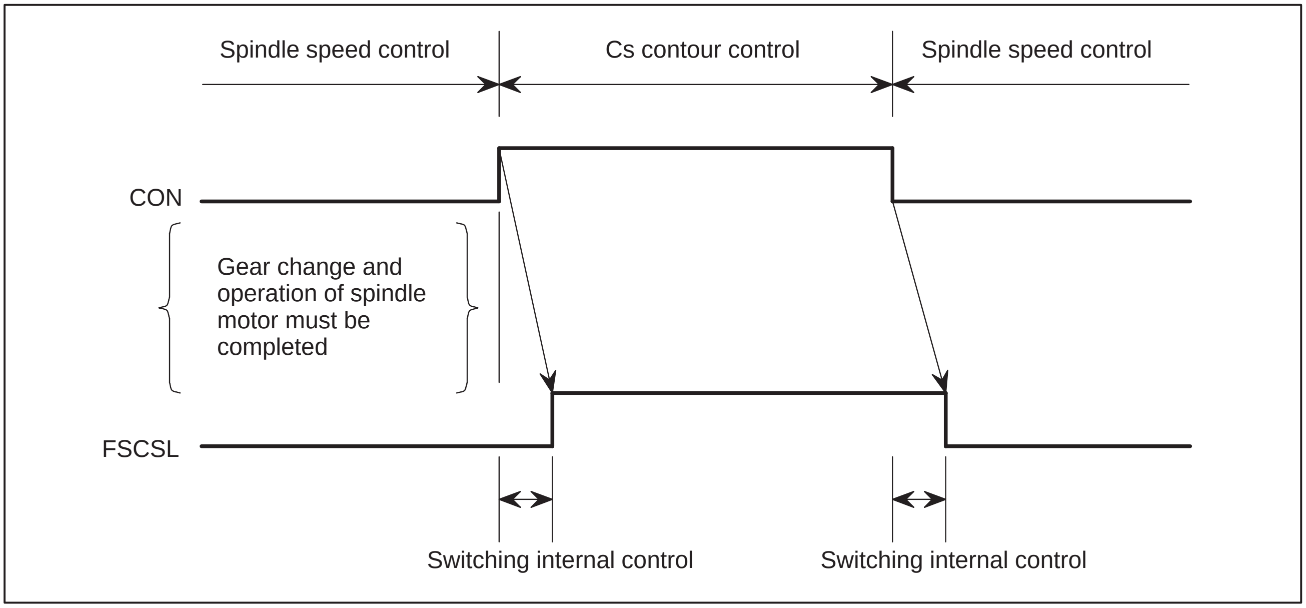
[Classification] Output signal

[Function] This signal indicates the axis is under Cs contour control.

[Output condition] Spindle speed control mode $\rightarrow$ 0

Cs contour control mode $\rightarrow$ 1

Time Chart



Notes

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.

A servo excessive error may be generated if the spindle motor is not ready for operation. (Signal SRVA, SFRA <G070#5, #6> or other required signals must be appropriately processed on the machine side).

Other signals

Gear select signal (Input)
GR1, GR2, <G028#1, #2>
Gear select signal (Output)
GR3O, GR2O, GR1O <F034#2, #1, #0>
(M series)

Refer to 9.3 "Spindle speed Control".

Clutch/Gear signal (Serial spindle)
CTH1A, CTH2A <G070#3, #2>

Refer to the manual of serial spindle.

These signals determine what parameter (loop gain, etc.) to be used for each gear position.

CTH1A and CTH2A 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

Analog spindle							Serial spindle		
T/M series with CSSC			M series without CSSC						
GR2	GR1	Gear selection	GR3O	GR2O	GR1O	Gear selection	CTH1A	CTH2A	Gear selection
0	0	1st stage	0	0	1	1st stage	0	0	1st stage
0	1	2nd stage	0	1	0	2nd stage	0	1	2nd stage
1	0	3rd stage	1	0	0	3rd stage	1	0	3rd stage
1	1	4th stage					1	1	4th stage

CSSC: Constant surface speed control

Notes

When the M series does not include the constant surface speed control option, and parameter No. 3706#4 GTT=0, GR1 and GR2 do not need to be input. Input CTH1A and CTH2A when gears are changed using GR10, GR2O and GR3O.

Cs contour control axis reference position return completion signal ZPx <F094>

[Classification] Output signal

[Function] This signal indicates that a reference position return has been made for the Cs contour control axis.

[Output condition] 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
G027	CON							
G028						GR2	GR1	
G070	MRDYA		SFRA	SRVA	CTH1A	CTH2A		
F034						GR3O	GR2O	GR1O
F044							FSCSL	
F094	ZP8	ZP7	ZP6	ZP5	ZP4	ZP3	ZP2	ZP1

Parameter

The following describes major parameters.

In addition, parameters such as axis feedrate, acceleration/deceleration, and display can be used. Also, digital servo parameters (No. 2000 ~) for the Cs contour axis are not required to be set.

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

[Data type] Bit

Type of controlled axis
0 : Linear axis
1 : Rotation axis

Note 1 Inch/metric conversion cannot be made to the rotation axis.
The machine coordinate values are rounded in 0° to 360°. Automatic reference position return (G28, G30) is made in the manual reference position return direction and the move amount does not exceed one rotation.

1020	Name of the axis used for programming for each axis

[Data type] Byte axis

Set the name of the program axis for each control axis, with one of the values listed in the following table:

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

Note 1 In the T series, when G code system A is used, neither U, V, nor W can be used as an axis name. Only when G code system B or C is used, U, V, and W can be used as axis names.

Note 2 The same axis name cannot be assigned to more than one axis.

Note 3 When the second auxiliary function is provided, address B cannot be used as an axis name. In the T series, when address A or C is used, set parameter CCR (No. 3405#4) to 0.

Any axis name can be used for Cs contour control axis except for above limitation.

1022	Setting of each axis in the basic coordinate system

[Data type] Byte axis

Only one axis can be set for each of the three basic axes X, Y, and Z, but two or more parallel axes can be set.

Set value	Meaning
0	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

Set 0 to the Cs contour control axis.

1023	Number of the servo axis for each axis

[Data type] Byte axis

Set the servo axis for each control axis.

Generally, the same number shall be assigned to the control axis and the corresponding servo axis.

Set –1 as the number of servo axis to the Cs contour control axis.

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

[Data type] Bit

ZPR Automatic setting of a coordinate system when the manual reference position return is performed
0 : Not set automatically
1 : Set automatically

1250	Coordinate value of the reference position used when automatic coordinate system setting is performed

[Data type] Two–word axis

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically.

1420	Rapid traverse rate for each axis

[Data type] Word axis

[Unit of data] 1 deg/min

[Valid data range] 30 to 24000 (IS–A, IS–B)
30 to 12000 (IS–C)

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1620	Time constant used in linear acceleration/deceleration or bell–shaped acceleration/deceleration in rapid traverse for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

1820	Command multiply for each axis (CMR)

[Data type] Byte axis

- When command multiply is 1/2 to 1/27

$$\text{Set value} = \frac{1}{(\text{Command multiply})} + 100 \text{ [Valid data range: 102 to 127]}$$

- When command multiply is 0.5 to 48

Set value = 2 $\times$ command multiply [Valid data range: 1 to 96]

1826	In–position width for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set the in–position width for each axis.

1828	Positioning deviation limit for each axis in movement

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for each axis.

1829	Positioning deviation limit for each axis in the stopped state

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set the positioning deviation limit in the stopped state for each axis.

1851	Backlash compensation value used for rapid traverse for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] −9999 to +9999

Set the backlash compensation value for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
3105	CNRST							

[Data type] Bit

CNRST When reference position return is completed, relative coordinate value of Cs contour control axis is
0 : Not cleared
1 : Cleared

	#7	#6	#5	#4	#3	#2	#1	#0
3700							NRF	

[Data type] Bit

NRF The first positioning command by G00 after the serial spindle is switched to Cs axis contouring control performs:
0 : Positioning after returning to the reference position.
1 : Normal positioning

3900	The number of servo axis that interpolates with Cs contour control axis

[Data type] Byte axis

[Valid data range] 0 to 8

Set the number of servo axis that interpolates with Cs contour control axis (1st group)

Note 1 Set 0 when there is no servo axis that interpolates with Cs contour control axis.

3901	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (High gear)
3902	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium high gear)
3903	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium low gear)
3904	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Low gear)

[Data type] Word axis

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 32767

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear (1st group)

3910	Number of servo axis that interpolates with Cs contour control

[Data type] Byte axis

[Valid data range] 0 to 8

Set the number of servo axis that interpolates with Cs contour control axis (2nd group)

Note 1 When there is no servo axis or only one servo axis that interpolates with Cs contour control axis, set this parameter to 0.

3911	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (High gear)
3912	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium high gear)
3913	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium low gear)
3914	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Low gear)

[Data type] Word axis

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 32767

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear (2nd group)

3920	Number of servo axis that interpolates with Cs contour control

[Data type] Byte axis

[Valid data range] 0 to 8

Set the number of servo axis that interpolates with Cs contour control axis (3rd group)

Note 1 When there is no servo axis or less than three servo axes that interpolates with Cs contour control axis, set this parameter to 0.

3921	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (High gear)
3922	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium high gear)
3923	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium low gear)
3924	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Low gear)

[Data type] Word axis

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 32767

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear (3rd group)

3930	Number of servo axis that interpolates with Cs contour control

[Data type] Byte

[Valid data range] 0 to 8

Set the number of servo axis that interpolates with Cs contour control axis (4th group)

Note 1 When there is no servo axis or less than four servo axes that interpolates with Cs contour control axis, set this parameter to 0.

3931	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (High gear)
3932	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium high gear)
3933	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium low gear)
3934	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Low gear)

[Data type] Word axis

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 32767

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear (3rd group)

3940	Number of servo axis that interpolates with Cs contour control

[Data type] Byte axis

[Valid data range] 0 to 8

Set the number of servo axis that interpolates with Cs contour control axis (5th group)

Note 1 When there is no servo axis or less than five servo axes that interpolates with Cs contour control axis, set this parameter to 0.

3941	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (High gear)
3942	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium high gear)
3943	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium low gear)
3944	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Low gear)

[Data type] Word axis

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 32767

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear (5th group)

4056	Gear ratio (HIGH)
4057	Gear ratio (MEDIUM HIGH)
4058	Gear ratio (MEDIUM LOW)
4059	Gear ratio (LOW)

[Data type] Word axis

[Unit of data] (Number of motor rotations to one spindle rotation) $\times$ 100

[Valid data range] 0 to 32767

Set the gear ratio between spindle and AC spindle motor.

Note 1 For which gear ratio is used in actual spindle operation, it depends on clutch/gear signal (serial spindle) CTH1A, CTH2A.

4065	Position gain at Cs contour control (High gear)
4066	Position gain at Cs contour control (Medium High gear)

4067	Position gain at Cs contour control (Medium Low gear)
4068	Position gain at Cs contour control (Low gear)

[Data type] Word axis

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 32767

Set the position gain at Cs contour control.

Note 1 For which position gain is used in actual spindle operation, it depends on clutch/gear signal (serial spindle) CTH1A, CTH2A.

4135	Grid shift value at Cs contour control

[Data type] Two–word

[Unit of data] 1 pulse unit (360000 p/rev)

[Valid data range] –360000 to +360000

Set the number of pulses from an issue of one–rotation signal to the machine zero point in Cs contour control.

Alarm and message

Number	Message	Description
194	SPINDLE COMMAND IN SYNCHRO–MODE	A contour control mode, spindle positioning (Cs–axis control) mode, or rigid tapping mode was specified during the serial spindle synchronous control mode. Correct the program so that the serial spindle synchronous control mode is released in advance.
197	C–AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cs contour control axis when the signal CON(G027#7) was off. Correct the program, or consult the PMC ladder diagram to find the reason the signal is not turned on.
751	FIRST SPINDLE ALARM DETECTION (AL–XX)	This alarm indicates in the NC that an alarm is generated in the spindle control unit of the system with the serial spindle. The alarm is displayed in form AL–XX (XX is a number). The alarm number XX is the number indicated on the spindle amplifier. The CNC holds this number and displays on the screen.
752	FIRST SPINDLE MODE CHANGE FAULT	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contouring, 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.

Note

Note 1 In the T series machines, the spindle contour control function and the spindle positioning function cannot be used at the same time. If both functions are specified simultaneously, the spindle positioning function takes precedence.

Note 2 In the spindle contour control mode, do not switch the spindle gears. When the gears need to be changed put the system in the spindle speed control mode first.

Reference item

FANUC CONTROL MOTOR AMPLIFIER α series DESCRIPTIONS (B–65152E)	11.6	Cs Contour Control Function
FANUC AC SPINDLE MOTOR series (Serial interface) DESCRIPTIONS	Appen-dix 3.2	Cs Contour Control Function Start–up Procedure

9.10 MULTI-SPINDLE CONTROL (T series)

General

In addition to the conventional (first) spindle, two other (second and third) spindles can be controlled. These additional spindles allow two-stage gear changes. An S code is used for a command to any of these spindles; which spindle is selected is determined by a signal from the PMC. The second and third spindle can change gears in 2 stages.

Also, the maximum spindle speed can be set to each spindle to clamp the spindle speed of each spindle (set by parameters No. 3772, 3802 and 3822).

When the second spindle is used, one position coder interface channel is added. Which position coder is selected is determined by a PMC signal. (The conventional and additional position coders are referred to as the first position coder and second position coder, respectively, throughout the remainder of this discussion.)

Selection between 1st position coder and 2nd position coder is made by a signal from PMC.

The spindle analog output option must be included in order to use the third spindle. The spindle serial output option is required to use multi-spindle control.

Control

Two multi-spindle control methods are available. Type A allows the SIND function (controlling the spindle motor speed based on the PMC) to be used only for the first spindle. Type B allows the SIND function to be used for each of the three spindles independently.

Basic control (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 SWS3 <G027#0-#2>). 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.

Each spindle also has a spindle stop signal (*SSTP1 to *SSTP3 <G027#3-#5>) to stop its rotation; an unused spindle can still be stopped.

There is a spindle enable signal to control each spindle; ENB <F001#4> controls the first spindle, while ENB2 and ENB3 <F038#2, #3> control the second and third spindles, respectively.

The PMC signal PC2SLC <G028#7> is used to select between the first and second position coders.

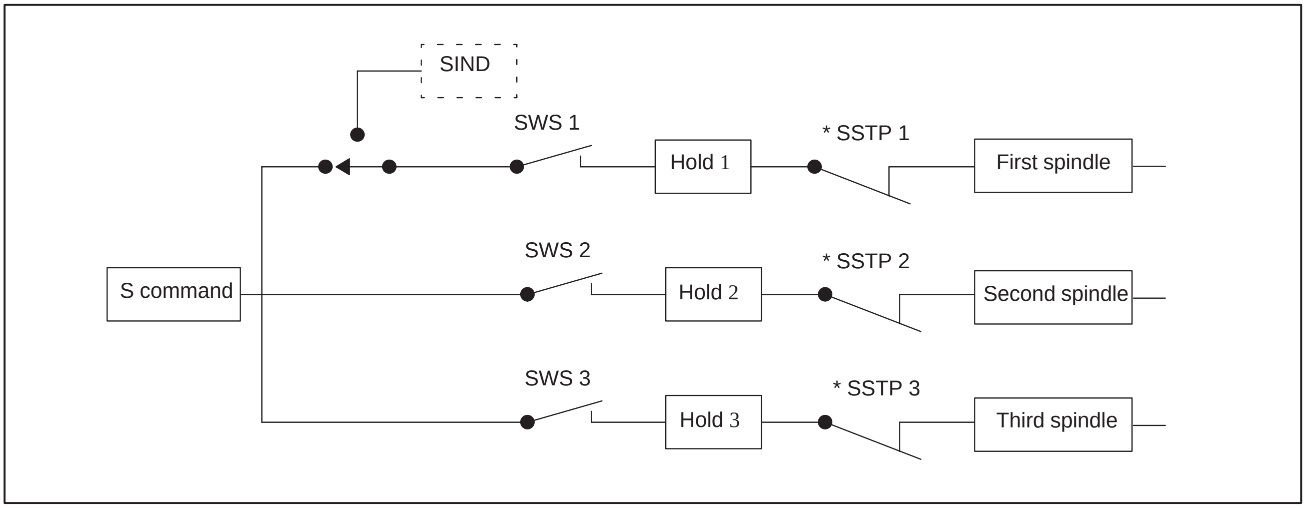
Multi-spindle control (TYPE-A)

When parameter MSI (No. 3709#2)=0, TYPE-A is used.

When the first spindle is selected with the SWS1 signal, the SIND signal <G033#7> is used to determine whether the spindle analog voltage is controlled by the PMC or CNC; then signals R01I to R12I <G0033#3 to G0032#0> are used to set that spindle's analog voltage. These signals do not affect the second and third spindles.

The PMC-based polarity (rotation direction) control signals SGN and SSIN <G033#5,#6> will function for any spindle selected by SWS1 to SWS3.

The concept of Type A multi-spindle control is outlined below.



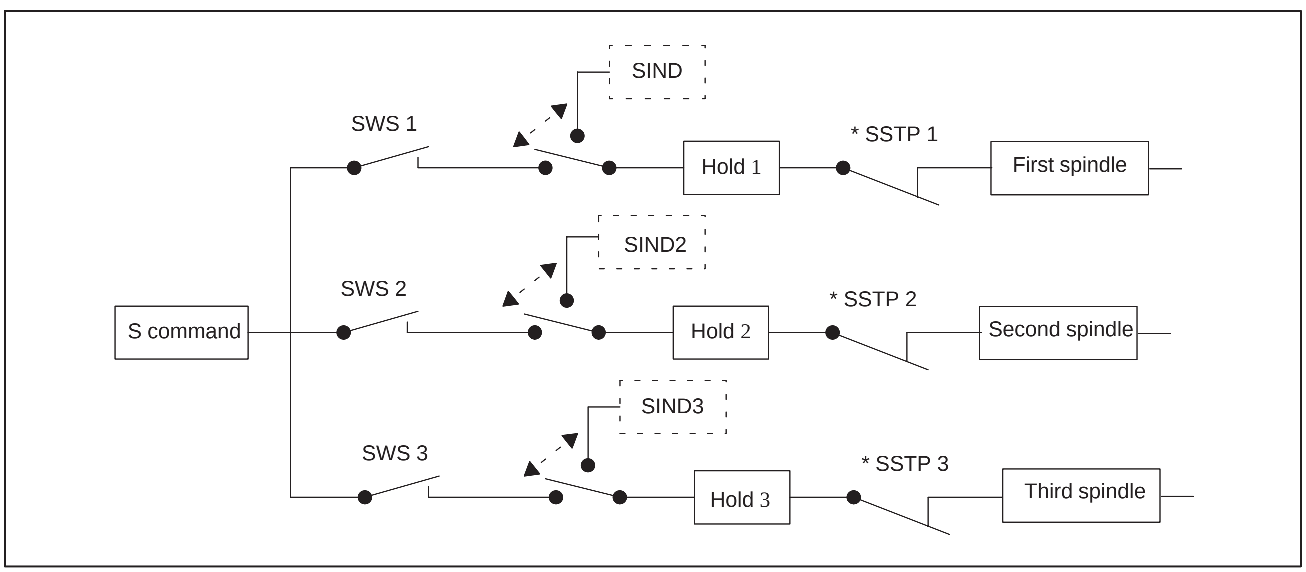
Multi-spindle control (TYPE-B)

Select Type B control by setting parameter MSI no. 3709#2 to “1”.

Each spindle has its own SIND, SSIN and SGN signals.
Each signal functions regardless of selection state of the spindle selection signals (SWS1 to SWS3).

When either the spindle selection signal or the SIND signal for the first, second or third spindle is set to “1”, the polarity control signals SSIN, SGN will function. The first spindle is controlled by the SGN and SSIN signals; the second and third spindles are controlled by SGN2, SGN3 <G035#5, G037#5>, and SSIN2, SSIN3 <G035#6, G037#6>, respectively.

The concept of Type B multi-spindle control outlined below.



Spindles to be controlled

In multi-spindle control, the first spindle is the first serial spindle, the second spindle is the second serial spindle, and the third spindle is an analog spindle.

A configuration is possible which does not connect the second or third spindles.

Connection of spindle

Spindle configuration when multi-spindle control is used:

Necessary option and parameter	· Multi-spindle control · Spindle serial output · Parameter SP2 (No. 3701#4) = 1 (to use second spindle) · Spindle analog output (to use third spindle)
Connection of each spindle	First spindle (CN11A) → Connect to JA7A (on CNC main CPU board). Second spindle (CN11A)→ CN11B connector on first spindle control unit. Set jumper S1 on first spindle control unit to B. Third spindle → Connect to JA8A connector on CNC main CPU board.
Connection of each position coder	First position coder: Feedback information obtained by position coder or equivalent sensor connected to first spindle control unit is fed to CNC via serial interface. Second position coder: Feedback information obtained by position coder or equivalent sensor connected to second spindle control unit is fed to CNC via serial interface through 1st spindle control unit Note) When second spindle is not connected, second position coder cannot be used.

For detailed information about serial spindle connection, refer to the manuals on the serial spindles.

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 three spindles if the spindle speed is within the range allowable for this function. (When the position coder is required, it can be installed on the 1st or 2nd spindle). The spindle selection signal (SWS 1–3) for the spindle must stay set at “1” during machining using this function.

• Spindle speed fluctuation detection

When the spindle speed fluctuation detection function is combined with multi-spindle control, two position coders can be used. Monitor the states of the second position coder selection signal (PL2SLC) and spindle selection signals (SWS 1–3).

• Actual spindle speed output

The actual spindle speed output function conveys speed information obtained from the selected position coder specified by the 2nd position coder selection signal (PC2SLC) to the PMC.
When the parameter HSO (No. 3709#5)=1, the difference of the feedback pulses between the 1st and 2nd position coder can be output irrespective of the state of 2nd position coder selection signal (PC2SLC).

- **Spindle positioning or Cs contour control**

When the spindle motor is used for positioning, as in the case of spindle positioning or Cs contour control, the first spindle functions as the positioning spindle. Switching to the positioning mode and positioning command are possible irrespective of the state of the selection signal of the first spindle (SWS1). This means that the first spindle cannot be controlled as a spindle in positioning mode, but the second and third spindles can be controlled as usual.

- **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, simplified synchronization control**

During spindle synchronization, polygon turning between spindles or simplified synchronization control, the second spindle operates in phase with the first spindle. Multi-spindle control for the first and third spindles can be used during synchronization control, but multi-spindle control for the second spindle is disabled.

- **Rigid tapping**

Using the spindle selection signal (SWS 1–3), rigid tapping can use either the first or second spindle as the rigid tap spindle. There are certain restrictions:

- Set the SWS 1 to 3 signals before directing rigid tapping;
- Do not switch the SWS 1 to 3 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.

- **Two-path control option**

Refer to 9.4 “Spindle Control for Two-path Lathe”.

Signal

Spindle Selection Signal SWS1, SWS2, SWS3 <G027#0, #1, #2>

[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.

SWS3 1 : Outputs a speed command to the third spindle.
0 : Outputs no speed command to the third spindle.

Individual spindle stop signal *SSTP1, *SSTP2, *SSTP3 <G027#3, #4, #5>

[Classification] Input signal

[Function] Effective only to multi-spindle, each spindle can be stopped by this signal.

***SSTP1**
1 : Does not set 0 rpm for output to the first spindle.
0 : Sets 0 rpm for output to the first spindle.

***SSTP2**
1 : Does not set 0 rpm for output to the second spindle.
0 : Sets 0 rpm for output to the second spindle.

***SSTP3**
1 : Does not set 0 rpm for output to the third spindle.
0 : Sets 0 rpm for output to the third spindle.

Gear select signal GR21 <G029#0> GR31 <G029#2>

[Classification] Input signal

[Function] Gear selection signals for 2nd and 3rd spindle when multi–spindle is equipped (2–stage). Use GR1 and GR2 <G028#1, #2> for the 1st spindle and up to 4–stage gears can be used.

GR21
1 : Selects the second-stage gear for the second spindle.
0 : Selects the first-stage gear for the second spindle.

GR31
1 : Selects the second-stage gear for the third spindle.
0 : Selects the first-stage gear for the third spindle.

2nd position coder selection signal PC2SLC <G028#7>

[Classification] Input signal

[Function] Position coder selection signal used for control.

PC2SLC
1 : Uses feedback pulses obtained by the second position coder for control.
0 : Uses feedback pulses obtained by the first position coder for control.

When the second position coder is not installed, do not switch this signal and always select the first position coder.

Spindle enable signal ENB2<F038#2> ENB3<F038#3>

[Classification] Output signal

[Function] These signals inform PMC of whether or not to perform output to the second and third spindles in multi–spindle control.

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 Section 9.11.)

[Output condition] ENB2 1 : Outputs a value other than 0 to the second spindle control unit.
0 : Outputs 0 to the second spindle control unit.

ENB3 1 : Outputs a value other than 0 to the third spindle control unit.
0 : Outputs 0 to the third spindle control unit.

Spindle control signal by PMC

1st spindle SIND, SSIN, SGN, <G033#7, #6, #5> (Input)
R12I~R01I<G033#3~G032#0> (Input)

2nd spindle SIND2, SSIN2, SGN2, <G035#7, #6, #5> (Input)
R12I2~R01I2<G035#3~G034#0> (Input)

3rd spindle SIND3, SSIN3, SGN3, <G037#7, #6, #5> (Input)
R12I3~R01I3<G037#3~G036#0> (Input)

[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 a 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 and third spindles.

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 and third spindles cannot be used.

For details of each signal, see Section 15.4.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G027			*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
G028	PC2SLC					GR2	GR1	
G029		*SSTP				GR31		GR21
G032	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
G033	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
G034	R08I2	R07I2	R06I2	R05I2	R04I2	R03I2	R02I2	R01I2
G035	SIND2	SSIN2	SGN2		R12I2	R11I2	R10I2	R09I2
G036	R08I3	R07I3	R06I3	R05I3	R04I3	R03I3	R02I3	R01I3

G037	SIND3	SSIN3	SGN3		R12I3	R11I3	R10I3	R09I3

F038					ENB3	ENB2		

Parameter

The parameters for the 1st spindle and the 1st position coder are the same as usual. This section describes the parameters which are added by this function.

	#7	#6	#5	#4	#3	#2	#1	#0
3701				SS2				

[Data type] Bit

SS2 The number of connections in serial spindle control
0 : 1
1 : 2

	#7	#6	#5	#4	#3	#2	#1	#0
3702							EMS	

[Data type] Bit

EMS Multi–spindle control is
0 : Used
1 : Not used

Note 1 If the multi–spindle control function is not required for one tool post in two–path control, specify this parameter for the tool post to which the multi–spindle control function need not be applied.

	#7	#6	#5	#4	#3	#2	#1	#0
3706	TCW	CWM	ORM		PCS		PG2	PG1

[Data type] Bit

PCS When multi–spindle control is applied to two tool posts in two–path control, this parameter specifies whether a position coder feedback signal from the other tool post is selectable, regardless of the state of the PC2SLC signal (bit 7 of G028/bit 7 of G1028) of the other tool post:
0 : Not selectable.
1 : Selectable. (To select a position coder for the other tool post, the SLPCA signal (bit 2 of G064) and the SLPCB signal (bit 3 of G064) are used.)

Note 1 Multi–spindle control based on the same serial spindle must be applied to both tool posts. Refer to 9.4 for details.

	#7	#6	#5	#4	#3	#2	#1	#0
3707							P22	P21

P22 and P21: Gear ratio of spindle to second position coder

Magnification	P22	P21
×1	0	0
×2	0	1
×4	1	0
×8	1	1

$$\text{Magnification} = \frac{\text{Number of spindle revolutions}}{\text{Number of position coder revolutions}}$$

	#7	#6	#5	#4	#3	#2	#1	#0
3709						MSI		

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)
1 : For each spindle irrespective of whether the spindle is selected (Each spindle has its own SIND signal).

3772	Maximum spindle speed

[Data type] Word type

[Unit of data] rpm

[Valid data range] 0 to 32767

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.

Note 1 When the constant surface speed control option is selected, the spindle speed is clamped at the maximum speed, regardless of whether the G96 mode or G97 mode is specified.

Note 2 When 0 is set in this parameter, the speed of the spindle is not clamped.

Note 3 When spindle speed command control is applied using the PMC, this parameter has no effect, and the speed of the spindle is not clamped.

Note 4 When the multi–spindle control option is selected (T series), set the maximum speed for each spindle in the following parameters:
Parameter No. 3772: Sets the maximum speed for the first spindle.
Parameter No. 3802: Sets the maximum speed for the second spindle.
Parameter No. 3822: Sets the maximum speed for the third spindle.

3802	Maximum speed of the second spindle

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

Parameter sets the maximum speed for the second spindle.
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.

Note 1 This parameter is valid when the multi–spindle control option is selected.

Note 2 When the constant surface speed control option is selected, the spindle speed is clamped at the specified maximum speed, regardless of whether the G96 mode or G97 mode is specified.

Note 3 When 0 is set in this parameter, the setting of parameter No. 3772 for the first spindle is used.
When 0 is set in parameter NO. 3772, the spindle speed is not clamped.

Note 4 When spindle speed command control is applied using the PMC, this parameter has no effect, and the spindle speed is not clamped.

3811	Maximum spindle speed for gear 1 of the second spindle
3812	Maximum spindle speed for gear 2 of the second spindle

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

Set the maximum spindle speed for each gear of the second spindle.

Note 1 These parameters are used for the multi–spindle control.

3820	Data for adjusting the gain of the analog output of the third–spindle speed

[Data type] Word

[Unit of data] 0.1%

[Valid data range] 700 to 1250

Set the data used for adjusting the gain of the analog output of the third spindle speed.

Note 1 This parameter is used for controlling the multi–spindles.

3821	Offset voltage compensation value of the analog output of the third spindle speed

[Data type] Word

[Unit of data] Velo

[Valid data range] –1024 to 1024

Set the offset voltage compensation value of the analog output of the third spindle speed.

1) Set 0 (standard setting) to this parameter.
2) Command a spindle speed that makes the spindle speed analog output 0.
3) Measure output voltage.
4) Set the following value to parameter No. 3821.

$$\text{Setting value} = \frac{-8191 \times \text{offset voltage (V)}}{12.5}$$

5) After the parameter has been set, command a spindle speed whose analog output becomes 0 and confirm the voltage becomes 0V.

3822	Maximum speed of the third spindle

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

This parameter sets the maximum speed for the third spindle.

When a command specifying a speed exceeding the maximum spindle speed is specified, or the spindle speed exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

Note 1 This parameter is valid when the multi–spindle control option is selected.

Note 2 When the constant surface speed control option is selected, the spindle speed is clamped at the specified maximum speed, regardless of whether the G96 mode or G97 mode is set.

Note 3 When 0 is set in this parameter, the setting of parameter No. 3772 for the first spindle is used. When 0 is set in parameter No. 3772, the spindle speed is not clamped.

Note 4 When spindle speed command control is applied using the PMC, this parameter has no effect, and the speed of the spindle is not clamped.

3831	Maximum spindle speed for gear 1 of the third spindle
3832	Maximum spindle speed for gear 2 of the third spindle

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

Set the maximum spindle speed for each gear of the third spindle.

Note 1 These parameters are used for the multi–spindle control.

Note

Note 1 The spindle orientation signal, spindle speed override signals, and spindle stop signal *SSTP only function for selected signals.

Note 2 If the primary spindle stop signal *SSTP for stopping all selected (SWS1 to SWS3) spindles' rotation is cleared, the speed command is restored. A spindle not selected by SWS1 to SWS3 and rotating at its previous speed, which is stopped using its respective command *SSTP1 to *SSTP3, cannot be restored to that speed when the signal is cleared.

Note 3 The S 12-bit code signals R01O to R12O outputs the state of a selected spindle. If two or more spindles are selected at the same time, the states of the first, second, and third spindles are output in this order.

Note 4 Do not switch between the first and second position coders while a function that uses position coder feedback information is being executed. That is, PMC signal PC2SLC <G028#7> cannot be used while, for instance, a command for feed per rotation or thread cutting is taking place.

Note 5 The multi-spindle function allows two 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 6 Type A multi-spindle control differs from Type B in the relationship between the SWS1 and SIND signals for the first spindle. In Type A, SIND functions only when SWS1 is set to “1”. In Type B, SIND functions whether SWS1 is “1” or “0”; each spindle is selected by either of its respective SWS or SIND signals being set to “1”.

Note 7 An SOR command has priority over S commands and SIND-based rotation control from the PMC, and will cause all selected spindle to perform orientation rotation.

9.11 RIGID TAPPING

9.11.1 General

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 (described in 9.11.2)
- Addition of a sequence to the PMC (described in 9.11.6 and 9.11.7)
- Setting of related parameters (described in 9.11.8)

This section provides an example of M series connection.

To avoid duplicate descriptions, assume the following unless noted otherwise:

- G code for a tapping cycle

  M series: G84 (G74) T series: G84 (G88)
- Gear selection method

  M series: M–type or T–type gear selection method
  T series: 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 5294, No. 5321 to No. 5324, etc.)
- M series: Up to three stages T series: Up to four stages
  (Shared by the second spindle. Up to two stages for the second spindle.)

Notes

1 The description given in this section covers up to the fourth axis.

2 Only when the M–type gear selection method is being used with the M series, the maximum spindle speed for rigid tapping must be set in parameter No. 3743, not parameter Nos. 5241 to 5244, regardless of the number of gear stages.

The descriptions given in this section (such as spindle gear switching and M–type/T–type) are based on the explanation given in Section 9.3. Refer to Section 9.3 as necessary.

Specification of M series/T series

- **Rigid tapping of M series**

The differences in the specifications for rigid tapping for the M series and T series are described below.

The tapping cycle G84 and the reverse tapping cycle G74 can be used to specify M series rigid tapping.

A tapping axis can be arbitrarily selected from the basic axes X, Y, and 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.

For rigid tapping, only the first spindle can be used.

The first spindle allows rigid tapping, even if the spindle is an analog or serial spindle.

- **Rigid tapping of T series**

The face tapping cycle G84 and the side tapping cycle G88 can be used to specify T series 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.

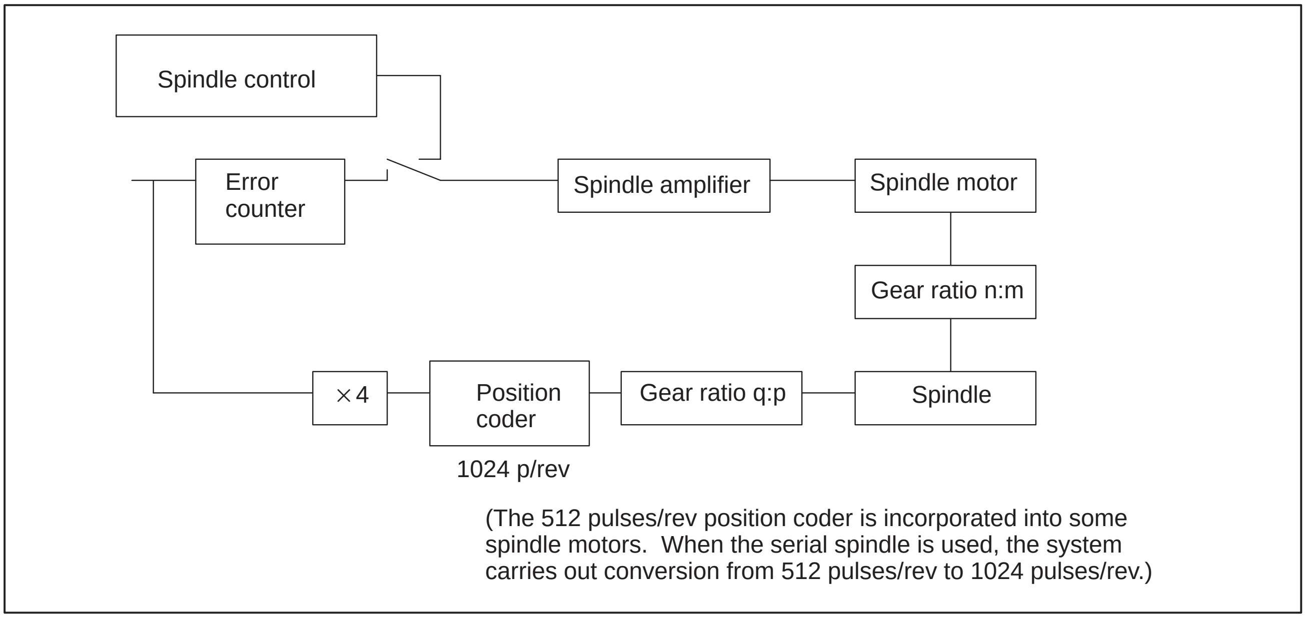
In addition to rigid tapping using the first spindle (either analog or serial), rigid tapping using the second spindle (serial) is also possible when multi–spindle control is being used.

For a two–path lathe, rigid tapping can be performed using a combination of the spindle and tapping axis selected in each path.

Rigid tapping using a mixture of paths is not allowed. For example, rigid tapping in combination of a tapping axis of tool post 1 and the spindle of tool post 2, by issuing a tapping command to tool post 1, is not supported.

9.11.2 Connection among Spindle, Spindle Motor, and Position Coder

As shown in the figure below a gear can be inserted between the spindle and spindle motor, and between the spindle and position coder.



(1) Gear between spindle and spindle motor

Up to three gear stages (M series) or four gear stages (1st spindle of T series), two gear stages (2nd spindle of T series) can be provided between the spindle and the spindle motor. The gear ratio is arbitrary. The spindle move distance per spindle motor rotation is different, based on the gear ratio. The speed command to the spindle motor must be adjusted. See (2), below, for additional information regarding a spindle motor incorporating a position coder.

(2) Gear between spindle and position coder

The position coder is used to detect the position of the spindle.
The gear ratio for the spindle and position coder is specified in the parameter sets No. 5221 to No. 5223 and No. 5231 to No. 5233, or parameter set PG1 and PG2 No. 3706 #0, #1, parameter P21, P22 (No. 3707 #0, #1) for 2nd spindle of T series. Which parameter set to use is specified by parameter VGR No. 5200 #1.

· **Arbitrary gear ratio (VGR=1)**

This is used if the gear ratio for the spindle motor and position coder (built-in or separate) is not 1:1, 1:2, 1:4, or 1:8, set VGR to 1 and set the gear ratio using parameter No. 5221 to 5224.

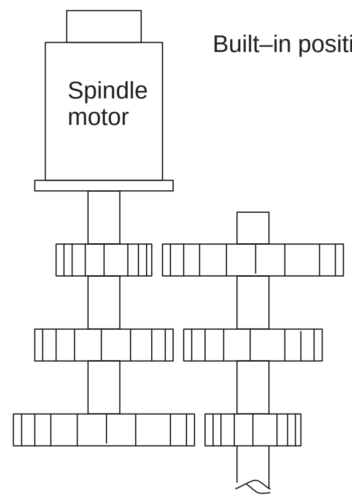
When position coder is mounted on a spindle, the gear ratio for the spindle motor and position coder cannot be changed by shifting the spindle motor and spindle gears. Parameters No. 5221 to 5224 must all specify the same value for the teeth of the individual spindle gears. Parameters No. 5231 to 5234 must all specify the same value for the teeth of individual position coder gears.

The 1024 or 512 pulses/rev position coder is built into the spindle motor. For the 512 pulses/rev version, specify double the number of teeth on each gear for the position coder. (Double the number of teeth need not be specified for the serial spindle.)

The M series allows up to three stages, regardless of which gear selection method has been selected. (Parameter Nos. 5224 and 5234 cannot be used.)

The T series supports up to two stages. (Set parameter Nos. 5221, 5222, 5231, and 5232.)

Example)



Parameter No.	Set value: 512p/rev Position coder	Set value: 1024p/rev Position coder	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	60 Note)	30	Number of teeth of the 1st gear for the position coder side
5232	100 Note)	50	Number of teeth of the 2nd gear for the position coder side
5233	140 Note)	70	Number of teeth of the 3rd gear for the position coder side

Note
Double value setting is not required for serial spindle.

- **Gear ratio is 1:1, 1:2, 1:4, 1:8 (VGR=0)**

If the gear ratio is either 1:1, 1:2, 1:4, and 1:8, it is set using parameters PG1 and PG2 (Nos. 3706 #0, #1). This applies if the position coder is mounted in a spindle or built into a spindle motor when only one stage gear is provided. .
For 2nd spindle of T series, set it to parameter P21, P22 (No.3707#0, #1).

Parameter		Gear ratio		Detection unit
PG2	PG1	Spindle	Position coder	
0	0	1	1	360/4096=0.08789 deg
0	1	1	2	360/4096×2=0.17578 deg
1	0	1	4	360/4096×4=0.35156 deg
1	1	1	8	360/4096×8=0.70313 deg

The spindle motor building in the 512 pulses/rev position coder uses the values set forth in the following table. A serial spindle does not require double-value setting; use the same values as for the spindle motor building in the 1024 pulses/rev position coder.

	Gear ratio		Parameter		Gear ratio of spindle to position coder	Detection unit (deg)
	Spindle motor	Spindle	PG2	PG1		
Built–in position coder 512p/rev	1	1	0	1	1:2	0.17578
	2	1	1	0	1:4	0.35156
	4	1	1	0	1:8	0.70313

(3) Rigid tapping and machines with multiple gears

If the M type gear selection method is selected, the CNC determines whether gears need changing using the gear change specification mentioned in section 9.3. If the gears need to be changed, the CNC generates the S function code read signal SF (F007#2) and gear selection signals GR10, GR20, and GR30 (F034#0-#2) to notify the PMC. Change gears using the PMC, based on these signals.

If the T type gear selection method is selected, the CNC does not process gear changes. When the CNC has the S function code, it outputs signal SF and S function code signals S00 to S31 (F022#0-F025#7) to the PMC. (However, parameter No. 3705 and its related parameters need to be set for S code and SF signal output). Using the PMC, determine whether gears need changing, and make the change if needed. Input gear selection signals GR1 and GR2 <G028#1,#2> or GR21 <G029#0> for 2nd spindle of T series for the selected gear, and notify the CNC of them.

To perform rigid tapping with the serial spindle, enter the clutch/gear selection signals CTH1 and CTH2 (G070#3,#2 for the first spindle, and G074#3, #2 for the second spindle) from the PMC. Notify the serial spindle control unit of these signals via the CNC, irrespective of the gear selection method.

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 9.3 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 $= \dfrac{\text{Maximum spindle motor speed} \times L\%}{\text{Low–speed gear ratio}}$
Medium speed gear	Maximum low–speed gear speed + 1 revolution	Maximum medium–speed gear speed $= \dfrac{\text{Maximum spindle motor speed} \times L\%}{\text{Medium speed gear ratio}}$
High–speed gear	Maximummedium–speed gear speed +1 revolution	Maximum high–speed gear speed $= \dfrac{\text{Maximum spindle motor speed} \times L\%}{\text{High–speed gear ratio}}$

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 =1).

Gear	Spindle speed range (during rigid tapping)	
	Lower limit	**Upper limit**
Low–speed gear	1 revolution	Maximum low–speed gear speed $= \dfrac{\text{Basic spindle motor speed} + \alpha}{\text{Low–speed gear ratio}}$
Medium speed gear	Maximum low–speed gear speed + 1 revolution	Maximum medium–speed gear speed $= \dfrac{\text{Basic spindle motor speed} + \alpha}{\text{Medium–speed gear ratio}}$
High–speed gear	Maximummedium–speed gear speed +1 revolution	Maximum high–speed gear speed $= \dfrac{\text{Basic spindle motor speed} + \alpha}{\text{High–speed gear ratio}}$

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 = 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 9.3, "Spindle Control" for details of the spindle gear change specifications.

The loop gain can be specified for each gear. Specify "0" for parameter No. 5280 and specify loop gains for each gear for parameter Nos. 5281 to 5284. Unless "0" is specified for parameter No. 5280, the loop gains for each gear are disabled, and the loop gain common to all gears, the value of parameter No. 5280, is enabled.

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 extraction time constant for each gear set. Specify the extraction time constant for each gear in parameter Nos. 5271 to 5274.

If bit 1 (VGR) of parameter No. 5200 is set to "1", the gear ratio for the spindle and position coder can be set to anywhere between 1:32767 and 32767:1 in one-increment units for three gear sets with M series, four gear sets with T series, or two gear sets with 2nd spindle of T series. However 1:8 to 8:1 is the recommended value.

9.11.3 Rigid Tapping Specification

- **Feed rate**

In rigid tapping mode, the tapping axis is fed at a rate specified by F; the spindle speed is $S \times 360$(deg/min). Override is invalid for both of them. An override of up to 200% can be applied to withdrawal operations by setting bit 4 (DOV) of parameter No. 5200 to "1", and setting an appropriate override value in parameter RGOVR of No. 5211. The time constant for withdrawal operations can be modified by bit 1 (TDR) of parameter No. 5201; when it is set to "1", the values in parameter Nos. 5271 to 5274 are used as the time constant for withdrawal.

- **Acceleration and deceleration**

Linear acceleration/deceleration is valid for rigid tapping.

- **Override**

Override is invalid for rigid tapping. Fixed override can be applied to withdrawal operations by setting bit 4 (DOV) of parameters No. 5200 or RGOVR of No. 5211.

- **Dry run**

Dry run is valid for G84 (G74). When the dry run is applied to the tapping axis speed of G84 (G74), tapping is performed. The spindle speed will match the dry run speed.

- **Machine lock**

Machine lock is valid for G84 (G74).
When G84 (G74) is executed with the machine locked, however the tapping 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, but G84 (G74) mode is not reset.

- **Feed hold, interlock, and single block**

The feed hold, interlock, and single block functions are effective for G84 (G74).
The feed hold and single block functions in rigid tapping mode can be nullified by setting bit 1 (FHD) of parameter No. 5200 to "1".
As with the machine lock signal, the feed hold and single block functions are also effective for the spindle indirectly, through tapping axis operations.

- **Operation mode**

G84 (G74) can be executed only in the MEM and MDI modes.

- **Manual feed**

Rigid tapping cannot be performed in the manual feed mode.

- **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 backlash compensation is normally made for the tapping axis.

9.11.4 Display Data on the Diagnosis Screen

For rigid tapping adjustment, the diagnosis screen displays information related to rigid tapping.

For part of the display data, the user can choose between two sets of data items relating to the synchronization of the spindle and tapping axis by setting bit 0 (DGN) of parameter No. 5204.

- **Common display data**

The following information items are displayed, regardless of the setting of bit 0 (DGN) of parameter No. 5204:

· Spindle position deviation → Diagnosis No. 450

· Number of command pulses distributed to the spindle (momentary value) → Diagnosis No. 451

· Cumulative number of command pulses distributed to the spindle → Diagnosis No. 454

- **Display of rigid tapping synchronization error (When DGN = 0)**

When bit 0 (DGN) of parameter No. 5204 is set to 0, the following information items are displayed.

(Diagnosis Nos. 452 and 453 are not displayed.)

· Spindle–converted move command difference → Diagnosis No. 455

· Spindle–converted position deviation difference → Diagnosis No. 456

· Synchronization error range → Diagnosis No. 457

Spindle–converted move command difference

$$= \Sigma \frac{\text{spindle move command}}{\text{gear ratio}} - \Sigma \frac{(\text{tapping axis move command}) \times 4096}{\text{thread lead}}$$

Spindle–converted position deviation difference

$$= \frac{\text{spindle position deviation}}{\text{gear ratio}} - \frac{(\text{tapping axis position deviation}) \times 4096}{\text{thread lead}}$$

Synchronization error range

= (maximum spindle–converted move position deviation difference on the positive side)

– (maximum spindle–converted position deviation difference on the negative side)

If a maximum allowable synchronization error range is set in parameter No. 5214, the position deviation alarm during spindle movement (alarm No. 741) 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.)

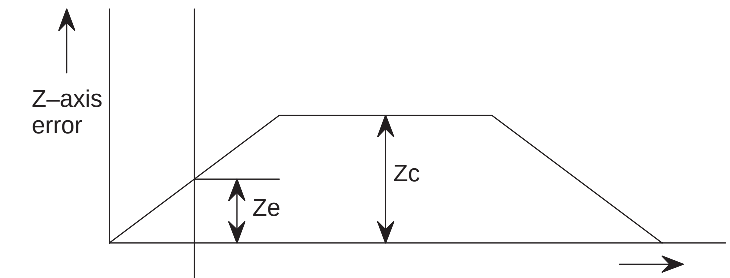
- **Rigid tapping error display (error difference display) (When DGN = 0)**

When bit 0 (DGN) of parameter No. 5204 is set to 1, the following information items are displayed. (Diagnosis Nos. 455, 456, and 457 are not displayed.)

· Momentary error difference between the spindle and tapping axis → Diagnosis No. 452

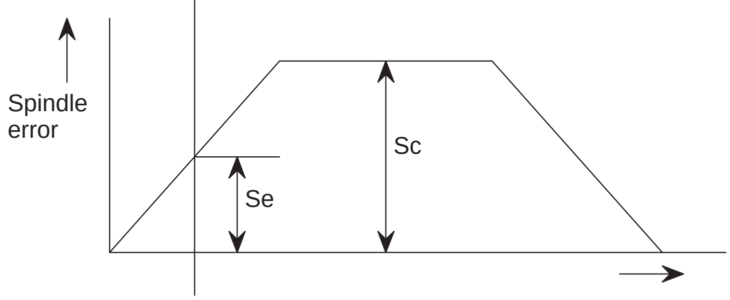
· Maximum error difference between the spindle and tapping axis → Diagnosis No. 453

Diagnosis No. 0452 is cleared to “0” when rigid tapping mode is set or canceled, and diagnosis No. 0453 is cleared to “0” in the positioning of the rigid tapping cycle.
The following figure shows the tapping axis as the Z axis.



$$Zc = \frac{\text{Speed}}{60} \times \frac{1}{\text{Gain}} \times \frac{1}{\text{Detection unit}} \times 10^2 \ \text{(Theoretical value)}$$

Ze = Z–axis error counts (measured value)



$$Sc = \frac{\text{Speed} \times 360}{60} \times \frac{1}{\text{Gain}} \times \frac{1}{\text{Detection unit}} \times 10^2 \ \text{(Theoretical value)}$$

Se = Spindle error counts (measured value)

Speed : mm/min or inch/min
Gain : 0.01 s^{-1}
Detection unit :mm, inch, or deg
Speed : rpm

$$\Delta Z = \frac{Ze}{Zc} \times 100 \ [\%]$$

$$\Delta S = \frac{Se}{Sc} \times 100 \ [\%]$$

The error difference between the spindle and Z axis can be obtained by ΔS–ΔZ. This value is sampled at intervals of 64 ms.

Diagnosis screen

- **Spindle position deviation**

0450	SPINDLE MOTION ERROR

Spindle position deviation during rigid tapping

[Unit] Pulse

- **Number of pulses distributed to the spindle**

0451	SPINDLE MOTION PULSE

Number of pulses distributed to the spindle during rigid tapping

[Unit] Pulse

- **Error difference between the spindle and tapping axis (momentary value)**

0452	RIGID ERROR

Momentary error difference between the spindle and tapping axis during rigid tapping (signed)

[Unit] %

Note 1 This data item is displayed only when bit 0 (DGN) of parameter No. 5204 is set to 1.

- **Error difference between the spindle and tapping axis (maximum)**

0453	RIGID ERROR(MAX)

Maximum error difference between the spindle and tapping axis during rigid tapping (absolute value)

[Unit] %

Note 1 This data item is displayed only when bit 0 (DGN) of parameter No. 5204 is set to 1.

- **Cumulative number of pulses distributed to the spindle during rigid tapping**

0454	SPINDLE PULSE(SUM)

Cumulative number of pulses distributed to the spindle during rigid tapping

[Unit] Pulse

- **Spindle–converted move command difference during rigid tapping (momentary value)**

0455	SYNC. PULSE(SUM)

Momentary spindle–converted move during command difference between the spindle and the tapping axis during rigid tapping

[Unit] Pulse

Note 1 This data item is displayed only when bit 0 (DGN) of parameter No. 5204 is set to 0.

- **Spindle–converted position deviation difference during rigid tapping (momentary value)**

0456	SYNC. ERROR

Momentary spindle–converted position deviation difference between the spindle and the tapping axis during rigid tapping

[Unit] Pulse

Note 1 This data item is displayed only when bit 0 (DGN) of parameter No. 5204 is set to 0.

- **Synchronization error range during rigid tapping (momentary value)**

0457	SYNC. WIDTH

Synchronization error range during rigid tapping (maximum value)

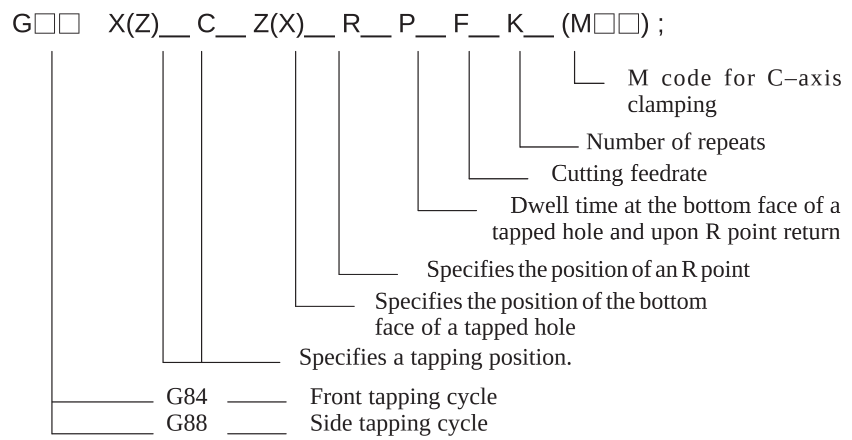
[Unit] Pulse

Note 1 This data item is displayed only when bit 0 (DGN) of parameter No. 5204 is set to 0.

9.11.5
Command format

Command format for the T series

The rigid tapping command format for the T series is described below. For an explanation of the command format used with the M series, refer to Section II.4.2 of the "FANUC Series 16–MB Operator's Manual."



The rigid tapping mode can be specified by using any 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.

Rigid tapping mode can be canceled by G80;. Note, however, that a G code for another canned cycle, or a group 01 G code can also cancel rigid tapping mode.

When rigid tapping is terminated by a command issued to cancel rigid tapping mode, the spindle stops. (Output to the spindle is equivalent to the specification of S0.)

A reset (by means of the RESET button or an external reset) can also cancel rigid tapping mode. Note, however, that canned cycle mode is not canceled by a reset.

- Specifying M29 before a block containing G84 (G88)

```
M29 S****;
G¢¢X (Z) __C__Z (X) __R__P__F__K__ (M¢¢) ;
X (Z) __C__;
X (Z) __C__;
 .
 .
G80;
```

Rigid tapping mode

- Specifying M29 and G84 (G88) in the same block (Note, however, that M29 and M□□ for C–axis clamping cannot be specified in the same block.)

```
G¢¢X (Z) __Z (X) __R__P__F__K__M29****;   ┐
X (Z) __C__;                              │ Rigid
X (Z) __C__;                              │ tapping
 .                                        │ mode
 .                                        │
G80;                                      ┘
```

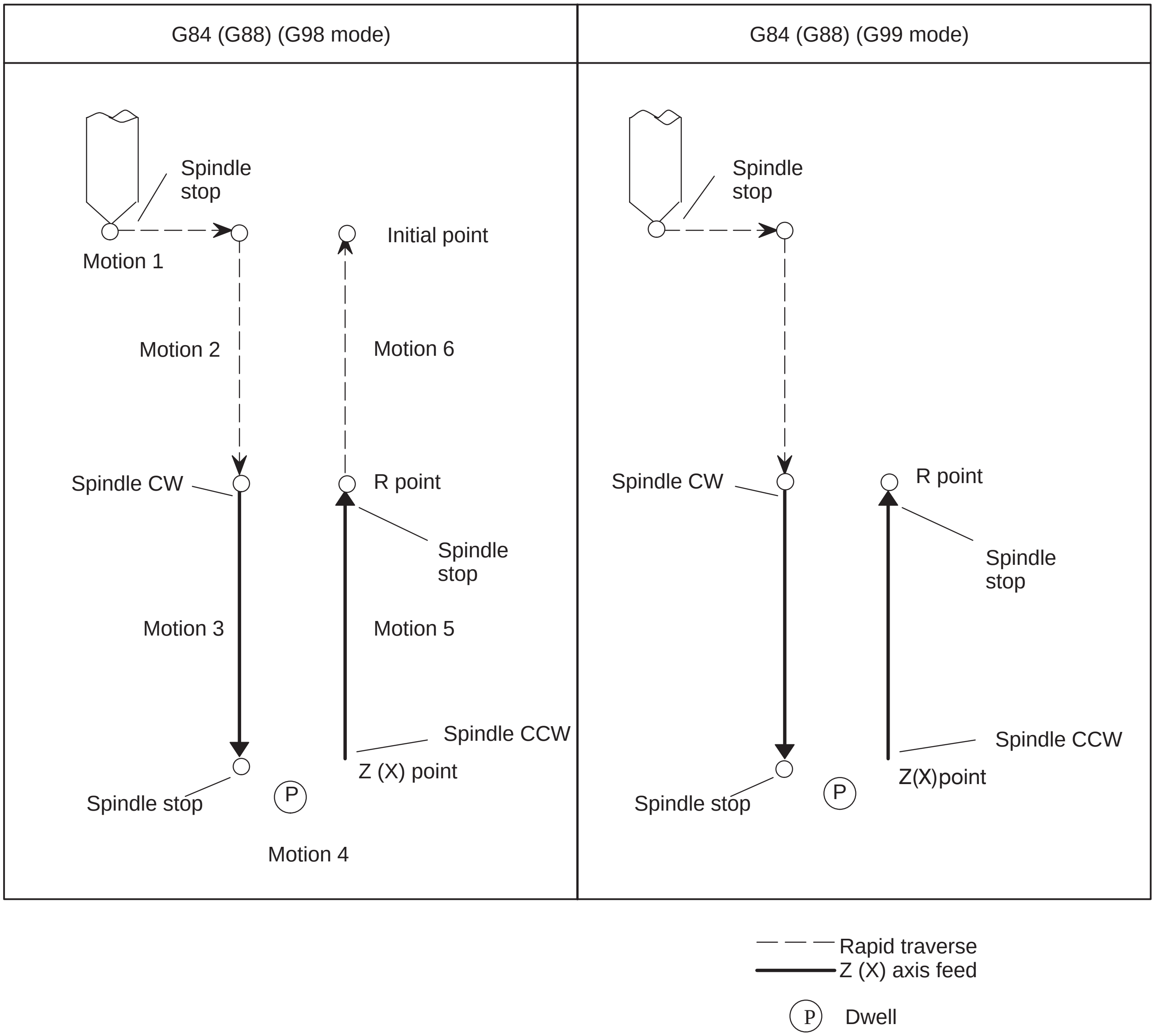
- Converting G84 (G88) to a G code for rigid tapping (by setting bit 0 (G84) of parameter No. 5200 to 1)

```
G¢¢X (Z) __C__Z (X) __R_P_F_K_S**** (M¢¢);   ┐
X (Z) __C__;                                 │ Rigid
X (Z) __C__;                                 │ tapping
 .                                           │ mode
 .                                           │
G80;                                         ┘
```

Notes on the T series

- In feed per minute mode, F_/S**** determines a thread lead. In feed per rotation mode, F_ specifies a thread lead.
- 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, P/S alarm No. 200 is issued in a block containing G84 (G88).
- F_ must specify a value that does not exceed the maximum cutting feedrate. When 0 is specified, P/S alarm No. 201 is issued.
- 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, P/S alarm Nos. 203 and 204 are issued, respectively.

G84·G85 (Tapping cycle)



Notes

1 G code system A does not include G88 (return to initial level) and G99 (return to R point level). Return to the initial level is always used.

2 During cutting feed along the Z–axis (X–axis), the feedrate override is assumed to be 100%. The spindle speed override is also assumed to be 100%. For a retract motion (motion 5), a fixed override of up to 200% can be applied by specifying bit 4 (DOV) of parameter No. 5200 and parameter No. 5211 (RGOVR).

Rigid tapping in feed per rotation mode

Rigid tapping is classified into two types: rigid tapping in feed per rotation mode (G99) and rigid tapping in feed per minute mode (G98).

Example)

The example below specifies rigid tapping in feed per rotation mode for cutting a thread with a lead of 1 mm at a spindle speed of 1,000 rpm.

```
O0001 ;
G99 ;
  .
  .
  .
M29  S1000 ;
G84  Z–100.   R–20.   F1. ;
  .
  .
  .
G80 ;
```

The example below specifies rigid tapping in feed per minute mode for cutting the same thread at the same spindle speed as above. (In feed per minute mode, F/S determines the thread lead.)

```
O0002 ;
G98 ;
  .
  .
  .
M29  S1000 ;
G84  Z–100.   R–20.   F1000 ;
  .
  .
  .
G80 ;
```

Units of F

	Metric input	Inch input	Remarks
G98	1 mm/min	0.01 inch/min	A fractional value can be specified.
G99	0.0001 mm/ rev	0.000001 inch/rev	A fractional value can be specified.

Notes

1 G98 and G99 are modal G codes. Upon power–up, G99 (feed per rotation mode) is set.

2 Even in feed per rotation mode, a pulse distribution command is converted to a feed per minute command. Thus, feed per rotation mode does not strictly implement feed per rotation. Accordingly, even if the spindle stops for some reason, the tapping axis (Z–axis or X–axis) does not stop.

9.11.6 Signal

9.11.6.1 Signals for the rigid tapping function

Rigid tapping signal RGTAP <G061#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) block.

Spindle rotation direction signals RGSPM, RGSPP <F065#1, #0> (M series only)

[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.

RGSPM 1 : Rotating in the reverse direction (CCW).
0 : Not rotating in the reverse direction.

[Output condition] 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 <F076#3>

[Classification] Output signal

[Function] This signal notifies the PMC that rigid tapping mode is set.

RTAP 1 : Rigid tapping mode is currently set.
0 : Rigid tapping mode is not currently set.

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.

9.11.6.2 Signals related to S code output

Spindle enable signal ENB <F001#4>

Second spindle enable signal ENB2 <F001#4> (T series only)

[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 <F022 to F025>

Spindle–speed function strobe signal SF <F007#2>

[Classification] Output signal

[Function] These signals send S codes specified for the CNC, in binary format, to the PMC.

[Output condition] When an S code is specified, the specified value is output, in binary format, with the signals. Upon the completion of output, 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 = 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

Note 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.

9.11.6.3 Signals related to gear switching

Gear selection signals (output) GR30, GR20, GR10 <F034#2, #1, #0> (M series only)

[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).

As 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 signals.

Reference information: The table below indicates the relationship between the output signals and gear selection.

	GR3O	GR2O	GR1O
1st (low) speed gear	×	×	;
2nd (medium) speed gear	×	;	×
3rd (high) speed gear	;	×	×

Gear selection signals (input) GR2, GR1 <G028#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.

	GR2	GR1
1st (low) speed gear	×	×
2nd (medium) speed gear	×	;
3rd (high) speed gear	;	×
4th (high) speed gear	;	;

← 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.

Gear selection signal (input) GR21 <G029#0> (T series only)

[Classification] Input signal

[Operation] When rigid tapping with the second spindle is being performed (for the T series only), the signal is used in a PMC sequence.

The signal notifies the CNC of spindle gear information when the second spindle has been selected.

The input signal is related to gear selection as described below.

GR21 1 : The second stage is currently selected as the second spindle gear.
0 : The first stage is currently selected as the second spindle gear.

	GR21
1st speed gear	×
2nd speed gear	;

When a serial spindle is used, the serial spindle clutch/gear selection signals (G070#3, #2 for the first spindle, and G074#3, #2 for the second spindle) must be set in addition to the setting of the gear selection signal described above.

9.11.6.4 Signals related to second spindle rigid tapping (T series only)

Gear selection signal (input)
GR21 <G029#0>
(T series only)

Signals related to multi–spindle control (T series only)

See the description of the signals related to gear switching, given above.

Spindle selection signals
SWS1, SWS2
<G027#0, #1>

Rigid tapping spindle selection signals
RGTSP2, RGTSP1
<G061#5, #4>

[Classification] Input signal

[Operation] SWS1 and SWS2 are used to transfer spindle commands when the multi–spindle control option 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.)

RGTSP2 and RGTSP1 are used to select a spindle used for rigid tapping, independently of the SWS1 and SWS2 signals, when the multi–spindle control option is being used. (The RGTSP2 and RGTSP1 signals can be used when bit 7 (SRS) of parameter No. 5200 is set to 1.)

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”	“1”
First spindle	“1”	“0”
Second spindle	“0”	“1”
P/S alarm No. 205 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”	“1”
First spindle	“1”	“0”
Second spindle	“0”	“1”
P/S alarm No. 205 is issued.	“0”	“0”

Note
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.

Spindle–by–spindle stop signals *SSTP1, *SSTP2 <G027#3, #4>

[Classification] Input signal

[Operation] These signals are used to stop each spindle when the multi–spindle control option 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 does not specify 0 rpm.
0 : The output to the first spindle specifies 0 rpm.

***SSTP2** 1 : The output to the second spindle does not specify 0 rpm.
0 : The output to the second spindle specifies 0 rpm.

Second position coder selection signal PC2SLC <G028#7>

[Classification] Input signal

[Operation] This signal is used to select the second position coder when the multi–spindle control option is being used. Note, however, that it cannot be used with a spindle selected to perform rigid tapping.

1 : Control is exercised using a feedback pulse signal from the second position coder.
0 : Control is exercised using a feedback pulse signal from the first position coder.

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.

9.11.6.5 Signal addresses

	#7	#6	#5	#4	#3	#2	#1	#0
G027				*SSTP2	*SSTP1		SWS2	SWS1
G028	PC2SLC					GR2	GR1	
G029								GR21
G061			RGTSP2	RGTSP1				RGTAP
F001				ENB				
F007						SF		
F034						GR3O	GR2O	GR1O
F038						ENB2		
F065							RGSPM	RGSPP
F076					RTAP			

9.11.6.6 Notes on interface with the PMC

The following describes some notes in designing the interface with the PMC.

Rigid tapping mode management and ENB (or ENB2)

The PMC must manage rigid tapping mode as follows: rigid tapping mode is set using M29, and is canceled upon the issue of a reset or at the falling edge of the spindle enable signal ENB in rigid tapping mode. ENB is used during rigid tapping in this way, so the spindle stop signal *SSTP must not be set to “0”.

However, *SSTP and SOR may be used for gear switching. To do so, ensure that the PMC does not cancel rigid tapping mode on a falling edge of ENB while *SSTP is “0”. Rigid tapping mode may be set on a rising edge of the RTAP signal instead of by using M29, and canceled on a falling edge of the RTAP signal instead of the ENB signal.

In rigid tapping using the second spindle with the T series, the ENB2 signal must be used for rigid tapping mode management.

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”.

For the same reason, the PMC's control over second spindle output must be disabled in rigid tapping mode by setting SIND2 to “0”.

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) 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. S**** is executed in a G84 block. Spindle output is equivalent to the specification of S0.

The timing chart is shown in the chart indicating the execution of G84 (G74).

Execution of G84 (G74)

When M29S****; is specified, S**** is read in, spindle output being equivalent to the specification of S0;. S**** is output when G84 (G74) 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 S**** is outside the previously selected gear range, the spindle–speed function strobe signal SF <F007#2> and gear selection signals (output) GR30, GR20, GR10 <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 stage is selected, the system assumes that the third gear has been selected.

T series: Up to four gear stages are supported for the first spindle, and up to two gear stages for the second spindle. (The gear selection for the second spindle is notified to the CNC by the signal GR21 <G029#0>.)

An S code is output in the first block (positioning to tapping position) of G84 (G74) execution. However, the spindle motor position loop is closed in the next block (R point positioning). Accordingly, spindle speed offset must be adjusted accurately until the position loop has been closed in the second block of G84 (G74) execution after the PMC activates the spindle motor with M29. Otherwise, the spindle motor may rotate slightly. (This applies only to an analog spindle. No offset adjustment is required for a serial spindle.)

Rigid tapping mode may be specified by specifying M29 before G84, specifying M29 and G84 in the same block, or by specifying G84 as a G code for rigid tapping. In each case, PMC processing is the same. (The M29 code is always output.)

9.11.7 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 (Fig. 9.11.7.1 (a) to Fig. 9.11.7.3 (d)) and apply the information it contains as necessary.

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. 9.11.7.1 (a)	Fig. 9.11.7.2 (a)	Fig. 9.11.7.3 (a)
	Performed	Fig. 9.11.7.1 (b)	Fig. 9.11.7.2 (b)	Fig. 9.11.7.3 (b)
T–type	Not performed	Fig. 9.11.7.1 (c)	Fig. 9.11.7.2 (c)	Fig. 9.11.7.3 (c)
	Performed	Fig. 9.11.7.1 (d)	Fig. 9.11.7.2 (d)	Fig. 9.11.7.3 (d)

Note

For more information about the M/T type gear selection method, see Section 9.3 SPINDLE CONTROL. Note the following:

T series:	<u>T–type</u>	only
M series:	<u>M–type</u>	when constant surface speed control is not being used and bit 4 (GTT) of parameter No. 3706 is set to 0
	<u>T–type</u>	when constant surface speed control is being used, or bit 4 (GTT) of parameter No. 3706 is set to 1

9.11.7.1
When M29 is specified before G84 (G74)

M type gear selection method

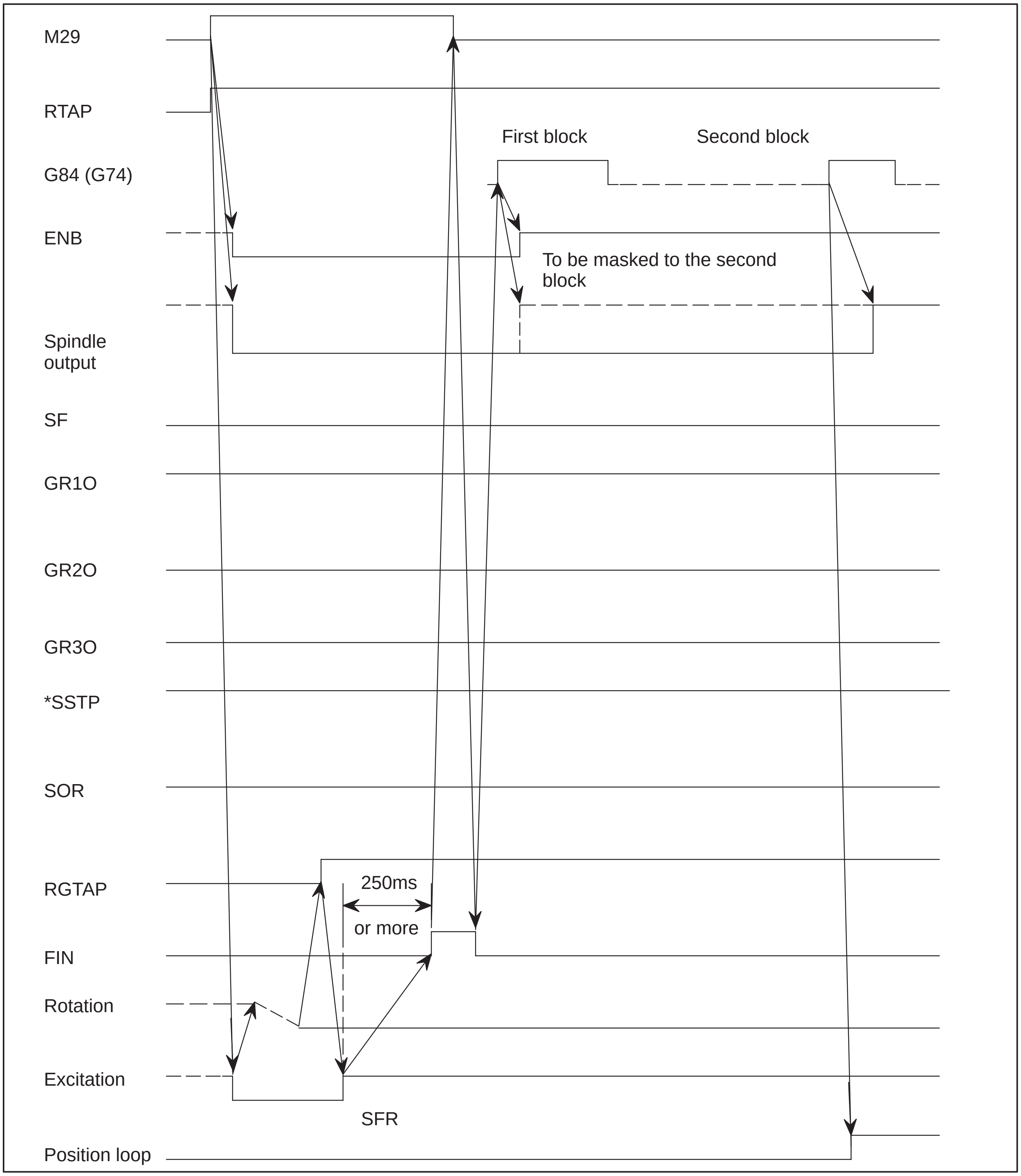
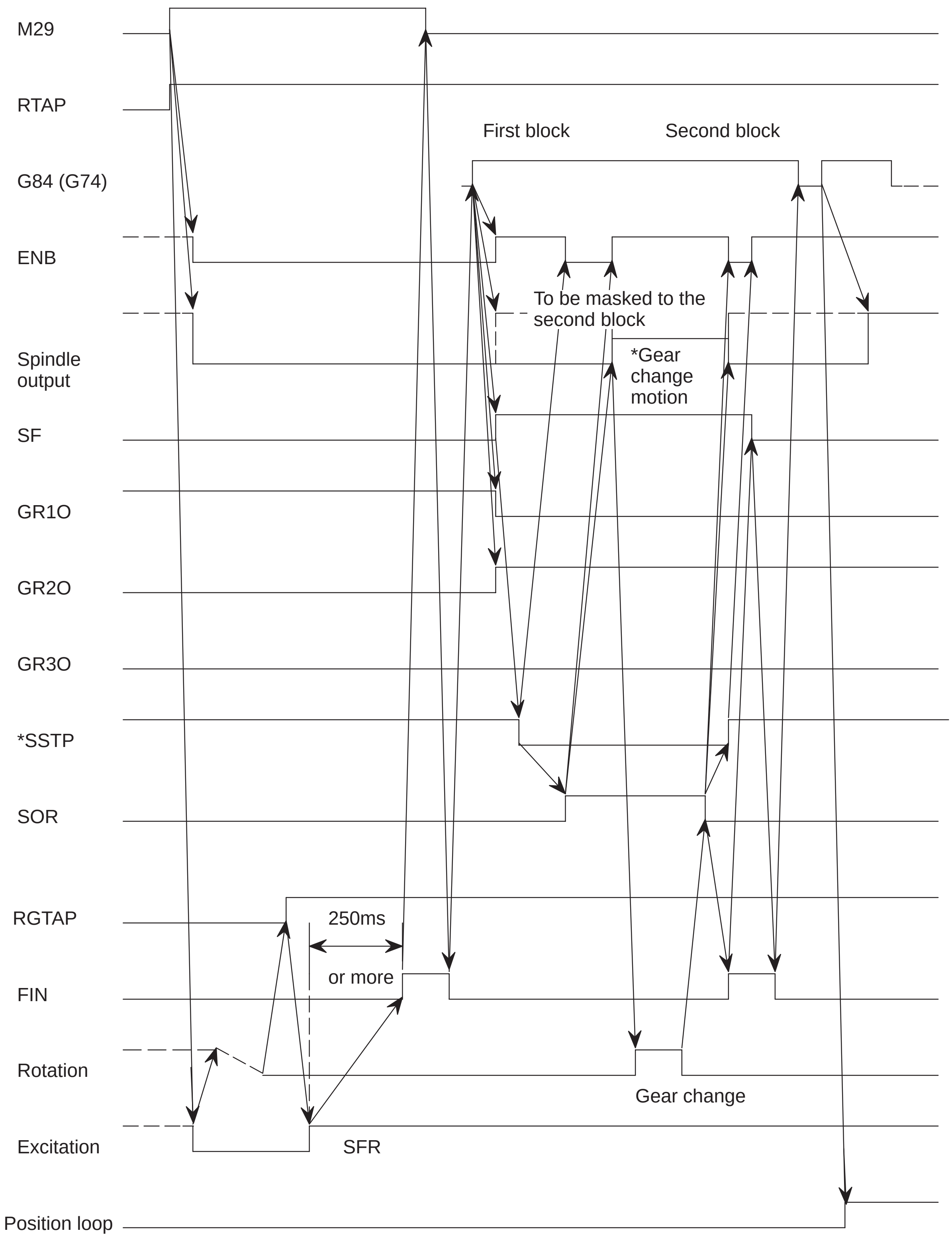


Fig. 9.11.7.1 (a) Gear is not changed



Note 1 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.

Fig. 9.11.7.1 (b) When gear change is performed (From low to middle gear)

T type gear selection method

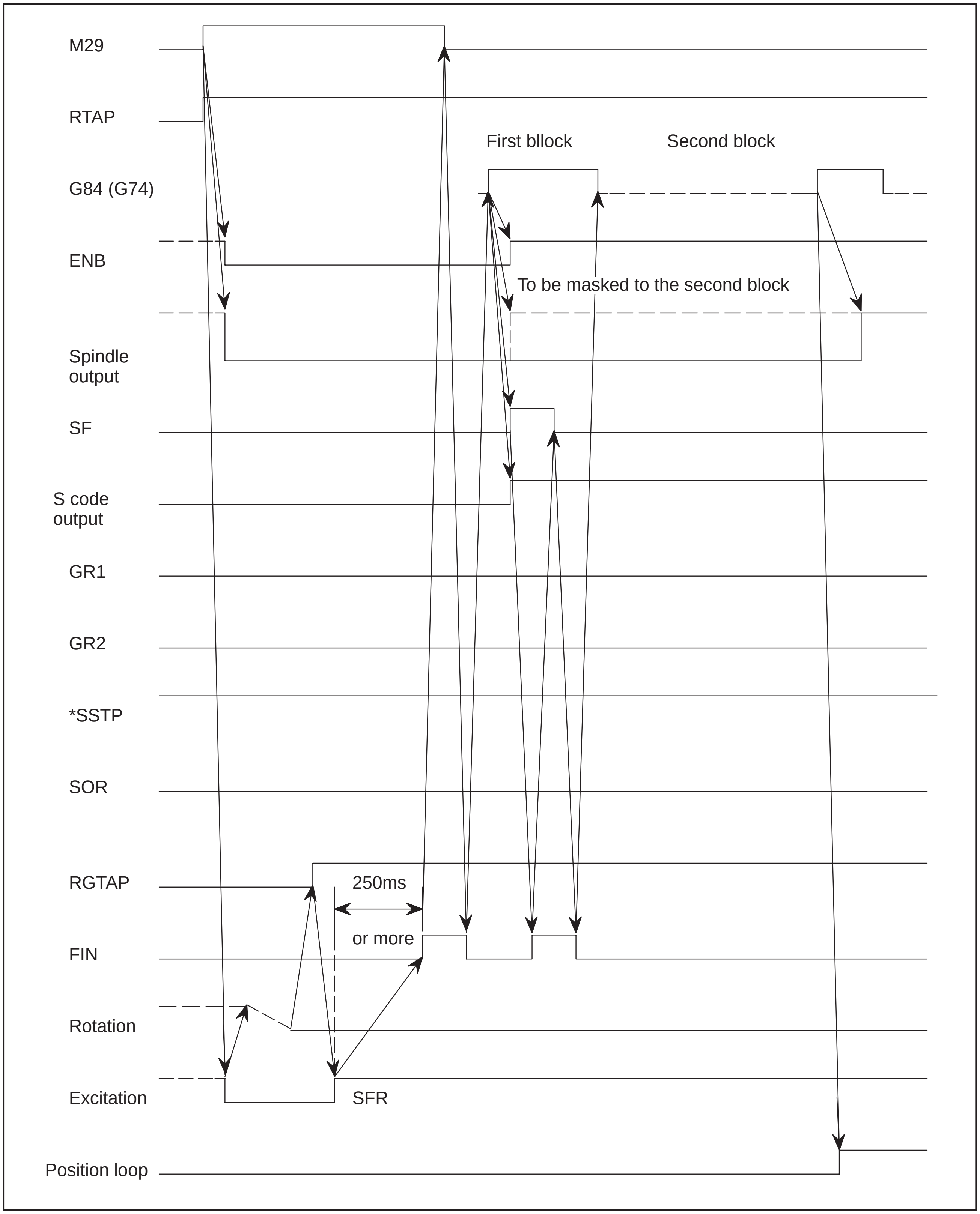
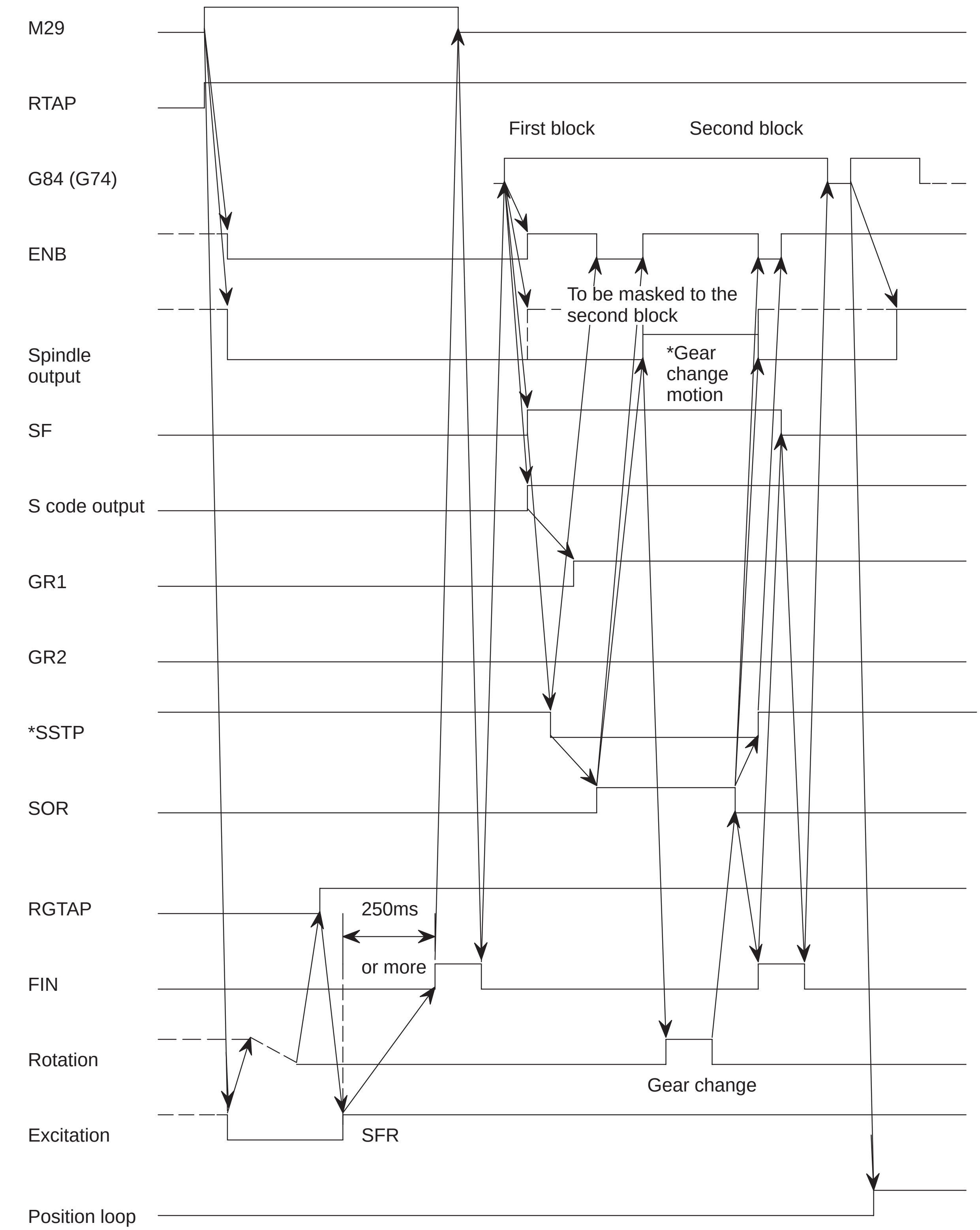


Fig. 9.11.7.1 (c) Gear change is not performed



Note 2 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.

Fig. 9.11.7.1 (d) When gear–change is performed (low to middle gear)

9.11.7.2 M29 and G84 (G74) are specified in the same block

M type gear selection

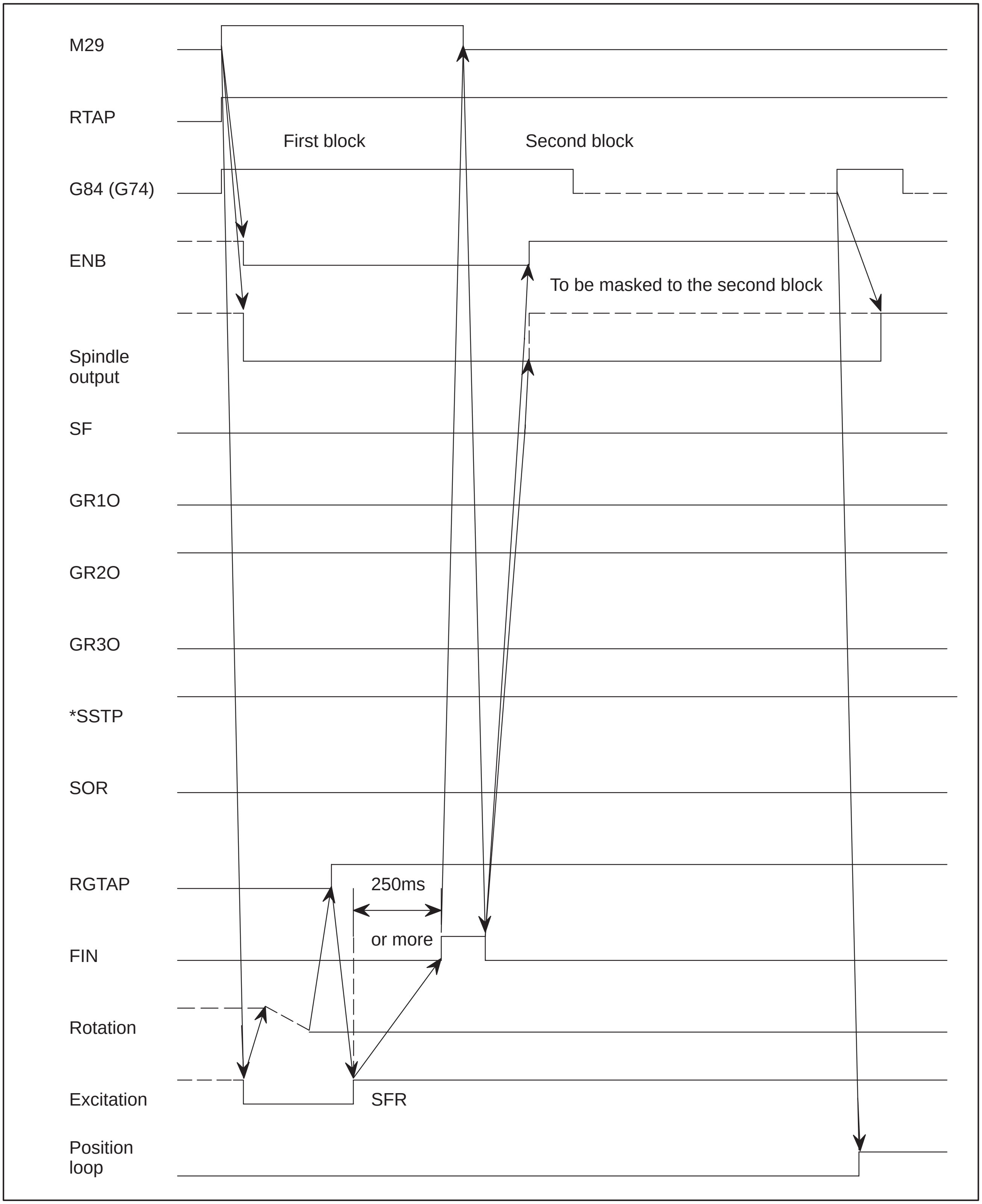
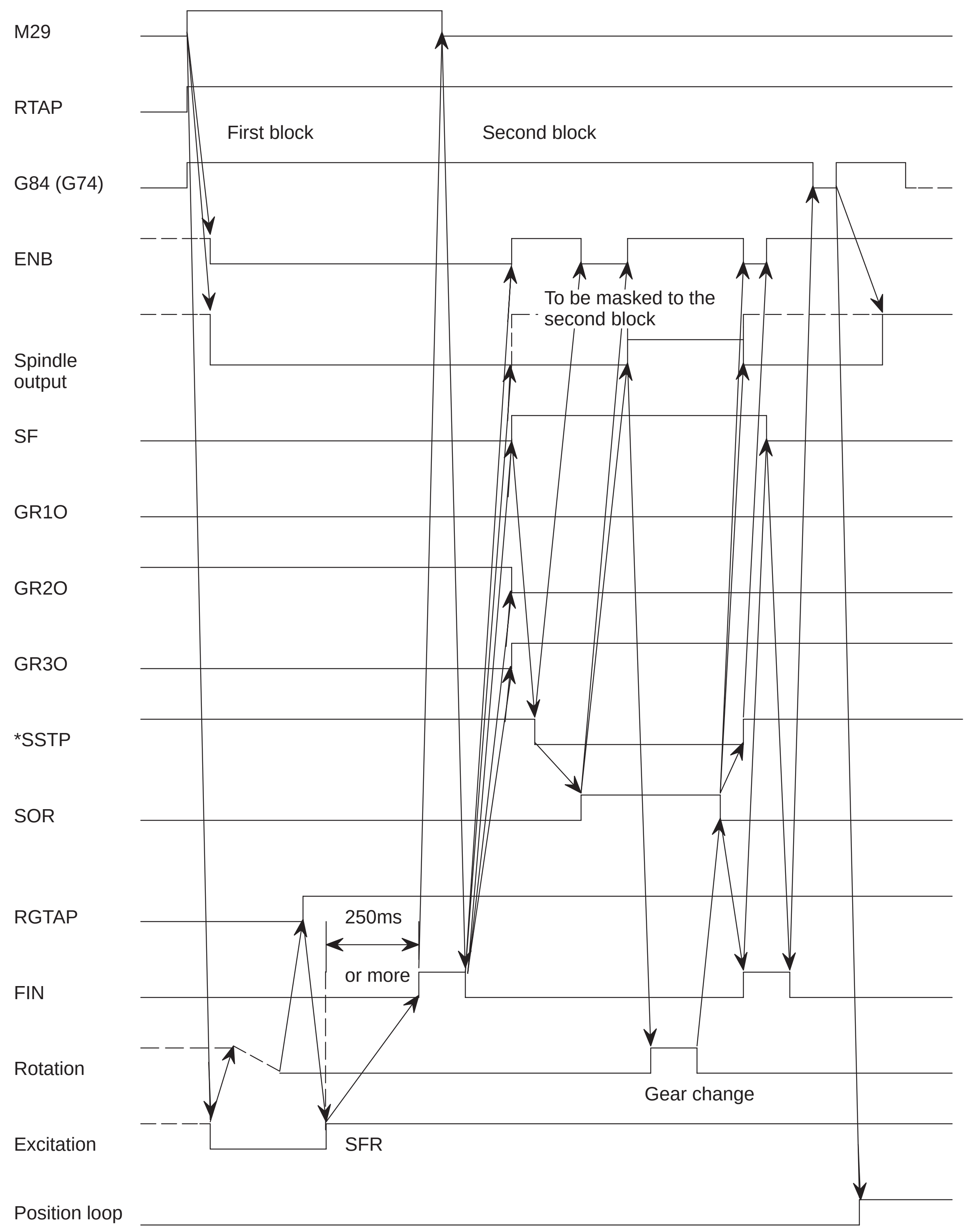


Fig. 9.11.7.2 (a) When gear–change is not performed



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.

Fig. 9.11.7.2 (b) When gear–change is performed (middle to high)

T type gear selection method

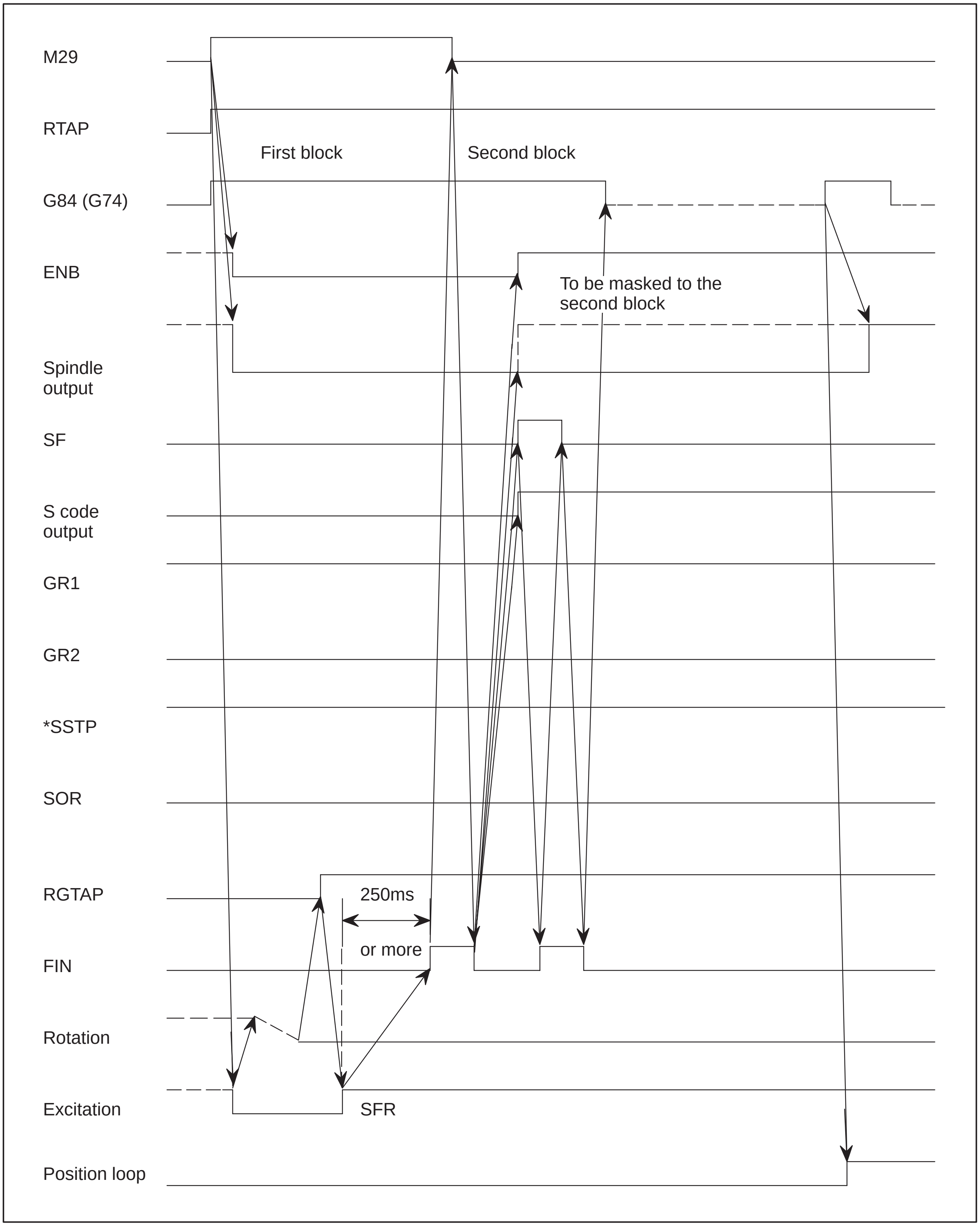
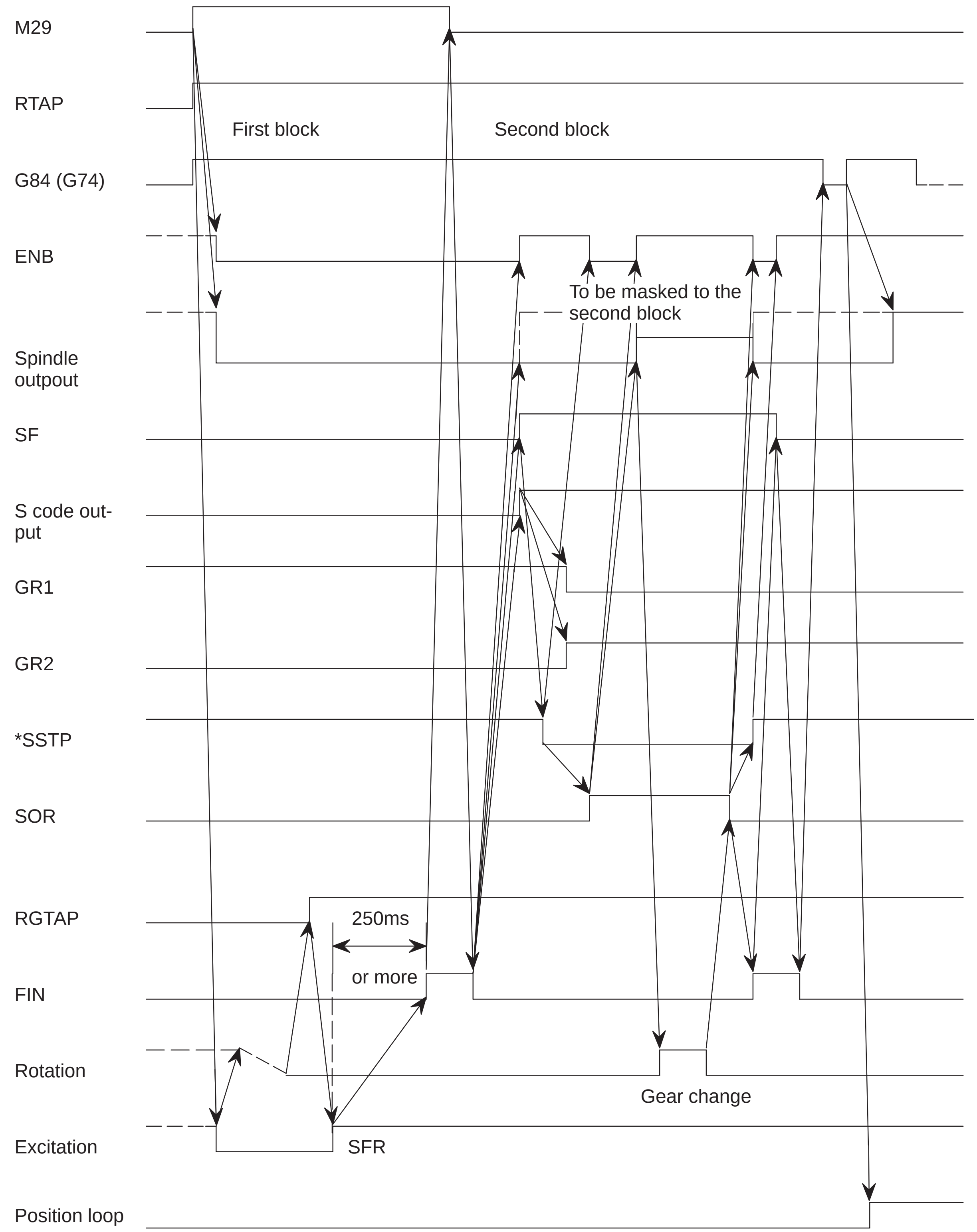


Fig. 9.11.7.2 (c) When gear change is not performed



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.

Fig 9.11. 7.2 (d) When gear–change is performed (middle to high gear)

9.11.7.3
Specifying G84 (G74) for rigid tapping by parameters

M type gear selection

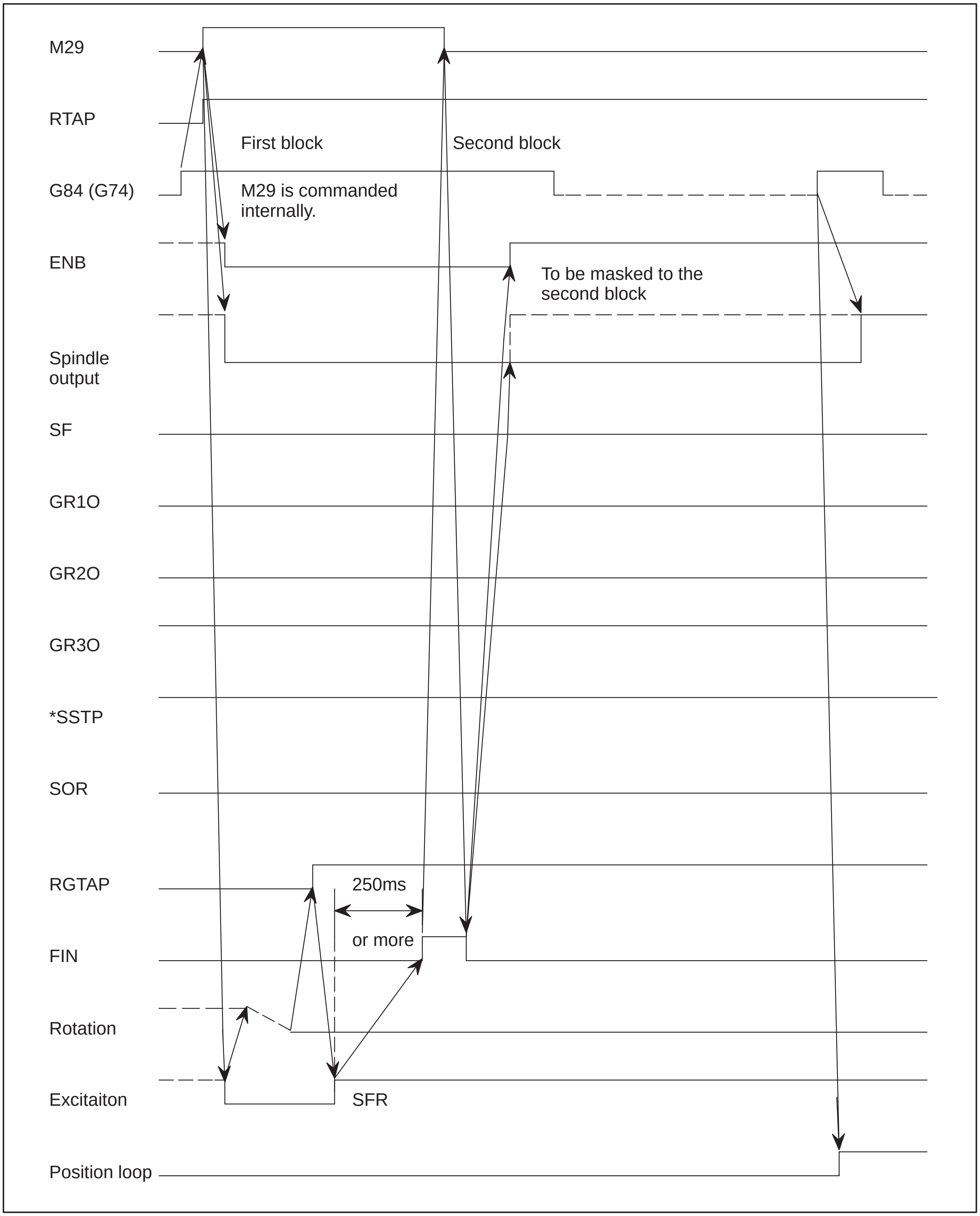
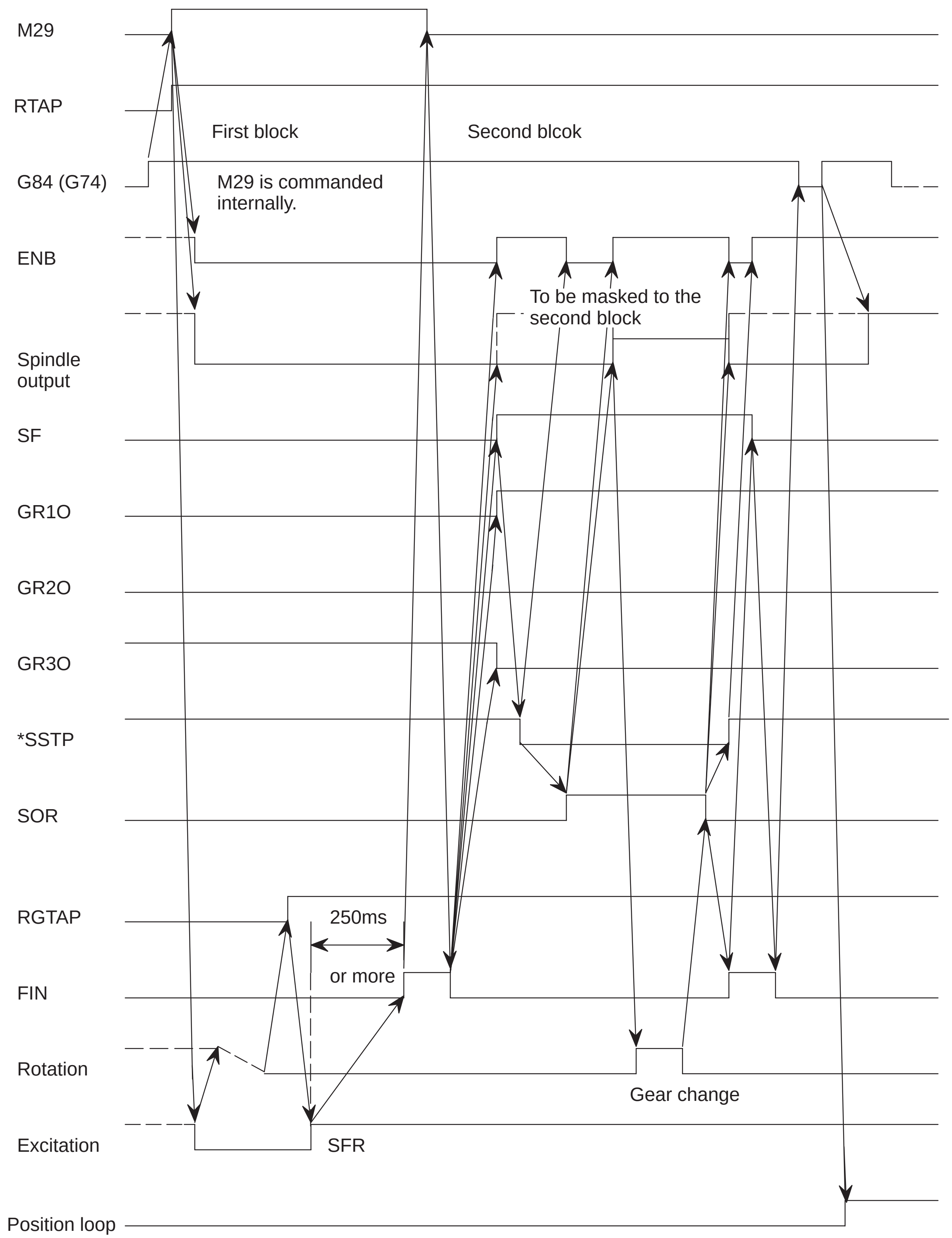


Fig. 9.11.7.3 (a) When gear–change is not performed



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.

Fig. 9.11.7.3 (b) When gear change is performed (high to low gear)

Type gear selection method

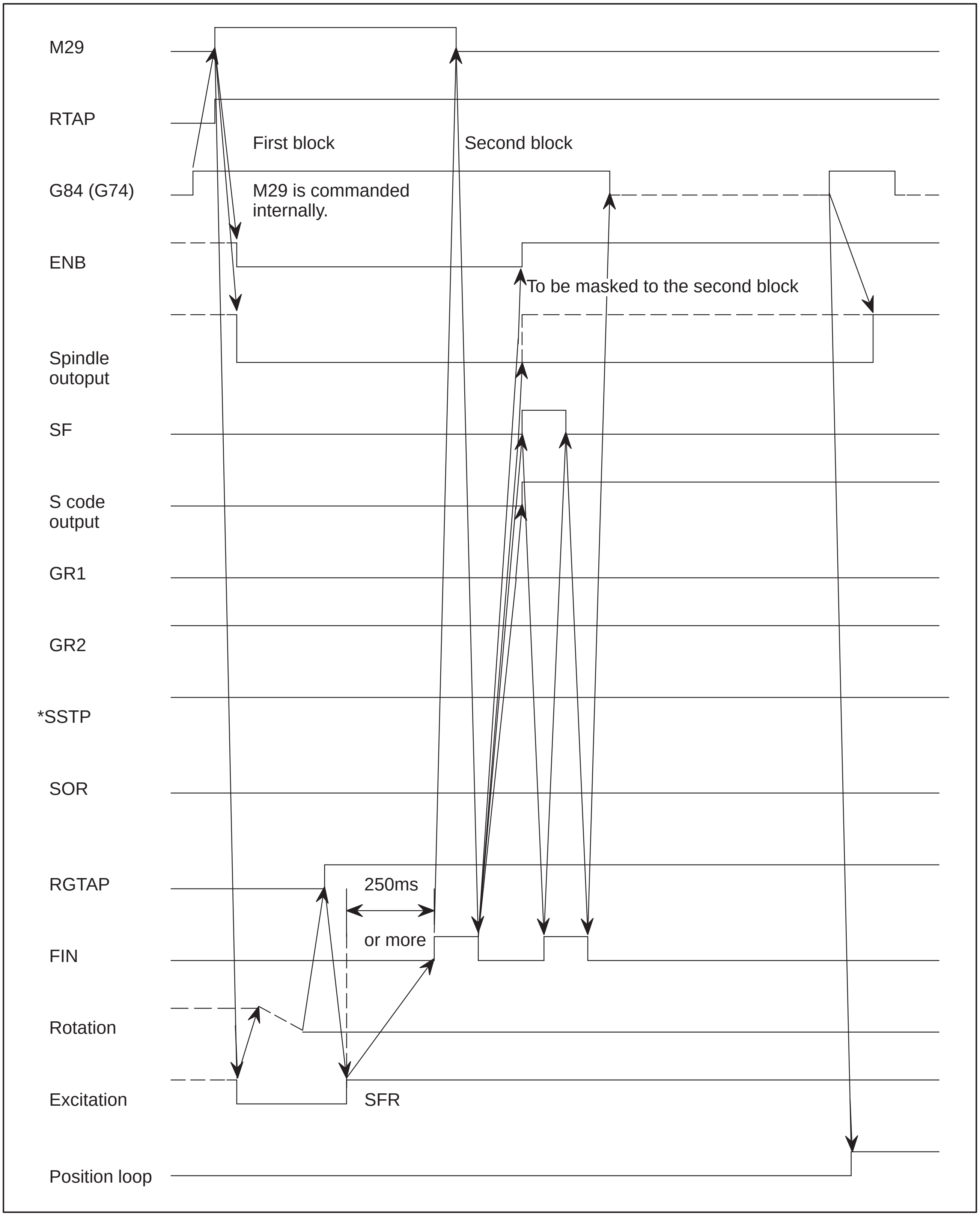
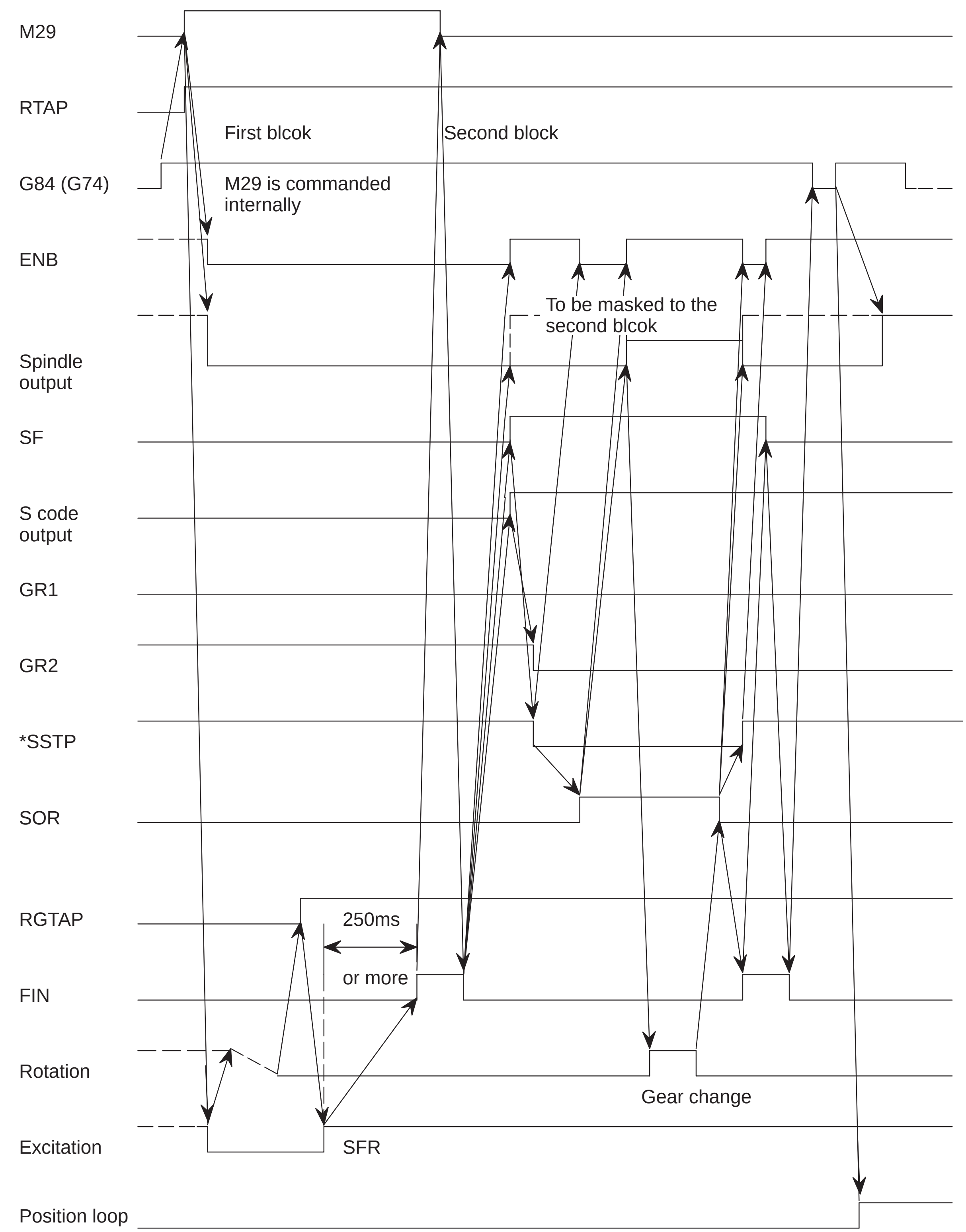


Fig. 9.11.7.3 (c) When gear change is not performed



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.

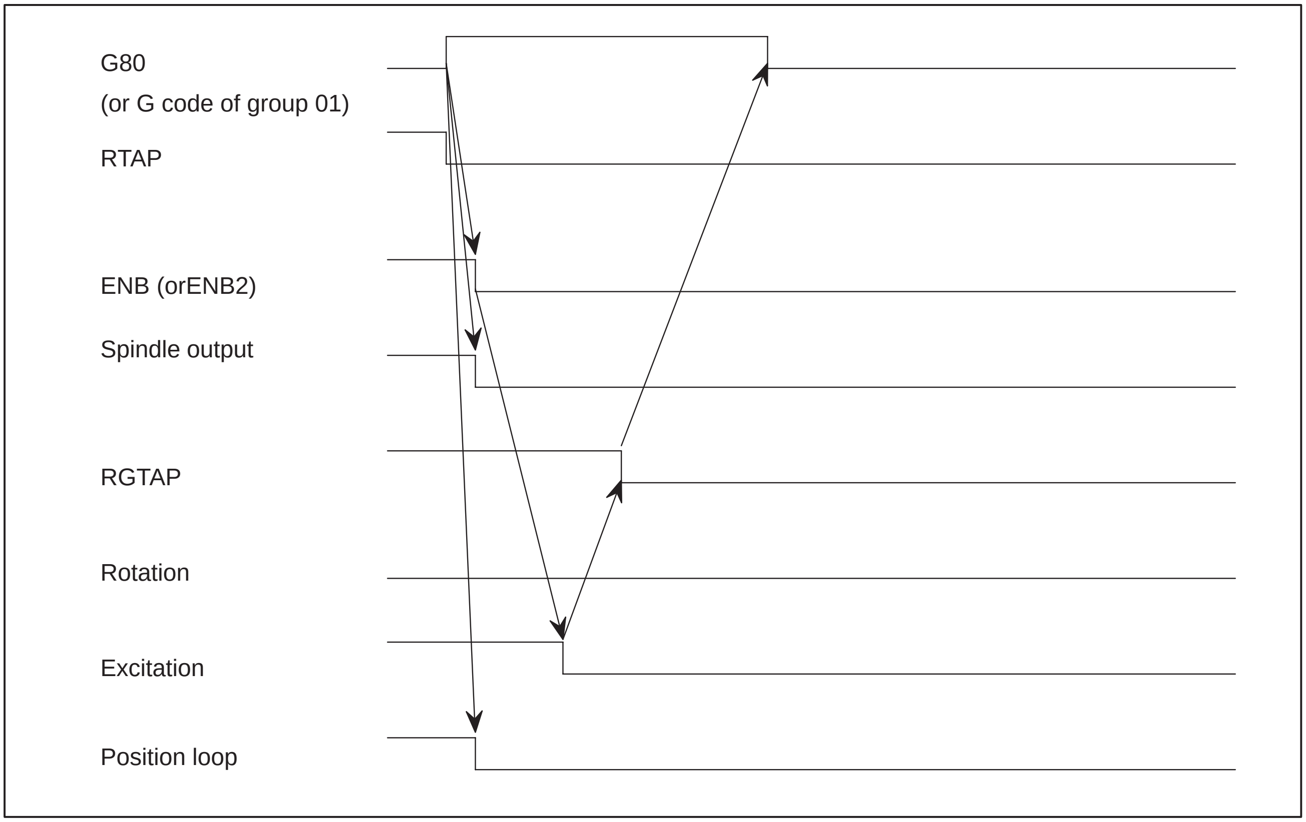
Fig. 9.11.7.3 (d) When gear–change is performed (high to low gear)

9.11.7.4 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.
The spindle output is produced in the same way as executing S0. Cancel the PMC rigid tapping mode at the falling edge of the ENB signal (ENB2 signal for 2nd spindle of T series) by de-energizing the spindle; then turn off the rigid tapping mode signal. The system goes to the next block after confirming that the signal is off.

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”.



Notes

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 a 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

3 Rigid tapping mode is canceled as described above regardless of the gear selection method of M–type or T–type.

9.11.8 Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS				ESF
		SFA	NSF		SGT			ESF

[Data type] Bit

ESF When the spindle control function (S analog outpu or S serial output) is used, and the consatant surface speed control function is used or bit 7 (GTT) of parameter No. 3705 is set to 1:
0 : S codes and SF are output for all S commands.
1 : S codes and SF are not output for an S command in constant surface speed control mode (G96 mode) or for an S command used to specify maximum spindle speed clamping (G50S–––;).

Note 1 For the T system, this parametar is enabled when bit 4 (EVS) of parameter No. 3705 is set to 1.
For the M series, SF is not output:

· For an S command used to specify maximum spindle speed clamping (G92S–––;) in constant surface speed control mode

· When bit 5 (NSF) of parameter No. 3705 is set to 1

SGT Gear switching method during tapping cycle (G84, G74)
0 : Method A (Same as normal gear switching method)
1 : Method B (Gears are switched during tapping cycle according to the spindle speed set in parameters 3761 and 3762)

EVS When the spindle control function (S analog output or S serial output) is used, S codes and SF are:
0 : Not output for an S command.
1 : Output for an S command.

Note 1 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.

NSF: When an S code command is issued in constant surface–speed control,
0 : SF is output.
1 : SF is not output:

SFA: The SF signal is output:
0 : When gears are switched
1 : Irrespective of whether gears are switched

3706	#7	#6	#5	#4	#3	#2	#1	#0
							PG2	PG1
				GTT			PG2	PG1

[Data type] Bit

PG2, PG1 Gear ratio of spindle to position coder

Magnification	PG2	PG1
×1	0	0
×2	0	1
×4	1	0
×8	1	1

$$\text{Magnification} = \frac{\text{Number of spindle revolutions}}{\text{Number of position coder revolutions}}$$

GTT Selection of a spindle gear selection method
0: Type M
1 : Type T

Note 1 Type M:
The gear selection signal is not entered. In response to an S command, the CNC selects a gear according to the speed range of each gear specified beforehand in a parameter. Then the CNC reports the selection of a gear by outputting the gear selection signal. The spindle speed corresponding to the gear selected by the gear selection signal is output.
Type T:
The gear selection signal is entered. The spindle speed corresponding to the gear selected by this signal is output.

Note 2 When the constant surface speed control option is selected, type T is selected, regardless of whether this parameter is specified.

3761	
	Spindle speed when switching from gear 1 to gear 2 during tapping

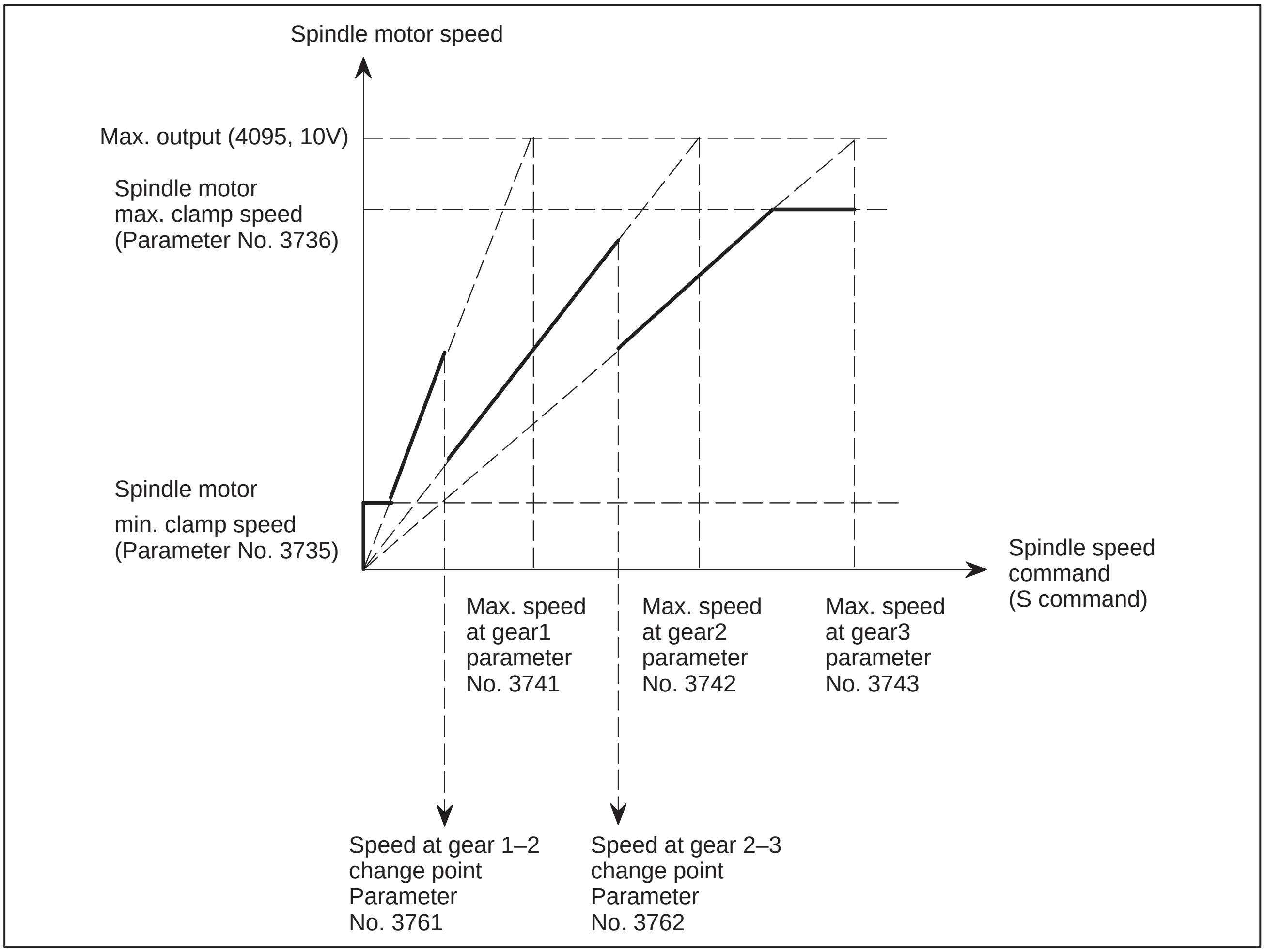
3762	
	Spindle speed when switching from gear 2 to gear 3 during tapping

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

When method B is selected (SGT,#3 of parameter 3705, is set to 1) for the tapping cycle gear switching method, set the spindle speed when the gears are switched.



	#7	#6	#5	#4	#3	#2	#1	#0
5101								
								FXY

[Data type] Bit

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 M series, this parameter is valid only for the drilling canned cycle in the Series 15 format.

	#7	#6	#5	#4	#3	#2	#1	#0
5200	SRS	FHD		DOV	SIG	CRG	VGR	G84
		FHD	PCP	DOV	SIG	CRG	VGR	G84

[Data type] Bit

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.)

VGR Any gear ratio between spindle and position coder in rigid tapping
0 : Not used (The gear ratio is set in parameter No. 3706.)
1 : Used (The gear ratio is set by parameters Nos. 5221 through 5224 and 5231 through 5234.)

CRG Rigid mode when a rigid mode cancel command is specified (G80, G01 group G code, reset, etc.)
0 : Canceled after rigid mode signal RGTAP is set to 0.
1 : Canceled before rigid mode signal RGTAP is set to 0.

SIG When gears are changed for rigid tapping, the use of SIND <G032 and G033> is
0 : Not permitted.
1 : Permitted.

DOV Override during extraction in rigid tapping
0 : Invalidated
1 : Validated (The override value is set in parameter No. 5211.)

PCP Rigid tapping
0 : Used as a high–speed peck tapping cycle
1 : Not used as a high–speed peck tapping cycle

FHD Feed hold and single block in rigid tapping
0 : Invalidated
1 : Validated

SRS To select a spindle used for rigid tapping in multi–spindle control:
0 : The spindle selection signals SWS1 and SWS2 (bits 0 and 1 of G027) are used. (These signals are used also for multi–spindle control.)
1 : The rigid tapping spindle selection signals RGTSP1 and RGTSP2 (bits 4 and 5 of G061) are used. (These signals are provided expressly for rigid tapping.)

	#7	#6	#5	#4	#3	#2	#1	#0
5201						TDR		
						TDR		NIZ

[Data type] Bit

NIZ Smoothing in rigid tapping is:
0 : Not performed.
1 : Performed.

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

	#7	#6	#5	#4	#3	#2	#1	#0
5202								
								ORI

Note 1 When this parameter is set, the power must be turned off before operation is continued.

[Data type] Bit

ORI When rigid tapping is started:
0 : Spindle orientation is not performed.
1 : Spindle orientation is performed.

Note 1 This parameter can be used only for a serial spindle.

The spindle orientation is a zero return in the serial spindle servo mode The stop position can be changed by serial spindle parameter No. 4073.

	#7	#6	#5	#4	#3	#2	#1	#0
5204								DGN

Note 1 When this parameter is set, the power must be turned off before operation is continued.

[Data type] Bit

DGN On the diagnosis screen:
0 : A rigid tapping synchronization error is displayed. (Nos. 455 to 457)
1 : An error difference between the spindle and tapping axis is displayed. (Nos. 452 and 453)

5210	Rigid tapping mode specification M code

[Data type] Byte

[Valid data range] 0 to 255

This parameter sets an M code that specifies the rigid tapping mode. To set an M code larger than 255, set it to parameter No. 5212.

Note 1 The M code is judged to be 29 (M29) when “0” is set.

5211	Override value during rigid tapping extraction

[Data type] Byte

[Unit of data] 1 %

[Valid data range] 0 to 200

The parameter sets the override value during rigid tapping extraction.

Note 1 The override value is valid when DOV in parameter No. 5200 is “1”.

5212	M code that specifies a rigid tapping mode

[Data type] Two–word

[Unit of data] Integer

[Valid data range] 0 to 65535

This parameter sets the M code that specifies the rigid tapping mode.

The M code that specifies the rigid tapping mode is usually set by parameter 5210. To use an M code whose number is greater than 255, specify the code number with parameter 5212.

Note 1 If the setting of this parameter is 0, the M code specifying the rigid tapping mode is determined by the setting of parameter 5210. Otherwise, it is determined by the setting of parameter 5212. The setting of parameter 5212 must always be within the above valid range.

5213	Return or clearance in peck tapping cycle

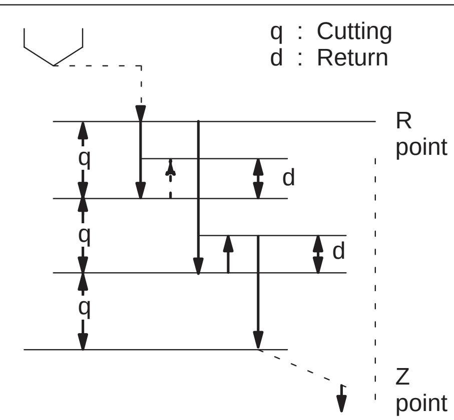
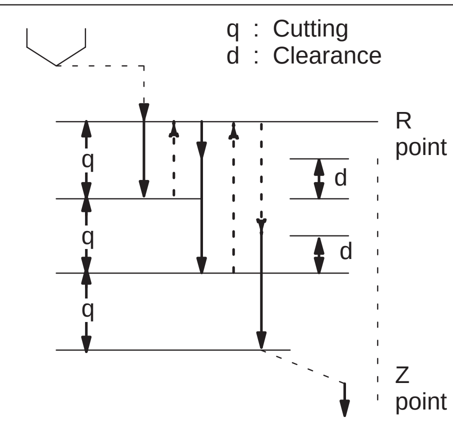
[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 32767

This parameter sets the return or clearance in the peck tapping cycle.

When bit PCP in parameter No. 5200 is 0 (High–speed peck tapping cycle)	When bit PCP in parameter No. 5200 is 1 (Peck tapping cycle)
	

5214	Rigid tapping synchronization error range setting

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter sets an allowable synchronization error range for rigid tapping.

When the synchronization error exceeds the allowable range set with this parameter, alarm No. 741 is issued. Note that when 0 is set with this parameter, no synchronization error check is performed.

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)

[Data type] Word

[Valid data range] 1 to 32767

These parameters set the number of gear teeth on the spindle side for every gear when any gear ratio is set in rigid tapping.

Note 1 This parameter is valid when VGR, #1 of parameter No. 5200, is “1”.

Set the same value to parameter Nos. 5221 to 5224 when the spindle has a position coder.

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)

[Data type] Word

[Valid data range] 1 to 32767

These parameters set the number of gear teeth on the position coder side for every gear when any gear ratio is set in rigid tapping.

Note 1 This parameter is valid when VGR, #1 of parameter No. 5200, is “1”.

Set the same value to parameter Nos. 5231 to 5234 when the spindle has a position coder.

A spindle motor incorporating the position coder uses a position coder with 2048 pulses per revolution. In this case, set the value that is two times as many as the actual number of gear teeth (because of conversion to 4096 pulses per revolution).

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)

[Data type] Two–word

[Unit of data] rpm

[Valid data range] Spindle and position coder gear ratio

1 : 1 0 to 7400
1 : 2 0 to 9999
1 : 4 0 to 9999
1 : 8 0 to 9999

These parameters set the maximum spindle speed for every gear in rigid tapping.

Note 1 In a system having one–stage gear, set the same value as parameter No. 5241 to parameter No. 5243. In a system having two–stage gear, set the same value as parameter No. 5242 to parameter No. 5243. If it is not set as such, P/S alarm no. 200 will be informed.
These are applicable to M series.

5261	Acceleration/deceleration time constant for every gear in rigid tapping (First gear)
5262	Acceleration/deceleration time constant for every gear in rigid tapping (Second gear)
5263	Acceleration/deceleration time constant for every gear in rigid tapping (Third gear)
5264	Acceleration/deceleration time constant for every gear in rigid tapping (Fourth gear)

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

These parameters set the spindle and tapping axis's time constant for every gear during linear acceleration/deceleration in rigid tapping.

Set the time required until a spindle speed reaches the maximum spindle speed (parameter Nos. 5241 and greater). The actual time constant is a proportional value between the maximum spindle speed and the specified spindle speed.

5271	Acceleration/deceleration time constant during extraction in rigid tapping (First gear)
5272	Acceleration/deceleration time constant during extraction in rigid tapping (Second gear)
5273	Acceleration/deceleration time constant during extraction in rigid tapping (Third gear)
5274	Acceleration/deceleration time constant during extraction in rigid tapping (Fourth gear)

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

These parameters set the linear acceleration/deceleration time constant of a spindle and tapping axis for every gear during extraction in rigid tapping.

Note 1 The time constant is valid when TDR, #2 of parameter No. 5201, is “1”.

5280	Position control loop gain of spindle and tapping axis in rigid tapping (Common in each gear)

[Data type] Word

[Unit of data] 0.01 s^{-1}

[Valid data range] 1 to 9999

This parameter sets the position control loop gain of a spindle and tapping axis in rigid tapping.

The loop gain setting significantly influences the screw precision. Perform a cutting test to adjust the loop gain and its multiplier to the optimum values.

Note 1 To change the loop gain for every gear, set this parameter value to “0” and set the loop gain for every gear to parameter Nos. 5281 through 5284. If this parameter value is not “0”, the loop gain for every gear is invalidated. This parameter then becomes a loop gain that is used in common for all gears.

5281	Position control loop gain of spindle and tapping axis in rigid tapping (First gear)
5282	Position control loop gain of spindle and tapping axis in rigid tapping (Second gear)
5283	Position control loop gain of spindle and tapping axis in rigid tapping (Third gear)
5284	Position control loop gain of spindle an tapping axis in rigid tapping (Fourth gear)

[Data type] Word

[Unit of data] 0.01 s^{-1}

[Valid data range] 1 to 9999

These parameters set the position control loop gain of a spindle and tapping axis for every gear in rigid tapping.

Note 1 To set the loop gain for every gear, set parameter No. 5280 to “0”.

5291	Spindle loop gain multiplier in the rigid tapping mode (for gear 1)
5292	Spindle loop gain multiplier in the rigid tapping mode (for gear 2)
5293	Spindle loop gain multiplier in the rigid tapping mode (for gear 3)
5234	Spindle loop gain multioplier in the rigid tapping mode (for gear4)

[Data type] Word

[Unit of data]

[Valid data range] 0 to 32767

Set the spindle loop gain multipliers for gears 1 to 4 in the rigid tapping mode. The thread precision depends on the multipliers. Find the most appropriate multipliers and 100P gain by conducting the cutting test.

Note 1 These parameters are used for analog spindles.

$$\text{Loop gain multiplier} = 2048 \times \frac{E}{L} \times \alpha \times 1000$$

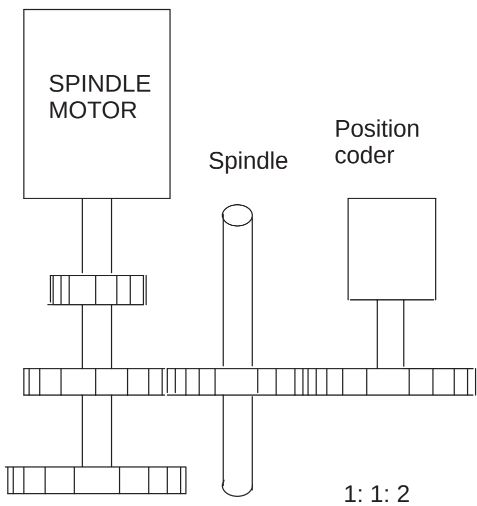
where;

E: Voltage in the velocity command at 1000 rpm

L: Rotation angle of the spindle per one rotation of the spindle motor

α: Unit used for the detection

Example)



When the spindle motor, spindle, and position coder are connected as shown left, let the variables be as follows:

E = 1.667 (V)

(A motor speed of 6000 rpm corresponds to 10 V.)

L = 360°

(One rotation of the spindle corresponds to one rotation of the spindle motor.)

α = La/4096
= 720°/4096
= 0.17578

La = 720°

(One rotation of the position coder corresponds to two rotations of the spindle = 360° × 2.)

4096 = The number of detected pulses per rotation of the position coder

} Gear ratio between the spindle and the position coder

1:1 . . . . 0.08789 degrees

1:2 . . . . 0.17578 degrees

1:4 . . . . 0.35156 degrees

1:8 . . . . 0.70313 degrees

Thus, Loop gain multiplier
= 2048 × 1.667/360 × 0.17578 × 1000 = 1667

Note 1 When the position coder which is built in a spindle motor sends 512 pulses per rotation, the unit used for the detection, α, is La/2048.

5300	In–position width of tapping axis in rigid tapping

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter sets the in–position width of a tapping axis in rigid tapping.

5301	In–position width of spindle in rigid tapping

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter sets the in–position width of a spindle in rigid tapping.

Note 1 The broad in–position width deteriorates the screw precision.

5310	Limit value of tapping axis positioning deviation during movement in rigid tapping

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter sets the limit value of a tapping axis positioning deviation during movement in rigid tapping.
To set a value larger than this value, set it to No. 5314.

Note 1 The setting value is represented in a 10–times unit when a high–resolution transducer is used.

5311	Limit value of spindle positioning deviation during movement in rigid tapping

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter sets the limit value of a spindle positioning deviation during movement in rigid tapping.

$$\text{Limit value} = \frac{S \times 360 \times 100 \times 1.5}{60 \times G \times \alpha}$$

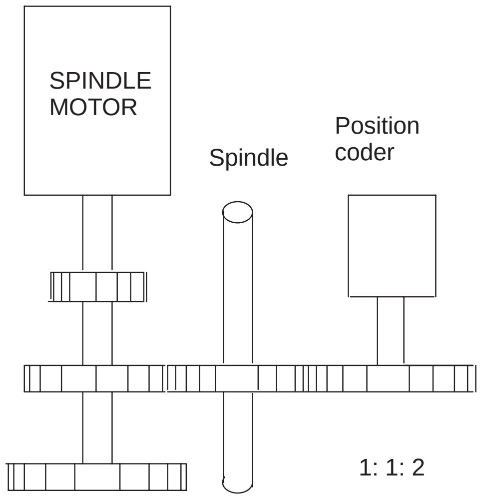
where

S: Maximum spindle speed in rigid tapping
(Setting value of parameter Nos. 5241 and greater)

G: Loop gain of rigid tapping axis
(Setting value of parameter Nos. 5280 and greater)

α: Detection unit

Example)



When the spindle motor, spindle, and position coder are connected as shown left, let the variables be as follows:

S = 3600
G = 3000
L = 360°
(One spindle rotation per spindle motor rotaion)
α = La / 4096
= 720°/4096
= 0.17578°
La= 720°
(One position coder rotation requires two spindle rotations =360° × 2)
4096=Detection pulse per positioncoder rotation

$$\text{Setting value} = \frac{3600 \times 360 \times 100 \times 1.5}{60 \times 3000 \times 0.17578}$$

$$= 6144$$

Note 1 The detection unit is α=La/2048 when the position coder built–in spindle motor uses a position coder of 512 pulses per revolution.

5312	Limit value of tapping axis positioning deviation during stop in rigid tapping

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter sets the limit value of a tapping axis positioning deviation during stop in rigid tapping.

5313	Limit value of spindle positioning deviation during stop in rigid tapping

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter sets the limit value of a spindle positioning deviation during stop in rigid tapping.

5314	Limit value of position deviation during movement along the tapping axis for rigid tapping

[Data type] Two–word

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Parameter 5310 usually sets the limit of positional deviation during movement along the tapping axis for rigid tapping. To specify a setting exceeding the valid range specified in parameter 5310 according to the resolution of the detector to be used, specify the limit value with parameter 5314.

Note 1 If the setting of this parameter is 0, the setting of parameter 5310 is enabled. Otherwise, the setting of parameter 5310 is disabled, and the setting of parameter 5314 is enabled.

5321	Spindle backlash in rigid tapping (First gear)
	Spindle backlash in rigid tapping
5322	Spindle backlash in rigid tapping (Second gear)
5323	Spindle backlash in rigid tapping (Third gear)
5324	Spindle backlash in rigid tapping (Fourth gear)

[Data type] Byte

[Unit of data] Detection unit

[Valid data range] 0 to 127

These parameters set the spindle backlash in rigid tapping.

9.11.9 Alarm and Message

Number	Message	Description
200	ILLEGAL S CODE COMMAND	In the rigid tapping, an S value is out of the range or is not specified. The maximum value for S which can be specified in rigid tapping is set in parameter (No.5241 to 5243). Change the setting in the parameter or modify the program.
201	FEEDRATE NOT FOUND IN RIGID TAP	In the rigid tapping, no F value is specified. Correct the program.
202	POSITION LSI OVERFLOW	In the rigid tapping, spindle distribution value is too large.
203	PROGRAM MISS AT RIGID TAPPING	In the rigid tapping, position for a rigid M code (M29) or an S command is incorrect. Modify the program.
204	ILLEGAL AXIS OPERATION	In the rigid tapping, an axis movement is specified between the rigid M code (M29) block and G84 or G74 block for M series (G84 or G88 block for T series). Modify the program.
205	RIGID MODE DI SIGNAL OFF	Rigid tapping signal (DGNG061#0) is not 1 when G84 or G74 block for M series (G84 or G88 block for T series) is executed though the rigid M code (M29) is specified. Consult the PMC ladder diagram to find the reason the signal is not turned on. Modify the program.
206	CAN NOT CHANGE PLANE (RIGID TAP)	Plane changeover was instructed in the rigid mode. Correct the program.
207	RIGID DATA MISMATCH	The specified distance was too short or too long in rigid tapping.
410	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis (axis 1–8 of rigid tapping axis) stops is larger than the set value. Note) Limit value must be set to parameter No.1829 for each axis.
411	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis (axis 1–8 of rigid tapping axis) moves is larger than the set value. Note) Limit value must be set to parameter No.1828 for each axis.
413	SERVO ALARM: n–th AXIS – LSI OVERFLOW	The contents of the error register for the n–th axis (axis 1–8 of rigid tapping axis) are beyond the range of -2^{31} to 2^{31}. This error usually occurs as the result of an improperly set parameters.

Number	Message	Description
740	RIGID TAP ALARM; EXCESS ERROR	Position deviation value of spindle at move exceeded a set value during rigid tapping.
741	RIGID TAP ALARM; EXCESS ERROR	Position deviation value of spindle at move exceeded a set value during rigid tapping or synchronous error exceeded a set value (parameter No. 5214) during rigid tapping.
742	RIGID TAP ALARM; LSI OVER FLOW	LSI overflow has occurred on the spindle side during rigid tapping.

9.11.10 Notes

NOTES ON SPINDLES

Note 1 A spindle pitch error is not compensated for in rigid tapping mode. Drift compensation is not made with an analog spindle.

Note 2 When using an analog spindle, set the spindle speed offset value parameter (No. 3731) accurately. For the standard system, a value within –8191 to 8191 must be specified in this parameter. To perform rigid tapping, a value within –1023 to 1023 must be specified.

If the spindle speed offset is set inaccurately, the spindle is stopped and placed in in–position wait state when tapping is started.

In rigid tapping with a serial spindle, no setting is required for parameter No. 3731. Be sure to set 0.

Note 3 When the threading and synchronous feed functions are enabled, the actual spindle speed during rigid tapping is indicated correctly. When an arbitrary gear ratio is used (by setting bit 1 (VGR) of parameter No. 5200 to 1), however, the actual spindle speed will not be indicated correctly in normal spindle mode.

When the T series is used, for example, information about the actual spindle speed is important for lathe machining. So, be particularly careful when using an arbitrary gear between the spindle and position coder.

Note 4 The maximum number of pulses that can be distributed to the spindle is:

· 32,767 pulses per 8 msec for a serial spindle

· 4,096 pulses per 8 msec for an analog spindle
(This information is displayed by selecting No. 451 on the diagnosis screen.)

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, P/S alarm No. 202 is issued.

Notes on using functions such as the spindle positioning function at the same time

Note 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.

Note 2 When the spindle positioning function is to be used at the same time

When the spindle positioning function 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 restriction does not apply, however, when multi–spindle control is applied; rigid tapping can be performed using the second spindle.

The spindle positioning function is effective for the first spindle only. This means that when spindle indexing is performed with the first spindle, rigid tapping can be specified with the second spindle.

Note 3 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 contorl 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, nullify jogging and manual handling feed of the Cs contouring axis in the PMC during rigid tapping.

(5) When the multi–spindle control is also available and the rigid tapping is performed on the 2nd spindle, the rigid tapping can be specified to the 2nd spindle during the Cs contouring control of the 1st spindle.

Position control loop gain switching and serial spindle parameters

In rigid tapping, the loop gain of the tapping axis is switched so that the loop gains for position control of the tapping axis and spindle match each other.

This switching processing is specified by parameter Nos. 5280, and 5281 to 5284. The contents of the processing vary with whether the spindle is an analog or serial spindle, as described below.

· When the spindle is an analog spindle, the loop gains of the spindle and tapping axis are switched according to the values set in these parameters.

· When the spindle is a serial spindle, the loop gain of the tapping axis is switched according to the values set in these parameters. The loop gain of the spindle depends on the values set in the serial spindle parameters and applied gear signals (CTH2, CTH1).

Accordingly, to perform rigid tapping with a serial spindle, the loop gain for position control of the spindle must be set in the serial spindle parameters used for rigid tapping.

When multi–spindle control is being used, rigid tapping can also be performed for the second spindle. For the serial spindle used for rigid tapping, set the parameters indicated below.

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 series (Serial Interface) DESCRIPTIONS (B–65042E) or FANUC AC SPINDLE MOTOR α series PARAMETER MANUAL (B–65160E).

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) (CFPGH)
4066	Position gain in the servo mode (MEDIUM HIGH) (CFPGMH)
4067	Position gain in the servo mode (MEDIUM LOW) (CFPGML)
4068	Position gain in the servo mode (LOW) (CFPGL)

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 65535

Set a servo loop gain in a servo mode (such as rigid tapping mode).

Note 1 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. 5280 and 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 (G070#3, #2 for the first spindle, and G074#3, #2 for the second spindle). 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	MEDIUM HIGH	4066		
1	0	MEDIUM LOW	4067	4045	4053
1	1	LOW	4068		

9.11.11 Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.14.2	RIGID TAPPING
CONNECTION MANUAL (This manual)	9.3	SPINDLE SPEED CONTROL
CONNECTION MANUAL (This manual)	9.10	MULTI–SPINDLE
FANUC AC SPINDLE MOTOR series (Serial Interface) DESCRIPTIONS (B–65042E)	Appen- dix 3.5	SERIAL SPINDLE RIGID TAPPING TECHNICAL MANUAL

9.12 SPINDLE SYNCHRONOUS CONTROL

General

This function enables the synchronous control of two spindles. 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.

Synchronous–spindle configuration

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.

The table below shows the synchronous spindle configuration.

	Master spindle	**Slave spindle**
T series/M series	First serial spindle	Second serial spindle
T series (two–path control)	First serial spindle at tool post 1	First serial spindle at tool post 2
M series (two–path control)	First serial spindle at each path	Second serial spindle at each axis

Supplementary description

For details of synchronous–spindle connection, see the description of serial spindles.

The following description relates to this CNC.

- Synchronous control of spindle phase is executed when the signal for controlling the spindle phases in synchronization is entered in spindle synchronization control mode (after output of the signal indicating that the synchronous control of spindle speed has been completed). The signal indicating that the synchronous control of spindle phase is completed is output when the difference between the error pulses of the two spindles does not exceed the number of pulses specified in parameter 4810 of the NC function.
  The positions of spindle phase synchronization can be specified in spindle parameter 4034 on each of tool post 1 and tool post2.

  When the two spindles are subject to spindle–phase synchronous–control (until the spindle–phase synchronous–control completion signal, FSPPH <F044, #3>, turns to “1”), they are not synchronized with each other.

  Do not specify spindle–phase synchronous control while the two spindles are holding a workpiece. Specifying this item causes phase synchronous control to start automatically.
- PMC signal, SYCAL <F044#4> is provided to monitor a synchronization errors between spindles for which spindle synchronization control or synchronous control of spindle phase is in effect. The synchronization error between the two spindles is always monitored. The SYCAL signal is set to 1 when the error (the absolute value of the error pulse) specified in parameter 4811 of tool post 1 is exceeded, and set to 0 when not exceeded.

- Constant surface speed control can be executed in synchronization control even while a workpiece is being held with the two spindles.
  However, if the speed is to change in excess of the specified time constant, the speed changes within the extent specified by time constant.
- The maximum speed in synchronization control is determined by the maximum speed of the spindle motor of master spindle (parameter 4020).

(Example) Maximum speed of the spindle motor of tool post 1: 6000 rpm
Maximum speed of the spindle motor of tool post 2: 4500 rpm

In the example above, a maximum spindle speed of 6,000 rpm is specified for a spindle of tool post 1, although a spindle–speed command can specify up to 12 bits, 4096. If 6,000 rpm is specified while synchronous control is specified, an overspeed alarm is issued for a spindle of tool post 2. Therefore, do not specify a value of more than 4,500 rpm in this case.

- Like the conventional spindle speed (S) command for which 4 or 5 digits are issued for the first spindle, the signal for specifying spindle speed can be generated when spindle synchronization control or synchronous control of spindle phase are in the process of being put into effect. The SIND, SSIN SSGN, R01l to R12l, *SSTP, and SOR signals are effective as usual.

  However, in the usual mode of spindle rotation control, spindle speed can be controlled by the PMC function when the following conditions are satisfied: The SIND signal is set to 1 and the SSIN, SSGN, and R01l to R12l signals are provided. When spindle synchronization control is in the process of being put into effect, something other than the R01l to R12l signals is required to control the spindle speed in synchronization. The maximum spindle gear speed must be properly set in parameters 3741, 3742, 3743 and 3744. When the value set in the parameter corresponding to the selected gear is 0, the rotations of the spindles are not synchronized even if a command is entered in the R01I to R12I signals.
- The S command for the master spindle and the PMC control signal for spindle control become effective when issued before spindle synchronization control or synchronous control of spindle phase are put into effect. The S command issued in synchronization control becomes effective for the first spindle immediately after synchronization control is canceled.
- The load may change due to cutting (or threading). When the load changes in spindle synchronization control, the spindle speed may change and the signal indicating that the synchronous control of spindle speed is completed may go off temporarily.
- Parameters No. 4800 #0 (for the master spindle) and #1 (for the slave spindle) are used to set the direction of rotation of the first spindle and second spindle, respectively.
- The gear ratio of the spindle to the position coder must be set to one–to–one.
- In spindle synchronization control , the compensation value for spindle speed offset (parameter 3731) is disabled.

- A spindle–phase synchronous control command is effective only in synchronous spindle control mode. The specified phase can be repeatedly changed under synchronous control.

Signal

See the manual of serial spindles.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
4800							ND2	ND1

[Data type] Bit type

ND1 In controlling the spindle synchronization, the direction of the first spindle (master spindle) motor rotation is:
0 : The direction indicated by the command sign
1 : The opposite direction to that indicated by the command sign

ND2 In controlling the spindle synchronization, the direction of the 2nd spindle (slave spindle) motor rotation is:
0 : The direction indicated by the command sign
1 : The opposite direction to that indicated by the command sign

4810	Error pulse between two spindles when synchronizing phases in the serial spindle synchronization control mode

[Data type] Byte type

[Unit of data] Pulse

[Valid data range] 0 to 255

Set the difference in error pulses between two spindles when synchronizing phases in the serial spindle synchronization control mode.

When the difference in error pulse between two spindles is within the value set in this parameter, the spindle phase synchronization completion signal FSPPH <F044#3> becomes “1”.

This parameter is used to check the difference in phase in synchronization control and to confirm the completion of synchronization in the serial spindle synchronization control mode.

4811	Allowable error count for the error pulses between two spindles in the serial spindle synchronization control mode

[Data type] Word type

[Unit of data] Pulse

[Valid data range] 0 to 32767

Set the allowable error count for the error pulses between two spindles in the serial spindle synchronization control mode.

This parameter is used to output the inter–spindle phase error detection signal SYCAL in the serial spindle synchronization control mode. The SYCAL <F044#4> signal becomes “1” when a phase error exceeding the value set in this parameter is found.

Alarm and message

Number	Message	Description
194	SPINDLE COMMAND IN SYNCHRO–MODE	A contour control mode, spindle positioning (Cs–axis control) mode, or rigid tapping mode was specified during the serial spindle synchronous control mode. Correct the program so that the serial spindle synchronous control mode is released in advance.

Note

Note 1 Signal SYCAL <F044#4> 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.

Reference item

FANUC AC SPINDLE MOTOR series (Serial interface) DESCRIPTIONS (B–65042/03)	Appendix 3.3	Start–up procedure for spindle synchronization control
FANUC CONTROL MOTOR AMPLIFIER α series DESCRIPTIONS (B–65162E/01)	11.7	Spindle synchronization control

9.13 SPINDLE ORIENTATION

General

This function stops the spindle at a specified position. The spindle can be stopped in either of the following two ways.

· The spindle is mechanically stopped by using stoppers.

· The spindle is stopped by applying a function of the spindle control unit.

Mechanical stop

To mechanically stop the spindle by using, for example, a stopper, rotate the spindle at a constant low speed and drive a stopper or pin into the spindle. The spindle can be rotated at a constant speed by applying either of the following methods.

· Spindle orientation signal (See 9.3, "Spindle Control.")

· Spindle output control by the PMC (See 15.4.)

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.

Serial spindle orientation by a position coder

In serial spindle orientation by a position coder, the stop position is specified either by a parameter or by the PMC (spindle orientation function with the stop position externally set).

Signal

Spindle orientation signals with the stop position externally set SHA00 to SHA11 for the first spindle <G078, G079> SHB00 to SHB11 for the second spindle <G080, G081>

[Classification] Input signal

[Function] This command is used for specifying a stop position with an absolute position within one rotation in the following equation:

$$= \frac{360}{4096} \times \sum_{i=0}^{\#n} (2_i \, Pi)$$

where
Pi = 0 when SHAi = 0
Pi = 1 when SHAi = 1

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G078	SHA07	SHA06	SHA05	SHA04	SHA03	SHA02	SHA01	SHA00
G079					SHA11	SHA10	SHA09	SHA08
G080	SHB07	SHB06	SHB05	SHB04	SHB03	SHB02	SHB01	SHB00
G081					SHB11	SHB10	SHB09	SHB08

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3702					OR2	OR1		

[Data type] Bit

OR1: Whether the stop–position external–setting type orientation function is used by the first spindle motor
0 : Not used
1 : Used

OR2 Whether the stop–position external–setting type orientation function is used by the second spindle motor
0 : Not used
1 : Used

Note

Note 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.

Note 2 When the spindle orientation function of stop position external setting type is used, the stop position parameters in spindle orientation with a position coder (No. 6531 and 6671) are invalid.

Note 3 Spindle orientation with the spindle positioning function differs from that described in this section. For details, see Section 9.8, “Spindle Positioning.”

Reference item

FANUC AC SPINDLE MOTOR series (Serial Interface) DESCRIPTIONS (B–65042E)	VIII.	Position coder method spindle orientation
FANUC CONTROL MOTOR AMPLIFIER α series DESCRIPTIONS (B–65162E)	11.1 12.1	Position coder method spindle orientation Magnetic sensor method spindle orientation

9.14 SPINDLE OUTPUT SWITCHING

General

Spindle output switching switches between the two 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, see also 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.

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.

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.

Note

Note 1) 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 shall not rotate with no drive 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.

Note 2) Output switching and gear switching

Spindle output switching ensures that the spindle motor demonstrates stable characteristics over a wide range, and eliminates the 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 (see 9.3) 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 <F034 #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.

- The parameters related to gear switching points, SGT, bit 3 of parameter No. 3705, and SGB, bit 2 of parameter No. 3761, 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 <G028 #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

CONNECTION MANUAL (This manual)	9.3 9.11	Spindle control Rigid tapping
FANUC AC SPINDLE MOTOR series (Serial Interface) DESCRIPTIONS (B–65042E)	XIII	Output switching control
FANUC CONTROL MOTOR AMPLIFIER α series DESCRIPTIONS (B–65162E/01)	11.9	Output switching control

10 TOOL FUNCTIONS

10.1 TOOL FUNCTION

General

(M series)

Selection of tools can be done by commanding tool numbers with up to an 8-digit numeral after address T.

(T series)

Selection of tools and offset amounts can be done by commanding tool numbers and offset numbers with up to an 8-digit numeral after address T. The offset number is specified with the last one or two digits of the T code. The tool number is specified with the remaining digits after excluding the one or two digits used to specify the offset number.

When the last one digit is used to specify the offset number:
(Parameter LD1 (No. 5002#0)=1)

T○○○○○○○ ○

Offset number

Tool number

When the last two digits are used to specify the offset number:
(Parameter LD1 (No. 5002#0)=0)

T○○○○○○ ○○

Offset number

Tool number

When a T code is specified, the code signal and strobe signal corresponding to the specified tool number are issued. The machine selects a tool according to the issued signals. The code signal is held until another T code is specified.

In a block, no more than one T code can be specified. The maximum number of digits that can follow T can be specified in parameter 3032. If this number is exceeded, an alarm occurs.

Signal

See Section 8.1.

Parameter

3032	Allowable number of digits for the T code

[Data type] Byte

[Valid data range] 1 to 8

Set the allowable numbers of digits for the T code.

	#7	#6	#5	#4	#3	#2	#1	#0
5002							LGN	LD1

[Data type] Bit

LD1 Offset number of tool offset (Wear offset number when option of tool geometry/wear compensation is selected)
0 : Specified using the lower two digits of a T code
1 : Specified using the lower one digit of a T code

LGN Geometry offset number of tool offset (When the option of tool geometry/wear compensation is selected, it is effective.)
0 : Is the same as wear offset number
1 : Specifies the geometry offset number by the tool selection number

	#7	#6	#5	#4	#3	#2	#1	#0
5006							TGC	

[Data type] Bit

TGC When a T code is specified in a block containing G50, G04, or G10:
0 : No alarm occurs.
1 : P/S alarm No. 254 occurs.

Alarm and message

Number	Message	Description
030	ILLEGAL OFFSET NUMBER (T series)	The offset number in T function specified for tool offset is too large. Modify the program.
043	ILLEGAL T–CODE COMMAND (M series)	In a system using the DRILL–MATE with an ATC, a T code was not specified together with the M06 code in a block. Alternatively, the Tcode was out of range.
245	T–CODE NOT ALOWED IN THIS BLOCK (T series)	One of the G codes, G50, G10, and G04, which cannot be specified in the same block as a T code, was specified with a T code.

Note

Note 1 When a move command and a tool function are specified in the same block, the commands are executed in one of the following two ways:

(i) Simultaneous execution of the move command and tool function commands.

(ii) Executing tool function commands upon completion of move command execution.
The selection of either (i) or (ii) depends on the sequence program of PMC.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.11.1	TOOL SELECTION FUNCTION
OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.11.1	TOOL SELECTION FUNCTION
CONNECTION MANUAL (This manual)	8	AUXILIARY FUNCTION

10.2 TOOL COMPENSATION VALUE/ TOOL COMPENSATION NUMBER/ TOOL COMPENSATION MEMORY

General

(M series)

Tool compensation values include tool geometry compensation values and tool wear compensation values (Fig. 10.2 (a)).
The geometry compensation and wear compensation can be unified to the tool compensation.

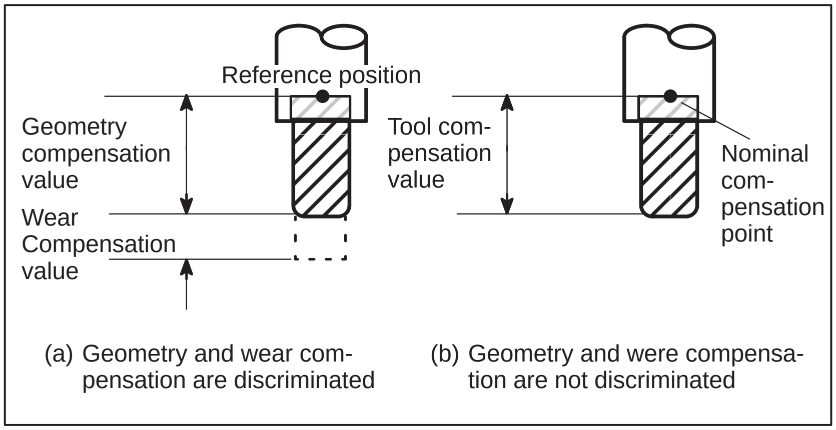


Fig. 10.2(a) Geometric compensation and wear compensation

Tool compensation values can be entered into CNC memory from the CRT/MDI panel or from a program.
A tool compensation value is selected from the CNC memory when the corresponding code is specified after address H or D in a program.
The value is used for tool length compensation, cutter compensation, or the tool offset.

- **Range of tool compensation value**

Tool offset amount range which can be set is as follows:

Increment system	Tool compensation (Geometry compensation)		Tool wear compensation	
	Metric input	Inch input	Metric input	Inch input
IS-B	±999.999 mm	±99.9999 inch	±99.999 mm	±9.9999 inch
IS-C	±999.9999 mm	±99.99999 inch	±99.9999 mm	±9.99999 inch

- **Tool compensation number**

The memory can hold 32, 64, 99, 200, 400, 499 or 999 sets of tool compensation values.

- **Tool compensation memory**

One of the tool compensation memory A/B/C can be selected according to offset amount.

- Tool compensation memory A
  There is no difference between geometry compensation memory and wear compensation memory in this tool compensation memory A. Therefore, amount of geometry offset and wear offset together is set as the offset memory. There is also no differences between cutter compensation (D code) and tool length compensation (H code).

- Tool compensation memory B

  Memory for geometry compensation and wear compensation is prepared separately in tool compensation memory B. Geometry compensation and wear compensation can thus be set separately. There is no difference between cutter compensation (D code) and tool length compensation (H code).

- Tool compensation memory C

  Memory for geometry compensation as well as wear compensation is prepared separately in tool compensation memory C. Geometry compensation and wear compensation can thus be set separately. Separate memories are prepared for cutter compensation (for D code) and for tool length compensation (for H code).

The above description is summarized as follows:

Tool compensation memory	Compensation amount
A	Tool compensation amount (Geometry compensation value + Wear compensation value)
B	Geometry compensation value
	Wear compensation value
C	Geometry compensation value for H code
	Geometry compensation value for D code
	Wear compensation value for H code
	Wear compensation value for D code

(T series)

Tool compensation values include tool geometry compensation values and tool wear compensation values (Fig. 10.2 (b)).
Tool compensation can be specified without differentiating compensation for tool geometry from that for tool wear (Fig. 10.2 (c)).

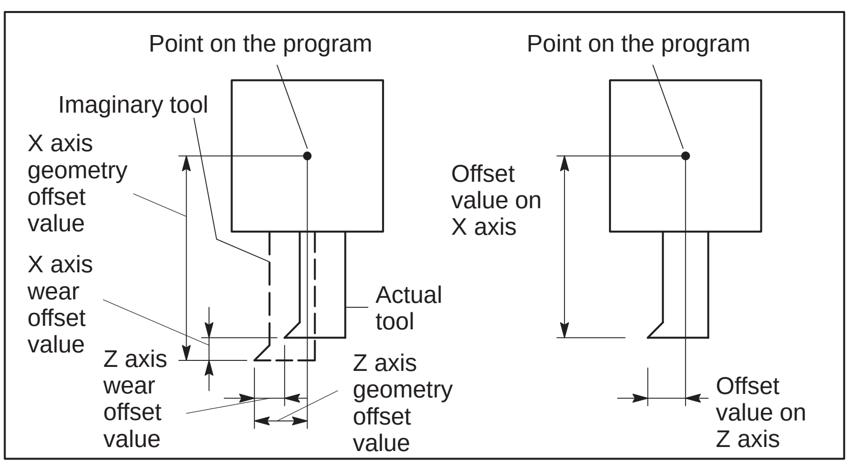


Fig. 10.2 (b) Difference the tool geometry offset from tool wear offset

Fig.10.2 (c) Not difference the tool geometry offset from tool wear offset

Tool compensation values can be entered into CNC memory from the CRT/MDI panel or from a program.
A tool compensation value is selected from the CNC memory when the corresponding code is specified after address T in a program.
The value is used for tool offset or tool nose radius compensation.

- **Range of tool compensation value**

Tool offset amount range which can be set is as follows:

Increment system	Tool compensation (geometry compensation, wear compensation)	
	Metric input	Inch input
IS-B	± 999.999 mm	± 99.9999 inch
IS-C	± 999.9999 mm	± 99.99999 inch

Range of tool compensation value can be expanded by option.

Increment system	Tool compensation value (geometry compensation, wear compensation)	
	Metric input	Inch input
IS–B	±9999.999 mm	±999.9999 inch
IS–C	±9999.9999 mm (±4000.0000 mm)	±999.99999 inch (±160.00000 inch)

Note 1 When parameter OIM (No. 5006#0)=1, the range in parenthesis is available.

Note 2 The above table does not apply to the B–axis offset in B–axis control function.

- **Tool compensation number**

The memory can hold 16, 32, 64 or 99 sets of tool compensation values.

- **Tool compensation memory**

There are two types of tool offset amount memory, which can be selected according to offset amount.

- Tool geometry/wear compensation option not specified

  There is no difference between geometry offset memory and wear offset memory. Therefore, amount of geometry offset and wear offset together is set as the offset memory.

- Tool geometry/wear compensation option specified

  Memory for geometry compensation and wear compensation is prepared separately. Geometry compensation and wear compensation can thus be set separately.

The above description is summarized as follws:

Tool compensation memory	Compensation amount
Without geometry/wear compensation	Tool compensation amount (Geometry compensation value + Wear compensation value)
With geometry/wear compensation	Geometry compensation
	Wear compensation

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3109							DWT	

[Data type] Bit

DWT Characters G and W in the display of tool wear/geometry compensation amount

0 : The characters are displayed at the left of each number.
1 : The characters are not displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3290							GOF	WOF

[Data type] Bit

WOF Setting the tool wear compensation value by MDI key input is:

0 : Not disabled
1 : Disabled (With parameter No. 3294 and No. 3295, set the offset number range in which updating the setting is to be disabled.)

GOF Setting the tool geometry compensation value by MDI key input is:

0 : Not disabled
1 : Disabled (With parameter No. 3294 and No. 3295, set the offset number range in which updating the setting is to be disabled.)

3294	Start number of tool offset values whose input by MDI is disabled
3295	Number of tool offset values (from the start number) whose input by MDI is disabled

[Data type] Word

When the modification of tool offset values by MDI key input is to be disabled using bit 0 (WOF) of parameter No. 3290 and bit 1 (GOF) of parameter No. 3290, parameter Nos. 3294 and 3295 are used to set the range where such modification is disabled. In parameter No. 3294, set the offset number of the start of tool offset values whose modification is disabled. In parameter No. 3295, set the number of such values.

Example:

The following setting disables the modification of both the tool geometry compensation values and tool wear compensation values corresponding to offset numbers 100 to 110:

Bit 1 (GOF) of parameter No. 3290 = 1 (Disables tool geometry compensation value modification.)

Bit 0 (WOF) of parameter No. 3290 = 1 (Disables tool wear compensation value modification.)

Parameter No. 3294 = 100

Parameter No. 3295 = 11

If bit 0 (WOF) of parameter No. 3290 is set to 0, the modification of the tool geometry compensation values alone is disabled. The tool wear compensation values may be modified.

	#7	#6	#5	#4	#3	#2	#1	#0
5002	WNP						LGN	

[Data type] Bit

LGN Geometry offset number of tool offset (When the option of tool geometry/wear compensation is selected, it is effective.)
0 : Is the same as wear offset number
1 : Specifies the geometry offset number by the tool selection number

WNP Imaginary tool tip direction used for tool nose radius compensation, when the geometry/wear compensation option is equipped, is the direction specified by:
0 : Geometry offset number
1 : Wear offset number

	#7	#6	#5	#4	#3	#2	#1	#0
5004							ORC	

[Data type] Bit

ORC Tool offset value
0 : Set by the diameter specification (Can be set in only the axis under diameter programming)
1 : Set by the radius specification

	#7	#6	#5	#4	#3	#2	#1	#0
5006								OIM

[Data type] Bit

OIM When the unit is switched between the inch and metric systems, automatic tool compensation value conversion is:
0 : Not performed
1 : Performed

5013	Maximum value of tool wear compensation

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range]

Increment system	IS–A	IS–B	IS–C
Metric input	0 to 99999	0 to 999999	0 to 9999999
Inch input	0 to 99999	0 to 999999	0 to 9999999

This parameter sets the maximum value of tool wear compensation. The following alarm or warning will be informed when the tool wear compensation (absolute value) exceeding this setting value is set.

Input from MDI	Too many digits
Input by G10	P/S 32 offset value is out of range by G10

5014	Maximum value of incremental input for tool wear compensation

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range]

Increment system	IS–A	IS–B	IS–C
Metric input	0 to 99999	0 to 999999	0 to 9999999
Inch input	0 to 99999	0 to 999999	0 to 9999999

This parameter sets the maximum value of tool wear compensation at an incremental input. If the incremental value exceeds the set value, the following alarm or warning message is indicated:

Input from MDI	Data is out of range
Input by G10	P/S 32 offset value is out of range by G10

Alarm and message

Number	Message	Description
032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.

Warning message	Content
DATA IS OUT OF RANGE	The value searched exceeds the permitted range.
TOO MANY DIGITS	The input value exceeds the permitted number of digits.

Note

Note 1 In the tow–path control, the number of specified tool compensation values equals the number of tool compensations for each tool post.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II 15.8	TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II 15.5	TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)

10.3 TOOL LIFE MANAGEMENT

General

When tools are classified into several groups, average tool life (No. of uses or time) is designated for each group. Whenever a tool is used, the usage time is subtracted from the tool life; when the tool life expires, the next tool in the group is selected. The tool sequence within a group is arranged in advance.

Signal

The end of a tool's life is reported by tool change signal TLCH or individual tool change signal TLCHI. Tool change signal TLCH is set to 1 at the end of the life of the last tool of a group. Individual tool change signal TLCHI is set to 1 at the end of the life of the current tool.

Tool change signal TLCH <F064#0>

[Classification] Output signal

[Function] Reports the end of the life of the last tool of a group.

[Output condition] The signal is set to 1 when:

· The life of the last tool of a group ends, after tool change has been performed each time the end of the life of each tool in a group is detected.

The signal is set to 0 when:

· Tool–change reset is completed for all groups in which no available tools remain.

Note 1 The TLCH signal turns to "1" when the CNC is reset by M02 or M30, for instance after the tool life, based on the frequency of times used, is reached. When tool life is specified by usage time, TLCH turns to "1" when the tool life limit is reached. The signal will change during machine operation, but machining will continue until the end of the program.

Tool change reset signal TLRST <G048#7>

[Classification] Input signal

[Function] Clears all executable data, including the life count of the group, *, and @.

To clear the data, specify a group number after replacing the worn–out tools that are displayed on the CRT. The data can also be cleared from the MDI.

[Output condition] When the signal is set to 1, the control unit operates as follows:

· Clears all executable data, including the life count of the group.

If the same group is specified after machining is resumed, the first tool in the group is selected.

Note 1 Tool change reset signal TLRST is valid only when the automatic operating signal OP is “0”.

Individual tool change signal TLCHI <F064#2>

[Classification] Output signal

[Function] Reports the end of the life of the current tool. The following processing can be programmed: A running program is interrupted by a tool-change program when the signal turns to “1”. Execution of the interrupted program is resumed when the tool is changed.

[Output condition] The signal is set to “1 ”when:

· The end of the life of the current tool is detected.

The signal is set to “0” when:

· Individual tool-change reset is executed.

Individual tool change reset signal TLRSTI <G048#6> (M series)

[Classification] Input signal

[Function] Sets the individual tool change signal TLCHI to “0”.

[Operation] When the signal is set to “1”, the control unit operates as follows:

· Sets the individual tool change signal to “0”.

Note 1 These signals are valid only when tool life management is performed on the basis of the tool life calculated in terms of time or cutting length.

Note 2 Individual tool change signal TLCHI is not cleared by reset.

Tool skip signal TLSKP <G048#5>

[Classification] Input signal

[Function] A tool which has not reached its lifespan may be changed by one of two methods:

(i) Designate the group number for the tool to be changed through the PMC, then turn the tool skip signal TLSKP to “1”. The next T-code command will pass over the first tool in the group for which the skip was designated, and select the second tool.

(ii) Turn the TLSKP signal to “1” without designating a group number, and the machine will skip to the next tool in the group currently in use.

Either of these methods is set using parameter SIG no. 6800#3. Tool life is counted from zero. When the TLSKP signal is “1” and the last tool in the group is being used, the TLCH signal turns to “1”.

[Operation] When the signal is set to “1”, the control unit operates as follows:

· Selects the next tool in the group for which a skip is specified with the next T code.

· Assumes the number of the group to which the current tool belongs.

Note 1 The cycle start lamp signal (STL) and feed hold lamp signal (SPL) must both be “0” before inputting the TLSKP signal.

New tool select signal TLNW <F064#1>

[Classification] Output signal

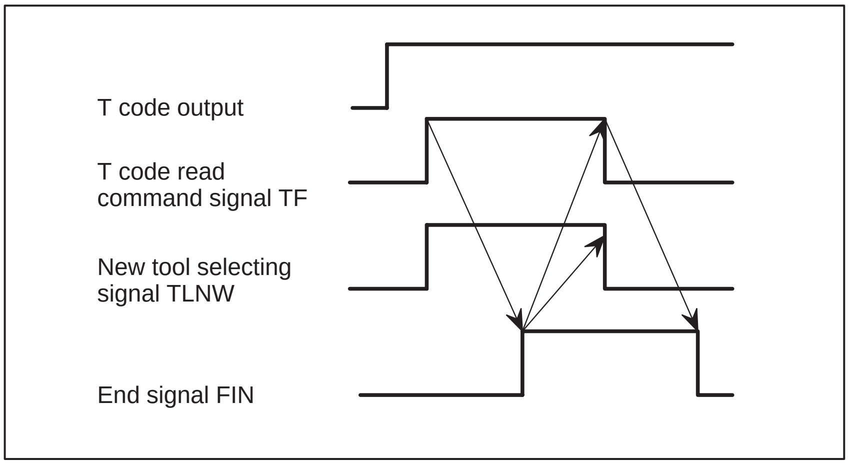
[Function] Reports that a new tool of a certain group is selected.

[Output condition] The signal is set to “1” when:

· A new tool of a certain group is selected.

The signal is set to “0” when:

· The completion signal is set to “1”.



Tool group number select signal TL01 to TL256 (M series) <G047#0 to G048#0> TL01 to TL 64 <G47#0 to #6> (T series)

[Classification] Input signal

[Function] When the TLRST and TLSKP signals are both input, the tool group number must be given in advance, using the tool group number signals TL01 to TL64 (T series) or TL01 to TL128 (M series).
Command the following value in binary form:
Tool group number to be specified –1

[Operation] When the signal is set to 1, the control unit operates as follows:

· Specifies a tool group number.

Tool life count override signal *TLV0 to *TLV9 <G049#0 to G050#1> (M series)

[Classification] Input signal

[Function] Overrides the life count (time) if bit 2 of parameter No. 6801 (LFV) is specified.

Each of the ten binary code signals has a unique override value that becomes valid when the signal is set to “0”. The life count is overridden by the sum of the valid override values. The override value can be specified in steps of 0.1, within the range of 0 to 99.9.

$$\text{Override value} = \sum_{i=0}^{9} \{2^i \times Vi\}$$

*TLV0	× 0.1
*TLV1	× 0.2
*TLV2	× 0.4
*TLV3	× 0.8
*TLV4	× 1.6
*TLV5	× 3.2
*TLV6	× 6.4
*TLV7	×12.8
*TLV8	×25.6
*TLV9	×51.2

(Example) When *TLV7, *TLV6, and *TLV3 are set to “0”, the override value is calculated as follows:

12.8 + 6.4 + 0.8 = 20.0

The life count is multiplied by 20.0.

[Operation] The actual cutting time is counted and multiplied by the override value obtained by the signals. The calculated time is used as the basis for tool-life management.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G047	TL128	TL64	TL32	TL16	TL08	TL04	TL02	TL01
G048	TLRST	TLRSTI	TLSKP					TL256
G049	*TLV7	*TLV6	*TLV5	*TLV4	*TLV3	*TLV2	*TLV1	*TLV0
G050							*TLV9	*TLV8
F064						TLCHI	TLNW	TLCH

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6800			SNG	GRS	SIG	LTM	GS2	GS1
	M6T	IGI	SNG	GRS	SIG	LTM	GS2	GS1

[Data type] Bit

GS1, GS2 This parameter sets the combination of the number of tool life groups which can be entered, and the number of tools which can be entered per group as shown in the table below.

Lower side ranges in M series column are for the tool life management of 512 pairs.

GS2	GS1	M series		T series	
		Group count	Tool count	Group count	Tool count
0	0	1– 16 1– 64	1– 16 1– 32	1– 16 1– 16	1– 16 1– 32
0	1	1– 32 1–128	1– 8 1– 16	1– 32 1– 32	1– 8 1– 16
1	0	1– 64 1–256	1– 4 1– 8	1– 64 1– 64	1– 4 1– 8
1	1	1–128 1–512	1– 2 1– 4	1– 16 1–128	1– 16 1– 4

LTM Tool life
0 : Specified by the number of times
1 : Specified by time

SIG Group number is
0 : Not input using the tool group signal during tool skip (The current group is specified.)
1 : Input using the tool group signal during tool skip

GRS Tool exchange reset signal
0 : Clears only the execution data of a specified group
1 : Clears the execution data of all entered groups

SNG Input of the tool skip signal when a tool that is not considered tool life management is selected.
0 : Skips the tool of the group used last or of the specified group (using SIG, #3 of parameter No. 6800).
1 : Ignores a tool skip signal

IGI Tool back number
0 : Not ignored
1 : Ignored

M6T T code in the same block as M06
0 : Judged as a back number
1 : Judged as a next tool group command

	#7	#6	#5	#4	#3	#2	#1	#0
6801		TLE					TSM	
	M6E	EXT			EMD	LFV		CUT

[Data type] Bit

CUT The tool life management using cutting distance is
0 : Not performed (Usually set this parameter to 0).
1 : Performed

TSM When a tool takes several tool numbers, life is counted in tool life management:
0 : For each of the same tool numbers.
1 : For each tool.

LFV Specifies whether life count override is enabled or disabled when the extended tool life management function is used.
0 : Disabled
1 : Enabled

EMD An asterisk (*) indicating that a tool has been expired is displayed,
0 : When the next tool is selected
1 : When the tool life is expired

TLE Tool life management data registration by G10 (T series) is:
0 : Performed after the data for all tool groups has been cleared.
1 : Performed on a specified group.

EXT Specifies whether the extended tool life management function is used.
0 : Not used
1 : Used

M6E When a T code is specified in the same block as M06
0 : The T code is processed as a return number or as a group number selected next. Either is set by parameter M6T No. 6800#7.
1 : The tool group life is counted immediately.

6810	
	Tool life management ignored number

[Data type] Word

[Valid data range] 0 to 9999

This parameter sets the tool life management ignored number.

When the set value is subtracted from a T code, a remainder is used as the tool group number of tool life management when a value exceeding the set value is specified in the T code.

6811	Tool life count restart M code

[Data type] Byte

[Valid data range] 0 to 255 (not including 01, 02, 30, 98, and 99)
When zero is specified, it is ignored.

When the life is specified by the number of times, the tool exchange signal is output when a tool life count restart M code is specified if tool life of at least one tool group is expired. A tool in life is selected in the specified group when a T code command (tool group command) is specified after the tool life count restart M code is specified. A tool life counter is then incremented by one.

When the life is specified by time, a tool in life is selected in the specified group when a T code command (tool group command) is specified after the tool life count restart M code is specified.

Alarm and message

Number	Message	Description
149	FORMAT ERROR IN G10L3	A code other than Q1,Q2,P1 or P2 was specified as the life count type in the extended tool life management.
150	ILLEGAL TOOL GROUP NUMBER	Tool Group No. exceeds the maximum allowable value. Modify the program.
151	TOOL GROUP NUMBER NOT FOUND	The tool group commanded in the machining program is not set. Modify the value of program or parameter.
152	NO SPACE FOR TOOL ENTRY	The number of tools within one group exceeds the maximum value registerable. Modify the number of tools.
153	T–CODE NOT FOUND	In tool life data registration, a T code was not specified where one should be. Correct the program.
154	NOT USING TOOL IN LIFE GROUP (M series)	When the group is not commanded, H99 or D99 was commanded. Correct the program.
155	ILLEGAL T–CODE IN M06 (M series)	In the machining program, M06 and T code in the same block do not correspond to the group in use. Correct the program.
	ILLEGAL T–CODE IN M06 (T series)	In the machining program, M06 and T code in the same block do not correspond to the group in use. Correct the program.
156	P/L COMMAND NOT FOUND	P and L commands are missing at the head of program in which the tool group is set. Correct the program.
157	TOO MANY TOOL GROUPS	The number of tool groups to be set exceeds the maximum allowable value. (See parameter No. 6800 bit 0 and 1) Modify the program.
158	ILLEGAL TOOL LIFE DATA	The tool life to be set is too excessive. Modify the setting value.
159	TOOL DATA SETTING INCOMPLETE	During executing a life data setting program, power was turned off. Set again.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.11.2	Tool Life Management Function
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.11.2	Tool Life Management Function

10.4 CUTTER COMPENSATION

10.4.1 CUTTER COMPENSATION B, C (M series)

General

When the tool is moved, the tool path can be shifted by the radius of the tool.
To make an offset as large as the radius of the tool, first create an offset vector with a length equal to the radius of the tool (start–up). The offset vector is perpendicular to the tool path. The tail of the vector is on the workpiece side and the head points to the center of the tool.
If a linear interpolation, corner offset (cutter compensation B only), or circular interpolation command is specified after start–up, the tool path can be shifted by the length of the offset vector during machining.
To return the tool to the start point at the end of machining, cancel the cutter compensation mode.

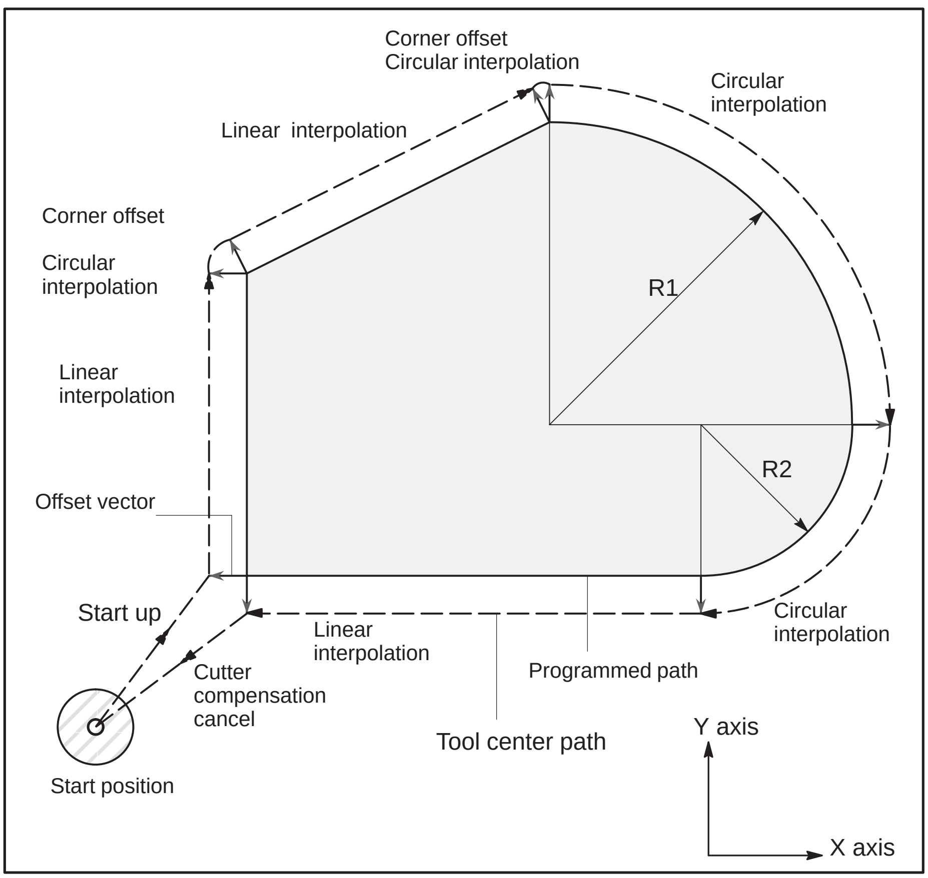


Fig. NO TAG (a) Outline of Cutter Compensation B

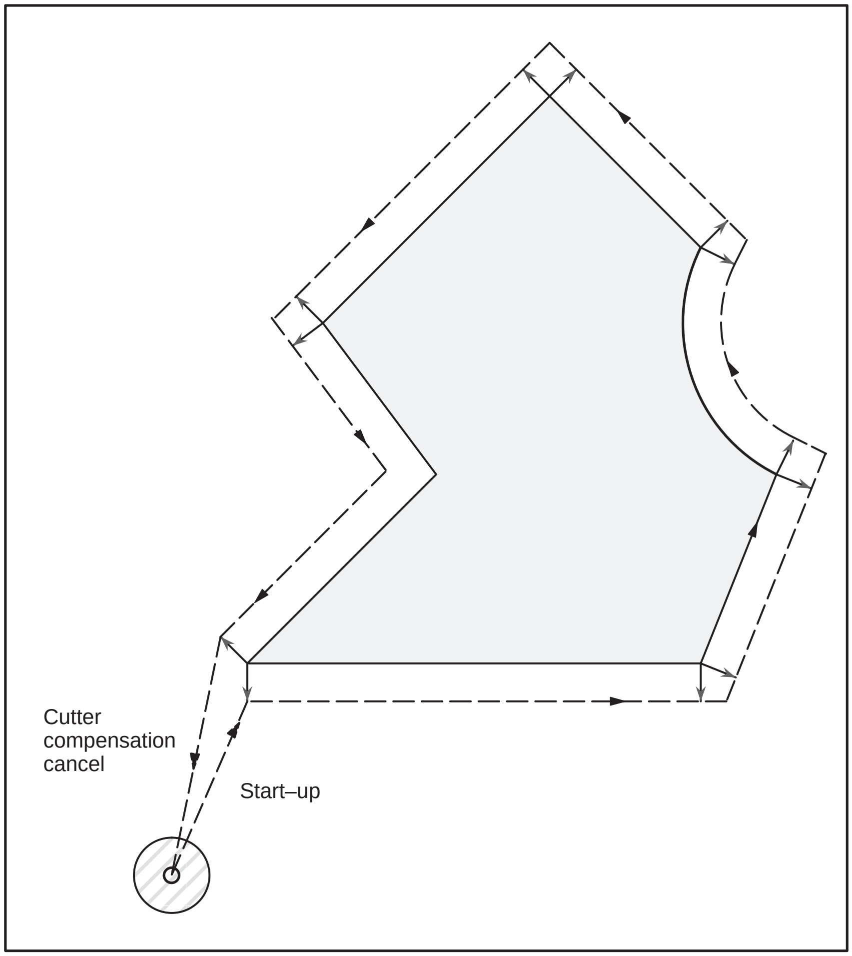


Fig. NO TAG (b) Outline of Cutter Compensation C

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5001						OFH		

[Data type] Bit

OFH Offset number of tool length compensation, cutter compensation and tool offset

0 : Specifies the tool length compensation using an H code, and cutter compensation C using a D code
Tool offset conforms to TPH in parameter No. 5001#5.

1 : Specifies the tool length compensation, cutter compensation and tool offset using H codes

	#7	#6	#5	#4	#3	#2	#1	#0
5003						CCN	SUV	SUP

[Data type] Bit

SUP Start up or cancel in cutter compensation C

0 : Type A
1 : Type B

SUV When G40, G41, and G42 are specified independently,

0 : The start up and cancel operation conforms to the standard specification.

1 : Moves by a distance corresponding to the offset vector which is vertical to the next block movement.

CCN When automatic reference position return (G28) is specified in the cutter compensation C mode (M series) or in tool nose radius compensation (T series):

0 : The cutter compensation vector is cancelled in movement to an intermediate position.

1 : The cutter compensation vector is not cancelled in movement to an intermediate position, but is cancelled in movement to the reference position.

5010	Limit value that ignores the vector when a tool moves on the outside of a corner during cutter compensation C

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 16383

This parameter sets the limit value that ignores a slight movement occurring when a tool moves on the outside of the corner during cutter compensation C.

Alarm and message

Number	Message	Description
033	NO SOLUTION AT CRC	A point of intersection cannot be determined for cutter compensation C. Modify the program.
034	NO CIRC ALLOWED IN ST–UP /EXT BLK	The start up or cancel was going to be performed in the G02 or G03 mode in cutter compensation C. Modify the program.
035	CAN NOT COMMANDED G39	G39 is commanded in cutter compensation B cancel mode or on the plane other than offset plane. Modify the program.
036	CAN NOT COMMANDED G31	Skip cutting (G31) was specified in cutter compensation mode. Modify the program.
037	CAN NOT CHANGE PLANE IN CRC	G40 is commanded on the plane other than offset plane in cutter compensation B. The plane selected by using G17, G18 or G19 is changed in cutter compensation C mode. Modify the program.

Number	Message	Description
038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in cutter compensation C because the arc start point or end point coincides with the arc center. Modify the program.
041	INTERFERENCE IN CRC	Overcutting will occur in cutter compensation C. Two or more blocks are consecutively specified in which functions such as the auxiliary function and dwell functions are performed without movement in the cutter compensation mode. Modify the program.
042	G45/G48 NOT ALLOWED IN CRC	Tool offset (G45 to G48) is commanded in cutter compensation. Modify the program.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.15.4	Cutter compensation B
	II.15.5	Cutter compensation C

10.4.2 TOOL NOSE RADIUS COMPENSATION (T series)

General

It is difficult to produce the compensation necessary to form accurate parts when using only the tool offset function due to tool nose roundness in taper cutting or circular cutting. The tool nose radius compensation function compensates automatically for the above errors.

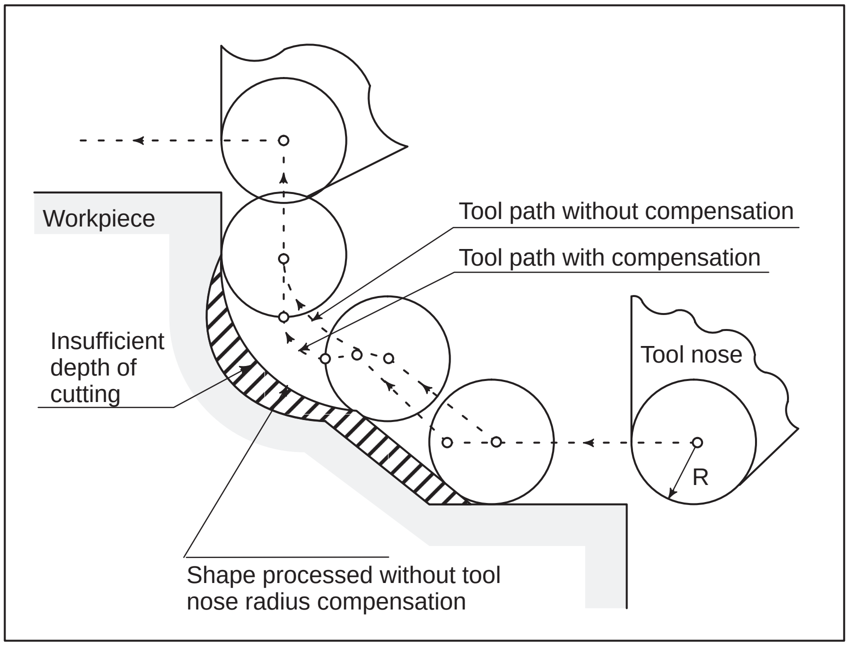


Fig. 10.4.2 Tool path of tool nose radius compensation

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5002	WNP							

[Data type] Bit

WNP Imaginary tool tip direction used for tool nose radius compensation, when the geometry/wear compensation option is equipped, is the direction specified by:
0 : Geometry offset number
1 : Wear offset number

	#7	#6	#5	#4	#3	#2	#1	#0
5003						CCN		

[Data type] Bit

CCN When automatic reference position return (G28) is specified in the cutter compensation C mode (M series) or in tool nose radius compensation (T series):

0 : The cutter compensation vector is cancelled in movement to an intermediate position.
1 : The cutter compensation vector is not cancelled in movement to an intermediate position, but is cancelled in movement to the reference position.

5010	Limit value that ignores the vector when a tool moves on the outside of a corner during too nose radius compensation

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 16383

This parameter sets the limit value that ignores a slight movement occurring when a tool moves on the outside of a corner during tool nose radius compensation.

Alarm and message

Number	Message	Description
033	NO SOLUTION AT CRC	A point of intersection cannot be determined for tool nose radius compensation. Modify the program. Modify the program.
034	NO CIRC ALLOWED IN ST–UP /EXT BLK	The start up or cancel was going to be performed in the G02 or G03 mode in tool nose radius compensation. Modify the program.
035	CAN NOT COMMANDED G31	Skip cutting (G31) was specified in tool nose radius compensation mode. Modify the program.
037	CAN NOT CHANGE PLANE IN NRC	The offset plane is switched in tool nose radius compensation. Modify the program.
038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in tool nose radius compensation because the arc start point or end point coincides with the arc center. Modify the program.
039	CHF/CNR NOT ALLOWED IN NRC	Chamfering or corner R was specified with a start–up, a cancel, or switching between G41 and G42 in tool nose radius compensation. The program may cause overcutting to occur in chamfering or corner R. Modify the program.

Number	Message	Description
040	INTERFERENCE IN G90/G94 BLOCK	Overcutting will occur in tool nose radius compensation in canned cycle G90 or G94. Modify the program.
041	INTERFERENCE IN NRC	Overcutting will occur in tool nose radius compensation. Modify the program.

Reference item

OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.15.2	Tool Nose Radius Compensation

11 PROGRAM COMMAND

11.1 DECIMAL POINT PROGRAMMING/ POCKET CALCULATOR TYPE DECIMAL POINT PROGRAMMING

General

Numerical values can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:
X, Y, Z, U, V, W, A, B, C, I, J, K, Q, R, and F . . . . M series
X, Y, Z, U, V, W, A, B, C, I, J, K, R, F. T series

There are two types of decimal point notation: calculator-type notation and standard notation.
When calculator-type decimal notation is used, a value without decimal point is considered to be specified in millimeters. When standard decimal notation is used, such a value is considered to be specified in least input increments. Select either calculator-type or standard decimal notation by using the DPI bit (bit 0 of parameter 3401). Values can be specified both with and without decimal point in a single program.

Program command	Pocket calculator type decimal point programming	Standard type decimal point programming
X1000 Command value without decimal point	1000mm Unit : mm	1mm Unit : Least input increment (0.001 mm)
X1000.0 Command value with decimal point	1000mm Unit : mm	1000mm Unit : mm

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3401								DPI

[Data type] Bit

DPI When a decimal point is omitted in an address that can include a decimal point
0 : The least input increment is assumed.
1 : The unit of mm, inches, or s is assumed. (Pocket calculator type decimal point programming)

Alarm and Message

Number	Message	Description
007	ILLEGAL USE OF DECIMAL POINT	Decimal point “ · ” input error (A decimal point was input after an address with which it can not be used. Or two decimal points were input.) Modify the program.

Reference Item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.9.4	Decimal point programming
OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.9.3	Decimal point programming

11.2 G CODE SYSTEM (T SERIES)

General

There are three G code systems : A,B, and C (Table 11.2). Select a G code system using bits 6 (GSB) and 7 (GSC) of parameter 3401.

Table 11.2 G code list (1/3)

G code A	G code B	G code C	Group	Function
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation CW or Helical interpolation CW
G03	G03	G03		Circular interpolation CW or Helical interpolation CCW
G04	G04	G04	00	Dwell
G05	G05	G05		High speed cycle cutting
G07.1 (G107)	G07.1 (G107)	G07.1 (G107)		Cylindrical interpolation
G10	G10	G10		Data setting
G12.1 (G112)	G12.1 (G112)	G12.1 (G112)	21	Polar coordinate interpolation mode
G13.1 (G113)	G13.1 (G113)	G13.1 (G113)		Polar coordinate interpolation cancel mode
G17	G17	G17	16	XpYp plane selection
G18	G18	G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G22	G22	G22	09	Stored stroke check function on
G23	G23	G23		Stored stroke check function off
G25	G25	G25	08	Spindle speed fluctuation detection off
G26	G26	G26		Spindle speed fluctuation detection on
G27	G27	G27	00	Reference position return check
G28	G28	G28		Reference position return
G30	G30	G30		2nd, 3rd and 4th reference position return
G30.1	G30.1	G30.1		Floating reference point return
G31	G31	G31		Skip function
G32	G33	G33	01	Thread cutting
G34	G34	G34		Variable–lead thread cutting
G36	G36	G36	00	Automatic tool compensation X
G37	G37	G37		Automatic tool compensation Z
G39	G39	G39		Corner circular interpolation
G40	G40	G40	07	Tool nose radius compensation cancel
G41	G41	G41		Tool nose radius compensation left
G42	G42	G42		Tool nose radius compensation right

Table 11.2 G code list (2/3)

G code A	G code B	G code C	Group	Function
G50	G92	G92	00	Coordinate system setting or max. spindle speed setting
G50.3	G92.1	G92.1		Workpiece coordinate system preset
G50.2 (G250)	G50.2 (G250)	G50.2 (G250)	20	Polygonal turning cancel
G51.2 (G251)	G51.2 (G251)	G51.2 (G251)		Polygonal turning
G52	G52	G52	00	Local coordinate system setting
G53	G53	G53		Machine coordinate system setting
G54	G54	G54	14	Workpiece coordinate system 1 selection
G55	G55	G55		Workpiece coordinate system 2 selection
G56	G56	G56		Workpiece coordinate system 3 selection
G57	G57	G57		Workpiece coordinate system 4 selection
G58	G58	G58		Workpiece coordinate system 5 selection
G59	G59	G59		Workpiece coordinate system 6 selection
G65	G65	G65	00	Macro calling
G66	G66	G66	12	Macro modal call
G67	G67	G67		Macro modal call cancel
G68	G68	G68	04	Mirror image for double turrets ON or balance cut mode
G69	G69	G69		Mirror image for double turrets OFF or balance cut mode cancel
G70	G70	G72	00	Finishing cycle
G71	G71	G73		Stock removal in turning
G72	G72	G74		Stock removal in facing
G73	G73	G75		Pattern repeating
G74	G74	G76		End face peck drilling
G75	G75	G77		Outer diameter/internal diameter drilling
G76	G76	G78		Multiple threading cycle
G71	G71	G72	01	Traverse grinding cycle (for grinding machine)
G72	G72	G73		Traverse direct constant–dimension grinding cycle (for grinding machine)
G73	G73	G74		Oscillation grinding cycle (for grinding machine)
G74	G74	G75		Oscillation direct constant–dimension grinding cycle (for grinding machine)
G80	G80	G80	10	Canned cycle for drilling cancel
G83	G83	G83		Cycle for face drilling
G84	G84	G84		Cycle for face tapping
G86	G86	G86		Cycle for face boring
G87	G87	G87		Cycle for side drilling
G88	G88	G88		Cycle for side tapping
G89	G89	G89		Cycle for side boring

Table 11.2 G code list (3/3)

G code			Group	Function
A	B	C		
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Thread cutting cycle
G94	G79	G24		End face turning cycle
G96	G96	G96	02	Constant surface speed control
G97	G97	G97		Constant surface speed control cancel
G98	G94	G94	05	Per minute feed
G99	G95	G95		Per revolution feed
—	G90	G90	03	Absolute programming
—	G91	G91		Incremental programming
—	G98	G98	11	Return to initial level
—	G99	G99		Return to R point level

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3401	GSC	GSB						

[Data type] Bit

GSB, GSC The G code system is set.

GSC	GSB	G code
0	0	G code system A
0	1	G code system B
1	0	G code system C

	#7	#6	#5	#4	#3	#2	#1	#0
3402		CLR			G91			G01

[Data type] Bit

G01 Mode entered when the power is turned on or when the control is cleared
0 : G00 mode (positioning)
1 : G01 mode (linear interpolation)

G91 When the power is turned on or when the control is cleared
0 : G90 mode (absolute command)
1 : G91 mode (incremental command)

CLR Reset button on the CRT/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 E of operator's manual (B–62444E).

Alarm and Message

Number	Message	Description
	IMPROPER G–CODE	An unusable G code or G code corresponding to the function not provided is specified. Modify the program.

Note

Note 1 If the CNC enters the clear state (see bit 6 (CLR) of parameter 3402) when the power is turned on or the CNC is reset, the modal G codes change as follows.

(1) G codes marked with ◤ in Table 11.2 are enabled.

(2) When the system is cleared due to power-on or reset, whichever specified, either G20 or G21, remains effective.

(3) G22 is set when the system is cleared due to power-on. When the system is cleared due to reset, whichever specified, either G22 or G23, remains effective.

(4) Setting bit 0 (G01) of parameter 3402 determines which code, either G00 or G01, is effective.

(5) Setting bit 3 (G91) of parameter 3402 determines which code, either G90 or G91, is effective.

Note 2 G codes of group 00 except G10 and G11 are single-shot G codes.

Note 3 Alarm 010 is displayed when a G code not listed in the G code list is specified or a G code without a corresponding option is specified.

Note 4 G codes of different groups can be specified in the same block. If G codes of the same group are specified in the same block, the G code specified last is valid.

Note 5 If a G code of group 01 is specified in a canned cycle, the canned cycle is canceled in the same way as when a G80 command is specified. G codes of group 01 are not affected by G codes for specifying a canned cycle.

Note 6 When G code system A is used for a canned cycle, only the initial level is provided at the return point.

Note 7 G codes are displayed for each group number.

Reference Item

OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.3	PREPARATORY FUNCTION (G FUNCTION)
	APPENDIX E	STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET

11.3 PROGRAM CONFIGURATION

General

A program consists of the following components:

Table 11.3 Program components

Components	Descriptions
Tape start	Symbol indicating the start of a program file
Leader section	Used for the title of a program file, etc.
Program start	Symbol indicating the start of a program
Program section	Commands for machining
Comment section	Comments or directions for the operator
Tape end	Symbol indicating the end of a program file

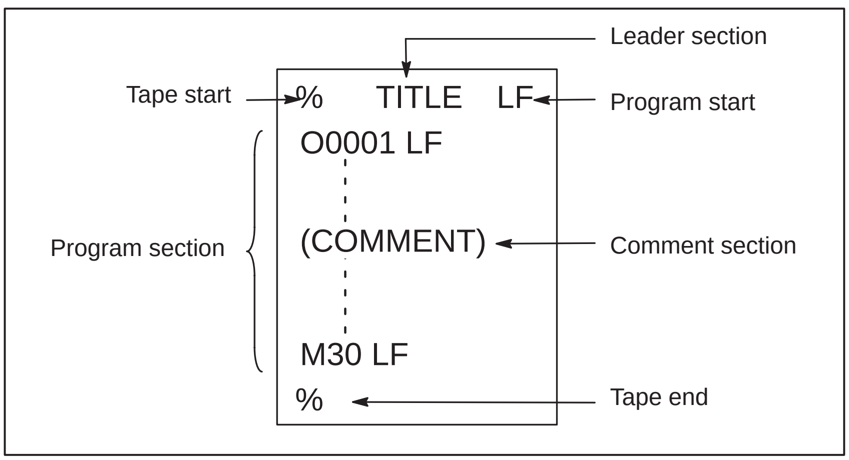


Fig. 11.3 Program configuration (Example of using ISO code)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0100							CTV	

Setting entry is acceptable.

[Data type] Bit

CTV: Character counting for TV check in the comment section of a program.
0 : Not performed
1 : Performed

	#7	#6	#5	#4	#3	#2	#1	#0
3201		NPE	N99					

[Data type] Bit

N99 With an M99 block, when bit 6 (NPE) of parameter No. 3201 = 0, program registration is assumed to be:
0 : Completed
1 : Not completed

NPE With an M02, M30, or M99 block, program registration is assumed to be:
0 : Completed
1 : Not completed

	#7	#6	#5	#4	#3	#2	#1	#0
3404		EOR				SBP		

[Data type] Bit

SBP Address P of the block including M198 in the subprogram call function
0 : Indicating a file number
1 : Indicating a program number

EOR When the end-of-record mark (%) is read during program execution:
0 : P/.S alarm No. 5010 occurs.
(Automatic operation is stopped, and the system enters the alarm state.)
1 : No alarm occurs.
(Automatic operation is stopped, and the system is reset.)

6030	M code that calls the program entered in file

[Data type] Byte

[Valid data range] 0, and 1 to 255

This parameter sets an M code that calls the program entered in a file.

Note 1 The M code is judged to be M198 when zero is specified as the setting value.

Alarm and Message

Number	Message	Description
001	TH PARITY ALARM	TH alarm (A character with incorrect parity was input).
002	TV PARITY ALARM	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective (when TVC, bit 0 of setting parameter 0000 is set to 1).
5010	END OF RECORD	The end of record (%) was specified.

Reference Item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.13	Program Configuration
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.13	Program Configuration

11.4 INCH/METRIC CONVERSION

General

Either inch or metric input can be selected by G code.

Signal

Inch input signal INCH <F002#0>

[Classification] Output signal

[Function] This signal indicates that inch input mode is in progress.

[Output condition] "1"indicates that the inch input mode (G20) is in progress, and "0" indicates that metric input mode (G21) is in progress.

This signal changes to the corresponding state when modes are switched using the setting data display on the MDI panel.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F002								INCH

Parameter

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

Setting entry is acceptable.

[Data type] Bit

INI Unit of input
0: In mm
1: In inches

	#7	#6	#5	#4	#3	#2	#1	#0
1001								INM

Note 1 When this parameter is set, the power must be turned off before operation is continued.

[Data type] Bit

INM Least command increment on the linear axis
0 : In mm (metric system machine)
1 : In inches (inch system machine)

1006	#7	#6	#5	#4	#3	#2	#1	#0
							ROSx	ROTx

Note 1 When this parameter is changed, turn off the power before continuing operation.

[Data type] Bit axis

ROTx, ROSx Setting linear or rotation axis

ROSx	ROTx	Description
0	0	Linear axis ● Inch/metric conversion is done. ● All coordinate values are linear axis type. (Not rounded in 0 to 360°) ● Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotation axis (A Type) ● Inch/metric conversion is not done. ● Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values and relative coordinate values are rounded or not rounded by parameter No. 1008 #0 and #2. ● Stored pitch error compensation is the rotation type. Refer to parameter No. 3624. ● Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	0	Setting is invalid (unused)
1	1	Rotation axis (B type) ● Inch/metric conversion is not done. ● Machine coordinate values is linear axis type and is not rounded in 0 to 360°. Absolute coordinate values and relative coordinate values are rounded or not rounded by parameter No. 1008 #0 and #2. ● Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) ● Cannot be used with the rotation axis roll over function and the index table indexing function (M series).

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

[Data type] Bit

ZPI Coordinates at the reference position when a coordinate system is set automatically

0 : Value set in parameter No. 1250 is used.

1 : For input in mm, the value set in parameter 1250 is used, or for input in inches, the value set in parameter No. 1251 is used.

1250	Coordinate value of the reference position used when automatic coordinate system setting is performed

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Linear axis (Metric input)	0.01	0.001	0.0001	mm
Linear axis (Inch input)	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] – 99999999 to 99999999

Set the coordinate value of the reference position on each axis used for setting a coordinate system automatically.

1251	Coordinate value of the reference position on each axis used for setting a coordinate system automatically when input is performed in inches

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] – 99999999 to 99999999

Set the coordinate value of the reference position on each axis used for setting a coordinate system automatically when input is performed in inches.

Note 1 This parameter is valid when ZPI in parameter 1201 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1403								MIF

[Data type] Bit

MIF Cutting feedrates at feed per minute is specified by F commands

0 : In units of 1 mm/min for millimeter machines or 0.01 inches/min for inch machines.

1 : In unit of 0.001 mm/min for millimeter machines or 0.00001 inches/min for inch machines.

Note 1 M series are not equipped with this parameter. Cutting feedrates are specified by F commands in units of 0.001 mm/min for millimeter machines or 0.00001 inches/min for inch machines.

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

[Data type] Bit

MCN Machine position is:

0 : Not displayed according to the unit of input.
(Regardless of whether input is made in mm or inches, the machine position is displayed in mm for millimeter machines, or in inches for inch machines.)

1 : Displayed according to the unit of input.
(When input is made in mm, the machine position is displayed in mm, and when input is made in inches, the machine position is displayed in inches accordingly.)

	#7	#6	#5	#4	#3	#2	#1	#0
3405								AUX

[Data type] Bit

AUX The least increment of the command of the second miscellaneous function specified with a decimal point
0 : Assumed to be 0.001
1 : Depending on the input increment.
(For input in mm, 0.001 is assumed, or for input in inches, 0.0001 is assumed.)

	#7	#6	#5	#4	#3	#2	#1	#0
5006								OIM

[Data type] Bit

OIM When the unit is switched between the inch and metric systems, automatic tool compensation value conversion is:
0 : Not performed
1 : Performed

	#7	#6	#5	#4	#3	#2	#1	#0
8003								PIM

[Data type] Bit

PIM When only the axes controlled by the PMC are used, the linear axis is:
0: Influenced by inch/millimeter input.
1: Not influenced by inch/millimeter input.

Note

Note 1 When the least input increment and the least command increment systems are different, the maximum error is half of the least command increment. This error is not accumulated.

Note 2 When switching inch input (G20) to metric input (G21) and vice versa, the tool compensation value must be re–set according to the least input increment.
However, when bit 0 (OIM) of parameter 5006 is 1, tool compensation values are automatically converted and need not be re–set.

Note 3 Reference position return is performed at a low speed for the first G28 command after the inch input is switched to the metric input or vice versa.

Note 4 The inch and metric input can also be switched using settings.

Reference Item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.9.3	Inch/metric conversion(G20, G21)
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.9.2	Inch/metric conversion(G20, G21)

11.5 HIGH SPEED CYCLE CUTTING

General

This function can convert the machining profile to a data group that can be distributed as pulses at high-speed by the macro compiler and macro executor. The function can also call and execute the data group as a machining cycle using the CNC command (G05 command).

Note 1 This function cannot be used for two-path control.

• Format

G05 P10○○○ L○○○ ;

P10○○○ is number of the cutting cycle to be called first:
P10001 to P10999
L○○○ is repetition count of the cutting cycle
(L1 applies when this parameter is omitted.) :
L1 to L999

Call and execute the data for the high speed cutting cycle specified by the macro compiler and macro executor using the above command.
Cycle data can be prepared for up to 999 cycles. Select the machining cycle by address P. More than one cycle can be called and executed in series using the cycle connection data in the header.
Specify the repetition count of the called machining cycle by address L. The repetition count in the header can be specified for each cycle.
The connection of cycles and their repetition count are explained below with an example.

Example) Assume the following:

Cycle 1 Cycle connection data 2 Repetition count 1
Cycle 2 Cycle connection data 3 Repetition count 3
Cycle 3 Cycle connection data 0 Repetition count 1
G05 P10001 L2 ;
The following cycles are executed in sequence:
Cycles 1, 2, 2, 2, 3, 1, 2, 2, 2, and3

• Number of control axes

Six axes maximum can be controlled. Six axes can be controlled simultaneously.

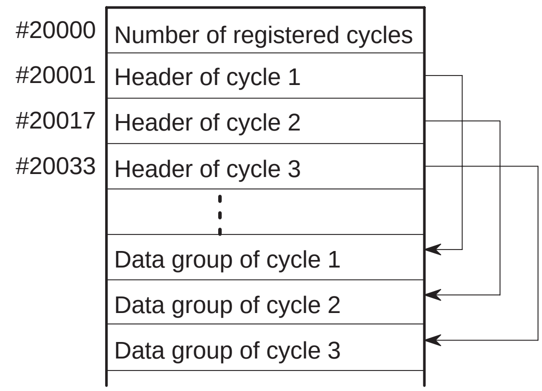
• Pulse distribution

Set the number of pulses per cycle in parameter 7501#4 to #6 as a macro variable (#20000 to #85535) for high speed cycle cutting using the macro compiler and macro executor.
The unit for the number of pulses is the least input increment.

- **Configuration of high–speed cycle cutting data**

Data for the high speed cycle cutting is assigned to variables (#20000 to #85535) for the high–speed cycle cutting by the macro compiler and macro executor.

Configuration of the high speed cycle cutting data



- **Number of Registered Cycles**

Specify the number of cycles (number of headers) of high-speed cycle cutting data. Values from 1 to 999 can be specified.

- **Header**

The header for high-speed cycle cutting data has the following configuration:

Header configuration

#20001/20017/20033..	Cycle repetition count
#20002/20018/20034..	Cycle connection data
#20003/20019/20035..	Number of data items
#20004/20020/20036..	Data type
#20005/20021/20037..	Variable assigned to the 1st axis data
#20006/20022/20038..	Variable assigned to the 2nd axis data
#20007/20023/20039..	Variable assigned to the 3rd axis data
#20008/20024/20040..	Variable assigned to the 4th axis data
#20009/20025/20041..	Variable assigned to the 5th axis data
#20010/20026/20042..	Variable assigned to the 6th axis data
#20011/20027/20043..	Total number of fixed data items for the 1st axis
#20012/20028/20044..	Total number of fixed data items for the 2nd axis
#20013/20029/20045..	Total number of fixed data items for the 3rd axis
#20014/20030/20046..	Total number of fixed data items for the 4th axis
#20015/20031/20047..	Total number of fixed data items for the 5th axis
#20016/20032/20048..	Total number of fixed data items for the 6th axis

Explanations

- **Cycle repetition count**

Specify the repetition count for the cycle. Values from 0 to 32767 can be specified. When 0 or 1 is specified, the cycle is executed once.

- **Cycle connection data**

Specify the number (1 to 999) of the cycle to be executed after the cycle. When no connection cycle exists because of the last cycle, specify 0.

- **Number of data items**

Specify the number of data items per cycle. Valid values are from 1 to 32767.
When a fixed data item is specified, the fixed data is repeated for the specified number of times in one cycle.

- **Data type**

15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
–	–	r6	r5	r4	r3	r2	r1	–	–	t6	t5	t4	t3	t2	t1

The bits from t1 to t6, corresponding to the 1st to 6th axes, have the following meanings:
0: Distribution data is always constant.
1: Distribution data is variable or fixed.
When the distribution data is variable or fixed, the bits from r1 to r6, corresponding to the 1st to 6th axes, have the following meanings:
0: Distribution data is read forward.
1: Distribution data is read backwards.
Because the data consists of bits, it is necessary to use a binary-coded decimal value when setting it using the macro compiler and macro executor.
Example)
When constant data is assigned to the 1st and 2nd axes and variable data is assigned to the 3rd and 4th axes, #20004 = 12; (t4 and t3: 1, t2 and t1: 0)

- **Variables assigned to data for the 1st to 6th axes**

·**Constant data**
When the corresponding data type bit (t6 to t1) is 0, specify "distribution data value".

·**Variable data**
When the corresponding data type bit (t6 to t1) is 1 and the total number of fixed data items = 0, specify "(Storing start data variable No. of the distribution data)/10".

·**Fixed data**
When the corresponding data type bit (t6 to t1) is 1 and the total number of fixed data items is other than 0, specify "(Storing start data variable No. of the distribution data)/10".
The applicable value for the variable data and fixed data is 2001 to 8553. It is not possible to start storing data in the executable format from a variable No. that is not a multiple of 10.
To read the distribution data backwards, set the variable No. of the data to be distributed last. For example, to read the distribution data in #25000 to #25999 backwards, set 25000 as the data assignment variable.

- **Total number of fixed data items for the first to 6th axes**

Set the length of the fixed data for the cycle.
The first address of the fixed data must be specified by the data assignment variable. When the total number of fixed data items = 0 and the corresponding data type bit (t6 to t1) is 1, the data is regarded as a variable data.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7501	IPC	IT2	IT1	IT0				CSP
	IPC	IT2	IT1	IT0				

[Data type] Bit

CSP Cs contouring control function dedicated to a piston lathe is
0: Not used.
1: Used.

IT0, IT1, IT2

IT2	IT1	IT0	
0	0	0	Interpolates the G05 data in 8ms
0	0	1	Interpolates the G05 data in 2ms
0	1	0	Interpolates the G05 data in 4ms
0	1	1	Interpolates the G05 data in 1ms
1	0	0	Interpolates the G05 data in 16ms

IPC

0: The system does not monitor whether a distribution process is stopped while high-speed cutting (G05) is performed with high-speed remote buffer A or B or in a high-speed cycle.
1: The system monitors whether a distribution process is stopped while high-speed machining (G05) is performed with high-speed remote buffer A or B or in a high-speed cycle cutting.
(Alarms 179 and 000 are simultaneously issued if the distribution process is stopped. In this case, the power must be turned off then on again.)

Note 1 The distribution process stops, when the host cannot send data with the high-speed remote buffer by the specified time.

	#7	#6	#5	#4	#3	#2	#1	#0
7505							HUNx	HSCx
								HSCx

Note 1 After setting this parameter, the power must be turned off then on again.

[Data type] Bit

HSCx Specifies whether each axis is used for high-speed distribution in a high-speed cycle cutting or with a high-speed remote buffer.
0: Used for high-speed distribution
1: Not used for high-speed distribution

HUNx Specifies whether the unit of data to be distributed during cutting in a high-speed cycle is ten times the least input increment.
0: The unit of data is the same as the least input increment.
1: The unit of data is ten times the least input increment.

Note 1 This parameter is used when a data item to be distributed exceeds a word in terms of the least input increment or the maximum travel speed.
CNC distributes ten times the value for cutting in a high–speed cycle for the axes in which HUNx of this parameter is set to 1. Therefore, set a value one tenth the value to be distributed for cutting in a high-speed cycle along the specified axes.

7510	Maximum number of simultaneously controlled axes when G05 is specified during high–speed cycle cutting or No. of controlled axes in high–speed remote buffer

[Data type] Word

[Unit of data] 1A6

This parameter sets the maximum number of simultaneous control axes when G05 is specified during high-speed cycle cutting or sets the number of control axes in a high-speed remote buffer.

7511	Extension of data variables used for cutting in a high–speed cycle

[Data type] Byte

[Valid data range] 0 to 8

This parameter sets the size for extending the data variables used for cutting in a high-speed cycle (Variables 200000 to 462143).

Set value	Explanation
0	Variable #200000 is not used. Conventional variables #200000 to #85535 are used.
1	Variables #200000 to #232767 are used.
2	Variables #200000 to #265535 are used.
3	Variables #200000 to #298303 are used.
4	Variables #200000 to #331072 are used.
5	Variables #200000 to #363839 are used.
6	Variables #200000 to #396607 are used.
7	Variables #200000 to #429375 are used.
8	Variables #200000 to #462143 are used.

Alarm and Message

Number	Message	Description
115	ILLEGAL VARIABLE NUMBER	The header contents are improper in a high–speed cycle cutting. This alarm is given in the following cases: 1. The header corresponding to the specified cutting cycle number called is not found. 2. The cycle connection data value is out of the allowable range (0 – 999). 3. The number of data in the header is out of the allowable range (0 – 32767). 4. The storing start data variable number of executable format data is out of the allowable range (#20000 – #85535). 5. The storing data variable number of executable format data is out of the allowable range (#85535). 6. The storing start data variable number of executable format data is overlapped with the variable number used in the header. Modify the program.
178	G05 COMMANDED IN G41/G42 MODE	G05 was commanded in the G41/G42 mode. Correct the program.
179	PARAM. (NO. 7510) SETTING ERROR	The number of controlled axes set by the parameter 7510 exceeds the maximum number. Modify the parameter setting value.

Note

Note 1 When the high-speed cycle cutting function is used, the extended RAM is necessary. The length of tape that can be specified is limited to 320 meters.

Note 2 Single block stop, dry run, feedrate override, automatic acceleration/deceleration and handle interruption are disabled during high-speed cycle cutting.

Note 3 Set the total number of distribution data items for one cycle to a multiple of the following values, according to the distribution cycle. This does not apply when the distribution cycle is 16 ms or 8 ms.
If the total number is not a multiple of one of the following values, movement in the remaining cycle becomes zero.

Distribution cycle 4 ms: Multiple of 2
Distribution cycle 2 ms: Multiple of 4
Distribution cycle 1 ms: Multiple of 8

For example, when all 41 data items (distribution cycle: 2 ms) are specified, movement is zero in the remaining 3 ms.

Reference Item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.20.1	High speed cycle cutting
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.19.1	High speed cycle cutting

11.6 CUSTOM MACRO

11.6.1 Custom Macro

General

Although subprograms are useful for repeating the same operation, the custom macro function also allows use of variables, arithmetic and logic operations, and conditional branches for easy development of general programs such as pocketing and user–defined canned cycles. A machining program can call a custom macro with a simple command, just like a subprogram.

Machining program

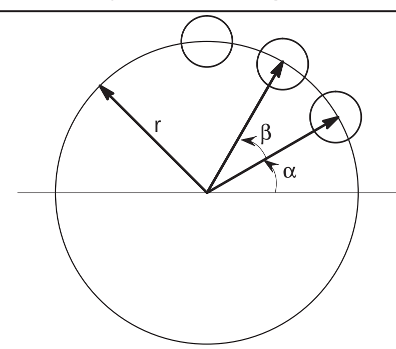
```
O0001 ;
  :
  :
  :
G65 P9010 R50.0 L2 ;
  :
  :
M30 ;
```

Custom macro

```
O9010 ;
#1=#18/2 ;
G01 G42 X#1 Y#1 F300 ;
G02 X#1 Y–#1 R#1 ;
  :
  :
  :
M99 ;
```

This means that a function of general use can be formed when programming a certain function as a custom macro. That is, programs can be written using variables for data that might change or be unknown. This can be further applied to group technology.

Similar workpieces can be collected as a group and a universal custom macro body can be programmed using variables applicable to each group. In this way, programming is not required for the workpieces in the group. The programmer only need to assign actual values to the variables.



Bolt hole circles as shown in the above figure can be made easily.

Once a custom macro body for the bolt hole circle is programmed and registered, the CNC can operate as if it has the bolt hole circle cutting function.

Programmers can use the bolt hole circle function by using the following command only:

(Example of calling bolt hole circle)

G65 P$\underline{p}$ R$\underline{r}$ A$\underline{\alpha}$ B$\underline{\beta}$ K$\underline{k}$;
P : Macro number of bolt hole circle
r : Radius
α : Start angle
β : Angle between circles
k : Number of circles

Signal

Custom Macro Input Signal UI000AUI015 <G054, G055>

[Classification] Input signal

[Function] No function is provided for the control unit. These signals can be read by a custom macro as a type of system variable, and are used for interface signals between custom macros and the PMC.
These signals correspond to system variables as indicated below.

Signals	Q'ty	Variables	Correspondence of values
UI000 UI001 UI002 UI003 : UI014 UI015	1 1 1 1 : 1 1	#1000 #1001 #1002 #1003 : #1014 #1015	"0" at "0" and "1" at "1"
UI000AUI015	16	#1032	16–bit binary code *1

**1 Variable value* $\#1032 = \sum_{i=0}^{15} \{\#[1000 + i] \times 2^{i}\}$

Custom Macro Output Signal UI000AUI015 <F054, F055> UO100AUO131 <F056AF059>

[Classification] Output signal

[Function] No function is provided for the control unit. These signals can be read or written by a custom macro as a type of system variable, and are used for interface signals between custom macros and the PMC.
These signals correspond to system variables as indicated below.

Signals	Q'ty	Variables	Correspondence of values
UO000 UO001 UO002 UO003 : UO014 UO015	1 1 1 1 : 1 1	#1100 #1101 #1102 #1103 : #1114 #1115	"0" at "0" and "1" at "1"
UO000AUO015 UO100AUO115	16 32	#1132 #1133	16–bit binary code *1 32–bit binary code *2

*1 Variable value $\#1132 = \sum_{i=0}^{15} \{\#[1100 + i] \times 2^i\}$

*2 Variable value $\#1133 = \sum_{i=0}^{30} \{2^i \times V_i\} - 2^{31} \times V_{31}$

Where Vi=0 when UO1i is H0Iand Vi=1 when UO1i is H1I

These system variables can be used on the left side of an assignment statement as well as on the right side.
The value assigned to the system variable used on the left side last is used for the value of the system variable to be assigned on the right side.

Signal Address

	#7	#6	#5	#4	#3	#2	#1	#0
G054	UI007	UI006	UI005	UI004	UI003	UI002	UI001	UI000
G055	UI015	UI014	UI013	UI012	UI011	UI010	UI009	UI008
F054	UO007	UO006	UO005	UO004	UO003	UO002	UO001	UO000
F055	UO015	UO014	UO013	UO012	UO011	UO010	UO009	UO008
F056	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100
F057	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108
F058	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
F059	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124

Parameter

- **Setting for single block stop**

	#7	#6	#5	#4	#3	#2	#1	#0
6000			SBM					

[Data type] Bit

SBM Custom macro statement
0 : Not stop the single block
1 : Stops the single block

- **Other settings**

	#7	#6	#5	#4	#3	#2	#1	#0
6001	CLV	CCV	TCS	CRO			PRT	

[Data type] Bit

PRT Reading zero when data is output using a DPRNT command
0 : Outputs a space
1 : Outputs no data

CRO ISO code output using a BPRNT command or a DPRNT command
0 : Outputs only LF after data is output
1 : Outputs LF and CR after data is output

TCS Custom macro (subprogram)
0 : Not called using a T code
1 : Called using a T code

CCV Custom macro’s common variables Nos. 100 through 149
0 : Cleared to “vacant” by reset
1 : Not cleared by reset

CLV Custom macro’s local variables #1 through #33
0 : Cleared to “vacant” by reset
1 : Not cleared by reset

● Setting when macro statement is input/output with EIA code

	#7	#6	#5	#4	#3	#2	#1	#0
6010	*7	*6	*5	*4	*3	*2	*1	*0

	#7	#6	#5	#4	#3	#2	#1	#0
6011	=7	=6	=5	=4	=3	=2	=1	=0

	#7	#6	#5	#4	#3	#2	#1	#0
6012	#7	#6	#5	#4	#3	#2	#1	#0

	#7	#6	#5	#4	#3	#2	#1	#0
6013	[7	[6	[5	[4	[3	[2	[1	[0

	#7	#6	#5	#4	#3	#2	#1	#0
6014	]7	]6	]5	]4	]3	]2	]1	]0

[Data type] Bit
These parameters are used to input/output macro statements.

The numeral of a suffix indicates the bit position in a code.

***0 to *7** Set the hole pattern of an EIA code indicating *.

=0 to =7 Set the hole pattern of an EIA code indicating =.

#0 to#7 Set the hole pattern of an EIA code indicating #.

[0 to [7 Set the hole pattern of an EIA code indicating [.

]0 to]7 Set the hole pattern of an EIA code indicating].

0 :Corresponding bit is 0

1 :Corresponding bit is 1.

● Setting an M code that calls a program entered in a file

6030	M code that calls the program entered in file

[Data type] Byte

[Valid data range] 0, and 1 to 255
This parameter sets an M code that calls the program entered in a file.

Note 1 The M code is judged to be M198 when zero is specified as the setting value.

• Setting G codes that call custom macros of program Nos.9010 to 9019

6050	G code that calls the custom macro of program number 9010
6051	G code that calls the custom macro of program number 9011
6052	G code that calls the custom macro of program number 9012
6053	G code that calls the custom macro of program number 9013
6054	G code that calls the custom macro of program number 9014
6055	G code that calls the custom macro of program number 9015
6056	G code that calls the custom macro of program number 9016
6057	G code that calls the custom macro of program number 9017
6058	G code that calls the custom macro of program number 9018
6059	G code that calls the custom macro of program number 9019

[Data type] Word

[Valid data range] 1 to 9999

These parameters set the G codes that call the custom macros of program numbers 9010 through 9019.

Note 1 Setting value 0 is invalid. No custom macro can be called by G00.

• Setting M codes that call subprograms of program Nos.9001 to 9009

6071	M code that calls the subprogram of program number 9001
6072	M code that calls the subprogram of program number 9002
6073	M code that calls the subprogram of program number 9003
6074	M code that calls the subprogram of program number 9004
6075	M code that calls the subprogram of program number 9005
6076	M code that calls the subprogram of program number 9006
6077	M code that calls the subprogram of program number 9007
6078	M code that calls the subprogram of program number 9008
6079	M code that calls the subprogram of program number 9009

[Data type] Two–word

[Valid data range] 1 to 99999999

These parameters set the M codes that call the subprograms of program numbers 9001 through 9009.

Note 1 Setting value 0 is invalid. No custom macro can be called by M00.

- **Setting M codes that call custom macros of no.9020 to 9029**

6080	M code that calls the custom macro of program number 9020
6081	M code that calls the custom macro of program number 9021
6082	M code that calls the custom macro of program number 9022
6083	M code that calls the custom macro of program number 9023
6084	M code that calls the custom macro of program number 9024
6085	M code that calls the custom macro of program number 9025
6086	M code that calls the custom macro of program number 9026
6087	M code that calls the custom macro of program number 9027
6088	M code that calls the custom macro of program number 9028
6089	M code that calls the custom macro of program number 9029

[Data type] Two–word

[Valid data range] 1 to 99999

These parameters set the M codes that call the custom macros of program numbers 9020 through 9029.

Note 1 Setting value 0 is invalid. No custom macro can be called by M00.

- **ASCII codes that call subprogram of program No. 9004**

6090	ASCII code that calls the subprogram of program number 9004
6091	ASCII code that calls the subprogram of program number 9005

[Data type] Byte

[Valid data range] 65 (A:41H) to 90 (Z:5AH)

These parameters set the ASCII codes that call subprograms in decimal.
Addresses that can be used are as follows:
T series : A, B, F, H, I, K, M, P, Q, R, S, T
M series: A, B, D, F, H, I, J, K, L, M, P, Q, R, S, T, X, Y, Z

Note 1 Set 0 when no subprogram is called

Alarm and message

Number	Message	Description
076	ADDRESS P NOT DEFINED	Address P (program number) was not commanded in the block which includes an M98, G65, or G66 command. Modify the program.
077	SUB PROGRAM NESTING ERROR	The subprogram was called in five folds. Modify the program.
078	NUMBER NOT FOUND	A program number or a sequence number which was specified by address P in the block which includes an M98, M99, M65 or G66 was not found. The sequence number specified by a GOTO statement was not found. Otherwise, a called program is being edited in background processing. Correct the program, or discontinue the background editing.
110	DATA OVERFLOW	The absolute value of fixed decimal point display data exceeds the allowable range. Modify the program.
111	CALCULATED DATA OVERFLOW	The result of calculation is out of the allowable range (-10^{47} to -10^{-29}, 0, and 10^{-29} to 10^{47}).
112	DIVIDED BY ZERO	Division by zero was specified. (including tan 90°)
113	IMPROPER COMMAND	A function which cannot be used in custom macro is commanded. Modify the program.
114	FORMAT ERROR IN MACRO	There is an error in other formats than <Formula>. Modify the program.
115	ILLEGAL VARIABLE NUMBER	A value not defined as a variable number is designated in the custom macro, or the header contents are improper in a high–speed cycle cutting. This alarm is given in the following cases: High speed cycle machining 1. The header corresponding to the specified machining cycle number called is not found. 2. The cycle connection data value is out of the allowable range (0 – 999). 3. The number of data in the header is out of the allowable range (0 – 32767). 4. The storing start data variable number of executable format data is out of the allowable range (#20000 – #85535). 5. The storing data variable number of executable format data is out of the allowable range (#85535). 6. The storing start data variable number of executable format data is overlapped with the variable number used in the header. Modify the program.

Number	Message	Description
116	WRITE PROTECTED VARIABLE	The left side of substitution statement is a variable whose substitution is inhibited. Modify the program.
118	PARENTHESIS NESTING ERROR	The nesting of bracket exceeds the upper limit (quintuple). Modify the program.
119	ILLEGAL ARGUMENT	The SQRT argument is negative, BCD argument is negative, or other values than 0 to 9 are present on each line of BIN argument. Modify the program.
122	DUPLICATE MACRO MODAL–CALL	The macro modal call is specified in double. Modify the program.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
124	MISSING END STATE-MENT	DO – END does not correspond to 1 : 1. Modify the program.
125	FORMAT ERROR IN MACRO	<Formula> format is erroneous. Modify the program.
126	ILLEGAL LOOP NUMBER	In DOn, 1≦ n ≦ 3 is not established. Modify the program.
127	NC, MACRO STATEMENT IN SAME BLOCK	NC and custom macro commands coex-ist. Modify the program.
128	ILLEGAL MACRO SE-QUENCE NUMBER	The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched. Modify the program.
129	ILLEGAL ARGUMENT ADDRESS	An address which is not allowed in <Argu-ment Designation > is used. Modify the program.
199	MACRO WORD UNDE-FINED	Undefined macro word was used. Modify the custom macro.

Note

Request:

Machine tool builders: You are requested to attach your custom macro program tape or program list to the CNC unit without fail.

If it is necessary to replace part program storage memory due to a failure, FANUC servicemen or end user in charge of maintenance should know the contents of your custom macro for the purpose of repairing the trouble immediately.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.16	Custom macro
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.16	Custom macro

11.6.2 Interruption Type Custom Macro

General

When a program is being executed, another program can be called by inputting an interrupt signal (UINT) from the machine. This function is referred to as an interruption type custom macro function. Program an interrupt command in the following format:

M96 P○○○○ ;	Enables custom macro interrupt
M97 ;	Disables custom macro interrupt

Use of the interruption type custom macro function allows the user to call a program during execution of an arbitrary block of another program. This allows programs to be operated to match situations which vary from time to time.

(1) When a tool abnormality is detected, processing to handle the abnormality is started by an external signal.

(2) A sequence of machining operations is interrupted by another machining operation without the cancellation of the current operation.

(3) At regular intervals, information on current machining is read.

Listed above are examples like adaptive control applications of the interruption type custom macro function.

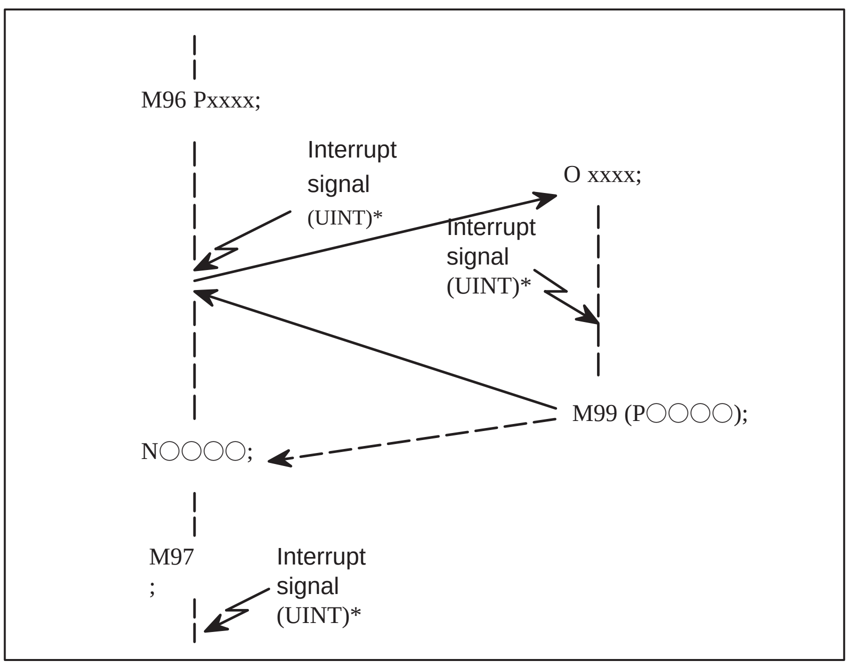


Fig 11.6.2 Interruption type custom macro function

When M96Pxxxx is specified in a program, subsequent program operation can be interrupted by an interrupt signal (UINT) input to execute the program specified by Pxxxx.
When the interrupt signal (UINT, marked by * in Fig. 11.6.2) is input during execution of the interrupt program or after M97 is specified, it is ignored.

Signal

Interrupt Signal for Custom Macro UNIT<G053#3>

[Classification] Input signal

[Function] This signal calls and executes a program in memory. During execution, a program in automatic operation is suspended.
To enable this signal to be accepted, a particular miscellaneous function must be specified in a command program for automatic operation. In addition, automatic operation must already be started to accept this signal.
The particular miscellaneous function code is set by parameter 6003, 6033 and 6034.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G053					UNIT			

Parameter

● Various Setting for Custom Macro

	#7	#6	#5	#4	#3	#2	#1	#0
6003	MUS	MCY	MSB	MPR	TSE	MIN	MSK	

[Data type] Bit

MSK Absolute coordinates at that time during custom macro interrupt
0 : Not set to the skip coordinates (system variables #5061 and later)
1 : Set to the skip coordinates (system variables #5601 and later)

MIN Custom macro interrupt
0 : Performed by interrupting an in–execution block (Custom macro interrupt type I)
1 : Performed after an in–execution block is completed (Custom macro interrupt type II)

TSE Custom macro interrupt signal UINT
0 : Edge trigger method (Rising edge)
1 : Status trigger method

MPR Custom macro interrupt valid/invalid M code
0 : M96/M97
1 : M code set using parameters (Nos. 6033 and 6034)

MSB Interrupt program
0 : Uses a dedicated local variable (Macro–type interrupt)
1 : Uses the same local variable as in the main program (Subprogram–type interrupt)

MCY Custom macro interrupt
0 : Not performed during cycle operation
1 : Performed during cycle operation

MUS Interrupt–type custom macro
0 : Not used
1 : Used

- **Setting M code that makes interruption effective and ineffective**

6033	M code that validates a custom macro interrupt
6034	M code that invalidates a custom macro interrupt

[Data type] Byte type

[Valid data range] 0A255

These parameters set the custom macro interrupt valid/invalid M codes.

Note 1 These parameters can be used when MPR, #4 of parameter No. 6003, is 1. M96 is used as a valid M code and M97 is used as an invalid M code when MPR is 0, irrespective of the state of this parameter.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.16.11	Interruption type custom macro
OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.16.11	Interruption type custom macro

11.6.3 Custom Macro Variables Common To Two–path Control (T series (Two–path control))

With two–path control, common variables are provided separately for each path; variable #n used with a path 1 is different from variable #n used with path 2. By parameter setting (No. 6036 and No. 6037), however, some or all of common variables #100 to # 149 and #500 to #531 can be made usable commonly by path 1 and path 2 so that such variables can be written or read for either path. Such variables are referred to as custom macro variables common between two paths.

Parameter

- **Setting the no. of custom macro variables common between two paths**

6036	Number of custom macro variables common between two paths (#100's)

[Data type] Byte

[Unit of data] Number of custom macro variables

[Valid data range] 0 to 50

The parameter specifies the number of variables commonly used for both paths 1 and 2 (custom macro variables common between two paths) that are included in custom macro variables 100 to 149.
The custom macro variables common between two paths can be written or read for either of the paths.

Example) When this parameter is set to 10, the custom macro variables are specified as follows:
Custom macro variables 100 to 109: Used commonly between two paths
Custom macro variables 110 to 149: Used independently for each path

Note 1 This parameter is dedicated to the 2–path control.

Note 2 When this parameter is set to 0, custom macro variables 100 to 149 are not used commonly between two paths.

Note 3 Custom macro variables that can be used as custom macro variables common between two paths are from 100 to 149. Custom macro variable 150 and subsequent custom macro variables cannot be used commonly between two paths, even if this parameter is set to 51 or more.

- **Setting the No. of custom macro variables common between two paths**

6037	Number of custom macro variables common between two paths (#500’s)

[Data type] Byte

[Unit of data] Number of custom macro variables

[Valid data range] 0 to 32

This parameter specifies the number of variables commonly used for both paths 1 and 2 (custom macro variables common between two paths) that are part of custom macro variables 500 to 531.
The custom macro variables common to tool posts can be written or read for either of the tool posts.

Example) When this parameter is set to 10, the custom macro variables are specified as follows:
Custom macro variables 500 to 509: Used commonly between two paths
Custom macro variables 510 to 531: Used independently for each path

Note 1 This parameter is dedicated to the 2–path control.

Note 2 When this parameter is set to 0, custom macro variables 500 to 531 are not used commonly between two paths.

Note 3 Custom macro variables that can be used as custom macro variables common between two paths are from 500 to 531. Custom macro variable 532 and subsequent custom macro variables cannot be used commonly for both tool posts, even if this parameter is set to 33 or more.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.22.3	Custom macro variables common to tool posts
OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.21.5	Custom macro variables common to tool posts

11.7 CANNED CYCLE (M series) / CANNED CYCLE FOR HOLE MACHINING (T series)

General

Canned cycles make it easier for the programmer to create programs. With a canned cycle, a frequently–used machining operation can be specified in a single block with a G function; without canned cycles, normally more than one block is required. In addition, the use of canned cycles can shorten the program to save memory.

Explanations

A canned cycle consists of a sequence of six operations.

Operation 1 Positioning a hole position
Operation 2 Rapid traverse up to point R level
Operation 3 Hole machining
Operation 4 Operation at the bottom of a hole
Operation 5 Retraction to point R level
Operation 6 Rapid traverse up to the initial point

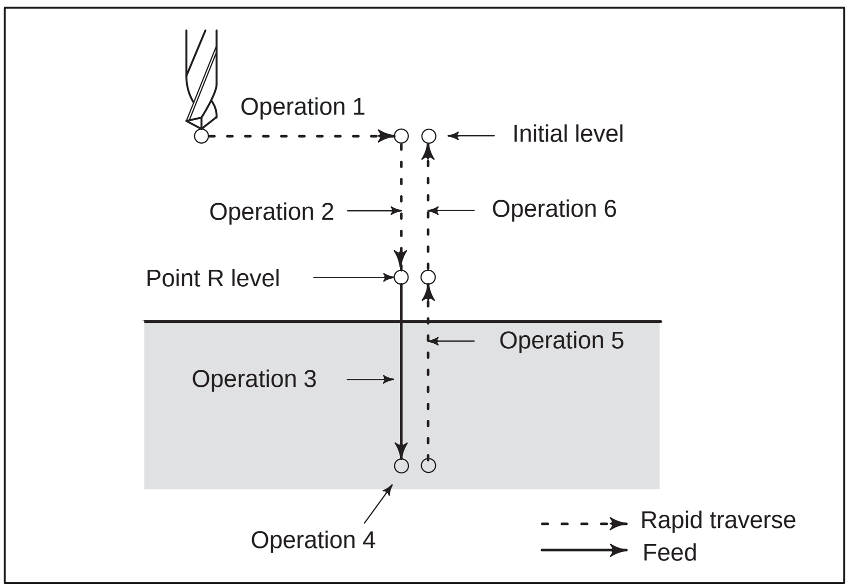


Fig. 11.7 (a) Canned cycle operation sequence

SPINDLE CONTROL

In some canned cycles, a spindle command to rotate the spindle in reverse direction may be output.

The following canned cycles require spindle control:

M series	T series
Reverse tapping cycle G74	Face tapping cycle (G84)
Fine boring cycle G76	Side tapping cycle (G88)
Tapping cycle G84	
Boring cycle G86	
Back boring cycle G87	
Boring cycle G88	

For spindle control, the following normal miscellaneous functions are used:
See the description of the miscellaneous functions.

M03: CW spindle rotation
M04: CCW spindle rotation
M05: Spindle stop
M19: Spindle orientation (M series)

When the rotation direction of the spindle is to be switched from one direction to the other (for example, when M04 is output during M03 operation), a parameter can specify whether to send M05 at the time switching.
Timing charts are described in the following page:

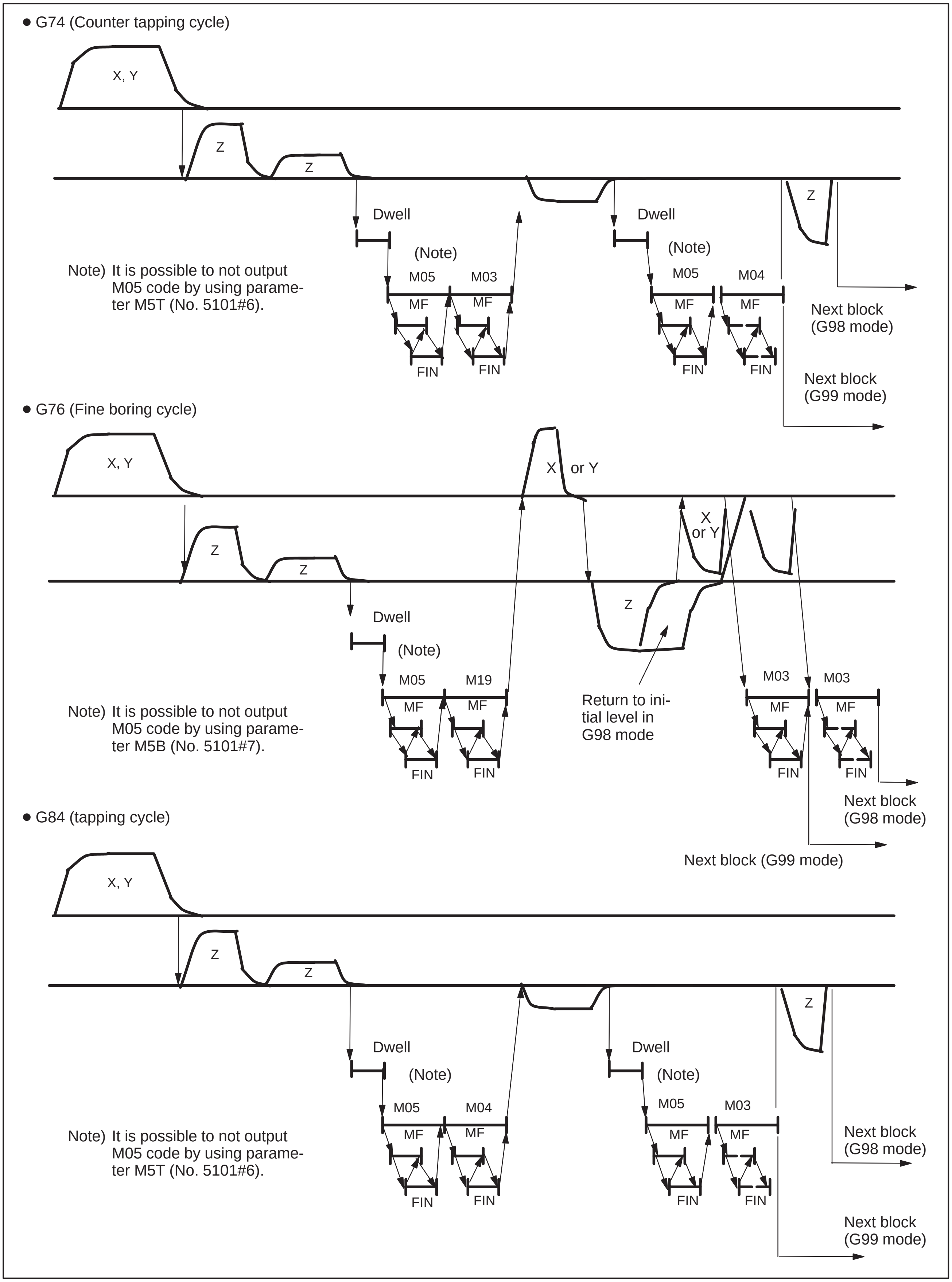


Fig. 11.7 (b) Canned cycle for M series (1/2)

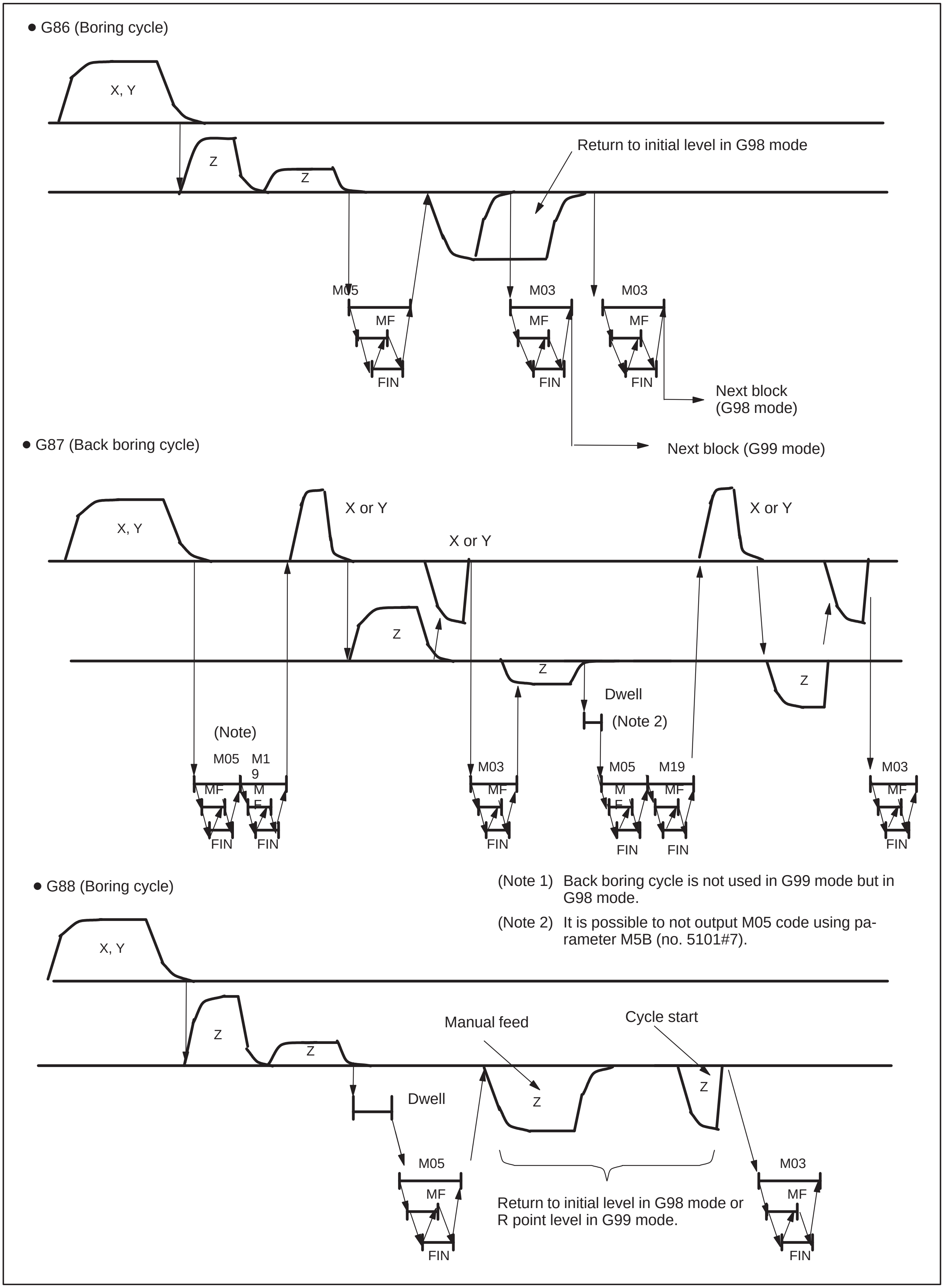
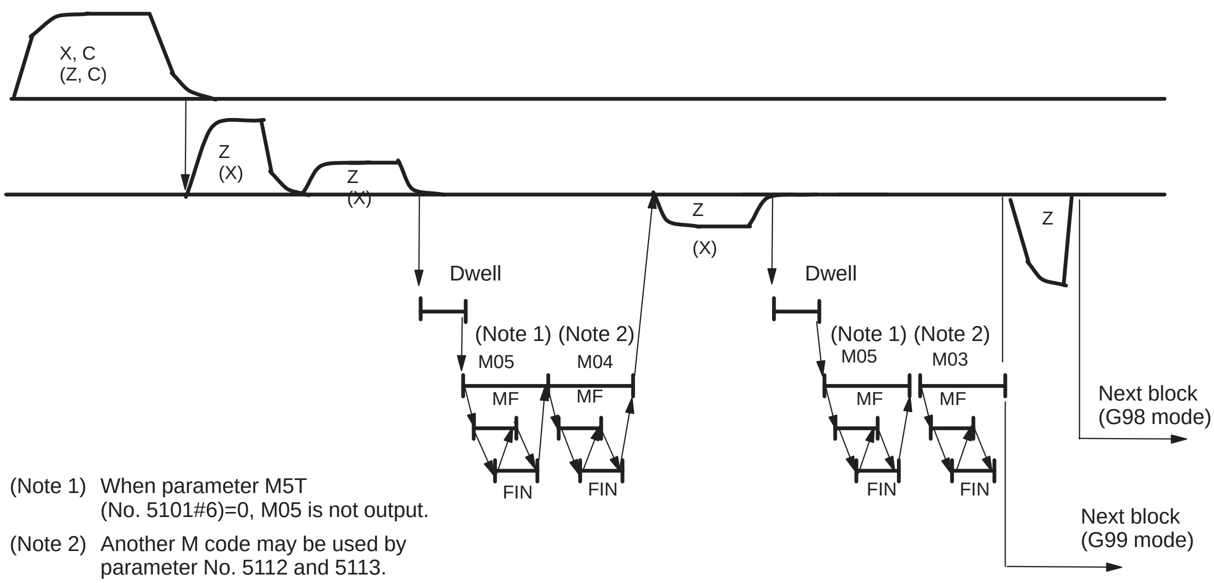


Fig. 11.7 (c) Canned cycle for M series (2/2)

- G84 (Face tapping cycle)
  G88 (Side tapping cycle)



(Note 1) When parameter M5T (No. 5101#6)=0, M05 is not output.

(Note 2) Another M code may be used by parameter No. 5112 and 5113.

Fig 11.7 (d) Canned cycle for T series

- **M code used for C–axis clamp/unclamp (T series)**

When an M code specified in parameter No.5110 for C–axis clamp/unclamp is coded in a program, the CNC issues the M code for C–axis clamp after the tool is positioned and before the tool is fed in rapid traverse to the point–R level. The CNC also issues the M code (M code C–axis clamp +1) for C–axis unclamp after the tool retracts to the point–R level. The tool dwells for the time specified in parameter No. 5111.

Tapping signal

During a tapping cycle, the tapping signal is output. The tapping signal is also output while the G code of the tapping cycle is valid.

Override

During tapping, cutting feedrate override is always set to 100%.

Feed hold

When the feed hold key is pressed during tapping, the movement is not stopped immediately but the movement is stopped when the tool is returned to level R.

Dry run

The TDR bit (bit 5 of parameter No. 1401) specifies whether dry run is valid during tapping.

Signal

Tapping signal TAP <F001, #5>

[Classification] Output signal

[Function] Reports that the system is in tapping mode.

[Output condition] The signal is set to 1 when:
– The system is in tapping cycle mode.
G74, G84: M series
G84, G88: T series
– The system is in tapping mode.
G63: M series
The signal is set to 0 when:
– The system is in neither tapping cycle mode nor tapping mode.
– A reset or emergency stop is specified.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F001			TAP					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5101		M5T			ILV	RTR		FXY
	M5B	M5T	RD2	RD1			EXC	FXY

[Data type] Bit

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 system, this parameter is valid only for the drilling canned cycle in the Series 15 format.

EXC G81
0 : Specifies a drilling canned cycle
1 : Specifies an external operation command

RTR G83 and G87
0 : Specify a high–speed peck drilling cycle
1 : Specify a peck drilling cycle

ILV Initial point position in drilling canned cycle
0 : Not updated by reset
1 : Updated by reset

RD2, RD1 Set the axis and direction in which the tool in drilling canned cycle G76 or G87 is got free. RD2 and RD1 are set as shown below by plane selection.

RD2	RD1	G17	G18	G19
0	0	+X	+Z	+Y
0	1	–X	–Z	–Y
1	0	+Y	+X	+Z
1	1	–Y	–X	–Z

M5T When a spindle rotates from the forward to the reverse direction and vice versa in tapping cycles G84 and G74 for M series (G84 and G88 for T series), befor M04 or M03 is output:
For T series
0 : Not output M05
1 : Outputs M05

For M series
0 : Outputs M05
1 : Not output M05

M5B In drilling canned cycles G76 and G87:
0 : Outputs M05 before an oriented spindle stops
1 : Not output M05 before an oriented spindle stops

	#7	#6	#5	#4	#3	#2	#1	#0
5102	RDI	RAB						

[Data type] Bit

RAB The R command for the drilling canned cycle in the Series 15 format is:
0 : Regarded as an incremental command
1 : Regarded as:
An absolute command in the case of G code system A
An absolute command in the case of G code system B or C when the G90 mode is specified.
An incremental command in the case of G code system B or C when the G91 mode is specified.

RDI The R command for the drilling canned cycle in the Series 15 format:
0 : Is regarded as the specification of a radius
1 : Follows the specification of a diameter/radius for the drilling axis

	#7	#6	#5	#4	#3	#2	#1	#0
5103								
								SIJ

[Data type] Bit

SIJ A tool shift value for the drilling canned cycle G76 or G87 is specified by:
0 : Address Q
1 : Address I, J, or K

[Data type] Byte

5110	C–axis clamp M code in drilling canned cycle

[Valid data range] 0 to 99

This parameter sets the C–axis clamp M code in a drilling canned cycle.

5111	Dwell time when C–axis unclamping is specified in drilling canned cycle

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 32767

This parameter sets the dwell time when C–axis unclamping is specified in a drilling canned cycle.

5112	Spindle forward–rotation M code in drilling canned cycle

[Data type] Byte

[Valid data range] 0 to 255

This parameter sets the spindle forward–rotation M code in a drilling canned cycle.

Note 1 M03 is output when “0” is set.

5113	Spindle reverse–rotation M code in drilling canned cycle

[Data type] Byte

[Valid data range] 0 to 255

This parameter sets the spindle reverse–rotation M code in a drilling canned cycle.

Note 1 M04 is output when “0” is set.

5114	Return or clearance value of drilling canned cycle G83
	Return value of high–speed peck drilling cycle G73

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.001	mm
Inch input	0.001	0.0001	0.0001	inch

[Valid data range] 0 to 32767

For 16–MB, this parameter sets the return value in high–speed peck drilling cycle G73 (G83 for 16–TB).

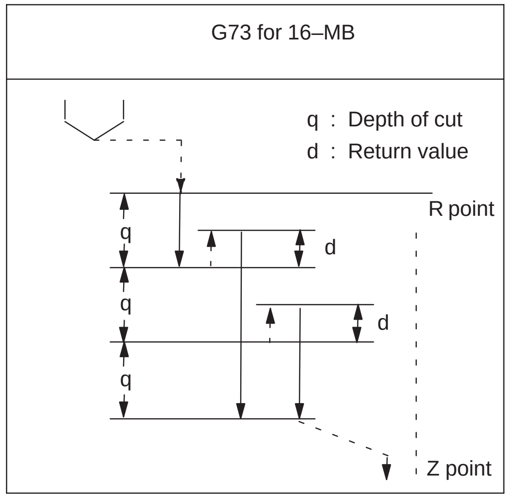


Fig. 11.7 (e) High–speed peck drilling cycle (G73) for 16–MB

For 16–TB, this parameter sets the return or clearance value in drilling canned cycle G83.

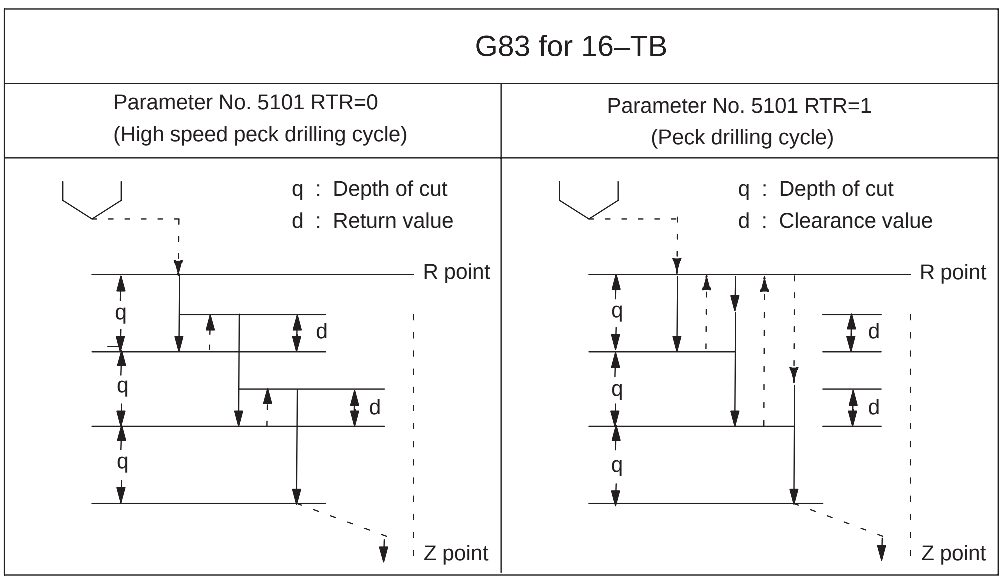


Fig. 11.7 (f) Drilling canned cycle (G83) for 16–TB

5115	Clearance canned cycle G83

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.001	mm
Inch input	0.001	0.0001	0.0001	inch

[Valid data range] 0 A32767

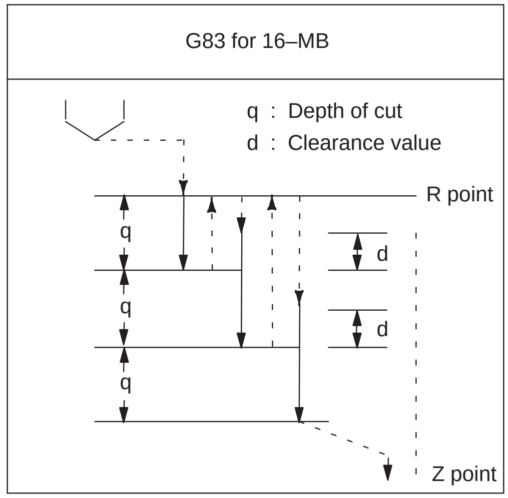


Fig. 11.7 (g) Peck drilling cycle (G83) for 16–MB

Alarm and Message

Number	Message	Description
044	G27–G30 NOT ALLOWED IN FIXED CYCLE (M series)	One of G27 to G30 is commanded in a canned cycle mode. Modify the program.

Note

Note 1 A parameter FXY (No. 5101#0) can be set to the Z axis always used as the drilling axis. When FXY=0, the Z axis is always the drilling axis.

Reference Item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.14.1	Canned cycle
OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.14.1	Canned cycle for hole machining

11.8 EXTERNAL MOTION FUNCTION (M SERIES)

General

Upon completion of positioning in each block in the program, an external operation function signal can be output to allow the machine to perform specific operation.

G81 IP_ ; (The IP_ is axis move command)

Every time positioning for the IP_ move command is completed, the CNC sends an external operation signal to the machine. An external operation signal is output for each positioning operation until canceled by G80 or a group 01 G code.

No external operation signals are output during execution of a block that contains neither X nor Y.

Basic procedure

1 Once positioning for a move command has been completed, the CNC sets the external operation signal EF to 1.
2 When the EF signal is set to 1, the PMC executes drilling or another operation. Once the operation has been completed, the PMC sets completion signal FIN to 1.
3 The CNC resets the EF signal to 0 upon the elapse of the time (TFIN) specified in parameter No. 3011 after the FIN signal is set to 1.
4 When the EF signal is set to 0, the PMC resets the FIN signal to 0.
5 The CNC starts executing the next block.

The timing diagram is shown below:

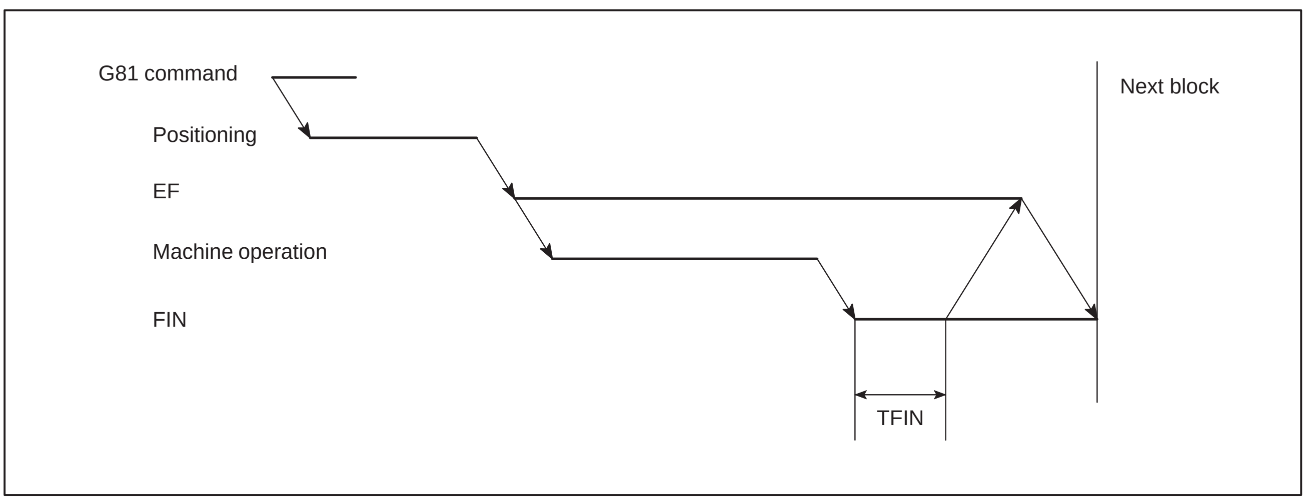


Fig. 11.8 (a) Timing diagram of basic procedure

Signal

External Operation Signal EF<F008#0>

[Classification] Output signal

[Function] Reports that the positioning of G81 has been completed in the external motion function, and that a special external operation is required.

[Output condition] For details of the output condition and procedure, see the ”basic procedure”, described previously.
For details of completion signal FIN, see section 8.1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F008								EF

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5101							EXC	

[Data type] Bit

EXC G81:
0 : Specifies a drilling canned cycle
1 : Specifies an external operation command

Note

Note 1 When this function is used, canned cycles (G73, G74, G76, and G82 to G89) cannot be used.

Note 2 When the high–speed M, S, T, or B interface is used, the signals used by this function are transferred in high–speed mode. See Section 8.4.

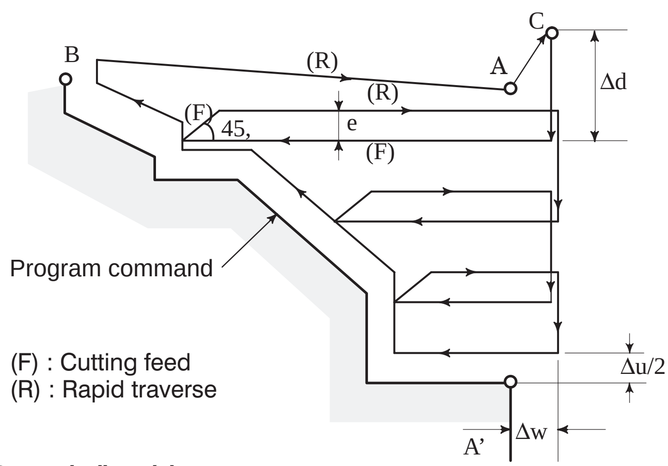
Reference Item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.14.8	External operation function

11.9 CANNED CYCLE (T SERIES)/MULTIPLE REPETITIVE CANNED CYCLE (T SERIES)

General

This option canned cycles to make CNC programming easy. For instance, the data of the finish work shape describes the tool path for rough machining. And also, a canned cycles for the thread cutting is available.
The following example shows stock removals in turning type I.
If a finished shape of A to A’ to B is given by a program as in the figure below, the specified area is removed by Δd (depth of cut), with finishing allowance Δu/2 and Δw left.



G71 U (Δd) R (e) ;
G71 P (ns) Q (nf) U (Δu) W (Δw) F (f) S (s) T (t)

N (ns)···········
················
F____
S____
T____
N (nf)···········;

The move command of a finished shape of A to A’ to B is specified in the blocks from sequence number ns to nf.

Δd : Depth of cut (radius designation)
Designate without sign. The cutting direction depends on the direction AA’. This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5132), and the parameter is changed by the program command.

e : Escaping amount
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5133), and the parameter is changed by the program command.

ns : Sequence number of the first block for the program of finishing shape.

nf : Sequence number of the last block for the program of finishing shape.

Δu : Distance and direction of finishing allowance in X direction (diameter / radius designation).

Δw : Distance and direction of finishing allowance in Z direction.

f,s,t : Any F , S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G71 block is effective.

Signal

Chamfering signal CDZ<G053#7>

[Classification] Input signal

[Function] Executes chamfering in a threading cycle. Specify the chamfering distance in parameter No. 5130.

[Operation] When the signal is set to 1, chamfering is not executed in the threading cycle.
When the signal is set to 0, chamfering is executed in the threading cycle.

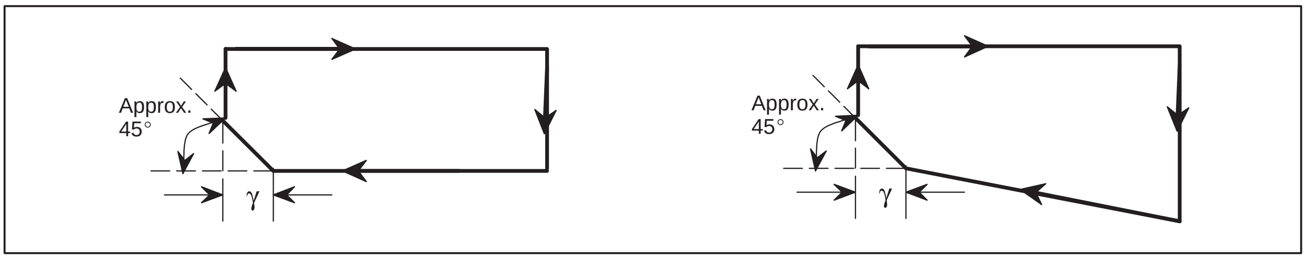


Fig. 11.9 (a) Straight thread cutting cycle **Fig. 11.9 (b)Taper thread cutting cycle**

Set the chamfering distance γ to the parameter No. 5130. When the optional multiple repetitive canned cycle is provided, the chamfering distance can be specified in G76. The chamfering angle is made smaller than 45° by the remaining pulses in the automatic acceleration/ deceleration circuit and servo system.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G053	CDZ							

Parameter

- **Various setting for multiple repetitive canned cycle**

	#7	#6	#5	#4	#3	#2	#1	#0
5102						QSR	MRC	

[Data type] Bit

MRC When a target figure other than a monotonically increasing or monotonically decreasing figure is specified in a multiple repetitive turning canned cycle (G71, G72):
0 : No alarm occurs.
1 : P/S alarm No. 064 is occurs.

Note 1 This parameter is valid for multiple repetitive turning canned cycle type I.

QSR Before a multiple repetitive canned cycle (G70 to G73) is started, a check to see if the program contains a block that has the sequence number specified in address Q is:
0 : Not made.
1 : Made. (If the sequence number specified in address Q cannot be found, an alarm occurs and the canned cycle is not executed.)

- **Chamfering distance in thread cutting cycles G76 and G92**

5130	Chamfering distance in thread cutting cycles G76 and G92

[Data type] Byte

[Unit of data] 0.1

[Valid data range] 0 to 127

This parameter sets the chamfering distance in thread cutting cycles G76 and G92.

- **Depth of cut in multiple repetitive canned cycles G71 and G72**

5132	Depth of cut in multiple repetitive canned cycles G71 and G72

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.001	mm
Inch input	0.001	0.0001	0.0001	inch

[Valid data range] 0A99999999

This parameter sets the depth of cut in multiple repetitive canned cycles G71 and G72.

- **Escape in multiple repetitive canned cycles G71 and G72.**

5133	Escape in multiple repetitive canned cycles G71 and G72.

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.001	mm
Inch input	0.001	0.0001	0.0001	inch

[Valid data range] 0A99999999

This parameter sets the escape in multiple repetitive canned cycles G71 and G72.

Escape in multiple repetitive canned cycles G73

5135	Escape in multiple repetitive canned cycle G73 in X–axis direction
5136	Escape in multiple repetitive canned cycle G73 in Z–axis direction

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.001	mm
Inch input	0.001	0.0001	0.0001	inch

[Valid data range] 0 to 99999999

This parameter sets the escape in multiple repetitive canned cycle G73 of an X, then Z axis.

Division count in multiple repetitive canned cycle G73

5137	Division count in multiple repetitive canned cycle G73

[Data type] Two–word

[Unit of data]

[Valid data range] 0 to 99999999

This parameter sets the division count in multiple repetitive canned cycle G73.

Return in multiple canned cycles G74 and G75

5139	Return in multiple canned cycles G74 and G75

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.001	mm
Inch input	0.001	0.0001	0.0001	inch

[Valid data range] 0 to 99999999

This parameter sets the return in multiple repetitive canned cycles G74 and G75.

- **Minimum depth of cut in multiple repetitive canned cycle G76**

5140	Minimum depth of cut in multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.001	mm
Inch input	0.001	0.0001	0.0001	inch

[Valid data range] 0 to 99999999

This parameter sets the minimum depth of cut in multiple repetitive canned cycle G76.

- **Finishing allowance in multiple repetitive canned cycle G76**

5141	Finishing allowance in multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.001	mm
Inch input	0.001	0.0001	0.0001	inch

[Valid data range] 0 to 99999999

This parameter sets the finishing allowance in multiple repetitive canned cycle G76.

- **Repetition count of final finishing in multiple repetitive canned cycle G76**

5142	Repetition count of final finishing in multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data] Cycle

[Valid data range] 1 to 99999999

This parameter sets the repetition count in multiple repetitive canned cycle G76.

- **Tool nose angle in multiple repetitive canned cycle G76.**

5143	Tool nose angle in multiple repetitive canned cycle G76.

[Data type] Two–word

[Unit of data] Degree

[Valid data range] 0 to 120 when FS15 tape format is used
0, 29, 30, 55, 60 and 80 when FS15 tape format is not used.

This parameter sets the tool nose angle in multiple repetitive canned cycle G76.

Alarm and Message

Number	Message	Description
061	ADDRESS P/Q NOT FOUND IN G70–G73	Address P or Q is not specified in G70, G71, G72, or G73 command. Modify the program.
062	ILLEGAL COMMAND IN G71–G76	1 The depth of cut in G71 or G72 is zero or negative value. 2 The repetitive count in G73 is zero or negative value. 3 The negative value is specified to Δi or Δk in G74 or G75. 4 A value other than zero is specified to address U or W, though Δi or Δk is zero in G74 or G75. 5 A negative value is specified to Δd, though the relief direction in G74 or G75 is determined. 6 Zero or a negative value is specified to the height of thread or depth of cut of first time in G76. 7 The specified minimum depth of cut in G76 is greater than the height of thread. 8 An unusable angle of tool tip is specified in G76. Modify the program.
063	SEQUENCE NUMBER NOT FOUND	The sequence number specified by address P in G70, G71, G72, or G73 command cannot be searched. Modify the program.
064	SHAPE PROGRAM NOT MONOTONOUSLY	A target shape which is not monotone increase or decrease was specified in a repetitive canned cycle (G71 or G72).
065	ILLEGAL COMMAND IN G71–G73	1 G00 or G01 is not commanded at the block with the sequence number which is specified by address P in G71, G72, or G73 command. 2. Address Z(W) or X(U) was commanded in the block with a sequence number which is specified by address P in G71 or G72, respectively. Modify the program.
066	IMPROPER G–CODE IN G71–G73	An unallowable G code was commanded beween two blocks specified by address P in G71, G72, or G73. Modify the program.

Number	Message	Description
067	CAN NOT ERROR IN MDI MODE	G70, G71, G72, or G73 command with address P and Q was specified. Modify the program.
069	FORMAT ERROR IN G70–G73	The final move command in the blocks specified by P and Q of G70, G71, G72, or G73 ended with chamfering or corner R.

Reference Item

OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.14.1 II.14.2	Canned cycle Multiple repetitive canned cycle

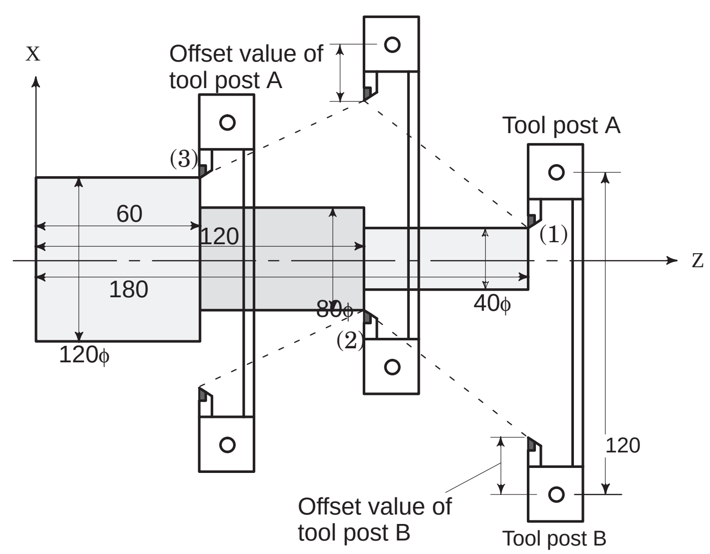
11.10 MIRROR IMAGE FOR DOUBLE TURRETS (T SERIES)

General

Mirror image can be applied to X–axis with G code.

G68 : Double turret mirror image on
G69 : Mirror image cancel

When G68 is designated, the coordinate system is shifted to the mating turret side, and the X–axis sign is reversed from the programmed command to perform symmetrical cutting. To use this function, set the distance between the two turrets to a parameter (No. 1290).
Program example for double turrets.



X40.0 Z180.0 T0101 ;	Position turret A at (1)
G68 ;	Shift the coordinate system by the distance A to B (120mm), and turn mirror image on.
X80.0 Z120.0 T0202 ;	Position turret B at (2)
G69 ;	Shift the coordinate system by the distance B to A, and cancel mirror image.
X120.0 Z60.0 T0101 ;	Position turret A at (3)

Parameter

- **Distance between two turrets**

1290	Distance between two turrets in mirror image

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 999999999

Set the distance between two turrets in mirror image.

Reference Item

OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.14.6	Mirror image for double turrets

11.11 INDEX TABLE INDEXING FUNCTION (M series)

General

By specifying indexing positions (angles) for the indexing axis (one rotation axis, A, B, or C), the index table of the machining center can be indexed.
Before and after indexing, the index table is automatically unclamped or clamped .

Basic Procedure

The control axis that indexes the index table can be named A, B or C. It will be referred to as “B” in the following discussion.

The positioning angle for the index table is commanded by the numerics following “B” in the program command, which is an independent block. Both absolute and incremental commands are possible, but the value after “B” is the integer times the numeric set by the parameter:

(Example)	G00G90B100000;	Absolute command (Positioning angle 10 degrees)
	G00G91B20.0;	Incremental command (Move distance 20 degrees)

There are two variations of the procedure (type A and type B) to set the index table position; the difference is in the ON/OFF timing of the position control servo. The sequence of events and the difference between the variations are described below, followed by time charts showing them graphically.

(1) Assume Bbbbb is ordered by the command program.

(2) The CNC turns the B axis unclamp signal BUCLP <F061#0> to “ 1 ”. (Type B -- When BUCLP is turned to “ 1 ”, the position control servo for the B axis is turned ON.)

(3) On the PMC side, the clamp of the B axis is released; when completed, the B axis unclamp completion signal *BEUCL <G038#6> turns to “0”.

(4) The CNC then turns the B axis unclamp signal BUCLP to “0”, to indicate it received the *BEUCL signal.

(5) When the PMC is notified that BUCLP has been turned to “0”, the PMC should turn *BEUCL to “1”.
In type B, B–axis unclamp signal BUCLP is turned to “0”, B–axis position control is made in servo–on state, B–axis is rotated, and the B axis is stopped at the specified position. B axis always moves at rapid traverse.

(6) When the B axis stops at the specified position, CNC turns B–axis clamp signal BCLP<F061#1>to 1. In type A, signal BCLP is set to “1” and B–axis position control is made in servo–off state.

(7) When BCLP is turned to “1” on the PMC side, the B axis is clamped mechanically (with a clutch or shot pin, for example). When the clamp is completed, the B axis clamp completion signal *BECLP <G038#7> is turned to “0”.

(8) When *BECLP is turned to “0”, the CNC then turns BCLP to “0”, informing it received the *BECLP signal. (Type B -- When BCLP turns to “0”, the B axis position control servo is turned off.)

(9) On the PMC side, when BCLP changes to “0”, *BECLP is turned to “1”. This completes the sequence.

The time charts for these operations are shown in the figures below.

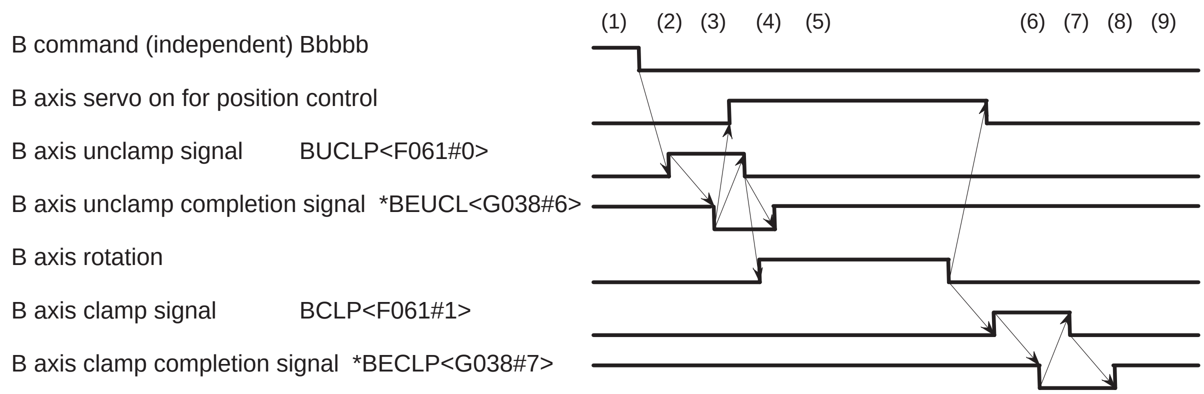


Fig. 11.11 (a) Time chart for positioning index table (type A)

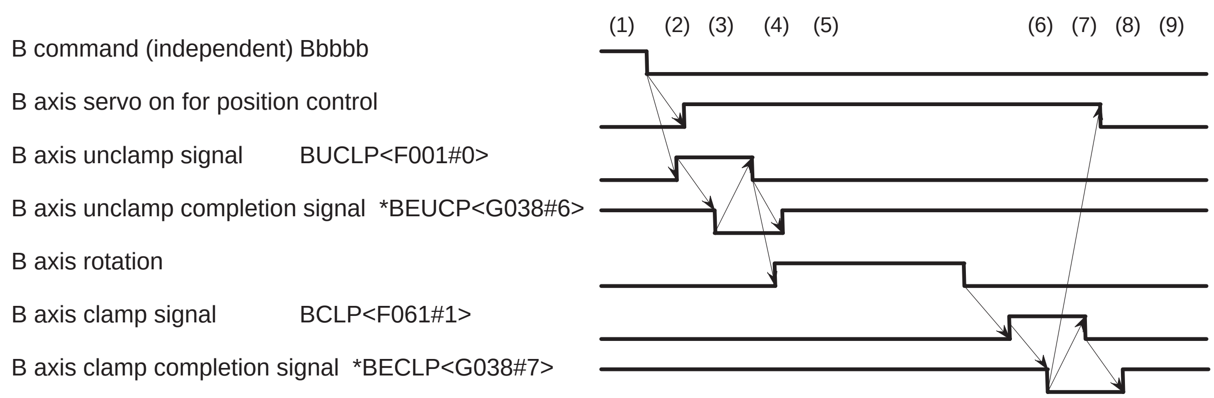


Fig. 11.11 (b) Time chart for positioning index table (type B)

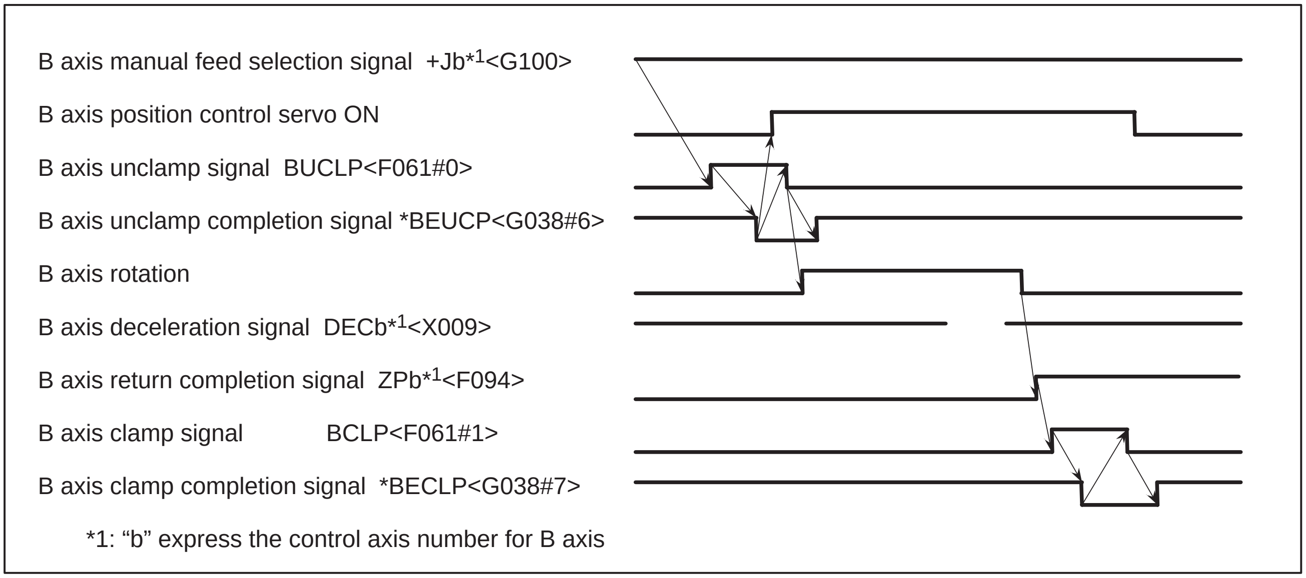


Fig. 11.11 (c) Manual reference position return of B axis time chart (type A)

Type A and Type B

As described in the basic procedure, type A differs from type B in that the servo used for B–axis position control is turned on or off at the different timing.

Type A is suitable for a system in which the B–axis is clamped with shot pins.

Type B is suitable for a system in which the B–axis is clamped with a clutch.

Minimum indexing angle

When the B–axis is clamped with shot pins, the mechanism can be indexed at only a limited number of positions. The minimum indexing angle can be specified in parameter No. 5512. If an angle which is not a multiple of this minimum indexing angle is specified in indexing, alarm No. 135 is issued.

Direction of rotation

The direction of rotation can be set to one of the following.

– Whichever direction has the shorter distance (INC, bit 3 of parameter No. 5500)
– Direction specified with a command
– Usually the positive direction. Only when a particular M code is specified in the same block, the axis rotates in the negative direction (parameter No. 5511).

Absolute/incremental programming

Setting G90, bit 4 of parameter No. 5500, specifies absolute programming, irrespective of G90/G91 mode.

Signal

B axis clamp signal BCLP<F061#1>

[Classification] Output signal

[Function] Instructs the PMC side to clamp the B axis mechanically with a clutch or shot pin.

[Output condition] The output condition and procedure are the same as those described in the basic procedure for positioning the index table.

B axis clamp completion signal BECLP<G038#7>

[Classification] Input signal

[Function] Notifies the CNC of completion of the B axis clamp operation.

[Operation] The operation and procedure are the same as those described in the basic procedure for positioning the index table.

B axis unclamp signal BUCLP<F061#0>

[Classification] Output signal

[Function] Instructs the PMC side to release the B axis from the mechanical clamp.

[Output condition] The output condition and procedure are the same as those described in the basic procedure for positioning the index table.

B axis unclamp completion signal *BEUCP<G038#6>

[Classification] Input signal

[Function] Notifies the CNC of completion of the release of the B axis from the mechanical clamp.

[Operation] The operation and procedure are the same as those described in the basic procedure for positioning the index table.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G038	*BECLP	*BEUCP						

	#7	#6	#5	#4	#3	#2	#1	#0
F061							BCLP	BUCLP

Parameter

- **Setting linear or rotation axis**

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

Note 1 When this parameter is changed, turn off the power before continuing operation.

[Data type] Bit axis

ROTx, ROSx Setting linear or rotation axis

ROSx	ROTx	Description
0	0	Linear axis ● Inch/metric conversion is done. ● All coordinate values are linear axis type. (Not rounded in 0 to 360°) ● Stored pitch error compensation is linear axis type (Refer topara-meter No. 3624)
0	1	Rotation axis (A Type) ● Inch/metric conversion is not done. ● Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values and relative coordinate values are rounded or not rounded by parameter No. 1008 #0 and #2. ● Stored pitch error compensation is the rotation type. Refer to parameter No. 3624. ● Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	0	Setting is invalid (unused)
1	1	Rotation axis (B type) ● Inch/metric conversion is not done. ● Machine coordinate values is linear axis type and is not rounded in 0 to 360°. Absolute coordinate values and relative coordinate values are rounded or not rounded by parameter No. 1008 #0 and #2. ● Stored pitch error compensation is linear axis type (Refer to pa-rameter No. 3624) ● Cannot be used with the rotation axis roll over function and the index table indexing function (M series).

- **Various setting for index table indexing**

	#7	#6	#5	#4	#3	#2	#1	#0
5500	IDX			G90	INC	ABS	REL	DDP

[Data type] Bit

DDP Selection of decimal–point input method of index table indexing axis
0 : Conventional method (Example IS–B: B1; = 0.001 deg)
1 : Pocket calculator method (Example IS–B: B1; = 1.000 deg)

REL Relative position display of index table indexing axis
0 : Not rounded by 360 degrees
1 : Rounded by 360 degrees

ABS Displaying absolute coordinate value of index table indexing axis
0 : Not rounded by 360 degrees
The index table indexing axis rotates 720 degrees (two rotations) when G90 B720.0; is specified from the 0–degree position. It rotates in reverse direction 720 degrees (two rotations) when G90 B0.; is specified. The absolute coordinate value then becomes 0 degree.
1 : Rounded by 360 degrees
The index table indexing axis is positioned in 40 degrees when G90 B400.0; is specified from the 0–degree position. The index table indexing axis does not rotate by two or more turns when this parameter is set to 1. It also does not move when G90 B720.0; is specified from the 0–degree position.

INC Rotation in the G90 mode when negative–direction rotation command M code (parameter No. 5511) is not set
0 : Not set to the shorter way around the circumference
1 : Set to the shorter way around the circumference (Set ABS, #2 of parameter No. 5500, to 1.)

G90 Index table indexing command
0 : Judged to be an absolute/increment command according to the G90/G91 mode
1 : Judged to be an absolute command

IDX Index table indexing sequence
0 : Type A
1 : Type B

- **Negative direction rotation command M code**

5511	Negative0direction rotation command M code

[Data type] Byte

[Valid data range] 0 to 255

0 : Not use an M code that sets the index table rotation to the negative direction. The rotation direction is specified using a command and parameter (INC, #3 of parameter No. 5500).
1 to 255:
Sets an M code that sets the index table rotation to the negative direction. The rotation is set to the negative direction only when an M code set here is specified in the same block as an index table indexing

command. If the M code is not specified in the same block, the rotation is always set to the positive direction.

Note 1 Set ABS, #2 of parameter No. 5500, to 1.

- **Unit of index table indexing angle**

5512	Unit of index table indexing angle

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm

[Valid data range] 0 to 360000

This parameter sets the unit of index table indexing angle. A P/S alarm is generated when movement other than integer multiple of the setting value is specified.

Note 1 If zero is specified as the setting value, any command can be specified irrespective of the unit of angle.

Alarm and Message

Number	Message	Description
135	SPINDLE ORIENTATION PLEASE	Without any spindle orientation , an attept was made for spindle indexing. Perform spindle orientation.
136	ILLEGAL AXIS COM-MAND	In index table indexing , another con-trol axis was instructed together with the B axis. Modify the program.

Note

Note 1 Specify a rotation axis as the index table indexing axis. (Set 1 in the ROTx bit (bit 0 of parameter No. 1006).)

Note 2 The secondary auxiliary function can be used if the address is different from that of the indexing axis.

Note 3 The servo off signal for the index table indexing axis is invalid.

Note 4 If the incremental command is used for indexing of the index table, the workpiece zero point offset value on the index table axis must always be 0. That is, the machine coordinate system must always agree with the workpiece coordinate system of the index table axis.

Note 5 Single direction positioning (G60) cannot be specified.

Note 6 While the index table is being positioned, input signals that reset the CNC, such as *ESP (emergency stop), ERS (external reset), and RRW (reset & rewind), are functional. When reset is applied to the CNC, this operation stops. Further, if *SP (automatic operation stop signal) turns to “0”, axis movement is stopped and the equipment enters the automatic operation stop state.

If a stop at an any position is not suitable for the machine, appropriate processing is required on the machine.

Note 7 If a reset occurs while the system is awaiting the completion of clamping or unclamping, the clamp or unclamp signal is cleared. The CNC exits from the completion wait status.

Note 8 Manual operation of jog feed, incremental feed and handle feed cannot be used with the B axis, but manual reference position return is possible. If reset is applied during the movement of B axis, the manual reference position return operation is performed.

Note 9 No movement can be performed by automatic return from the reference position (G29), return to the second reference position (G30), or selection of the machine coordinate system (G53).

Note 10 The dry run signal DRN is ineffective during positioning of the B axis.

Note 11 The machine lock signal MLK is functional during positioning of the B axis. However, while the B axis is moving, after the movement ends, the MLK is functional.

Reference Item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.15.11	Index table indexing function

11.12 SCALING (M SERIES)

General

A programmed figure can be magnified or reduced (scaling).
The dimensions specified with X_, Y_, and Z_ can each be scaled up or down with the same or different rates of magnification.
The magnification rate can be specified in the program.
Unless specified in the program, the magnification rate specified in the parameter is applied.

- **Scaling up or down along all axes at the same rate of magnification**

Least input increment of scaling magnification is: 0.001 or 0.00001.
It depends on parameter SCR (No. 5400#07) which value is selected. If scaling P is not specified on the block of scaling (G51X_Y_Z_P_ ;), the scaling magnification set to parameter (No. 5411) is applicable. If X,Y,Z are omitted, the tool position where the G51 command was specified serves as the scaling center.

SCALING UP OR DOWN ALONG ALL AXES AT THE SAME RATE OF MAGNIFICATION

Format		Meaning of command	
G51X_Y_Z_P_ ;	Scaling start	**X_Y_Z_** :	Absolute command for center coordinate value of scaling
:	Scaling is effective. (Scaling mode)	**P_** :	Scaling magnification
G50 ;	Scaling cancel		

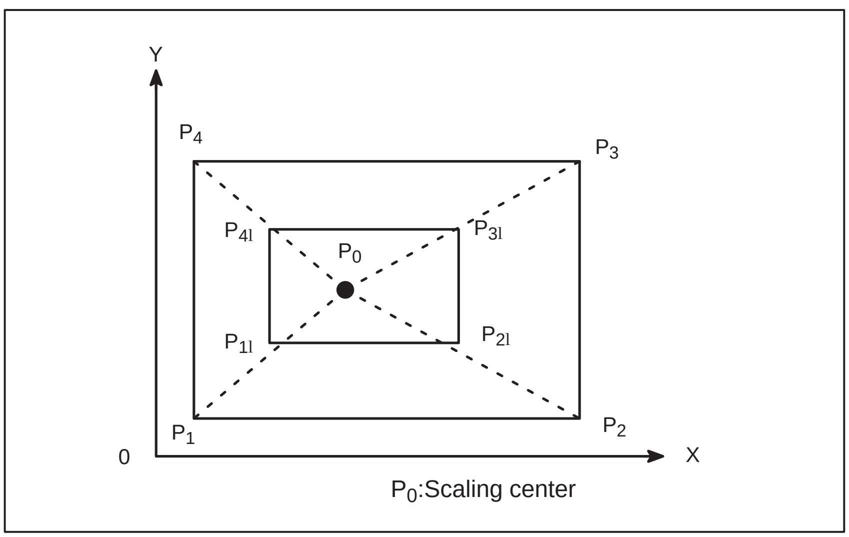


Fig.11.12(a) Scaling (P_1 P_2 P_3 P_4→P_1'P_2'P_3'P_4')

- **Scaling of each axis, programmable mirror image (negative magnification)**

Each axis can be scaled by different magnifications. Also when a negative magnification is specified, a mirror image is applied. First of all, set a parameter XSC (No. 5400#6) which validates each axis scaling (mirror image).
Then, set parameter SCLx (No. 5401#0) to enable scaling along each axis.
Least input increment of scaling magnification of each axis (I, J, K) is 0.001 or 0.00001(set parameter SCR (No. 5400#7)).
Magnification is set to parameter 5421 within the range ±0.00001 to ±9.99999 or ±0.001 to ±9.999.
If a negative value is set, mirror image is effected.
If magnification I, J or K is not commanded, a magnification value set to parameter (No. 5421) is effective. However, a value other than 0 must be set to the parameter.

SCALING UP OR DOWN ALONG EACH AXES AT A DIFFERENT RATE OF MAGNIFICATION (MIRROR IMAGE)

Format		Meaning of command
G51_X_Y_Z_I_J_K_;	Scaling start	**X_Y_Z_:** Absolute command for center coordinate value of scaling **I_J_K_:** Scaling magnification for X axis, Y axis and Z axis respectively
⋮	Scaling is effective. (Scaling mode)	
G50	Scaling cancel	

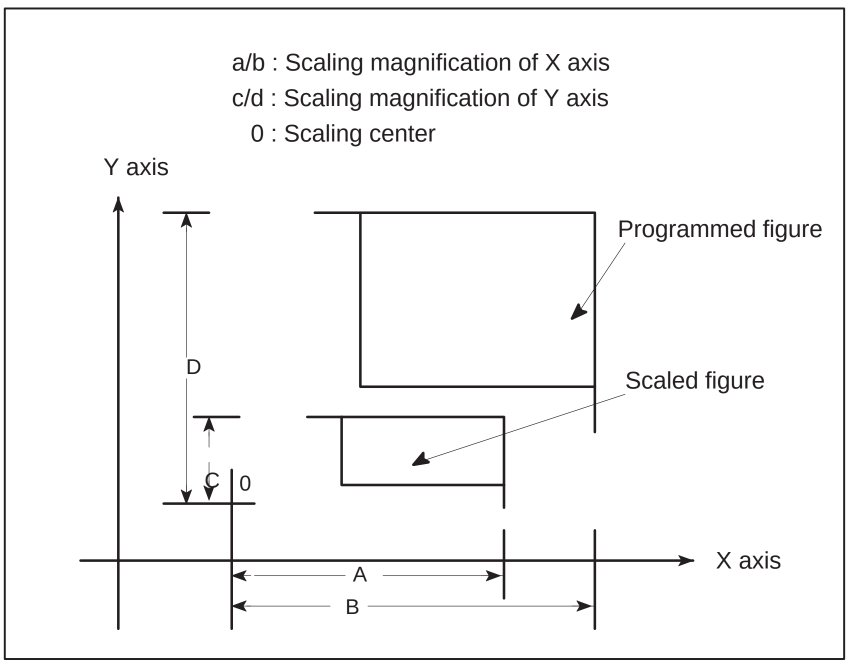


Fig11.12(b) Scaling of each axis

Parameter

- **Setting valid/invalid and magnification of scaling**

	#7	#6	#5	#4	#3	#2	#1	#0
5400	SCR	XSC						

[Data type] Bit

XSC Axis scaling and programmable mirror image
0 : Invalidated (The scaling magnification is specified by P.)
1 : Validated

SCR Scaling magnification unit
0 : 0.00001 times (1/100,000)
1 : 0.001 times

- **Valid/invalid setting to each axis scaling**

	#7	#6	#5	#4	#3	#2	#1	#0
5401								SCLx

[Data type] Bit axis

Scaling for every axis
0 : Invalidated
1 : Validated

- **Magnification used when scaling magnification is not specified**

5411	Magnification used when scaling magnification is not specified

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] 0.001 or 0.00001 times (Selected using SCR, #7 of parameter No. 5400)

[Valid data range] 1 to 999999
This parameter sets the scaling magnification. This setting value is used when a scaling magnification (P) is not specified in the program.

Note 1 Parameter No. 5421 becomes valid when scaling for every axis is valid. (XSC, #6 of parameter No. 5400 is “1”.)

- **Scaling magnification for every axis**

5421	Scaling magnification for every axis

[Data type] Two–word

[Unit of data] 0.001 or 0.00001 times (Selected using SCR, #7 of parameter No. 5400)

[Valid data range] $-999999 \sim -1$, $1 \sim 999999$

This parameter sets the scaling magnification for every axis.

Alarm and Message

Number	Message	Description
141	CAN NOT COMMAND G51 IN CRC	G51 (Scaling ON) is commanded in the tool offset mode. Modify the program.
142	ILLEGAL SCALE RATE	Scaling magnification is commanded in other than 1 – 999999. Correct the scaling magnification setting (G51 P_p, or parameter 5411 or 5421).
143	SCALED MOTION DATA OVERFLOW	The scaling results, move distance, coordinate value and circular radius exceed the maximum command value. Correct the program or scaling magnification.

Reference Item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.15.9	Scaling (G50, G51)

11.13 COORDINATE SYSTEM ROTATION

General

A programmed shape can be rotated. By using this function it becomes possible, for example, to modify a program using a rotation command when a workpiece has been placed with some angle rotated from the programmed position on the machine. Further, when there is a pattern comprising some identical shapes in the positions rotated from a shape, the time required for programming and the length of the program can be reduced by preparing a subprogram of the shape and calling it after rotation.

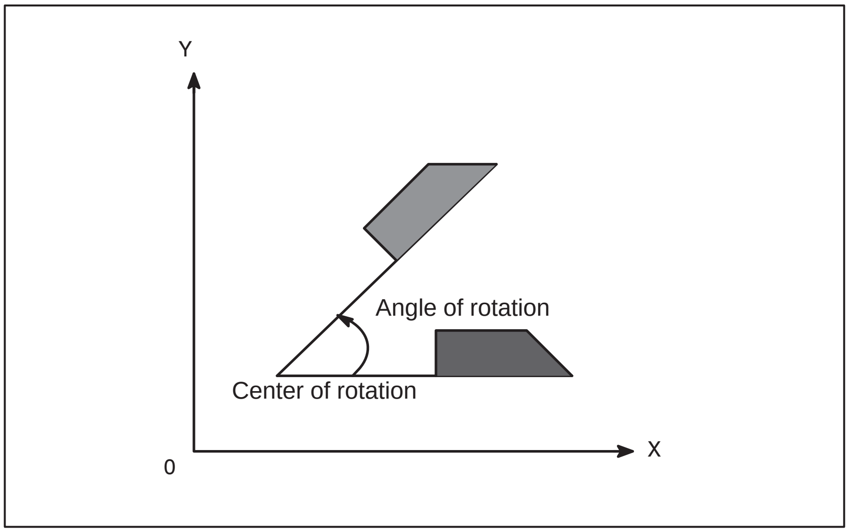


Fig. 11.13 (a) Coordinate system rotation

FORMAT

$\left\{\begin{array}{l}\textbf{G17}\\ \textbf{G18}\\ \textbf{G19}\end{array}\right\}$ **G68 (G68.1)**α_β_**R_** ; Start rotation of a coordinate system.

Coordinate system rotation mode
(The coordinate system is rotated.)

G69 (G69.1); Coordinate system rotation cancel command

Note:G68/G69 for M series, G68.1/G69.1 for T series.

MEANING OF COMMAND

G17 (G18 or G19) : Select the plane in which contains the figure to be rotated.

α_β_ Absolute command for two of the x_,y_,and Z_ axes that correspond to the current plane selected by a command (G17, G18, or G19). The command specifies the coordinates of the center of rotation for the values specified subsequent to G68.

R_ Angular displacement with a positive value indicates counter clockwise rotation. Parameter 5400#0 selects whether the specified angular displacement is always considered an absolute value or is considered an absolute or incremental value depending on the specified G code (G90 or G91).

Least input increment : 0.001 deg
Valid data range : –360,000 to 360,000

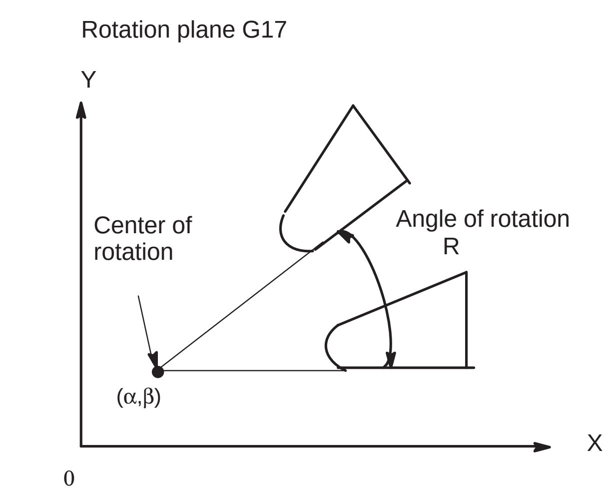


Fig. 11.13 (b) Coordinate system rotation

Parameter

● Angle specification method of coordinate system rotation

	#7	#6	#5	#4	#3	#2	#1	#0
5400								RIN

[Data type] Bit

RIN Coordinate rotation angle command (R)
0 : Specified by an absolute method
1 : Specified by G90 or G91

● Angular displacement used when no angular displacement is specified for coordinate system rotation

5410	Angular displacement used when no angular displacement is specified for coordinate system rotation

[Data type] Two–word

[Unit of data] 0.001 degrees

[Valid data range] –360000 to 360000

This parameter sets the angular displacement for coordinate system rotation. When the angular displacement for coordinate system rotation is not specified with address R in the block where G68 is specified, the setting of this parameter is used as the angular displacement for coordinate system rotation.

Alarm and Message

Number	Message	Description
144	ILLEGAL PLANE SE-LECTED	The coordinate rotation plane and arc or cutter compensation C plane must be the same. Modify the program.

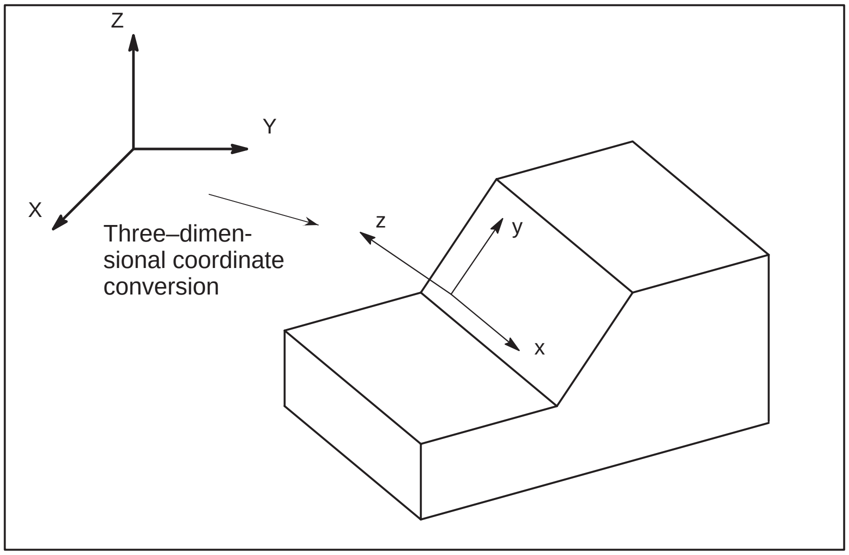
Reference Item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.15.10	Coordinate system rotation (G68, G69)
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.15.7	Coordinate system rotation (G68.1, G69.1)

11.14 THREE–DIMENSIONAL COORDINATE CONVERSION (M series)

General

The coordinate system can be rotated about an axis by specifying the center of rotation, direction of the axis of rotation, and angular displacement. This coordinate conversion function is quite useful for three–dimensional machining using a diesinking machine. By applying three–dimensional coordinate conversion to a program generated for machining on XY plane, identical machining can be executed on a desired plane.



Parameter

- **Setting relative position and absolute position**

	#7	#6	#5	#4	#3	#2	#1	#0
3104	DAC	DAL	DRC	DRL				

[Data type] Bit

DRL Relative position
0 : The actual position displayed takes into account tool length offset.
1 : The programmed position displayed does not take into account tool length offset.

DRC Relative position
0 : The actual position displayed takes into account cutter compensation.
1 : The programmed position displayed does not take into account cutter compensation.

DAL Absolute position
0 : The actual position displayed takes into account tool length offset.
1 : The programmed position displayed does not take into account tool length offset.

DAC Absolute position
0 : The actual position displayed takes into account cutter compensation.
1 : The programmed position displayed does not take into account cutter compensation.

- **Setting absolute coordinates in the three–dimensional coordinate conversion mode**

	#7	#6	#5	#4	#3	#2	#1	#0
3106		DAK						

[Data type] Bit

DAK When absolute coordinates are displayed in the three–dimensional coordinate conversion mode:
0 : Coordinates in the program coordinate system are displayed.
1 : Coordinates in the workpiece coordinate system are displayed.

Alarm and Message

Number	Message	Description
047	ILLEGAL AXIS SELECT	For startup of three–dimensional tool compensation or three–dimensional coordinate conversion, two or more axes were specified in the same direction (basic and parallel axes.)
048	BASIC 3 AXIS NOT FOUND	For startup of three–dimensional tool compensation or three–dimensional coordinate conversion, the three basic axes used when X_p, Y_p, and Z_p are omitted were not specified in parameter No. 1022.
5043	TOO MANY G68 NESTING	G68, three–dimensional coordinate conversion, was specified three times or more.
5044	G68 FORMAT ERROR	A format error occurred in a block including G68. The alarm is issued when any of the following takes place. (1) In a block including G68, any of I, J, and K is not specified (the option for coordinate rotation is not provided). (2) In a block including G68, all of I, J, and K is 0. (3) In a block including G68, R is not specified.

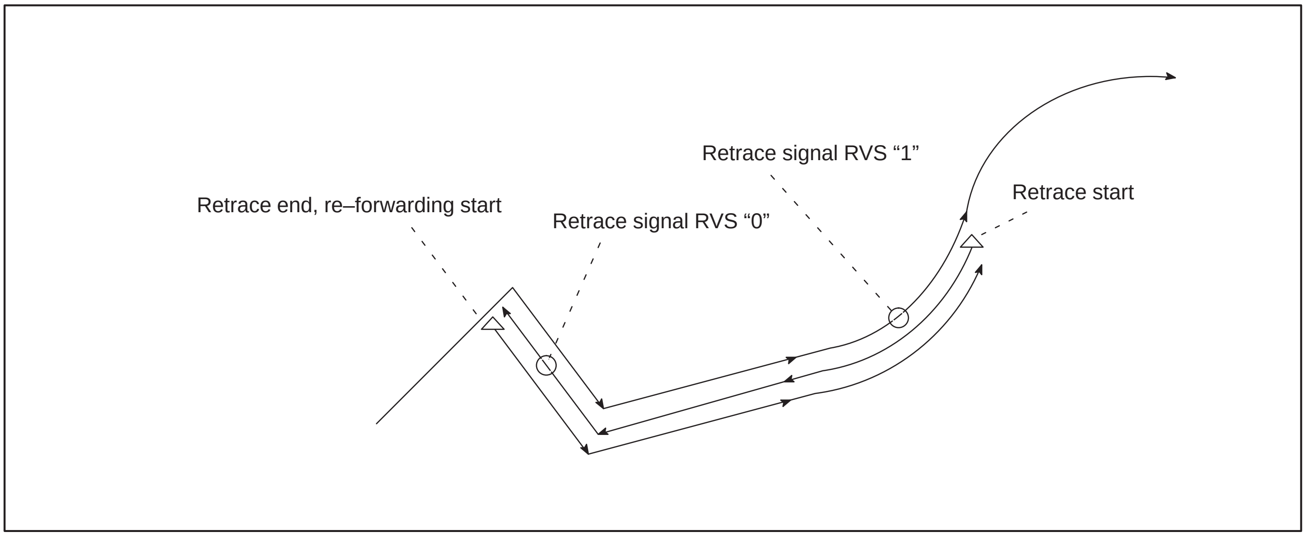
Reference Item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.14.10	Three–dimensional coordinate conversion

11.15 RETRACE (M SERIES)

General

A tool can retrace the tool path along which the tool has moved. This operation is referred to as retrace. In addition, a tool can move forward again along the path that the tool has retraced. This operation is referred to as re–forward. The tool can then resume machining according to the program when it returns to the position where it started retrace.



By using the retrace signal RVS, which is an input signal from the PMC to the control unit, retrace and re–forward can be performed. Retrace is performed when the retrace signal RVS turns to “1”. Re–forward operation is performed when the retrace signal turns to “0”. In retrace, a tool can retrace blocks that have been executed in the automatic operation mode (memory command, tape command, manual data input), but a limit is imposed on the number of blocks that can be retraced. For detailed information, refer to the operator’s manual.
Whether to use the same federate as specified for normal operation or use a feedrate dedicated to retrace can be determined by setting parameter No. 1414.

Signal

Retrace signal (RVS) <G007#0>

[Classification] Input signal

[Function] Directs the control unit to retrace the tool along the path which the tool was moved in automatic operation (memory command, tape command, manual data input).

[Operation] When RVS turns to “1” while the tool is being moved forward (hereafter referred to as forward), the tool retraces the tool path along which it was moved. Note, however, that the tool does not start retrace immediately after this signal turns to “1” ; the tool starts retrace after the block performing the current forward operation is executed. When RVS turns to “0” while retrace is in progress, the tool switches from retrace operation to re–forward. In this case also, the tool does not start re–forward to resume machining immediately after this signal turns to “0” retrace; the tool starts re–forward after retracing all commands of the block currently subject to retrace.

The movement of a tool can be immediately switched from forward to retrace or from retrace to re–forward. For this purpose, first turn the feed hold signal *SP to “0” to stop automatic operation. Then, change the state of RVS after the automatic operation start in–progress signal STL turns to “0” and the automatic operation stop state is set. And the feed hold lamp SPL turns to “1”. Next, turn feed hold signal *SP the automatic operation start signal ST from “1” to “0” to start automatic operation. Then, the tool can switch its movement to retrace or re–forward during execution of a block.

Retrace–in–progress signal (RVSL) <F082#2>

[Classification] Output signal

[Function] Notifies the PMC that retrace is in progress.

[Output condition] This signal turns to "1"when:

· Tool is in retrace with the retrace signal RVS turned to "1".

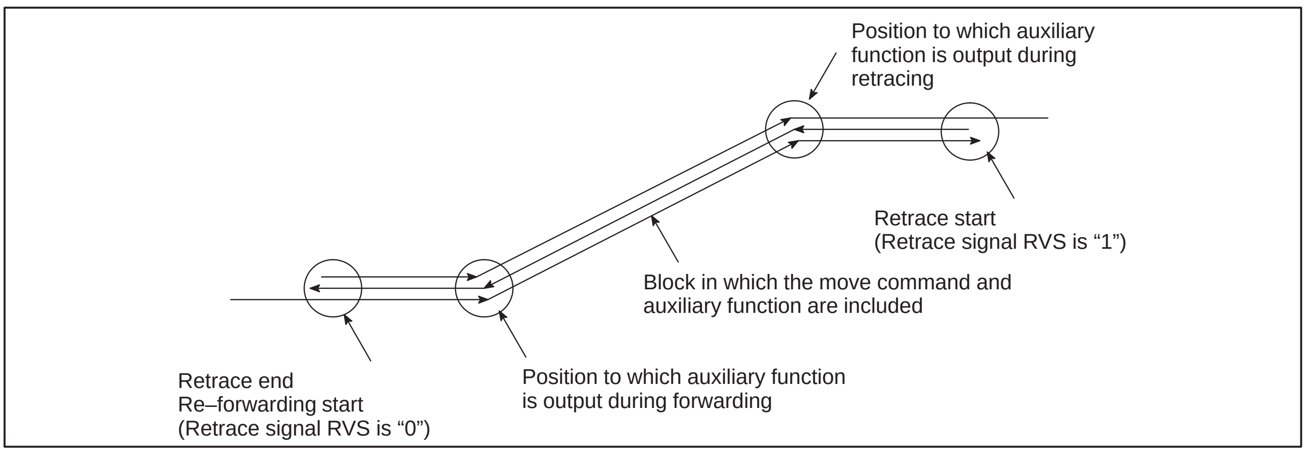
This signal turns to "0"

· The tool is in forward or re–forward with the retrace signal RVS turned to "0".

· The tool is at stop because of no block to retrace during retracing.

When the tool is in retrace, the M functions, S functions, T functions, and second auxiliary functions are executed in the same way as when the tool is moving forward. During retrace, this signal can be used on the PMC, if required, to prevent these functions from being executed in the same way as when the tool is moving forward.

Particularly when an M function, S function, T function, or second auxiliary function is specified in a block containing a move command, the positions where the code signals and strobe signal are output differ, depending on whether the tool is in forward (or re–forward) or retrace. So, take action on the PMC by using this signal and distribution end signal DEN as required.



Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G007								RVS

	#7	#6	#5	#4	#3	#2	#1	#0
F082						RVSL		

Parameter

1414	Feedrate for retrace

[Data type] Two–word

This parameter sets the feedrate for retrace when the retrace function is used.

(1) For rapid traverse

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 240000	6 to 100000
Inch machine	0.1 inch/min	6 to 96000	6 to 48000
Rotation axis	1 deg/min	6 to 240000	6 to 100000

Note 1 When 0 is set in this parameter, the rapid traverse rate that is set in parameter No. 1420 is used for retrace.

(2) For cutting feed

When a value other than 0 is specified in this parameter, the same feedrate as an F command specified using the value without a decimal point is set and is used for retrace. When 0 is specified in this parameter, the programmed feedrate (F command) is used for retrace.

Alarm and Message

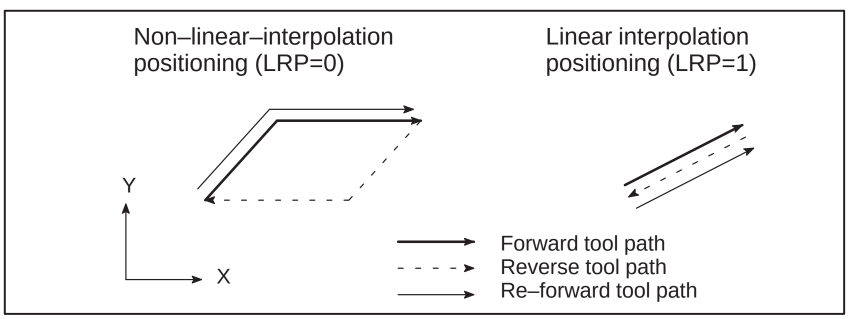
While a tool is in retrace, the retrace–in–progress signal RVSL is sent, and the character string RVRS blinks on the CRT screen to signal that the tool is currently in retrace. When a tool is in re–forward, the character string RTRY blinks to signal that the tool is currently in re–forward. The indication of RTRY continues until the tool returns to the block where retrace was started, that is, until forward movement is resumed. When there are no more retraceable blocks, the character string RVED blinks to signal that no further retrace can take place.

Note

Note 1 Positioning (G00)

If non–linear–interpolation positioning is executed (the LRP bit (bit 1 of parameter No. 1401) is set to 0), the retrace tool path will not agree with the forward tool path. The re–forward tool path agrees with the forward tool path.

If the linear interpolation positioning is executed (the LRP bit (bit 1 of parameter No. 1401) is set to 1), the retrace tool path agrees with the forward tool path.



Reference Item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	III.4.9	Retrace function

11.16 MACRO COMPILER/EXECUTER

General

There are two types of NC programs; those which, once created, are scarcely changed, and those which are changed for each machining type. The former are programs created by the custom macro, and the latter are machining programs. If programs of these types are executed simultaneously, a battery may run out or the custom macro may be destroyed by error operation.
Such problems can be solved by this function. The custom macro created by a machine tool builder is converted to an execute-form program, be stored in the FLASH ROM module, and be executed.

Features

(1) Since the program is stored after converted to an execute-form program, the execution speed is high. The machining time is then reduced, and the precision is improved.
(2) Since the program is stored in FLASH ROM, there is no problem of battery extinction or custom macro destruction by error operation. The reliability is improved.
(3) Since the stored program is not displayed on a program screen, the know-how of the machine tool builder is protected.
(4) Since the custom macro is stored in FLASH ROM, the program edit memory can be used efficiently.
(5) The user can call the macro easily without knowing the stored program. A custom macro can be created and executed in the program edit memory as usual.
(6) An original screen can be created by using the graphic display or selecting screens by the soft key. The machine tool builder can extend the control function by using such functions as machining program creation and edit control, reader/punch interface control, and PMC data read/write functions.

Note

Note 1 When the macro executor is attached, the order-made macro cannot be specified.

Reference Item

Macro compiler/executer programming manual (B–61803E–1)

11.17 SMALL HOLE PECK DRILLING CYCLE (M series)

General

This is a repetition of a peck drilling cycle in which when an overload torque detection signal (skip signal is used) is received with an arbor having an overload torque detection function during drilling, the tool is retracted, and the spindle speed and cutting feedrate are changed to retry machining.

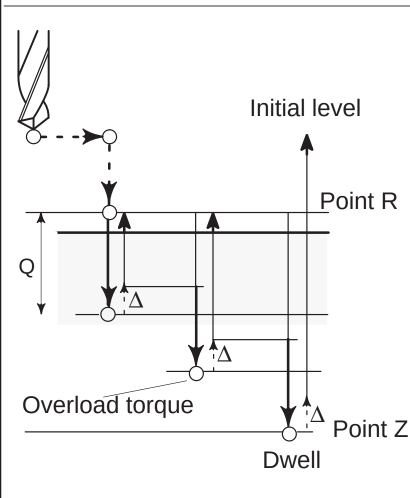
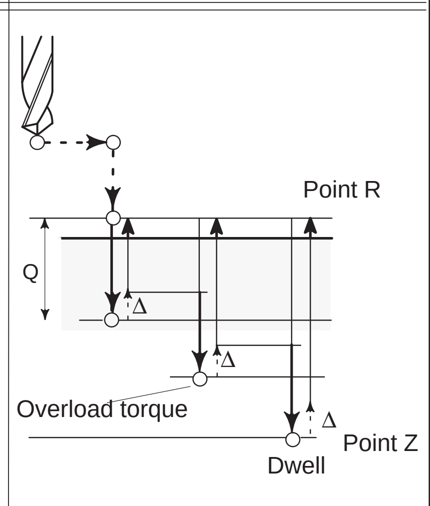
The cycle is realized by:

* X– and Y–axis positioning
* Positioning at point R along the Z–axis
* Cutting along the Z–axis (first time, depth of cut Q, incremental)

Repeated until point Z is reached:

Retracting
(bottom of hole → minimum clearance Δ , incremental)
Retracting
(→ to point R, absolute)
Forwarding
(point R → to point with hole bottom + clearance Δ, absolute)
Cutting
(second and subsequent times, cut of depth Q + Δ, incremental)

* Dwell
* Return to point R along the Z–axis (or initial point) = end of cycle

G83(**G98**)	**G83**(**G99**)
	

Change of cutting conditions

The cutting conditions are changed at each pecking operation (forwarding → cutting → retracting) during one G83 cycle. (Appropriate setting of bits 1 and 2 of parameter No. 5160 can specify that the cutting conditions are not to be changed.)

- **Changing of cutting feedrate**

The cutting feedrate programmed with the F word is changed during each of the second and subsequent cutting operations. Parameter Nos. 5166 and 5167 specify the ratio of change for a case in which a skip signal was received during the previous cutting operation and a case in which no skip signal was received during the previous cutting operation, respectively.

$$\boxed{\text{Cutting feed rate} = F \times \alpha}$$

(First time) $\alpha = 1.0$

(Second and subsequent times) $\alpha = \alpha \times \beta \div 100$
(where β is the ratio of change for the first time)

Skip during the previous cutting: β = b1% (parameter No. 5166)
No skip during the previous cutting: β = b2% (parameter No. 5167)

When the ratio α at which the cutting feedrate is changed becomes less than the ratio specified in parameter No. 5168, changing the cutting feedrate is discontinued. The upper limit to the newly specified cutting feedrate is the maximum cutting feedrate.

- **Changing of spindle speed**

The cutting feedrate programmed with the S word is changed during each of the second and subsequent cutting operations. Parameter Nos. 5164 and 5165 specify the ratio of change for a case in which a skip signal was received during the previous cutting operation and a case in which no skip signal was received during the previous cutting operation, respectively.

$$\boxed{\text{Spindle speed} = S \times \gamma}$$

(First time) $\gamma = 1.0$

(Second and subsequent times) $\gamma = \gamma \times \delta \div 100$
(where δ is the ratio of change for the first time)

Skip during the previous cutting: δ = d1% (parameter No. 5164)
No skip during the previous cutting: δ = d2% (parameter No. 5165)

If the cutting feedrate is clamped at the lower limit, the spindle speed is not changed. The upper limit to the newly specified spindle speed is the maximum S analog data.

Signal

Overload torque signal SKIP <X004#7>

[Classification] Input signal

[Function] Retracts a tool if an overload torque is applied it.

[Operation] When this signal becomes “1”, the control unit operates as follows:

- Assuming that an overload torque is applied to a tool, retracts it, and changes the spindle speed and cutting feedrate, then retries machining.
- This signal is valid, when the drill axis is between points R and Z and is moving forward or in cutting operation (so that the tool can retract).

Note 1 This signal is used also as a skip signal. (See Section 14.3.)

Small–diameter peck drilling in progress signal PECK2 <F066#5>

[Classification] Output signal

[Function] Indicates whether small–diameter peck drilling is in progress.

[Output condition] This signal becomes “1” under the following conditions.

- When the tool returns from drill axis point R positioning to R point/initial level, after it was positioned at the hole position along a nondrill axis.
- The signal does not become “1” during positioning at the hole position.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X004	SKIP							

	#7	#6	#5	#4	#3	#2	#1	#0
F066			PECK2					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5160						NOL	OLS	

[Data type] Bit

CHNRM When an overload torque signal is received in a peck drilling cycle of a small diameter, the feed and spindle speed are
0 : Not changed.
1 : Changed.

CHSKP When the depth of cut per action is satisfied although no overload torque signal is received in a peck drilling cycle of a small diameter, the feed and spindle speed are:

0 : Not changed.
1 : Changed.

5163	M code that specifies the peck drilling cycle mode of a small diameter

[Data type] Two–word

[Unit of data]

[Valid data range] 1 to 99999999

This parameter sets an M code that specifies the peck drilling cycle mode of a small diameter.

5164	Percentage of the spindle speed to be changed when the tool is retracted after an overload torque signal is received

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 255

This parameter sets the percentage of the spindle speed to be changed when the tool is retracted because the overload torque signal is received in a peck drilling cycle of a small diameter.

$S2 = S1 \times d1 \div 100$
S1: Spindle speed to be chaged
S2: Spindle speed changed
d1 is set as a percentage.

5165	Percentage of the spindle speed to be changed when the tool is retracted without an overload torque signal received

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 255

This parameter sets the percentage of the spindle speed to be changed when the tool is retracted without the overload torque signal received in a peck drilling cycle of a small diameter.

$S2 = S1 \times d2 \div 100$
S1: Spindle speed to be chaged
S2: Spindle speed changed
d2 is set as a percentage.

5166	Percentage of cutting feedrate to be changed when the tool is retracted after an overload torque signal is received

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 255

This parameter sets the percentage of the cutting feedrate to be changed when the tool is retracted because the overload torque signal is received in a peck drilling cycle of a small diameter.

$F2 = F1 \times b1 \div 100$
F1: Cutting feedrate to be changed
F2: Changed cutting feedrate
b1 is set as a percentage.

5167	Percentage of the cutting feedrate to be changed when the tool is retracted without an overload torque signal received

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 255

This parameter sets the percentage of the cutting feedrate tot be changed when the tool is retracted without the overload torque signal received in a peck drilling cycle of a small diameter.

$F2 = F1 \times b2 \div 100$
F1: Cutting feedrate to be changed
F2: Changed cutting feedrate
b2 is set as a percentage.

5168	Lower limit of the percentage of the cutting feedrate in a peck drilling cycle of a small diameter

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 255

This parameter sets the lower limit of the percentage of the cutting feedrate changed repeatedly in a peck drilling cycle of a small diameter to the specified cutting feedrate.

$FL = F \times b3 \div 100$
F: Specified cutting feedrate
FL: Changed cutting feedrate
Set b3 as a percentage.

5170	Number of the macro variable to which the total number of retractions during cutting is output

[Data type] Word

[Valid data range] 100 to 149

This parameter sets the number of the macro variable to which the total number of times the tool is retracted during cutting in a peck drilling cycle mode of a small diameter is output.

Note 1 The total number cannot be output to common variables 500 to 599.

5171	Number of the macro variable to which the total umber of retractions because of an overload signal is output

[Data type] Word

[Valid data range] 100 to 149

This parameter sets the common variable number of the custom macro to which the number of times the tool is retracted after the overload signal is received during cutting in a peck drilling cycle mode of a small diameter is output.

Note 1 The total number cannot be output to common variables 500 to 599.

5172	Speed of retraction to point R when no address I is issued

[Data type] Word

[Unit of data] mm/min

[Valid data range] 0 to 400

This parameter sets the speed of retraction to point R when no address I is issued in a peck drilling cycle of a small diameter.

5173	Speed of advancing to the position just before the bottom of a hole when no address I is issued

[Data type] Word

[Unit of data] mm/min

[Valid data range] 0 to 400

This parameter sets the speed of advancing to the position just before the bottom of a previously machined hole when no address I is issued in a peck drilling cycle of a small diameter.

5174	Clearance in a peck drilling cycle of a small diameter

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Linear axis (millimeter input)	0.01	0.001	0.0001	mm
Linear axis (inch input)	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 32767

This parameter sets the clearance in a peck drilling cycle of a small diameter.

Alarm and message

• Diagnostic display

520	Total number of retract operations during cutting since G83 was issued
521	Total number of retract operations due to reception of the overload torque signal since G83 was issued

The indications of DGN Nos. 520 and 521 are cleared by G80.

522	Coordinates at which the drill axis started retracting (least input command)
523	Difference between the previous and current coordinates at which the drill axis started retracting (least input increment: previous – current)

Note

Note 1 Forwarding or retracting is not performed by rapid traverse positioning. Instead, it is performed with the same interpolation as for cutting feed. This means exponential acceleration/deceleration is performed; however, the tool life management function does not count the tool life during forwarding or retracting.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II. 14.1.7	Small hole peck drilling cycle

12 DISPLAY / SET / EDIT

12.1 DISPLAY/SET

12.1.1 Clock Function

General

Time is displayed in the hour/minute/second format on each display screen. Some screens allows display of the year, month, and day.
The custom macro system variable can be used to read the time. The time will be told through the window on the PMC side.

Time information can be read and written.

System variables for time information

Variable number	Function
#3001	This variable functions as a timer that counts in 1–millisecond increments at all times. When the power is turned on, the value of this variable is reset to 0. When 65535 milliseconds is reached, the value of this timer returns to 0.
#3002	This variable functions as a timer that counts in 1–hour increments when the cycle start lamp is on. This timer preserves its value even when the power is turned off. When 1145324.612 hours is reached, the value of this timer returns to 0.
#3011	This variable can be used to read the current date (year/month/day). Year/month/day information is converted to an apparent decimal number. For example, March 28, 1993 is represented as 19930328.
#3012	This variable can be used to read the current time (hours/minutes/seconds). Hours/minutes/seconds information is converted to an apparent decimal number. For example, 34 minutes and 56 seconds after 3 p.m. is represented as 153456.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.11.4.5	Displaying and Setting Run Time,Parts Count, and Time
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.11.4.9	Displaying and Setting Run Time,Parts Count, and Time

12.1.2 Displaying Operation History

General

This function displays a history of the key and signal operations, performed by the NC operator, upon the occurrence of a failure or NC alarm. The history can also be displayed for previously generated NC alarms.

The following history data is recorded:

(1) MDI key operation sequences, performed by the NC operator
Example: A ~Z, <POS>, <PAGE ↑>, [SF1]

(2) On/off status transitions of selected input and output signals
Example: G0000.7↑ , SBK ↑

(3) NC alarm information
Example: P/S0010

(4) Time (date, time) stamp
Example: 92/01/20
09:15:30

The history data can be output to an input/output device, connected via the reader/punch interface. Previously output history data can be input from an input/output device.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3106	OHS			OPH				

[Data type] Bit

OPH The operation history screen is:
0 : Not displayed.
1 : Displayed.

OHS Operation history sampling is:
0 : Performed.
1 : Not performed.

3122	Time interval used to record time data in operation history

[Data type] Word

[Unit of data] Minutes

[Valid data range] 0 to 1439

Time data is recorded in operation history at set intervals. When 0 is specified in this parameter, 10 minutes is assumed as the default. However, note that time data is not recorded if there is no data to be recorded at the specified time.

Note

Note 1 While the operation history screen is displayed, history data is not recorded.

Note 2 When the duration of the on/off state of an input signal is 16 msec or shorter, that state is not recorded in the history. In addition, note that some signals are not recorded in the history.

Note 3 Once the memory area becomes full, the oldest data is deleted to allow new data to be recorded. The memory area can store information corresponding to approximately 8000 key strokes.

Note 4 Recorded history data is held in memory even after the power is turned off. Note, however, that the history data is erased by performing an entire–memory clear operation.

Note 5 For the operation history function, sampling is disabled when bit 7 (OPS) of parameter No. 3106 is set to 1.

Note 6 Be careful to set the date and time correctly.

When the time is recorded at regular intervals, but no data is recorded within a defined period, the time for that period is not recorded.

To input and output operation history data, the reader/punch interface option is required.

12.1.3 Help Function

General

The help function displyas on the screen detailed information about alarms issued in the CNC and about CNC operations. The following information is displayed.

- **Detailed information of alarms**

When the CNC is operated incorrectly or an erroneous machining program is executed, the CNC enters the alram state. The help screen displays detailed information about the alarm that has been issued and how to reset it. The detailed information is displayed only for a limited number of P/S alarms. These alarms are often misunderstood and are rather difficult to understand.

- **Operation method**

If you are not sure about a CNC operation, refer to the help screen for information about each operation.

- **Parameter table**

When setting or referring to a system parameter, if you are not sure of the number of the parameter, the help screen displays a list of parameter Nos. for each function.

Note

Note 1 The user cannot switch the screen display from the PMC screen or CUSTOM screen to the help screen.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.13	HELP FUNCTION
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.13	HELP FUNCTION

12.1.4 Displaying Alarm History

General

Up to 25 of the most recent CNC alarms are stored and displayed on the screen.

The following information items are displayed.

(1) The date the alarm was issued

(2) Alarm No.

(3) Alarm message (some contains no message)

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.7.2	ALARM HISTORY DISPLAY
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.7.2	ALARM HISTORY DISPLAY

12.1.5 Servo Tuning Screen

General

On the servo tuning screen, parameters required for basic adjustment of the servo motor and statuses being monitored are listed for each axis.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3111								SVS

[Data type] Bit

SVS Servo tuning screen
0 : Not displayed
1 : Displayed

Reference item

Maintenance Manual (B–62445E)	5.2	Servo Tuning Screen

12.1.6 Spindle Tuning Screen

General

On the spindle tuning screen, parameters required for basic adjustment of the serial spindle and statuses being monitored are listed. The screen is only for the main spindle connected to the first amplifier.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3111						SVP	SPS	

[Data type] Bit type

SPS Spindle tuning screen
0 : Not displayed
1 : Displayed

SVP Synchronization errors displayed on the spindle tuning screen
0 : Instantaneous values are displayed.
1 : Peak–hold values are displayed.

Reference item

MAINTENANCE MANUAL (B–62445E)	6.4	Spindle tuning screen

12.1.7 Waveform Diagnosis Display

General

Waveform diagnosis is classified into two main types.

(1) One–shot type

One–shot waveform diagnosis provides graphs of waveforms to illustrate changes in the following data.

In one–shot waveform diagnosis, the start of data collection can be triggered by the rising or falling edge of a machine signal.

This function is designed to facilitate the adjustment of servo motors and spindle motors.

a. Servo motor error value along each axis, number of distributed pulses, torque, feedrate, current, and thermal simulation data

b. Combined feedrate for the first, second, and third axes

c. Spindle motor speed, load meter reading, and position deviation, converted to spindle position

d. On/off status of the machine signal, specified by a signal address

(2) Stored type

In stored waveform diagnosis, changes in the following data are recorded. When a servo alarm is issued, the recorded data can be read and displayed graphically as a waveforms.

In stored waveform diagnosis, the end of data collection can be triggered by the rising or falling edge of a machine signal.

This function is designed to facilitate the determination of a faulty location.

Stored data can be output via the reader/punch interface.

a. Servo motor error value along each axis, number of distributed pulses, torque, feedrate, current, and thermal simulation data

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3112								SGD

[Data type] Bit

SGD Servo waveform
0 : Not displayed
1 : Displayed

3120	Time from the output of an alarm to the termination of sampling (waveform diagnosis function)

[Data type] Word

[Unit of data] ms

[Valid data range] 1 to 32760

Note

Note 1 Once the memory area becomes full, the oldest data is deleted to allow new data to be recorded (stored type).

Note 2 Waveform diagnosis data is held in memory even after the power is turned off (stored type).

Note 3 Be careful to set the date and time correctly.

Note 4 To output stored type waveform data, the reader/punch interface option is required.

Note 5 Waveform diagnosis is enabled when bit 0 (SGD) of parameter No. 3112 is set to 1. To perform waveform diagnosis, a graphics board is required.

Reference item

Maintenance Manual (B–62445E)	1.8	Waveform diagnostic display

12.1.8 Self–diagnosis

General

When a breakdown occurs, in order to quickly determine the cause, the following should be done.

First, it has to be determined as to whether the breakdown occurred in the NC internal section, or the PMC or machine side.

There are times when it appears that a breakdown has occurred even when the breakdown has not actually occurred. For example, when the machinery ceases to operate because it is waiting for an external signal.

In this case, the condition of the interface between the CNC and PMC, or between the CNC and the machinery, and the conditions within the CNC need to be investigated.

The NC checks the following itself.

1) Abnormality of detection system
2) Abnormality of position control unit
3) Abnormality of servo system
4) Overheat
5) Abnormality of CPU
6) Abnormality of ROM
7) Abnormality of RAM
8) Abnormality in data transfer between CRT/MDI
9) Abnormality of part program storage memory
10) Abnormality in tape reader read function
11) Abnormality in data transfer between PMC

Input/output signals from PMC to CNC, or vice versa, and inner status of the NC can be displayed on the CRT screen.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.11.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.11.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN

12.1.9 Display of Hardware and Software Configuration

General

The required hardware/software configuration for CNC maintenance can be displayed on the CRT.

The system configuration screen displays the following information:

(1) Printed circuit board configuration

The type and function of the printed circuit board mounted in each slot are displayed. When a CPU is mounted on a printed circuit board, the software series and edition are also displayed.

(2) Software configuration

The series and editions of installed software, including the CNC software, servo software, PMC management software, and ladder programs, are displayed. For the CNC software, assembly information is also displayed.

(3) Module configuration

The configuration of the modules or hardware mounted on each printed circuit board (such as the type of a module or the hardware, and whether it is mounted) is displayed.

Reference item

Maintenance Manual (B–62445E)	1.3	System Configuration Screen

12.1.10 Position Display Neglect

General

Indication of the current position can be suppressed by setting bit 0 (NDPx) of parameter No. 3115, or by issuing the position indication ignore signal.

Bit 1 (NDAx) of parameter No. 3115 enables the display of positions in the machine coordinate system only.

Signal

Position Display Neglect Signal NPOS1ANPOS8<G198>

[Classification] Input signal

[Function] Disables the display of the current position.
A separate signal is provided for each controlled axis. The number at the end of each signal name denotes the controlled axis number.

NPOS1

1···1st axis current position display
2···2nd axis current position display
3···3rd axis current position display
4···4th axis current position display
5···5th axis current position display
6···6th axis current position display
7···7th axis current position display
8···8th axis current position display

[Operation] While a signal is set to 0, the current position in the corresponding axis is displayed. While the signal is set to 1, the current position along the corresponding axis is not displayed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G198	NPOS8	NPOS7	NPOS6	NPOS5	NPOS4	NPOS3	NPOS2	NPOS1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3115							NDAx	NDPx

[Data type] Bit axis

NDPx Display of the current position for each axis
0 : The current position is displayed.
1 : The current position is not displayed.

NDAx Position display using absolute coordinates and relative coordinates is:
0 : Performed.
1 : Not performed. (Machine coordinates are displayed.)

12.1.11 Run Hour and Parts Count Display

General

This function displays the integrated power–on time, the integrated cycle operation time, the integrated cutting time and timer (started by an input signal from PMC) on the CRT display screen. The integrated cycle operation time, the integrated cutting time and timer can be altered and preset, using the MDI.
In addition to the above, this function displays the count of the total number of parts machined, the number of parts required and the number of parts machined on the CRT screen. Each time M02, M30 or a parameter set M code is executed, the count of the total number of parts machined and the number of parts machined in memory is incremented by 1.
If a program is prepared so as to execute M02, M30 or a parameter set M code each time one part machining is completed, the number of parts machined can be counted automatically.
If the count of the number of parts machined reaches the number of parts required, a signal is output to the PMC side.
It is possible to change and preset the number of parts required and the number of parts machined using MDI.

Signal

Target part count reached signal PRTSF<F062#7>

[Classification] Output signal

[Function] Reports to the PMC that the specified number of parts have been machined.

[Output condition] The PRTSF signal is set to 1 when:

- Machining of the specified number of parts has been completed.

  When 0 (infinity) is set as the required number of parts, this signal is not output.

The PRTSF signal is set to 0 when:

- Machining of the specified number of parts has not yet been completed.
- The system is reset.

General–purpose integrating meter start signal TMRON <G053#0>

[Classification] Input signal

[Function] The CNC has an integrating meter which is started by an input signal from the PMC, as well as integrating meters for counting the automatic operation time and counting cutting time. The count for this integrating meter can be displayed on the CRT. The count can be preset by MDI operation.

[Operation] When the signal is set to 1, the integrating meter starts counting.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G053								TMRON
F062	PRTSF							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6700								PCM

[Data type] Bit

PGM M code that counts the total number of machined parts and the number of machined parts

0 : M02, or M30, or an M code specified by parameter No. 6710

1 : Only M code specified by parameter No. 6710

6710	M code that counts the total number of machined parts and the number of machined parts

[Data type] Byte

[Valid data range] 0 to 255 except 98 and 99

The total number of machined parts and the number of machined parts are counted (+1) when the M code set is executed.

Note 1 Set value 0 is invalid (the number of parts is not counted for M00). Data 98 and 99 cannot be set.

6711	Number of machined parts

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One piece

[Valid data range] 0 to 99999999

The number of machined parts is counted (+1) together with the total number of machined parts when the M02, M30, or a M code specified by parameter No. 6710 is executed.

6712	Total number of machined parts

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One piece

[Valid data range] 0 to 99999999

This parameter sets the total number of machined parts.

The total number of machined parts is counted (+1) when M02, M30, or an M code specified by parameter No. 6710 is executed.

6713	Number of required parts

Setting entry is acceptable.

[Data type] Word

[Unit of data] One piece

[Valid data range] 0 to 9999

This parameter sets the number of required machined parts.

Required parts finish signal PRTSF is output to PMC when the number of machined parts reaches the number of required parts. The number of parts is regarded as infinity when the number of required parts is zero. The PRTSF signal is then not output.

6750	Integrated value of power–on period

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One minute

[Valid data range] 0 to 99999999

This parameter displays the integrated value of power–on period.

6751	Operation time (integrated value of time during automatic operation)

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One ms

[Valid data range] 0 to 60000

6752	Operation time (integrated value of time during automatic operation)

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One minute

[Valid data range] 0 to 99999999

This parameter displays the integrated value of time during automatic operation (neither stop nor hold time included).

6753	Integrated value of cutting time

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One ms

[Valid data range] 1 to 60000

6754	Integrated value of cutting time

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One minute

[Valid data range] 0 to 99999999

This parameter displays the integrated value of a cutting time that is performed in cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03).

6755	Integrated value of general–purpose integrating meter drive signal (TMRON) ON time

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One ms

[Valid data range] 0 to 60000

6756	Integrated value of general–purpose integrating meter drive signal (TMRON) ON time

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One minute

[Valid data range] 0 to 99999999

This parameter displays the integrated value of a time while input signal TMRON from PMC is on.

6757	Operation time (integrated value of one automatic operation time)

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One ms

[Valid data range] 0 to 60000

6758	Operation time (integrated value of one automatic operation time)

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One minute

[Valid data range] 0 to 99999999

This parameter displays the one automatic operation drive time (neither stop nor hold state included). The operation time is automatically preset to 0 during the power–on sequence and the cycle start from the reset state.

Reference item

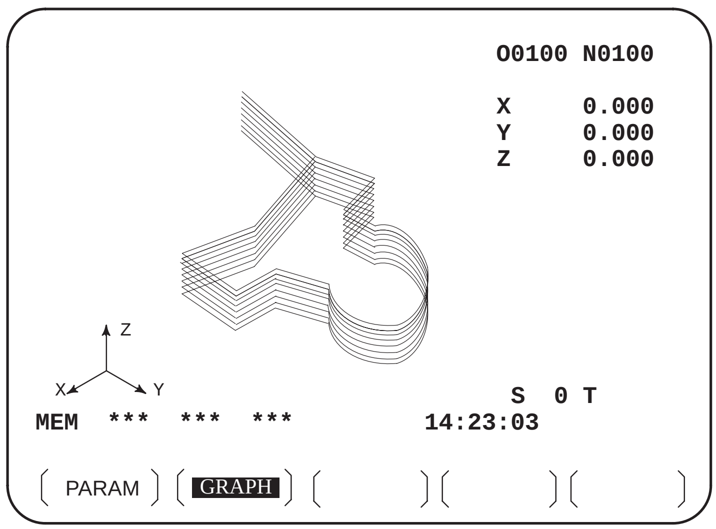
OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.11.4.5	Displaying and Setting Run Time,Parts Count, and Time
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.11.4.9	Displaying and Setting Run Time,Parts Count, and Time

12.1.12 Graphic Display/ Dynamic Graphic Display

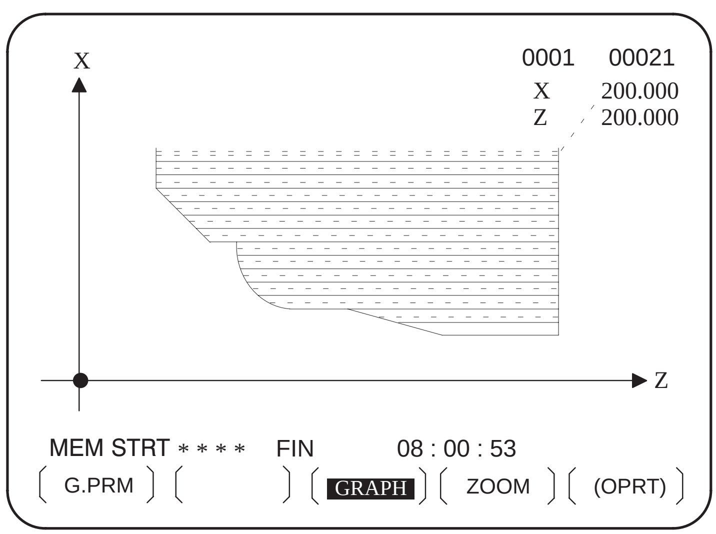
General

Graphic Display

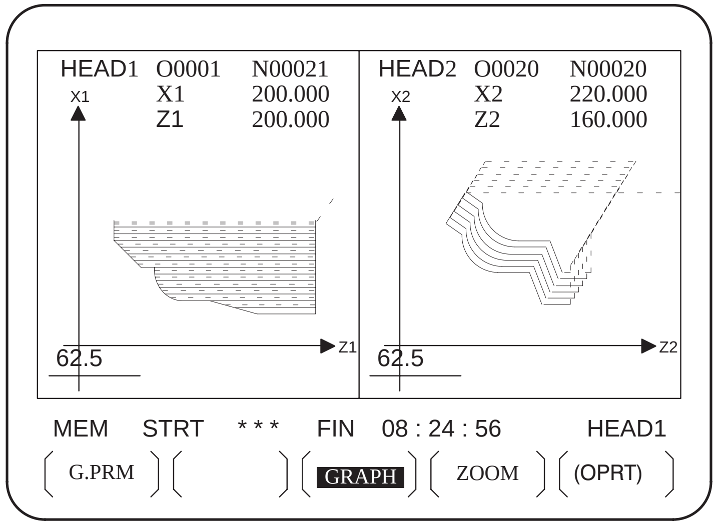
It is possible to draw the programmed tool path on the CRT screen, which makes it possible to check the progress of machining, while observing the path on the CRT screen.
In addition, it is also possible to enlarge/reduce the screen.
The drawing coordinates (parameter) and graphic parameters must be set before a tool path can be displayed.
With T series (Two–path control) the tool paths of two tool posts are displayed on the same screen, one on the right and the other on the left.



M series



T series



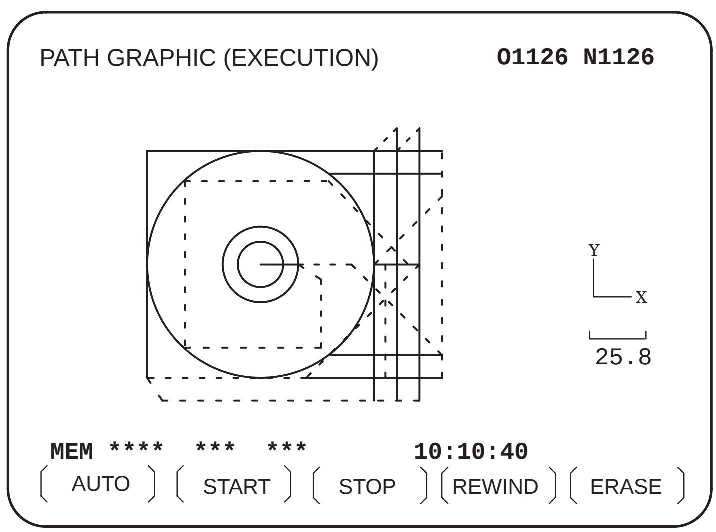
T series (Two–path control)

Dynamic graphic display (M series)

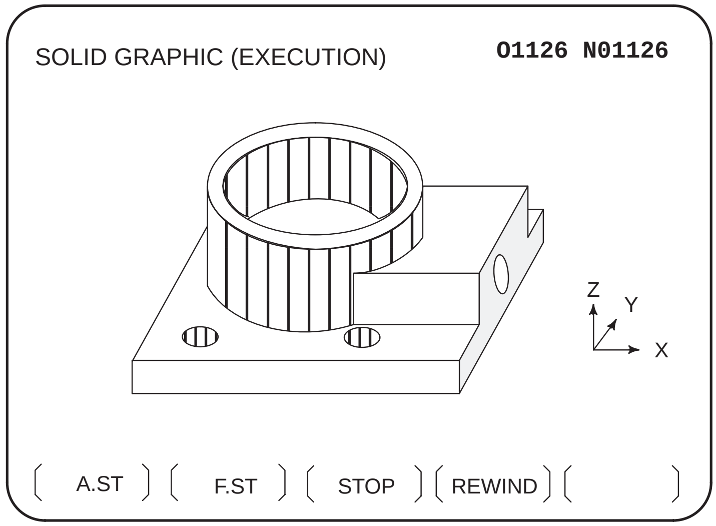
There are the following two functions in Dynamic Graphics.

Path graphic	This is used to draw the path of tool center commanded by the part program.
Solid graphic	This is used to draw the workpiece figure machined by tool movement commanded by the part program.

The path graphic function is used to precisely check the part program for drawing the tool path with a line. The solid graphic function is used to draw the workpiece figure to be machined with a program. Thus, it is easy to recognize roughly the part program. These two functions can be used freely by switching them.



Tool path Graph



Part Machined

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3003	MVG							

[Data type] Bit

MVG While drawing using the dynamic graphics function (with no machine movement), the axis–in–movement signal is:
0 : Output
1 : Not output

	#7	#6	#5	#4	#3	#2	#1	#0
6500		NZM			DPA	GUL	SPC	GRL
			DPO					

[Data type] Bit

GRL Graphic display (2–path control)
0 : Tool post 1 is displayed on the left, and tool post 2 is displayed on the right.
1 : Tool post 1 is displayed on the right, and tool post 2 is displayed on the left.

SPC Graphic display (2–path control) is done
0 : on two spindles and two tool posts
1 : on one spindle and two tool posts

GUL
0 : The positions of X1– and X2–axes are not replaced with each other in the coordinate system specified with parameter 6509. (2–path control)
1 : The positions of X1– and X2–axes are replaced with each other in the coordinate system specified with parameter 6509. (2–path control)

DPA Current position display on the graphic display screen
0 : Displays the actual position to ensure tool nose radius compensation
1 : Displays the programmed position

DPO Current position on the solid drawing (machining profile drawing) or tool path drawing screen
0 : Not appear
1 : Appears

NZM
0 : The screen image is not enlarged by specifying the center of the screen and magnification. (Screen image enlargement by a conventional method is enabled.)
1 : The screen image is enlarged by specifying the center of the screen and magnification. (Screen image enlargement by the conventional method is disabled.)

	#7	#6	#5	#4	#3	#2	#1	#0
6501			CSR					
			CSR	FIM	RID	3PL	TLC	ORG

[Data type] Bit

ORG Movement when coordinate system is altered during drawing
0 : Draws in the same coordinate system
1 : Draws in the new coordinate system (only for the path drawing)

TCL In solid drawing
0 : Not compensate the tool length
1 : Compensates the tool length

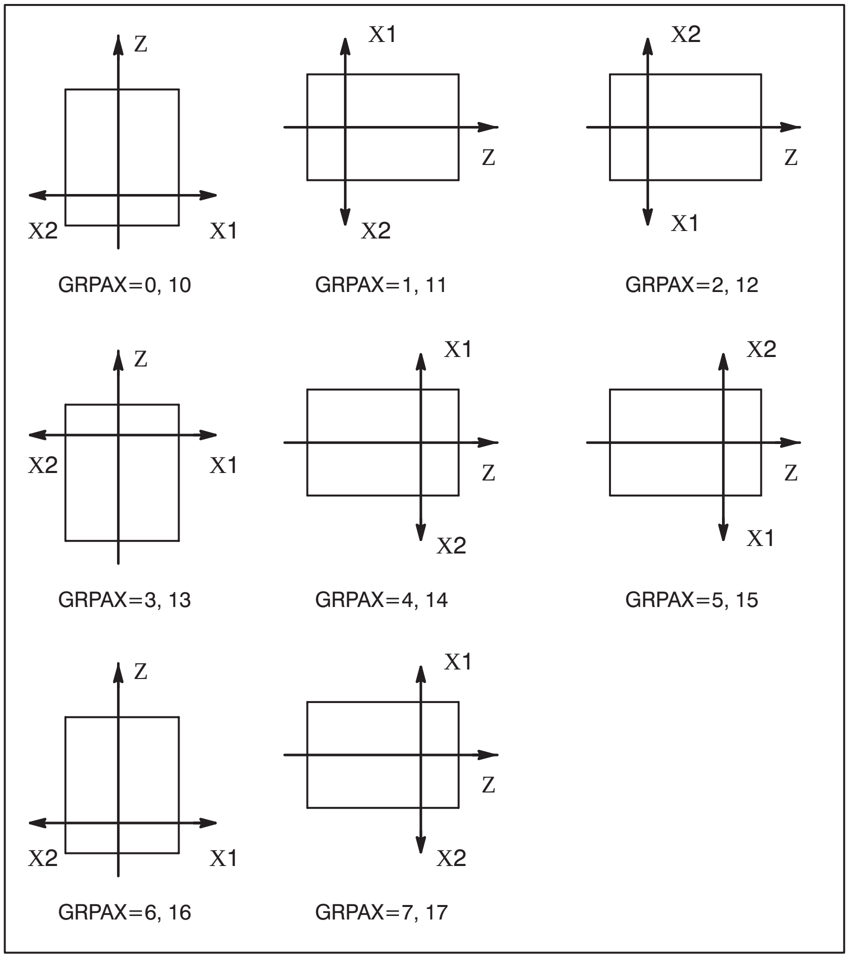
3PL Tri–plane drawing in solid drawing
0 : Drawn by the first angle projection
1 : Drawn by the third angle projection

RID In solid drawing
0 : Draws a plane without edges.
1 : Draws a plane with edges.

FIM Machining profile drawing in solid drawing
0 : Displayed in the coarse mode
1 : Displayed in the fine mode

CSR While the screen image is enlarged, the shape of the graphic cursor is:
0 : A square.
1 : An X.

6509	Coordinate system for drawing a single spindle (2–path control)



[Data type] Byte

[Valid data range] 0 to 7 and 10 to 17 (However, 0 to 7 are the same settings as 10 to 17.)

This parameter sets the coordinate system for drawing a single spindle (bit 1 of parameter 6500 = 1) for 2–path control.

The following shows the relationship between the settings and the drawing coordinate systems:

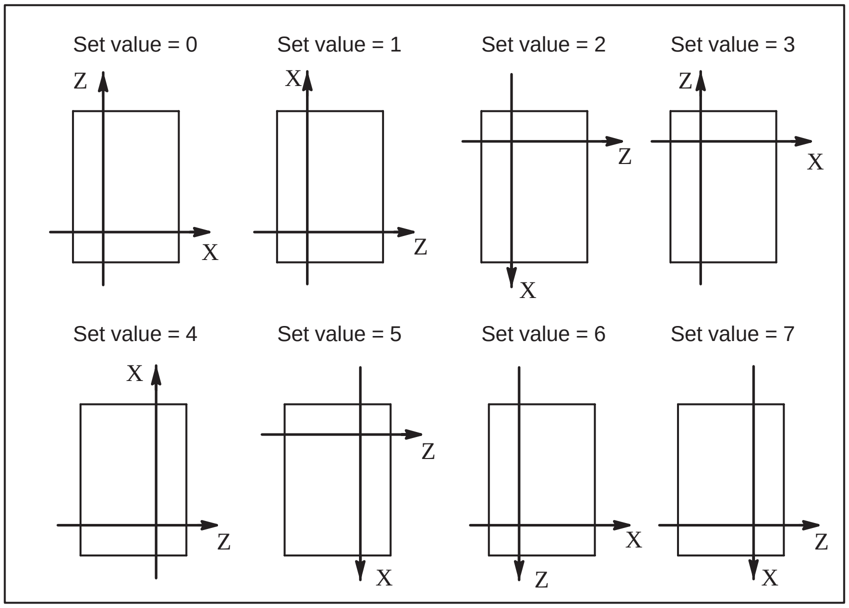
6510	Drawing coordinate system

[Data type] Byte

[Valid data range] 0 to 7

This parameter specifies the drawing coordinate system for the graphic function.

The following show the relationship between the set values and the drawing coordinate systems.



Note 1 This parameter is specified for each tool post in the 2–path control. A different drawing coordinate system can be selected for each tool post.

6511	Right margin in solid drawing
6512	Left margin in solid drawing
6513	Upper margin in solid drawing
6514	Lower margin in solid drawing

[Data type] Word

[Unit of data] Dot

These parameters set the machining profile drawing position in margins on the CRT screen. The unit is a dot.

Parameter No.	Margin area	Standard set value			
		DPO=0		DPO=1	
		9” CRT	14” CRT	9” CRT	14” CRT
6511	Right	0	0	200	100
6512	Left	0	0	0	0
6513	Upper	25	32	25	32
6514	Lower	0	10	0	10

Set DPO with parameter No. 6500#5.

6515	Change in cross–section position in tri–plane drawing

[Data type] Byte type

[Unit of data] Dot

[Valid data range] 0 to 10

This parameter sets the change in the cross–section position when a soft key is continuously pressed in tri–plane drawing. When zero is specified, it is set to 1.

Note

Note 1 When the dynamic graphics function is used, the graphics function cannot be used. (M series)

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	III.12.1	GRAPHICS FUNCTION
	III.12.2	DYNAMIC GRAPHIC DISPLAY
OPERATOR’S MANUAL (For Lathe) (B–62444E)	III.12.1	GRAPHICS FUNCTION

12.1.13 Displaying Operating Monitor

General

The reading on the load meter can be displayed for each servo axis and the serial spindle.

- **Display of the servo axes**

The reading on the load meter can be displayed for up to three servo axes by setting parameters 3151 to 3153.

- **Display of the spindle axes**

When serial spindles are used, the reading on the load meter and speedometer can be displayed only for the main serial spindle.

- **Speedometer**

Although the speedometer normally indicates the speed of the spindle motor, it can also be used to indicate the speed of the spindle by setting bit 6 (OPS) of parameter 3111 to 1.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3111		OPS	OPM					

[Data type] Bit

OPM Operating monitor
0: Not displayed
1: Displayed

OPS The speedometer on the operating monitor screen indicates:
0: Spindle motor speed
1: Spindle speed

3151	Number of the axis for which the first load meter for the servo motor is used
3152	Number of the axis for which the second load meter for the servo motor is used
3153	Number of the axis for which the third load meter for the servo motor is used

[Data type] Byte

[Valid data range] 0, 1, . . . , the number of control axes

Set the numbers of the axes for which measurement values on the load meters for the three servo motors are displayed. When only two load meters are used, set the third axis number to 0.

3160	Rated load of the load meter for each axis

[Data type] Word axis

[Valid data range] 0 to 255

The rated load of the motor corresponding to each load meter is set.

2086	Rated current parameter (RTCURR)

[Data type] Word axis

4127	Load meter displayed value for maximum output

[Data type] Word axis

Note

Note 1 The reading on the load meter depends on servo parameter 2086 and spindle parameter 4127.
These parameters are set by the automatic setting.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.11.1.8	Operating Monitor Display
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.11.1.8	Operating Monitor Display

12.1.14 Stamping the Machining Time

General

When a program is executed, its main program machining time is displayed on the program machining time display screen. The machining time can be displayed, in hours, minutes, and seconds format, for up to 10 main programs. Upon the execution of the eleventh and subsequent programs, previous data is cleared, starting from the oldest.

The time between the first start operation after a reset being performed in memory operation mode, until another reset is performed, is counted. When no reset is performed during operation, the time from the start of operation until M02 (M30) is encountered is counted. While the duration of the operation stop state is not being counted, the duration of the wait for completion of an M, S, T, or B function is counted.

The displayed machining time can be inserted (stamped) as a comment for a program stored in memory. The machining time is placed as a comment after the program number.

The machining time inserted after a program number can be displayed on the program directory screen (by setting bit 0 (NAM) of parameter No. 3107 to 1). Using this screen, the user can determine the machining time required for each program. This data is useful for on–site process planning.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3107								NAM

[Data type] Bit

NAM Program list
0 : Only program numbers are displayed.
1 : Program numbers and program names are displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3404			M02					

[Data type] Bit

M02 When M02 is specified in memory operation
0 : M02 is sent to the machine, and the program is positioned at tits beginning automatically. As completion signal FIN is returned without resetting or without resetting and rewinding, the program is restarted from the beginning.
1 : M02 is only sent to the machine. The program is positioned at its beginning by the reset and rewind signal.

Note

Note 1 When M02 does not reset the control unit, and completion signal FIN is sent to continuously reexecute the program from the beginning (when bit 5 (M02) of parameter No. 3404 is set to 0), counting of machining time is terminated by completion signal FIN.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.11.2.6	Stamping the machining time
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.11.2.6	Stamping the machining time

12.1.15 Software Operator's Panel

General

The software operator's panel function replaces part of the control switches on the machine operator's panel with soft switches which can be turned on or off using the CRT/MDI of the control unit.

The control switches for the functions listed in the following table can be replaced with soft switches. Also available are eight general–purpose soft switches which can be used additionally by the machine tool builder. These eight general–purpose soft switches can be optionally named by the machine tool builder. For control switches in groups 1 to 7, parameter (no.7200) can be used to select whether the control switches on the machine operator's panel or soft switches on the CRT/MDI of the control unit are used for each group.

Group1 :Mode selection

Group2 :Selection of jog feed axis, jog rapid traverse

Group3 :Selection of manual pulse generator feed axis, selection of manual pulse magnification

Group4 :Jog federate, federate override, rapid traverse override

Group5 :Optional block skip, single block, machine lock, dry run

Group6 :Protect key

Group7 :Feed hold

Group8 :General purpose

The states of all soft switches are informed to the PMC by output signals. Based on these output signals, the PMC should turn "1" or "0" input signals related to soft switch functions. In other words, turning "1" the soft switch assigned to single block operation, for example, does not cause the control unit to select single block operation internally. Single block operation is selected when the PMC turns to "1" the input signal for single block operation, instead.

Signal

Group	Function	Output signal	Related input signal
1	Mode selection	MD1O <F073#0> MD2O <F073#1> MD4O <F073#2> ZRNO <F073#4>	MD1 MD2 MD4 ZRN
2	Jog axis select	+J10 – +J40 –J10 – –J40 <F081>	+J1 – +J4 –J1 – –J4
	Jog rapid	RTO <F077#6>	RT
3	Handle feed	HS1AO <F077#0> HS1BO <F077#1> HS1CO <F077#2> HS1DO <F077#3>	HS1A HS1B HS1C HS1D
	Handle feed magnification	MP1O <F076#0> MP2O <F076#1>	MP1 MP2
4	Jog feed rate	*JV0O – *JV15O <F079, F080>	*JV0 – *JV15
	Feed rate override	*FV0O – *FV7O <F078>	*FV0 – *FV7
	Rapid traverse override	ROV1O <F076#4> ROV2O <F076#5>	ROV1 ROV2
5	Optional block skip	BDTO <F075#2>	BDT
	Single block	SBKO <F075#3>	SBK
	Machine lock	MLKO <F075#4>	MLK
	Dryrun	DRNO <F075#5>	DRN
6	Protect key	KEYO <F075#6>	KEY1 – KEY4
7	Feed hold	SPO <F075#7>	*SP
8	General purpose (Switch from 1st line to the 8th line on CRT)	OUT0 – OUT7 <F072>	

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F072	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
F073				ZRNO		MD4O	MD2O	MD1O
F074								
F075	SPO	KEYO	DRNO	MLKO	SBKO	BDTO		

	#7	#6	#5	#4	#3	#2	#1	#0
F076			ROV2O	ROV1O			MP2O	MP1O
F077		RTO			HS1DO	HS1CO	HS1BO	HS1AO
F078	*FV7O	*FV6O	*FV5O	*FV4O	*FV3O	*FV2O	*FV1O	*FV0O
F079	*JV7O	*JV6O	:JV5O	*JV4O	*JV3O	*JV2O	*JV1O	*JV0O
F080	*JV15O	*JV14O	*JV13O	*JV12O	*JV11O	*JV10O	*JV9O	*JV8O
F081	–J4O	+J4O	–J3O	+J3O	–J2O	+J2O	–J1O	+J1O

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7200		OP7	OP6	OP5	OP4	OP3	OP2	OP1

[Data type] Bit

OP1 Mode selection on software operator's panel
0 : Not performed
1 : Performed

OP2 JOG feed axis select and JOG rapid traverse buttons on software operator's panel
0 : Not performed
1 : Performed

OP3 Manual pulse generator's axis select and manual pulse generator's magnification switches on software operator's panel
0 : Not performed
1 : Performed

OP4 JOG speed override and rapid traverse override switches on software operator's panel
0 : Not performed
1 : Performed

OP5 Optional block skip, single block, machine lock, and dry run switches on software operator's panel
0 : Not performed
1 : Performed

OP6 Protect key on software operator's panel
0 : Not performed
1 : Performed

OP7 Feed hold on software operator's panel
0 : Not performed
1 : Performed

7210	Job–movement axis and its direction on software operator's panel V↑W

[Data type] Byte

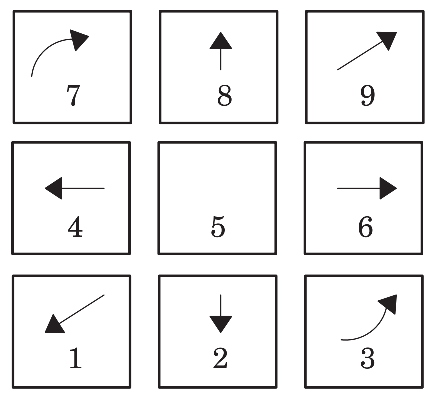
7211	Job–movement axis and its direction on software operator’s panel V↓W
7212	Job–movement axis and its direction on software operator’s panel V→W
7213	Job–movement axis and its direction on software operator’s panel V←W
7214	Job–movement axis and its direction on software operator’s panel V↙W
7215	Job–movement axis and its direction on software operator’s panel V↗W
7216	Job–movement axis and its direction on software operator’s panel V⤴W
7217	Job–movement axis and its direction on software operator’s panel V↷W

[Valid data range] 0 to 8

On software operator’s panel, set a feed axis corresponding to an arrow key on the CRT/MDI panel when jog feed is performed.

Set value	Feed axis and direction
0	Not moved
1	First axis, positive direction
2	First axis, negative direction
3	Second axis, positive direction
4	Second axis, negative direction
5	Third axis, positive direction
6	Third axis, negative direction
7	Fourth axis, positive direction
8	Fourth axis, negative direction

Arrow keys on the CRT/MDI panel



Example

Under X, Y, and Z axis configuration, to set arrow keys to feed the axes in the direction specified as follows, set the parameters to the values given below. [8 ↑] to the positive direction of the Z axis, [2 ↓] to the negative direction of the Z axis, [6 →] to the positive direction of the X axis [4←] to the negative direction of the X axis, [1↙] to the positive direction of the Y axis, [9↗] to the negative direction of the Y axis

Parameter No. 7210 = 5 (Z axis, positive direction)

Parameter No. 7211 = 6 (Z axis, negative direction)

Parameter No. 7212 = 1 (X axis, positive direction)

Parameter No. 7213 = 2 (X axis, negative direction)

Parameter No. 7214 = 3 (Y axis, positive direction)

Parameter No. 7215 = 4 (Y axis, negative direction)

Parameter No. 7216 = 0 (Not used)

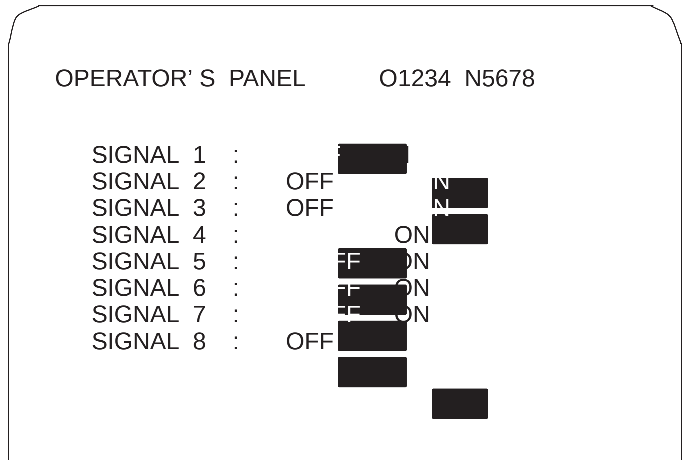
Parameter No. 7217 = 0 (Not used)

7220	Name of general–purpose switch on software operator’s panel
⋮	⋮
7283	Name of general–purpose switch on software operator’s panel

[Data type] Byte

Example

These parameters set the names of the general–purpose switches (SIGNAL 1 through SIGNAL 8) on the software operator’s panel as described below.



These names are set using character codes that are displayed in parameter Nos. 7220 to 7283.

Parameter No. 7220:

Sets the character code (083) corresponding to S of SIGNAL 1.

Parameter No. 7221:

Sets the character code (073) corresponding to I of SIGNAL 1.

Parameter No. 7222:

Sets the character code (071) corresponding to G of SIGNAL 1.

Parameter No. 7223:

Sets the character code (078) corresponding to N of SIGNAL 1.

Parameter No. 7224:

Sets the character code (065) corresponding to A of SIGNAL 1.

Parameter No. 7225:

Sets the character code (076) corresponding to L of SIGNAL 1.

Parameter No. 7226:

Sets the character code (032) corresponding to (space) of SIGNAL 1.

Parameter No. 7227:

Sets the character code (049) corresponding to 1 of SIGNAL 1.

Parameter Nos. 7228 to 7235:

Set the character codes of SIGNAL 2 shown in the figure above.

Parameter Nos. 7236 to 7243:

Set the character codes of SIGNAL 3 shown in the figure above.

Parameter Nos. 7244 to 7251:

Set the character codes of SIGNAL 4 shown in the figure above.

Parameter Nos. 7252 to 7259:

Set the character codes of SIGNAL 5 shown in the figure above.

Parameter Nos. 7260 to 7267:

Set the character codes of SIGNAL 6 shown in the figure above.

Parameter Nos. 7268 to 7275:

Set the character codes of SIGNAL 7 shown in the figure above.

Parameter Nos. 7276 to 7283:

Set the character codes of SIGNAL 8 shown in the figure above.

The character codes are shown in character code list on the following page.

Character to Code Correspondence Table

Char- acter	Code	Com- ment	Char- acter	Code	Comment	Char- acter	Code	Com- ment	Char- acter	Code	Comment
A	065		6	054		⍰	177		⍰	209	
B	066		7	055		⍰	178		⍰	210	
C	067		8	056		⍰	179		⍰	211	
D	068		9	057		⍰	180		⍰	212	
E	069			032	Space	⍰	181		⍰	213	
F	070		!	033	Exclamation mark	⍰	182		⍰	214	
G	071		I	034	Quotation marks	⍰	183		⍰	215	
H	072		#	035	Shape	⍰	184		⍰	216	
I	073		p	036	Dollar mark	⍰	185		⍰	217	
J	074		%	037	Percent	⍰	186		⍰	218	
K	075		&	038	Ampersand	⍰	187		⍰	219	
L	076		’	039	Apostrophe	⍰	188		⍰	220	
M	077		(	040	Left parenthesis	⍰	189		⍰	166	
N	078		)	041	Right parenthesis	⍰	190		⍰	221	
O	079		*	042	Asterisk	⍰	191		⍰	167	
P	080		+	043	Positive sign	⍰	192		⍰	168	
Q	081		,	044	Comma	⍰	193		⍰	169	
R	082		–	045	Negative sign	⍰	194		⍰	170	
S	083		.	046	Period	⍰	195		⍰	171	
T	084		/	047	Slash	⍰	196		⍰	172	
U	085		:	058	Colon	⍰	197		⍰	173	
V	086		;	059	Semicolon	⍰	198		⍰	174	
W	087		<	060	Left angle bracket	⍰	199		⍰	175	
X	088		=	061	Equal sign	⍰	200		+	222	Dakuten
Y	089		>	062	Right angle bracket	⍰	201		,	223	Han dakuten
Z	090		?	063	Question mark	⍰	202		#	161	Full stop
0	048		@	064	Commercial at mark	⍰	203		V	162	Left quotation mark
1	049		[	091	Left square bracket	⍰	204		W	163	Right quotation mark
2	050		^	092		⍰	205		”	164	Comma
3	051		o	093	Yen mark	⍰	206		·	165	Centered dot
4	052		]	094	Right square bracket	⍰	207			000	Space
5	053		_	095	Underline	⍰	208				

Note
The “dakuten” and “han dakuten” in Katakana also correspond to one character.

Note

Note 1 Only the modes shown below can be selected by soft switches. When the mode for DNC operation is to be equipped, for example, all control switches for mode selection should be on the machine operator's panel or a general–purpose soft switch should be used to select the mode for DNC operation.

Soft switches available for mode selection

· Manual data input
· Automatic operation
· Memory edit
· Manual handle feed / incremental feed
· Jog feed
· Manual reference position return

Note 2 Only one soft switch is available for protection key. However, four input signals are available for protection key (KEY1, KEY2, KEY3 and KEY4). Generally, four input signals are simultaneously turned to "1" or "0" according to the state of the soft switch for protection keys.

Note 3 When the soft switch for feed hold is turned on, output signal SPO is turned to "1". At this time, the PMC turns feed hold signal *SP to "0".

In contrast to the above, when the soft switch for feed hold is turned off, output signal SPO is turned "0" and the PMC turns signal *SP to "1". For soft switches other than feed hold and general soft switches, when an output signal informing the state of a soft switch is turned to "1", the corresponding input signal is turned to "1".

Note 4 The following table lists the override values which can be selected by soft switches for jog feedrate.

	*JV00 – *JV150 (*JV0 – *JV150) 15 12 8 4 0 bit				Override values (%)
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	0 1 0 1	0.1
2	1 1 1 1	1 1 1 1	1 1 1 1	0 0 0 1	0.14
3	1 1 1 1	1 1 1 1	1 1 1 0	1 0 1 1	0.2
4	1 1 1 1	1 1 1 1	1 1 1 0	0 1 0 0	0.27
5	1 1 1 1	1 1 1 1	1 1 0 1	1 0 1 0	0.37
6	1 1 1 1	1 1 1 1	1 1 0 0	1 0 1 1	0.52
7	1 1 1 1	1 1 1 1	1 0 1 1	0 1 1 1	0.72
8	1 1 1 1	1 1 1 1	1 0 0 1	1 0 1 1	1.0
9	1 1 1 1	1 1 1 1	0 1 1 1	0 0 1 1	1.4
10	1 1 1 1	1 1 1 1	0 0 1 1	0 1 1 1	2.0
11	1 1 1 1	1 1 1 0	1 1 1 1	0 0 0 1	2.7
12	1 1 1 1	1 1 1 0	1 0 0 0	1 1 0 1	3.7
13	1 1 1 1	1 1 0 1	1 1 1 1	0 1 1 1	5.2
14	1 1 1 1	1 1 0 1	0 0 1 0	1 1 1 1	7.2
15	1 1 1 1	1 1 0 0	0 0 0 1	0 1 1 1	10.0
16	1 1 1 1	1 0 1 0	1 0 0 0	0 1 1 1	14.0
17	1 1 1 1	1 0 0 0	0 0 1 0	1 1 1 1	20.0
18	1 1 1 1	0 1 0 1	0 1 1 1	0 0 1 1	27.0
19	1 1 1 1	0 0 0 1	1 0 0 0	1 0 1 1	37.0
20	1 1 1 0	1 0 1 1	1 0 1 0	1 1 1 1	52.0
21	1 1 1 0	0 0 1 1	1 1 0 1	1 1 1 1	72.0
22	1 1 0 1	1 0 0 0	1 1 1 0	1 1 1 1	100.0
23	1 1 0 0	1 0 0 1	0 1 0 0	1 1 1 1	140.0
24	1 0 1 1	0 0 0 1	1 1 0 1	1 1 1 1	200.0

Note 5 The following table lists the override values which can be selected by soft switches for feedrate override.

	*FV0O – *FV7O (*FV0 – *FV7) 7 4 0	Override values (%)

0	1 1 1 1	1 1 1 1	0
1	1 1 1 1	0 1 0 1	10
2	1 1 1 0	1 0 1 1	20
3	1 1 1 0	0 0 0 1	30
4	1 1 0 1	0 1 1 1	40
5	1 1 0 0	1 1 0 1	50
6	1 1 0 0	0 0 1 1	60
7	1 0 1 1	1 0 0 1	70
8	1 0 1 0	1 1 1 1	80
9	1 0 1 0	0 1 0 1	90
10	1 0 0 1	1 0 1 1	100
11	1 0 0 1	0 0 0 1	110
12	1 0 0 0	0 1 1 1	120
13	0 1 1 1	1 1 0 1	130
14	0 1 1 1	0 0 1 1	140
15	0 1 1 0	1 0 0 1	150
16	0 1 0 1	1 1 1 1	160
17	0 1 0 1	0 1 0 1	170
18	0 1 0 0	1 0 1 1	180
19	0 1 0 0	0 0 0 1	190
20	0 0 1 0	0 1 1 1	200

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.11.4.9	Displaying and Setting the Software Operator's Panel
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.11.4.12	Displaying and Setting the Software Operator's Panel

12.1.16 Multi–language Display

General

The CRT or LCD screens are displayed in a parameter–set language.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3102		SPN	HNG	ITA	CHI	FRN	GRM	JPN

[Data type] Bit type

Note 1 When this parameter is set, turn off the power once.

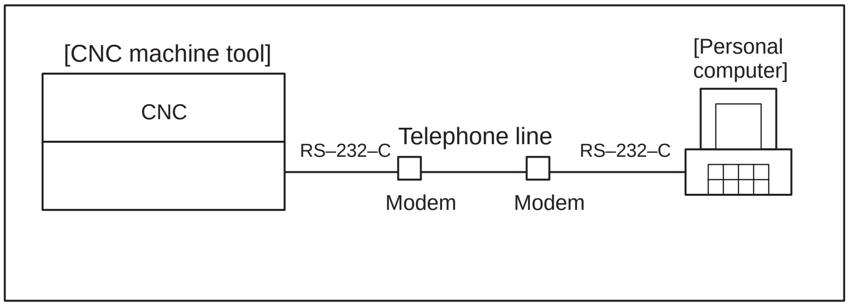
The language used in the display on the CRT is selected.

SPN	HNG	ITA	CHI	FRN	GRM	JPN	CRT display language
0	0	0	0	0	0	0	English
0	0	0	0	0	0	1	Japanese
0	0	0	0	0	1	0	German
0	0	0	0	1	0	0	French
0	0	0	1	0	0	0	Chinese (Taiwanese)
0	0	1	0	0	0	0	Italian
0	1	0	0	0	0	0	Hangul
1	0	0	0	0	0	0	Spanish

12.1.17 Remote Diagnosis

General

The remote diagnosis function allows CNC status monitoring and modification to CNC data to be performed remotely by menu–based operation. The remote diagnosis function, operating under MS–DOS, is installed on a standard personal computer, connected as a service terminal to the CNC via the RS–232–C interface, over a telephone line, and so on.



The remote diagnosis terminal software is sold separately.

The remote diagnosis function provides the following capabilities:

a. CNC programs

a–1 Computer → CNC

(1) CNC command data for verification
(2) Searching for a specified program
(3) Part program
(4) Deleting a specified program
(5) Deleting all programs

a–2 CNC → computer

(1) Part program
(2) Displaying a program directory
(3) Program number of a program being executed
(4) Sequence number of a sequence being executed

b. Computer → CNC

(1) Parameter
(2) Pitch error data
(3) Tool offset value
(4) Custom macro variable
(5) Selecting a display screen
(6) Memory contents
(7) PMC data
(8) Displaying a specified message
(9) All parameters

c. CNC → computer
  (1) Alarm information
  (2) Machine position
  (3) Absolute position
  (4) Skip position
  (5) Servo delay
  (6) Acceleration/deceleration delay
  (7) Diagnosis
  (8) Parameter
  (9) Tool life management data
  (10) Display screen status
  (11) Modal information
  (12) Pitch error data
  (13) Tool offset value
  (14) Custom macro variable
  (15) Memory contents
  (16) Ladder program
  (17) Actual feedrate
  (18) Status
  (19) A/D conversion data
  (20) PMC data
  (21) Screen character data
  (22) Printed circuit board information
  (23) Ladder title
  (24) Series and edition of PMC/ladder
  (25) All parameters

d. File function selection
  (1) Listing files
  (2) Referencing a file
  (3) Deleting a file
  (4) Copying a file
  (5) Renaming a file
  (6) Linking a file
  (7) Changing the current directory
  (8) Creating a directory
  (9) Deleting a directory

Note 1 An arrow ”→” indicates the direction of data flow.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0002								RDG

Setting entry is acceptable.

[Data type] Bit

RDG Remote diagnose
0 : Not performed
1 : Performed

Note 1 Set this bit to 0 when the remote diagnosis functions is not used. When this bit is set to 1, never modify the parameters related to remote diagnosis (parameter Nos. 0201 to 0223).

	#7	#6	#5	#4	#3	#2	#1	#0
0201						NCR	ASC	SB2

[Data type]

SB2 Number of stop bits
0 : 1 bit
1 : 2 bit

ASC Data output code
0 : ISO Code
1 : ASCII Code

NCR EOB (End of Block) is output as
0 : “LF” “CR” “CR”
1 : “LF”

0203	Band rate (For remote diagnosis)

[Data type] Byte

Set value	Baud rate
1	50
2	100
3	110
4	150
5	200
6	300

Set value	Baud rate
7	600
8	1200
9	2400
10	4800
11	9600

0204	Channel used for remote diagnosis

[Data type] Byte

[Valid data range] 0, 1, 2

Interface used for remote diagnosis

0, 1: RS–232–C Serial Port 1 (Channel 1)
2 : RS–232–C Serial Port 2 (Channel 2)

0206	Device ID number for remote diagnosis

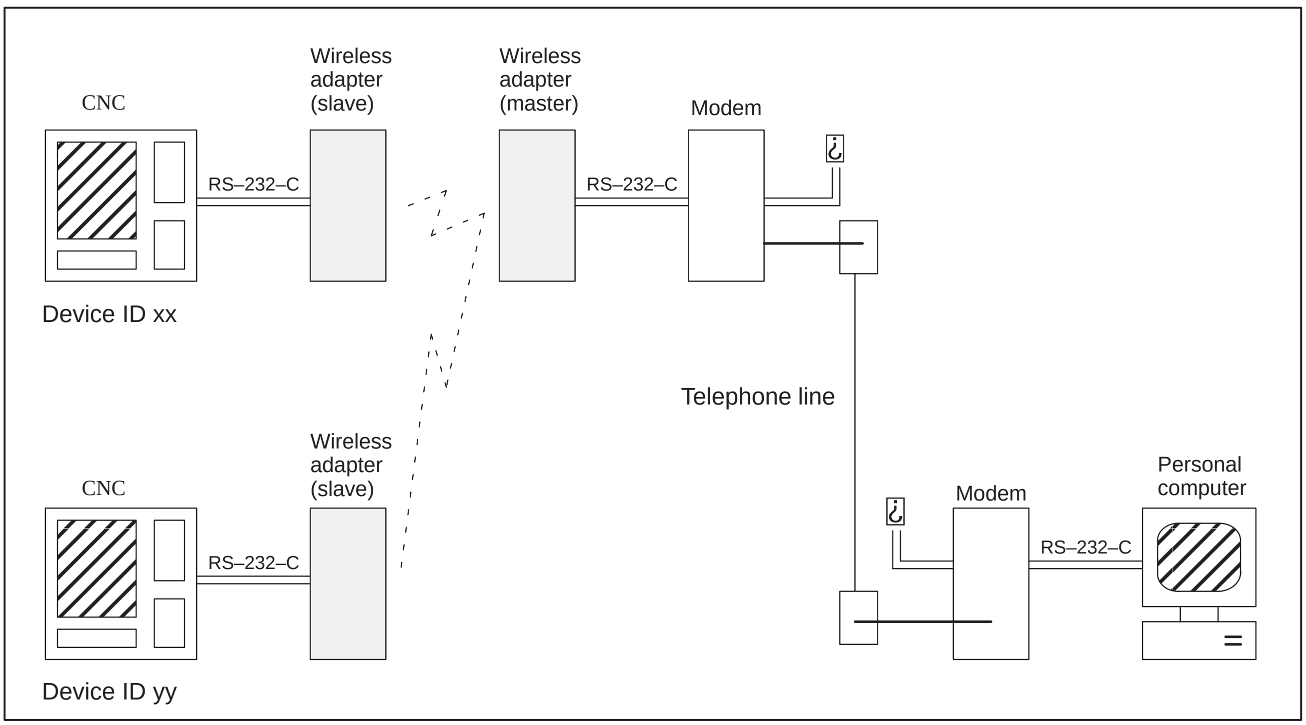
[Data type] Byte

[Valid data range] 0 to 20

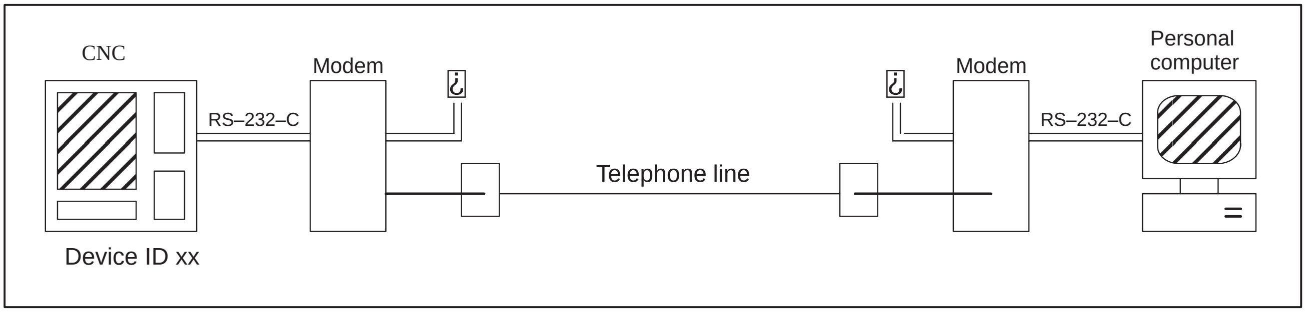
This parameter sets a device identifier (ID) for identifying each CNC with which the host computer is to communicate.

With the remote diagnosis function, multiple CNCs can be diagnosed via a single telephone line by using wireless adapters. Besides wireless adapter device numbers, a device ID can be assigned to each CNC to check that the correct CNC to be diagnosed is selected.

When wireless adapters are used



When wireless adapters are not used



0211	Password 1 for remote diagnose
0212	Password 2 for remote diagnose
0213	Password 3 for remote diagnose

[Valid data range] 1 to 99999999

These parameters set passwords for using the remote diagnosis function.

With the remote diagnosis function, three types of passwords are available for protecting data. These passwords help to prevent unauthorized persons from accessing system parameters and machining programs.

Password 1: Sets a password for all services of the remote diagnosis function. (No remote diagnosis function services are available until this password is entered on the host computer (personal or other)).

Password 2: Sets a password for part programs. (Program–related operations such as program data input/output and check cannot be performed until this password is entered on the host computer (personal or other)).

Password 3: Sets a password for parameters. (Parameter–related operations such as parameter data input/output cannot be performed until this password is entered on the host computer (personal or other)).

Note 1 Once a value other than 0 is set as a password, the password cannot be modified until the same value is set in the corresponding keyword parameter (parameter Nos. 221 to 223.) When a value other than 0 is set as a password, the parameter screen does not display the value of the password; only blanks are displayed. Care must be taken in setting a password.

0221	Key word 1 for remote diagnosis
0222	Key word 2 for remote diagnosis
0223	Key word 3 for remote diagnosis

[Valid data range] 1 to 99999999

These parameters set the keywords for passwords used with the remote diagnosis function.

Keyword 1: Keyword for password 1 (parameter No. 211)
Keyword 2: Keyword for password 2 (parameter No. 212)
Keyword 3: Keyword for password 3 (parameter No. 213)

When a value other than 0 is specified as a password (parameter Nos. 221 to 223), the password cannot be modified until the same value is set in the corresponding keyword parameter.

Note 1
- Upon power–up, the keyword parameters are set to 0.
- The parameter screen does not display any set keyword values; only blanks are displayed.

12.1.18 External Operator Message Logging and Display

General

External operator messages can be logged in a history file.

These messages can be displayed on the external operator message history screen.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3112						OMH		

[Data type] Bit

OMH The external operator message history screen is:
0 : Not displayed.
1 : Displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3113	MS1	MS0						MHC

MHC External operator message history data:
0 : Cannot be cleared.
1 : Can be cleared.
(Such data can be cleared using the [CLEAR] soft key.)

MS0, MS1 A combination of the number of characters preserved as external operator message history data and the number of history data items is set according to the table below.

MS1	MS0	Number of history data characters	Number of history data items
0	0	255	8
0	1	200	10
1	0	100	18
1	1	50	32

Note 1 When the values of MS0 and MS1 are changed, all preserved external operator message history data is cleared.

Reference item

OPERATOR'S MANUAL (Fot Machining Center) (B–62454E)	III.11.7.1	External operator message logging and display
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.11.7.1	External operator message logging and display

12.2 EDIT

12.2.1 Part Program Storage Length

General

One of the following part program length can be selected.

FS16/160 (m)	—	20	40	80	160	320	640	1280	2560	5120
FS18/180 (m)	10	20	40	80	160	320	640	1280	—	—

Alarm and message

Number	Message	Description
070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	IV.11.3.1	Displaying Memory Used and a List of Programs
OPERATOR'S MANUAL (For Lathe) (B–62444E)	IV.11.3.1	Displaying Memory Used and a List of Programs

12.2.2 No. of Registered Programs

General

One of the following no. of registered programs can be selected.

63/125/200/400/1000

Alarm and message

Number	Message	Description
072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 63 (basic), 125 (option), 200 (option), 400 (option), or 1000(option). Delete unnecessary programs and execute program registeration again.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	IV.11.3.1	Displaying Memory Used and a List of Programs
OPERATOR'S MANUAL (For Lathe) (B–62444E)	IV.11.3.1	Displaying Memory Used and a List of Programs

12.2.3 Memory Protection Key

General

A key called the data protection key is used to prevent part programs, offset values, parameters, and setting data from being registered, modified, or deleted erroneously.

Signal

Memory protection signal KEY1AKEY4 <G046#3A#6>

[Classification] Input signal

[Function] Enables the changing of the memory contents from the MDI panel. Four signals are provided. The operations that can be performed on the contents of memory by each signal vary depending on the setting of bit 7 (KEY) of parameter No. 3290.

When KEY = 0

- KEY1: Enables the input of tool compensation values and the workpiece zero point offset values.
- KEY2: Enables the input of setting data and macro variables.
- KEY3: Enables program loading and editing.
- KEY4: (Reserved)

When KEY = 1

- KEY1: Enables program loading and editing, as well as the input of PMC parameters.
- KEY2 to KEY4: Not used

[Operation] When a signal is set to 0, the associated operations are disabled.
When a signal is set to 1, the associated operations are enabled.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G046		KEY4	KEY3	KEY2	KEY1			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3290	KEY							

KEY For memory protection keys:
0 : The KEY1, KEY2, KEY3, and KEY4 signals are used.
1 : Only the KEY1 signal is used.

Note 1 The functions of the signals depend on whether KEY = 0 or KEY = 1.

Alarm and message

Warning message	Contents
WRITE PROTECT	Key input is invalid because of memory protect signal.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	IV.11	SETTING AND DISPLAYING DATA
OPERATOR’S MANUAL (For Lathe) (B–62444E)	IV.11	SETTING AND DISPLAYING DATA

12.2.4 Password Function

General

The password function locks NE9 (bit 4 of parameter No. 3202), used to protect program Nos. 9000 to 9999, by using the PASSWD (No. 3210) and KEYWD (No. 3211) parameters. When NE9 is locked, NE9 cannot be set to 0. Therefore, the protection for programs numbered 9000 to 9999 cannot be released unless the correct keyword is entered.

NE9 is locked when different values are set in the PASSWD and KEYWD parameters. The values set in the two parameters are not displayed. NE9 is unlocked when the value preset in the PASSWD parameter is set in the KEYWD parameter. When 0 is indicated for the PASSWD parameter, a value has not yet been set for PASSWD.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3202				NE9				

[Data type] Bit

NE9 Editing of subprograms with program numbers 9000 to 9999
0 : Not inhibited
1 : Inhibited

The following edit operations are disabled:

(1) Program deletion (Even when deletion of all programs is specified, programs with program numbers 9000 to 9999 are not deleted.)
(2) Program output (Even when outputting all programs is specified, programs with program numbers 9000 to 9999 are not output.)
(3) Program number search
(4) Program editing after registration
(5) Program registration
(6) Program collation
(7) Displaying programs

3210	Password(PASSWD)

[Data type] Two–word

Set a secret number to this parameter. Its value is not displayed.

Note 1 This parameter show 0, when no value is set to this parameter. Once a key is lock, parameter NE9 cannot become 0 and PASSWD cannot be changed unless you perform an unlock operation or perform the memory all clear operation. When an attempt is made to modify the password by MDI input operation in this state, the warning message ”WRITE PROTECTED” is displayed to indicate that the password cannot be modified. When an attempt is made to modify the password with G10 (programmable parameter input), P/S alarm No. 231 is issued.

3211	Keyword(KEYWD)

[Data type] Two–word

When the value set as the password (set in parameter No. 3210) is set to this parameter, the locked state is released and the user can now modify the password and the value set in bit 4 (NE9) of parameter No. 3202 becomes 0.

Note 1 The value set in this parameter is not displayed. When the power is turned off, this parameter is set to 0.

Alarm and message

Number	Message	Description
231	ILLEGAL FORMAT IN G10 OR L50	Any of the following errors occurred in the specified format at the programmable–parameter input. 1) Address N or R was not entered. 2) A number not specified for a parameter was entered. 3) The axis number was too large. 4) An axis number was not specified in the axis–type parameter. 5) An axis number was specified in the parameter which is not an axis type. 6) An attempt was made to reset bit 4 of parameter 3202 (NE9) or change parameter 3210 (PSSWD) when they are protected by a password. Correct the program.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	IV.9.9	PASSWORD FUNCTION
OPERATOR'S MANUAL (For Lathe) (B–62444E)	IV.9.9	PASSWORD FUNCTION

12.2.5 Background Editing

General

Editing a program while executing another program is called background editing. The method of editing is the same as for ordinary editing (foreground editing).
A program edited in the background should be registered in foreground program memory.
During background editing, all programs cannot be deleted at once.

Alarm and message

Number	Message	Description
???	BP/S alarm	BP/S alarm occurs in the same number as the P/S alarm that occurs in ordinary program edit. (070, 071, 072, 073, 074 085,086,087 etc.)
140	BP/S alarm	It was attempted to select or delete in the background a program being selected in the foreground. Use background editing correctly.
239	BP/S alarm	Background editing was performed while the external punch was being executed in external I/O device control.
240	BP/S alarm	Background editing was done while in MDI operation.

Note 1 Alarm in background edit is displayed in the key input line of the background edit screen instead of the ordinary alarm screen and is resettable by any of the MDI key operation.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.9.8	BACKGROUND EDITING
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.9.8	BACKGROUND EDITING

12.2.6 Playback

General

When the playback option is selected, the **TEACH IN JOG** mode (TJOG) and **TEACH IN HANDLE** mode (THND) are added. In these modes, a machine position along the X, Y, and Z axes obtained by manual operation is stored in memory as a program position to create a program. The words other than X, Y, and Z, which include O, N, G, R, F, M, S, T, P, Q, and EOB, can be stored in memory in the same way as in **EDIT** mode.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7100							THD	

[Data type] Bit type

THD Manual pulse generator in TEACH IN JOG mode
0 : Valid
1 : Invalid

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.10.3	CREATING PROGRAMS IN TEACH IN MODE
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.10.3	CREATING PROGRAMS IN TEACH IN MODE
CONNECTION MANUAL (This manual)	2.6	MODE SELECTION

12.2.7 Conversational Programming with Graphic Function

General

Programs can be created block after block on the conversational screen while displaying the G code menu.
Blocks in a program can be modified, inserted, or deleted using the G code menu and converstional screen.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.10.4	CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION
OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.10.4	CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION

13 INPUT/OUTPUT OF DATA

13.1 READER/PUNCHER INTERFACE

General

The data shown below can be input/output through reader/puncher interface.

1. Program
2. Offset data
3. Parameter
4. Pitch error compensation data
5. Custom macro common variables.

Parameter

This CNC has three channels of input/output device interfaces. The input/output device to be used is specified by setting the channel connected to that device in setting parameter I/O CHANNEL.
The specified data, such as a baud rate and the number of stop bits, of an input/output device connected to a specific channel must be set in parameters for that channel in advance.
For channel 1, two combinations of parameters to specify the input/output device data are provided.
The following shows the interrelation between the input/output device interface parameters for the channels.

Input/output channel number (parameter No. 0020)
↓

0020	I/O CHANNEL

Specify a channel for an input/output device.
I/ O CHANNEL
=0 : Channel1
=1 : Channel1
=2 : Channel2
=3 : Channel3

I/O CHANNEL	Parameter	Data
I/O CHANNEL=0 (channel 1)	0101	Stop bit and other data
	0102	Number specified for the input/output device
	0103	Baud rate
I/O CHANNEL=1 (channel 1)	0111	Stop bit and other data
	0112	Number specified for the input/output device
	0113	Baud rate
I/O CHANNEL=2 (channel 2)	0121	Stop bit and other data
	0122	Number specified for the input/output device
	0123	Baud rate
I/O CHANNEL=3 (channel 3)	0131	Stop bit and other data
	0132	Number specified for the input/output device
	0133	Baud rate
	0134	Selection of protocol
	0135	Selection of RS–422 or RS–232C, and other data

	#7	#6	#5	#4	#3	#2	#1	#0
0000							ISO	TVC

Setting entry is acceptable.

[Data type] Bit

TVC TV check
0 : Not performed
1 : Performed

ISO Code used for data output
0 : EIA code
1 : ISO code

●

0020	I/O CHANNEL: Selection of an input/output device

Setting entry is acceptable.

[Data type] Byte

[Valid data range] 0 to 3

0 : Select the device of channel 1 (I/O device connected to JD5A of Main CPU board)

1 : Select the device of channel 1 (I/O device connected to JD5A of Main CPU board)

2 : Select the device of channel 2 (I/O device connected to JD5B of Main CPU board)

3 : Select the device of channel 3 (I/O device connected to OPTION 1 board)

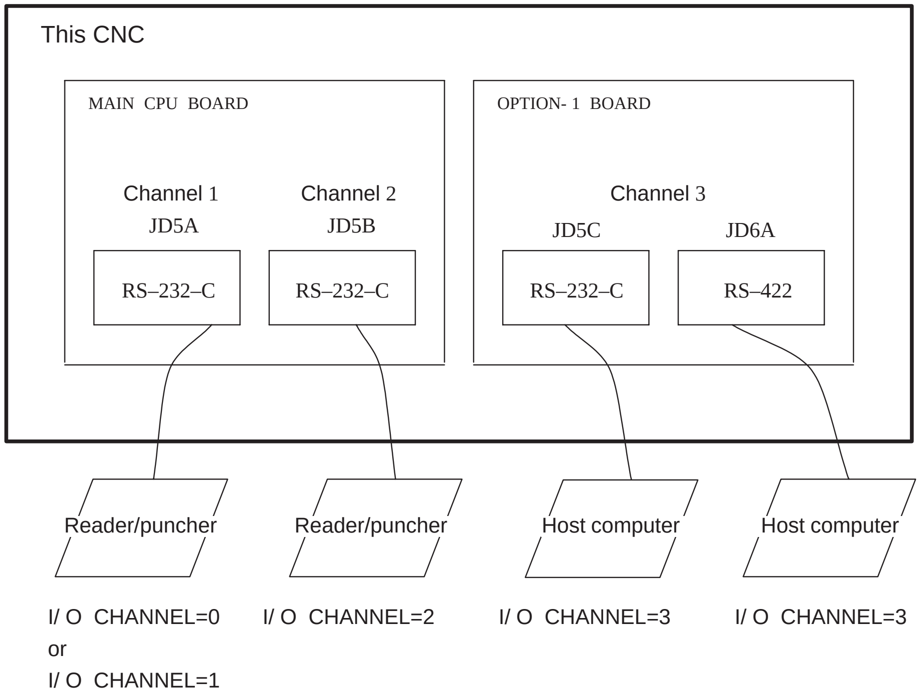
This CNC has three channels for interfacing I/O devices. Set the channels for interfacing I/O devices. Set the channel to which an I/O device to be used is connected.

Note 1 The input/output device used can be selected also on the setting screen. Using the setting screen is a more common method for selecting the device.

Note 2 The specified data, such as a baud rate and the number of stop bits, of an input/output device connected to a specific channel must be set in parameters for that channel in advance.

I/O CHANNEL=0 and I/O CHANNEL=1 both refer to channel 1. For each, parameters to set the baud rate, the number of stop bits, and other data are provided separately.

Note 3 JD5A, JD5B, JD5C and JD6A are the connectors on the PCB.



(1) Parameters common to all channels

	#7	#6	#5	#4	#3	#2	#1	#0
0100	ENS				NCR		CTV	

Setting entry is acceptable.

[Data type] Bit type

CTV Character counting for TV check in the comment section of a program.
0 : Not performed
1 : Performed

NCR Output of the end of block (EOB) in ISO code
0 : LF, CR, CR are output.
1 : Only LF is output.

ENS Action taken when a NULL code is found during read of EIA code
0 : An alarm is generated.
1: The NULL code is ignored.

(2) Parameters for channel 1 (I/O CHANNEL=0)

	#7	#6	#5	#4	#3	#2	#1	#0
0101	NFD				ASI			SB2

[Data type] Bit type

SB2 The number of stop bits
0 : 1
1 : 2

ASI Code used at data input
0 : EIA or ISO code (automatically distinguished)
1 : ASCII code

NFD Feed before and after the data at data output
0 : Output
1 : Not output

Note 1 When input/output devices other than the FANUC PPR are used, set NFD to 1.

0102	Number specified for the input/output device (when the I/O CHANNEL is set to 0)

[Data type] Byte

Set the number specified for the input/output device used when the I/O CHANNEL is set to 0, with one of the set values listed in Table 13.1 (a).

Table 13.1 (a)

Set value	Input/output device
0	RS–232–C (Used control codes DC1 to DC4)
1	FANUC CASSETTE ADAPTOR 1 (FANUC CASSETTE B1/ B2)
2	FANUC CASSETTE ADAPTOR 3 (FANUC CASSETTE F1)
3	FANUC PROGRAM FILE Mate, FANUC FA Card Adaptor FANUC FLOPPY CASSETTE ADAPTOR, FANUC Handy File FANUC SYSTEM P-MODEL H
4	RS–232–C (Not used control codes DC1 to DC4)
5	Portable tape reader
6	FANUC PPR FANUC SYSTEM P-MODEL G, FANUC SYSTEM P-MODEL H

0103	Baud rate (when the I/O CHANNEL is set to 0)

[Data type] Byte

Set baud rate of the input/output device used when the I/O CHANNEL is set to 0, with a set value in Table 13.1 (b).

Table 13.1 (b)

Set value	Baud rate (bps)	Set value	Baud rate (bps)
1	50	7	600
2	100	8	1200
3	110	9	2400
4	150	10	4800
5	200	11	9600
6	300	12	19200

(3) Parameters for channel 1 (I/O CHANNEL=1)

	#7	#6	#5	#4	#3	#2	#1	#0
0111	NFD				ASI			SB2

[Data type] Bit

These parameters are used when I/O CHANNEL is set to 1. The meanings of the bits are the same as for parameter 0101.

0112	Number specified for the input/output device (when I/O CHANNEL is set to 1)

[Data type] Byte

Set the number specified for the input/output device used when the I/O CHANNEL is set to 1, with one of the set values listed in Table 13.1 (a).

0113	Baud rate (when I/O CHNNEL is set to 1)

[Data type] Byte

Set the baud rate of the input/output device used when I/O CHANNEL is set to 1, with a value in Table 13.1 (b).

(4) Parameters for channel 2 (I/O CHANNEL=2)

	#7	#6	#5	#4	#3	#2	#1	#0
0121	NFD				ASI			SB2

[Data type] Bit

These parameters are used when I/O CHANNEL is set to 2. The meanings of the bits are the same as for parameter 0101.

0122	Number specified for the input/output device (when I/O CHANNEL is set to 2)

[Data type] Byte

Set the number specified for the input/output device used when I/O CHANNEL is set to 2, with a value in Table 13.1 (a).

0123	Baud rate (when the I/O CHANNEL is set to 2)

[Data type] Byte

Set the baud rate of the input/output device used when I/O CHANNEL is set to 2, with a value in Table 13.1 (b).

Alarm and message

Number	Message	Description
001	TH PARITY ALARM	TH alarm (A character with incorrect parity was input). Correct the tape.
002	TV PARITY ALARM	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective.
085	COMMUNICATION ERROR	When entering data in the memory by using Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect.
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
087	BUFFER OVERFLOW	When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read. I/O unit or P.C.B. is defective.
233	DEVICE BUSY	When an attempt was made to use a unit such as that connected via the RS–232–C interface, other users were using it.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	III.8	DATA INPUT/OUTPUT
OPERATOR’S MANUAL (For Lathe) (B–62444E)	III.8	DATA INPUT/OUTPUT

13.2 REMOTE BUFFER

Refer to Remote Buffer DESCRIPTIONS (B–61802E–1) for detailed informetion of remote buffer.

13.3 DNC1 INTERFACE

General

Refer to FANUC DNC1 DESCRIPTIONS(B–61782E) for detailed information of DNC1 interface.

Parameter

0020	I/O CHANNEL: Selection of an input/output device

Setting entry is acceptable.

[Data type] Byte

Set value. :10

0133	Baud rate

[Data type] Byte

The baud rate of HDLC is fixed to 460 kbps for DNC1.
Set following value:

Set value. :51

Note 1 When this parameter is set, the power must be turned off before operation is continued.

0141	System for connection between the CNC and host (DNC1 interface)

[Data type] Byte

[Valid data range] 1 or 2

This parameter specifies the system for connection (DNC1 interface) between the CNC and host.

Set value
1 : Point–to–point connection
2 : Multipoint connection

Note 1 When this parameter is set, the power must be turned off before operation is continued.

0142	Station address of the CNC (DNC1 interface)

[Data type] Byte

[Valid data range] 2 to 52

This parameter specifies the station address of the CNC when the CNC is connected via the DNC1 interface using multipoint connection.

Note 1 When this parameter is set, the power must be turned off before operation is continued.

MAP SCREEN

1. Communication screen

1.1 Operational Proceclure

1) Press function key <SYSTEM>.
2) Press right–end soft key repeatedly until the soft key [C–SERV],[C–OPER] are displayed shown below:

```
MDI    ***  STOP  ***  ***  ***  12 : 34 : 53

[C–OPER][C–SERV][            ][            ][ ( OPT  ) ]
```

1.2 Description of screen

1.2.1 Setting screen (one page)

Press soft key [C–OPER] and the following screen is displayed.

```
COMMUNICATION  OPERATION                        O0001   N00000

   DNC  FILE  SELECTION





>
MDI    ***  STOP  ***  ***  ***  12 :  34 : 53

[C–OPER][C–SERV][            ][            ][ ( OPT  ) ]
```

1.2.2 Service Screen

Press soft key [C–SERV] and the following screen is displayed.
Three pages are available and one of the pages is selected by page key.

```
COMMUNICATION  PARAMETER                              O0001    N00000
 NC    APPLICATION  NAME



 HOST  APPLICATION  NAME



>
   MDI      ***   STOP   ***   ***   ***   12 : 34 : 53

[C–OPER][C–SERV][          ][          ][ (  OPT   )  ]
```

```
COMMUNICATION  PARAMETER                              O0001    N00000
    CNC STATUS ( UNSOLICITED STATUS )
        RISING      UPPER    word      00000000   11111111
                    LOWER    word      11111111   11111111
        FALLING     UPPER    word      00000000   00000000
                    LOWER    word      01010100   00000010
    INFORMATION REPORT MASK
                    UPPER    word      00000000   00000000
                    LOWER    word      00000000   00000000
    CNC ALARM ( INFORMATION REPORT )
                    UPPER    word      11110001   00000000
                    LOWER    word      01000001   10000111

>
   MDI      ***   STOP   ***   ***   ***   12 : 34 : 53

[C–OPER][C–SERV][          ][          ][ (  OPT  )  ]
```

```
COMMUNICATION  PARAMETER                                O0001    N00000
    PASCAL STACK ADDRESS
                  UPPER LIMIT                            01ABC000
                  LOWER LIMIT                            01ABC0FF
    SERVICE MODE      1
                                              01010100   00000010
    SERVICE MODE      2
                                              01000000   00000001
    FILE REQUEST TIME OUT
                                                         12345678
    REMOTE REQUEST TIME OUT
                                                         12345678

>
  MDI     ***   STOP   ***   ***   ***   12 : 34 : 53

[C–OPER][C–SERV][              ][              ][ (  OPT  )  ]
```

1.3 Entry Method

1) Setting screen
   DNC file selection String input
2) Service screen page 1
   NC application String input
   Host application String input
3) Service screen page 2
   Status post enable Bit input
   Status post mask Bit input
   Alarm post Bit input
4) Service screen page 3
   Pascal stack address Hexadecimal input
   Service mode Bit input
   File request time-out Integer input
   Remote request time-out Integer input

String input: Details are described in Section 1.4, ”String Input Mode.”

Integer input: Entered as positive integers from 0 to 99999999

Bit input: Entered as 0/1; 00000000 to 11111111

Hexadecimal input: Entered as a hexadecimal number from 00000000 to FFFFFFFF

[Setting procedure]

1 Put the system in the MDI mode.
2 Cause the setting screen or service screen to appear, and press the [OPT] soft key.
3 Move the cursor to the item you want to specify, using the page and cursor keys.
4 Enter the setting value from the keypad, and press the [INPUT] soft key.

1.4 String Entry Mode

1) On the string input screen, the cursor is placed on top.

```
COMMUNICATION  PARAMETER                         O0001    N00000
■ DNC   FILE  SELECTION
 The cursor is positioned here at first.
```

2) Press the [(OPT)] key, and the following soft keys will appear on the screen.

```
[  STRING  ][  INPUT  ][  CLEAR  ][  INS. CH  ][  DEL. CH  ]
```

3) Press the [STRING] key, then it enters the string input mode.

```
        DNC   FILE  SELECTION
             ■
            ↑
            └  When the cursor is here, it is the string input mode.
>
MDI     ***   STOP   ***   ***   ***   12 : 34 : 53
[  STRING  ][  INPUT  ][  CLEAR  ][  INS. CH  ][  DEL. CH  ]
```

4) Specify the DNC file name.
(Example) O1000. PRG

5) Press the [INPUT] soft key to input the values.

```
   DNC   FILE  SELECTION
           O1000.PRG■

>
  MDI      ***   STOP   ***   ***   ***   12 : 34 : 53
[  STRING  ][  INPUT  ][  CLEAR  ][  INS. CH  ][  DEL. CH  ]
```

6) Deleting the DNC file name

- If you want to delete the entire name, press the [CLEAR] soft key.
- If you want to delete a letter at the cursor, press the [DEL.CH] soft key.

(Example) ABCDEFG■

Cursor

To delete letter E, move the cursor to that letter:
ABCDEFG
Press [DEL.CH], and the result will be:
ABCDFG

7) Overwriting

(Example) When you want to overwrite letters starting at C with letters XYZ, move the cursor to letter C.

ABCDEFG
Enter ”XYZ,” then press the [INPUT] soft key.
ABXYZFG

8) Inserting letters

(Example) To insert string ABC after string ABC, move the cursor to letter D.

ABCDEFG
Enter letters ABC.

ABCDEFG

Press the [INS.CH] soft key.
ABCABC EFG

9) Canceling the string input mode
If you press the [STRING] soft key, the cursor goes back to the position shown below, and the string input mode is canceled.
■ DNC file selection
O1000. PRG

(Note 1) The page keys do not work during the string input mode.

(Note 2) Switching the CNC mode during a string input mode cancels the string input mode.

3. PARAMETER DESCRIPTION

(1) Setting screen

- DNC file selection
  To start DNC operation, specify a file name in the host computer.
  Format: Oxxxx. PRG (where xxxx is a four–digit decimal number.)

(2) Service screen

- NC and host application names
  Specify these parameters with NC and host application names in the string input mode.
- Status post enable

Rising change	Upper word:	11111111	11111111
	Lower word:	01110000	10111110
Falling change	Upper word:	00000000	00000000
	Lower word:	11111111	11111111

Each bit in this parameter specifies whether to send the CNC status information to the DNC1 board according to a local request.
The CNC status information consists of a bit pattern listed in Table A. Each bit in the bit pattern corresponds to the bits in the rising and falling change words.
Setting value 0: Mask
1: Post
The rising change word specifies that a status information bit change from 0 to 1 be posted, and the falling change word, 1 to 0.

(Note) Use the lower word only.

○ CNC status information bit pattern (Table A)
Lower word

15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00

Bit position: Signal name	— CNC status information —
00: RWD	Rewind signal
01: AL	Alarm output signal
02: RST	Resetting signal
03: SPL	Feed hold lamp signal
04: STL	Cycle start lamp signal
05: OP	Cycle operating signal
06: SA	Servo ready
07: MA	CNC ready
08:	Not used
09:	Not used
10:	Not used
11:	Not used
12: M00	M00 decode output signal (*)
13: M01	M01 decode output signal (*)
14: M02	M02 decode output signal (*)
15: M30	M03 decode output signal (*)

* For the T series (two–path control), bits 08 to 11 correspond to M00 to M30 at HEAD2 respectively, and bits 12 to 15, at HEAD1.

- Status post mask
  Not used
- Alarm post
  This parameter specifies whether the bit position of a CNC alarm is posted to the host when a status change occurs according to a local request.

  Setting value
  0: Not to post
  1: To post

  The relationships between the parameter bits and alarms are as follows:

Upper word bit parameter

31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16

Lower word bit parameter

15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00

Bit number:	Alarm type
01:	P/S alarm
02:	Overheat alarm
05:	P/S 100 alarm
06:	Overtravel
12:	Servo alarm
13:	P/S 101 alarm
14:	P/S 000 alarm
16:	Battery alarm

- Service mode 2
  The DNC1 all–file directory information read function works as follows:
  00000000 00000001: Only file numbers are read.
  00000000 00000000: File numbers and sizes are read.
- File request time-out
- Remote request time-out
  This parameter specifies when a time-out is to occur for a request from the host.

  Parameter setting: 0 to 99999999
  Unit of time: ms

(Note 1) If value 0 is specified in the parameter, a conventional fixed time of about 25.6 seconds is specified as a time-out value.

(Note 2) The time for the time-out is counted in 32 ms units internally. The actual time for a time-out to occur is calculated as:

Actual time-out time = (parameter setting ÷ 32 + 1)* x 32 [ms]*

The term enclosed in parentheses is rounded down at a decimal point.

13.4 DNC2 INTERFACE

Refer to an item of FANUC DNC2 DESCRIPTIONS (B–61992E) for detailed information of DNC2 interface.

13.5 EXTERNAL I/O DEVICE CONTROL

General

It is possible to request from the outside that a program be registered, collated, or output.

- Registeration/Collation
  As triggered by the external read start signal EXRD, the background edit function saves programs from an external input unit onto tape and verifies them.
- Output
  As triggered by the external punch start signal EXWT, the background edit function outputs all programs stored in the part program memory to an external output device.

Signal

External Read Start Signal <G058#1>

[Classification] Input signal

[Function] Programs are registered through the reader/punch interface or remote buffer. Or the read programs are collated with programs already stored in the part program memory.

[Operation] When this signal becomes logical 1, the CNC operates as follows:

- In all modes other than the MDI mode, the background edit function reads programs from an external input device, and register them on the part program memory or collates them with programs already registered in the part program memory.
  (The memory protection key KEY3 <G046#5> determines whether to register or collate.)
- Bit 1 (RAL) of parameter No. 3201 selects whether to register all programs in a file or one program at a time. Bit 0 (RDL) of parameter No. 3201 can be used to delete all programs previously stored in the part program memory. However, it is impossible to delete programs protected by bits 0 (NE8) and 4 (NE9) of parameter No. 3202.
- When programs are being registed or collated, the read/punch busy signal (RPBSY) is kept to be logical 1.
- When the background processing–activated signal BGEACT is logical 1 (for example, during background editing or MDI mode), the external read start signal EXRD is ignored.
- When programs are being registered or collated, if the system is reset or the external read/punch stop signal EXSTP becomes logical 1, the registeration or collation is discontinued.
- If the foreground processing is already using the reader/punch interface (for example, during DNC operation or program reading in the edit mode), the external read start signal EXRD is ignored.

- There are some other conditions to determine whether a program can be registered or collated. For example, a program cannot be registered or collated, if a program with the same program number is being executed in the foreground processing.

External Punch Start Signal <G058#3>

[Classification] Input signal

[Function] Programs stored in the part program memory are output to an external unit via the reader/punch interface.

[Operation] When this signal becomes logical 1, the CNC operates as follows:

- In all modes other than the MDI mode, the background edit function outputs all programs stored in the part program memory to an external output device.
- When programs are being output, the read/punch busy signal RPBSY becomes logical 1.
- When the background processing–activated signal BGEACT is logical 1 (for example, during background editing or MDI mode), the external punch start signal EXWT is ignored.
- When programs are being output, if the system is reset or the external read/punch stop signal EXSTP becomes logical 1, the output is discontinued.
- If the foreground processing is already using the reader/punch interface (for example, during DNC operation or program reading in the edit mode), the external punch start signal EXWT is ignored.
- There are some other conditions to determine whether all programs can be output. For example, a program cannot be output, if it is running or protected by bits 0 (NE8) and 4 (NE9) of parameter No. 3202.

External Read/Punch Stop Signal EXSTP <G058#2>

[Classification] Input signal

[Function] When the external read/punch stop signal becomes logical 1, it stops program registeration, collation, or output via the reader/punch interface and program registeration and collation via the remote buffer.

[Operation] When this signal becomes logical 1, the CNC operates as follows:

- The program registeration, collation or output triggered by the external read or punch start signal is stopped immediately.

Background editing signal BGEACT <F053#4>

[Classification] Output signal

[Function] This signal indicates that the background edit function is operating.

[Output condition] This signal becomes logical 1 when:

- The [BG EDIT] soft key is pressed to put the CNC in the background edit mode.
- The MDI mode is selected.
- The external read or punch start signal starts program registeration, collation, or output.
- Program upload or download is started by the DNC1, DNC2, or MMC.

This signal becomes logical 0 when:

- The [BG END] soft key is pressed to terminate the background edit mode.
- The CNC shifts from the MDI mode to another mode.
- Program registeration or output triggered by the external read or punch start signal ends either normally or abnormally (reset or requested by the EXSTP signal).
- Program upload or download started by the DNC1, DNC2, or MMC is ended.

Read/punch busy signal RPBSY <F053#2>

[Classification] Output signal

[Function] This signal indicates that program registeration, collation, or output triggered by the external read or punch start signal is under way.

[Output condition] This signal becomes logical 1, when:

- The external read or punch start signal triggers program registeration, collation, or output.

This signal becomes logical 0, when:

- Program registeration collation or output triggered by the external read or punch start signal ends either normally or abnormally (reset or requested by the EXSTP signal).

Read/punch alarm signal
RPALM <F053#3>

[Classification] Output signal

[Function] This signal indicates that an alarm condition has occurred during program registeration, collation, or output triggered by the external read or punch start signal.

[Output condition] This signal becomes logical 1, when:

- An alarm condition occurs during program registeration, collation, or output triggered by the external read or punch start signal.

This signal becomes logical 0, when:

- The system is reset, or the external read/punch stop signal EXSTP is input.

Signal Address

	#7	#6	#5	#4	#3	#2	#1	#0
G058					EXWT	EXSTP	EXRD	

	#7	#6	#5	#4	#3	#2	#1	#0
F053				BGEACT	RPALM	RPBSY		

Parameter

Input/output channel number (parameter No. 0020)
↓

0020	I/O CHANNEL	I/O CHANNEL=0 (channel 1)	0101	Stop bit and other data
			0102	Number specified for the input/output device
			0103	Baud rate
		I/O CHANNEL=1 (channel 1)	0111	Stop bit and other data
			0112	Number specified for the input/output device
			0113	Baud rate
		I/O CHANNEL=2 (channel 2)	0121	Stop bit and other data
			0122	Number specified for the input/output device
			0123	Baud rate
		I/O CHANNEL=3 (channel 3)	0131	Stop bit and other data
			0132	Number specified for the input/output device
			0133	Baud rate
			0134	Selection of protocol
			0135	Selection of RS–422 or RS–232C, and other data

Specify a channel for an input/output device.

I/ O CHANNEL
- =0 : Channel1
- =1 : Channel1
- =2 : Channel2
- =3 : Channel3

	#7	#6	#5	#4	#3	#2	#1	#0
3201		NPE	N99			REP	RAL	RDL

[Data type] Bit

RDL When a program is registered by input/output device external control
- 0 : The new program is registered following the programs already registered.
- 1 : All registered programs are deleted, then the new program is registered.
  Note that programs which are protected from being edited are not deleted.

RAL When programs are registered through the reader/puncher interface
- 0 : All programs are registered.
- 1 : Only one program is registered.

REP Action in response to an attempt to register a program whose number is the same as that of an existing program
- 0 : An alarm is generated.
- 1 : The existing program is deleted, then the new program is registered.
  Note that if the existing program is protected from being edited, it is not deleted, and an alarm is generated.

N99 With an M99 block, when bit 6 (NPE) of parameter No. 3201 = 0, program registration is assumed to be:
0 : Completed
1 : Not completed

NPE With an M02, M30, or M99 block, program registration is assumed to be:
0 : Completed
1 : Not completed

	#7	#6	#5	#4	#3	#2	#1	#0
3202				NE9				NE8

[Data type] Bit type

NE8 Editing of subprograms with program numbers 8000 to 8999
0 : Not inhibited
1 : Inhibited

The following edit operations are disabled:

(1) Program deletion (Even when deletion of all programs is specified, programs with program numbers 8000 to 8999 are not deleted.)

(2) Program output (Even when outputting all programs is specified, programs with program numbers 8000 to 8999 are not output.)

(3) Program number search

(4) Program editing after registration

(5) Program registration

(6) Program collation

(7) Displaying programs

NE9 Editing of subprograms with program numbers 9000 to 9999
0 : Not inhibited
1 : Inhibited

The following edit operations are disabled:

(1) Program deletion (Even when deletion of all programs is specified, programs with program numbers 9000 to 9999 are not deleted.)

(2) Program punching (Even when punching of all programs is specified, programs with program numbers 9000 to 9999 are not punched.)

(3) Program number search

(4) Program editing after registration

(5) Program registration

(6) Program collation

(7) Displaying programs

Alarm and message

Number	Message	Description
079	BP/S ALARM	In memory or program collation,a program in memory does not agree with that read from an external I/O device. Check both the programs in memory and those from the external device.
085	BP/S ALARM	When entering data in the memory by using Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect.
086	BP/S ALARM	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
087	BP/S ALARM	When entering data in the memory by using reader /puncher interface, though the read terminate command is specified, input does not stop after 10 characters read. I/O unit or P.C.B. is defective.
180	BP/S ALARM	Remote buffer connection alarm has generated. Confirm the number of cables, parameters and I/O device.
233	BP/S ALARM	When an attempt was made to use a unit such as that connected via the RS–232–C interface, other users were using it.
239	BP/S ALARM	While punching was being performed with the function for controlling external I/O units ,background editing was performed.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	III.8.4	PROGRAM INPUT/OUTPUT
OPERATOR’S MANUAL (For Lathe) (B–62444E)	III.8.4	PROGRAM INPUT/OUTPUT

13.6 SIMULTANEOUS INPUT AND OUTPUT OPERATIONS (M series)

General

While an automation operation is being performed, a program input from an I/O device connected to the reader/punch interface can be executed and stored in memory.
Similarly, a program stored in memory can be executed and output through the reader/punch interface at the same time.

Basic procedure for input and run simultaneous operation

(1) Search the head of a program (file) you want to run and input.
(2) Select the DNC operation mode.
(3) Set the input and run simultaneous mode select signal STRD to logical 1.
(4) Activate automatic operation.
(5) The system repeats to input and run one block of data alternately.

Basic procedure for output and run simultaneous operation

(1) Select a program you want to run and output.
(2) Select the DNC operation mode.
(3) Set the output and run simultaneous mode select signal STWD to logical 1.
(4) Activate automatic operation.
(5) The system repeats to output and run one block of data alternately.

Signal

Input and run simultaneous mode select signal STRD <G058#5>

[Classification] Input signal

[Function] When this signal becomes logical 1, the control unit:

- Selects the input and run simultaneous mode.
  To select the input and run simultaneous mode, it is necessary to select the DNC operation mode and to set this signal to logical 1.

Output and run simultaneous mode select signal STWD <G058#6>

[Classification] Input signal

[Function] When this signal becomes logical 1, the control unit:

- Selects the output and run simultaneous mode.
  To select the output and run simultaneous mode, it is necessary to select the DNC operation mode and to set this signal to logical 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G058		STWD	STRD					

Alarm and message

Number	Message	Description
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
210	CAN NOT COMAND M198/M199	M198 and M199 are executed in the schedule operation. M198 is executed in the DNC operation. Modify the program.
222	DNC OP. NOT ALLOWED IN BG.–EDIT	Input and output are executed at a time in the background edition. Execute a correct operation.

Note

Note 1 M198 (file access) cannot be executed in the input, output and run simultaneous mode. An attempt to do so results in alarm No. 210.

Note 2 A macro control command cannot be executed in the input, output and run simultaneous mode. An attempt to do so results in alarm No. 123.

Note 3 If an alarm condition occurs during the input, output and run simultaneous mode, a block being processed when the alarm condition occurs and all blocks before that are input or output.

Note 4 In the output and run simultaneous mode, if a device used is a floppy disk drive or FA card, the file name is the execution program number.

Note 5 When a program is being executed in the output and run simultaneous mode, if a subprogram is called, only the main program is output.

13.7 EXTERNAL PROGRAM INPUT

General

By using the external program input start signal, a program can be loaded from an input unit into CNC memory.
When an input unit such as the FANUC Handy File or FANUC Floppy Cassette is being used, a file can be searched for using the workpiece number search signals, after which the program can be loaded into CNC memory.

Signal

External program input start signal MINP <G058#0>

[Classification] Input signal

[Function] This signal starts loading of a program from an input unit into CNC memory.

[Operation] When the signal is set to 1, the control unit operates as follows:

- When memory operation mode is set, but no automatic operation is being performed and program loading is not inhibited by the setting of the memory protection key, the CNC deletes all currently loaded programs, then loads a program from the external input unit into CNC memory.
- When the FANUC Handy File or FANUC Floppy Cassette is being used as the input unit, a desired file can be searched for using the workpiece number search signals (PN1 to PN16), after which the program can be loaded into CNC memory.
  File numbers are indicated using the workpiece number search signals, as follows:

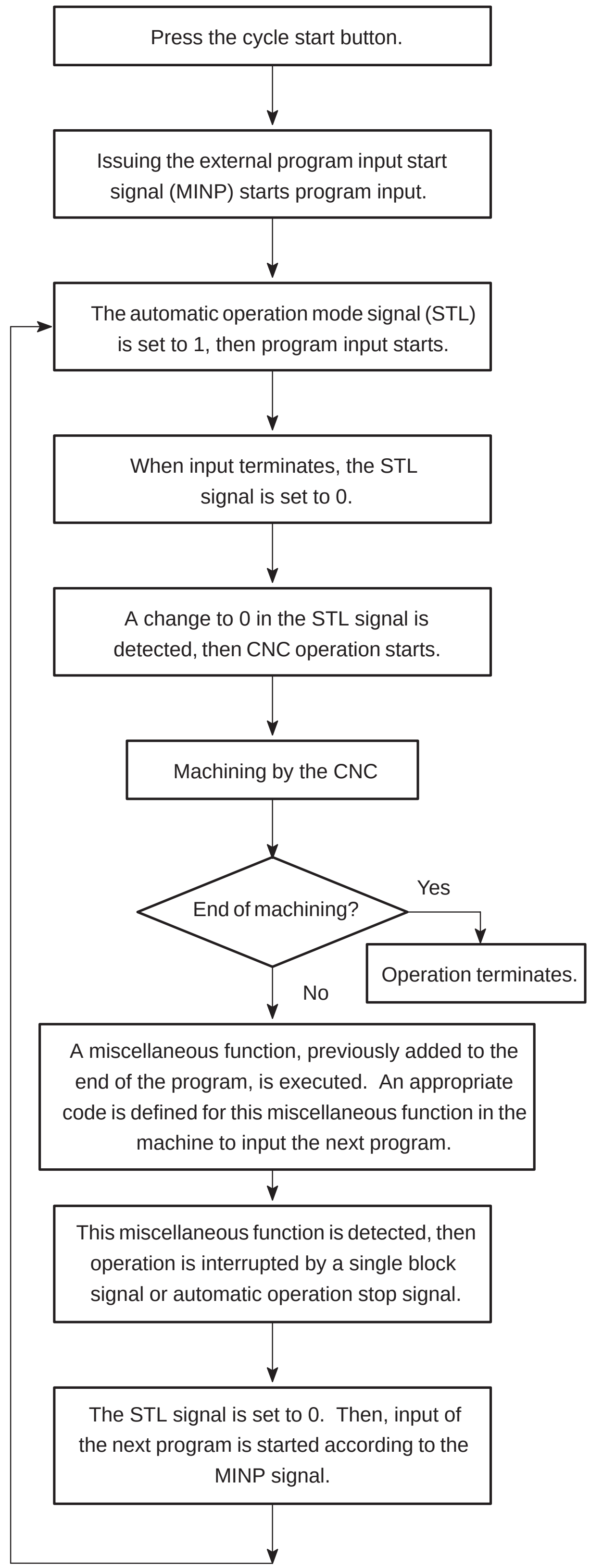
Workpiece no. search signal					File no.
PN16	PN8	PN4	PN2	PN1	
0	0	0	0	0	00
0	0	0	0	1	01
0	0	0	1	0	02
0	0	0	1	1	03
0	0	1	0	0	04
0	0	1	0	1	05
0	0	1	1	0	06
0	0	1	1	1	07
0	1	0	0	0	08
0	1	0	0	1	09

Workpiece no. search signal					File no.
PN16	PN8	PN4	PN2	PN1	
0	1	0	1	0	10
0	1	0	1	1	11
0	1	1	0	0	12
0	1	1	0	1	13
0	1	1	1	0	14
0	1	1	1	1	15
1	0	0	0	0	16
1	0	0	0	1	17
1	0	0	1	0	18
1	0	0	1	1	19
1	0	1	0	0	20
1	0	1	0	1	21
1	0	1	1	0	22
1	0	1	1	1	23
1	1	0	0	0	24
1	1	0	0	1	25
1	1	0	1	0	26
1	1	0	1	1	27
1	1	1	0	0	28
1	1	1	0	1	29
1	1	1	1	0	30
1	1	1	1	1	31

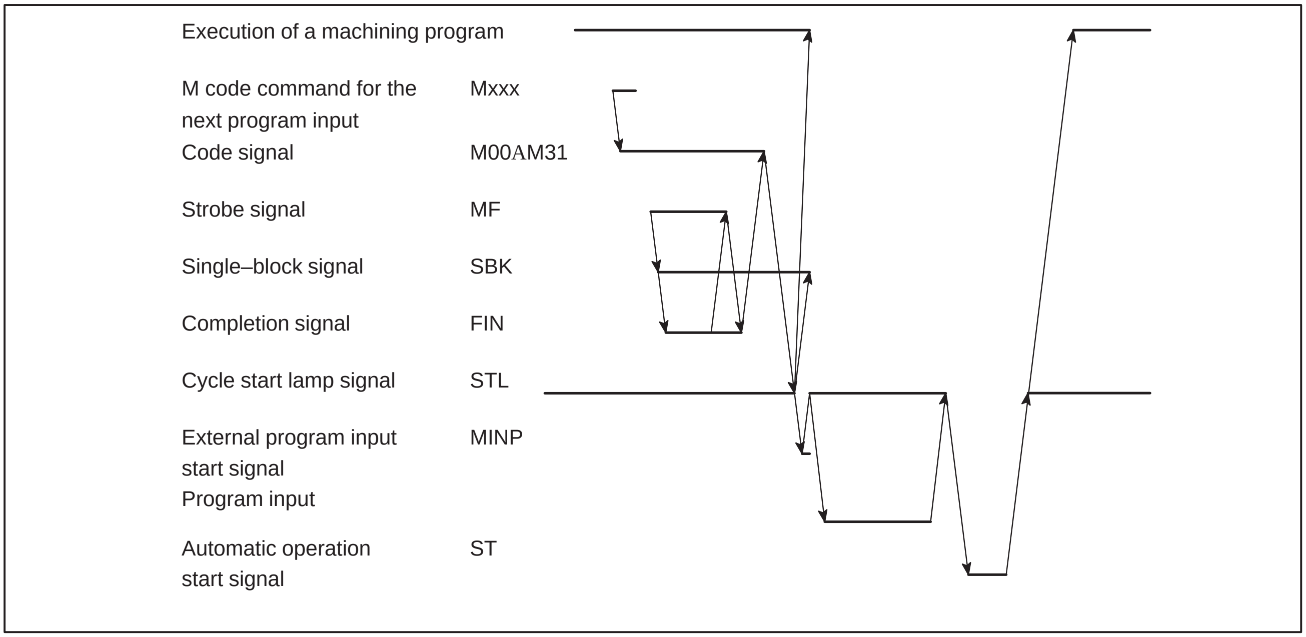
File No. 00 is used for special specification; specifying file No. 00 means that no search operation is to be performed. Therefore, numbers 01 to 31 can be assigned to files.

[Application] This function is applicable to the following case:
When a program to be used for machining is too large to be loaded into CNC memory, the program is divided into several segments. These segments are loaded into memory and executed, one by one.

In this case, the general operation flow is as shown below.



The timing chart for data reading is shown below.



Note 1 While a program is being input, the automatic operation mode signal STL is set to 1. Upon termination of program input, STL is set to 0.

Note 2 The M code used for input of the next program must not be buffered.

Signal Address

	#7	#6	#5	#4	#3	#2	#1	#0
G058								MINP

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3201	MIP							

[Data type] Bit type

MIP Specifies whether to load a program into memory according to the external program input start signal (MINP).
0 : Does not load a program into memory.
1 : Loads a program into memory.

Note

Note 1 A program can be input according to the external program input start signal only when the program has only one program number.
To read programs having multiple program numbers, reset the CNC each time the CNC reads one program. After reset, search for a desired program by using the workpiece number search signals, then input the program according to the external program input start signal.

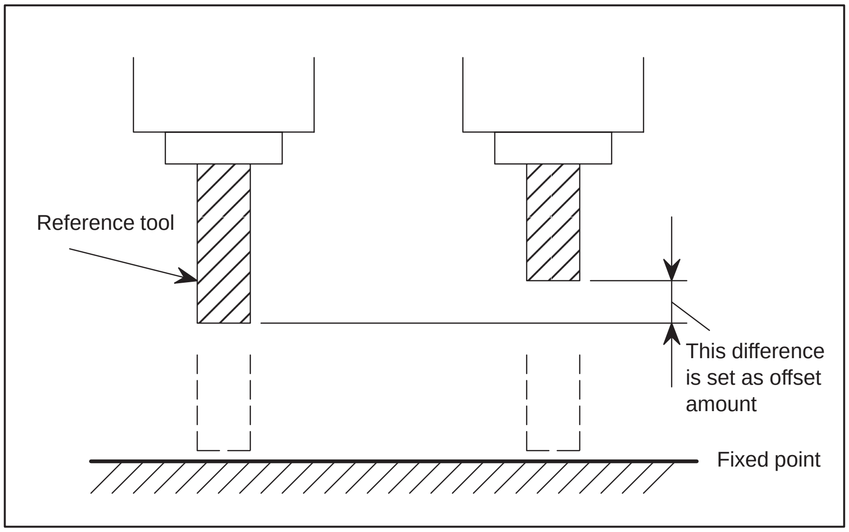
14 MEASUREMENT

14.1 TOOL LENGTH MEASUREMENT (M SERIES)

General

The value displayed as a relative position can be set in the offset memory as an offset value by a soft key.

Call offset value display screen on the CRT. Relative positions are also displayed on this screen. Reset the displayed relative position to zero. Set the tool for measurement at the same fixed point on the machine by a manual operation. The relative position display at this point shows difference between the reference tool and the tool measured and the relative position display value is then set as offset amounts.



Reference Item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	III.11.4.2	Tool Length Measurement

14.2 AUTOMATIC TOOL LENGTH MEASUREMENT (M SERIES) / AUTOMATIC TOOL OFFSET (T SERIES)

General

When a tool is moved to the measurement position by execution of a command given to the CNC, the CNC automatically measures the difference between the current coordinate value and the coordinate value of the command measurement position and uses it as the offset value for the tool. When the tool has been already offset, it is moved to the measurement position with that offset value. If the CNC judges that further offset is needed after calculating the difference between the coordinate values of the measurement position and the commanded coordinate values, the current offset value is further offset.

Signal

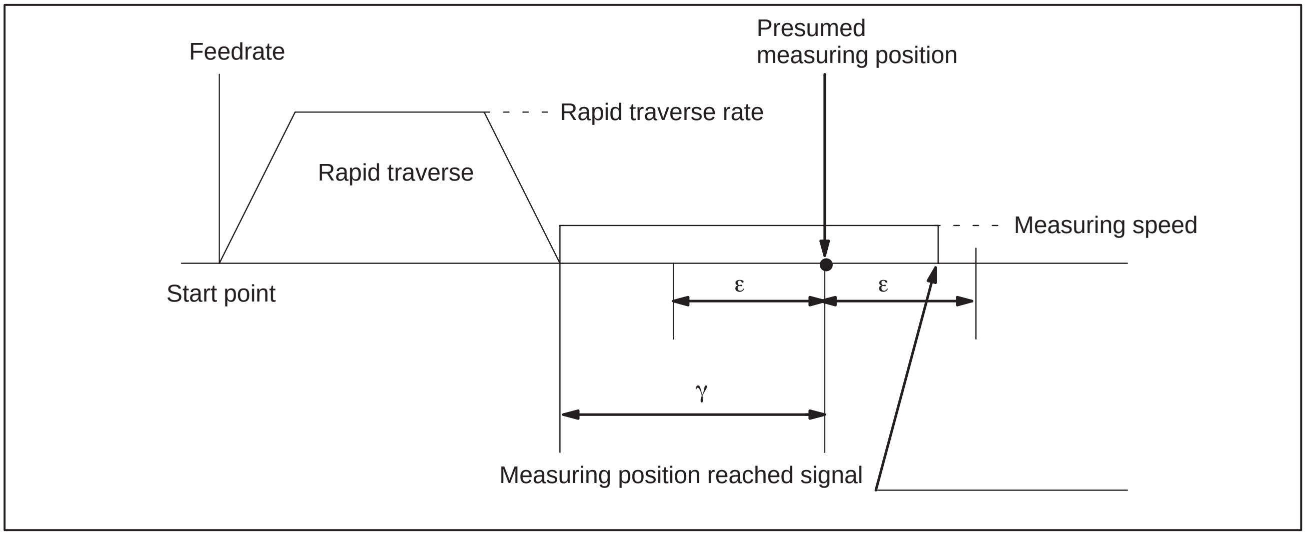
Measuring position reached signals XAE<X004#0>, YAE<X004#1>, ZAE<X004#2>(M series) XAE<X004#0>, ZAE<X004#1>(T series)

[Classification] Input signal

[Function] If the measuring position specified by a program command differs from the measuring position which a tool has reached in practice, that is, the position at the moment the measuring position reached signal has just been turned ”1”, the difference in the coordinate value is added to the current tool compensation value to update the compensation value. The tool is first fed to the specified measuring position by rapid traverse in a block where one of the following commands has been specified:

G37 (M series)
G36, G37 (T series)

The tool decelerates and temporarily stops at the distance O before the measuring position.
The tool then moves to the measuring position at the speed preset by a parameter no. 6241. If the measuring position reached signal corresponding to the G code is turned ”1” after the tool has approached within distance ε of the measuring position and before the tool overshoots the measuring position by distance ε, the control unit updates the compensation value and terminates the move command for the block. If the measuring position reached signal is not turned ”1” even after the tool has overshot the measuring position by distance ε, the control unit enters an alarm state and terminates the move command for the block without updating the compensation value.



[Operation] When the signal is turned ”1”, the control unit operates as follows:

- Reads the position of the tool along the axis currently specified and updates the current compensation value based on the difference between the specified measuring position and the read measuring position in the following case: When the measuring position reached signal corresponding to the G code is turned on in a block where G36 (T series) or G37 is specified after the tool is within distance ε of the measuring position specified by a program and before the tool overshoots the measuring position by distance ε. The control unit then stops the tool, and terminates the move command for the block.
- Enters an alarm state and terminates the move command for the block without updating the compensation value in the following case: When the measuring position reached signal corresponding to the command is turned ”1” in a block where G36 (T series), G37 is specified after the tool is within distance γ of the measuring position but before the tool is within distance ε of the measuring position.
- The control unit does not monitor the measuring position reached signal for its rising edge but monitors the state of the signal. If the signal remains ”1” when the next corresponding automatic tool length measurement (automatic tool compensation) is specified, the control unit enters an alarm state when the tool is within distance γ of the measuring position.

Notes
1. The measuring position reached signal requires at least 10 msec.
2. The CNC directly inputs the measuring position reached signals from the machine tool; the PMC does not process them.
3. If automatic tool compensation nor automatic tool length measurement is not used, the PMC can use the signal terminals corresponding to the measuring position reached signal as the general-purpose input signals.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X004							ZAE	XAE
						ZAE	YAE	XAE

Parameter

6241	Feedrate during measurement of automatic tool compensation
	Feedrate during measurement of tool length automatic compensation

[Data type] Word type

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotation axis	1 deg/min	6 to 15000	6 to 12000

This parameter sets the feedrate during measurement of automatic tool compensation (T series) and tool length automatic compensation (M series).

6251	γ value on X axis during automatic tool compensation
	γ value during tool length automatic compensation

6252	γ value on Z axis during automatic tool compensation

[Data type] Two–word type

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1 to 99999999

These parameters set the γ value during automatic tool compensation (T series) or tool length automatic compensation (M series).

Note 1 Set a radius value irrespective of whether the diameter programming or the radius programming is specified.

6254	ε value on X axis during automatic tool compensation
	ε value during tool length automatic compensation

6255	ε value on Z axis during tool automatic compensation

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1 to 99999999

These parameters set the ε value during automatic tool compensation (T series) or tool length automatic compensation (M series).

Note 1 Set a radius value irrespective of whether the diameter programming or the radius programming is specified.

Alarm and Message

Number	Message	Description
080	G37 ARRIVAL SIGNAL NOT ASSERTED (M series)	In the automatic tool length measurement function (G37), the measurement position reached signal (XAE, YAE, or ZAE) is not turned on within an area specified in parameter 6254 (value ε). This is due to a setting or operator error.
	G37 ARRIVAL SIGNAL NOT ASSERTED (T series)	In the automatic tool compensation function (G36, G37), the measurement position reached signal (XAE or ZAE) is not turned on within an area specified in parameter 6254, and 6255 (value ε). This is due to a setting or operator error.
081	OFFSET NUMBER NOT FOUND IN G37 (M series)	Tool length automatic measurement (G37) was specified without a H code. (Automatic tool length measurement function) Modify the program.
	OFFSET NUMBER NOT FOUND IN G37 (T series)	Automatic tool compensation (G36, G37) was specified without a T code. (Automatic tool compensation function) Modify the program.

Number	Message	Description
082	H–CODE NOT ALLOWED IN G37 (M series)	H code and automatic tool compensation (G37) were specified in the same block. (Automatic tool length measurement function) Modify the program.
	T–CODE NOT ALLOWED IN G37 (T series)	T code and automatic tool compensation (G36, G37) were specified in the same block. (Automatic tool compensation function) Modify the program.
083	ILLEGAL AXIS COMMAND IN G37 (M series)	In automatic tool length measurement, an invalid axis was specified or the command is incremental. Modify the program.
	ILLEGAL AXIS COMMAND IN G37 (T series)	In automatic tool compensation (G36, G37), an invalid axis was specified or the command is incremental. Modify the program.

Note

Note 1 Measurement speed, γ, and $\varepsilon$are set as parameters.ε must be positive numbers so that $\gamma>\varepsilon$.

Note 2 The compensation value is updated by the following formula:

New compensation value = (Current compensation value)+[(Current position of the tool along the specified axis when the measuring position reached signal is turned on) – (specified measuring position)]

The following compensation values are updated:

(1) In a M series, the compensation value corresponding to the tool compensation number selected by an H code.

When offset memory A is used, the offset value is changed.

When offset memory B is used, the tool wear compensation value is changed.

When offset memory C is used, the tool wear compensation value for the H code is changed.

(2) In a T series, the compensation value corresponding to the tool compensation number selected by a T code and to the specified axis (X, Z) in G36, G37.

Note 3 The maximum measuring error is calculated as shown below.

$$ERRmax = Fm \times \frac{1}{60} \times \frac{4}{1000}$$

ERRmax: Maximum measuring error (mm)

Fm : Measuring feedrate (mm/min)

If Fm = 100 mm/min, for example, ERRmax = 0.007 mm

Note 4 After the measuring position reached signal has been detected, the tool moves for a maximum of 20 msec, then stops. Values for calculating the compensation amount, that is the coordinate of the tool where the tool reached the measuring position are not those obtained after stop, but those obtained at the position where the measuring position reached signal was detected.
The overtravel amount for 20 msec is calculated as follows.

$$Qmax = Fm \times \; \times \frac{1}{60} \quad \frac{1}{1000} \; (20 + Ts)$$

Qmax: Maximum overtravel amount (mm)

Fm : Measuring feedrate (mm/min)

Ts : Servo time constant [msec] (1/loop gain)

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.15.2	AUTOMATIC TOOL LENGTH MEASUREMENT (G37)
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.15.6	AUTOMATIC TOOL OFFSET (G36, G37)

14.3 SKIP FUNCTION

14.3.1 Skip Function

General

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

The coordinate values when the skip signal is turned on can be used in a custom macro because they are stored in the custom macro system variable #5061 to #5068, as follows:

#5061 X axis coordinate value
#5062 Z axis coordinate value
#5063 3rd axis coordinate value
:
:
#5068 8th axis coordinate value

Signal

Skip Signal SKIP<X004#7> SKIPP<G006#6> (T series)

[Classification] Input signal

[Function] This signal terminates skip cutting. That is, the position where a skip signal turns to ”1” in a block containing G31 is stored in a custom macro variable, and the move command of the block is terminated at the same time.

[Operation] When a skip signal turns to “1”, the control unit functions as described below.

(1) When a block contains a skip cutting command G31, the control unit reads and stores the current position of the specified axis at that time. The control unit stops the axis, then cancels the remaining distance that the block was supposed to be moved.

(2) The skip signal is monitored not for a rising edge, but for its state. So, if a skip signal continues to be “1”, a skip condition is assumed to be satisfied immediately when the next skip cutting is specified.

Notes
1 The skip signal requires at least 10 msec.
2 The CNC directly reads the skip signal SKIP<X004#7> from the machine tool; the PMC no longer requires to process the signal.
3 If the skip function G31 is not used, the PMC can use the signal terminal SKIP<X004#7> corresponding to the skip signal as a general purpose input signal.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X004	SKIP							

	#7	#6	#5	#4	#3	#2	#1	#0
G006		SKIPP						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6200	SKF						SK0	GSK
	SKF						SK0	

[Data type] Bit

SKF Dry run, override, and automatic acceleration/deceleration for G31 skip command
0 : Disabled
1 : Enabled

SK0 This parameter specifies whether the skip signal is made valid under the state of the skip signal SKIP (bit 7 of X004) and the multistage skip signals (bits 0 to 7 of X004) (for the T series only).
0 : Skip signal is valid when these signals are 1.
1 : Skip signal is valid when these signals are 0.

GSK In skip cutting (G31), the signal SKIPP (bit 6 of G006) is:
0 : Not used as a skip signal.
1 : Used as a skip signal.

Alarm and message

Number	Message	Description
035	CAN NOT COMMANDED G31 (T series)	Skip cutting (G31) was specified in tool nose radius compensation mode. Modify the program.
036	CAN NOT COMMANDED G31 (M series)	Skip cutting (G31) was specified in cutter compensation mode. Modify the program.

Note

Note 1 Disable feedrate override, dry run, and automatic acceleration/deceleration (enabled with parameter No. 6200#7 SKF=1) when the feedrate per minute is specified, allowing for reducing an error in the position of the tool when a skip signal is input. These functions are enabled when the feedrate per rotation is specified.

Note 2 The G31 block is set to G01 mode. The feedrate is specified by an F code.

Note 3 When the measuring motion is made by utilizing the skip signal, program a constant feedrate; otherwise, if the feedrate changes, the measuring error will be noticeable. With a constant feedrate, the maximum measuring error can be calculated as follows:

$$ERRmax = Fm \times \frac{1}{60} \times \frac{4}{1000}$$

ERRmax : Maximum measuring error (mm or inch)

Fm : Measuring feedrate (mm/min or inch/min)

Note 4 Overtravel amount Qmax after skip signal has been turned to ”1” is calculated by the following:

$$Qmax = Fm \times \frac{1}{60} \times \frac{1}{1000} (20^{\#1} + Tc + Ts)$$

Qmax : Overtravel amount (mm or inch)

Fm : Feedrate (mm/min or inch/min)

Tc : Cutting time constant (ms) (Normally 0)

Ts : Servo time constant (ms) (1/1000 loop gain)

#1 : The value becomes 28 when the skip signal SKIPP <G006#6> is used. (Also it changes according to the precessing time of ladder program).

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–62454E)	II.4.11	SKIP FUNCTION(G31)
OPERATOR’S MANUAL (For Lathe) (B–62444E)	II.4.11	SKIP FUNCTION(G31)

14.3.2 High–speed Skip Signal

General

The skip function operates based on a high-speed skip signal (HDI0 – HDI7 : connected directly to the NC; not via the PMC) instead of an ordinary skip signal (X004#7). In this case, up to eight signals can be input.

Delay and error of skip signal input is 0 - 2 msec at the NC side (not considering those at the PMC side).

This high-speed skip signal input function keeps this value to 0.1 msec or less, thus allowing high precision measurement.

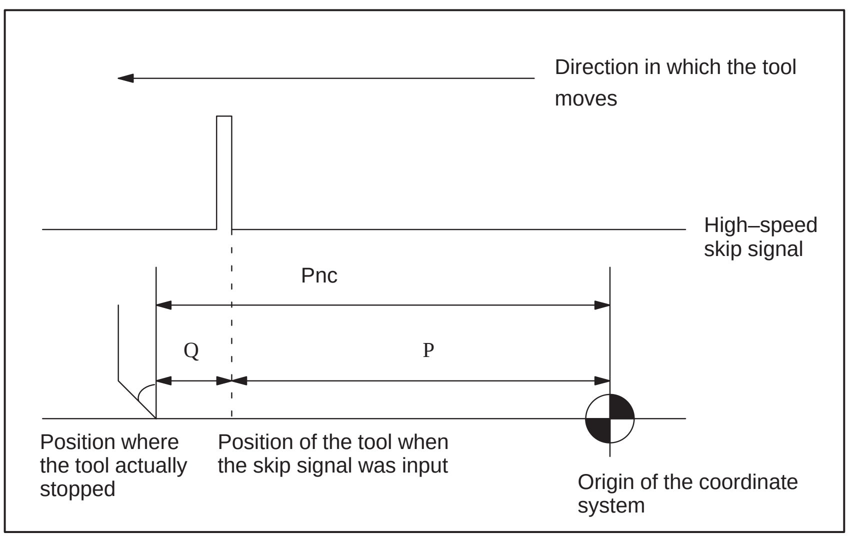
Acceleration / Deceleration and Servo Delay Compensation (Type A/B)

The skip function causes the NC to memorize the “current position” of the tool when a skip signal is input. However, the “current position” includes a delay in the servo system. In other words, the ”current position” deviates by the distance corresponding to the servo delay from the position where the tool actually was when the skip signal was input. This deviation can be calculated from the positional error in the servo system and the number of remaining pulses due to feedrate acceleration/ deceleration in the NC. Taking this deviation into account eliminates the necessity to include the servo delay in a measurement error.

The deviation of the “current position” can be compensated for by either of the following two types, using parameter SEA (bit 0 of parameter No. 6201) or parameter SEB (bit 1 of parameter No. 6201).

(1) Type A: The deviation is calculated from the cutting time constant and the servo time constant (loop gain).

(2) Type B: The deviation is assumed to be a sum of the number of remaining pulses due to acceleration/deceleration caused when the skip signal is turned on, and the positional error.



Pnc: Position where the tool actually stopped after the skip signal was input [mm/inch]

P: Distance to be measured [mm/inch]

Q: Servo delay [mm/inch]

Under the conditions shown above, the NC calculates the following equation using parameter SEA (bit 0 of parameter No. 6201) or SEB (bit 1 of parameter No. 6201):

$$P = Pnc - Q$$

For type A (SEA), the deviation is calculated by:

$$Q = Fm \times 1/60 \times (\alpha \times Tc/1000 + Ts/1000)$$

where

Fm: Feedrate [mm/min or inch/min]

Tc: Cutting time constant [ms]

Parameter No. 1622: Exponential acceleration/deceleration

Parameter No. 1628: Linear acceleration/deceleration after interpolation

If parameter SKF (bit 7 of parameter No. 6200) = 0, Tc = 0.

Ts: Servo time constant [ms]

Assuming that the loop gain (parameter No. 1825) is G (unit: 1/s):

Ts = 1000/G

α: = 1 Exponential acceleration/deceleration

= 1/2 Linear acceleration/deceleration after interpolation

> Note
> 1 For type A (SEA), the skip signal must be turned on when the tool moves at constant feedrate.

Signal

High Speed Skip Staus Signal HDO0AHDO7<F122>

[Classification] Output signal

[Function] This signal informs the PMC of the input status of the high-speed skip signal. The signal-to-bit correspondence is as follows:

High-speed skip signal		Bit name
HDI0	——	HDO0
HDI1	——	HDO1
HDI2	——	HDO2
HDI3	——	HDO3
HDI4	——	HDO4
HDI5	——	HDO5
HDI6	——	HDO6
HDI7	——	HDO7

[Output condition] Each bit is set to 1 when:

- The corresponding high-speed skip signal is logical 1.

Each bit is set to 0 when:

- The corresponding high-speed skip signal is logical 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F122	HDO7	HDO6	HDO5	HDO4	HDO3	HDO2	HDO1	HDO0

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6200		SRE		HSS				

[Data type] Bit

HSS

0 : The skip function does not use high-speed skip signals.
1 : The skip function uses high-speed skip signals.

SRE When a high-speed skip signal is used:
0 : The signal is considered to be input at the rising edge (0 → 1).
1 : The signal is considered to be input at the falling edge (1 → 0).

[Data type] Bit

SEA When a high speed skip signal goes on while the skip function is used, acceleration/deceleration and servo delay are:

	#7	#6	#5	#4	#3	#2	#1	#0
6201				IGX			SEB	SEA

0 : Ignored.
1 : Considered and compensated (type A).

SEB When a high speed skip signal goes on while the skip function is used, acceleration/deceleration and servo delay are:
0 : Ignored.
1 : Considered and compensated (type B).

IGX When the high-speed skip function is used, SKIP (bit 7 of X004), SKIPP (bit 6 of G006), and +MIT1 to –MIT2 (bits 2 to 5 of X004) are:
0 : Enabled as skip signals.
1 : Disabled as skip signals.

Note 1 SKIPP (bit 6 of G006) and +MIT1 to –MIT2 (bits 2 to 5 of X004) are enabled only when bit 0 (GSK) of parameter No. 6200 is set to 1 and bit 3 (MIT) of parameter No. 6200 is set to 1. Note also that these signals are enabled only for the T series.

	#7	#6	#5	#4	#3	#2	#1	#0
6202	1S8	1S7	1S6	1S5	1S4	1S3	1S2	1S1

1S1A1S8

Specify which high-speed skip signal is enabled when the G31 skip command is issued. The bits correspond to the following signals:
0 : The skip signal corresponding to the bit is disabled.
1 : The skip signal corresponding to the bit is enabled.

1S1 —— HDI0
1S2 —— HDI1
1S3 —— HDI2
1S4 —— HDI3
1S5 —— HDI4
1S6 —— HDI5
1S7 —— HDI6
1S8 —— HDI7

Note 1 HDI0 to HDI7 are high-speed skip signals.

Reference item

CONNECTION MANUAL (Hardware) (B–62443E)	7.8	HIGH SPEED DI SIGNAL INTERFACE

14.3.3 Multi–step Skip (T series)

General

In a block specifying P1 to P4 after G31, the multistage skip function stores coordinates in a custom macro variable when a skip signal (8 points) or high-speed skip signal (8 points) is turned on.

Also in a block specifying Q1 to Q4 after G04, the multistage skip function skips a dwell when the skip signal (8 points) or high speed skip signal (8 points) has turned on.

A skip signal from equipment such as a fixed-dimension size measuring instrument can be used to skip programs being executed.

In plunge grinding, for example, a series of operations from rough machining to spark-out can be performed automatically by applying a skip signal each time rough machining, semi-fine machining, fine-machining, or spark-out operation is completed.

Signal

Skip signal SKIP, SKIP2 to SKIP8 <X004#2, #6, #0, #1>

[Classification] Input signal

[Function] These signals terminate skip cutting. That is, the position where a skip signal turns to ”1” in a command program block containing G31P1 (or G31), G31P2, or G31P3, G31P4 is stored in a custom macro variable, and the move command of the block is terminated at the same time. Furthermore, in a block containing G04, G04Q1, G04Q2, G04Q3 or G04Q4, the dwell command of the block is terminated.

In either case, until all other commands (such as miscellaneous functions) of the block are completed, machining never proceeds to the next block.

Which of the eight skip signals is applicable to blocks containing the G codes can be determined by parameter (no. 6202 to 6206). The eight skip signals can correspond to the G codes on a one-to-one basis. One skip signal can also be made applicable to multiple G codes. Conversely, multiple skip signals can be made applicable to one G code.

[Operation] When a skip signal turns to ”1”, the control unit functions as described below.

(1) When a block contains a G code from (G31, G31P1 to P4) for skip cutting, and the skip signal is made applicable by parameter setting to the command, the control unit reads and stores the current position of the specified axis at that time. The control unit stops the axis, then cancels the remaining distance that the block was supposed to be moved.

(2) When a block contains a G04, or G04Q1 to Q4 code for dwell, and the skip signal is made applicable by parameter setting to the command, the control unit stops dwell operation, and cancels any remaining dwell time.

(3) The skip signal is monitored not for a rising edge, but for its state. So, if a skip signal continues to be ”1”, a skip condition is assumed to be satisfied immediately when the next skip cutting or dwell operation is specified.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
X004	SKIP	ESKIP / SKIP6	–MIT2 / SKIP5	+MIT2 / SKIP4	–MIT1 / SKIP3	+MIT1 / SKIP2	ZAE / SKIP8	XAE / SKIP7	(T series)
	SKIP	ESKIP / SKIP6	SKIP5	SKIP4	SKIP3	ZAE / SKIP2	YAE / SKIP8	XAE / SKIP7	(M series)

Notes
1 If the automatic tool compensation option is used, SKIP5 to SKIP8 cannot be used.
2 SKIP2 to SKIP6 are at the same addresses as skip signal ESKIP (axis control by PMC) and axial manual feed interlock signals +MIT1, –MIT1, +MIT2, and –MIT2 (direct input B for tool compensation measurements). Be careful when using both. (T series)
3 SKIP2 and SKIP6 to SKIP8 are at the same addresses as skip signal ESKIP (axis control by PMC) and measurement position arrival signal XAE, YAE, and ZAE (tool length automatic measurement). Be careful when using both. (M series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6200			SLS	HSS				

[Data type] Bit

HSS
0 : The skip function does not use high-speed skip signals.
1 : The skip function uses high-speed skip signals.

SLS
0 : The multi–step skip function does not use high-speed skip signals while skip signals are input.
1 : The multi–step skip function uses high-speed skip signals while skip signals are input.

Note 1 Skip signals (SKIP and SKIP2 to SKIP8) do not depend on the setting of this parameter. They are always enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
6202	1S8	1S7	1S6	1S5	1S4	1S3	1S2	1S1
6203	2S8	2S7	2S6	2S5	2S4	2S3	2S2	2S1
6204	3S8	3S7	3S6	3S5	3S4	3S3	3S2	3S1
6205	4S8	4S7	4S6	4S5	4S4	4S3	4S2	4S1
6206	DS8	DS7	DS6	DS5	DS4	DS3	DS2	DS1

[Data type] Bit

1S1A1S8, 2S1A2S8, 3S1A3S8, 4S1A4S8, DS1ADS8

Specify which skip signal is enabled when the skip command (G31, or G31P1 to G31P4) and the dwell command (G04, G04Q1 to G04Q4) are issued with the multi–step skip function.

The following table shows the correspondence between the bits, input signals, and commands.

The settings of the bits have the following meanings:

0 : The skip signal corresponding to the bit is disabled.
1 : The skip signal corresponding to the bit is enabled.

Multi–step skip function

Command / Input signal	G31 G31P1 G04Q1	G31P2 G04Q2	G31P2 G04Q2	G31P4 G04Q4	G04
SKIP/HDI0	1S1	2S1	3S1	4S1	DS1
SKIP2/HDI1	1S2	2S2	3S2	4S2	DS2
SKIP3/HDI2	1S3	2S3	3S3	4S3	DS3
SKIP4/HDI3	1S4	2S4	3S4	4S4	DS4
SKIP5/HDI4	1S5	2S5	3S5	4S5	DS5
SKIP6/HDI5	1S6	2S6	3S6	4S6	DS6
SKIP7/HDI6	1S7	2S7	3S7	4S7	DS7
SKIP8/HDI7	1S8	2S8	3S8	4S8	DS8

Note 1 HDI0 to HDI7 are high-speed skip signals.

Note

Note 1 The skip cutting commands G31 P1, G31 P2, G31 P3, and G31 P4 are all identical, except that they correspond to different skip signals. The tool moves along the specified axis until the SKIP signal is set to "1" or the end point of the specified movement is reached, while performing linear interpolation. The feedrate is specified in the program. G31 is the same as G31 P1.

Dwell commands G04, G04 Q1, G04 Q2, G04 Q3, and G04 Q4 are also identical, except that they correspond to different skip signals.

When no Qn command follows the G04 command, and when DS1 to DS8, bit 0 to bit 7 of parameter No. 6206, are not set, dwell is not skipped.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–62454E)	II.4.12	Multi–step Skip (G31)
OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.4.12	Multi–step Skip (G31)

14.3.4 Torque Limit Skip (T series)

General

Specifying a move command after G31 P99 (or G31 P98) with a motor torque limit set (for example, specifying a torque limit on the PMC window) allows the same cutting feed as that specified with G01 to be performed.

While the tool is moved with a motor torque limit set during cutting feed, skip is performed when a signal indicating that the motor torque limit has been reached is input as a result of an operation such as pushing something against the motor.

● Basic operations

When the motor torque limit is reached or the SKIP signal <X0004#7> is input during the execution of G31 P99, the execution of the next block starts without executing the remaining portion of the move command.

When the motor torque limit is reached during the execution of G31 P98, the execution of the next block starts without executing the remaining portion of the move command. (The SKIP signal does not affect the execution of G31 P98.)

When no torque limit is specified before executing G31 P99 or P98, the move command is executed without performing the skip operation.

For G31 P99 and P98, the coordinate, indicating the position to which the tool is to be positioned after skip, is stored in the system variable of the custom macro.

Alarm No. 244 occurs if errors have accumulated to an amount (32767) that cannot be corrected in one distribution before the torque–limit–reached signal is input during the execution of G31 P99 or P98.

Signal

Torque limit reached signals TRQL1 to TRQL8 <F114>

[Classification] Output signal

[Function] Indicates that the torque limit has been reached.

[Output condition] Set to “1” when:

· The torque limit has been reached for the corresponding axis.

Set to “0” when:

· The torque limit has not been reached for the corresponding axis

Numbers 1 to 8 indicate the corresponding axis numbers.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F114	TRQL8	TRQL7	TRQL6	TRQL5	TRQL4	TRQL3	TRQL2	TRQL1

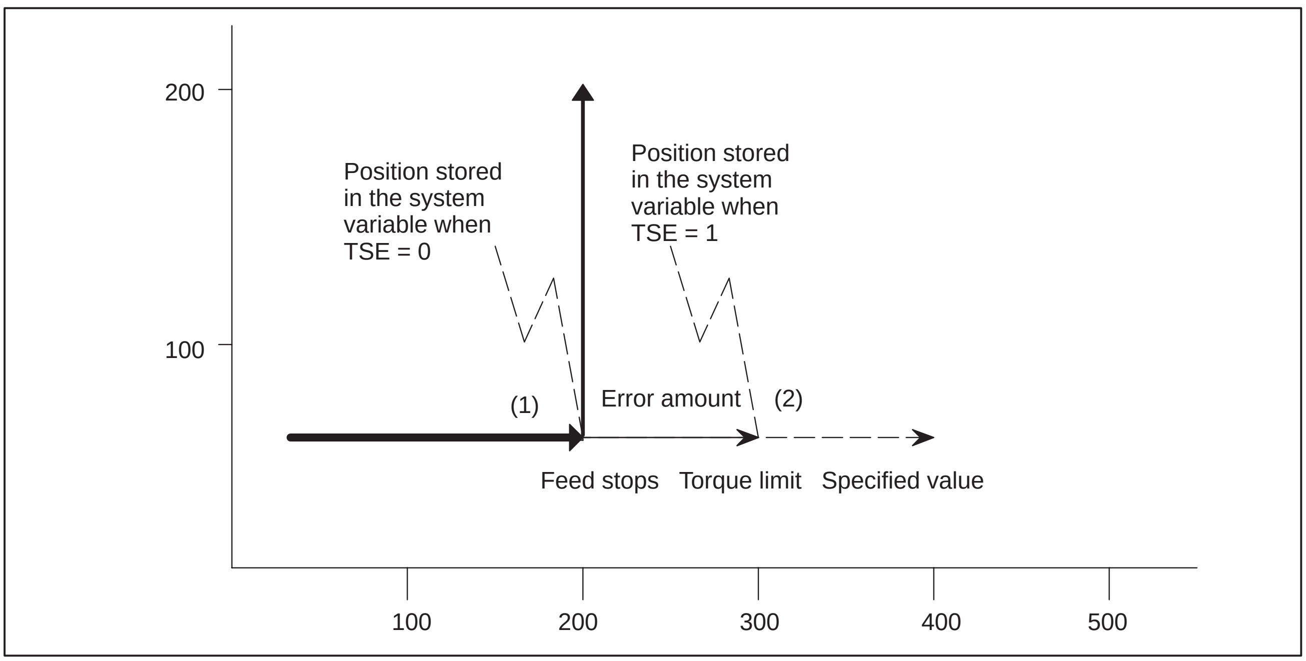
Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6201					TSA	TSE		

[Data type] Bit type

TSE When a skip operation is performed by the G31 P99 or P98 command used to specify torque limit skip:
0 : Corrects servo errors. (1)
1 : Does not correct servo errors. (2)

TSA Selects the axes to be monitored for whether the torque limit has been reached during the execution of the G31 P99 or 98 command used to specify torque limit skip:
0 : Monitors all axes.
1 : Monitors the axes specified by G31 P99 or P98.



Alarm and message

Number	Message	Description
015	TOO MANY AXES COMMANDED	In a block where the skip function activated by the torque–limit reached signal (G31, P99/P98) was specified, either moving the machine along an axis was not specified, or moving the machine along multiple axes was specified. Specify movement only along one axis.
244	P/S ALARM	In the skip function activated by the torque limit signal, the number of accumulated erroneous pulses exceed 32767 before the signal was input. Therefore, the pulses cannot be corrected with one distribution. Change the conditions, such as feedrates along axes and torque limit, and try again.

Note

Note 1 Always specify the torque limit before executing the G31 P99 or P98 command. Otherwise, the move command is executed without performing the skip operation.

Note 2 With the G31 P99 command, the skip operation is also triggered by the SKIP signal. However, do not use high–speed skip.

Note 3 Cancel tool nose radius compensation by using G40 before executing G31 P99 or P98.

Note 4 Specify 0 in SKF, bit 7 of parameter No. 6200, to disable dry run, override, and automatic acceleration/deceleration for skip command G31.

Reference item

OPERATOR'S MANUAL (For Lathe) (B–62444E)	II.4.13	TORQUE LIMIT SKIP

14.4 ENTERING COMPENSATION VALUES

14.4.1 Input of Offset Value Measured A (T series)

General

This is a function of setting an offset value by key-inputting a workpiece diameter manually cut and measured from the MDI keyboard.

First the workpiece is cut in the longitudinal or in the cross direction manually. When the position record signal is turned ”1” (prepare a button on the machine operator’s panel) on completion of the cutting, the workpiece coordinate value of X axis and Z axis at that time is recorded in the CNC. Then, withdraw the tool, stop the spindle, and measure the diameter if the cutting was on the longitudinal direction or measure the distance from the standard face if the cutting was on the facing. (The standard face is made as Z = 0.) When the measured value is entered on the offset value display screen, NC inputs the difference between the input measured value and the coordinate value recorded in NC, as the offset value of the offset number.

If you release the tool without moving the tool in the axis along which an offset value is entered but moves the tool along the other axis, an offset value can be set without using the position record signal.

The workpiece coordinate system can be shifted using the technique of directly inputting the measured value for offset. This technique is used when the coordinate system planned in the program does not match with the coordinate system actually set.

The procedures are the same as those for direct input for offset, except a difference of using the standard tool on the work shift screen.

Signal

Position record signal (PRC) <G040#6>

[Classification] Input signal

[Function] This signal is prepared for the function of input of offset value measured A. It is used to store in the control unit the data on the positions of the tool for tentative cutting. After measuring a dimension of the workpiece, input the measured value by the specified manual operation. The difference is then stored as the specified tool compensation value.

[Operation] The control unit stores the current position along X and Z axes when the signal turns to “1”.

Note
To use this signal, set parameter PRC (No.5005#2) to 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G040		PRC						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5005						PRC		

[Data type] Bit

PRC Direct input of tool offset value and workpiece coordinate-system shift value
0 : Not use a PRC signal
1 : Uses a PRC signal

Reference item

OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.11.4.2	Direct Input of Tool Offset Measured

14.4.2 Input of Tool Offset Value Measured B (T series)

General

When the touch sensor is provided, the tool offset value is automatically settable in the tool offset memory, by moving the tool to make contact with the touch sensor during manual operation. The workpiece coordinate system shift amount is also automatically settable.

- **Touch sensor**

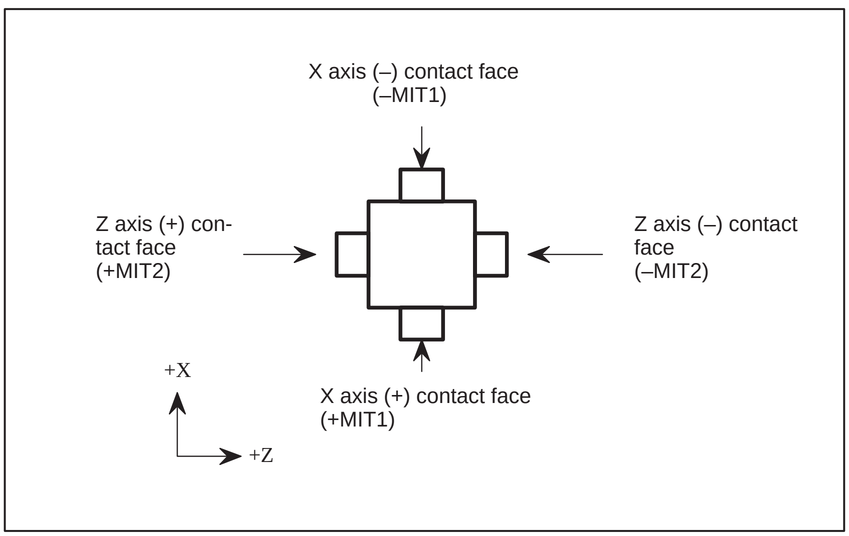
This touch sensor makes contact in two directions in each axis, and outputs four signals when it detects a tool contact. The contact faces are selected according to the tool nose figures to be measured.

a) +MIT1 (+MITX) : Contact to X-axis (+) contact face (Contact in X + direction)

b) –MIT1 (–MITX) : Contact to X-axis (–) contact face (Contact in X – direction)

c) +MIT2 (+MITZ) : Contact to Z-axis (+) contact face (Contact in Z + direction)

d) –MIT2 (–MITZ) : Contact to Z-axis (–) contact face (Contact in Z – direction)



Touch sensor

● Setting tool offset value

Determine a specific point on the machine tool as the measuring reference position. In advance, set the distance from this point to the measuring position (contact face of the touch sensor) as a reference value, using parameter nos. 5015-5018. Select the tool whose offset value is to be measured, and bring it to touch the sensor, receiving a contact detection signal (tool compensation value write signal). The mechanical coordinate value is the distance from the tool nose position of the measuring tool at the mechanical reference (home) position to the measuring position; set the difference between this value and the reference value (parameter setting) into the tool offset value memory as the tool geometry offset value. The corresponding tool wear offset value becomes 0.

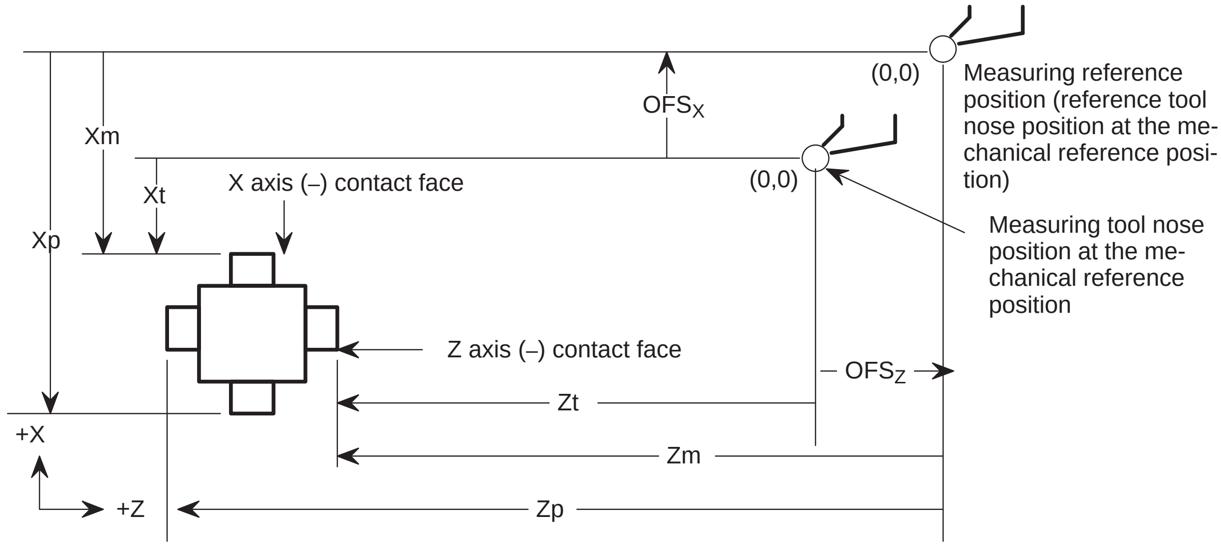
(Tool offset value to be set)

= (Mechanical coordinate value when tool compensation value write signal has become "1") – (Reference value (parameter value) corresponding to the tool compensation value write signal)

The tool offset value to be set differs according to the method of determining the measuring reference position.

Example 1

The difference between the reference tool nose tip position and the measuring tool nose tip position is settable as the tool offset value. Define the reference tool nose tip position at the mechanical reference position (machine zero position) as the measuring reference position, then set the distances Xp, Zp, Xm, Zm, from the measuring reference position to the contact faces of the sensor as parameters.

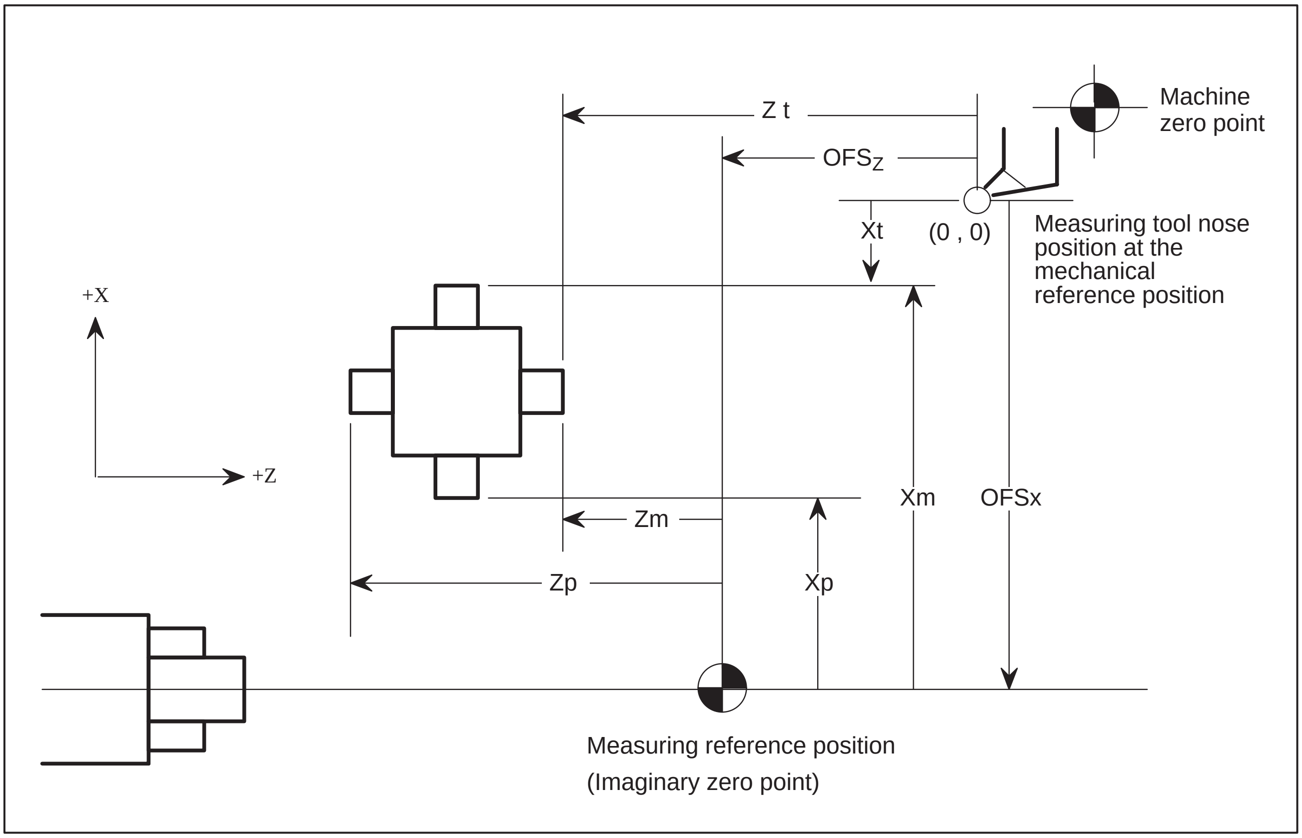


XP	:	Distance from the measuring reference position to X-axis (+) contact face (parameter no. 5015)
Xm	:	Distance from the measuring reference position to X-axis (–) contact face (parameter no. 5016)
Zp	:	Distance from the measuring reference position to Z-axis (+) contact face (parameter no. 5017)
Zm	:	Distance from the measuring reference position to Z-axis (–) contact face (parameter no. 5018)
Xt	:	X-axis direction moving distance of the measuring tool up to the contact face of sensor (X-axis machine coordinate value)
Zt	:	Z-axis direction moving distance of the measuring tool up to the contact face of sensor (Z-axis machine coordinate value)
		(when Xt and Zt touch the X-axis (–) contact face and Z-axis (–) contact face in the above figure)
OFSx	:	Tool offset value to be set (X-axis): OFSx = Xt - Xm
OFSz	:	Tool offset value to be set (Z-axis): OFSz = Zt - Zm

When the reference tool nose tip position is set as the measuring reference position

Example 2

The measuring reference point may be an imaginary point (imaginary point zero), as shown in the figure below. The difference between the imaginary zero point and the measuring tool nose tip position at the mechanical reference point is settable as the tool offset value of the measuring tool, by setting the distances from the imaginary zero point to the respective contact faces as parameters.



When the imaginary zero position is set as the measuring reference position

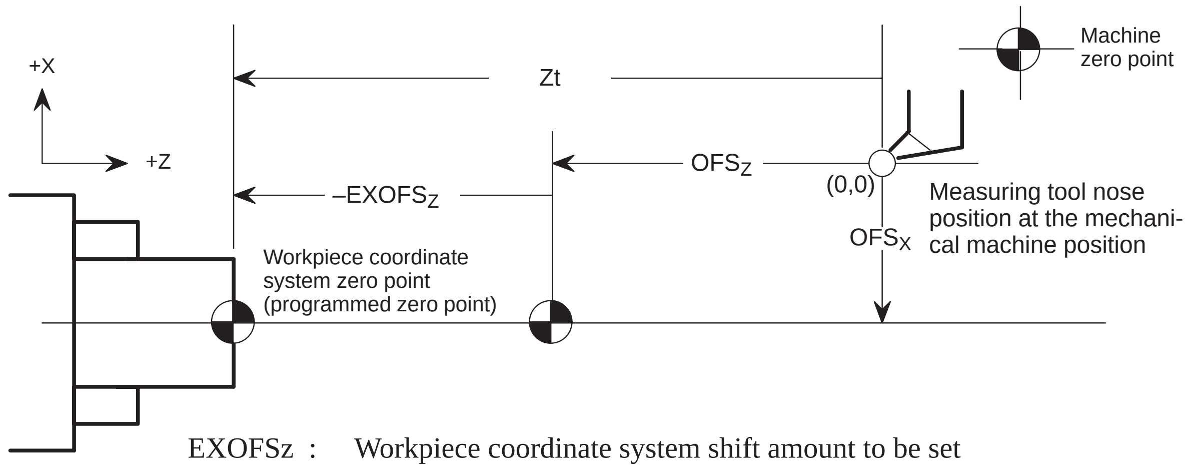
- **Setting the workpiece coordinate system shift amount**

The workpiece coordinate system shift amount for the Z-axis is settable as follows: Bring the tool to touch the workpiece end face. Subtract the tool geometry offset value of the tool (the value shifted in the coordinate system by the tool geometry offset) from the machine coordinate value (the distance from the measuring tool nose tip position at the mechanical reference position (machine zero point) to the workpiece end face).The result is set as the workpiece coordinate system shift value.

(Z axis workpiece coordinate system shift amount to be set (EXOFSz))

= (Z axis tool geometry offset value of the corresponding tool (OFSz)) – (Z axis machine coordinate value(Zt))

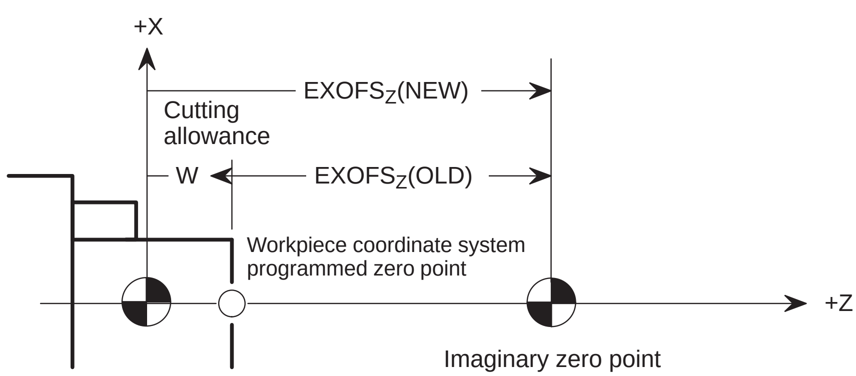
Using the above methods, the workpiece coordinate system is set with the workpiece end face (the contact point of the sensor) specified as the programmed zero point of the workpiece coordinate system of the Z-axis.



EXOFSz	:	Workpiece coordinate system shift amount to be set
OFSz	:	Tool geometry offset value
Zt	:	Mechanical coordinate value (Distance to the workpiece end face)

Setting of workpiece coordinate system shift amount

To deviate the programmed zero point of the workpiece coordinate system from the workpiece end face, such as adding a cutting allowance, use the incremental input of the workpiece coordinate system shift amount in MDI operation. By setting the distance from the programmed zero point to the workpiece end face with a sign, the numeric value input is added to the preset amount.



$EXOFS_Z$(OLD)	:	Workpiece coordinate system shift amount being set by the function of input of tool offset value measured B
W	:	Cutting allowance (incremental input value)
EXOFSZ(NEW)	:	Workpiece coordinate system shift amount after adding the cutting allowance.

Setting of cutting allowance

Basic Procedure to Set Tool Offset Value

1 Execute manual reference position return.
By executing manual reference position return, a machine coordinate system is established.
The tool offset value is computed on the machine coordinate system.

2 Select manual handle mode or manual continuous feed mode and set the tool compensation value write mode select signal GOQSM to ”1”.
The CRT display is automatically changed to the tool offset screen (geometry), and the ”OFST” indicator starts blinking in the status indication area in the bottom of the screen, which informs that the tool compensation value writing mode is ready.

3 Select a tool to be measured.

4 When the cursor does not coincide with the tool offset number desired to be set, move the cursor to the desired offset number by page key and cursor key.
Besides, the cursor can also be coincided with the tool offset number desired to set automatically by the tool offset number input signals (when parameter QNI(No.5005#5)=1).
In this case, the position of the cursor cannot be changed on the tool compensation screen using page keys or cursor keys.

5 Near the tool to the sensor by manual operation.

6 Place the tool edge to a contacting surface of the sensor by manual handle feed.
Bring the tool edge in contact with the sensor. This causes the tool compensation value writing signals (+MIT1, –MIT1, +MIT2 or –MIT2) to input to CNC.
The tool compensation value writing signal is set to ”1”, and the :

- The axis is interlocked in this direction and its feeding is stopped.
- The tool offset value extracted by the tool offset memory (tool geometry offset value) which corresponds to the offset number shown by the cursor is set up.

7 For both X-axis and Z-axis, their offset value are set by the operations **5** and **6**.

8 Repeat operations **3** to **7** for necessary tools.

9 Set the tool compensation value writing mode signal GOQSM to ”0”.
The writing mode is canceled and the blinking ”OFST” indicator light goes off.

Basic Procedure to Set Workpiece Coordinate Shift Value

1 Set the tool geometry offset values of each tool in advance.

2 Execute manual reference position return.
By executing manual reference position return, the machine coordinate system is established.
The workpiece coordinate system shifting amount is computed based on the machine coordinate system of the tool.

3 Set the workpiece coordinate system shifting amount writing mode select signal WOQSM to “1”.
The CRT display is automatically switches to the workpiece shifting screen, the ”WFST” indicator starts blinking at the status indicator area in the bottom of the screen, which inform that the workpiece coordinate system shifting amount writing mode is ready.

4 Select a tool to be measured.

5 Check tool offset numbers.
The tool offset number corresponding to the tool required for measurement, shall be set in the parameter (No.5020) in advance.
Besides the tool offset number can be set automatically by setting the tool offset number input signal (with parameter QNI(No.5005#5)=1).

6 Manually approach the tool to an end face of the workpiece.

7 Place the tool edge to the end face (sensor) of the workpiece by manual handle feed.
When the tool edge contacts with the end face of the workpiece, input the workpiece coordinate system shift amount signal WOSET.
The workpiece coordinate system shifting amount on the Z–axis is automatically set.

8 Release the tool.

9 Set the workpiece coordinate system shift amount write mode select signal WOQSM to ”0”.
The writing mode is canceled and the blinking ”WSFT” indicator light goes off.

Signal

Tool offset write mode select signal (GOQSM <G039#7>)

[Classification] Input signal

[Function] Selects the mode for writing tool compensation.

[Operation] When this signal is turned ”1” in a manual operation mode, the mode for writing tool compensation is selected. The control unit then automatically switches the screen displayed on the CRT to the tool geometry compensation screen and blinks the OFST status display in the bottom of the screen to notify that the mode has been changed to the mode for writing tool compensation.

Tool offset write signal
+MIT1, +MIT2
<X004#2, #4>
–MIT1, –MIT2
<X004#3, #5>

[Classification] Input signal

[Function] Each of these signals inhibits the tool from being fed along the corresponding axis during manual operation. When signal GOQSM for selecting the mode for writing tool compensation is set to ”1”, the manual feed is inhibited and also the tool geometry compensation along the axis is automatically calculated and the result is set in tool compensation memory.

[Operation] When these signals are turned ”1”, the control unit operates as follows:

- Inhibits tools from being fed along the corresponding axis during manual operation.

  +MIT1 : Inhibits the tool from being manually fed in the positive direction along the X-axis.

  –MIT1 : Inhibits the tool from being manually fed in the negative direction along the X-axis.

  +MIT2 : Inhibits the tool from being manually fed in the positive direction along the Z-axis.

  –MIT2 : Inhibits the tool from being manually fed in the negative direction along the Z-axis.

  +MIT3 : Inhibits the tool from being manually fed in the positive direction along the Y-axis.

  –MIT3 : Inhibits the tool from being manually fed in the negative direction along the Y-axis.

- When signal GOQSM for selecting the mode for writing tool compensation is turned ”1”, the manual feed interlock signal also automatically calculates the tool geometry compensation for the tool compensation number pointed to by the cursor and sets the result in tool compensation memory.

Note 1 This signal is used as the manual feed interlock signal in each axis direction.

Tool offset number select signals
OFN0AOFN5, OFN6
<G039#0A#5, G040#0>

[Classification] Input signal

[Function] Selects the tool offset number.

[Operation] When the mode for writing tool compensation is selected, the cursor is automatically positioned on the tool geometry compensation number selected by these signals.
A tool offset number is specifed by 7–bit binary number. Number 0 to 98 corresponds to the compensation number 1 to 99.

Note 1 This signal is available only when parameter QNI (No. 5005#5) =1.

Workpiece coordinate system shift value write mode select signal (WOQSM <G039#6>)

[Classification] Input signal

[Function] Selects the mode for writing the shift amount for the workpiece coordinate system.

[Operation] When this signal is turned ”1” in a manual operation mode, the mode for writing the shift amount for the workpiece coordinate system is selected. The control unit then automatically switches the screen displayed on the CRT to the WORK SHIFT screen and blinks the OFST status display in the bottom of the screen to notify that the mode has been changed to the mode for writing the shift amount for the workpiece coordinate system. However, this is not performed when the mode for writing tool compensation values is selected.

Workpiece coordinate system shift value write signal (WOSET <G040#7>)

[Classification] Input signal

[Function] Automatically calculates and sets the shift amount for the workpiece coordinate system.

[Operation] When this signal turns to ”1” in the mode for writing the shift amount for the workpiece coordinate system, it triggers automatic calculation and setting of the shift amount for the workpiece coordinate system.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X004			–MIT2	+MIT2	–MIT1	+MIT1		
			SKIP5	SKIP4	SKIP3	SKIP2		

Note
Since the same addresses are used for both +MIT1, –MIT1, +MIT2, –MIT2 and skip signals SKIP2 to SKIP5 (multi–step skip), be careful for using two kinds of signals.

	#7	#6	#5	#4	#3	#2	#1	#0
G039	GOQSM	WOQSM	OFN5	OFN4	OFN3	OFN2	OFN1	OFN0

G040	WOSET							OFN6

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3003					DIT			

[Data type] Bit

DIT Interlock for each axis direction
0 : Enabled
1 : Disabled

	#7	#6	#5	#4	#3	#2	#1	#0
5005			QNI					

[Data type] Bit

QNI In the function of input of offset value measured B
0 : Not automatically select the tool offset number
1 : Automatically selects a tool offset number

5015	Distance (XP) betweeen reference position and X axis + contact surface
5016	Distance (XM) betweeen reference position and X axis – contact surface
5017	Distance (ZP) betweeen reference position and Z axis + contact surface
5018	Distance (ZM) betweeen reference position and Z axis – contact surface

[Data type] Two–word

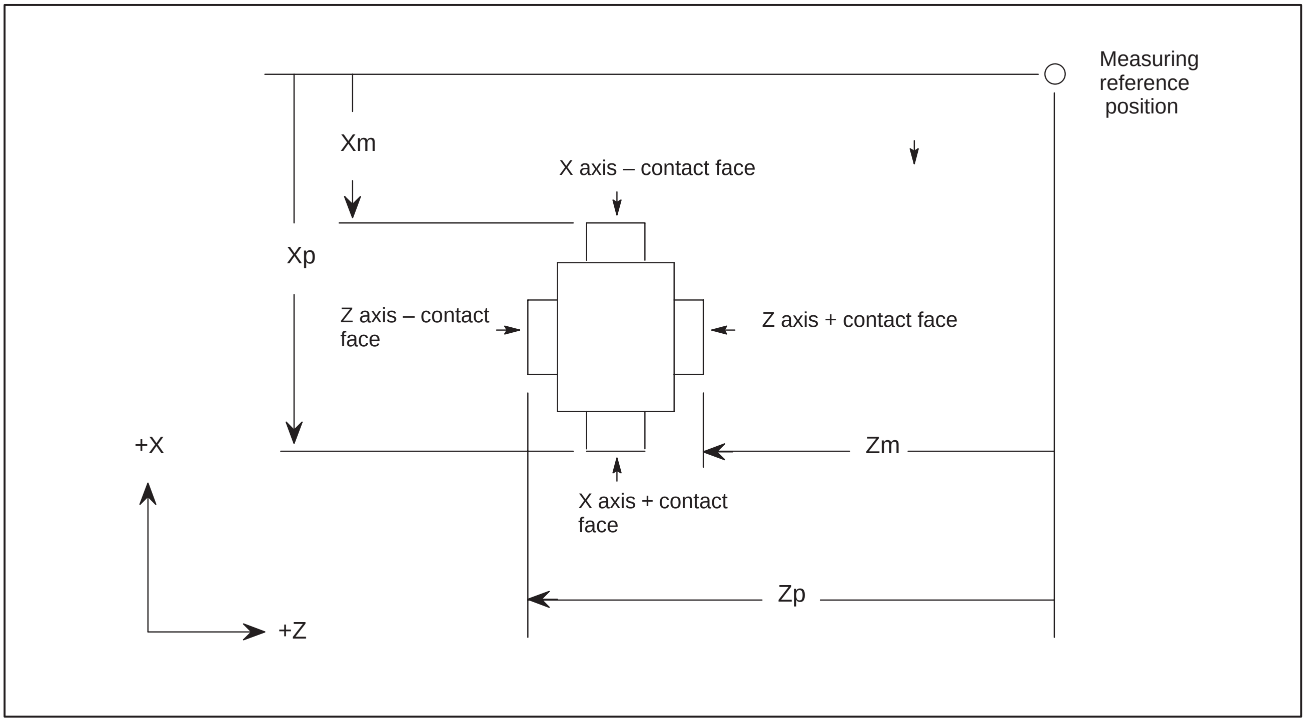
[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] –99999999 – 99999999

These parameters are related to the function of input of tool offset value measured B.

They set the distance (with sign) between the measurement reference position and sensor contact surface. For an axis under diameter programming, set it by a diameter value.



5020	Tool offset number used for the input of tool offset value measured B

[Data type] Byte

[Valid data range] 0 to the number of tools to be compensated.

Set tool offset number used for the input of tool offset value measured B function (i.e. when workpiece coordinate system shift value is set). (The tool offset number corresponding to the measured tool shall be set in advance.) This parameter is valid when the tool offset number is not selected automatically (QNI, #5 of parameter 5005, is zero).

Reference item

OPERATOR'S MANUAL (For Lathe) (B–62444E)	III.11.4.3	Direct Input of Tool Offset Measured B

14.4.3 Direct Input of Workpiece Zero Point Offset Value Measured

General

By directly entering the measured deviation of the actual coordinate system from a programmed work coordinate system, the workpiece zero point offset at the cursor is automatically set so that a command value matches the actual measurement.

15 PMC CONTROL FUNCTION

15.1 PMC AXIS CONTROL/PMC AXIS SPEED CONTROL FUNCTION

General

The PMC can directly control any given axis, independently of the CNC. In other words, moving the tool along axes that are not controlled by the CNC is possible by entering commands, such as those specifying moving distance and feedrate, from the PMC. This enables the control of turrets, pallets, index tables and other peripheral devices using any given axes of the CNC.

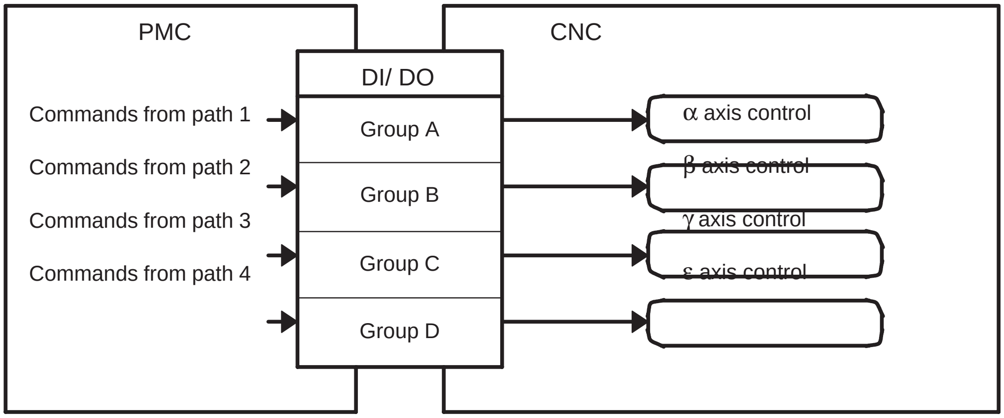
Whether the CNC or PMC controls an axis is determined by the input signal provided for that particular axis.

The PMC can directly control the following operations:

(1) Rapid traverse with moving distance specified
(2) Cutting feed – feed per minute, with moving distance specified
(3) Cutting feed – feed per revolution, with moving distance specified
(4) Skip – feed per minute, with moving distance specified
(5) Dwell
(6) Jog feed
(7) Reference position return
(8) 1st reference position return
(9) 2nd reference position return
(10) 3rd reference position return
(11) 4th reference position return
(12) External pulse synchronization – Main spindle
(13) External pulse synchronization – first manual handle
(14) External pulse synchronization – second manual handle
(15) External pulse synchronization – third manual handle (for M series only)
(16) Feedrate control
(17) Auxiliary function
(18) Selection of the machine coordinate system

The PMC is provided with four paths to control these operations using input and output signals.

By issuing commands through these four paths, the PMC can simultaneously control multiple axes separately. Use parameter No. 8010 to determine which path controls which axis. Commands may be issued through one path to two or more axes, thus allowing the PMC to control multiple axes using one path.



In the following description, input/output signals from the four paths are called group A (path 1), group B (path 2), group C (path 3), and group D (path 4), respectively.

The name of an input/output signal used for PMC axis control always contains a lowercase g, as in EBUFg. However, there is no such signal as EBUFg. The actual signal names represented by EBUFg are EBUFA, EBUFB, EBUFC, and EBUFD, which respectively correspond to signals of group A (path 1), group B (path 2), group C (path 3), and group D (path 4).

Basic procedure

(1) In parameter No. 8010, specify which DI/DO signal group (A, B, C, or D) is to be used for PMC axis control on a per–axis basis.

When using the same group for simultaneously controlling two or more axes, check that the settings of the parameters related to feedrate (rapid traverse rate, acceleration/deceleration time constant, diameter/radius, linear axis/rotation axis, etc.) are identical for each axis to be controlled.

(2) To enable direct PMC axis control, set each control axis selection signal (EAX1 to EAX8), that corresponds to an axis to be controlled, to 1.

(3) Determine the operation.

The axis control command signals (EC0g to EC6g) specify the type of operation. The axis control feedrate signals (EIF0g to EIF15g) specify the feedrate. The axis control data signals (EID0g to EID31g) specify the moving distance and other data.

❍ These signals, together with block stop prohibition signal EMSBKg (described later), determine one complete operation, which is tantamount to one block executed during CNC–controlled automatic operation. These signals may be collectively called the axis control block data signals.

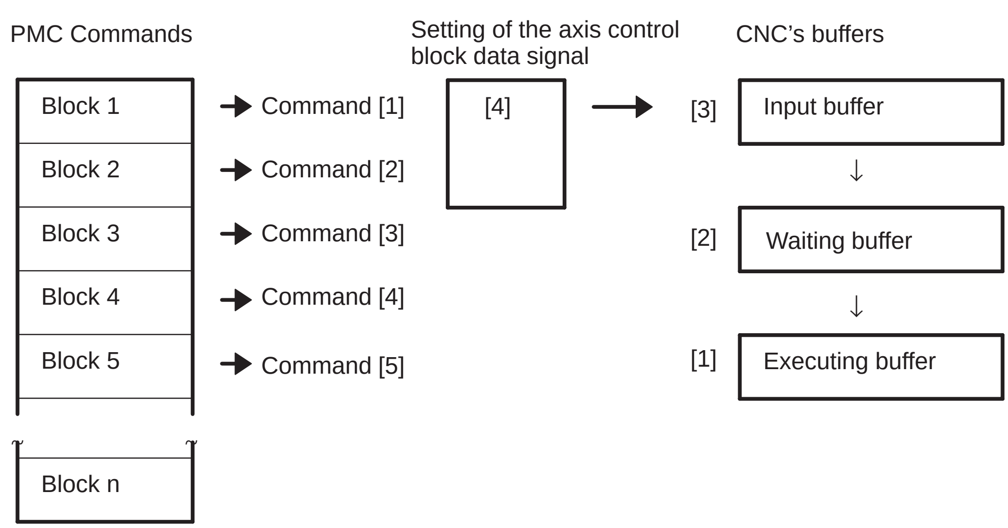
} List of Signals Determining Data, Tantamount to One Block for PMC Axis Control

Generic name	Signal name	Symbol	Data type
Axis control block data signals	Block stop prohibition signal	EMSBKg	Bit
	Axis control command signal	EC0gAEC6g	Byte
	Axis control feedrate signal	EIF0gAEIF15g	Word
	Axis control data signal	EID0gAEID31g	Two words

(4) When the data governing a complete operation (one block) is determined, reverse the logical state of axis control command read signal EBUFg (i.e., from “0” to “1” or vice versa). Note that, for this to occur, axis control command read completion signal EBSYg must be in the same logical state as EBUFg.

- The CNC is capable of storing axis control commands from the PMC in its buffer so that multiple operations can be performed in series, under the control of the PMC. This allows the CNC to accept a new command block from the PMC during the execution of another block if the buffer has free space.

  The following figure illustrates an example in which command [1] is being executed, commands [2] and [3] are stored in the buffers, and command [4] has been issued (the axis control block data signal is set).



When the execution of command [1] is completed:

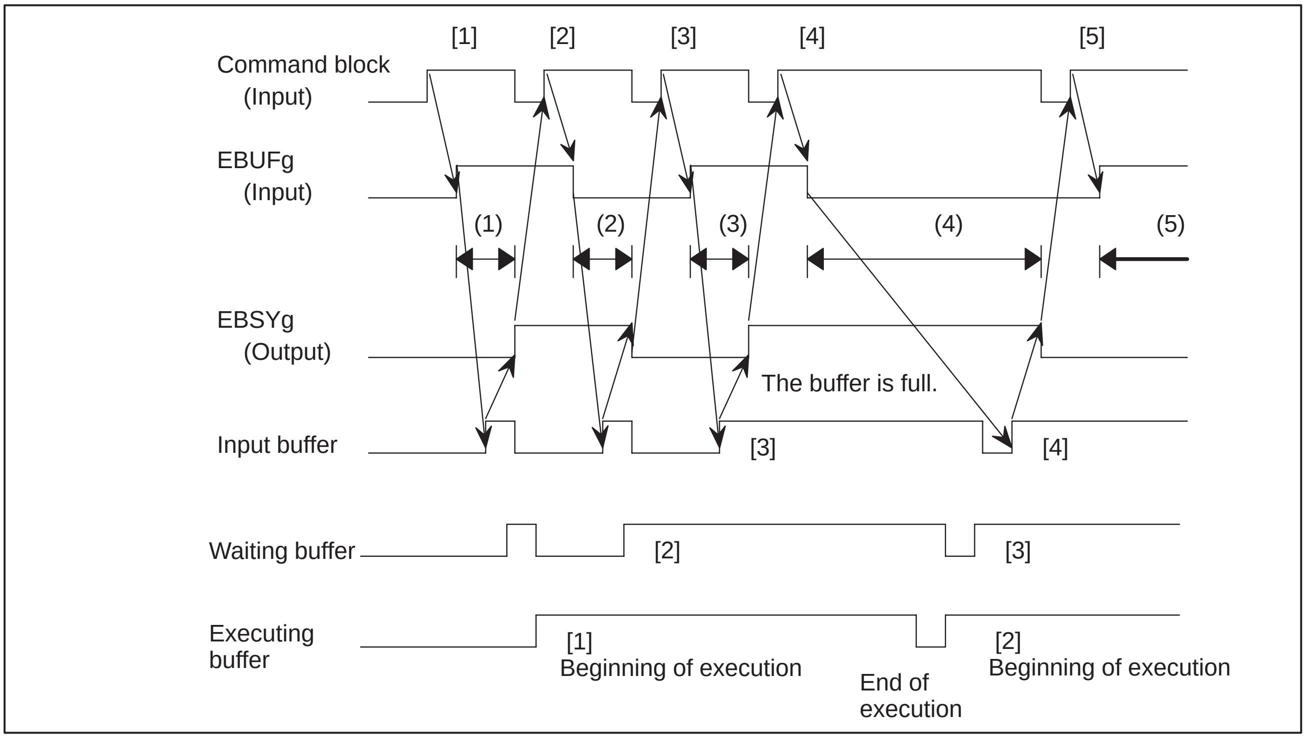
· command [2] is transferred from the waiting buffer to the executing buffer;

· command [3] is transferred from the input buffer to the waiting buffer; and

command [4] is transferred to the input buffer as the command block (axis control block data signal).

Upon the reception of command [4] by the input buffer, the PMC can issue command [5] to the CNC (the axis control block data signal is set).

The timing chart for the command operation is shown below.



(1), (2), (3), (4), (5) : A new block cannot be issued during these intervals (while EBUFg and EBSYg are in different logical states).

- The status of the CNC buffer can be determined by the exclusive OR of axis control command read signal EBUFg, input from the PMC, and axis control command read completion signal EBSYg, output from the CNC.

EBUFg ‖ EBSYg	Exclusive OR (XOR)	CNC buffer status
0 ‖ 0 1 ‖ 1	0	The previous block has already been read into the CNC buffer. The PMC can issue the next block.
0 ‖ 1 1 ‖ 0	1	The previous block has not yet been read completely. It is just being read or waiting for the CNC buffer to become available. Do not issue the next block, nor reverse the logical state of EBUFg. Reversing the EBUFg state invalidates any block that has been already issued.

(5) Repeat steps (3) and (4) until all the blocks have been issued.

When the final block has been issued, set control axis selection signals EAX1 to EAX8 to “0”. Before setting these signals to “0”, however, check that the blocks stored in the CNC’s input, waiting, and executing buffers have all been executed. Setting the signals to “0” while a block is being executed, or while a block remains in any of these buffers, results in the issue of a P/S alarm. This alarm suspends the current block execution and invalidates the blocks stored in the input and waiting buffers.

To ensure no block is being executed, or that there are no blocks remaining in the input or waiting buffer, check that control axis selection status signal *EAXSL is set to “0”.

For those axes that are always subject to PMC control, such as those controlling turrets, pallets, and ATCs, ensure that the EAX1 to EAX8 signals are always set to “1”. There is no need to set these signals to “0” after issuing commands from the PMC to the CNC. When all command blocks have been executed (there are no blocks remaining to be executed), the CNC automatically stops execution.

(6) When control axis selection signals EAX1 to EAX8 are set to “0”, control is returned to the CNC.

Signal

Signal list

No.	Symbol	Signal name
1	EAX1 – EAX8	Control axis selection signals
2	EC0g – EC6g	Axis control command signals
3	EIF0g – EIF15g	Axis control feedrate signals
4	EID0g – EID31g	Axis control data signals
5	EBUFg	Axis control command read signal
6	EBSYg	Axis control command read completion signal
7	ECLRg	Reset signal
8	ESTPg	Axis control temporary stop signal
9	ESBKg	Block stop signal
10	EMSBKg	Block stop disable signal
11	EM11g – EM48g	Auxiliary function code signals
12	EMFg	Auxiliary function strobe signal
13	EFINg	Auxiliary function completion signal
14	ESOFg	Servo–off signal
15	EMBUFg	Buffering disable signal
16	*EAXSL	Control axis selection status signal
17	EINPg	In–position signal
18	ECKZg	Following zero checking signal
19	EIALg	Alarm signal
20	EGENg	Axis moving signal
21	EDENg	Auxiliary function executing signal
22	EOTNg	Negative–direction overtravel signal
23	EOTPg	Positive–direction overtravel signal
24	*FV0E – *FV7E	Feedrate override signals
25	OVCE	Override cancellation signal
26	ROV1E, ROV2E	Rapid traverse override signals
27	DRNE	Dry run signal
28	RTE	Manual rapid traverse selection signal
29	EOV0	Override 0% signal
30	ESKIP	Skip signal
31	EADEN1 – EADEN8	Distribution completion signals
32	EABUFg	Buffer full signal
33	EACNT1 – EACNT8	Controlling signals

Signal Detail

1 Control axis selection signals EAX1 to EAX8

[Classification] Input signal

[Function] When the signal is set to "1", the corresponding axis becomes subject to PMC control.

When the signal is set to "0", PMC control becomes invalid. Changing the setting of the control axis selection signal is possible only when control axis selection status signal *EAXSL is set to "0". Changing the setting when *EAXSL is set to "1" results in the issue of a P/S alarm (No. 139). Alarm signal EIALg is set to "1".

When NCC, bit 5 of parameter No. 8001, is set to "0", a command issued from the CNC is executed while the control axis selection signal is set to "1" and signal *EAXSL is set to "0". When the parameter is set to "1", the same attempt results in the issue of a P/S alarm (No. 139). Note that the command is invalidated when the tool is moving along the axis in manual continuous feed mode.
If the control axis selection signal is set to "1" while the CNC is currently executing a command, a P/S alarm is generated. In manual continuous feed mode, setting this signal to "1" suspends the execution of the command. While *EAXSL is set to "0", the status of alarm signal EIALg does not change to 1 when the control axis selection signal is set to 1 and a P/S alarm (No. 139) is generated. In this case, the axis can be controlled from the PMC, even when the CNC is in the alarm status.

Note 1 After setting control axis selection signals EAX1 to EAX8 to 1, it takes at least 8 msec before the PMC can issue commands to the CNC.

2 Axis control command signals EC0g to EC6g

[Classification] Input signal

[Function] Specifies the following operations through each path.

Axis control command (hexadecimal code)	Operation
00h	Rapid traverse (linear acceleration/deceleration)
	Performs the same operation as G00, used by the CNC.
01h	Cutting feed – feed per minute (exponential acceleration/deceleration or linear acceleration/deceleration after interpolation)
	Performs the same operation as G94 G01, used by the CNC.

Axis control command (hexadecimal code)	Operation
02h	Cutting feed – feed per revolution (exponential acceleration/ deceleration or linear acceleration/deceleration after inter-polation)
	Performs the same operation as G95 G01, used by the CNC.
03h	Skip – feed per minute
	Performs the same operation as G31 G01, used by the CNC.
04h	Dwell
	Performs the same operation as G04, used by the CNC.
05h	Reference position return
	Moves the tool in the direction of reference position return specified by ZMIx, bit 5 of parameter No. 1006, in rapid tra-verse mode, then performs the same operation as manual reference position return, done by the CNC.
06h	Jog feed (exponential acceleration/deceleration)
	Moves the tool in the specified direction in jog feed mode. Performs the same operation as that of JOG feed, done by the CNC.
07h	1st reference position return
	Performs the same operation as done when positioning the tool to the reference position from the intermediate point specified by G28 of the CNC.
08h	2nd reference position return
	Performs the same operation as done when positioning the tool to the reference position from the intermediate point specified by G30 P2 of the CNC.
09h	3rd reference position return
	Performs the same operation as done when positioning the tool to the reference position from the intermediate point specified by G30 P3 of the CNC.
0Ah	4th reference position return
	Performs the same operation as done when positioning the tool to the reference position from the intermediate point specified by G30 P4 of the CNC.
0Bh	External pulse synchronization – main spindle
	Synchronizes with the main spindle.
0Dh	External pulse synchronization – 1st manual handle
	Synchronizes with the 1st manual handle.

Axis control command (hexadecimal code)	Operation
0Eh	External pulse synchronization – 2nd manual handle
	Synchronizes with the second manual handle.
0Fh	External pulse synchronization – 3rd manual handle
	Synchronizes with the 3rd manual handle.
10h	Speed command (linear acceleration/deceleration)
	Performs jog feed at the specified speed.
12h	Auxiliary function
	Performs the same function as the miscellaneous function (M function), used by the CNC.
20h	Machine coordinate system selection
	Performs the same operation as G53, used by the CNC.

Rapid traverse rate

When using the rapid traverse command (EC0g to EC6g: 00h), the feedrate can be specified in either the same parameter as that used by the CNC (No. 1420) or the PMC's axis interface feedrate signals EIF0g to EIF15g. This can be set with RPD, bit 0 of parameter No. 8002.

Cutting feed – feed per revolution

When using the cutting feed – feed per revolution command (EC0g to EC6g: 02h), the optional function for threading in synchronous feed mode is necessary in the case of the M series.

This operation cannot be performed when IT0 to IT2, bit 4 to 6 of parameter No. 7501, specify high–speed cycle machining.

Reference position return

The reference position return command (EC0g to EC6g: 05h) enables the following operation: When DLZ, bit 1 of parameter No. 1002, specifying reference position return without dogs for all axes, or DLZx, bit 1 of parameter No. 1005, specifying reference position return without dogs for each axis, is valid and the tool has not been returned to the reference position since the power was turned on, move each axis in the direction specified by the jog feed command (EC0g to EC6g: 06h) (position the tool to a point near the reference position) and issue the reference position return command (EC0g to EC6g: 05h). This returns the tool to the reference position (positions the tool to the grid nearest the current position) without using the deceleration signal for reference position return.

Note that, when positioning the tool to a point near the reference position, the tool must be moved in the direction of reference position return at such a speed that the servo position error exceeds the value of parameter No. 1836.

The direction of the grid relative to the proximate position depends on ZMIx, bit 5 of parameter No. 1006.

After the reference position has been established, reference position return can be performed at high speed by issuing the reference position return command (EC0g to EC6g: 05h), irrespective of the reference position return direction specified by ZMIx, bit 5 of parameter No. 1006.

Reference position return without dogs

When using the 1st reference position return command (EC0g to EC6g: 07h), if DLZ, bit 1 of parameter No. 1002, specifying reference position return without dogs for all axes, or DLZx, bit 1 of parameter No. 1005, specifying reference position return without dogs on a per–axis basis is valid and the tool has not been returned to the reference position since the power was turned on, issuing the 1st reference position return command (EC0g to EC6g: 07h) results in the issue of a P/S alarm (No. 090).

1st to 4th reference position return

When using the 1st to 4th reference position return commands (EC0g to EC6g: 07h to 0Ah), the feedrate can be specified using RPD, bit 0 of parameter No. 8002, in the same manner as when using the rapid traverse command (EC0g to EC6g: 00h).

Note that, in the case of the 1st reference position return, if the tool has not been manually returned to the reference position after the power was turned on, the feedrate specified by parameter No. 1424 applies.

External pulse synchronization

When using the external pulse synchronization commands (EC0g to EC6g: 0Bh, 0Dh to 0Fh), the tool moves backwards if the external pulse has a negative value. When a manual handle interrupt is executed for the axis to which the external pulse is being applied, the moving distance is the sum of the external pulse and the interrupt pulse.

Display of remaining distance

When using the jog feed command (EC0g to EC6g: 06h) and the external pulse synchronization command (EC0g to EC6g: 0Bh, 0Dh to 0Fh), the displayed remaining distance is always “0”.

Speed command

When using the speed command (EC0g to EC6g: 10h), specify the axis to be controlled as a rotation axis in ROT, bit 0 of parameter No. 1006.

While position control is being executed for the jog feed command (EC0g to EC6g: 06h), the speed command (EC0g to EC6g: 10h) exerts speed control over the servo motor, thus allowing the speed to be dynamically changed during jog feed. This makes this command suitable for driving a rotation tool with a servo motor.

A linear acceleration/deceleration time constant can be set for each axis, using parameter No. 8028.

Note that, while jog feed is being executed by the speed command, no coordinate system values are changed. This will result in the loss of the tool position. Therefore, after jog feed has been completed, always return the tool to the reference position before executing the move command.

Machine coordinate system selection

The machine coordinate system selection command (EC0g to EC6g: 20h) performs absolute positioning to move the tool in rapid traverse to a specified position on the machine coordinate system. This command is used to move the tool to a position specifically defined for the machine, such as the tool exchange position.

This command can be used, irrespective of whether the optional function for workpiece coordinate system setting is provided.

For a rotation axis, short cut rotation can also be specified. When using this command for the T series, cancel the tool offset and the tool nose radius compensation. For the M series, cancel cutter compensation, tool length compensation, and tool offset.

The machine coordinate system must be set before attempting to use this command. After turning on the power, return the tool to the reference position either manually or by using G28. When an absolute position detector is provided, returning the tool to the reference position is not necessary because the tool position will be stored in memory.

The following table shows the correspondence between the axis control commands and their data:

Command block		
Operation	**Axis control code signal EC0g to EC6g**	**Command data**
Rapid traverse	00h	Total moving distance EID0g to EID31g Rapid traverse rate EIF0g to EIF15g The rapid traverse rate is valid when PRD, bit 0 of parameter No. 8002, is set to “1”.
Cutting feed – feed per minute	01h	Total moving distance EID0g to EID31g Feedrate EIF0g to EIF15g
Skip – feed per minute	03h	
Cutting feed – feed per revolution	02h	Total moving distance EID0g to EID31g Feed per rotation EIF0g to EIF15g
Dwell	04h	Dwell time EID0 to EID31g
Reference position return	05h	None
Jog feed	06h	Feed direction EID31g Jog feedrate EIF0g to EIF15g
1st reference position return	07h	Rapid traverse rate EIF0g to EIF15g The rapid traverse rate is valid when PRD, bit 0 of parameter No. 8002, is set to “1”.
2nd reference position return	08h	
3rd reference position return	09h	
4th reference position return	0Ah	

Operation	Axis control code signal EC0g to EC6g	Command data
External pulse synchronization – main spindle	0Bh	Pulse weight EIF0g to EIF15g
External pulse synchronization – manual handle	0Dh	
	0Eh	
	0Fh <For M series only>	
Speed command	10h	Jog feedrate EIF0g to EIF15g
Auxiliary function	12h	Auxiliary function code EID0g to EID15g
Machine coordinate system setting	20h	Machine coordinate system setting (absolute value) EID0g to EID31g Rapid traverse rate EIF0g to EIF15g The rapid traverse rate setting is effective when bit 0 (RPD) of parameter 8002 is set to “1”.

3 Axis control feedrate signals EIF0g to EIF15g

[Classification] Input signal

[Function]

(1) Rapid traverse (EC0g to EC6g: 00h)

(2) 1st reference position return (EC0g to EC6g: 07h)

(3) 2nd reference position return (EC0g to EC6g: 08h)

(4) 3rd reference position return (EC0g to EC6g: 09h)

(5) 4th reference position return (EC0g to EC6g: 0Ah)

(6) Machine coordinate system selection (EC0g to EC6g: 20h)

For these commands, signals EIF0g to EIF15g are used to specify the rapid traverse rate, in binary format, when bit 0 (RPD) of parameter 8002 is set to “1”. For 1st reference position return, however, the rapid traverse rate specified with parameter No. 1424 is used if manual reference position return has not been performed after the power was first turned on.

[Unit of data]

		Data unit		Unit
		IS–B	IS–C	
Linear axis	Metric machine	1		mm/min
	Inch machine	0.1		inch/min
Rotation axis		1		deg/min

[Valid data range] Specify data within the range given in the following table.

		Data range		Unit
		IS–B	IS–C	
Linear axis	Metric machine	30 – 15000	30 – 12000	mm/min
	Inch machine	30 – 6000	30 – 4800	inch/min
Rotation axis		30 – 15000	30 – 12000	deg/min

(7) Cutting feed – feed per minute (EC0g to EC6g: 01h)

(8) Skip – feed per minute (EC0g to EC6g: 03h)

For these commands, the signals are used to specify, in binary format, the feedrate along an axis. The specified feedrate can be magnified by ten by the setting of bit 3 (F10) of parameter No. 8002.

[Unit of data] When bit 3 (F10) of parameter No. 8002 is set to 0

		Data unit		Unit
		IS–B	IS–C	
Linear axis	Metric machine	1	0.1	mm/min
	Inch machine	0.01	0.01	inch/min
Rotation axis		1	0.1	deg/min

When bit 3 (F10) of parameter No. 8002 is set to 1

		Data unit		Unit
		IS–B	IS–C	
Linear axis	Metric machine	10	1	mm/min
	Inch machine	0.1	0.01	inch/min
Rotation axis		10	1	deg/min

[Valid data range] 1 to 65535
(Actual values must fall within the ranges given in the following table.)

		Data range		Unit
		IS–B	IS–C	
Linear axis	Metric machine	1 – 100000	0.1 – 12000.0	mm/min
	Inch machine	0.01 – 4000.00	0.001 – 480.000	inch/min
Rotation axis		1 – 100000	0.1 – 12000.0	deg/min

Note 1 When “0” is specified, the CNC continues to perform buffering without moving the tool. In such a case, release the buffering by issuing reset signal ECLRg.

Note 2 Cutting feedrate clamp is disabled.

(9) Cutting feed – feed per rotation (EC0g to EC6g: 02h)

For this command, the signals are used to specify the amount by which the tool is moved for every rotation of the spindle.

<For T series>

[Unit of data] The data increment depends on the settings of bits 6 (FR1) and 7 (FR2) of parameter No. 8002, as listed in the following table.

Parameter		Metric input (mm/rev)	Inch input (inch/rev)	Rotation axis (deg/rev)
FR2	FR1			
1	1	0.0001	0.000001	0.0001
0	0			
0	1	0.001	0.00001	0.001
1	0	0.01	0.0001	0.01

[Valid data range] 1 to 65535
(Actual values must fall within the ranges given in the following table.)

		Data range		Unit
		IS–B	IS–C	
Linear axis	Metric input	0.0001 – 500.0000		mm/rev
	Inch input	0.000001 – 9.999999		inch/rev
Rotation axis		0.0001 – 500.0000		deg/rev

<For M series>

[Unit of data] The data unit depends on the settings of bits 6 (FR1) and 7 (FR2) of parameter No. 8002, as listed in the following table.

Parameter		Metric input (mm/rev)	Inch input (inch/rev)	Rotation axis (deg/rev)
FR2	FR1			
1	1	0.01	0.0001	0.01
0	0			
0	1	0.1	0.001	0.1
1	0	1	0.01	1

[Valid data range] 1 to 65535
(Actual values must fall within the ranges given in the following table.)

		Data range		Unit
		IS–B	IS–C	
Linear axis	Metric input	0.01 – 500.00		mm/rev
	Inch input	0.0001 – 9.9999		inch/rev
Rotation axis		0.01 – 500.00		deg/rev

Note 1 The specified feedrate can be magnified by 1, 10, or 100 by setting bits 6 (FR1) and 7 (FR2) of parameter No. 8002 accordingly.

Note 2 The value of parameter No. 8022 is used as the upper limit for clamping the feedrate.

Note 3 Override for the feedrate is effective. Dry run is invalid.

(10) External pulse synchronization – main spindle
(EC0g to EC6g: 0Bh)

(11) External pulse synchronization – first manual handle
(EC0g to EC6g: 0Dh)

(12) External pulse synchronization – second manual handle
(EC0g to EC6g: 0Eh)

(13) External pulse synchronization – third manual handle
(EC0g to EC6g: 0Fh)

For these commands, the signals are used to specify the weight of the external pulses. A weight range of ±1/256 to ±127 can be set by using signals EIF0g to EIF7g for the figures after the decimal point. When a negative weight is specified, the tool is moved in the reverse direction. When a new pulse weight is specified while the tool is moving in synchronization with external pulses, inverting signal EBUFg causes the tool to move with the new pulse weight.

As commands for (10) to (13) are executed without buffering, axis control command read completion signal EBSYg usually need not be checked.

Note 1 The pulse weight is clamped according to the value set for parameter No. 1424 (parameter for the manual rapid traverse rate for each axis).

Note 2 When bit 7 (NDI) of parameter No. 8004 is set to diameter programming, the tool moves with double pulse weight. In this case, set bit 3 (DIAx) of parameter No. 1006 to 1 and the tool moves in accordance with a command like in a radius programming.

(14) Jog feed (EC0g to EC6g: 06h)

Set the feedrate as the same as for cutting feed – feed per minute (EC0g to EC6g: 01h). The feedrate can be changed during jog feed.

Specify the feedrate with signals EIF0g to EIF15g, and invert the axis control command read signal EBUFg during jog feed, then the tool moves at the new feedrate.

As commands for jog feed are executed without buffering, axis control command read completion signal EBSYg usually need not be checked.

The specified feedrate can be magnified by 10 by setting bit 3 (F10) of parameter No. 8002, and by 200 by setting bit 2 (JFM) of parameter No. 8004.

} Maximum feedrate (with override of 254%)

	IS–B		IS–C	
	Metric input	**Inch input**	**Metric input**	**Inch input**
Magnified by 1	166458mm/min	1664.58inch/min	16645mm/min	166.45inch/min
Magnified by 10	1664589mm/min	16645.89 inch/min	166458mm/min	1664.58inch/min
Magnified by 200 (Note 3)	1966050mm/min	1966605.00 inch/min	196605mm/min	19660.50 inch/min

Note 1 Magnification of 200 is valid only for the jog feed command (EC0g to EC6g: 06h).

Note 2 The actual speed may not be displayed correctly, depending on the feedrate.

Note 3 The maximum feedrate depends on whether override is applied or canceled. The following table lists the maximum feedrate when override is canceled.

	IS–B		IS–C	
	Metric input	**Inch input**	**Metric input**	**Inch input**
Magnified by 1	65535mm/min	655.35inch/min	6553mm/min	65.53inch/min
Magnified by 10	655350mm/min	6553.50inch/min	65535mm/min	655.35inch/min
Magnified by 200	13107000 mm/min	131070.00 inch/min	1310700mm/min	13107.00 inch/min

(15) Speed command (EC0g to EC6g: 10h)

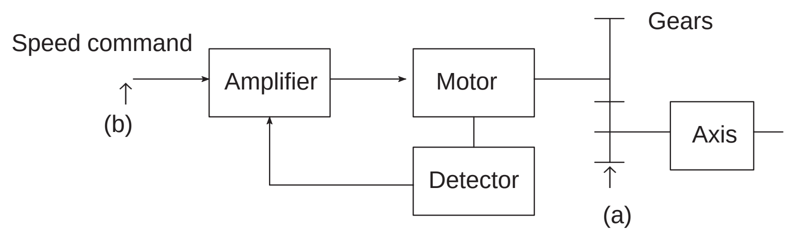
For this command, the signals are used to specify, in binary format, the servo motor speed.

Specify a positive value for rotation in the forward direction. Specify a negative value (twos complement) for rotation in the reverse direction.

When a new servo motor speed is specified, inverting the axis control command read signal EBUFg accelerates or decelerates the servo motor until it attains the new speed.

Data range	Unit
–32768A+32767	rpm

Note 1 The servo motor speed may contain a slight error, as follows:



(a) The speed command for PMC axis control requires specification of the servo motor speed, not the feedrate along an axis. To specify a feedrate along the axis when gears are used to link the servo motor and axis, the feedrate must be converted to a rotation speed of the servo motor speed. As the motor speed must be specified with an integer, the converted speed is subject to a round–off error.

(b) The minimum increment for specifying the motor speed is calculated by the following formula and rounded to the nearest integer:

$$\text{Fmin} = \frac{P \times 2}{15} \times \frac{1}{1000}$$

Fmin : Minimum increment for the motor speed

P : Number of pulses per rotation of the detector for velocity feedback

Specify the speed command using the value calculated by the following formula:

$$F = \frac{N \times P \times 2}{15} \times \frac{1}{1000}$$

F : Speed command value (integer)

N : Servo motor speed (rpm)

P : Number of detector pulses issued per rotation for velocity feedback

Note 2 In speed command mode, the speed after acceleration/deceleration is specified to the servo control unit. The loop gain for position control is invalid.

4 Axis control data signals EID0g to EID31g

[Classification] Input signal

[Function]

[Unit of data]

	IS–B	IS–C	Unit
Metric input Degree input	0.001	0.0001	mm deg
Inch input	0.0001	0.00001	inch

[Valid data range]

(1) Rapid traverse (EC0g to EC6g: 00h)

(2) Cutting feed – feed per minute (EC0g to EC6g: 01h)

(3) Cutting feed – feed per rotation (EC0g to EC6g: 02h)

(4) Skip – feed per minute (EC0g to EC6g: 03h)

For these commands, signals EID0g to EID31g are used to specify, in binary format, the incremental moving distance, according to the input increment used for the axis.

	IS–B	IS–C	Unit
Metric input Degree input	±99999.999	±9999.9999	mm deg
Inch input	±9999.9999	±999.99999	inch

Bit 7 (NDI) of parameter No. 8004 can be set such that the tool is moved by the double moving distance in diameter mode, specified with bit 3 (DIAx) of parameter No. 1006. In this way, the tool is moved in the same way as in radius mode.

(5) Dwell (EC0g to EC6g: 04h)

For this command, the signals are used to specify, in binary format, the dwell time.

Data range	Unit
1A9999999	ms

When diamete programming is used by bit 3 (DIAx) of parameter No. 1006, dwell is executed double the specified time. If Bit 7 (NDI) of parameter No. 8004 is set to 1, the tool dwells as specified by a command value as in radius programming.

Note 1 When the increment system IS–C is used, the least input increment for the dwell time can be set to 0.1 ms, according to the setting of bit 1 (DWE) of parameter No. 8002.

(6) Jog feed (EC0g to EC6g: 06h)

For this command, signal EID31g is used to specify the direction of jog feed, as follows:

0: Positive direction

1: Negative direction

Signals EID0g to EID30g are undefined.

(7) Auxiliary functions (EC0g to EC6g: 12h)

For this command, the signals are used to specify, in binary format, an auxiliary function code to be sent to the PMC. The auxiliary function code can be specified using either one or two bytes, depending on the setting of bit 6 (AUX) of parameter No. 8001, in signals EID0g to EID15g.

(8) Machine coordinate system selection (EC0g to EC6g: 20h)

For this command, the signals are used to specify, in binary format, an absolute machine coordinate, according to the increment system used by the axis.

Example: For absolute value 10000

Input increment	inch	1.0000	mm	10.000
Output increment	mm	25.400	inch	0.3937

The direction of rotation can be specified about a rotation axis with a parameter. To enable the roll–over function, set bit 0 (ROAx) of parameter No. 1008 to 1. Then, select whether the tool is to be rotated in the direction corresponding to the sign of the specified value, or in whichever direction minimizes the distance to the end point, using bit 1 (RABx) of parameter No. 1008. The moving distance per rotation must be set in parameter No. 1260.

5 Axis control command read signal EBUFg

[Classification] Input signal

[Function] Directs the CNC to read a block of command data for PMC axis control. See “Basic procedure” for details of the operation performed when this signal is set from “0” to “1” or from “1” to “0”.

6 Axis control command read completion signal EBSYg

[Classification] Output signal

[Function] Notifies the system that the CNC has read a block of command data for PMC axis control and has stored the block in the input buffer. See "Basic procedure" for details of the output conditions and the procedure.

7 Reset signal ECLRg

[Classification] Input signal

[Function] Resets the corresponding PMC–controlled axis.
When this signal is set to "1", the following is performed:

(1) When the tool is moving along the axis: Decelerates and stops the tool.

(2) When the tool is dwelling: Stops the operation.

(3) When an auxiliary function is being executed: Stops the operation.

Simultaneously, all buffered commands are canceled. Any control command is ignored while this signal is set to "1".

The jog feed command (EC0g to EC6g: 06h) and external pulse synchronization command (EC0g to EC6g: 0Bh, 0Dh to 0Fh) can be terminated by setting reset signal ECLRg to "1". When these commands are terminated, the servo motor decelerates and stops, the axis moving signal EGENg is set to "0", and the control axis selection status signal *EAXSL is set to "0". Confirm that the control axis selection status signal *EAXSL has been set to "0" before issuing the next command. Do not set reset signal ECLRg to "0" until the control axis selection status signal *EAXSL has been set to "0".

The speed command (EC0g to EC6g: 10h) can also be terminated by setting the reset signal ECLRg to "1". When this command is terminated, the servo motor decelerates and stops, and the axis moving signal EGENg is set to "0". Confirm that the axis moving signal EGENg has been set to "0" before issuing the next command. Do not attempt to set the reset signal ECLRg to "0" until the axis moving signal EGENg has been set to "0".

8 Axis control temporary stop signal ESTPg

[Classification] Input signal

[Function] When this signal is set to "1", the following is performed:

(1) When the tool is moving along the axis: Decelerates and stops the tool.

(2) When the tool is dwelling: Stops the operation.

(3) When an auxiliary function is being executed: Stops the operation when auxiliary function completion signal EFINg is input.

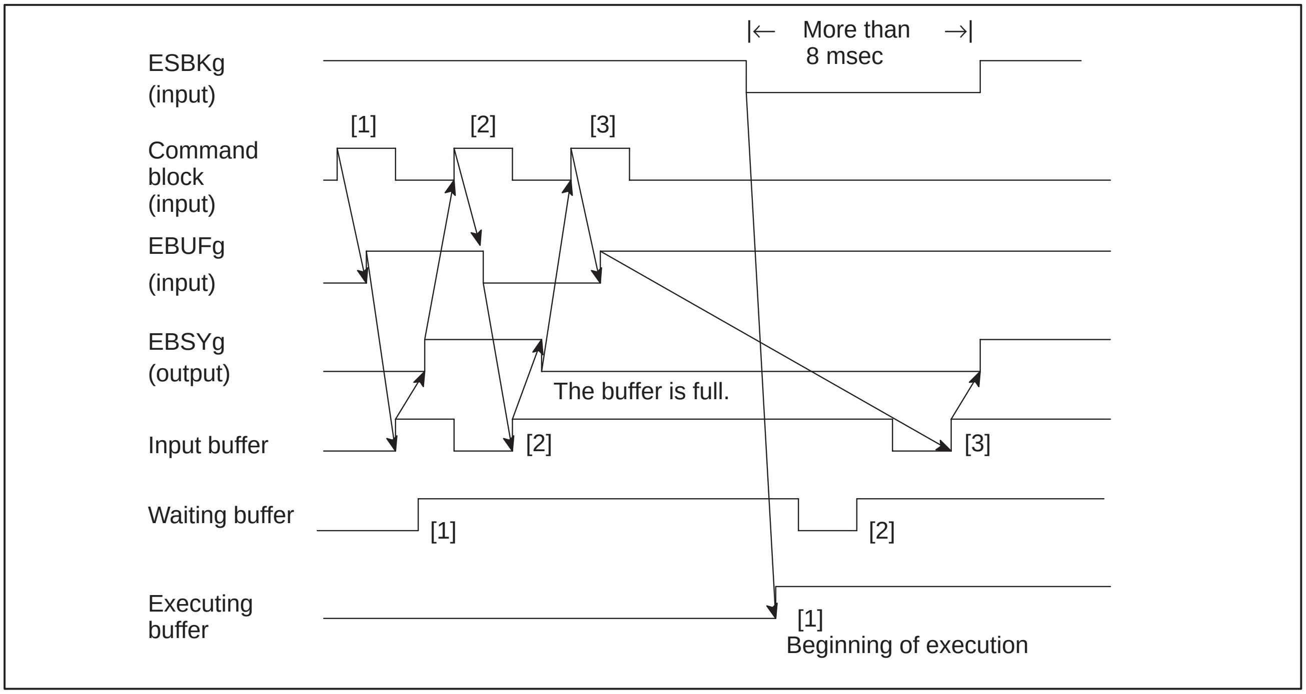
The stopped operation can be restarted by setting this signal to "0".

9 Block stop signal ESBKg
10 Block stop disable signal EMSBKg

[Classification] Input signal

[Function] When block stop signal ESBKg is set to “1” during the execution of a command issued from the PMC, axis control is stopped after the block being executed is completed. When this signal is set to “0”, the buffered command is executed. Block stop signal ESBKg is disabled when block stop disable signal EMSBKg is set to “1” for the block.

The timing chart for the command operation is shown below.



11 Auxiliary function code signals EM11g to EM48g

[Classification] Output signal

12 Auxiliary function strobe signal EMFg

[Classification] Output signal

13 Auxiliary function completion signal EFINg

[Classification] Input signal

[Function] When an auxiliary function command (EC0g to EC6g: 12h) is issued by the PMC, the auxiliary function code is specified in a byte (using signals EID0g to EID7g) or two bytes (using signals EID0g to EID15g), depending on the setting of bit 6 (AUX) of parameter No. 8001.

The CNC sends the auxiliary function code specified in signals EID0g to EID7g (and EID8g to EID15g) to auxiliary function code signals EM11g to EM28g (and EM31g to EM48g) and awaits auxiliary function completion signal EFINg. When the auxiliary function completion signal EFINg is returned, the CNC starts executing the next block.
The timings for sending the auxiliary function code signals and auxiliary function strobe signal, as well as for receiving the auxiliary function completion signal, are the same as those for the miscellaneous functions (M functions) under the control of the CNC. See “Auxiliary function executing signal” for details.

14 Servo–off signal ESOFg

[Classification] Input signal

[Function] When this signal is set to “1”, the servo motor for the corresponding PMC–controlled axis is turned off (servo–off state).

When this signal is set to “0”, the servo motor is turned on.

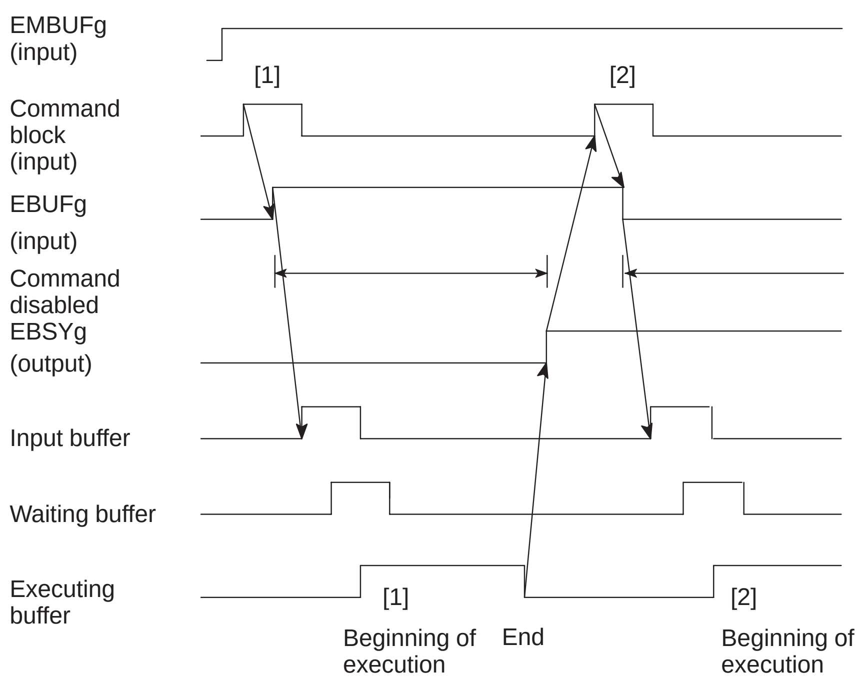
15 Buffering disable signal EMBUFg

[Classification] Input signal

[Function] When this signal is set to “1”, commands from the PMC are not read while the executing, waiting, or input buffer contains a block. If this signal is set to “1” when any of these buffers contain a block, that block is executed but subsequent commands are read only when the buffers are all empty.

To discriminate the buffering disabled condition, the CNC outputs the axis control command read completion signal (EBSYg) only when a command is read when all buffers are empty.

The timing chart for the command operation is shown below.



Buffering is disabled, regardless of buffering disable signal EMBUFg, for the following commands:

(1) Skip–feed per minute (EC0g to EC6g: 03h)

(2) Reference position return (EC0g to EC6g: 05h)

(3) 1st reference position return (EC0g to EC6g: 07h)

(4) 2nd reference position return (EC0g to EC6g: 08h)

(5) 3rd reference position return (EC0g to EC6g: 09h)

(6) 4th reference position return (EC0g to EC6g: 0Ah)

(7) Machine coordinate system selection (EC0g to EC6g: 20h)

The following commands, for which reset signal ECLRg is used for termination, operate as though buffering had been disabled. That is, the subsequently specified block is not executed but canceled:

(1) Jog feed (EC0g to EC6g: 06h)

(2) External pulse synchronization – main spindle (EC0g to EC6g: 0Bh)

(3) External pulse synchronization – first manual handle (EC0g to EC6g: 0Dh)

(4) External pulse synchronization – second manual handle (EC0g to EC6g: 0Eh)

(5) External pulse synchronization – third manual handle (EC0g to EC6g: 0Fh)

(6) Speed command (EC0g to EC6g: 10h)

16 Control axis selection status signal *EAXSL

[Classification] Output signal

[Function] When this signal is set to “0”, control axis selection signals EAX1 to EAX8 can be changed.

This signal is set to 1 in the following cases:

(1) When the tool is moving along a PMC–controlled axis

(2) When a block is being read into a buffer

(3) When the servo–off signal ESOFg is set to “1”

When this signal is set to “1”, control axis selection signals EAX1 to EAX8 cannot be changed. Any attempt to change these signals results in the output of P/S alarm No. 139.

If an attempt to change signals EAX1 to EAX8 is made when servo–off signal ESOFg is “1”, P/S alarm No. 139 occurs and cannot be released simply by setting reset signal ECLRg to “1”. In such a case, restore signals EAX1 to EAX8 or set servo–off signal ESOFg to “0” before setting reset signal ECLRg to “1”.

When a command is issued for any of the four paths with PMC axis control, signal *EAXSL is set to “1” to disable axis selection. Thus, changing signals EAX1 to EAX8 results in the output of P/S alarm No. 139. For paths for which commands are not issued, however, axis selection is enabled if bit 5 (DSL) of parameter No. 8004 is set accordingly.

17 In–position signal EINPg

[Classification] Output signal

[Function] This signal is set to “1” when the corresponding PMC–controlled axis is in the in–position state.

When the tool is decelerated, in–position check is performed to disable the next command until the tool enters the in–position area. The in–position check, however, can be skipped using bit 6 (NCI) of parameter No. 8004 to reduce the cycle time.

When the axis fed at a very low speed, in –position signal might turn “1”.

18 Following zero checking signal ECKZg

[Classification] Output signal

[Function] This signal is set to “1” when following zero check or in–position check is being performed for the corresponding PMC–controlled axis.

19 Alarm signal EIALg

[Classification] Output signal

[Function] This signal is set to “1” when a servo alarm, overtravel alarm, or P/S alarm No. 130 or 139 occurs for the corresponding PMC–controlled axis. This signal is set to “0” when reset signal ECLRg is set to “1” after the alarm is released, as described below.

- Servo alarm

  Eliminate the cause of the alarm, then reset the CNC.

- Overtravel alarm

  Move the tool into the area within the stored stroke limit, then reset the CNC.

  The following commands can be used to move the tool into the area within the stored stroke limit during an overtravel alarm:

(1) Rapid traverse (EC0g to EC6g: 00h)

(2) Cutting feed–feed per minute (EC0g to EC6g: 01h)

(3) Cutting feed–feed per rotation (EC0g to EC6g: 02h)

(4) Jog feed (EC0g to EC6g: 06h)

(5) External pulse synchronization – first manual handle (EC0g to EC6g: 0Dh)

(6) External pulse synchronization – second manual handle (EC0g to EC6g: 0Eh)

(7) External pulse synchronization – third manual handle (EC0g to EC6g: 0Fh)

- P/S alarm (130 or 139)

  Reset the CNC. See “Alarms and messages” for details.

Reset signal ECLRg cannot be used to reset the CNC in the above cases. Use the reset button on the setting panel, external reset signal ERS, or emergency stop signal *ESP.

20 Axis moving signal EGENg

[Classification] Output signal

[Function] This signal is set to “1” when the tool is moving along the corresponding PMC–controlled axis according to commands such as rapid traverse (EC0g to EC6g: 00h) and cutting feed (EC0g to EC6g: 01h).

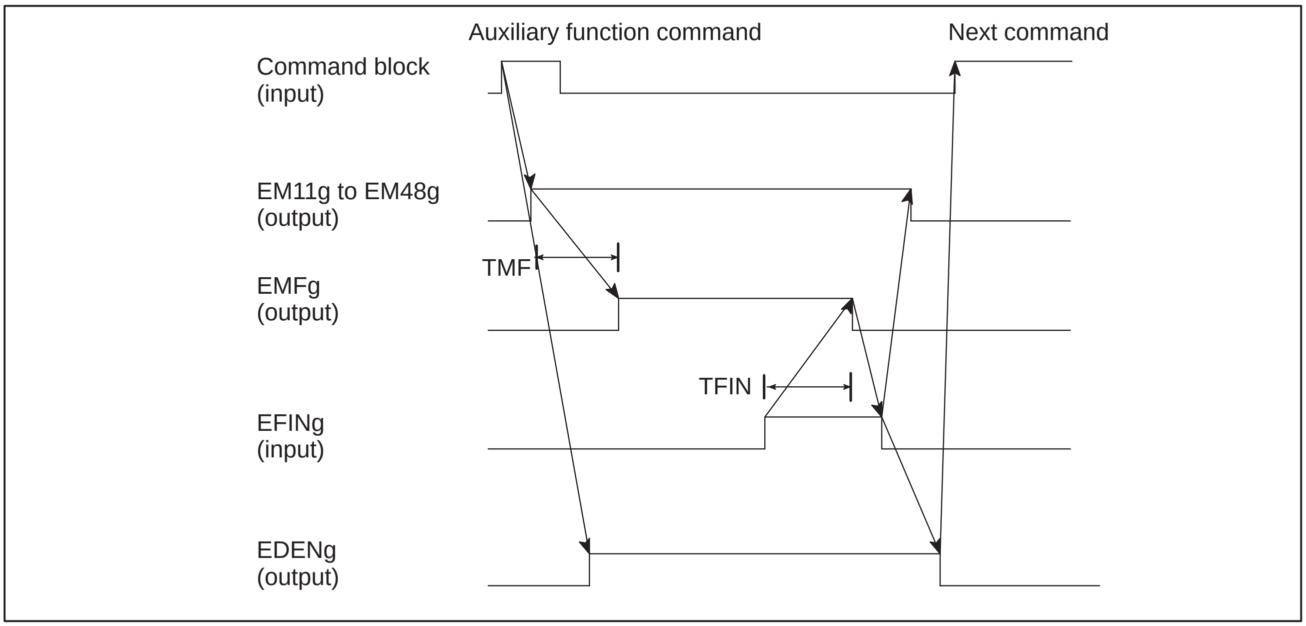
Note 1 This signal is set to “0” when distribution for the axis is completed (the signal is set to “0” during deceleration).

21 Auxiliary function executing signal EDENg

[Classification] Output signal

[Function] When an auxiliary function (EC0g to EC6g: 12h) is specified by the PMC, this signal is set to “1” during the period from when auxiliary function codes EID0g to EID15g are sent to auxiliary function code signals EM11g to EM48g until auxiliary function completion signal EFINg is returned.

The timing chart for the command operation is shown below.



TMF and TFIN are set with parameters 3010 and 3011.

22 Negative–direction overtravel signal EOTNg
23 Positive–direction overtravel signal EOTPg

[Classification] Output signal

[Function] These signals are set to “1” when an overtravel alarm is detected. When the stroke limit in the negative direction is exceeded, signal EOTNg is set to “1”. When the stroke limit in the positive direction is exceeded, signal EOTPg is set to “1”. Simultaneously, alarm signal EIALg is set to “1”.

These signals are set to “0” when the overtravel alarm is released and reset signal ECLRg is set to “1”. See “Alarm signal EIALg” for details of how to release an overtravel alarm.

24 Feedrate override signals *FV0E to *FV7E

[Classification] Input signal

[Function] Like the CNC's feedrate override signals *FV0 to *FV7, these signals can be used to select the override for the cutting feedrate, in steps of 1% from 0 to 254%, independently of the CNC using bit 2 (OVE) of parameter No. 8001.

These signals form an eight–bit binary code and correspond to the override value as follows:

$$\text{Override value} = \sum_{i=0}^{7} |2^i \times Vi|\%$$

Vi = 0 when signal *FViE is 1
Vi = 1 when signal *FViE is 0

That is, each signal has the following significance:

*FV7E = 128%, *FV3E = 8% ,
*FV6E = 64%, *FV2E = 4% ,
*FV5E = 32%, *FV1E = 2% ,
*FV4E = 16%, *FV0E = 1%

When all signals are set to "0", the override is regarded as being 0%, as well as when all signals are "1".

25 Override cancellation signal OVCE

[Classification] Input signal

[Function] When override is enabled, independently of the CNC, by setting bit 2 (OVE) of parameter No. 8001, setting this signal to "1" fixes the cutting feed override to 100%. This signal does not affect the rapid traverse override.

26 Rapid traverse override signals ROV1E and ROV2E

[Classification] Input signal

[Function] These signals can be used to select the override for the rapid traverse rate, independently of the CNC, by setting bit 2 (OVE) of parameter No. 8001.

Rapid traverse override signals		Override value
ROV2E	ROV1E	
0	0	100%
0	1	50%
1	0	25%
1	1	F0

F0 is the minimum feedrate specified with parameter No. 1421.

27 Dry run signal DRNE
28 Manual rapid traverse selection signal RTE

[Classification] Input signal

[Function] These signals can be used to perform dry run or manual rapid traverse, independently of the CNC, by setting bit 2 (OVE) of parameter No. 8001. When dry run signal DRNE is set to "1", the specified rapid traverse rate and cutting feedrate are ignored and the tool moves at the dry run speed (set in parameter No. 1410) multiplied by the specified override. Bit 3 (RDE) of parameter No. 8001 can be used to specify whether to enable or disable dry run for rapid traverse.

When manual rapid traverse selection signal RTE is set to "1" during dry run, the tool moves at the rapid traverse rate for rapid traverse and at the maximum jog feedrate for cutting feed. When the signal is set to "0", the tool moves at the jog feedrate. When dry run signal DRNE is set to "0", the specified rapid traverse rate or cutting feedrate is restored.

Manual rapid traverse select signal	Command from PMC	
	Rapid traverse	Feed
1	Rapid traverse rate	Maximum jog feedrate
0	Jog feedrate(*)	Jog feedrate

* Can also be set to the rapid traverse rate with bit 3 (RDE) of parameter No. 8001.

29 Override 0% signal EOV0

[Classification] Output signal

[Function] This signal is set to “1” when the feedrate override is 0%.

30 Skip signal ESKIP

[Classification] Input signal

[Function] When this signal is set to “1”, the block being executed is immediately stopped and the next block is executed. Bit 7 (SKE) of parameter No. 8001 can be used to select whether to use signal SKIP, which is the common skip signal for the PMC and CNC, or PMC–specific skip signal ESKIP.

31 Distribution completion signals EADEN1 to EADEN8

[Classification] Output signal

[Function] These signals are set to “0” when the tool is moving with a command from the PMC. The signals are set to “1” when the tool is not moving, except when it is stopped by the issue of an axis control temporary stop signal ESTPg during the execution of a move command.

32 Buffer full signal EABUFg

[Classification] Output signal

[Function] This signal is set to “1” when the input buffer contains a command block.

33 Controlling signals EACNT1 to EACNT8

[Classification] Output signal

[Function] When the control axis selection status signal *EAXSL is set to “1”, signal EACNTn corresponding to the axis being controlled is set to “1”.

Signal address

MT→CNC

ADDRESS

	#7	#6	#5	#4	#3	#2	#1	#0
X004	SKIP	ESKIP						

PMC→CNC

ADDRESS

	#7	#6	#5	#4	#3	#2	#1	#0
G136	EAX8	EAX7	EAX6	EAX5	EAX4	EAX3	EAX2	EAX1
G150	DRNE	RTE	OVCE				ROV2E	ROV1E
G151	*FV7E	*FV6E	*FV5E	*FV4E	*FV3E	*FV2E	*FV1E	*FV0E

For group A

	#7	#6	#5	#4	#3	#2	#1	#0
G142	EBUFA	ECLRA	ESTPA	ESOFA	ESBKA	EMBUFA		EFINA
G143	EMSBKA	EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A
G144	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G145	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A
G146	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G147	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G148	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A
G149	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A

For group B

	#7	#6	#5	#4	#3	#2	#1	#0
G154	EBUFB	ECLRB	ESTPB	ESOFB	ESBKB	EMBUFB		EFINB
G155	EMSBKB	EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B
G156	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B
G157	EIF15B	EIF14B	EIF13B	EIF12B	EIF11B	EIF10B	EIF9B	EIF8B
G158	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G159	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G160	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G161	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B

For group C

	#7	#6	#5	#4	#3	#2	#1	#0
G166	EBUFC	ECLRC	ESTPC	ESOFC	ESBKC	EMBUFC		EFINC
G167	EMSBKC	EC6C	EC5C	EC4C	EC3C	EC2C	EC1C	EC0C
G168	EIF7C	EIF6C	EIF5C	EIF4C	EIF3C	EIF2C	EIF1C	EIF0C
G169	EIF15C	EIF14C	EIF13C	EIF12C	EIF11C	EIF10C	EIF9C	EIF8C
G170	EID7C	EID6C	EID5C	EID4C	EID3C	EID2C	EID1C	EID0C
G171	EID15C	EID14C	EID13C	EID12C	EID11C	EID10C	EID9C	EID8C
G172	EID23C	EID22C	EID21C	EID20C	EID19C	EID18C	EID17C	EID16C
G173	EID31C	EID30C	EID29C	EID28C	EID27C	EID26C	EID25C	EID24C

For group D

	#7	#6	#5	#4	#3	#2	#1	#0
G178	EBUFD	ECLRD	ESTPD	ESOFD	ESBKD	EMBUFD		EFIND
G179	EMSBKD	EC6D	EC5D	EC4D	EC3D	EC2D	EC1D	EC0D
G180	EIF7D	EIF6D	EIF5D	EIF4D	EIF3D	EIF2D	EIF1D	EIF0D
G181	EIF15D	EIF14D	EIF13D	EIF12D	EIF11D	EIF10D	EIF9D	EIF8D
G182	EID7D	EID6D	EID5D	EID4D	EID3D	EID2D	EID1D	EID0D
G183	EID15D	EID14D	EID13D	EID12D	EID11D	EID10D	EID9D	EID8D
G184	EID23D	EID22D	EID21D	EID20D	EID19D	EID18D	EID17D	EID16D
G185	EID31D	EID30D	EID29D	EID28D	EID27D	EID26D	EID25D	EID24D

CNC→PMC

ADDRESS

	Address	#7	#6	#5	#4	#3	#2	#1	#0
	F112	EADEN8	EADEN7	EADEN6	EADEN5	EADEN4	EADEN3	EADEN2	EADEN1
	F129	*EAXSL		EOV0					
For group A	F130	EBSYA	EOTNA	EOTPA	EGENA	EDENA	EIALA	ECKZA	EINPA
For group A	F131							EABUFA	EMFA
For group A	F132	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A
For group B	F133	EBSYB	EOTNB	EOTPB	EGENB	EDENB	EIALB	ECKZB	EINPB
For group B	F134							EABUFB	EMFB
For group B	F135	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B
For group C	F136	EBSYC	EOTNC	EOTPC	EGENC	EDENC	EIALC	ECKZC	EINPC
For group C	F137							EABUFC	EMFC
For group C	F138	EM28C	EM24C	EM22C	EM21C	EM18C	EM14C	EM12C	EM11C
For group D	F139	EBSYD	EOTND	EOTPD	EGEND	EDEND	EIALD	ECKZD	EINPD
For group D	F140							EABUFD	EMFD
For group D	F141	EM28D	EM24D	EM22D	EM21D	EM18D	EM14D	EM12D	EM11D
Group A	F142	EM48A	EM44A	EM42A	EM41A	EM38A	EM34A	EM32A	EM31A
Group B	F145	EM48B	EM44B	EM42B	EM41B	EM38B	EM34B	EM32B	EM31B
Group C	F148	EM48C	EM44C	EM42C	EM41C	EM38C	EM34C	EM32C	EM31C
Group D	F151	EM48D	EM44D	EM42D	EM41D	EM38D	EM34D	EM32D	EM31D

	#7	#6	#5	#4	#3	#2	#1	#0
F182	EACNT8	EACNT7	EACNT6	EACNT5	EACNT4	EACNT3	EACNT2	EACNT1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3105							PCF	

[Data type] Bit

PCF Whether movement along PMC–controlled axes is included in the actual speed display
0 : Included
1 : Not included

Note 1 This parameter is valid when the same axis is controlled alternately by the CNC and PMC.

	#7	#6	#5	#4	#3	#2	#1	#0
8001	SKE	AUX	NCC		RDE	OVE		MLE

[Data type] Bit

MLE Whether machine lock signal MLK is valid for PMC–controlled axes
0 : Valid
1 : Invalid

Note 1 Each–axis machine lock signals MLK1 to MLK8 are always valid, regardless of the setting of this parameter.

OVE Signals related to dry run and override used in PMC axis control
0: Same signals as those used for the CNC
(1) Feedrate override signals *FV0 to *FV7
(2) Override cancellation signal OVC
(3) Rapid traverse override signals ROV1 and ROV2
(4) Dry run signal DRN
(5) Rapid traverse selection signal RT
1: Signals specific to the PMC
(1) Feedrate override signals *FV0E to *FV7E
(2) Override cancellation signal OVCE
(3) Rapid traverse override signals ROV1E and ROV2E
(4) Dry run signal DRNE
(5) Rapid traverse selection signal RTE

RDE Whether dry run is valid for rapid traverse in PMC axis control
0 : Invalid
1 : Valid

NCC When a travel command is issued for a PMC–controlled axis (selected by a controlled–axis selection signal) according to the program:
0 : P/S alarm 139 is issued while the PMC controls the axis with an axis control command. While the PMC does not control the axis, a CNC command is enabled.
1 : P/S alarm 139 is issued unconditionally.

AUX The number of bytes for the code of an auxiliary function (12H) command to be output is
0 : 1 (0 to 255)

1 : 2 (0 to 65535)

SKE Skip signal during axis control by the PMC
0 : Uses the same signal SKIP (X004#7) as CNC.
1 : Uses dedicated axis control signal ESKIP (X004#6) used by the PMC.

	#7	#6	#5	#4	#3	#2	#1	#0
8002	FR2	FR1	PF2	PF1	F10	SUE	DWE	RPD

[Data type] Bit

RPD Rapid traverse rate for PMC–controlled axes
0 : Feedrate specified with parameter No. 1420
1 : Feedrate specified with the feedrate data in an axis control command

DWE Minimum time which can be specified in a dwell command in PMC axis control when the increment system is IS–C
0 : 1 ms
1 : 0.1 ms

SUE Whether acceleration/deceleration is performed for an axis that is synchronized with external pulses, for external pulse synchronization commands in PMC axis control
0 : Performed (exponential acceleration/deceleration)
1 : Not performed

F10 Least increment for the feedrate for cutting feed (per minute) in PMC axis control

F10	Metric input	Inch input
0	1 mm/min	0.01 inch/min
1	10 mm/min	0.1 inch/min

PF1, PF2 Set the the feedrate unit of feed per minute in PMC axis control

PF2	PF1	Feedrate unit
0	0	1/1
0	1	1/10
1	0	1/100
1	1	1/1000

FR1, FR2 Set the feedrate unit for feed per rotation for an axis controlled by the PMC.

<table>
<tr><th>FR2</th><th>FR1</th><th>Metric input</th><th>Inch input</th></tr>
<tr><td>0</td><td>0</td><td rowspan="2">0.0001 mm/rev</td><td rowspan="2">0.000001 inch/rev</td></tr>
<tr><td>1</td><td>1</td></tr>
<tr><td>0</td><td>1</td><td>0.001 mm/rev</td><td>0.00001 inch/rev</td></tr>
<tr><td>1</td><td>0</td><td>0.01 mm/rev</td><td>0.0001 inch/rev</td></tr>
</table>

	#7	#6	#5	#4	#3	#2	#1	#0
8003								PIM

[Data type] Bit

PIM If a linear axis is controlled solely by the PMC, the commands for that axis are:
0 : Affected by inch/metric input.
1 : Not affected by inch/metric input.

	#7	#6	#5	#4	#3	#2	#1	#0
8004	NDI	NCI	DSL	G8R	G8C	JFM	NMT	CMV

[Data type] Bit

CMV If the PMC issues a command for an axis after the completion of a movement along that axis according to a command issued by the CNC, but before receiving the signal indicating that the miscellaneous function is completed in the same block,
0 : P/S alarm No. 130 occurs.
1 : The axis is handled as a PMC axis and the specified movement is executed.

NMT If the CNC issues a command that does not result in any movement along a PMC–controlled axis while another command, specified for the axis, is being processed,
0 : P/S alarm No. 130 occurs.
1 : No alarm occurs.

JFM Specifies the units used to specify the feedrate for jog feed (06H) for a PMC–controlled axis.

Increment system	JFM	Metric input	Inch input
IS–B	0	1 mm/min	0.01 inch/min
	1	200 mm/min	2.00 inch/min
IS–C	0	0.1 mm/min	0.001 inch/min
	1	20 mm/min	0.200 inch/min

G8C If a cutting feed command is specified for a PMC–controlled axis, look–ahead control is:
0 : Disabled.
1 : Enabled.

Note 1 The above setting is effective only when the NAHx bit (bit 7 of parameter No. 1819) is set to 0.

G8R If a rapid traverse command is specified for a PMC–controlled axis, look–ahead control is:
0 : Disabled.
1 : Enabled.

Note 1 The above setting is effective only when the NAHx bit (bit 7 of parameter No. 1819) is set to 0.

DSL If an axis exchange is attempted while the selection of a PMC–controlled axis is inhibited,

0 : The attempt fails and a P/S alarm No. 139 occurs.
1 : The axis exchange is executed for the axes that belong to an unspecified path.

NCI During deceleration along an axis controlled by the PMC, the in–position check is:
0 : Performed.
1 : Not performed.

NDI If diameter programming is used for a PMC–controlled axis, the data is specified in:
0 : Radius.
1 : Diameter.

Note 1 The above setting is effective only when the DIAx bit (bit 3 of parameter No. 1006) is set to 1.

8010	Selection of the DI/DO group for each axis controlled by the PMC

[Data type] Byte axis

[Valid data range] 1 to 4

Specify the DI/DO group to be used to specify a command for each PMC–controlled axis.

Value	Description
1	DI/DO group A (G142 to G153) is used.
2	DI/DO group B (G154 to G165) is used.
3	DI/DO group C (G166 to G177) is used.
4	DI/DO group D (G178 to G189) is used.

Note 1 If another value is specified, the axis is not PMC–controlled.

8022	Maximum feedrate for feed per rotation along a PMC–controlled axis

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Specify the maximum feedrate for feed per rotation along a PMC–controlled axis

8028	Linear acceleration/deceleration time constant for jog feed specified by the speed command for each PMC–controlled axis

[Data type] Word axis

[Unit of data] ms/1000 rpm

[Valid data range] 0 to +32767

Specify, for each PMC–controlled axis, the time needed to increase or decrease the speed of the servo motor by 1000 rpm, that is, the time constant of linear acceleration/deceleration for jog feed according to the speed specified for that axis.

Note 1 If 0 is specified, the system does not control the acceleration/deceleration.

Alarm and message

A servo alarm or overtravel alarm for a PMC–controlled axis is detected in the same way as an alarm for a CNC–controlled axis.

If an alarm occurs, the alarm is handled by applying the normal procedure, alarm signal EIALg being set to “1” to inform the PMC of the alarm.

(If an overtravel alarm occurs, either negative overtravel signal EOTNg or positive overtravel signal EOTPg is also set to “1”.)

If the PMC issues a command for a CNC–controlled axis, a P/S alarm No. 130 occurs.

Commands issued by the PMC are effective if the axis is in feed hold or single block stop mode. The command results in the issue of an alarm if cutting feed is executed with an override of 0%, or if the interlock is enabled.

If the CNC issues a command for a PMC–controlled axis, a P/S alarm No. 130 occurs.

If the PMC issues a movement command for an axis in the plane of polar coordinate interpolation in polar coordinate interpolation mode (G12.1), a P/S alarm No. 130 occurs.

Number	Message	Description
0130	ILLEGAL AXIS OPERATION	An axis control command was given by PMC to an axis controlled by CNC. Or an axis control command was given by CNC to an axis controlled by PMC. Modify the program.
0139	CAN NOT CHANGE PMC CONTROL AXIS	A PMC controlled axis was again selected. Or, the CNC issued a control command for an axis that has been set as a PMC–controlled axis for which no command has been specified. Or, an axis under control of CNC was selected by PMC.

Note

Note 1 The mode selection, CNC reset, and other CNC statuses have no effect.

Note 2 Feed hold, single block stop, reset, or interlock of one or all axes, performed by the CNC, does not affect a PMC–controlled axis. Similar control is possible by using the equivalent signals issued from the PMC.

Note 3 Emergency stop or machine lock is enabled. Machine lock can be disabled if the MLE bit (bit 0 of parameter No. 8001) is specified accordingly. However, machine lock for an individual axis is always enabled.

Note 4 The mirror image functions (setting, parameter, input signal) are disabled.

Note 5 In consecutive cutting feed blocks, a new block starts its operation without waiting for the following zero of the servo acceleration/deceleration. In other than the above blocks, a new block starts its operation after the following zero of the servo acceleration/deceleration is confirmed.

Note 6 The actual speed excluding the effect of the movement along a PMC–controlled axis can be displayed if the PCF bit (bit 1 of parameter No. 3105) is specified.

Note 7 Commands for a linear axis that is controlled solely by the PMC (not used as an axis controlled by the CNC) are not affected by inch/metric input if the PIM bit (bit 0 of parameter No. 8003) is specified accordingly. The current position display, also, is not affected by inch/metric input.

Note 8 For a PMC–controlled axis, manual absolute mode is always set. If the PMC starts control of an axis after manual intervention (manual continuous feed, manual handle feed, etc.) is performed during automatic operation while manual absolute mode is not set (*ABSM is set to 1), manual absolute mode is set.

Note 9 If an absolute pulse coder is used, a specified reference position is retained in memory, even after the power is turned off.

Note 10 If the index table indexing function of the M series is added, the PMC cannot control the fourth axis.

15.2 EXTERNAL DATA INPUT

General

The following signals are used to send data from the PMC to the NC.

Signal name	Signal code
Data signal for external data input (input)	ED0 to ED15
Address signal for external data input (input)	EA0 to EA6
Read signal for external data input (input)	ESTB
Read completion signal for external data input (output)	EREND
Search completion signal for external data input (output)	ESEND

The basic external data input procedure is described below:

(1) The PMC sets address signals EA0 to EA6 that indicate the data type and data signals ED0 to ED15.

(2) The PMC sets read signal ESTB to 1.

(3) When the ESTB signal is set to 1, the control unit reads the address.

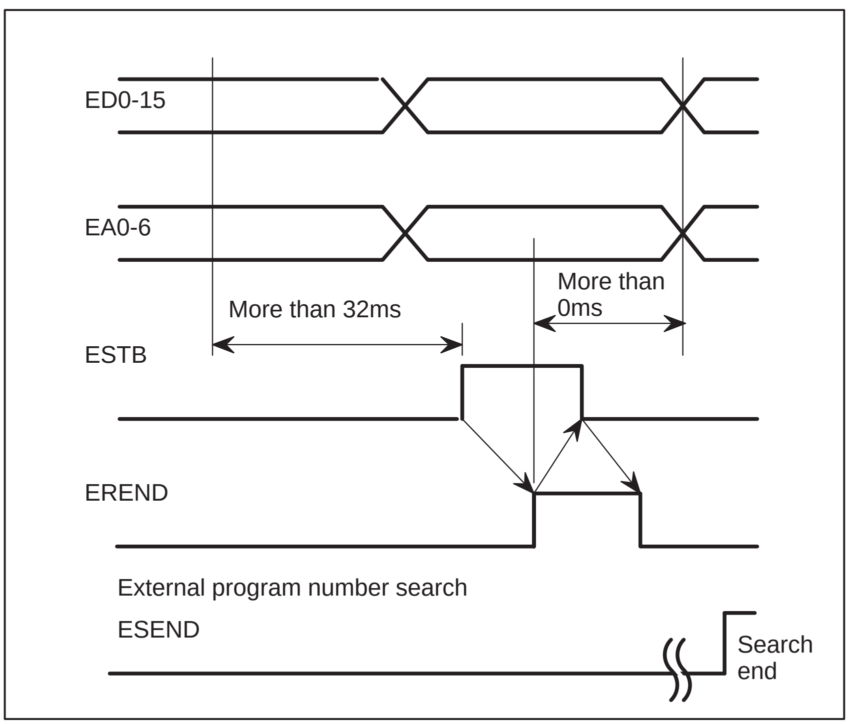
(4) After reading the address, the control unit sets read completion signal EREND to 1.

(5) When the EREND signal is set to 1, the PMC sets the ESTB signal to 0.

(6) When the ESTB signal is set to 0, the control unit sets the EREND signal to 0.

This completes the data input procedure. New data can now be entered.

The timing diagram is shown below:



Kind of data accessed by external data input

No.	Item	E S E E E T A A A B 6 5 4	E E E E A A A A 3 2 1 0	ED15 to ED0			
				15141312	1110 9 8	7 6 5 4	3 2 1 0
1	External program number search	1 0 0 0	××××	Program number(BCD4 digits)			
				0A9	0A9	0A9	0A9
2	External tool compensation	1 0 0 1	××××	Offset value(BCD 4 digits with sign)			
				±0A7	0A9	0A9	0A9
3	External workpiece coordinate system shift	1 0 1 0	axis code	Shift value(BCD 4 digits with sign)			
				±0A7	0A9	0A9	0A9
4	External machine coordinate system shift	1 0 1 1	axis code	Machine coordinate system shift value(binary) ±0A9999			
5	Alarm set	1 1 0 0	0 0 0 0	Alarm No.(binary)			
					0A999		
	Alarm clear	1 1 0 0	0 0 0 1	Alarm No.(binary)			
					0A999		
	Operator message list	1 1 0 0	0 1 0 0	Message No.(binary)			
					0A999		
	Operator message clear	1 1 0 0	0 1 0 1	Message No.(binary)			
					0A999		
	Message	1 1 0 0	0×1 1	character(Character code)			
6	Substitute No. of parts required	1 1 1 0	0 0 0 0	No. of parts required(BDC4 digits)			
				0A9	0A9	0A9	0A9
	Substitute No. of parts machined	1 1 1 0	0 0 0 1	No. of parts machined(BDC4 digits)			
				0A9	0A9	0A9	0A9

Note 1 Input an axis code according to the list below.

Axis	EA3 to EA0			
	3	2	1	0
1st axis	0	0	0	0
2nd axis	0	0	0	1
3rd axis	0	0	1	0
4th axis	0	0	1	1
5th axis	0	1	0	0
6th axis	0	1	0	1
7th axis	0	1	1	0
8th axis	0	1	1	1

Note 2 Though bits EA4 to EA6 distinguish one set of data from another, the machine must be interlocked in order to prevent other function data being fed during a process for which they are invalid.

1) External Program Number Search

A program number (1 to 9999) is specified from the outside and is selected in the CNC memory.

For machines that can load several kinds of workpieces, this function can automatically select for execution the program corresponding to a specific workpiece.

Data for the external program number search is accepted regardless of the mode, but the search execution can be made only in the reset state.

The ESEND signal switches from “0” to “1” at the end of the external program number search. This signal does not turn to “0” unless the cycle start or reset signal is input, or another search is made. Use ESEND to make a cycle start signal after the search.

Note 1 The external program number search is valid when parameter ESR no. 6300#4=1.

Note 2 The reset state is when the automatic operation lamp is off. If the start button is pushed in the cycle operation stop or hold state, search execution starts from the actual position indicated by the pointer.

Note 3 When there is not a program stored in memory corresponding to the set program number, alarm no. 59 will be activated.

Note 4 Program search is not made if the program number is set to “0”. When the start button is pushed, execution starts from the position indicated by the pointer, instead.

2) External tool compensation

These signals provide for changing the tool compensation amount via the PMC. When the offset number is specified by a program, data input from the PMC is added to the offset amount. The offset amount can also be used as input data itself by specifying the input signal.

When the machine tool is equipped with automatic tools or workpiece measuring functions, the offset amount can be corrected using this function, by inputting the error from the correct value into the CNC via PMC.

If the tool compensation amount is externally input when offset number 0 is specified in a program (a offset cancel) in T series, the workpiece coordinate system shifts by the entered quantity. The external tool compensation amount is 0 - +/-7.999mm or +/-0.7999 inch at a time.

Data specification method in external tool compensation (For M series)

Address							Data															
EA6	EA5	EA4	EA3	EA2	EA1	EA0	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
0	0	1	ABS / INC	0	0	0	sign	MSB ← Offset amount → LSB														

sign: 0 : + / 1 : –

Specified by BCD 4-digit code (0000A7999)

Specify “000”

0 : Incremental specification (Input value is added to present compensation amount)
1 : Absolute specification (Input value is replaced with present compensation amount)

Specify “001” for external tool offset

Data specification method in external tool compensation (For T series)

Address							Data															
EA6	EA5	EA4	EA3	EA2	EA1	EA0	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
0	0	1	ABS / INC	W / G	X/Z/R		sign	MSB ← Offset amount → LSB														

sign: 0 : + / 1 : –

Specified by BCD 4-digit code (0000A7999)

EA1	EA0	Description
0	0	Specification of X axis offset
0	1	Specification of Z axis offset
1	0	Specification of tool nose radius compensation amount

0 : Specification of tool wear offset amount
1 : Specification of tool geometry offset amount

0 : Incremental specification (Input value is added to present compensation amount)
1 : Absolute specification (Input value is replaced with present compensation amount)

Specify “001” for external tool offset

3) External workpiece coordinate system shift

The external workpiece coordinate system shift adjusts the workpiece coordinate system depending on the shift amount set via PMC. Each axis (parameter no. 1220) has this shift amount, and it is added to all the workpiece coordinate systems for use. The shift amount is not lost by cut off of the power supply. It is not added incrementally, but each input shift amount makes a new shift amount. The amount that can be input is 0-+/-7.999 mm or +/-0.7999 inch.

4) External machine coordinate system

The machine coordinate system can be shifted by inputting shift value. When the shift amount is input, compensation is immediately applied to the corresponding axis and the machine starts operation. The position accuracy can be improved by combining this function with the sensor.
The specification method for the axis to be shifted is the same as that for the external workpiece coordinate system shift.
The compensation value is specified for the signals ED0 to ED15 by a binary code ranging from 0 to $\pm$9999. This compensation value should be absolute and the amount which the machine actually moves on input is the difference from the previously stored value. When a large amount of compensation is applied at a time, an alarm such as ”excessive error on stop” may occur. In this case, input the compensation amount several times.

5) External alarm message

- External alarm message
  The external alarm message holds the CNC under an alarm condition by sending an alarm number from the external unit, as well as a message that is displayed on the CRT screen of the CNC. Up to four alarm numbers and messages can be sent at a time; the alarm number ranges from 0 to 999, and the CNC displays it with 1000 added. The message for one alarm number can be up to 15 characters long. The alarm condition is reset by external data.
- External operator message
  The external operator message sends the operator message and number from the external unit to the CNC, with a display on the CRT screen of the CNC.
  Only one message can be transmitted, with a potential message length of 255 characters. The alarm number ranges from 0 to 999; from 0 to 99, the CNC adds 2000 to the number, while from 100 to 999 the number is not displayed, only the message is displayed.

Data specification method in external message

Item	EA6	EA5	EA4	EA3	EA2	EA1	EA0	ED15 to ED0 (binary)
Alarm set	1	0	0	0	0	0	0	Alarm No.
Alarm clear	1	0	0	0	0	0	1	Alarm No.
Operator message list	1	0	0	0	1	0	0	Message No.
Operator message clear	1	0	0	0	1	0	1	Message No.
Message	1	0	0	0	×	1	1	Character (Note)

Note 1 Two characters are sent at a time (see ISO code given in Table below).

ED15 - ED8 . . . . Character code in 1st character.

ED7 - ED0 . . . . Character code in 2nd character.

If sending only one character, fill the second slot with a code smaller than 20 and it will be ignored.

Character code table

b8				0	0	0	0	1	1	1	1
b7				0	0	1	1	0	0	1	1
b6				1	1	0	0	1	1	0	0
b5				0	1	0	1	0	1	0	1
b4	b3	b2	b1								
0	0	0	0	S	0	@	P		–	⍰	⍰
0	0	0	1	P	1	A	Q	#	⍰	⍰	⍰
0	0	1	0	!	2	B	R	V	⍰	⍰	⍰
0	0	1	1	I	3	C	S	W	⍰	⍰	⍰
0	1	0	0	#	4	D	T	”	⍰	⍰	⍰
0	1	0	1	$	5	E	U	·	⍰	⍰	⍰
0	1	1	0	%	6	F	V	⍰	⍰	⍰	⍰
0	1	1	1	&	7	G	W	⍰	⍰	⍰	⍰
1	0	0	0	’	8	H	X	⍰	⍰	⍰	⍰
1	0	0	1	(	9	I	Y	⍰	⍰	⍰	⍰
1	0	1	0	)	:	J	Z	⍰	⍰	⍰	⍰
1	0	1	1	*	;	K	[	⍰	⍰	⍰	⍰
1	1	0	0	+	<	L	o	⍰	⍰	⍰	⍰
1	1	0	1	,	=	M	]	⍰	⍰	⍰	⍰
1	1	1	0	=	>	N	^	⍰	⍰	⍰	m
1	1	1	1	.	?	O	_	⍰	⍰	⍰	#
				/							

SP : Space code

6) Substituting No. of parts required and No. of parts, machined

Substitution is possible for the No. of parts required and the No. of parts machined.

Data specifiecation method for No. of parts required and No. of parts machined.

Address							Data		
EA6	EA5	EA4	EA3	EA2	EA1	EA0	ED15	. . .	ED0
1	1	0	0	0	0	0/1	MSB LSB Value of aprt nos.		

BCD 4–digit code (0000A9999)

0 : No. of parts required

1 : No. of parts machined

Set to 110000.

Signals

The signals indicate the entered data.
The use of the 16 code signals varies with the data type.

Data signals for external data input ED0 to ED15 <G000, G001>

[Classification] Input signal

Address signals for external data input EA0 to EA6 <G002>

[Classification] Input signal

[Function] The signals indicate the type of the entered data.

Read signal for external data input ESTB <G002#7>

[Classification] Input signal

[Function] The signal reports that the address and data are set in external data input.

When the signal is set to 1, the control unit reads the address and data for external data input.

[Operation] The ”basic procedure” describes the procedure for, and operation of, the control unit when the signal turns to ”1”.

Read completion signal for external data input EREND <F060#0>

[Classification] Output signal

[Function] The signal reports that the control unit has finished reading the entered data.

[Operation] The output condition and procedure are described in the "basic procedure."

Search completion signal for external data input ESEND <F060#1>

[Classification] Output signal

[Function] The signal reports that program number search, specified by external data input, has been completed.

[Output condition] The signal is set to 1 when:

The program number search specified by external data input is completed.

The signal is set to 0 when:

· An automatic operation is started.
· A reset occurs.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G000	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
G001	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
G002	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0
F060							ESEND	EREND

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6300				ESR				

[Data type] Bit type

ESR External program number search
0 : Disabled
1 : Enabled

Alarm and Message

Number	Message	Description
059	PROGRAM NUMBER NOT FOUND	In an external program number search or external workpiece number search, a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Check the program number and external signal. Or discontinue the background eiting.
131	TOO MANY EXTERNAL ALARM MESSAGES	Five or more alarms have generated in external alarm message. Consult the PMC ladder diagram to find the cause.
132	ALARM NUMBER NOT FOUND	No alarm No. concerned exists in external alarm message clear. Check the PMC ladder diagram.
133	ILLEGAL DATA IN EXT. ALARM MSG	Small section data is erroneous in external alarm message or external operator message. Check the PMC ladder diagram.

15.3 EXTERNAL WORKPIECE NUMBER SEARCH

General

When several part programs are stored in program storage memory, a program can be searched with the workpiece number search signals PN1 to PN16 from the machine side.

When the cycle operation is actuated in the memory operation mode under reset status, the workpiece number (program number) specified by PN1 to PN16 is searched and executed from the beginning.

Signal

Workpiece Number Search Signal PN1, PN2, PN4, PN8, PN16 <G009#0 to #4>

[Classification] Input signal

[Function] Select the number of a workpiece to be machined in the memory mode. Five code signals are provided. These signals are set as binary code to designate a workpiece number as follows:

Workpiece number search signal					Workpiece number
PN16	PN8	PN4	PN2	PN1	
0	0	0	0	0	00
0	0	0	0	1	01
0	0	0	1	0	02
0	0	0	1	1	03
0	0	1	0	0	04
0	0	1	0	1	05
0	0	1	1	0	06
0	0	1	1	1	07
0	1	0	0	0	08
0	1	0	0	1	09
0	1	0	1	0	10
0	1	0	1	1	11
0	1	1	0	0	12
0	1	1	0	1	13
0	1	1	1	0	14
0	1	1	1	1	15

Workpiece number search signal					Workpiece number
PN16	PN8	PN4	PN2	PN1	
1	0	0	0	0	16
1	0	0	0	1	17
1	0	0	1	0	18
1	0	0	1	1	19
1	0	1	0	0	20
1	0	1	0	1	21
1	0	1	1	0	22
1	0	1	1	1	23
1	1	0	0	0	24
1	1	0	0	1	25
1	1	0	1	0	26
1	1	0	1	1	27
1	1	1	0	0	28
1	1	1	0	1	29
1	1	1	1	0	30
1	1	1	1	1	31

Workpiece number 00 is used for special designation “no search”. Thus, a workpiece number ranges from 01 to 31.

Note 1 These signals are also used to specify a file number for file search during external program input. See Section 13.7, “External Program Input.”

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G009				PN16	PN8	PN4	PN2	PN1

Alarm and Message

Number	Message	Description
059	PROGRAM NUMBER NOT FOUND	In an external program number search or external workpiece number search, a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Check the program number and external signal. Or discontinue the background eiting.

Note

Note 1 This function can be used only in memory operation. It cannot be used during manual data input.

Note 2 Select the program number from O001 ~ O031.

Note 3 Program numbers from O001 to O031 can be used. However, programs corresponding to all the program numbers do not have to be stored in memory.

Note 4 When a program corresponding to the specified program number is not stored in memory, an alarm (No. 59) is activated when the start button is pressed.

Note 5 Program search is performed only when the start button is pressed in the reset state. When the CNC is in the automatic operation stop state (single block stop, etc.) or pause state (feedhold stop, etc.), program search is not performed even if the start button is pressed and execution is started from the point specified by the present execution pointer.

Note 6 To restart program halfway through, press the start button after sequence number search in MEM mode. The workpiece number search is not performed; program execution starts from the block which is searched by sequence number search, because the OP signal is set by sequence number search in MEM mode and the CNC reset state is released.

Note 7 When the start button is pressed with all PN1 to PN16 “0”, program search is not performed but execution is started from the point specified by the present execution pointer. To restart operation from the start of a program which cannot be searched by this function, perform the usual program number search operation (CRT/MDI panel operation), turn all the PN1 to PN16 to “0” and press the start button

15.4 SPINDLE OUTPUT CONTROL BY THE PMC

General

The PMC can control the speed and polarity of each spindle motor, connected by the optional spindle serial output/spindle analog output function.

The first, second, and third spindles all 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.

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. If the multispindle control function is added, the CNC can also control the second and third spindle motors.

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 an SINDx signal

· Setting the spindle motor speed data, calculated by the PMC, in 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 setting (for example, the maximum speed clamp parameter) related to the spindle speed command of the CNC spindle control function.

→ If the multispindle control function is added, however, the spindle stop signal *SSTPx <G0027, #3, #4, #5> can be used to stop a PMC–controlled spindle.

The spindle motor speed data is obtained from the following expression. Its value can range from 0 to 4095:

$$\text{Spindle motor speed data} = \frac{\text{Spindle motor speed}}{\text{Maximum spindle motor speed}} \times 4095$$

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.

$$\text{Spindle motor speed data} = \frac{\text{Spindle speed}}{\text{Maximum spindle speed}} \times 4095$$

By using this expression, the spindle motor speed data can be obtained easily.

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 require 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 <F022 to F025> and SF signal <F007, #2> can be output only when many 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 <F036, #0 to F037, #3> are output to the first spindle motor. The output data is calculated from the results of the CNC spindle control. (See Section 9.3.)

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.

Sample application 1)

Controlling the first and second spindles of a lathe system

→ 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)

Using a lathe's orientation function 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 those 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 <G078, #0 to G079, #3> and perform the orientation.

Signal

PMC spindle control signals
For the first spindle: SIND, SSIN, SGN <G033#7, #6, #5>
R01I to R12I <G032#0 to G033#3>
For the second spindle: SIND2, SSIN2, SGN2 <G035#7, #6, #5>
R01I2 to R12I2 <G034#0 to G035#3>
For the third spindle: SIND3, SSIN3, SGN3 <G037#7, #6, #5>
R01I3 to R12I3 <G036#0 to G037#3>

[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 if the multispindle control function is not provided, these signals allow the second or third spindle to be controlled.

When the multispindle control function and type A are being used (if the MSI bit, bit 2 of parameter No. 3709, is set to 0), the signals for the second and third spindles 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.

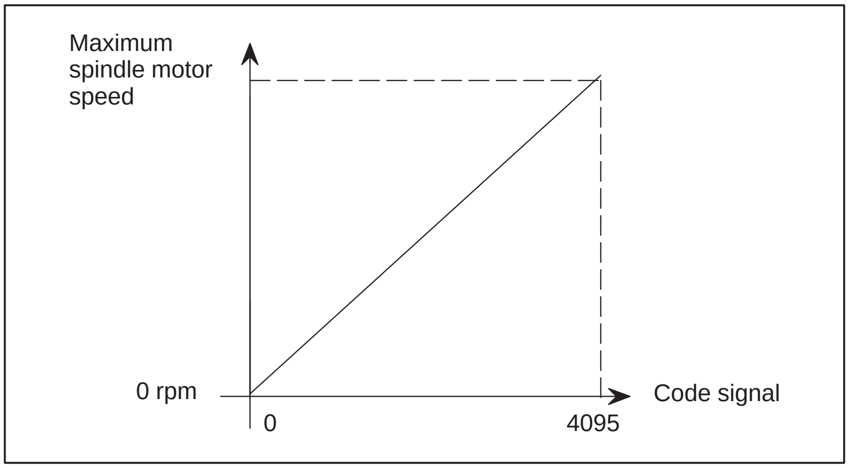
0: The spindle motor is controlled according to speed commands issued by the CNC. The spindle speed specified with the S command is output.

- 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.

$$\text{Value to be specified} = \frac{\text{Spindle motor speed}}{\text{Maximum spindle motor speed}} \times 4095$$

(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 TCW and CWM bits (bits 7 and 6 of parameter No. 3706) 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.

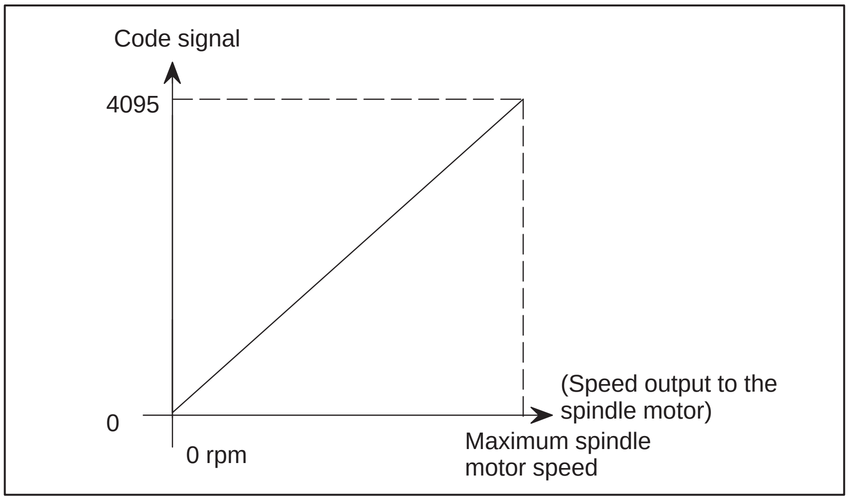
Twelve code signals corresponding to the S value R01O to R12O <G036#0 to G037#3>

[Classification] Output signal

[Function] The S value, specified in the CNC part program, is converted to the speed output to the spindle motor that is required to control the connected spindle. The converted value is sent to the PMC with twelve code signals, in proportion 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.

(See Section 9.3 for an explanation of the relationship between the CNC spindle control and the speed output to the spindle motor.)



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3701				SS2				

[Data type] Bit

SS2 The number of connections in serial spindle control
0 : 1
1 : 2

	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS				ESF
		SFA	NSF					ESF

[Data type] Bit

ESF When the spindle control function (S analog output or S serial output) is used, and the constant surface speed control function is used or bit 7 (GTT) of parameter No. 3705 is set to 1:
0 : S codes and SF are output for all S commands.
1 : S codes and SF are not output for an S command in constant surface speed control mode (G96 mode) or for an S command used to specify maximum spindle speed clamping (G50S–––;).

Note 1 For the T series, this parameter is enabled when bit 4 (EVS) of parameter No. 3705 is set to 1. For the M series, SF is not output:
(1) For an S command used to specify maximum spindle speed clamping (G92S–––;) in constant surface speed control mode
(2) When bit 5 (NSF) of parameter No. 3705 is set to 1

EVS When the spindle control function (S analog output or S serial output) is used, S codes and SF are:
0 : Not output for an S command.
1 : Output for an S command.

Note 1 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.

NSF: When an S code command is issued in constant surface–speed control,
0 : SF is output.
1 : SF is not output.

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		

[Data type] Bit

MSI In multi–spindle control, the SIND signal is valid:
0 : Only when the first spindle is selected. (SIND signal for 2nd and 3rd spindle become invalid)
1 : For each spindle irrespective of whether the spindle is selected. (Each spindle has its own SIND signal.)

3821	Offset–voltage compensation value of the analog output of the third–spindle speed

[Data type] Word

[Unit of data] Velo

[Valid data range] –1024 to 1024

Set the offset–voltage compensation value of the analog output of the third–spindle speed.

[Setting method]

(1) Set 0 (standard value)

(2) Specify a spindle speed at which the spindle speed analog output becomes 0.

(3) Measure output voltage.

(4) Set the following value to parameter No. 3731.

$$\text{Set value} = \frac{-8191 \times \text{offset voltage (V)}}{12.5}$$

(5) After the parameter has been set, command again a spindle speed where the spindle speed analog output become 0 and confirm that voltage becomes 0V.

Note

Note 1 If the spindle fails to move after the PMC issues the spindle motor speed command, check the following:

Type A is selected (the MSI bit, bit 2 of parameter No. 3709, is set to 0) when the multispindle control function is used.

→ The second or third spindle cannot be controlled. The first spindle can be controlled only when the spindle selection signal SWS1 is set to “1”.

The spindle stop signal for each axis is set to “0” when the multispindle control function is being used.

→ Spindle stop signal for each axis *SSTPx <G027, #3, #4, #5> 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.

Note 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.

Note 3 For an explanation of connecting the second or third spindle, see Sections 9.2 and 9.10.

Note 4 If the multispindle control function is not being used, the CNC does not issue any commands to the second and third spindles. The output polarity is controlled by the SGNx signal. It is not affected by the SSINx signal.

The speed output to the spindle motor can be controlled only when the SINDx signal is set to “1”.

15.5 EXTERNAL KEY INPUT

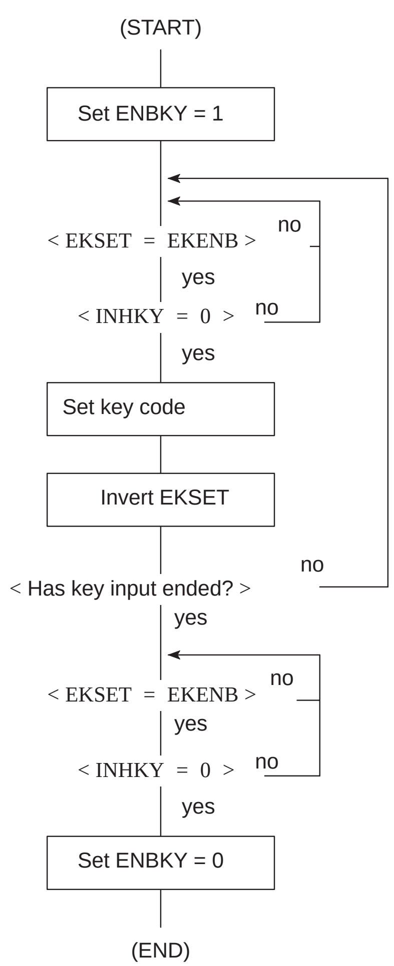
General

MDI key codes can be sent from the PMC to CNC by means of interface signals. This allows the CNC to be controlled in the same way as when the operator performs CRT/MDI key operation.

Control is realized by exchanging the following interface signals between the PMC and CNC:

Signal name	Abbreviation
External key input mode selection signal (input)	ENBKY
Key code signals (input)	EKC0AEKC7
Key code read signal (input)	EKSET
Key code read completion signal (output)	EKENB
Key input disable signal (output)	INHKY
Program screen display mode signal (output)	PRGDPL

The processing flow in the PMC is shown below.



1) Set the external key input control mode.

2) Check whether a key code can be set.

3) When a key code can be set:
(1) Set the key code to be input.
(2) Invert the (EKSET) signal to instruct the CNC to read the key code.

4) Repeat the operation for the key data to be input

5) After reporting the last data, check that the CNC has read the key data.

6) Release external key input control mode.

Note 1 Read processing is controlled by exclusive–ORing (XOR) the key code read signal (EKSET) with the read completion signal (EKENB). When the EKSET and EKENB signals differ in their logic, the CNC reads the input key code. Once reading has been completed, the CNC inverts the EKENB signal to match its logic with that of the EKSET signal.

In the PMC, on the other hand, a new key code cannot be set while the EKSET and EKENB signals differ in their logic.

Signals

External key input mode selection signal ENBKY <G066#1>

[Classification] Input signal

[Function] While this signal is turned on “1”, external key input control is enabled. In external key input control mode, any CRT/MDI key operations are ignored.

Key code read signal EKSET <G066#7>

[Classification] Input signal

[Function] This signal instructs the CNC to read the input key code.

Key code signals EKC0 to EKC7 <G098>

[Classification] Input signal

[Function] These signals set an input key code. (See the MDI key code table.)

Key input disable signal INHKY<F053#0>

[Classification] Output signal

[Function] While this signal is 1, no key code is accepted in external key input control mode.

Program screen display mode signal PRGDPL <F053#1>

[Classification] Output signal

[Function] This signal is on “1” while the CNC is displaying a program screen.

Key code read completion signal EKENB <F053#7>

[Classification] Output signal

[Function] This signal reports that the CNC has read a key code.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G066	EKSET						ENBKY	

	#7	#6	#5	#4	#3	#2	#1	#0
G098	EKC7	EKC6	EKC5	EKC4	EKC3	EKC2	EKC1	EKC0

	#7	#6	#5	#4	#3	#2	#1	#0
F053	EKENB						PRGDPL	INHKY

MDI key code table

Codes in the table are given in hexadecimal. For example, A corresponds to 41H in hexadecimal. RESET corresponds to 90H in hexadecimal.

(Note 1) and (Note 2) are explained below:

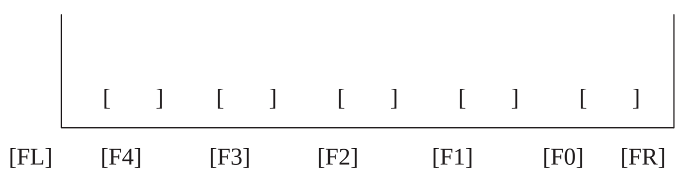
Note 1 For the small keyboard, 0EDH is assigned to GRAPH/CUSTOM.

For the standard keyboard, 0EDH is assigned to GRAPH. 0EEH is assigned to CUSTOM.

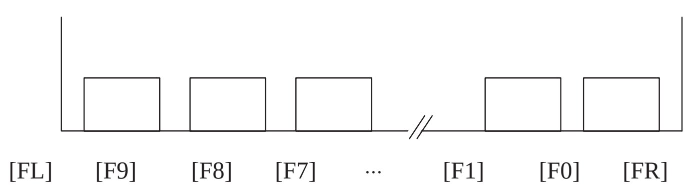
Note 2 Handling of the soft keys

[F0] to [F9], [FR], and [FL] in the key code table are the key codes for the soft keys. They are associated with the MDI keys as shown below.

(9 ”CRT) [F0] A[F4] and [FR], [FL]



(8.4” LCD/9.5” LCD/14”CRT) [F0]A[F9] and [FR], [FL]



MDI Key Code Table(00HA7FH)

	0	1	2	3	4	5	6	7
0			Space	0	@	P		
1				1	A	Q		
2				2	B	R		
3			#	3	C	S		
4				4	D	T		
5				5	E	U		
6			&	6	F	V		
7				7	G	W		
8			(	8	H	X		
9			)	9	I	Y		
A	; (EOB)		*		J	Z		
B			+		K	[		
C			,		L			
D			–	=	M	]		
E			.		N			
F			/	?	O			

MDI Key Code Table(80HA0FFH)

	8	9	A	B	C	D	E	F
0		RESET						[F0] (Note2)
1								[F1] (Note2)
2								[F2] (Note2)
3								[F3] (Note2)
4		INSERT						[F4] (Note2)
5		DELETE						[F5] (Note2)
6	CAN	ALTER						[F6] (Note2)
7								[F7] (Note2)
8	Cursor →	INPUT					POS	[F8] (Note2)
9	Cursor ←						PROG	[F9] (Note2)
A	Cur-sor↓	HELP					OFF-SET SET-TING	
B	Cur-sor↑						SYS-TEM	
C							MES-SAEG	
D							GRAPH (CUSTOM) (Note1)	
E	PAGE↓						CUS-TOM (Note1)	[FR] (Note2)
F	PAGE↑						FAPT	[FL] (Note2)

15.6 DIRECT OPERATION BY PMC/MMC

General

Activating memory operation in memory operation mode (MEM) with the direct operation select signal set to 1 enables machining (direct operation) while reading a program stored in the PMC–RC or MMC.

Signal

Direct operation select signal DMMC <G042#7>

[Classification] Input signal

[Function] Selects the mode (direct operation mode) for performing machining while reading a program stored in the PMC–RC and MMC.

[Operation] When this signal is set to 1, the control unit operates as follows:

· When memory operation mode (MEM) is not selected, the control unit ignores this signal.

· When memory operation mode (MEM) is selected, the control unit selects direct operation mode and enables direct operation.

Reference Item

FANUC PMC–MODEL RC/RC3/NB PROGRAMMING MANUAL (C LANGUAGE) (B–61863E–1)

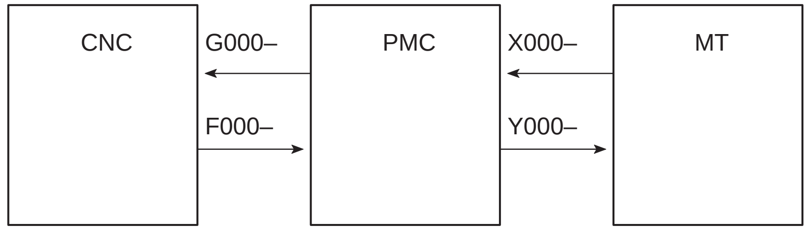
APPENDIX

A INTERFACE BETWEEN CNC AND PMC

A.1 LIST OF ADDRESSES

A.1.1 List of Addresses (One–Path Control)

Interface addresses among CNC, PMC and Machine Tool are as follows:



Following shows table of addresses:

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 T series or M series does not have this signal. Upper part is for T series and lower part is for M series.

[Example 1]

Signals EXLM and ST are common signals, STLK is for T series only and RLSOT and RVS are for M series only.

	#7	#6	...	#2	#1	#0	
G007	RLSOT	SA		ST	STLK	RVS	T series M series

MT → PMC

Address	7	6	5	4	3	2	1	0	
X000									
X001									
X002									
X003									
X004	SKIP	ESKIP SKIP6	–MIT2 SKIP5	+MIT2 SKIP4	–MIT1 SKIP3	+MIT1 SKIP2	ZAE SKIP8	XAE SKIP7	(T series)
	SKIP	ESKIP SKIP6	SKIP5	SKIP4	SKIP3	ZAE SKIP2	YAE SKIP8	XAE SKIP7	(M series)
X005									
X006									
X007									
X008				*ESP					
X009	*DEC8	*DEC7	*DEC6	*DEC5	*DEC4	*DEC3	*DEC2	*DEC1	
X010									
X011									
X012									

(Bit number)

PMC → CNC

Address	Bit 7	6	5	4	3	2	1	0
G000	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
G001	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
G002	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0
G003								
G004			MFIN3	MFIN2	FIN			
G005	BFIN	AFL		BFIN	TFIN	SFIN	EFIN	MFIN
G006		SKIPP		OVC		*ABSM		SRN
G007	RLSOT	EXLM	*FLWU			ST	STLK	RVS
G008	ERS	RRW	*SP	*ESP				*IT
G009				PN16	PN8	PN4	PN2	PN1
G010	*JV7	*JV6	*JV5	*JV4	*JV3	*JV2	*JV1	*JV0
G011	*JV15	*JV14	*JV13	*JV12	*JV11	*JV10	*JV9	*JV8
G012	*FV7	*FV6	*FV5	*FV4	*FV3	*FV2	*FV1	*FV0
G013	*AFV7	*AFV6	*AFV5	*AFV4	*AFV3	*AFV2	*AFV1	*AFV0
G014							ROV2	ROV1
G015								
G016	F1D							
G017								
G018	HS2D	HS2C	HS2B	HS2A	HS1D	HS1C	HS1B	HS1A
G019	RT		MP2	MP1	HS3D	HS3C	HS3B	HS3A
G020								
G021								
G022								
G023	ALNGH							
G024								

	7	6	5	4	3	2	1	0
G025								
G026								
G027	CON		*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
G028	PC2SLC	SPSTP	*SCPF	*SUCPF		GR2	GR1	
G029		*SSTP	SOR	SAR		GR31		GR21
G030	SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0
G031								
G032	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
G033	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
G034	R08I2	R07I2	R06I2	R05I2	R04I2	R03I2	R02I2	R01I2
G035	SIND2	SSIN2	SGN2		R12I2	R11I2	R10I2	R09I2
G036	R08I3	R07I3	R06I3	R05I3	R04I3	R03I3	R02I3	R01I3
G037	SIND3	SSIN3	SGN3		R12I3	R11I3	R10I3	R09I3
G038	*BECLP	*BEUCP			SPPHS	SPSYC		*PLSST
G039	GOQSM	WOQSM	OFN5	OFN4	OFN3	OFN2	OFN1	OFN0
G040	WOSET	PRC						OFN6
G041	HS2ID	HS2IC	HS2IB	HS2IA	HS1ID	HS1IC	HS1IB	HS1IA
G042	DMMC				HS3ID	HS3IC	HS3IB	HS3IA
G043	ZRN		DNCI			MD4	MD2	MD1
G044							MLK	BDT1
G045	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2
G046	DRN	KEY4	KEY3	KEY2	KEY1		SBK	
G047	TL128	TL64	TL32	TL16	TL08	TL04	TL02	TL01
G048	TLRST	TLRSTI	TLSKP					TL256
G049	*TLV7	*TLV6	*TLV5	*TLV4	*TLV3	*TLV2	*TLV1	*TLV0

	7	6	5	4	3	2	1	0
G050							*TLV9	*TLV8
G051								
G052	RMTDI7	RMTDI6	RMTDI5	RMTDI4	RMTDI3	RMTDI2	RMTDI1	RMTDI0
G053	CDZ	SMZ			UINT			TMRON
G054	UI007	UI006	UI005	UI004	UI003	UI002	UI001	UI000
G055	UI015	UI014	UI013	UI012	UI011	UI010	UI009	UI008
G056								
G057								
G058		STWD	STRD		EXWT	EXSTP	EXRD	MINP
G059							TRRTN	TRESC
G060	*TSB							
G061			RGTSP2	RGTSP1				RGTAP
G062								
G063								
G064								
G065								
G066	EKSET						ENBKY	IGNVRY
G067								
G068								
G069								
G070	MRDYA	ORCMA	SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
G071	RCHA	RSLA		SOCNA	MCFNA	SPSLA	*ESPA	ARSTA
G072								
G073								
G074	MRDYB	ORCMB	SFRB	SRVB	CTH1B	CTH2B	TLMHB	TLMLB

	7	6	5	4	3	2	1	0
G075	RCHB	RSLB		SOCNB	MCFNB	SPSLB	*ESPB	ARSTB
G076								
G077								
G078	SHA07	SHA06	SHA05	SHA04	SHA03	SHA02	SHA01	SHA00
G079					SHA11	SHA10	SHA09	SHA08
G080	SHB07	SHB06	SHB05	SHB04	SHB03	SHB02	SHB01	SHB00
G081					SHB11	SHB10	SHB09	SHB08
G082	Reserve for order made macro							
G083	Reserve for order made macro							
G084								
G085								
G086								
G087								
G088								
G089								
G090								
G091								
G092								
G093								
G094								
G095								
G096	HROV	*HROV6	*HROV5	*HROV4	*HROV3	*HROV2	*HROV1	*HROV0
G097								
G098	EKC7	EKC6	EKC5	EKC4	EKC3	EKC2	EKC1	EKC0
G099								

	7	6	5	4	3	2	1	0
G100	+J8	+J7	+J6	+J5	+J4	+J3	+J2	+J1
G101								
G102	–J8	–J7	–J6	–J5	–J4	–J3	–J2	–J1
G103								
G104								
G105								
G106	MI8	MI7	MI6	MI5	MI4	MI3	MI2	MI1
G107								
G108	MLK8	MLK7	MLK6	MLK5	MLK4	MLK3	MLK2	MLK1
G109								
G110	+LM8	+LM7	+LM6	+LM5	+LM4	+LM3	+LM2	+LM1
G111								
G112	–LM8	–LM7	–LM6	–LM5	–LM4	–LM3	–LM2	–LM1
G113								
G114	*+L8	*+L7	*+L6	*+L5	*+L4	*+L3	*+L2	*+L1
G115								
G116	*–L8	*–L7	*–L6	*–L5	*–L4	*–L3	*–L2	*–L1
G117								
G118	*+ED8	*+ED7	*+ED6	*+ED5	*+ED4	*+ED3	*+ED2	*+ED1
G119								
G120	*–ED8	*–ED7	*–ED6	*–ED5	*–ED4	*–ED3	*–ED2	*–ED1
G121								
G122	PK8	PK7	PK6	PK5	PK4	PK3	PK2	PK1
G123								
G124	DTCH8	DTCH7	DTCH6	DTCH5	DTCH4	DTCH3	DTCH2	DTCH1

	7	6	5	4	3	2	1	0
G125								
G126	SVF8	SVF7	SVF6	SVF5	SVF4	SVF3	SVF2	SVF1
G127								
G128								
G129								
G130	*IT8	*IT7	*IT6	*IT5	*IT4	*IT3	*IT2	*IT1
G131								
G132					+MIT4	+MIT3	+MIT2	+MIT1
G133								
G134					–MIT4	–MIT3	–MIT2	–MIT1
G135								
G136	EAX8	EAX7	EAX6	EAX5	EAX4	EAX3	EAX2	EAX1
G137								
G138	SYNC8	SYNC7	SYNC6	SYNC5	SYNC4	SYNC3	SYNC2	SYNC1
G139								
G140	SYNCJ8	SYNCJ7	SYNCJ6	SYNCJ5	SYNCJ4	SYNCJ3	SYNCJ2	SYNCJ1
G141								
G142	EBUFA	ECLRA	ESTPA	ESOFA	ESBKA	EMBUFA		EFINA
G143	EMSBKA	EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A
G144	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G145	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A
G146	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G147	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G148	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A
G149	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A

	7	6	5	4	3	2	1	0
G150	DRNE	RTE	OVCE				ROV2E	ROV1E
G151	*FV7E	*FV6E	*FV5E	*FV4E	*FV3E	*FV2E	*FV1E	*FV0E
G152								
G153								
G154	EBUFB	ECLRB	ESTPB	ESOFB	ESBKB	EMBUFB		EFINB
G155	EMSBKB	EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B
G156	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B
G157	EIF15B	EIF14B	EIF13B	EIF12B	EIF11B	EIF10B	EIF9B	EIF8B
G158	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G159	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G160	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G161	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B
G162								
G163								
G164								
G165								
G166	EBUFC	ECLRC	ESTPC	ESOFC	ESBKC	EMBUFC		EFINC
G167	EMSBKC	EC6C	EC5C	EC4C	EC3C	EC2C	EC1C	EC0C
G168	EIF7C	EIF6C	EIF5C	EIF4C	EIF3C	EIF2C	EIF1C	EIF0C
G169	EIF15C	EIF14C	EIF13C	EIF12C	EIF11C	EIF10C	EIF9C	EIF8C
G170	EID7C	EID6C	EID5C	EID4C	EID3C	EID2C	EID1C	EID0C
G171	EID15C	EID14C	EID13C	EID12C	EID11C	EID10C	EID9C	EID8C
G172	EID23C	EID22C	EID21C	EID20C	EID19C	EID18C	EID17C	EID16C
G173	EID31C	EID30C	EID29C	EID28C	EID27C	EID26C	EID25C	EID24C
G174								

	7	6	5	4	3	2	1	0
G175								
G176								
G177								
G178	EBUFD	ECLRD	ESTPD	ESOFD	ESBKD	EMBUFD		EFIND
G179	EMSBKD	EC6D	EC5D	EC4D	EC3D	EC2D	EC1D	EC0D
G180	EIF7D	EIF6D	EIF5D	EIF4D	EIF3D	EIF2D	EIF1D	EIF0D
G181	EIF15D	EIF14D	EIF13D	EIF12D	EIF11D	EIF10D	EIF9D	EIF8D
G182	EID7D	EID6D	EID5D	EID4D	EID3D	EID2D	EID1D	EID0D
G183	EID15D	EID14D	EID13D	EID12D	EID11D	EID10D	EID9D	EID8D
G184	EID23D	EID22D	EID21D	EID20D	EID19D	EID18D	EID17D	EID16D
G185	EID31D	EID30D	EID29D	EID28D	EID27D	EID26D	EID25D	EID24D
G186								
G187								
G188								
G189								
G190								
G191								
G192	IGVRY8	IGVRY7	IGVRY6	IGVRY5	IGVRY4	IGVRY3	IGVRY2	IGVRY1
G193								
G194								
G195								
G196								
G197								
G198	NPOS8	NPOS7	NPOS6	NPOS5	NPOS4	NPOS3	NPOS2	NPOS1
G199								

	7	6	5	4	3	2	1	0
G200								
G201								
G202								
G203								
G204								
G205								
G206								
G207								
G208								
G209								
G210								
G211								
G212								
G213								
G214								
G215								
G216								
G217								
G218								
G219								
G220								
G221								
G222								

CNC → PMC

Address	Bit 7	6	5	4	3	2	1	0
F000	OP	SA	STL	SPL				RWD
F001	MA		TAP	ENB	DEN	BAL	RST	AL
F002	MDRN	CUT		SRNMV	THRD	CSS	RPDO	INCH
F003	MTCHIN	MEDT	MMEM	MRMT	MMDI	MJ	MH	MINC
F004			MREF	MAFL	MSBK	MABSM	MMLK	MBDT1
F005	MBDT9	MBDT8	MBDT7	MBDT6	MBDT5	MBDT4	MBDT3	MBDT2
F006								
F007	BF			BF	TF	SF	EFD	MF
F008			MF3	MF2				EF
F009	DM00	DM01	DM02	DM30				
F010	M07	M06	M05	M04	M03	M02	M01	M00
F011	M15	M14	M13	M12	M11	M10	M09	M08
F012	M23	M22	M21	M20	M19	M18	M17	M16
F013	M31	M30	M29	M28	M27	M26	M25	M24
F014	M207	M206	M205	M204	M203	M202	M201	M200
F015	M215	M214	M213	M212	M211	M210	M209	M208
F016	M307	M306	M305	M304	M303	M302	M301	M300
F017	M315	M314	M313	M312	M311	M310	M309	M308
F018								
F019								
F020								
F021								
F022	S07	S06	S05	S04	S03	S02	S01	S00
F023	S15	S14	S13	S12	S11	S10	S09	S08
F024	S23	S22	S21	S20	S19	S18	S17	S16

	7	6	5	4	3	2	1	0
F025	S31	S30	S29	S28	S27	S26	S25	S24
F026	T07	T06	T05	T04	T03	T02	T01	T00
F027	T15	T14	T13	T12	T11	T10	T09	T08
F028	T23	T22	T21	T20	T19	T18	T17	T16
F029	T31	T30	T29	T28	T27	T26	T25	T24
F030	B07	B06	B05	B04	B03	B02	B01	B00
F031	B15	B14	B13	B12	B11	B10	B09	B08
F032	B23	B22	B21	B20	B19	B18	B17	B16
F033	B31	B30	B29	B28	B27	B26	B25	B24
F034						GR3O	GR2O	GR1O
F035								SPAL
F036	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F037					R12O	R11O	R10O	R09O
F038					ENB3	ENB2	SUCLP	SCLP
F039								
F040	AR7	AR6	AR5	AR4	AR3	AR2	AR1	AR0
F041	AR15	AR14	AR13	AR12	AR11	AR10	AR09	AR08
F042								
F043								
F044				SYCAL	FSPPH	FSPSY	FSCSL	
F045	ORARA	TLMA	LDT2A	LDT1A	SARA	SDTA	SSTA	ALMA
F046					RCFNA	RCHPA	CFINA	CHPA
F047								
F048								
F049	ORARB	TLMB	LDT2B	LDT1B	SARB	SDTB	SSTB	ALMB

	7	6	5	4	3	2	1	0
F050					RCFNB	RCHPB	CFINB	CHPB
F051								
F052								
F053	EKENB			BGEACT	RPALM	RPBSY	PRGDPL	INHKY
F054	UO007	UO006	UO005	UO004	UO003	UO002	UO001	UO000
F055	UO015	UO014	UO013	UO012	UO011	UO010	UO009	UO008
F056	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100
F057	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108
F058	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
F059	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124
F060							ESEND	EREND
F061							BCLP	BUCLP
F062	PRTSF							
F063	PSYN					PSAR	PSE2	PSE1
F064						TLCHI	TLNW	TLCH
F065							RGSPM	RGSPP
F066	EXHPCC	MMPCC	PECK2					G08MD
F067								
F068								
F069	RMTDO7	RMTDO6	RMTDO5	RMTDO4	RMTDO3	RMTDO2	RMTDO1	RMTDO0
F070	PSW08	PSW07	PSW06	PSW05	PSW04	PSW03	PSW02	PSW01
F071							PSW10	PSW09
F072	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
F073				ZRNO		MD4O	MD2O	MD1O
F074								

	7	6	5	4	3	2	1	0
F075	SPO	KEYO	DRNO	MLKO	SBKO	BDTO		
F076			ROV2O	ROV1O	RTAP		MP2O	MP1O
F077		RTO			HS1DO	HS1CO	HS1BO	HS1AO
F078	*FV7O	*FV6O	*FV5O	*FV4O	*FV3O	*FV2O	*FV1O	*FV0O
F079	*JV7O	*JV6O	*JV5O	*JV4O	*JV3O	*JV2O	*JV1O	*JV0O
F080	*JV15O	*JV14O	*JV13O	*JV12O	*JV11O	*JV10O	*JV9O	*JV8O
F081	– J4O	+ J4O	– J3O	+ J3O	– J2O	+ J2O	– J1O	+ J1O
F082						RVSL		
F083								
F084								
F085								
F086								
F087								
F088								
F089								
F090						ABTSP2	ABTSP1	ABTQSV
F091								
F092			TRSPS		TRACT			
F093								
F094	ZP8	ZP7	ZP6	ZP5	ZP4	ZP3	ZP2	ZP1
F095								
F096	ZP28	ZP27	ZP26	ZP25	ZP24	ZP23	ZP22	ZP21
F097								
F098	ZP38	ZP37	ZP36	ZP35	ZP34	ZP33	ZP32	ZP31
F099								

	7	6	5	4	3	2	1	0
F100	ZP48	ZP47	ZP46	ZP45	ZP44	ZP43	ZP42	ZP41
F101								
F102	MV8	MV7	MV6	MV5	MV4	MV3	MV2	MV1
F103								
F104	INP8	INP7	INP6	INP5	INP4	INP3	INP2	INP1
F105								
F106	MVD8	MVD7	MVD6	MVD5	MVD4	MVD3	MVD2	MVD1
F107								
F108	MMI8	MMI7	MMI6	MMI5	MMI4	MMI3	MMI2	MMI1
F109								
F110	MDTCH8	MDTCH7	MDTCH6	MDTCH5	MDTCH4	MDTCH3	MDTCH2	MDTCH1
F111								
F112	EADEN8	EADEN7	EADEN6	EADEN5	EADEN4	EADEN3	EADEN2	EADEN1
F113								
F114	TRQL8	TRQL7	TRQL6	TRQL5	TRQL4	TRQL3	TRQL2	TRQL1
F115								
F116	FRP8	FRP7	FRP6	FRP5	FRP4	FRP3	FRP2	FRP1
F117								
F118	SYN8O	SYN7O	SYN6O	SYN5O	SYN4O	SYN3O	SYN2O	SYN1O
F119								
F120	ZRF8	ZRF7	ZRF6	ZRF5	ZRF4	ZRF3	ZRF2	ZRF1
F121								
F122	HDO7	HDO6	HDO5	HDO4	HDO3	HDO2	HDO1	HDO0
F123								
F124								

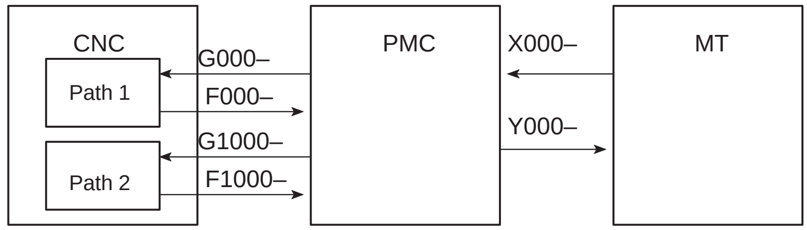
	7	6	5	4	3	2	1	0
F125								
F126								
F127								
F128								
F129	*EAXSL		EOV0					
F130	EBSYA	EOTNA	EOTPA	EGENA	EDENA	EIALA	ECKZA	EINPA
F131							EABUFA	EMFA
F132	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A
F133	EBSYB	EOTNB	EOTPB	EGENB	EDENB	EIALB	ECKZB	EINPB
F134							EABUFB	EMFB
F135	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B
F136	EBSYC	EOTNC	EOTPC	EGENC	EDENC	EIALC	ECKZC	EINPC
F137							EABUFC	EMFC
F138	EM28C	EM24C	EM22C	EM21C	EM18C	EM14C	EM12C	EM11C
F139	EBSYD	EOTND	EOTPD	EGEND	EDEND	EIALD	ECKZD	EINPD
F140							EABUFD	EMFD
F141	EM28D	EM24D	EM22D	EM21D	EM18D	EM14D	EM12D	EM11D
F142	EM48A	EM44A	EM42A	EM41A	EM38A	EM34A	EM32A	EM31A
F143								
F144								
F145	EM48B	EM44B	EM42B	EM41B	EM38B	EM34B	EM32B	EM31B
F146								
F147								
F148	EM48C	EM44C	EM42C	EM41C	EM38C	EM34C	EM32C	EM31C
F149								

	7	6	5	4	3	2	1	0
F150								
F151	EM48D	EM44D	EM42D	EM41D	EM38D	EM34D	EM32D	EM31D
F152								
F153								
F154								
F155								
F156								
F157								
F158								
F159								
F160								
F161								
F162								
F163								
F164								
F165								
F166								
F167								
F168								
F169								
F170								
F171								
F172								
F173								
F174								

	7	6	5	4	3	2	1	0
F175								
F176								
F177								
F178								
F179								
F180								
F181								
F182	EACNT8	EACNT7	EACNT6	EACNT5	EACNT4	EACNT3	EACNT2	EACNT1
F183								
F184								
F185								
F186								
F187								
F188								
F189								
F190								
F191								
F192								
F193								
F194								
F195								

A.1.2 List of Addresses (Two–path control)

Interface addresses among CNC, PMC and Machine Tool are as follows:



Signals addresses for each path are usually assigned as follows:

Signal address	Contents
G000–G255	Signals on path 1(PMC→CNC)
F000–F255	Signals on path 1(CNC→PMC)
G1000–G1255	Signals on path 2(PMC→CNC)
F1000–F1255	Signals on path 2(CNC→PMC)

However, for the signals common to both paths, those signals are assigned to path 1. Interface signals between the CNC and PMC are as shown below:

The signals with suffix #1 are those for path 1 and the signals with suffix #2 are those for path 2.

MT → PMC

Address	Bit number 7	6	5	4	3	2	1	0	
X000									
X001									
X002									
X003									
X004	SKIP #1	ESKIP SKIP6 #1	–MIT2 #1 SKIP5 #1	+MIT2 #1 SKIP4 #1	–MIT1 #1 SKIP3 #1	+MIT1 #1 SKIP2 #1	ZAE #1 SKIP8 #1	XAE #1 SKIP7 #1	(Tseries)
	SKIP #1	ESKIP SKIP6 #1	SKIP5 #1	SKIP4 #1	SKIP3 #1	ZAE #1 SKIP2 #1	YAE #1 SKIP8 #1	XAE #1 SKIP7 #1	(Mseries)
X005									
X006									
X007		*DEC7 #2	*DEC6 #2	*DEC5 #2	*DEC4 #2	*DEC3 #2	*DEC2 #2	*DEC1 #2	
X008				*ESP					
X009		*DEC7 #1	*DEC6 #1	*DEC5 #1	*DEC4 #1	*DEC3 #1	*DEC2 #1	*DEC1 #1	
X010									
X011									
X012									
X013	SKIP #2	SKIP6 #2	–MIT2 #2 SKIP5 #2	+MIT2 #2 SKIP4 #2	–MIT1 #2 SKIP3 #2	+MIT1 #2 SKIP2 #2	ZAE #2 SKIP8 #2	XAE #2 SKIP7 #2	(Tseries)
	SKIP #2	SKIP6 #2	SKIP5 #2	SKIP4 #2	SKIP3 #2	ZAE #2 SKIP2 #2	YAE #2 SKIP8 #2	XAE #2 SKIP7 #2	(Mseries)

PMC → CNC

Address	Bit number 7	6	5	4	3	2	1	0
G000	ED7 #1	ED6 #1	ED5 #1	ED4 #1	ED3 #1	ED2 #1	ED1 #1	ED0 #1
G001	ED15 #1	ED14 #1	ED13 #1	ED12 #1	ED11 #1	ED10 #1	ED9 #1	ED8 #1
G002	ESTB #1	EA6 #1	EA5 #1	EA4 #1	EA3 #1	EA2 #1	EA1 #1	EA0 #1
G003								
G004			MFIN3 #1	MFIN2 #1	FIN #1			
G005	BFIN #1	AFL #1		BFIN #1	TFIN #1	SFIN #1	EFIN #1	MFIN #1
G006		SKIPP #1		OVC #1		*ABSM #1		SRN #1
G007	RLSOT #1	EXLM #1	*FLWP #1			ST #1	STLK #1	RVS #1
G008	ERS #1	RRW #1	*SP #1	*ESP #1				*IT #1
G009				PN16 #1	PN8 #1	PN4 #1	PN2 #1	PN1 #1
G010	*JV7 #1	*JV6 #1	*JV5 #1	*JV4 #1	*JV3 #1	*JV2 #1	*JV1 #1	*JV0 #1
G011	*JV15 #1	*JV14 #1	*JV13 #1	*JV12 #1	*JV11 #1	*JV10 #1	*JV9 #1	*JV8 #1
G012	*FV7 #1	*FV6 #1	*FV5 #1	*FV4 #1	*FV3 #1	*FV2 #1	*FV1 #1	*FV0 #1
G013	*AFV7 #1	*AFV6 #1	*AFV5 #1	*AFV4 #1	*AFV3 #1	*AFV2 #1	*AFV1 #1	*AFV0 #1
G014							ROV2 #1	ROV1 #1
G015								
G016	F1D #1							
G017								
G018	HS2D #1	HS2C #1	HS2B #1	HS2A #1	HS1D #1	HS1C #1	HS1B #1	HS1A #1
G019	RT #1		MP2 #1	MP1 #1	HS3D #1	HS3C #1	HS3B #1	HS3A #1
G020								
G021								
G022								
G023	ALNGH #1							
G024								

	7	6	5	4	3	2	1	0
G025								
G026								
G027	CON #1		*SSTP3#1	*SSTP2#1	*SSTP1#1	SWS3 #1	SWS2 #1	SWS1 #1
G028	PC2SLC#1	SPSTP#1	*SCPF#1	*SUCPF#1		GR2 #1	GR1 #1	
G029		*SSTP#1	SOR #1	SAR #1		GR31 #1		GR21 #1
G030	SOV7 #1	SOV6 #1	SOV5 #1	SOV4 #1	SOV3 #1	SOV2 #1	SOV1 #1	SOV0 #1
G031								
G032	R08I #1	R07I #1	R06I #1	R05I #1	R04I #1	R03I #1	R02I #1	R01I #1
G033	SIND #1	SSIN #1	SGN #1		R12I #1	R11I #1	R10I #1	R09I #1
G034	R08I2 #1	R07I2 #1	R06I2 #1	R05I2 #1	R04I2 #1	R03I2 #1	R02I2 #1	R01I2 #1
G035	SIND2#1	SSIN2#1	SGN2#1		R12I2 #1	R11I2 #1	R10I2 #1	R09I2 #1
G036	R08I3 #1	R07I3 #1	R06I3 #1	R05I3 #1	R04I3 #1	R03I3 #1	R02I3 #1	R01I3 #1
G037	SIND3#1	SSIN3#1	SGN3#1		R12I3 #1	R11I3 #1	R10I3 #1	R09I3 #1
G038	*BECLP #1	*BEUCP #1			SPPHS / SPPHS #1	SPSYC / SPSYC #1		*PLSST#1
G039	GOQSM#1	WOQSM#1	OFN5 #1	OFN4 #1	OFN3 #1	OFN2 #1	OFN1 #1	OFN0 #1
G040	WOSET#1	PRC #1						OFN6 #1
G041	HS2ID#1	HS2IC#1	HS2IB#1	HS2IA#1	HS1ID#1	HS1IC#1	HS1IB#1	HS1IA#1
G042					HS3ID#1	HS3IC#1	HS3IB#1	HS3IA#1
G043	ZRN #1		DNCI #1			MD4 #1	MD2 #1	MD1 #1
G044							MLK #1	BDT1 #1
G045	BDT9 #1	BDT8 #1	BDT7 #1	BDT6 #1	BDT5 #1	BDT4 #1	BDT3 #1	BDT2 #1
G046	DRN #1	KEY4 #1	KEY3 #1	KEY2 #1	KEY1 #1		SBK #1	
G047	TL128 #1	TL64 #1	TL32 #1	TL16 #1	TL08 #1	TL04 #1	TL02 #1	TL01 #1
G048	TLRST#1	TLRSTI#1	TLSKP#1					TL256 #1
G049	*TLV7 #1	*TLV6 #1	*TLV5 #1	*TLV4#1	*TLV3 #1	*TLV2 #1	*TLV1 #1	*TLV0 #1

	7	6	5	4	3	2	1	0
G050							*TLV9 #1	*TLV8 #1
G051								
G052								
G053	CDZ #1	SMZ #1			UINT #1			TMRON#1
G054	UI007 #1	UI006 #1	UI005 #1	UI004 #1	UI003 #1	UI002 #1	UI001 #1	UI000 #1
G055	UI015 #1	UI014 #1	UI013 #1	UI012 #1	UI011 #1	UI010 #1	UI009 #1	UI008 #1
G056								
G057								
G058					EXWT #1	EXSTP #1	EXRD #1	MINP#1
G059							TRRTN #1	TRESC #1
G060	*TSB #1							
G061			RGTSP2#1	RGTSP1#1				RGTAP#1
G062								
G063					SLSPB	SLSPA	NOWT	HEAD
G064					SLPCB	SLPCA		
G065								
G066	EKSET						ENBKY	IGNVRY#1
G067								
G068								
G069								
G070	MRDYA#1	ORCMA#1	SFRA #1	SRVA #1	CTH1A#1	CTH2A#1	TLMHA#1	TLMLA#1
G071	RCHA #1	RSLA #1		SOCNA#1	MCFNA#1	SPSLA#1	*ESPA#1	ARSTA#1
G072								
G073								
G074	MRDYB#1	ORCMB#1	SFRB #1	SRVB #1	CTH1B#1	CTH2B#1	TLMHB#1	TLMLB#1

	7	6	5	4	3	2	1	0
G075	RCHB#1	RSLB #1		SOCNB#1	MCFNB#1	SPSLB#1	*ESPB#1	ARSTB#1
G076								
G077								
G078	SHA07#1	SHA06#1	SHA05#1	SHA04#1	SHA03#1	SHA02#1	SHA01#1	SHA00#1
G079					SHA11#1	SHA10#1	SHA09#1	SHA08#1
G080	SHB07#1	SHB06#1	SHB05#1	SHB04#1	SHB03#1	SHB02#1	SHB01#1	SHB00#1
G081					SHB11#1	SHB10#1	SHB09#1	SHB08#1
G082	Reserved for order made macro							
G083	Reserved for order made macro							
G084								
G085								
G086								
G087								
G088								
G089								
G090								
G091								
G092								
G093								
G094								
G095								
G096	HROV#1	*HROV6#1	*HROV5#1	*HROV4#1	*HROV3#1	*HROV2#1	*HROV1#1	*HROV0#1
G097								
G098	EKC7	EKC6	EKC5	EKC4	EKC3	EKC2	EKC1	EKC0
G099								

	7	6	5	4	3	2	1	0
G100		+J7 #1	+J6 #1	+J5 #1	+J4 #1	+J3 #1	+J2 #1	+J1 #1
G101								
G102		–J7 #1	–J6 #1	–J5 #1	–J4 #1	–J3 #1	–J2 #1	–J1 #1
G103								
G104								
G105								
G106		MI7 #1	MI6 #1	MI5 #1	MI4 #1	MI3 #1	MI2 #1	MI1 #1
G107								
G108		MLK7 #1	MLK6 #1	MLK5 #1	MLK4 #1	MLK3 #1	MLK2 #1	MLK1 #1
G109								
G110		+LM7 #1	+LM6 #1	+LM5 #1	+LM4 #1	+LM3 #1	+LM2 #1	+LM1 #1
G111								
G112		–LM7 #1	–LM6 #1	–LM5 #1	–LM4 #1	–LM3 #1	–LM2 #1	–LM1 #1
G113								
G114		*+L7 #1	*+L6 #1	*+L5 #1	*+L4 #1	*+L3 #1	*+L2 #1	*+L1 #1
G115								
G116		*–L7 #1	*–L6 #1	*–L5 #1	*–L4 #1	*–L3 #1	*–L2 #1	*–L1 #1
G117								
G118		*+ED7 #1	*+ED6 #1	*+ED5 #1	*+ED4 #1	*+ED3 #1	*+ED2 #1	*+ED1 #1
G119								
G120		*–ED7 #1	*–ED6 #1	*–ED5 #1	*–ED4 #1	*–ED3 #1	*–ED2 #1	*–ED1 #1
G121								
G122		PK7 #1	PK6 #1	PK5 #1	PK4 #1	PK3 #1	PK2 #1	PK1 #1
G123								
G124		DTCH7 #1	DTCH6 #1	DTCH5 #1	DTCH4 #1	DTCH3 #1	DTCH2 #1	DTCH1 #1

	7	6	5	4	3	2	1	0
G125								
G126		SVF7 #1	SVF6 #1	SVF5 #1	SVF4 #1	SVF3 #1	SVF2 #1	SVF1 #1
G127								
G128		MIX7	MIX6	MIX5	MIX4	MIX3	MIX2	MIX1
G129								
G130		*IT7 #1	*IT6 #1	*IT5 #1	*IT4 #1	*IT3 #1	*IT2 #1	*IT1 #1
G131								
G132					+MIT4 #1	+MIT3 #1	+MIT2 #1	+MIT1 #1
G133								
G134					–MIT4 #1	–MIT3 #1	–MIT2 #1	–MIT1 #1
G135								
G136		EAX7 #1	EAX6 #1	EAX5 #1	EAX4 #1	EAX3 #1	EAX2 #1	EAX1 #1
G137								
G138		SYNC7#1	SYNC6#1	SYNC5#1	SYNC4#1	SYNC3#1	SYNC2#1	SYNC1#1
G139								
G140		SYNCJ7#1	SYNCJ6#1	SYNCJ5#1	SYNCJ4#1	SYNCJ3#1	SYNCJ2#1	SYNCJ1#1
G141								
G142	EBUFA#1	ECLRA#1	ESTPA#1	ESOFA#1	ESBKA#1	EMBUFA#1		EFINA#1
G143	EMSBKA#1	EC6A #1	EC5A #1	EC4A #1	EC3A #1	EC2A #1	EC1A #1	EC0A #1
G144	EIF7A #1	EIF6A #1	EIF5A #1	EIF4A #1	EIF3A #1	EIF2A #1	EIF1A #1	EIF0A #1
G145	EIF15A#1	EIF14A#1	EIF13A#1	EIF12A#1	EIF11A#1	EIF10A#1	EIF9A #1	EIF8A #1
G146	EID7A#1	EID6A#1	EID5A#1	EID4A#1	EID3A#1	EID2A#1	EID1A#1	EID0A#1
G147	EID15A#1	EID14A#1	EID13A#1	EID12A#1	EID11A#1	EID10A#1	EID9A#1	EID8A#1
G148	EID23A#1	EID22A#1	EID21A#1	EID20A#1	EID19A#1	EID18A#1	EID17A#1	EID16A#1
G149	EID31A#1	EID30A#1	EID29A#1	EID28A#1	EID27A#1	EID26A#1	EID25A#1	EID24A#1

	7	6	5	4	3	2	1	0
G150	DRNE#1	RTE #1	OVCE#1				ROV2E#1	ROV1E#1
G151	*FV7E#1	*FV6E#1	*FV5E#1	*FV4E#1	*FV3E#1	*FV2E#1	*FV1E#1	*FV0E#1
G152								
G153								
G154	EBUFB#1	ECLRB#1	ESTPB#1	ESOFB#1	ESBKB#1	EMBUFB#1		EFINB#1
G155	EMSBKB#1	EC6B #1	EC5B #1	EC4B #1	EC3B #1	EC2B #1	EC1B #1	EC0B #1
G156	EIF7B#1	EIF6B#1	EIF5B#1	EIF4B#1	EIF3B#1	EIF2B#1	EIF1B#1	EIF0B#1
G157	EIF15B#1	EIF14B#1	EIF13B#1	EIF12B#1	EIF11B#1	EIF10B#1	EIF9B#1	EIF8B#1
G158	EID7B#1	EID6B#1	EID5B#1	EID4B#1	EID3B#1	EID2B#1	EID1B#1	EID0B#1
G159	EID15B#1	EID14B#1	EID13B#1	EID12B#1	EID11B#1	EID10B#1	EID9B#1	EID8B#1
G160	EID23B#1	EID22B#1	EID21B#1	EID20B#1	EID19B#1	EID18B#1	EID17B#1	EID16B#1
G161	EID31B#1	EID30B#1	EID29B#1	EID28B#1	EID27B#1	EID26B#1	EID25B#1	EID24B#1
G162								
G163								
G164								
G165								
G166	EBUFC#1	ECLRC#1	ESTPC#1	ESOFC#1	ESBKC#1	EMBUFC#1		EFINC#1
G167	EMSBKC#1	EC6C #1	EC5C #1	EC4C #1	EC3C #1	EC2C #1	EC1C #1	EC0C #1
G168	EIF7C#1	EIF6C#1	EIF5C#1	EIF4C#1	EIF3C#1	EIF2C#1	EIF1C#1	EIF0C#1
G169	EIF15C#1	EIF14C#1	EIF13C#1	EIF12C#1	EIF11C#1	EIF10C#1	EIF9C#1	EIF8C#1
G170	EID7C#1	EID6C#1	EID5C#1	EID4C#1	EID3C#1	EID2C#1	EID1C#1	EID0C#1
G171	EID15C#1	EID14C#1	EID13C#1	EID12C#1	EID11C#1	EID10C#1	EID9C#1	EID8C#1
G172	EID23C#1	EID22C#1	EID21C#1	EID20C#1	EID19C#1	EID18C#1	EID17C#1	EID16C#1
G173	EID31C#1	EID30C#1	EID29C#1	EID28C#1	EID27C#1	EID26C#1	EID25C#1	EID24C#1
G174								

	7	6	5	4	3	2	1	0
G175								
G176								
G177								
G178	EBUFD[#1]	ECLRD[#1]	ESTPD[#1]	ESOFD[#1]	ESBKD[#1]	EMBUFD[#1]		EFIND[#1]
G179	EMSBKD[#1]	EC6D [#1]	EC5D [#1]	EC4D [#1]	EC3D [#1]	EC2D [#1]	EC1D [#1]	EC0D [#1]
G180	EIF7D[#1]	EIF6D[#1]	EIF5D[#1]	EIF4D[#1]	EIF3D[#1]	EIF2D[#1]	EIF1D[#1]	EIF0D[#1]
G181	EIF15D[#1]	EIF14D[#1]	EIF13D[#1]	EIF12D[#1]	EIF11D[#1]	EIF10D[#1]	EIF9D[#1]	EIF8D[#1]
G182	EID7D[#1]	EID6D[#1]	EID5D[#1]	EID4D[#1]	EID3D[#1]	EID2D[#1]	EID1D[#1]	EID0D[#1]
G183	EID15D[#1]	EID14D[#1]	EID13D[#1]	EID12D[#1]	EID11D[#1]	EID10D[#1]	EID9D[#1]	EID8D[#1]
G184	EID23D[#1]	EID22D[#1]	EID21D[#1]	EID20D[#1]	EID19D[#1]	EID18D[#1]	EID17D[#1]	EID16D[#1]
G185	EID31D[#1]	EID30D[#1]	EID29D[#1]	EID28D[#1]	EID27D[#1]	EID26D[#1]	EID25D[#1]	EID24D[#1]
G186								
G187								
G188								
G189								
G190		OVLS7[#1]	OVLS6[#1]	OVLS5[#1]	OVLS4[#1]	OVLS3[#1]	OVLS2[#1]	OVLS1[#1]
G191								

	7	6	5	4	3	2	1	0
G192		IGVRY7#1	IGVRY6#1	IGVRY5#1	IGVRY4#1	IGVRY3#1	IGVRY2#1	IGVRY1#1
G193								
G194								
G195								
G196								
G197								
G198		NPOS7#1	NPOS6#1	NPOS5#1	NPOS4#1	NPOS3#1	NPOS2#1	NPOS1#1
G199								
G200								
G201								
G202								
G203								
G204								
G205								
G206								
G207								
G208								
G209								
G210								
G211								
G212								
G213								
G214								
G215								
G216								

	7	6	5	4	3	2	1	0
G217								
G218								
G219								

PMC → CNC

Address	Bit 7	6	5	4	3	2	1	0
G1000	ED7[#2]	ED6[#2]	ED5[#2]	ED4[#2]	ED3[#2]	ED2[#2]	ED1[#2]	ED0[#2]
G1001	ED15[#2]	ED14[#2]	ED13[#2]	ED12[#2]	ED11[#2]	ED10[#2]	ED9[#2]	ED8[#2]
G1002	ESTB[#2]	EA6[#2]	EA5[#2]	EA4[#2]	EA3[#2]	EA2[#2]	EA1[#2]	EA0[#2]
G1003								
G1004			MFIN3[#2]	MFIN2[#2]	FIN[#2]			
G1005	BFIN[#2]	AFL[#2]		BFIN[#2]	TFIN[#2]	SFIN[#2]	EFIN[#2]	MFIN[#2]
G1006		SKIPP[#2]		OVC[#2]		*ABSM[#2]		SRN[#2]
G1007	RLSOT[#2]	EXLM[#2]	*FLWP[#2]			ST[#2]	STLK[#2]	RVS[#2]
G1008	ERS[#2]	RRW[#2]	*SP[#2]	*ESP[#2]				*IT[#2]
G1009				PN16[#2]	PN8[#2]	PN4[#2]	PN2[#2]	PN1[#2]
G1010	*JV7[#2]	*JV6[#2]	*JV5[#2]	*JV4[#2]	*JV3[#2]	*JV2[#2]	*JV1[#2]	*JV0[#2]
G1011	*JV15[#2]	*JV14[#2]	*JV13[#2]	*JV12[#2]	*JV11[#2]	*JV10[#2]	*JV9[#2]	*JV8[#2]
G1012	*FV7[#2]	*FV6[#2]	*FV5[#2]	*FV4[#2]	*FV3[#2]	*FV2[#2]	*FV1[#2]	*FV0[#2]
G1013	*AFV7[#2]	*AFV6[#2]	*AFV5[#2]	*AFV4[#2]	*AFV3[#2]	*AFV2[#2]	*AFV1[#2]	*AFV0[#2]
G1014							ROV2[#2]	ROV1[#2]
G1015								
G1016	F1D[#2]							
G1017								
G1018	HS2D[#2]	HS2C[#2]	HS2B[#2]	HS2A[#2]	HS1D[#2]	HS1C[#2]	HS1B[#2]	HS1A[#2]
G1019	RT[#2]		MP2[#2]	MP1[#2]	HS3D[#2]	HS3C[#2]	HS3B[#2]	HS3A[#2]
G1020								
G1021								
G1022								
G1023	ALNGH[#2]							
G1024								

	7	6	5	4	3	2	1	0
G1025								
G1026								
G1027	CON#2		*SSTP3#2	*SSTP2#2	*SSTP1#2	SWS3#2	SWS2#2	SWS1#2
G1028	PC2SLC#2	SPSTP#2	*SCPF#2	*SUCPF#2		GR2#2	GR1#2	
G1029		*SSTP#2	SOR#2	SAR#2		GR31#2		GR21#2
G1030	SOV7#2	SOV6#2	SOV5#2	SOV4#2	SOV3#2	SOV2#2	SOV1#2	SOV0#2
G1031								
G1032	R08I#2	R07I#2	R06I#2	R05I#2	R04I#2	R03I#2	R02I#2	R01I#2
GI033	SIND#2	SSIN#2	SGN#2		R12I#2	R11I#2	R10I#2	R09I#2
G1034	R08I2#2	R07I2#2	R06I2#2	R05I2#2	R04I2#2	R03I2#2	R02I2#2	R01I2#2
G1035	SIND2#2	SSIN2#2	SGN2#2		R12I2#2	R11I2#2	R10I2#2	R09I2#2
G1036	R08I3#2	R07I3#2	R06I3#2	R05I3#2	R04I3#2	R03I3#2	R02I3#2	R01I3#2
G1037	SIND3#2	SSIN3#2	SGN3#2		R12I3#2	R11I3#2	R10I3#2	R09I3#2
G1038	*BECLP #2	*BEUCP #2			SPPHS#2	SPSYC#2		*PLSST#2
G1039	GOQSM#2	WOQSM#2	OFN5#2	OFN4#2	OFN3#2	OFN2#2	OFN1#2	OFN0#2
G1040	WOSET#2	PRC#2						OFN6#2
G1041	HS2ID#2	HS2IC#2	HS2IB#2	HS2IA#2	HS1ID#2	HS1IC#2	HS1IB#2	HS1IA#2
G1042					HS3ID#2	HS3IC#2	HS3IB#2	HS3IA#2
G1043	ZRN#2		DNCI#2			MD4#2	MD2#2	MD1#2
G1044							MLK#2	BDT1#2
G1045	BDT9#2	BDT8#2	BDT7#2	BDT6#2	BDT5#2	BDT4#2	BDT3#2	BDT2#2
G1046	DRN#2	KEY4#2	KEY3#2	KEY2#2	KEY1#2		SBK#2	
G1047	TL128#2	TL64#2	TL32#2	TL16#2	TL08#2	TL04#2	TL02#2	TL01#2
G1048	TLRST#2	TLRSTI#2	TLSKP#2					TL256#2
G1049	*TLV7 #2	*TLV6 #2	*TLV5 #2	*TLV4#2	*TLV3 #2	*TLV2 #2	*TLV1 #2	*TLV0 #2

	7	6	5	4	3	2	1	0
G1050							*TLV9 #2	*TLV8 #2
G1051								
G1052								
G1053	CDZ#2	SMZ#2			UINT#2			TMRON#2
G1054	UI007#2	UI006#2	UI005#2	UI004#2	UI003#2	UI002#2	UI001#2	UI000#2
G1055	UI015#2	UI014#2	UI013#2	UI012#2	UI011#2	UI010#2	UI009#2	UI008#2
G1056								
G1057								
G1058					EXWT#2	EXSTP#2	EXRD#2	MINP#2
G1059							TRRTN#2	TRESC#2
G1060	*TSB#2							
G1061			RGTSP2#2	RGTSP1#2				RGTAP#2
G1062								
G1063								
G1064								
G1065								
G1066								IGNVRY#2
G1067								
G1068								
G1069								
G1070	MRDYA#2	ORCMA#2	SFRA#2	SRVA#2	CTH1A#2	CTH2A#2	TLMHA#2	TLMLA#2
G1071	RCHA#2	RSLA#2		SOCNA#2	MCFNA#2	SPSLA#2	*ESPA#2	ARSTA#2
G1072								
G1073								
G1074	MRDYB#2	ORCMB#2	SFRB#2	SRVB#2	CTH1B#2	CTH2B#2	TLMHB#2	TLMLB#2

	7	6	5	4	3	2	1	0
G1075	RCHB#2	RSLB#2		SOCNB#2	MCFNB#2	SPSLB#2	*ESPB#2	ARSTB#2
G1076								
G1077								
G1078	SHA07#2	SHA06#2	SHA05#2	SHA04#2	SHA03#2	SHA02#2	SHA01#2	SHA00#2
G1079					SHA11#2	SHA10#2	SHA09#2	SHA08#2
G1080	SHB07#2	SHB06#2	SHB05#2	SHB04#2	SHB03#2	SHB02#2	SHB01#2	SHB00#2
G1081					SHB11#2	SHB10#2	SHB09#2	SHB08#2
G1082	Reserved for order made macro							
G1083	Reserved for order made macro							
G1084								
G1085								
G1086								
G1087								
G1088								
G1089								
G1090								
G1091								
G1092								
G1093								
G1094								
G1095								
G1096	HROV#2	*HROV6#2	*HROV5#2	*HROV4#2	*HROV3#2	*HROV2#2	*HROV1#2	*HROV0#2
G1097								
G1098								
G1099								

	7	6	5	4	3	2	1	0
G1100		+J7#2	+J6#2	+J5#2	+J4#2	+J3#2	+J2#2	+J1#2
G1101								
G1102		–J7#2	–J6#2	–J5#2	–J4#2	–J3#2	–J2#2	–J1#2
G1103								
G1104								
G1105								
G1106		MI7#2	MI6#2	MI5#2	MI4#2	MI3#2	MI2#2	MI1#2
G1107								
G1108		MLK7#2	MLK6#2	MLK5#2	MLK4#2	MLK3#2	MLK2#2	MLK1#2
G1109								
G1110		+LM7 #2	+LM6 #2	+LM5 #2	+LM4 #2	+LM3 #2	+LM2 #2	+LM1 #2
G1111								
G1112		–LM7 #2	–LM6 #2	–LM5 #2	–LM4 #2	–LM3 #2	–LM2 #2	–LM1 #2
G1113								
G1114		*+L7#2	*+L6#2	*+L5#2	*+L4#2	*+L3#2	*+L2#2	*+L1#2
G1115								
G1116		*–L7#2	*–L6#2	*–L5#2	*–L4#2	*–L3#2	*–L2#2	*–L1#2
G1117								
G1118		*+ED7#2	*+ED6#2	*+ED5#2	*+ED4#2	*+ED3#2	*+ED2#2	*+ED1#2
G1119								
G1120		*–ED7#2	*–ED6#2	*–ED5#2	*–ED4#2	*–ED3#2	*–ED2#2	*–ED1#2
G1121								
G1122		PK7#2	PK6#2	PK5#2	PK4#2	PK3#2	PK2#2	PK1#2
G1123								
G1124		DTCH7#2	DTCH6#2	DTCH5#2	DTCH4#2	DTCH3#2	DTCH2#2	DTCH1#2

	7	6	5	4	3	2	1	0
G1125								
G1126		SVF7#2	SVF6#2	SVF5#2	SVF4#2	SVF3#2	SVF2#2	SVF1#2
G1127								
G1128								
G1129								
G1130		*IT7#2	*IT6#2	*IT5#2	*IT4#2	*IT3#2	*IT2#2	*IT1#2
G1131								
G1132					+MIT4 #2	+MIT3#2	+MIT2 #2	+MIT1 #2
G1133								
G1134					–MIT4 #2	–MIT3 #2	–MIT2 #2	–MIT1 #2
G1135								
G1136		EAX7#2	EAX6#2	EAX5#2	EAX4#2	EAX3#2	EAX2#2	EAX1#2
G1137								
G1138		SYNC7#2	SYNC6#2	SYNC5#2	SYNC4#2	SYNC3#2	SYNC2#2	SYNC1#2
G1139								
G1140		SYNCJ7#2	SYNCJ6#2	SYNCJ5#2	SYNCJ4#2	SYNCJ3#2	SYNCJ2#2	SYNCJ1#2
G1141								
G1142	EBUFA#2	ECLRA#2	ESTPA#2	ESOFA#2	ESBKA#2	EMBUFA#2		EFINA#2
G1143	EMSBKA#2	EC6A#2	EC5A#2	EC4A#2	EC3A#2	EC2A#2	EC1A#2	EC0A#2
G1144	EIF7A#2	EIF6A#2	EIF5A#2	EIF4A#2	EIF3A#2	EIF2A#2	EIF1A#2	EIF0A#2
G1145	EIF15A#2	EIF14A#2	EIF13A#2	EIF12A#2	EIF11A#2	EIF10A#2	EIF9A#2	EIF8A#2
G1146	EID7A#2	EID6A#2	EID5A#2	EID4A#2	EID3A#2	EID2A#2	EID1A#2	EID0A#2
G1147	EID15A#2	EID14A#2	EID13A#2	EID12A#2	EID11A#2	EID10A#2	EID9A#2	EID8A#2
G1148	EID23A#2	EID22A#2	EID21A#2	EID20A#2	EID19A#2	EID18A#2	EID17A#2	EID16A#2
G1149	EID31A#2	EID30A#2	EID29A#2	EID28A#2	EID27A#2	EID26A#2	EID25A#2	EID24A#2

	7	6	5	4	3	2	1	0
G1150	DRNE#2	RTE#2	OVCE#2				ROV2E#2	ROV1E#2
G1151	*FV7E#2	*FV6E#2	*FV5E#2	*FV4E#2	*FV3E#2	*FV2E#2	*FV1E#2	*FV0E#2
G1152								
G1153								
G1154	EBUFB#2	ECLRB#2	ESTPB#2	ESOFB#2	ESBKB#2	EMBUFB#2		EFINB#2
G1155	EMSBKB#2	EC6B#2	EC5B#2	EC4B#2	EC3B#2	EC2B#2	EC1B#2	EC0B#2
G1156	EIF7B#2	EIF6B#2	EIF5B#2	EIF4B#2	EIF3B#2	EIF2B#2	EIF1B#2	EIF0B#2
G1157	EIF15B#2	EIF14B#2	EIF13B#2	EIF12B#2	EIF11B#2	EIF10B#2	EIF9B#2	EIF8B#2
G1158	EID7B#2	EID6B#2	EID5B#2	EID4B#2	EID3B#2	EID2B#2	EID1B#2	EID0B#2
G1159	EID15B#2	EID14B#2	EID13B#2	EID12B#2	EID11B#2	EID10B#2	EID9B#2	EID8B#2
G1160	EID23B#2	EID22B#2	EID21B#2	EID20B#2	EID19B#2	EID18B#2	EID17B#2	EID16B#2
G1161	EID31B#2	EID30B#2	EID29B#2	EID28B#2	EID27B#2	EID26B#2	EID25B#2	EID24B#2
G1162								
G1163								
G1164								
G1165								
G1166	EBUFC#2	ECLRC#2	ESTPC#2	ESOFC#2	ESBKC#2	EMBUFC#2		EFINC#2
G1167	EMSBKC#2	EC6C#2	EC5C#2	EC4C#2	EC3C#2	EC2C#2	EC1C#2	EC0C#2
G1168	EIF7C#2	EIF6C#2	EIF5C#2	EIF4C#2	EIF3C#2	EIF2C#2	EIF1C#2	EIF0C#2
G1169	EIF15C#2	EIF14C#2	EIF13C#2	EIF12C#2	EIF11C#2	EIF10C#2	EIF9C#2	EIF8C#2
G1170	EID7C#2	EID6C#2	EID5C#2	EID4C#2	EID3C#2	EID2C#2	EID1C#2	EID0C#2
G1171	EID15C#2	EID14C#2	EID13C#2	EID12C#2	EID11C#2	EID10C#2	EID9C#2	EID8C#2
G1172	EID23C#2	EID22C#2	EID21C#2	EID20C#2	EID19C#2	EID18C#2	EID17C#2	EID16C#2
G1173	EID31C#2	EID30C#2	EID29C#2	EID28C#2	EID27C#2	EID26C#2	EID25C#2	EID24C#2
G1174								

	7	6	5	4	3	2	1	0
G1175								
G1176								
G1177								
G1178	EBUFD#2	ECLRD#2	ESTPD#2	ESOFD#2	ESBKD#2	EMBUFD#2		EFIND#2
G1179	EMSBKD#2	EC6D#2	EC5D#2	EC4D#2	EC3D#2	EC2D#2	EC1D#2	EC0D#2
G1180	EIF7D#2	EIF6D#2	EIF5D#2	EIF4D#2	EIF3D#2	EIF2D#2	EIF1D#2	EIF0D#2
G1181	EIF15D#2	EIF14D#2	EIF13D#2	EIF12D#2	EIF11D#2	EIF10D#2	EIF9D#2	EIF8D#2
G1182	EID7D#2	EID6D#2	EID5D#2	EID4D#2	EID3D#2	EID2D#2	EID1D#2	EID0D#2
G1183	EID15D#2	EID14D#2	EID13D#2	EID12D#2	EID11D#2	EID10D#2	EID9D#2	EID8D#2
G1184	EID23D#2	EID22D#2	EID21D#2	EID20D#2	EID19D#2	EID18D#2	EID17D#2	EID16D#2
G1185	EID31D#2	EID30D#2	EID29D#2	EID28D#2	EID27D#2	EID26D#2	EID25D#2	EID24D#2
G1186								
G1187								
G1188								
G1189								
G1190		OVLS7#2	OVLS6#2	OVLS5#2	OVLS4#2	OVLS3#2	OVLS2#2	OVLS1#2
G1191								
G1192		IGVRY7#2	IGVRY6#2	IGVRY5#2	IGVRY4#2	IGVRY3#2	IGVRY2#2	IGVRY1#2
G1193								
G1194								
G1195								
G1196								
G1197								
G1198		NPOS7#2	NPOS6#2	NPOS5#2	NPOS4#2	NPOS3#2	NPOS2#2	NPOS1#2
G1199								

	7	6	5	4	3	2	1	0
G1200								
G1201								
G1202								
G1203								
G1204								
G1205								
G1206								
G1207								
G1208								
G1209								
G1210								
G1211								
G1212								
G1213								
G1214								
G1215								
G1216								
G1217								
G1218								
G1219								

CNC → PMC

Address	7	6	5	4	3	2	1	0
F000	OP#1	SA#1	STL#1	SPL#1				RWD#1
F001	MA#1		TAP#1	ENB#1	DEN#1	BAL#1	RST#1	AL#1
F002	MDRN#1	CUT#1		SRNMV#1	THRD#1	CSS#1	RPDO#1	INCH#1
F003	MTCHIN#1	MEDT#1	MMEM#1	MRMT#1	MMDI#1	MJ#1	MH#1	MINC#1
F004			MREF#1	MAFL#1	MSBK#1	MABSM#1	MMLK#1	MBDT1#1
F005	MBDT9#1	MBDT8#1	MBDT7#1	MBDT6#1	MBDT5#1	MBDT4#1	MBDT3#1	MBDT2#1
F006								
F007	BF#1			BF#1	TF#1	SF#1	EFD#1	MF#1
F008			MF3#1	MF2#1				EF#1
F009	DM00#1	DM01#1	DM02#1	DM30#1				
F010	M07#1	M06#1	M05#1	M04#1	M03#1	M02#1	M01#1	M00#1
F011	M15#1	M14#1	M13#1	M12#1	M11#1	M10#1	M09#1	M08#1
F012	M23#1	M22#1	M21#1	M20#1	M19#1	M18#1	M17#1	M16#1
F013	M31#1	M30#1	M29#1	M28#1	M27#1	M26#1	M25#1	M24#1
F014	M207#1	M206#1	M205#1	M204#1	M203#1	M202#1	M201#1	M200#1
F015	M215#1	M214#1	M213#1	M212#1	M211#1	M210#1	M209#1	M208#1
F016	M307#1	M306#1	M305#1	M304#1	M303#1	M302#1	M301#1	M300#1
F017	M315#1	M314#1	M313#1	M312#1	M311#1	M310#1	M309#1	M308#1
F018								
F019								
F020								
F021								
F022	S07#1	S06#1	S05#1	S04#1	S03#1	S02#1	S01#1	S00#1
F023	S15#1	S14#1	S13#1	S12#1	S11#1	S10#1	S09#1	S08#1
F024	S23#1	S22#1	S21#1	S20#1	S19#1	S18#1	S17#1	S16#1

	7	6	5	4	3	2	1	0
F025	S31#1	S30#1	S29#1	S28#1	S27#1	S26#1	S25#1	S24#1
F026	T07#1	T06#1	T05#1	T04#1	T03#1	T02#1	T01#1	T00#1
F027	T15#1	T14#1	T13#1	T12#1	T11#1	T10#1	T09#1	T08#1
F028	T23#1	T22#1	T21#1	T20#1	T19#1	T18#1	T17#1	T16#1
F029	T31#1	T30#1	T29#1	T28#1	T27#1	T26#1	T25#1	T24#1
F030	B07#1	B06#1	B05#1	B04#1	B03#1	B02#1	B01#1	B00#1
F031	B15#1	B14#1	B13#1	B12#1	B11#1	B10#1	B09#1	B08#1
F032	B23#1	B22#1	B21#1	B20#1	B19#1	B18#1	B17#1	B16#1
F033	B31#1	B30#1	B29#1	B28#1	B27#1	B26#1	B25#1	B24#1
F034						GR3O#1	GR2O#1	GR1O#1
F035								SPAL#1
F036	R08O#1	R07O#1	R06O#1	R05O#1	R04O#1	R03O#1	R02O#1	R01O#1
F037					R12O#1	R11O#1	R10O#1	R09O#1
F038					ENB3#1	ENB2#1	SUCLP#1	SCLP#1
F039								
F040	AR7#1	AR6#1	AR5#1	AR4#1	AR3#1	AR2#1	AR1#1	AR0#1
F041	AR15#1	AR14#1	AR13#1	AR12#1	AR11#1	AR10#1	AR09#1	AR08#1
F042								
F043								
F044				SYCAL SYCAL#1	FSPPH FSPPH#1	FSPSY FSPSY#1	FSCSL#1	
F045	ORARA#1	TLMA#1	LDT2A#1	LDT1A#1	SARA#1	SDTA#1	SSTA#1	ALMA#1
F046					RCFNA#1	RCHPA#1	CFINA#1	CHPA#1
F047								
F048								
F049	ORARB#1	TLMB#1	LDT2B#1	LDT1B#1	SARB#1	SDTB#1	SSTB#1	ALMB#1

	7	6	5	4	3	2	1	0
F050					RCFNB[#1]	RCHPB[#1]	CFINB[#1]	CHPB[#1]
F051								
F052								
F053	EKENB			BGEACT[#1]	RPALM[#1]	RPBSY[#1]	PRGDPL	INHKY
F054	UO007[#1]	UO006[#1]	UO005[#1]	UO004[#1]	UO003[#1]	UO002[#1]	UO001[#1]	UO000[#1]
F055	UO015[#1]	UO014[#1]	UO013[#1]	UO012[#1]	UO011[#1]	UO010[#1]	UO009[#1]	UO008[#1]
F056	UO107[#1]	UO106[#1]	UO105[#1]	UO104[#1]	UO103[#1]	UO102[#1]	UO101[#1]	UO100[#1]
F057	UO115[#1]	UO114[#1]	UO113[#1]	UO112[#1]	UO111[#1]	UO110[#1]	UO109[#1]	UO108[#1]
F058	UO123[#1]	UO122[#1]	UO121[#1]	UO120[#1]	UO119[#1]	UO118[#1]	UO117[#1]	UO116[#1]
F059	UO131[#1]	UO130[#1]	UO129[#1]	UO128[#1]	UO127[#1]	UO126[#1]	UO125[#1]	UO124[#1]
F060							ESEND[#1]	EREND[#1]
F061							BCLP[#1]	BUCLP[#1]
F062	PRTSF[#1]							
F063	PSYN[#1]	WATO[#1]				PSAR[#1]	PSE2[#1]	PSE1[#1]
F064	TIALM	TICHK	COSP			TLCHI[#1]	TLNW[#1]	TLCH[#1]
F065							RGSPM[#1]	RGSPP[#1]
F066			PECK2[#1]					G08MD[#1]
F067								
F068								
F069								
F070	PSW08[#1]	PSW07[#1]	PSW06[#1]	PSW05[#1]	PSW04[#1]	PSW03[#1]	PSW02[#1]	PSW01[#1]
F071							PSW10[#1]	PSW09[#1]
F072	OUT7[#1]	OUT6[#1]	OUT5[#1]	OUT4[#1]	OUT3[#1]	OUT2[#1]	OUT1[#1]	OUT0[#1]
F073				ZRNO[#1]		MD4O[#1]	MD2O[#1]	MD1O[#1]
F074								

	7	6	5	4	3	2	1	0
F075	SPO#1	KEYO#1	DRNO#1	MLKO#1	SBKO#1	BDTO#1		
F076			ROV2O#1	ROV1O#1	RTAP#1		MP2O#1	MP1O#1
F077		RTO#1			HS1DO#1	HS1CO#1	HS1BO#1	HS1AO#1
F078	*FV7O#1	*FV6O#1	*FV5O#1	*FV4O#1	*FV3O#1	*FV2O#1	*FV1O#1	*FV0O#1
F079	*JV7O#1	*JV6O#1	*JV5O#1	*JV4O#1	*JV3O#1	*JV2O#1	*JV1O#1	*JV0O#1
F080	*FV15O#1	*FV14O#1	*FV13O#1	*FV12O#1	*FV11O#1	*FV10O#1	*FV9O#1	*FV8O#1
F081	–J4O#1	+J4O#1	–J3O#1	+J3O#1	–J2O#1	+J2O#1	–J1O#1	+J1O#1
F082						RVSL#1		
F083								
F084								
F085								
F086								
F087								
F088								
F089								
F090						ABTSP2#1	ABTSP1#1	ABTQSV#1
F091								
F092			TRSPS#1		TRACT#1			
F093								
F094		ZP7#1	ZP6#1	ZP5#1	ZP4#1	ZP3#1	ZP2#1	ZP1#1
F095								
F096		ZP27#1	ZP26#1	ZP25#1	ZP24#1	ZP23#1	ZP22#1	ZP21#1
F097								
F098		ZP37#1	ZP36#1	ZP35#1	ZP34#1	ZP33#1	ZP32#1	ZP31#1
F099								

	7	6	5	4	3	2	1	0
F100		ZP47#1	ZP46#1	ZP45#1	ZP44#1	ZP43#1	ZP42#1	ZP41#1
F101								
F102		MV7#1	MV6#1	MV5#1	MV4#1	MV3#1	MV2#1	MV1#1
F103								
F104		INP7#1	INP6#1	INP5#1	INP4#1	INP3#1	INP2#1	INP1#1
F105								
F106		MVD7#1	MVD6#1	MVD5#1	MVD4#1	MVD3#1	MVD2#1	MVD1#1
F107								
F108		MMI7#1	MMI6#1	MMI5#1	MMI4#1	MMI3#1	MMI2#1	MMI1#1
F109								
F110		MDTCH7#1	MDTCH6#1	MDTCH5#1	MDTCH4#1	MDTCH3#1	MDTCH2#1	MDTCH1#1
F111								
F112		EADEN7#1	EADEN6#1	EADEN5#1	EADEN4#1	EADEN3#1	EADEN2#1	EADEN1#1
F113								
F114		TRQL7#1	TRQL6#1	TRQL5#1	TRQL4#1	TRQL3#1	TRQL2#1	TRQL1#1
F115								
F116		FRP7#1	FRP6#1	FRP5#1	FRP4#1	FRP3#1	FRP2#1	FRP1#1
F117								
F118		SYN7O#1	SYN6O#1	SYN5O#1	SYN4O#1	SYN3O#1	SYN2O#1	SYN1O#1
F119								
F120		ZRF7#1	ZRF6#1	ZRF5#1	ZRF4#1	ZRF3#1	ZRF2#1	ZRF1#1
F121								
F122	HDO7#1	HDO6#1	HDO5#1	HDO4#1	HDO3#1	HDO2#1	HDO1#1	HDO0#1
F123								
F124								

	7	6	5	4	3	2	1	0
F125								
F126								
F127								
F128								
F129	*EAXSL[#1]		EOV0[#1]					
F130	EBSYA[#1]	EOTNA[#1]	EOTPA[#1]	EGENA[#1]	EDENA[#1]	EIALA[#1]	ECKZA[#1]	EINPA[#1]
F131							EABUFA[#1]	EMFA[#1]
F132	EM28A[#1]	EM24A[#1]	EM22A[#1]	EM21A[#1]	EM18A[#1]	EM14A[#1]	EM12A[#1]	EM11A[#1]
F133	EBSYB[#1]	EOTNB[#1]	EOTPB[#1]	EGENB[#1]	EDENB[#1]	EIALB[#1]	ECKZB[#1]	EINPB[#1]
F134							EABUFB[#1]	EMFB[#1]
F135	EM28B[#1]	EM24B[#1]	EM22B[#1]	EM21B[#1]	EM18B[#1]	EM14B[#1]	EM12B[#1]	EM11B[#1]
F136	EBSYC[#1]	EOTNC[#1]	EOTPC[#1]	EGENC[#1]	EDENC[#1]	EIALC[#1]	ECKZC[#1]	EINPC[#1]
F137							EABUFC[#1]	EMFC[#1]
F138	EM28C[#1]	EM24C[#1]	EM22C[#1]	EM21C[#1]	EM18C[#1]	EM14C[#1]	EM12C[#1]	EM11C[#1]
F139	EBSYD[#1]	EOTND[#1]	EOTPD[#1]	EGEND[#1]	EDEND[#1]	EIALD[#1]	ECKZD[#1]	EINPD[#1]
F140							EABUFD[#1]	EMFD[#1]
F141	EM28D[#1]	EM24D[#1]	EM22D[#1]	EM21D[#1]	EM18D[#1]	EM14D[#1]	EM12D[#1]	EM11D[#1]
F142	EM48A[#1]	EM44A[#1]	EM42A[#1]	EM41A[#1]	EM38A[#1]	EM34A[#1]	EM32A[#1]	EM31A[#1]
F143								
F144								
F145	EM48B[#1]	EM44B[#1]	EM42B[#1]	EM41B[#1]	EM38B[#1]	EM34B[#1]	EM32B[#1]	EM31B[#1]
F146								
F147								
F148	EM48C[#1]	EM44C[#1]	EM42C[#1]	EM41C[#1]	EM38C[#1]	EM34C[#1]	EM32C[#1]	EM31C[#1]
F149								

	7	6	5	4	3	2	1	0
F150								
F151	EM48D#1	EM44D#1	EM42D#1	EM41D#1	EM38D#1	EM34D#1	EM32D#1	EM31D#1
F152								
F153								
F154								
F155								
F156								
F157								
F158								
F159								
F160								
F161								
F162								
F163								
F164								
F165								
F166								
F167								
F168								
F169								
F170								
F171								
F172								
F173								
F174								

	7	6	5	4	3	2	1	0
F175								
F176								
F177								
F178								
F179								
F180								
F181								
F182		EACNT7#1	EACNT6#1	EACNT5#1	EACNT4#1	EACNT3#1	EACNT2#1	EACNT1#1
F183								
F184								
F185								
F186								
F187								
F188								
F189								
F190								
F191								
F192								
F193								
F194								
F195								

PMC → CNC

	7	6	5	4	3	2	1	0
F1000	OP#2	SA#2	STL#2	SPL#2				RWD#2
F1001	MA#2		TAP#2	ENB#2	DEN#2	BAL#2	RST#2	AL#2
F1002	MDRN#2	CUT#2		SRNMV#2	THRD#2	CSS#2	RPDO#2	INCH#2
F1003	MTCHIN#2	MEDT#2	MMEM#2	MRMT#2	MMDI#2	MJ#2	MH#2	MINC#2
F1004			MREF#2	MAFL#2	MSBK#2	MABSM#1	MMLK#2	MBDT1#2
F1005	MBDT9#2	MBDT8#2	MBDT7#2	MBDT6#2	MBDT5#2	MBDT4#2	MBDT3#2	MBDT2#2
F1006								
F1007	BF#2			BF#2	TF#2	SF#2	EFD#2	MF#2
F1008			MF3#2	MF2#2				EF#2
F1009	DM00#2	DM01#2	DM02#2	DM30#2				
F010	M07#2	M06#2	M05#2	M04#2	M03#2	M02#2	M01#2	M00#2
F1011	M15#2	M14#2	M13#2	M12#2	M11#2	M10#2	M09#2	M08#2
F1012	M23#2	M22#2	M21#2	M20#2	M19#2	M18#2	M17#2	M16#2
F1013	M31#2	M30#2	M29#2	M28#2	M27#2	M26#2	M25#2	M24#2
F1014	M207#2	M206#2	M205#2	M204#2	M203#2	M202#2	M201#2	M200#2
F1015	M215#2	M214#2	M213#2	M212#2	M211#2	M210#2	M209#2	M208#2
F1016	M307#2	M306#2	M305#2	M304#2	M303#2	M302#2	M301#2	M300#2
F1017	M315#2	M314#2	M313#2	M312#2	M311#2	M310#2	M309#2	M308#2
F1018								
F1019								
F1020								
F1021								
F1022	S07#2	S06#2	S05#2	S04#2	S03#2	S02#2	S01#2	S00#2
F1023	S15#2	S14#2	S13#2	S12#2	S11#2	S10#2	S09#2	S08#2
F1024	S23#2	S22#2	S21#2	S20#2	S19#2	S18#2	S17#2	S16#2

	7	6	5	4	3	2	1	0
F1025	S31#2	S30#2	S29#2	S28#2	S27#2	S26#2	S25#2	S24#2
F1026	T07#2	T06#2	T05#2	T04#2	T03#2	T02#2	T01#2	T00#2
F1027	T15#2	T14#2	T13#2	T12#2	T11#2	T10#2	T09#2	T08#2
F1028	T23#2	T22#2	T21#2	T20#2	T19#2	T18#2	T17#2	T16#2
F1029	T31#2	T30#2	T29#2	T28#2	T27#2	T26#2	T25#2	T24#2
F1030	B07#2	B06#2	B05#2	B04#2	B03#2	B02#2	B01#2	B00#2
F1031	B15#2	B14#2	B13#2	B12#2	B11#2	B10#2	B09#2	B08#2
F1032	B23#2	B22#2	B21#2	B20#2	B19#2	B18#2	B17#2	B16#2
F1033	B31#2	B30#2	B29#2	B28#2	B27#2	B26#2	B25#2	B24#2
F1034						GR3O#2	GR2O#2	GR1O#2
F1035								SPAL#2
F1036	R08O#2	R07O#2	R06O#2	R05O#2	R04O#2	R03O#2	R02O#2	R01O#2
F1037					R12O#2	R11O#2	R10O#2	R09O#2
F1038					ENB3#2	ENB2#2	SUCLP#2	SCLP#2
F1039								
F1040	AR7#2	AR6#2	AR5#2	AR4#2	AR3#2	AR2#2	AR1#2	AR0#2
F1041	AR15#2	AR14#2	AR13#2	AR12#2	AR11#2	AR10#2	AR09#2	AR08#2
F1042								
F1043								
F1044				SYCAL#2	FSPPH#2	FSPSY#2	FSCSL#2	
F1045	ORARA#2	TLMA#2	LDT2A#2	LDT1A#2	SARA#2	SDTA#2	SSTA#2	ALMA#2
F1046					RCFNA#2	RCHPA#2	CFINA#2	CHPA#2
F1047								
F1048								
F1049	ORARB#2	TLMB#2	LDT2B#2	LDT1B#2	SARB#2	SDTB#2	SSTB#2	ALMB#2

	7	6	5	4	3	2	1	0
F1050					RCFNB#2	RCHPB#2	CFINB#2	CHPB#2
F051								
F1052								
F1053				BGEACT#2	RPALM#2	RPBSY#2		
F1054	UO007#2	UO006#2	UO005#2	UO004#2	UO003#2	UO002#2	UO001#2	UO000#2
F1055	UO015#2	UO014#2	UO013#2	UO012#2	UO011#2	UO010#2	UO009#2	UO008#2
F1056	UO107#2	UO106#2	UO105#2	UO104#2	UO103#2	UO102#2	UO101#2	UO100#2
F1057	UO115#2	UO114#2	UO113#2	UO112#2	UO111#2	UO110#2	UO109#2	UO108#2
F1058	UO123#2	UO122#2	UO121#2	UO120#2	UO119#2	UO118#2	UO117#2	UO116#2
F1059	UO131#2	UO130#2	UO129#2	UO128#2	UO127#2	UO126#2	UO125#2	UO124#2
F1060							ESEND#2	EREND#2
F1061							BCLP#2	BUCLP#2
F1062	PRTSF#2							
F1063	PSYN#2	WATO#2				PSAR#2	PSE2#2	PSE1#2
F1064						TLCHI#2	TLNW#2	TLCH#2
F1065							RGSPM#2	RGSPP#2
F1066			PECK2#2					G08MD#2
F1067								
F1068								
F1069								
F1070	PSW08#2	PSW07#2	PSW06#2	PSW05#2	PSW04#2	PSW03#2	PSW02#2	PSW01#2
F1071							PSW10#2	PSW09#2
F1072	OUT7#2	OUT6#2	OUT5#2	OUT4#2	OUT3#2	OUT2#2	OUT1#2	OUT0#2
F1073				ZRNO#2		MD4O#2	MD2O#2	MD1O#2
F1074								

	7	6	5	4	3	2	1	0
F1075	SPO[#2]	KEYO[#2]	DRNO[#2]	MLKO[#2]	SBKO[#2]	BDTO[#2]		
F1076			ROV2O[#2]	ROV1O[#2]	RTAP[#2]		MP2O[#2]	MP1O[#2]
F1077		RTO[#2]			HS1DO[#2]	HS1CO[#2]	HS1BO[#2]	HS1AO[#2]
F1078	*FV7O[#2]	*FV6O[#2]	*FV5O[#2]	*FV4O[#2]	*FV3O[#2]	*FV2O[#2]	*FV1O[#2]	*FV0O[#2]
F1079	*JV7O[#2]	*JV6O[#2]	*JV5O[#2]	*JV4O[#2]	*JV3O[#2]	*JV2O[#2]	*JV1O[#2]	*JV0O[#2]
F1080	*FV15O[#2]	*FV14O[#2]	*FV13O[#2]	*FV12O[#2]	*FV11O[#2]	*FV10O[#2]	*FV09O[#2]	*FV08O[#2]
F1081	–J4O[#2]	+J4O[#2]	–J3O[#2]	+J3O[#2]	–J2O[#2]	+J2O[#2]	–J1O[#2]	+J1O[#2]
F1082						RVSL[#2]		
F1083								
F1084								
F1085								
F1086								
F1087								
F1088								
F1089								
F1090						ABTSP2[#2]	ABTSP1[#2]	ABTQSV[#2]
F1091								
F1092			TRSPS[#2]		TRACT[#2]			
F1093								
F1094		ZP7[#2]	ZP6[#2]	ZP5[#2]	ZP4[#2]	ZP3[#2]	ZP2[#2]	ZP1[#2]
F1095								
F1096		ZP27[#2]	ZP26[#2]	ZP25[#2]	ZP24[#2]	ZP23[#2]	ZP22[#2]	ZP21[#2]
F1097								
F1098		ZP37[#2]	ZP36[#2]	ZP35[#2]	ZP34[#2]	ZP33[#2]	ZP32[#2]	ZP31[#2]
F1099								

	7	6	5	4	3	2	1	0
F1100		ZP47#2	ZP46#2	ZP45#2	ZP44#2	ZP43#2	ZP42#2	ZP41#2
F1101								
F1102		MV7#2	MV6#2	MV5#2	MV4#2	MV3#2	MV2#2	MV1#2
F1103								
F1104		INP7#2	INP6#2	INP5#2	INP4#2	INP3#2	INP2#2	INP1#2
F1105								
F1106		MVD7#2	MVD6#2	MVD5#2	MVD4#2	MVD3#2	MVD2#2	MVD1#2
F1107								
F1108		MMI7#2	MMI6#2	MMI5#2	MMI4#2	MMI3#2	MMI2#2	MMI1#2
F1109								
F1110		MDTCH7#2	MDTCH6#2	MDTCH5#2	MDTCH4#2	MDTCH3#2	MDTCH2#2	MDTCH1#2
F1111								
F1112		EADEN7#2	EADEN6#2	EADEN5#2	EADEN4#2	EADEN3#2	EADEN2#2	EADEN1#2
F1113								
F1114		TRQL7#2	TRQL6#2	TRQL5#2	TRQL4#2	TRQL3#2	TRQL2#2	TRQL1#2
F1115								
F1116		FRP7#2	FRP6#2	FRP5#2	FRP4#2	FRP3#2	FRP2#2	FRP1#2
F1117								
F1118		SYN7O#2	SYN6O#2	SYN5O#2	SYN4O#2	SYN3O#2	SYN2O#2	SYN1O#2
F1119								
F1120		ZRF7#2	ZRF6#2	ZRF5#2	ZRF4#2	ZRF3#2	ZRF2#2	ZRF1#2
F1121								
F1122	HDO7#2	HDO6#2	HDO5#2	HDO4#2	HDO3#2	HDO2#2	HDO1#2	HDO0#2
F1123								
F1124								

	7	6	5	4	3	2	1	0
F1125								
F1126								
F1127								
F1128								
F1129	*EAXSL#2		EOV0#2					
F1130	EBSYA#2	EOTNA#2	EOTP#2	EGENA#2	EDENA#2	EIALA#2	ECKZA#2	EINPA#2
F1131							EABUFA#2	EMFA#2
F1132	EM28A#2	EM24A#2	EM22A#2	EM21A#2	EM18A#2	EM14A#2	EM12A#2	EM11A#2
F1133	EBSYB#2	EOTNB#2	EOTB#2	EGENB#2	EDENB#2	EIALB#2	ECKZB#2	EINPB#2
F1134							EABUFB#2	EMFB#2
F1135	EM28B#2	EM24B#2	EM22B#2	EM21B#2	EM18B#2	EM14B#2	EM12B#2	EM11B#2
F1136	EBSYC#2	EOTNC#2	EOTC#2	EGENC#2	EDENC#2	EIALC#2	ECKZC#2	EINPC#2
F1137							EABUFC#2	EMFC#2
F1138	EM28C#2	EM24C#2	EM22C#2	EM21C#2	EM18C#2	EM14C#2	EM12C#2	EM11C#2
F1139	EBSYD#2	EOTND#2	EOTD#2	EGEND#2	EDEND#2	EIALD#2	ECKZD#2	EINPD#2
F1140							EABUFD#2	EMFD#2
F1141	EM28D#2	EM24D#2	EM22D#2	EM21D#2	EM18D#2	EM14D#2	EM12D#2	EM11D#2
F1142	EM48A#2	EM44A#2	EM42A#2	EM41A#2	EM38A#2	EM34A#2	EM32A#2	EM31A#2
F1143								
F1144								
F1145	EM48B#2	EM44B#2	EM42B#2	EM41B#2	EM38B#2	EM34B#2	EM32B#2	EM31B#2
F1146								
F1147								
F1148	EM48C#2	EM44C#2	EM42C#2	EM41C#2	EM38C#2	EM34C#2	EM32C#2	EM31C#2
F1149								

	7	6	5	4	3	2	1	0
F1150								
F1151	EM48D#2	EM44D#2	EM42D#2	EM41D#2	EM38D#2	EM34D#2	EM32D#2	EM31D#2
F1152								
F1153								
F1154								
F1155								
F1156								
F1157								
F1158								
F1159								
F1160								
F1161								
F1162								
F1163								
F1164								
F1165								
F1166								
F1167								
F1168								
F1169								
F1170								
F1171								
F1172								
F1173								
F1174								

	7	6	5	4	3	2	1	0
F1175								
F1176								
F1177								
F1178								
F1179								
F1180								
F1181								
F1182		EACNT7#2	EACNT6#2	EACNT5#2	EACNT4#2	EACNT3#2	EACNT2#2	EACNT1#2
F1183								
F1184								
F1185								
F1186								
F1187								
F1188								
F1189								
F1190								
F1191								
F1192								
F1193								
F1194								

A.2
LIST OF SIGNALS

A.2.1
List of Signals
(In order of functions)

○ : Available
● : Available only with 2–path control
– : Unavailable

Function	Signal name	Symbol	Address	T series	M series	Item
Abnormal load detection	Servo axis abnormal load detected signal	ABTQSV	F090#1	○	○	2.10
	First–spindle abnormal load detected signal	ABTSP1	F090#1	○	○	
	Second–spindle abnormal load detected signal	ABTSP2	F090#2	○	○	
Actual spindle speed output	Actual spindle speed signal	AR0 to AR15	F040,F041	○	-	9.7
ALARM signal	Alarm signal	AL	F001#0	○	○	2.4
	Battery alarm signal	BAL	F001#2	○	○	
Automatic tool length measurement (M series)/Automatic tool offset (T series)	Measuring position reached signal	XAE	X004#0	○	○	14.2
		YAE	X004#1	-	○	
		ZAE	X004#2	-	○	
		ZAE	X004#1	○	-	
Auxiliary function lock	Auxiliary function lock signal	AFL	G005#6	○	○	8.2
	Auxiliary function lock check signal	MAFL	F004#4	○	○	
Canned cycle (M series)/Canned cycle for hole machining (T series)	Tapping signal	TAP	F001#5	○	○	11.7
Canned cycle (T series)/Multiple repetitive canned cycle (T series)	Chamferring signal	CDZ	G053#7	○	-	11.9
Chuck/tailstock barrier (T series)	Tail stock barrier select signal	*TSB	G060#7	○	-	2.3.4
CNC ready signal	CNC Ready signal	MA	F001#7	○	○	2.2
	Servo ready signal	SA	F00#6	○	○	
Constant surface speed control	Constant surface speed signal	CSS	F002#2	○	○	9.5
Controlled axes detach	Controlled axis detach signal	DTCH1 to DTCH8	G124	○	○	1.2.4
	Controlled axis detach staus signal	MDTCH1 to MDTCH8	F110	○	○	
Cs contour control	Cs contour control change signal	CON	G027#7	○	○	9.9
	Cs contour control change completion signal	FSCSL	F044#1	○	○	

Function	Signal name	Symbol	Address	T series	M series	Item
Custom macro	Input signal for custom macro	UI000 to UI015	G054,G055	○	○	11.6.1
	Output signal for custom macro	UO000 to UO015	F054,F055	○	○	
		UO100 to UO131	F056 to F059	○	○	
Cycle start/feed hold	Cycle start signal	ST	G007#2	○	○	5.1
	Feed hold signal	*SP	G008#5	○	○	
	Automatic operation signal	OP	F000#7	○	○	
	Cycle start lamp signal	STL	F000#5	○	○	
	Feed hold lamp signal	SPL	F000#4	○	○	
Direct operation by PMC/MMC	Direct operation select signal	DMMC	G042#7	○	○	15.6
DNC operation	DNC operation select signal	DNCI	G043#5	○	○	5.11
	DNC operation selection confirm signal	MRMT	F003#4	○	○	
Dry run	Dry run signal	DRN	G046#7	○	○	5.3.2
	Dry run check signal	MDRN	F002#7	○	○	
Emergency stop	Emergency stop signal	*ESP	G008#4	○	○	2.1
		*ESP	G008#4	○	○	
Error detect (T series)	Error detect signal	SMZ	G053#6	○	-	7.2.6.3
External data input	Data signal for external data input	ED0 to ED15	G000,G001	○	○	15.2
	Address signal for external data input	EA0 to EA6	G002#0 to #6	○	○	
	Read signal for external data input	ESTB	G002#7	○	○	
	Read completion signal for external data input	EREND	F060#0	○	○	
	Search completion signal for external data input	ESEND	F060#1	○	○	
External deceleration	External deceleration signal	*+ED1 to *+ED8	G118	○	○	7.1.9
		*-ED1 to *-ED8	G120	○	○	
External I/O device control	External read start signal	EXRD	G058#1	○	○	13.5
	External punch start signal	EXWT	G058#3	○	○	
	External read/punch stop signal	EXSTP	G058#2	○	○	
	Background editing signal	BGEACT	F053#4	○	○	
	Read/punch busy signal	RPBSY	F053#2	○	○	
	Read/punch alarm signal	RPALM	F053#3	○	○	

Function	Signal name	Symbol	Address	T series	M series	Item
External key input	External key input mode selection signal	ENBKY	G066#1	○	○	15.5
	Key code signal	EKC0 to EKC7	G098	○	○	
	Key code read signal	EKSET	G066#7	○	○	
	Key code read completion signal	EKENB	F053#7	○	○	
	Key input disable signal	INHKY	F053#0	○	○	
	Program screen display mode signal	PRGDPL	F053#1	○	○	
External motion function (M series)	External operation signal	EF	F008#0	-	○	11.8
Feedrate override	Feedrate override signal	*FV0 to *FV7	G012	○	○	7.1.7.2
External program input	External program input start signal	MINP	G058#0	○	○	13.7
External workpiece number search	Workpiece number search signal	PN1, PN2, PN4, PN8, PN16	G009#0 to 4	○	○	15.3
Floating reference position return	Floating reference position return end signal	FRP1 to FRP8	F116	○	○	4.6
Follow–up	Follow–up signal	*FLWU	G007#5	○	○	1.2.7
F1–digit feed (M series)	F1–digit feed select signal	F1D	G016#7	-	○	7.1.5
High–precision contour control by RISC (M series)	HPCC mode signal	MHPCC	F066#6	-	○	7.1.14
	HPCC operation signal	EXHPCC	F066#7	-	○	
High–speed skip signal	High–speed skip status signal	HDO0 to HDO7	F122	○	○	14.3.2
High–speed M/S/T/B interface	Miscellaneous function completion signal	MFIN	G005#0	○	○	8.4
	Spindle function completion signal	SFIN	G005#2	○	○	
	Tool function completion signal	TFIN	G005#3	○	○	
	2nd auxiliary function completion signal	BFIN	G005#4	○	-	
		BFIN	G005#7	-	○	
	2nd M function completion signal	MFIN2	G004#4	○	○	
	3rd M function completion signal	MFIN3	G004#5	○	○	
	External operation signal for high–speed interface	EFD	F007#1	-	○	
	External operation function completion signal	EFIN	G005#1	-	○	
Inch/metric conversion	Inch input signal	INCH	F002#0	○	○	11.4

Function	Signal name	Symbol	Address	T series	M series	Item
Index table indexing function (M series)	B–axis clamp signal	BCLP	F061#1	-	○	11.11
	B–axis clamp completion signal	*BECLP	G038#7	-	○	
	B–axis unclamp signal	BUCLP	F061#0	-	○	
	B–axis unclamp completion signal	*BEUCP	G038#6	-	○	
In–position check	In–position signal	INP1 to INP8	F104	○	○	7.2.6.1
Input of offset value measured A (T series)	Position record signal	PRC	G040#6	○	-	14.4.1
Input of tool offset value measured B (T series)	Tool offset value write mode select signal	GOQSM	G039#7	○	-	14.4.2
	Tool offset value write signal	+MIT1,+MIT2	X004#2,#4	○	-	
		-MIT1,-MIT2	X004#3,#5	○	-	
	Tool offset number select signal	OFN0 to OFN5,OFN6	G039#0 to #5,G040#0	○	-	
	Workpiece coordinate system shift value write mode select signal	WOQSM	G039#6	○	-	
	Workpiece coordinate system shift value write signal	WOSET	G040#7	○	-	
Interlock/start lock	Start lock signal	STLK	G007#1	○	-	2.5
	Interlock signal	*IT	G008#0	○	○	
	Interlock signal for each axis	*IT1 to *IT8	G130	○	○	
	Interlock signal for each axis and direction	+MIT1, +MIT2	X004#2,#4	○	-	
		-MIT1,-MIT2	X004#3,#5	○	-	
	Interlock signal for each axis and direction	+MIT1 to +MIT4	G132#0 to #3	-	○	
		-MIT1 to -MIT4	G134#0 to #3	-	○	
Interruption type custom macro	Interrupt signal for custom macro	UINT	G053#3	○	○	11.6.2
Jog feed/incremental feed	Feed axis and direction selection signal	+J1 to +J8	G100	○	○	3.1
		-J1 to -J8	G102	○	○	
	Manual feedrate override signal	*JV0 to *JV15	G010,G011	○	○	
	Manual rapid traverse selection signal	RT	G019#7	○	○	
Look–ahead control	Lock–ahead control mode signal	G08MD	F066#0	-	○	7.1.13
Machine lock	All–axis Machine lock signal	MLK	G044#1	○	○	5.3.1
	Each–axis machine lock signal	MLK1 to MLK8	G108	○	○	
	All–axis machine lock check signal	MMLK	F004#1	○	○	
Manual absolute on/off	Manual absolute signal	*ABSM	G006#2	○	○	5.4
	Manual absolute check signal	MABSM	F004#2	○	○	

Function	Signal name	Symbol	Address	T series	M series	Item
Manual handle feed	Manual handle feed axis selection signal	HS1A to HS1D	G018#0 to #3	○	○	3.2
		HS2A to HS2D	G018#4 to #7	○	○	
		HS3A to HS3D	G019#0 to #3	-	○	
	Manual handle feed amount selection signal (incremental feed signal)	MP1,MP2	G019#4,#5	○	○	
Manual handle interruption	Manual handle interrupt axis selection signal	HS1IA to HS1ID	G041#0 to #3	○	○	3.3
		HS2IA to HS2ID	G041#4 to #7	○	○	
		HS3IA to HS3ID	G042#0 to #3	-	○	
Manual reference position return	Manual reference position return selection signal	ZRN	G043#7	○	○	4.1
	Manual reference position return selection check signal	MREF	F004#5	○	○	
	Reference position return deceleration signal	*DEC1 to *DEC8	X009	○	○	
	Reference position return end signal	ZP1 to ZP8	F094	○	○	
	Reference position establishment signal	ZRF1 to ZRF8	F120	○	○	
Memory protection key	Memory protection signal	KEY1 to KEY4	G046#3 to #6	○	○	12.2.3
Mirror image	Mirror image signal	MI1 to MI8	G106	○	○	1.2.6
	Mirror image check signal	MMI1 to MMI8	F108	○	○	
Miscellaneous function/2nd auxiliary function	Miscellaneous function code signal	M00 to M31	F010 to F013	○	○	8.1
	Miscellaneous function strobe signal	MF	F007#0	○	○	
	Decode M signal	DM00	F009#7	○	○	
		DM01	F009#6	○	○	
		DM02	F009#5	○	○	
		DM30	F009#4	○	○	
	Spindle speed code signal	S00 to S31	F022 to F025	○	○	
	Spindle speed strobe signal	SF	F007#2	○	○	
	Tool function code signal	T00 to T31	F026 to F029	○	○	
	Tool function strobe signal	TF	F007#3	○	○	
	2nd auxiliary function code signal	B00 to B31	F030 to F033	○	○	
	2nd auxiliary function strobe signal	BF	F007#4	○	-	
		BF	F007#7	-	○	
	End signal	FIN	G004#3	○	○	
	Distribution end signal	DEN	F001#3	○	○	

Function	Signal name	Symbol	Address	T series	M series	Item
Mode selection	Mode selection signal	MD1,MD2,MD4	G043#0 to #2	○	○	2.6
	Manual data input select check signal	MMDI	F003#3	○	○	
	Automatic operation select check signal	MMEM	F003#5	○	○	
	Memory edit select check signal	MEDT	F003#6	○	○	
	Manual handle feed select check signal	MH	F003#1	○	○	
	Incremental feed select check signal	MINC	F003#0	○	○	
	Jog feed select check signal	MJ	F003#2	○	○	
	TEACH IN select check signal	MTCHIN	F003#7	○	○	
Multi–spindle control (T series)	Spindle selection signal	SWS1	G027#0	○	-	9.10
		SWS2	G027#1	○	-	
		SWS3	G027#2	○	-	
	Individual spindle stop signal	*SSTP1	G027#3	○	-	
		*SSTP2	G027#4	○	-	
		*SSTP3	G027#5	○	-	
	Gear select signal (input)	GR21	G029#0	○	-	
		GR31	G029#2	○	-	
	2nd position coder selection signal	PC2SLC	G028#7	○	-	
	Spindle enable signal	ENB2	F038#2	○	-	
		ENB3	F038#3	○	-	
Skip signal	Skip signal	SKIP2 to SKIP6,SKIP7,SKIP8	X004#2 to #6,#0,#1	○	○	14.3.3
Multiple M commands in a single block	2nd M function code signal	M200 to M215	F014 to F015	○	○	8.3
	3rd M function code signal	M300 to M315	F016 to F017	○	○	
	2nd M function strobe signal	MF2	F008#4	○	○	
	3rd M function strobe signal	MF3	F008#5	○	○	
Optional block skip/ addition of optional block skip	Optional block skip signal	BDT1,BDT2 to BDT9	G044#0,G045	○	○	5.5
	Optional block skip check signal	MBDT1,MBDT2 to MBDT9	F004#0,F005	○	○	
Outputting the movement state of an axis	Axis moving signal	MV1 to MV8	F102	○	○	1.2.5
	Axis moving direction signal	MVD1 to MVD8	F106	○	○	
Override cancel	Override cancel signal	OVC	G006#4	○	○	7.1.7.4
Overtravel signal	Overtravel signal	*+L1 to *+L8	G114	○	○	2.3.1
		*-L1 to *-L8	G116	○	○	
Path selection	Path selection signal (Tool post selection signal)	HEAD	G063#0	●	●	2.7

Function	Signal name	Symbol	Address	T series	M series	Item
PMC axis control/ PMC axis speed control function	Control axis selection signal (PMC axis control)	EAX1 to EAX8	G136	○	○	15.1
	Axis control command signal (PMC axis control)	EC0A to EC6A	G143#0 to #6	○	○	
		EC0B to EC6B	G155#0 to #6	○	○	
		EC0C to EC6C	G167#0 to #6	○	○	
		EC0D to EC6D	G179#0 to #6	○	○	
	Axis control feedrate signal (PMC axis control)	EIF0A to EIF15A	G144,G145	○	○	
		EIF0B to EIF15B	G156,G157	○	○	
		EIF0C to EIF15C	G168,G169	○	○	
		EIF0D to EIF15D	G180,G181	○	○	
	Axis control data signal (PMC axis control)	EID0A to EID31A	G146 to G149	○	○	
		EID0B to EID31B	G158 to G161	○	○	
		EID0C to EID31C	G170 to G173	○	○	
		EID0D to EID31D	G182 to G185	○	○	
	Axis control command read signal (PMC axis control)	EBUFA	G142#7	○	○	
		EBUFB	G154#7	○	○	
		EBUFC	G166#7	○	○	
		EBUFD	G178#7	○	○	
	Axis control command read completion signal (PMC axis control)	EBSYA	F130#7	○	○	
		EBSYB	F133#7	○	○	
		EBSYC	F136#7	○	○	
		EBSYD	F139#7	○	○	
	Reset signal (PMC axis control)	ECLRA	G142#6	○	○	
		ECLRB	G154#6	○	○	
		ECLRC	G166#6	○	○	
		ECLRD	G178#6	○	○	
	Axis control temporary stop signal (PMC axis control)	ESTPA	G142#5	○	○	
		ESTPB	G154#5	○	○	
		ESTPC	G166#5	○	○	
		ESTPD	G178#5	○	○	
	Block stop signal (PMC axis control)	ESBKA	G142#3	○	○	
		ESBKB	G154#3	○	○	
		ESBKC	G166#3	○	○	
		ESBKD	G178#3	○	○	
	Block stop disable signal (PMC axis control)	EMSBKA	G143#7	○	○	
		EMSBKB	G155#7	○	○	
		EMSBKC	G167#7	○	○	
		EMSBKD	G179#7	○	○	

Function	Signal name	Symbol	Address	T series	M series	Item
PMC axis control/ PMC axis speed control function	Auxiliary function code signal (PMC axis control)	EM11A to EM48A	F132,F142	○	○	15.1
		EM11B to EM48B	F135,F145	○	○	
		EM11C to EM48C	F138,F148	○	○	
		EM11D to EM48D	F141,F151	○	○	
	Auxiliary function strobe signal (PMC axis control)	EMFA	F131#0	○	○	
		EMFB	F134#0	○	○	
		EMFC	F137#0	○	○	
		EMFD	F140#0	○	○	
	Auxiliary function completion signal (PMC axis control)	EFINA	G142#0	○	○	
		EFINB	G154#0	○	○	
		EFINC	G166#0	○	○	
		EFIND	G178#0	○	○	
	Servo off signal (PMC axis control)	ESOFA	G142#4	○	○	
		ESOFB	G154#4	○	○	
		ESOFC	G166#4	○	○	
		ESOFD	G178#4	○	○	
	Buffering disable signal (PMC axis control)	EMBUFA	G142#2	○	○	
		EMBUFB	G154#2	○	○	
		EMBUFC	G166#2	○	○	
		EMBUFD	G178#2	○	○	
	Control axis selection status signal (PMC axis control)	*EAXSL	F129#7	○	○	
	In–position signal (PMC axis control)	EINPA	F130#0	○	○	
		EINPB	F133#0	○	○	
		EINPC	F136#0	○	○	
		EINPD	F139#0	○	○	
	Following zero checking signal (PMC axis control)	ECKZA	F130#1	○	○	
		ECKZB	F133#1	○	○	
		ECKZC	F136#1	○	○	
		ECKZD	F139#1	○	○	
	Alarm signal (PMC axis control)	EIALA	F130#2	○	○	
		EIALB	F133#2	○	○	
		EIALC	F136#2	○	○	
		EIALD	F139#2	○	○	
	Axis moving signal (PMC axis control)	EGENA	F130#4	○	○	
		EGENB	F133#4	○	○	
		EGENC	F136#4	○	○	
		EGEND	F139#4	○	○	

Function	Signal name	Symbol	Address	T series	M series	Item
PMC axis control/ PMC axis speed control function	Auxiliary function executing signal (PMC axis control)	EDENA	F130#3	○	○	15.1
		EDENB	F133#3	○	○	
		EDENC	F136#3	○	○	
		EDEND	F139#3	○	○	
	Negative–direction overtravel signal (PMC axis control)	EOTNA	F130#6	○	○	
		EOTNB	F133#6	○	○	
		EOTNC	F136#6	○	○	
		EOTND	F139#6	○	○	
	Positive–direction overtravel signal (PMC axis control)	EOTPA	F130#5	○	○	
		EOTPB	F133#5	○	○	
		EOTPC	F136#5	○	○	
		EOTPD	F139#5	○	○	
	Feedrate override signal (PMC axis control)	*FV0E to *FV7E	G151	○	○	
	Override cancellation signal (PMC axis control)	OVCE	G150#5	○	○	
	Rapid traverse override signal (PMC axis control)	ROV1E,ROV2E	G150#0,#1	○	○	
	Dry run signal (PMC axis control)	DRNE	G150#7	○	○	
	Manual rapid traverse selection signal (PMC axis control)	RTE	G150#6	○	○	
	Override 0% signal (PMC axis control)	EOV0	F129#5	○	○	
	Skip signal (PMC axis control)	ESKIP	X004#6	○	○	
	Distribution completion signal (PMC axis control)	EADEN1 to EADEN8	F112	○	○	
	Buffer full signal (PMC axis control)	EABUFA	F131#1	○	○	
		EABUFB	F134#1	○	○	
		EABUFC	F137#1	○	○	
		EABUFD	F140#1	○	○	
	Controling signal (PMC axis control)	EACNT1 to EACNT8	F182	○	○	
Polygonal turning	Polygon synchronization under way signal	PSYN	F063#7	○	-	6.10.1

Function	Signal name	Symbol	Address	T series	M series	Item
Polygonal turning with two spindles	Polygon spindle stop signal	*PLSST	G038#0	○	-	6.10.2
	Spindle polygon speed arrival signal	PSAR	F063#2	○	-	
	Master axis not arrival signal	PSE1	F063#0	○	-	
	Polygon synchronous axis not arrival signal	PSE2	F063#1	○	-	
Position switch	Position switch signal	PSW01 to PSW10	F070#0 to F071#1	○	○	1.2.9
Position display neglect	Position display neglect signal	NPOS1 to NPOS8	G198	○	○	12.1.10
Program restart	Program restart signal	SRN	G006#0	○	○	5.7
	Program restart under way signal	SRNMV	F002#4	○	○	
Rapid traverse override	Rapid traverse override signal	ROV1,ROV2	G014#0,#1	○	○	7.1.7.1
	1% step rapid traverse override select signal	HROV	G096#7	○	○	
	1% step rapid traverse override signal	*HROV0 to *HROV6	G096#0 to #6	○	○	
Reset and rewind	External reset signal	ERS	G008#7	○	○	5.2
	Reset & rewind signal	RRW	G008#6	○	○	
	Resetting signal	RST	F001#1	○	○	
	Rewinding signal	RWD	F000#0	○	○	
Remote buffer	Input signal for remote buffer	RMTDI0 to RMTDI7	G052	○	○	13.2
	Output signal for remote buffer	RMTDO0 to RMTDO7	F069	○	○	
Retrace (M series)	Retrace signal	RVS	G007#0	-	○	11.15
	Retrace–in–progress signal	RVSL	F082#2	-	○	
Rigid tapping	Rigid tapping signal	RGTAP	G061#0	○	○	9.11
	Spindle rotation direction signal	RGSPP	F065#0	-	○	
		RGSPM	F065#1	-	○	
	Rigid tapping in–progress signal	RTAP	F076#3	○	○	
	Rigid tapping spindle selection signal	RGTSP1,RGTSP2	G061#4,#5	○	-	
Run hour and parts count display	Target part count reached signal	PRTSF	F062#7	○	○	12.1.11
	General–purpose integrating meter start signal	TMRON	G053#0	○	○	
Second reference position return/crd, 4th reference position return	2nd reference position return end signal	ZP21 to ZP28	F096	○	○	4.5
	3rd reference position return end signal	ZP31 to ZP38	F098	○	○	
	4th reference position return end signal	ZP41 to ZP48	F100	○	○	
Second feedrate override	2nd feedrate override signal	*AFV0 to *AFV7	G013	○	○	7.1.7.3
Servo off (mechanical handle)	Servo off signal	SVF1 to SVF8	G126	○	○	1.2.8

Function	Signal name	Symbol	Address	T series	M series	Item
Single block	Single block signal	SBK	G046#1	○	○	5.3.3
	Single block check signal	MSBK	F004#3	○	○	
Simultaneous input and output operations (M series)	Input and run simultaneous mode select signal	STRD	G058#5	-	○	13.6
	Output and run simultaneous mode select signal	STWD	G058#6	-	○	
Simple synchronous control	Simple synchronous axis select signal	SYNC1 to SYNC8	G138	○	○	1.6
	Simple synchronous manual feed axis select signal	SYNCJ1 to SYNCJ8	G140	-	○	
Skip function	Skip signal	SKIP	X004#7	○	○	14.3.1
		SKIPP	G006#6	○	-	
Small hole peck drilling cycle	Overload torque signal	SKIP	X004#7	-	○	11.17
	Small–diameter peck drilling in progress signal	PECK2	F066#5	-	○	11.17
Software operator's panel	Software operator's panel signal (MD1)	MD1O	F073#0	○	○	12.1.15
	Software operator's panel signal (MD2)	MD2O	F073#1	○	○	
	Software operator's panel signal (MD4)	MD4O	F073#2	○	○	
	Software operator's panel signal (ZRN)	ZRNO	F073#4	○	○	
	Software operator's panel signal (+J1 to +J4)	+J1O to +J4O	F081#0,#2,#4,#6	○	○	
	Software operator's panel signal (-J1 to -J4)	-J1O to -J4O	F081#1,#3,#5,#7	○	○	
	Software operator's panel signal (RT)	RTO	F077#6	○	○	
	Software operator's panel signal (HS1A)	HS1AO	F077#0	○	○	
	Software operator's panel signal (HS1B)	HS1BO	F077#1	○	○	
	Software operator's panel signal (HS1C)	HS1CO	F077#2	○	○	
	Software operator's panel signal (HS1D)	HS1DO	F077#3	○	○	

Function	Signal name	Symbol	Address	T series	M series	Item
Software operator's panel	Software operator's panel signal (MP1)	MP1O	F076#0	○	○	12.1.15
	Software operator's panel signal (MP2)	MP2O	F076#1	○	○	
	Software operator's panel signal (*JV0 to *JV15)	*JV0O to *JV15O	F079,F080	○	○	
	Software operator's panel signal (*FV0 to *FV7)	*FV0O to *FV7O	F078	○	○	
	Software operator's panel signal (ROV1)	ROV1O	F076#4	○	○	
	Software operator's panel signal (ROV2)	ROV2O	F076#5	○	○	
	Software operator's panel signal (BDT)	BDTO	F075#2	○	○	
	Software operator's panel signal (SBK)	SBKO	F075#3	○	○	
	Software operator's panel signal (MLK)	MLKO	F075#4	○	○	
	Software operator's panel signal (DRN)	DRNO	F075#5	○	○	
	Software operator's panel signal (KEY1 to KEY4)	KEYO	F075#6	○	○	
	Software operator's panel signal (*SP)	SPO	F075#7	○	○	
	Software operator's panel general–purpose switch signal	OUT0 to OUT7	F072	○	○	
Spindle orientation	Spindle orientation external stop position command signal	SHA00 to SHA11	G078#0 to G079#3	○	○	9.13
		SHB00 to SHB11	G080#0 to G081#3	○	○	
Spindle output control by the PMC	Spindle motor speed command select signal	SIND	G033#7	○	○	15.4
		SIND2	G035#7	○	○	
		SIND3	G037#7	○	○	
Spindle output control by the PMC	Spindle motor speed command signal	R01I to R12I	G032#0 to G033#3	○	○	15.4
		R01I2 to R12I2	G034#0 to G035#3	○	○	
		R01I3 to R12I3	G036#0 to G037#3	○	○	
	Spindle motor command polarity select signal	SSIN	G033#6	○	○	
		SSIN2	G035#6	○	○	
		SSIN3	G037#6	○	○	
	Spindle motor command polarity select signal	SGN	G033#5	○	○	
		SGN2	G035#5	○	○	
		SGN3	G037#5	○	○	

Function	Signal name	Symbol	Address	T series	M series	Item
Spindle positioning (T series)	Spindle stop complete signal	SPSTP	G028#6	○	-	9.8
	Spindle unclamp signal	SUCLP	F038#1	○	-	
	Spindle unclamp completion signal	*SUCPF	G028#4	○	-	
	Spindle clamp signal	SCLP	F038#0	○	-	
	Spindle clamp completion signal	*SCPF	G028#5	○	-	
	Spindle orientation completion signal	ZPX	F094	○	-	
Spindle serial output/ spindle analog output	Torque limit command LOW signal (serial spindle)	TLMLA	G070#0	○	○	9.2
		TLMLB	G074#0	○	○	
	Torque limit command HIGH signal (serial spindle)	TLMHA	G070#1	○	○	
		TLMHB	G074#1	○	○	
	Clutch/gear signal (serial spindle)	CTH1A,CTH2A	G070#3,#2	○	○	
		CTH1B,CTH2B	G074#3,#2	○	○	
	CCW command signal (serial spindle)	SRVA	G070#4	○	○	
		SRVB	G074#4	○	○	
	CW command signal (serial spindle)	SFRA	G070#5	○	○	
		SFRB	G074#5	○	○	
	Orientation command signal (serial spindle)	ORCMA	G070#6	○	○	
		ORCMB	G074#6	○	○	
	Machine ready signal (serial spindle)	MRDYA	G070#7	○	○	
		MRDYB	G074#7	○	○	
	Alarm reset signal (serial spindle)	ARSTA	G071#0	○	○	
		ARSTB	G075#0	○	○	
	Emergency stop signal (serial spindle)	*ESPA	G071#1	○	○	
		*ESPB	G075#1	○	○	

Function	Signal name	Symbol	Address	T series	M series	Item
Spindle serial output/ spindle analog output	Spindle select signal (serial spindle)	SPSLA	G071#2	○	○	9.2
	Spindle select signal (serial spindle)	SPSLB	G075#2	○	○	
	Power line switch completion signal (serial spindle)	MCFNA	G071#3	○	○	
		MCFNB	G075#3	○	○	
	Soft start/stop cancel signal (serial spindle)	SOCNA	G071#4	○	○	
		SOCNB	G075#4	○	○	
	Output switch request signal (serial spindle)	RSLA	G071#6	○	○	
		RSLB	G075#6	○	○	
	Power line status check signal (serial spindle)	RCHA	G071#7	○	○	
		RCHB	G075#7	○	○	
	Alarm signal (serial spindle)	ALMA	F045#0	○	○	
		ALMB	F049#0	○	○	
	Speed zero signal (serial spindle)	SSTA	F045#1	○	○	
		SSTB	F049#1	○	○	
	Speed detection signal (serial spindle)	SDTA	F045#2	○	○	
		SDTB	F049#2	○	○	
	Speed arrival signal (serial spindle)	SARA	F045#3	○	○	
		SARB	F049#3	○	○	
	Load detection signal 1 (serial spindle)	LDT1A	F045#4	○	○	
		LDT1B	F049#4	○	○	
	Load detection signal 2 (serial spindle)	LDT2A	F045#5	○	○	
		LDT2B	F049#5	○	○	
	Torque limit signal (serial spindle)	TLMA	F045#6	○	○	
		TLMB	F049#6	○	○	
	Orientation completion signal (serial spindle)	ORARA	F045#7	○	○	
		ORARB	F049#7	○	○	
	Power line switch signal (serial spindle)	CHPA	F046#0	○	○	
		CHPB	F050#0	○	○	
	Spindle switch completion signal (serial spindle)	CFINA	F046#1	○	○	
		CFINB	F050#1	○	○	
	Output switch signal (serial spindle)	RCHPA	F046#2	○	○	
		RCHPB	F050#2	○	○	
	Output switch completion signal (serial spindle)	RCFNA	F046#3	○	○	
		RCFNB	F050#3	○	○	

Function	Signal name	Symbol	Address	T series	M series	Item
Spindle speed control	Spindle stop signal	*SSTP	G029#6	○	○	9.3
	Spindle orientation signal	SOR	G029#5	○	○	
	Spindle speed override signal	SOV0 to SOV7	G030	○	○	
	Spindle speed arrival signal	SAR	G029#4	○	○	
	Spindle enable signal	ENB	F001#4	○	○	
	Gear selection signal (output)	GR1O,GR2O,GR3O	F034#0 to #2	-	○	
	Gear selection signal (input)	GR1,GR2	G028#1,#2	○	○	
	S12–bit code signal	R01O to R12O	F036#0 to F037#3	○	○	
Spindle speed control for two–path lathe	Spindle command select signal	SLSPA,SLSPB	G063#2,#3	●	-	9.4
	Spindle feedback select signal	SLPCA,SLPCB	G064#2,#3	●	-	
	Spindle command signal	COSP	F064#5	●	-	
Spindle speed fluctuation detection	Spindle fluctuation detection alarm signal	SPAL	F035#0	○	○	9.6
Spindle synchronous control	Spindle synchronous control signal	SPSYC	G038#2	○	○	9.12
	Spindle phase synchronous control signal	SPPHS	G038#3	○	○	
	Spindle synchronous speed control completion signal	FSPSY	F044#2	○	○	
	Spindle phase synchronous control completion signal	FSPPH	F044#3	○	○	
	Spindle synchronous control alarm signal	SYCAL	F044#4	○	○	
Status output signal	Rapid traversing signal	RPDO	F002#1	○	○	2.8
	Cutting feed signal	CUT	F002#6	○	○	
Stored stroke limit	Stored stroke limit select signal	EXLM	G007#6	○	○	2.3.2
	Stroke limit external setting signal	+LM1 to +LM8	G110	-	○	
		<LM1 to <LM8	G110	-	○	
	Stroke limit release signal	RLSOT	G007#7	-	○	
Synchronous control	Synchronous control axis selection signals	SYNC to SYNC8	G138	○	-	1.8
	Parking signals	PK1 to PK8	G122	○	-	1.8
	Synchronous control under way signals	SYN1O to SYN8O	F118	○	-	1.8
	Synchronous control axis selection signals	SYNC to SYNC7	G138#0 to #6	●	-	1.9
	Synchronous/composite/superimposed control under way signals	SYN1O to SYN7O	F118#0 to #6	●	-	1.9
Thread cutting	Thread cutting signal	THRD	F002#3	○	○	6.4.1
Tool axis direction handle feed function (M series)	Tool axis direction handle feed mode signal	ALNGH	G023#7	-	○	3.4

Function	Signal name	Symbol	Address	T series	M series	Item
Tool post interface check (T series, two–path control)	Tool post interference check signal	TICHK	F064#6	●	-	2.3.5
	Tool post interference alarm signal	TIALM	F064#7	●	-	
Tool retraction and return	Tool retraction signal	TRESC	G059#0	○	○	5.8
	Tool retraction mode signal	TRACT	F092#3	○	○	
	Tool return signal	TRRTN	G059#1	○	○	
	Tool return completion signal	TRSPS	F092#5	○	○	
Tool life management	Tool change signal	TLCH	F064#0	○	○	10.3
	Tool change reset signal	TLRST	G048#7	○	○	
	Individual tool change signal	TLCHI	F064#2	-	○	
	Individual tool change reset signal	TLRST	IG048#6	-	○	
	Tool skip signal	TLSKP	G048#5	○	○	
	New tool select signal	TLNW	F064#1	○	○	
	Tool group number select signal	TL01 to TL256	G047#0 to G048#0	-	○	
		TL01 to TL64	G047#0 to #6	○	-	
	Tool life count override signal	*TLV0 to *TLV9	G049#0 to G050#1	-	○	
Torque limit skip (T series)	Torque limit reached signal	TRQL1 to TRQL8	F114	○	-	14.3.4
VRDY off alarm ignore signal	All–axis VRDY OFF alarm ignore signal	IGNVRY	G066#0	○	○	2.9
	Each–axis VRDY OFF alarm ignore signal	IGVRY1 to IGVRY8	G192	○	○	
Waiting M code (two–path control)	No–wait signal	NOWT	G063#1	●	●	8.5
	Waiting signal	WATO	F063#6	●	●	

A.2.2
List of Signals (In order of symbols)

○	:	Available
●	:	Available only with 2–path control
–	:	Unavailable

Group	Symbol	Signal name	Address	T series	M series	Refer-ence item
*	*+ED1 to *+ED8	External deceleration signal	G118	○	○	7.1.9
	*+L1 to *+L8	Overtravel signal	G114	○	○	2.3.1
	*-ED1 to *-ED8	External deceleration signal	G120	○	○	7.1.9
	*-L1 to *-L8	Overtravel signal	G116	○	○	2.3.1
	*ABSM	Manual absolute signal	G006#2	○	○	5.4
	*AFV0 to *AFV7	2nd feedrate override signal	G013	○	○	7.1.7.3
	*BECLP	B–axis clamp completion signal	G038#7	-	○	11.11
	*BEUCP	B–axis unclamp completion signal	G038#6	-	○	11.11
	*DEC1 to *DEC8	Deceleration signal for reference position return	X009	○	○	4.1
	*EAXSL	Control axis selection status signal(PMC axis control)	F129#7	○	○	15.1
	*ESP	Emergency stop signal	X008#4	○	○	2.1
	*ESP		G008#4	○	○	2.1
	*ESPA	Emergency stop signal (serial spindle)	G071#1	○	○	9.2
	*ESPB		G075#1	○	○	9.2
	*FLWU	Follow–up signal	G007#5	○	○	1.2.7
	*FV0 to *FV7	Feedrate override signal	G012	○	○	7.1.7.2
	*FV0E to *FV7E	Feedrate override signal (PMC axis control)	G151	○	○	15.1
	*FV0O to *FV7O	Software operator's panel signal(*FV0 to *FV7)	F078	○	○	12.1.15
	*HROV0 to *HROV6	1% step rapid traverse override signal	G096#0 to #6	○	○	7.1.7.1
	*IT	Interlock signal	G008#0	○	○	2.5
	*IT1 to *IT8	Interlock signal for each axis	G130	○	○	2.5
	*JV0 to *JV15	Manual feedrate override signal	G010,G011	○	○	3.1
	*JV0O to *JV15O	Software operator's panel signal(*JV0 to *JV15)	F079,F080	○	○	12.1.15
	*PLSST	Polygon spindle stop signal	G038#0	○	-	6.10.2
	*SCPF	Spindle clamp completion signal	G028#5	○	-	9.8
	*SP	Feed hold signal	G008#5	○	○	5.1
	*SSTP	Spindle stop signal	G029#6	○	○	9.3
	*SSTP1	Stop signal in each spindle	G027#3	○	-	9.10

Group	Symbol	Signal name	Address	T series	M series	Refer-ence item
*	*SSTP2	Stop signal in each spindle	G027#4	○	-	9.10
	*SSTP3		G027#5	○	-	9.10
	*SUCPF	Spindle unclamp completion signal	G028#4	○	-	9.8
	*TLV0 to *TLV9	Tool life count override signal	G049#0 to G050#1	-	○	10.3
	*TSB	Tailstock barrier select signal	G060#7	○	-	2.3.4
+	+J1 to +J8	Feed axis and direction selection signal	G100	○	○	3.1
	+J1O to +J4O	Software operator's panel signal(+J1 to +J4)	F081#0,#2,#4,#6	○	○	12.1.15
	+LM1 to +LM8	Stroke limit external setting signal	G110	-	○	2.3.2
	+MIT1,+MIT2	Manual feed interlock signal for each axis	X004#2,#4	○	-	2.5
	+MIT1,+MIT2	Tool offset write signal	X004#2,#4	○	-	14.4.2
	+MIT1 to +MIT4	Interlock signal for each axis and direction	G132#0 to #3	-	○	2.5
<	−J1 to -J8	Feed axis and direction selection signal	G102	○	○	3.1
	−J1O to −J4O	Software operator's panel signal(−J1 to −J4)	F081#1,#3,#5,#7	○	○	12.1.15
	−LM1 to −LM8	Stroke limit external setting signal	G112	-	○	2.3.2
	−MIT1,−MIT2	Manual feed interlock signal for each axis	X004#3,#5	○	-	2.5
	−MIT1,−MIT2	Tool offset write signal		○	-	14.4.2
	−MIT1 to −MIT4	Interlock signal for each axis and direction	G134#0 to #3	-	○	2.5
A	ABTQSV	Servo axis abnormal load detected signal	F090#1	○	○	
	ABTSP1	First−spindle abnormal load detected signal	F090#1	○	○	2.10
	ABTSP2	Second−spindle abnormal load detected signal	F090#2	○	○	
	AFL	Miscellaneous function lock signal	G005#6	○	○	8.2
	AL	Alarm signal	F001#0	○	○	2.4
	ALMA	Alarm signal (serial spindle)	F045#0	○	○	9.2
	ALMB		F049#0	○	○	9.2
	ALNGH	Tool axis direction handle feed mode signal	G023#7	-	○	3.4
	AR0 to AR15	Actual spindle speed signal	F040,F041	○	-	9.7
	ARSTA	Alarm reset signal (serial spindle)	G071#0	○	○	9.2
	ARSTB		G075#0	○	○	9.2

Group	Symbol	Signal name	Address	T series	M series	Reference item
B	B00 to B31	2nd auxiliary function code signal	F030 to F033	○	○	8.1
	BAL	Battery alarm signal	F001#2	○	○	2.4
	BCLP	B–axis clamp signal	F061#1	-	○	11.11
	BDT1,BDT2 to BDT9	Optional block skip signal	G044#0,G045	○	○	5.5
	BDTO	Software operator's panel signal(BDT)	F075#2	○	○	12.1.15
	BF	2nd auxiliary function strobe signal	F007#4	○	-	8.1
	BF		F007#7	-	○	8.1
	BFIN	2nd auxiliary function completion signal	G005#4	○	-	8.4
	BFIN		G005#7	-	○	8.4
	BGEACT	Background editing signal	F053#4	○	○	13.5
	BUCLP	B–axis unclamp signal	F061#0	-	○	11.11
C	CDZ	Chamferring signal	G053#7	○	-	11.9
	CFINA	Spindle switch completion signal(serial spindle)	F046#1	○	○	9.2
	CFINB		F050#1	○	○	9.2
	CHPA	Power line switch signal (serial spindle)	F046#0	○	○	9.2
	CHPB		F050#0	○	○	9.2
	CON	Cs contour control change signal	G027#7	○	○	9.9
	COSP	Spindle command signal	F064#5	●	-	9.4
	CSS	Constant surface speed signal	F002#2	○	○	9.5
	CTH1A,CTH2A	Clutch/gear signal (serial spindle)	G070#3,#2	○	○	9.2
	CTH1B,CTH2B		G074#3,#2	○	○	9.2
	CUT	Cutting feed signal	F002#6	○	○	2.8
D	DEN	Distribution end signal	F001#3	○	○	8.1
	DM00	Decode M signal	F009#7	○	○	8.1
	DM01		F009#6	○	○	8.1
	DM02		F009#5	○	○	8.1
	DM30		F009#4	○	○	8.1
	DMMC	Direct operation select signal	G042#7	○	○	15.6
	DNCI	DNC operation select signal	G043#5	○	○	5.11
	DRN	Dry run signal	G046#7	○	○	5.3.2
	DRNE	Dry run signal (PMC axis control)	G150#7	○	○	15.1
	DRNO	Software operator's panel signal(DRN)	F075#5	○	○	12.1.15
	DTCH1 to DTCH8	Controlled axis detach signal	G124	○	○	1.2.4

Group	Symbol	Signal name	Address	T series	M series	Refer-ence item
E	EA0 to EA6	Address signal for external data input	G002#0 to #6	○	○	15.2
	EABUFA	Buffer full signal (PMC axis control)	F131#1	○	○	15.1
	EABUFB		F134#1	○	○	15.1
	EABUFC		F137#1	○	○	15.1
	EABUFD		F140#1	○	○	15.1
	EACNT1 to EACNT8	Controlling signal (PMC axis control)	F182	○	○	15.1
	EADEN1 to EADEN8	Distribution completion signal(PMC axis control)	F112	○	○	15.1
	EAX1 to EAX8	Control axis select signal (PMC axis control)	G136	○	○	15.1
	EBSYA	Axis control command read completion signal (PMC axis control)	F130#7	○	○	15.1
	EBSYB		F133#7	○	○	15.1
	EBSYC		F136#7	○	○	15.1
	EBSYD		F139#7	○	○	15.1
	EBUFA	Axis control command read signal(PMC axis control)	G142#7	○	○	15.1
	EBUFB		G154#7	○	○	15.1
	EBUFC		G166#7	○	○	15.1
	EBUFD		G178#7	○	○	15.1
	EC0A to EC6A	Axis control command signal (PMC axis control)	G143#0 to #6	○	○	15.1
	EC0B to EC6B		G155#0 to #6	○	○	15.1
	EC0C to EC6C		G167#0 to #6	○	○	15.1
	EC0D to EC6D		G179#0 to #6	○	○	15.1
	ECKZA	Following zero checking signal (PMC axis control)	F130#1	○	○	15.1
	ECKZB		F133#1	○	○	15.1
	ECKZC		F136#1	○	○	15.1
	ECKZD		F139#1	○	○	15.1
	ECLRA	Reset signal (PMC axis control)	G142#6	○	○	15.1
	ECLRB		G154#6	○	○	15.1
	ECLRC		G166#6	○	○	15.1
	ECLRD		G178#6	○	○	15.1
	ED0 to ED15	Data signal for external data input	G000,G001	○	○	15.2
	EDENA	Auxiliary function executing signal (PMC axis control)	F130#3	○	○	15.1
	EDENB		F133#3	○	○	15.1
	EDENC		F136#3	○	○	15.1
	EDEND		F139#3	○	○	15.1
	EF	External operation signal	F008#0	-	○	11.8
	EFD	External operation signal for high–speed interface	F007#1	-	○	8.4

Group	Symbol	Signal name	Address	T series	M series	Refer-ence item
E	EFIN	External operation function completion signal	G005#1	-	○	8.4
	EFINA	Auxiliary function completion signal (PMC axis control)	G142#0	○	○	15.1
	EFINB		G154#0	○	○	15.1
	EFINC		G166#0	○	○	15.1
	EFIND		G178#0	○	○	15.1
	EGENA	Axis moving signal (PMC axis control)	F130#4	○	○	15.1
	EGENB		F133#4	○	○	15.1
	EGENC		F136#4	○	○	15.1
	EGEND		F139#4	○	○	15.1
	EIALA	Alarm signal (PMC axis control)	F130#2	○	○	15.1
	EIALB		F133#2	○	○	15.1
	EIALC		F136#2	○	○	15.1
	EIALD		F139#2	○	○	15.1
	EID0A to EID31A	Axis control data signal (PMC axis control)	G146 to G149	○	○	15.1
	EID0B to EID31B		G158 to G161	○	○	15.1
	EID0C to EID31C		G170 to G173	○	○	15.1
	EID0D to EID31D		G182 to G185	○	○	15.1
	EIF0A to EIF15A	Axis control feedrate signal (PMC axis control)	G144,G145	○	○	15.1
	EIF0B to EIF15B		G156,G157	○	○	15.1
	EIF0C to EIF15C		G168,G169	○	○	15.1
	EIF0D to EIF15D		G180,G181	○	○	15.1
	EINPA	In–position signal (PMC axis control)	F130#0	○	○	15.1
	EINPB		F133#0	○	○	15.1
	EINPC		F136#0	○	○	15.1
	EINPD		F139#0	○	○	15.1
	EKC0 to EKC7	Key code signal	G098	○	○	15.5
	EKENB	Key code read completion signal	F053#7	○	○	15.5
	EKSET	key code read signal	G066#7	○	○	15.5
	EM11A to EM48A	Auxiliary function code signal (PMC axis control)	F132,F142	○	○	15.1
	EM11B to EM48B		F135,F145	○	○	15.1
	EM11C to EM48C		F138,F148	○	○	15.1
	EM11D to EM48D		F141,F151	○	○	15.1
	EMBUFA	Buffering disable signal (PMC axis control)	G142#2	○	○	15.1
	EMBUFB		G154#2	○	○	15.1
	EMBUFC		G166#2	○	○	15.1
	EMBUFD		G178#2	○	○	15.1

Group	Symbol	Signal name	Address	T series	M series	Reference item
E	EMFA	Auxiliary function strobe signal (PMC axis control)	F131#0	○	○	15.1
	EMFB		F134#0	○	○	15.1
	EMFC		F137#0	○	○	15.1
	EMFD		F140#0	○	○	15.1
	EMSBKA	Block stop disable signal (PMC axis control)	G143#7	○	○	15.1
	EMSBKB		G155#7	○	○	15.1
	EMSBKC		G167#7	○	○	15.1
	EMSBKD		G179#7	○	○	15.1
	ENB	Spindle enable signal	F001#4	○	○	9.3
	ENB2		F038#2	○	-	9.10
	ENB3		F038#3	○	-	9.10
	ENBKY	External key input mode selection signal	G066#1	○	○	15.5
	EOTNA	Negative–direction overtravel signal (PMC axis control)	F130#6	○	○	15.1
	EOTNB		F133#6	○	○	15.1
	EOTNC		F136#6	○	○	15.1
	EOTND		F139#6	○	○	15.1
	EOTPA	Positive–direction overtravel signal (PMC axis control)	F130#5	○	○	15.1
	EOTPB		F133#5	○	○	15.1
	EOTPC		F136#5	○	○	15.1
	EOTPD		F139#5	○	○	15.1
	EOV0	Override 0% signal (PMC axis control)	F129#5	○	○	15.1
	EREND	Read completion signal for external data input	F060#0	○	○	15.2
	ERS	External reset signal	G008#7	○	○	5.2
	ESBKA	Block stop signal (PMC axis control)	G142#3	○	○	15.1
	ESBKB		G154#3	○	○	15.1
	ESBKC		G166#3	○	○	15.1
	ESBKD		G178#3	○	○	15.1
	ESEND	Search completion signal for external data input	F060#1	○	○	15.2
	ESKIP	Skip signal (PMC axis control)	X004#6	○	○	15.1
	ESOFA	Servo off signal (PMC axis control)	G142#4	○	○	15.1
	ESOFB		G154#4	○	○	15.1
	ESOFC		G166#4	○	○	15.1
	ESOFD		G178#4	○	○	15.1
	ESTB	Read signal for external data input	G002#7	○	○	15.2

Group	Symbol	Signal name	Address	T series	M series	Refer-ence item
E	ESTPA	Axis control temporary stop signal (PMC axis control)	G142#5	○	○	15.1
	ESTPB		G154#5	○	○	15.1
	ESTPC		G166#5	○	○	15.1
	ESTPD		G178#5	○	○	15.1
	EXHPCC	HPCC operation signal	F066#7	-	○	7.1.14
	EXLM	Stored stroke limit select signal	G007#6	○	○	2.3.2
	EXRD	External read start signal	G058#1	○	○	13.5
	EXSTP	External read/punch stop signal	G058#2	○	○	13.5
	EXWT	External punch start signal	G058#3	○	○	13.5
F	F1D	F1–digit feed select signal	G016#7	-	○	7.1.5
	FIN	Completion signal	G004#3	○	○	8.1
	FRP1 to FRP8	Floating reference position return end signal	F116	○	○	4.6
	FSCSL	Cs contour control change completion signal	F044#1	○	○	9.9
	FSPPH	Spindle phase synchronous control completion signal	F044#3	○	○	9.12
	FSPSY	Spindle synchronous speed control completion signal	F044#2	○	○	9.12
G	G08MD	Lock–ahead control mode signal	F066#0	-	○	7.1.13
	GOQSM	Tool offset value write mode select signal	G039#7	○	-	14.4.2
	GR1,GR2	Gear selection signal (input)	G028#1,#2	○	○	9.3
	GR1O,GR2O,GR3O	Gear selection signal (output)	F034#0 to #2	-	○	9.3
	GR21	Gear selection signal (input)	G029#0	○	-	9.10
	GR31		G029#2	○	-	9.10
H	HDO0 to HDO7	High–speed skip status signal	F122	○	○	14.3.2
	HEAD	Path selection signal (Tool post selection signal)	G063#0	●	●	2.7
	HROV	1% step rapid traverse override select signal	G096#7	○	○	7.1.7.1
	HS1A to HS1D	Manual handle feed axis selection signal	G018#0 to #3	○	○	3.2
	HS1AO	Software operator's panel signal(HS1A)	F077#0	○	○	12.1.15
	HS1BO	Software operator's panel signal(HS1B)	F077#1	○	○	12.1.15
	HS1CO	Software operator's panel signal(HS1C)	F077#2	○	○	12.1.15
	HS1DO	Software operator's panel signal(HS1D)	F077#3	○	○	12.1.15
	HS1IA to HS1ID	Manual handle interruption axis select signal	G041#0 to #3	○	○	3.3

Group	Symbol	Signal name	Address	T series	M series	Refer-ence item
H	HS2A to HS2D	Manual handle feed axis selection signal	G018#4 to #7	○	○	3.2
	HS2IA to HS2ID	Manual handle interruption axis select signal	G041#4 to #7	○	○	3.3
	HS3A to HS3D	Manual handle feed axis selection signal	G019#0 to #3	-	○	3.2
	HS3IA to HS3ID	Manual handle interruption axis select signal	G042#0 to #3	-	○	3.3
I	IGNVRY	All–axis VRDY OFF alarm ignore signal	G066#0	○	○	2.9
	IGVRY1 to IGVRY8	Each–axis VRDY OFF alarm ignore signal	G192	○	○	2.9
	INCH	Inch input signal	F002#0	○	○	11.4
	INHKY	Key input disable signal	F053#0	○	○	15.5
	INP1 to INP8	In–position signal	F104	○	○	7.2.6.1
K	KEY1 to KEY4	Memory protect signal	G046#3 to #6	○	○	12.2.3
	KEYO	Software operator's panel signal(KEY1 to KEY4)	F075#6	○	○	12.1.15
L	LDT1A	Load detection signal 1 (serial spindle)	F045#4	○	○	9.2
	LDT1B		F049#4	○	○	9.2
	LDT2A	Load detection signal 2 (serial spindle)	F045#5	○	○	9.2
	LDT2B		F049#5	○	○	9.2
M	M00 to M31	Miscellaneous function code signal	F010 to F013	○	○	8.1
	M200 to M215	2nd M function code signal	F014 to F015	○	○	8.3
	M300 to M315	3rd M function code signal	F016 to F017	○	○	8.3
	MA	CNC ready signal	F001#7	○	○	2.2
	MABSM	Manual absolute check signal	F004#2	○	○	5.4
	MAFL	Miscellaneous function lock check signal	F004#4	○	○	8.2
	MBDT1,MBDT2 to MBDT9	Optional block skip check signal	F004#0,F005	○	○	5.5
	MCFNA	Power line switch completion signal (serial spindle)	G071#3	○	○	9.2
	MCFNB		G075#3	○	○	9.2
	MD1,MD2,MD4	Mode selection signal	G043#0 to #2	○	○	2.6
	MD1O	Software operator's panel signal(MD1)	F073#0	○	○	12.1.15
	MD2O	Software operator's panel signal(MD2)	F073#1	○	○	12.1.15
	MD4O	Software operator's panel signal(MD4)	F073#2	○	○	12.1.15
	MDRN	Dry run check signal	F002#7	○	○	5.3.2

Group	Symbol	Signal name	Address	T series	M series	Refer-ence item
M	MDTCH1 to MDTCH8	Controlled axis detach status signal	F110	○	○	1.2.4
	MEDT	Memory edit select check signal	F003#6	○	○	2.6
	MF	Auxiliary function strobe signal	F007#0	○	○	8.1
	MF2	2nd M function strobe signal	F008#4	○	○	8.3
	MF3	3rd M function strobe signal	F008#5	○	○	8.3
	MFIN	Auxiliary function completion signal	G005#0	○	○	8.4
	MFIN2	2nd M function completion signal	G004#4	○	○	8.4
	MFIN3	3rd M function completion signal	G004#5	○	○	8.4
	MH	Manual handle feed select check signal	F003#1	○	○	2.6
	MHPCC	HPCC mode signal	F066#6	-	○	7.1.14
	MI1 to MI8	Mirror image signal	G106	○	○	1.2.6
	MINC	Incremental feed select check signal	F003#0	○	○	2.6
	MINP	External program input start signal	G058#0	○	○	13.7
	MIX1 to MIX7	Composite control axis selection signals	G128#0 to #6	●	–	1.9
	MJ	JOG feed select check signal	F003#2	○	○	2.6
	MLK	All–axis machine lock signal	G044#1	○	○	5.3.1
	MLK1 to MLK8	Each–axis machine lock signal	G108	○	○	5.3.1
	MLKO	Software operator's panel signal(MLK)	F075#4	○	○	12.1.15
	MMDI	Manual data input select check signal	F003#3	○	○	2.6
	MMEM	Automatic operation select check signal	F003#5	○	○	2.6
	MMI1 to MMI8	Mirror image check signal	F108	○	○	1.2.6
	MMLK	All–axis machine lock check signal	F004#1	○	○	5.3.1
	MP1,MP2	Manual handle feed amount selection signal (incremental feed signal)	G019#4,#5	○	○	3.2
	MP1O	Software operator's panel signal(MP1)	F076#0	○	○	12.1.15
	MP2O	Software operator's panel signal(MP2)	F076#1	○	○	12.1.15
	MRDYA	Machine ready signal (serial spindle)	G070#7	○	○	9.2
	MRDYB		G074#7	○	○	9.2
	MREF	Manual reference position return selection check signal	F004#5	○	○	4.1
	MRMT	DNC operation select check signal	F003#4	○	○	5.11

Group	Symbol	Signal name	Address	T series	M series	Refer- ence item
M	MSBK	Single block check signal	F004#3	○	○	5.3.3
	MTCHIN	TEACH IN select check signal	F003#7	○	○	2.6
	MV1 to MV8	Axis moving signal	F102	○	○	1.2.5
	MVD1 to MVD8	Axis moving direction signal	F106	○	○	1.2.5
N	NOWT	No–wait signal	G063#1	●	●	8.5
	NPOS1 to NPOS8	Position display neglect signal	G198	○	○	12.1.10
O	OFN0 to OFN5,OFN6	Tool offset number select signal	G039#0 to #5,G040#0	○	-	14.4.2
	OP	Automatic operation signal	F000#7	○	○	5.1
	ORARA	Orientation completion signal (serial spindle)	F045#7	○	○	9.2
	ORARB		F049#7	○	○	9.2
	ORCMA	Orientation command signal (serial spindle)	G070#6	○	○	9.2
	ORCMB		G074#6	○	○	9.2
	OUT0 to OUT7	Software operator's panel general–purpose switch signal	F072	○	○	12.1.15
	OVC	Override cancel signal	G006#4	○	○	7.1.7.4
	OVCE	Override cancellation signal (PMC axis control)	G150#5	○	○	15.1
	OVLS1 to OVLS7	Superimposed control axis selection signals	G190#0 to #6	●	-	1.9
P	PC2SLC	2nd position coder selection signal	G028#7	○	-	9.10
	PECK2	Small–diameter peck drilling in progress signal	F066#5	-	○	11.17
	PK1 to PK8	Parking signals	G122	○	-	1.8
P	PK1 to PK7	Parking signals	G122#0 to #6	●	-	1.9
	PN1,PN2,PN4,PN8, PN16	Workpiece number search signal	G009#0 to 4	○	○	15.3
	PRC	Position record signal	G040#6	○	-	14.4.1
	PRGDPL	program screen display mode signal	F053#1	○	○	15.5
	PRTSF	Target parts count reached signal	F062#7	○	○	12.1.11
	PSAR	Spindle polygon speed arrival signal	F063#2	○	-	6.10.2
	PSE1	Master axis not arrival signal	F063#0	○	-	6.10.2
	PSE2	Polygon synchronous axis not arrival signal	F063#1	○	-	6.10.2
	PSW01 to PSW10	Position switch signal	F070#0 to F071#1	○	○	1.2.9
	PSYN	Polygon synchronization under way signal	F063#7	○	-	6.10.1

Group	Symbol	Signal name	Address	T series	M series	Refer-ence item
R	R01I to R12I	Spindle motor speed command signal	G032#0 to G033#3	○	○	15.4
	R01I2 to R12I2		G034#0 to G035#3	○	○	15.4
	R01I3 to R12I3		G036#0 to G037#3	○	○	15.4
	R01O to R12O	S12–bit code signal	F036#0 to F037#3	○	○	9.3
	RCFNA	Output switch completion signal(serial spindle)	F046#3	○	○	9.2
	RCFNB		F050#3	○	○	9.2
	RCHA	Power line status check signal (serial spindle)	G071#7	○	○	9.2
	RCHB		G075#7	○	○	9.2
	RCHPA	Output switch signal (serial spindle)	F046#2	○	○	9.2
	RCHPB		F050#2	○	○	9.2
	RGSPM	Spindle rotation direction signal	F065#1	-	○	9.11
	RGSPP		F065#0	-	○	9.11
	RGTAP	Rigid tapping signal	G061#0	○	○	9.11
	RGTSP1,RGTSP2	Rigid tapping spindle selection signal	G061#4,#5	○	-	9.11
	RLSOT	Stroke limit release signal	G007#7	-	○	2.3.2
	RMTDI0 to RMTDI7	Input signal for remote buffer	G052	○	○	13.2
	RMTDO0 to RMTDO7	Output signal for remote buffer	F069	○	○	13.2
	ROV1,ROV2	Rapid traverse override signal	G014#0,#1	○	○	7.1.7.1
	ROV1E,ROV2E	Rapid traverse override signal(PMC axis control)	G150#0,#1	○	○	15.1
	ROV1O	Software operator's panel signal(ROV1)	F076#4	○	○	12.1.15
	ROV2O	Software operator's panel signal(ROV2)	F076#5	○	○	12.1.15
	RPALM	Read/punch alarm signal	F053#3	○	○	13.5
	RPBSY	Read/punch busy signal	F053#2	○	○	13.5
	RPDO	Rapid traversing signal	F002#1	○	○	2.8
	RRW	Reset&rewind signal	G008#6	○	○	5.2
	RSLA	Output switch request signal (serial spindle)	G071#6	○	○	9.2
	RSLB		G075#6	○	○	9.2
	RST	Reset signal	F001#1	○	○	5.2
	RT	Manual rapid traverse selection signal	G019#7	○	○	3.1
	RTAP	Rigid tapping in–progress signal	F076#3	○	○	9.11

Group	Symbol	Signal name	Address	T series	M series	Reference item
R	RTE	Manual rapid traverse selection signal (PMC axis control)	G150#6	○	○	15.1
	RTO	Software operator's panel signal(RT)	F077#6	○	○	12.1.15
	RVS	Retrace signal	G007#0	-	○	11.15
	RVSL	Retrace–in–progress signal	F082#2	-	○	11.15
	RWD	Rewinding signal	F000#0	○	○	5.2
S	S00 to S31	Spindle speed code signal	F022 to F025	○	○	8.1
	SA	Servo ready signal	F000#6	○	○	2.2
	SAR	Spindle speed arrival signal	G029#4	○	○	9.3
	SARA	Speed arrival signal (serial spindle)	F045#3	○	○	9.2
	SARB		F049#3	○	○	9.2
	SBK	Single block signal	G046#1	○	○	5.3.3
	SBKO	Software operator's panel signal(SBK)	F075#3	○	○	12.1.15
	SCLP	Spindle clamp signal	F038#0	○	-	9.8
	SDTA	Speed detection signal (serial spindle)	F045#2	○	○	9.2
	SDTB		F049#2	○	○	9.2
	SF	Spindle speed strobe signal	F007#2	○	○	8.1
	SFIN	Spindle function completion signal	G005#2	○	○	8.4
	SFRA	CW command signal (serial spindle)	G070#5	○	○	9.2
	SFRB		G074#5	○	○	9.2
	SGN	Spindle motor command polarity select signal	G033#5	○	○	15.4
	SGN2		G035#5	○	○	15.4
	SGN3		G037#5	○	○	15.4
	SHA00 to SHA11	Spindle orientation external stop position command signal	G078#0 to G079#3	○	○	9.13
	SHB00 to SHB11		G080#0 to G081#3	○	○	9.13
	SIND	Spindle motor speed command select signal	G033#7	○	○	15.4
	SIND2		G035#7	○	○	15.4
	SIND3		G037#7	○	○	15.4
	SKIP	Skip signal	X004#7	○	○	14.3.1
		Overload torque signal	X004#7	-	○	11.17
	SKIP2 to SKIP6, SKIP7,SKIP8	Skip signal	X004#2 to #6, #0,#1	○	○	14.3.3
	SKIPP		G006#6	○	-	14.3.1
	SLPCA,SLPCB	Spindle return select signal	G064#2,#3	●	-	9.4
	SLSPA,SLSPB	Spindle command select signal	G063#2,#3	●	-	9.4
	SMZ	Error detect signal	G053#6	○	-	7.2.6.3

Group	Symbol	Signal name	Address	T series	M series	Reference item
S	SOCNA	Soft start/stop cancel signal (serial spindle)	G071#4	○	○	9.2
	SOCNB		G075#4	○	○	9.2
	SOR	Spindle orientation signal	G029#5	○	○	9.3
	SOV0 to SOV7	Spindle speed override signal	G030	○	○	9.3
	SPAL	Spindle fluctuation detection alarm signal	F035#0	○	○	9.6
	SPL	Feed hold lamp signal	F000#4	○	○	5.1
	SPO	Software operator's panel signal(*SP)	F075#7	○	○	12.1.15
	SPPHS	Spindle phase synchronous control signal	G038#3	○	○	9.12
	SPSLA	Spindle select signal (serial spindle)	G071#2	○	○	9.2
	SPSLB		G075#2	○	○	9.2
	SPSTP	Spindle stop complete signal	G028#6	○	-	9.8
	SPSYC	Spindle synchronous control signal	G038#2	○	○	9.12
	SRN	Program restart signal	G006#0	○	○	5.7
	SRNMV	Program restart under way signal	F002#4	○	○	5.7
	SRVA	CCW command signal (serial spindle)	G070#4	○	○	9.2
	SRVB		G074#4	○	○	9.2
	SSIN	Spindle motor command polarity select signal	G033#6	○	○	15.4
	SSIN2		G035#6	○	○	15.4
	SSIN3		G037#6	○	○	15.4
	SSTA	Speed zero signal (serial spindle)	F045#1	○	○	9.2
	SSTB		F049#1	○	○	9.2
	ST	Cycle start lamp signal	G007#2	○	○	5.1
	STL	Cycle start signal	F000#5	○	○	5.1
	STLK	Start lock signal	G007#1	○	-	2.5
	STRD	Input and run simultaneous mode select signal	G058#5	-	○	13.6
	STWD	Output and run simultaneous mode select signal	G058#6	-	○	13.6
	SUCLP	Spindle unclamp signal	F038#1	○	-	9.8
	SVF1 to SVF8	Servo off signal	G126	○	○	1.2.8
	SWS1	Spindle selection signal	G027#0	○	-	9.10
	SWS2		G027#1	○	-	9.10
	SWS3		G027#2	○	-	9.10
	SYCAL	Spindle synchronous control alarm signal	F044#4	○	○	9.12

Group	Symbol	Signal name	Address	T series	M series	Reference item
S	SYN1O to SYN8O	Synchronous control under way signals	F118	○	-	1.8
	SYN1O to SYN7O	Synchronous/composite/superimposed control under way signals	F118#0 to #6	●	-	1.9
	SYNC1 to SYNC8	Simple synchronous axis select signal	G138	○	○	1.6
	SYNC to SYNC8	Synchronous control axis selection signals	G138	○	-	1.8
	SYNC to SYNC7	Synchronous control axis selection signals	G138#0 to #6	●	-	1.9
	SYNCJ1 to SYNCJ8	Simple synchronous manual feed axis select signal	G140	-	○	1.6
T	T00 to T31	Tool function code signal	F026 to F029	○	○	8.1
	TAP	Tapping signal	F001#5	○	○	11.7
	TF	Tool function strobe signal	F007#3	○	○	8.1
	TFIN	Tool function completion signal	G005#3	○	○	8.4
	THRD	Thread cutting signal	F002#3	○	○	6.4.1
	TIALM	Tool post interference alarm signal	F064#7	●	-	2.3.5
	TICHK	Tool post interference check signal	F064#6	●	-	2.3.5
	TL01 to TL64	Tool group number select signal	G047#0 to #6	○	-	10.3
	TL01 to TL256		G047#0 to G048#0	-	○	10.3
	TLCH	Tool change signal	F064#0	○	○	10.3
	TLCHI	Individual tool change signal	F064#2	-	○	10.3
	TLMA	Torque limit signal (serial spindle)	F045#6	○	○	9.2
	TLMB		F049#6	○	○	9.2
	TLMHA	Torque limit command HIGH signal (serial spindle)	G070#1	○	○	9.2
	TLMHB		G074#1	○	○	9.2
	TLMLA	Torque limit command LOW signal(serial spindle)	G070#0	○	○	9.2
	TLMLB		G074#0	○	○	9.2
	TLNW	New tool select signal	F064#1	○	○	10.3
	TLRST	Tool change reset signal	G048#7	○	○	10.3
	TLRSTI	Individual tool change reset signal	G048#6	-	○	10.3
	TLSKP	Tool skip signal	G048#5	○	○	10.3
	TMRON	General–purpose integrating meter start signal	G053#0	○	○	12.1.11
	TRACT	Tool retraction mode signal	F092#3	○	○	5.8
	TRESC	Tool retraction signal	G059#0	○	○	5.8
	TRQL1 to TRQL8	Torque limit reached signal	F114	○	-	14.3.4
	TRRTN	Tool return signal	G059#1	○	○	5.8
T	TRSPS	Tool return completion signal	F092#5	○	○	5.8

Group	Symbol	Signal name	Address	T series	M series	Refer-ence item
U	UI000 to UI015	Input signal for custom macro	G054,G055	○	○	11.6.1
	UINT	Interrupt signal for custom macro	G053#3	○	○	11.6.2
	UO000 to UO015	Output signal for custom macro	F054,F055	○	○	11.6.1
	UO100 to UO131		F056 to F059	○	○	11.6.1
W	WATO	Waiting signal	F063#6	●	●	8.5
	WOQSM	Workpiece coordinate system shift value write mode select signal	G039#6	○	-	14.4.2
	WOSET	Workpiece coordinate system shift value write signal	G040#7	○	-	14.4.2
X	XAE	Measuring position reached signal	X004#0	○	○	14.2
Y	YAE		X004#1	-	○	14.2
Z	ZAE		X004#1	○	-	14.2
	ZAE		X004#2	-	○	14.2
	ZP1 to ZP8	Reference position return end signal	F094	○	○	4.1
	ZP21 to ZP28	2nd reference position return end signal	F096	○	○	4.5
	ZP31 to ZP38	3rd reference position return end signal	F098	○	○	4.5
	ZP41 to ZP48	4th reference position return end signal	F100	○	○	4.5
	ZPX	Spindle orientation completion signal	F094	○	-	9.8
	ZRF1 to ZRF8	Reference position establishment signal	F120	○	○	4.1
	ZRN	Manual reference position return selection signal	G043#7	○	○	4.1
	ZRNO	Software operator's panel signal(ZRN)	F073#4	○	○	12.1.15

A.2.3
List of Signals (In order of addresses)

○ : Available
● : Available only with 2–path control
– : Unavailable

Address	Signal name	Symbol	T series	M series	Refer-ence Item
X004#0	Measuring position reached signal	XAE	○	○	14.2
X004#1		YAE	-	○	14.2
X004#1		ZAE	○	-	14.2
X004#2		ZAE	-	○	14.2
X004#2,#4	Manual feed interlock signal for each axis	+MIT1,+MIT2	○	-	2.5
X004#2,#4	Tool offset write signal	+MIT1,+MIT2	○	-	14.4.2
X004#2 to #6,#0,#1	Skip signal	SKIP2 to SKIP6, SKIP7,SKIP8	○	○	14.3.3
X004#3,#5	Manual feed interlock signal for each axis	-MIT1,-MIT2	○	-	14.4.2
X004#3,#5	Tool offset write signal	-MIT1,-MIT2	○	-	14.4.2
X004#6	Skip signal (PMC axis control)	ESKIP	○	○	15.1
X004#7	Skip signal	SKIP	○	○	14.3.1
X004#7	Overload torque signal	SKIP	-	○	11.17
X008#4	Emergency stop signal	*ESP	○	○	2.1
X009	Reference position return deceleration signal	*DEC1 to *DEC8	○	○	4.1
G000,G001	Data signal for external data input	ED0 to ED15	○	○	15.2
G002#0 to #6	Address signal for external data input	EA0 to EA6	○	○	15.2
G002#7	Read signal for external data input	ESTB	○	○	15.2
G004#3	End signal	FIN	○	○	8.1
G004#4	2nd M function completion signal	MFIN2	○	○	8.4
G004#5	3rd M function completion signal	MFIN3	○	○	8.4
G005#0	Auxiliary function completion signal	MFIN	○	○	8.4
G005#1	External operation function completion signal	EFIN	-	○	8.4
G005#2	Spindle function completion signal	SFIN	○	○	8.4
G005#3	Tool function completion signal	TFIN	○	○	8.4
G005#4	2nd auxiliary function completion signal	BFIN	○	-	8.4
G005#6	Auxiliary function lock signal	AFL	○	○	8.2
G005#7	2nd auxiliary function completion signal	BFIN	-	○	8.4
G006#0	Program restart signal	SRN	○	○	5.7
G006#2	Manual absolute signal	*ABSM	○	○	5.4
G006#4	Override cancel signal	OVC	○	○	7.1.7.4
G006#6	Skip signal	SKIPP	○	-	14.3.1
G007#0	Retrace signal	RVS	-	○	11.15
G007#1	Start lock signal	STLK	○	-	2.5

Address	Signal name	Symbol	T series	M series	Reference Item
G007#2	Cycle start signal	ST	○	○	5.1
G007#5	Follow–up signal	*FLWU	○	○	1.2.7
G007#6	Stored stroke limit select signal	EXLM	○	○	2.3.2
G007#7	Stroke limit release signal	RLSOT	-	○	2.3.2
G008#0	Interlock signal	*IT	○	○	2.5
G008#4	Emergency stop signal	*ESP	○	○	2.1
G008#5	Feed hold signal	*SP	○	○	5.1
G008#6	Reset & rewind signal	RRW	○	○	5.2
G008#7	External reset signal	ERS	○	○	5.2
G009#0 to 4	Workpiece number search signal	PN1,PN2,PN4, PN8,PN16	○	○	15.3
G010,G011	Manual feedrate override signal	*JV0 to *JV15	○	○	3.1
G012	Feedrate override signal	*FV0 to *FV7	○	○	7.1.7.2
G013	2nd feedrate override signal	*AFV0 to *AFV7	○	○	7.1.7.3
G014#0,#1	Rapid traverse override signal	ROV1,ROV2	○	○	7.1.7.1
G016#7	F1–digit feed select signal	F1D	-	○	7.1.5
G018#0 to #3	Manual handle feed axis selection signal	HS1A to HS1D	○	○	3.2
G018#4 to #7		HS2A to HS2D	○	○	3.2
G019#0 to #3		HS3A to HS3D	-	○	3.2
G019#4,#5	Manual handle feed amount selection signal (incremental feed signal)	MP1,MP2	○	○	3.2
G019#7	Manual rapid traverse selection signal	RT	○	○	3.1
G023#7	Tool axis direction handle feed mode signal	ALNGH	-	○	3.4
G027#0	Spindle selection signal	SWS1	○	-	9.10
G027#1		SWS2	○	-	9.10
G027#2		SWS3	○	-	9.10
G027#3	Stop signal in each spindle	*SSTP1	○	-	9.10
G027#4		*SSTP2	○	-	9.10
G027#5		*SSTP3	○	-	9.10
G027#7	Cs contour control switch signal	CON	○	○	9.9
G028#1,#2	Gear selection signal (input)	GR1,GR2	○	○	9.3
G028#4	Spindle unclamp completion signal	*SUCPF	○	-	9.8
G028#5	Spindle clamp completion signal	*SCPF	○	-	9.8
G028#6	Spindle stop complete signal	SPSTP	○	-	9.8
G028#7	2nd position coder selection signal	PC2SLC	○	-	9.10
G029#0	Gear select signal (input)	GR21	○	-	9.10
G029#2		GR31	○	-	9.10
G029#4	Spindle speed arrival signal	SAR	○	○	9.3
G029#5	Spindle orientation signal	SOR	○	○	9.3

Address	Signal name	Symbol	T series	M series	Refer- ence Item
G029#6	Spindle stop signal	*SSTP	○	○	9.3
G030	Spindle speed override signal	SOV0 to SOV7	○	○	9.3
G032#0 to G033#3	Spindle motor speed command signal	R01I to R12I	○	○	15.4
G033#5	Spindle motor command polarity select signal	SGN	○	○	15.4
G033#6	Spindle motor command polarity select signal	SSIN	○	○	15.4
G033#7	Spindle motor speed command select signal	SIND	○	○	15.4
G034#0 to G035#3	Spindle motor speed command signal	R01I2 to R12I2	○	○	15.4
G035#5	Spindle motor command polarity select signal	SGN2	○	○	15.4
G035#6	Spindle motor command polarity select signal	SSIN2	○	○	15.4
G035#7	Spindle motor speed command select signal	SIND2	○	○	15.4
G036#0 to G037#3	Spindle motor speed command signal	R01I3 to R12I3	○	○	15.4
G037#5	Spindle motor command polarity select signal	SGN3	○	○	15.4
G037#6	Spindle motor command polarity select signal	SSIN3	○	○	15.4
G037#7	Spindle motor speed command select signal	SIND3	○	○	15.4
G038#0	Polygon spindle stop signal	*PLSST	○	-	6.10.2
G038#2	Spindle synchronous control signal	SPSYC	○	○	9.12
G038#3	Spindle phase synchronous control signal	SPPHS	○	○	9.12
G038#6	B–axis unclamp completion signal	*BEUCP	-	○	11.11
G038#7	B–axis clamp completion signal	*BECLP	-	○	11.11
G039#0 to #5,G040#0	Tool offset number select signal	OFN0 to OFN5, OFN6	○	-	14.4.2
G039#6	Workpiece coordinate system shift value write mode select signal	WOQSM	○	-	14.4.2
G039#7	Tool offset value write mode select signal	GOQSM	○	-	14.4.2
G040#6	Position record signal	PRC	○	-	14.4.1
G040#7	Workpiece coordinate system shift value write signal	WOSET	○	-	14.4.2
G041#0 to #3	Manual handle interrupt axis selection signal	HS1IA to HS1ID	○	○	3.3
G041#4 to #7	Manual handle interrupt axis selection signal	HS2IA to HS2ID	○	○	3.3
G042#0 to #3	Manual handle interrupt axis selection signal	HS3IA to HS3ID	-	○	3.3
G042#7	Direct operation select signal	DMMC	○	○	15.6
G043#0 to #2	Mode selection signal	MD1,MD2,MD4	○	○	2.6
G043#5	DNC operation select signal	DNCI	○	○	5.11
G043#7	Manual reference position return selection signal	ZRN	○	○	4.1
G044#0,G045	Optional block skip signal	BDT1, BDT2 to BDT9	○	○	5.5
G044#1	All–axis machine lock signal	MLK	○	○	5.3.1
G046#1	Single block signal	SBK	○	○	5.3.3

Address	Signal name	Symbol	T series	M series	Refer- ence Item
G046#3 to #6	Memory protect signal	KEY1 to KEY4	○	○	12.2.3
G046#7	Dry run signal	DRN	○	○	5.3.2
G047#0 to #6	Tool group number select signal	TL01 to TL64	○	-	10.3
G047#0 to G048#0		TL01 to TL256	-	○	10.3
G048#5	Tool skip signal	TLSKP	○	○	10.3
G048#6	Individual tool change reset signal	TLRSTI	-	○	10.3
G048#7	Tool change reset signal	TLRST	○	○	10.3
G049#0 to G050#1	Tool life count override signal	*TLV0 to *TLV9	-	○	10.3
G052	Input signal for remote buffer	RMTDI0 to RMTDI7	○	○	13.2
G053#0	General–purpose integrating meter start signal	TMRON	○	○	12.1.11
G053#3	Interrupt signal for custom macro	UINT	○	○	11.6.2
G053#6	Error detect signal	SMZ	○	-	7.2.6.3
G053#7	Chamferring signal	CDZ	○	-	11.9
G054,G055	Input signal for custom macro	UI000 to UI015	○	○	11.6.1
G058#0	External start signal program input	MINP	○	○	13.7
G058#1	External read start signal	EXRD	○	○	13.5
G058#2	External read/punch stop signal	EXSTP	○	○	13.5
G058#3	External punch start signal	EXWT	○	○	13.5
G058#5	Input and run simultaneous mode select signal	STRD	-	○	13.6
G058#6	Output and run simultaneous mode select signal	STWD	-	○	13.6
G059#0	Tool retraction signal	TRESC	○	○	5.8
G059#1	Tool return signal	TRRTN	○	○	5.8
G060#7	Tail stock barrier select signal	*TSB	○	-	2.3.4
G061#0	Rigid tapping signal	RGTAP	○	○	9.11
G061#4,#5	Rigid tap spindle select signal	RGTSP1, RGTSP2	○	-	9.11
G063#0	Path selection signal (Tool post selection signal)	HEAD	●	●	2.7
G063#1	No–wait signal	NOWT	●	●	8.5
G063#2,#3	Spindle command select signal	SLSPA,SLSPB	●	-	9.4
G064#2,#3	Spindle feedback select signal	SLPCA,SLPCB	●	-	9.4
G066#0	All–axis VRDY OFF alarm ignore signal	IGNVRY	○	○	2.9
G066#1	External key input mode selection signal	ENBKY	○	○	15.5
G066#7	Key code read signal	EKSET	○	○	15.5
G070#0	Torque limit command LOW signal (serial spindle)	TLMLA	○	○	9.2
G070#1	Torque limit command HIGH signal (serial spindle)	TLMHA	○	○	9.2
G070#3,#2	Clutch/gear signal (serial spindle)	CTH1A,CTH2A	○	○	9.2
G070#4	CCW command signal (serial spindle)	SRVA	○	○	9.2

Address	Signal name	Symbol	T series	M series	Refer-ence Item
G070#5	CW command signal (serial spindle)	SFRA	○	○	9.2
G070#6	Orientation command signal (serial spindle)	ORCMA	○	○	9.2
G070#7	Machine ready completion signal (serial spindle)	MRDYA	○	○	9.2
G071#0	Alarm reset signal (serial spindle)	ARSTA	○	○	9.2
G071#1	Emergency stop signal (serial spindle)	*ESPA	○	○	9.2
G071#2	Spindle select signal (serial spindle)	SPSLA	○	○	9.2
G071#3	Power line switch completion signal (serial spindle)	MCFNA	○	○	9.2
G071#4	Soft start stop cancel signal (serial spindle)	SOCNA	○	○	9.2
G071#6	Output switch request signal (serial spindle)	RSLA	○	○	9.2
G071#7	Power line status check signal (serial spindle)	RCHA	○	○	9.2
G074#0	Torque limit command LOW signal (serial spindle)	TLMLB	○	○	9.2
G074#1	Torque limit command HIGH signal (serial spindle)	TLMHB	○	○	9.2
G074#3,#2	Clutch/gear signal (serial spindle)	CTH1B,CTH2B	○	○	9.2
G074#4	CCW command signal (serial spindle)	SRVB	○	○	9.2
G074#5	CW command signal (serial spindle)	SFRB	○	○	9.2
G074#6	Orientation command signal (serial spindle)	ORCMB	○	○	9.2
G074#7	Machine ready completion signal (serial spindle)	MRDYB	○	○	9.2
G075#0	Alarm reset signal (serial spindle)	ARSTB	○	○	9.2
G075#1	Emergency stop signal (serial spindle)	*ESPB	○	○	9.2
G075#2	Spindle select signal (serial spindle)	SPSLB	○	○	9.2
G075#3	Power line switch completion signal (serial spindle)	MCFNB	○	○	9.2
G075#4	Soft start stop cancel signal (serial spindle)	SOCNB	○	○	9.2
G075#6	Output switch request signal (serial spindle)	RSLB	○	○	9.2
G075#7	Power line status check signal (serial spindle)	RCHB	○	○	9.2
G078#0 to G079#3	Spindle orientation external stop position command signal	SHA00 to SHA11	○	○	9.13
G080#0 to G081#3		SHB00 to SHB11	○	○	9.13
G096#0 to #6	1% step rapid traverse override signal	*HROV0 to *HROV6	○	○	7.1.7.1
G096#7	1% step rapid traverse override select signal	HROV	○	○	7.1.7.1
G098	Key code signal	EKC0 to EKC7	○	○	15.5
G100	Feed axis and direction selection signal	+J1 to +J8	○	○	3.1
G102		-J1 to -J8	○	○	3.1
G106	Mirror image signal	MI1 to MI8	○	○	1.2.6
G108	Each–axis machine lock signal	MLK1 to MLK8	○	○	5.3.1
G110	Stroke limit external setting signal	+LM1 to +LM8	-	○	2.3.2
G112		-LM1 to -LM8	-	○	2.3.2
G114	Overtravel signal	*+L1 to *+L8	○	○	2.3.1
G116		*-L1 to *-L8	○	○	2.3.1

Address	Signal name	Symbol	T series	M series	Reference Item
G118	External deceleration signal	*+ED1 to *+ED8	○	○	7.1.9
G120		*-ED1 to *-ED8	○	○	7.1.9
G122	Parking signals	PK1 to PK8	○	-	1.8
G122#0 to #6		PK1 to PK7	●	–	1.9
G124	Controlled axis detach signal	DTCH1 to DTCH8	○	○	1.2.4
G126	Servo off signal	SVF1 to SVF8	○	○	1.2.8
G128#0 to #6	Composite control axis selection signals	MIX1 to MIX7	●	–	1.9
G130	Interlock signal for each axis	*IT1 to *IT8	○	○	2.5
G132#0 to #3	Interlock signal for each axis and direction	+MIT1 to +MIT4	-	○	2.5
G134#0 to #3		-MIT1 to -MIT4	-	○	2.5
G136	Control axis select signal (PMC axis control)	EAX1 to EAX8	○	○	15.1
G138	Simple synchronous axis select signal	SYNC1 to SYNC8	○	○	1.6
G140	Simple synchronous manual feed axis select signal	SYNCJ1 to SYNCJ8	-	○	1.6
G138	Synchronous control axis selection signals	SYNC to SYNC8	○	-	1.8
G138#0 to #6		SYNC to SYNC7	●	-	1.9
G142#0	Auxiliary function completion signal (PMC axis control)	EFINA	○	○	15.1
G142#2	Buffering disable signal (PMC axis control)	EMBUFA	○	○	15.1
G142#3	Block stop signal (PMC axis control)	ESBKA	○	○	15.1
G142#4	Servo off signal (PMC axis control)	ESOFA	○	○	15.1
G142#5	Axis control temporary stop signal (PMC axis control)	ESTPA	○	○	15.1
G142#6	Reset signal (PMC axis control)	ECLRA	○	○	15.1
G142#7	Axis control command read signal (PMC axis control)	EBUFA	○	○	15.1
G143#0 to #6	Axis control command signal (PMC axis control)	EC0A to EC6A	○	○	15.1
G143#7	Block stop disable signal (PMC axis control)	EMSBKA	○	○	15.1
G144,G145	Axis control feedrate signal (PMC axis control)	EIF0A to EIF15A	○	○	15.1
G146 to G149	Axis control data signal (PMC axis control)	EID0A to EID31A	○	○	15.1
G150#0,#1	Rapid traverse override signal (PMC axis control)	ROV1E,ROV2E	○	○	15.1
G150#5	Override cancel signal (PMC axis control)	OVCE	○	○	15.1
G150#6	Manual rapid traverse selection signal (PMC axis control)	RTE	○	○	15.1
G150#7	Dry run signal (PMC axis control)	DRNE	○	○	15.1
G151	Feedrate override signal (PMC axis control)	*FV0E to *FV7E	○	○	15.1
G154#0	Auxiliary function completion signal (PMC axis control)	EFINB	○	○	15.1

Address	Signal name	Symbol	T series	M series	Refer-ence Item
G154#2	Buffering disable signal (PMC axis control)	EMBUFB	○	○	15.1
G154#3	Block stop signal (PMC axis control)	ESBKB	○	○	15.1
G154#4	Servo off signal (PMC axis control)	ESOFB	○	○	15.1
G154#5	Axis control temporary stop signal (PMC axis control)	ESTPB	○	○	15.1
G154#6	Reset signal (PMC axis control)	ECLRB	○	○	15.1
G154#7	Axis control command read signal (PMC axis control)	EBUFB	○	○	15.1
G155#0 to #6	Axis control command signal (PMC axis control)	EC0B to EC6B	○	○	15.1
G155#7	Block stop disable signal (PMC axis control)	EMSBKB	○	○	15.1
G156,G157	Axis control feedrate signal (PMC axis control)	EIF0B to EIF15B	○	○	15.1
G158 to G161	Axis control data signal (PMC axis control)	EID0B to EID31B	○	○	15.1
G166#0	Auxiliary function completion signal (PMC axis control)	EFINC	○	○	15.1
G166#2	Buffering disable signal (PMC axis control)	EMBUFC	○	○	15.1
G166#3	Block stop signal (PMC axis control)	ESBKC	○	○	15.1
G166#4	Servo off signal (PMC axis control)	ESOFC	○	○	15.1
G166#5	Axis control temporary stop signal (PMC axis control)	ESTPC	○	○	15.1
G166#6	Reset signal (PMC axis control)	ECLRC	○	○	15.1
G166#7	Axis control command read signal (PMC axis control)	EBUFC	○	○	15.1
G167#0 to #6	Axis control command signal (PMC axis control)	EC0C to EC6C	○	○	15.1
G167#7	Block stop disable signal (PMC axis control)	EMSBKC	○	○	15.1
G168,G169	Axis control feedrate signal (PMC axis control)	EIF0C to EIF15C	○	○	15.1
G170 to G173	Axis control data signal (PMC axis control)	EID0C to EID31C	○	○	15.1
G178#0	Auxiliary function completion signal (PMC axis control)	EFIND	○	○	15.1
G178#2	Buffering disable signal (PMC axis control)	EMBUFD	○	○	15.1
G178#3	Block stop signal (PMC axis control)	ESBKD	○	○	15.1
G178#4	Servo off signal (PMC axis control)	ESOFD	○	○	15.1
G178#5	Axis control temporary stop signal (PMC axis control)	ESTPD	○	○	15.1
G178#6	Reset signal (PMC axis control)	ECLRD	○	○	15.1
G178#7	Axis control command read signal (PMC axis control)	EBUFD	○	○	15.1
G179#0 to #6	Axis control command signal (PMC axis control)	EC0D to EC6D	○	○	15.1
G179#7	Block stop disable signal (PMC axis control)	EMSBKD	○	○	15.1
G180,G181	Axis control feedrate signal (PMC axis control)	EIF0D to EIF15D	○	○	15.1
G182 to G185	Axis control data signal (PMC axis control)	EID0D to EID31D	○	○	15.1
G190#0 to #6	Superimposed control axis selection signals	OVLS1 to OVLS7	●	-	1.9
G192	Each–axis VRDY OFF alarm ignore signal	IGVRY1 to IGVRY8	○	○	2.9

Address	Signal name	Symbol	T series	M series	Refer-ence Item
G198	Position display neglect signal	NPOS1 to NPOS8	○	○	12.1.10
F000#0	Rewinding signal	RWD	○	○	5.2
F000#4	Feed hold lamp signal	SPL	○	○	5.1
F000#5	Cycle start lamp signal	STL	○	○	5.1
F000#6	Servo ready completion signal	SA	○	○	2.2
F000#7	Automatic operation signal	OP	○	○	5.1
F001#0	Alarm signal	AL	○	○	2.4
F001#1	Resetting signal	RST	○	○	5.2
F001#2	Battery alarm signal	BAL	○	○	2.4
F001#3	Distribution end signal	DEN	○	○	8.1
F001#4	Spindle enable signal	ENB	○	○	9.3
F001#5	Tapping signal	TAP	○	○	11.7
F001#7	CNC signal	MA	○	○	2.2
F002#0	Inch input signal	INCH	○	○	11.4
F002#1	Rapid traversing signal	RPDO	○	○	2.8
F002#2	Constant surface speed signal	CSS	○	○	9.5
F002#3	Thread cutting signal	THRD	○	○	6.4.1
F002#4	Program restart under way signal	SRNMV	○	○	5.7
F002#6	Cutting feed signal	CUT	○	○	2.8
F002#7	Dry run check signal	MDRN	○	○	5.3.2
F003#0	Incremental feed select check signal	MINC	○	○	2.6
F003#1	Manual handle feed select check signal	MH	○	○	2.6
F003#2	Jog feed select check signal	MJ	○	○	2.6
F003#3	Manual data input select check signal	MMDI	○	○	2.6
F003#4	DNC operation selection confirm signal	MRMT	○	○	5.11
F003#5	Automatic operation select check signal	MMEM	○	○	2.6
F003#6	Memory edit select check signal	MEDT	○	○	2.6
F003#7	TEACH IN select check signal	MTCHIN	○	○	2.6
F004#0,F005	Optional block skip check signal	MBDT1,MBDT 2 to MBDT9	○	○	5.5
F004#1	All–axis machine lock check signal	MMLK	○	○	5.3.1
F004#2	Manual absolute check signal	MABSM	○	○	5.4
F004#3	Single block check signal	MSBK	○	○	5.3.3
F004#4	Auxiliary function lock check signal	MAFL	○	○	8.2
F004#5	Manual reference position return selection check signal	MREF	○	○	4.1
F007#0	Miscellaneous function strobe signal	MF	○	○	8.1
F007#1	Externaloperation signal for high–speed interface	EFD	-	○	8.4
F007#2	Spindle–speed function strobe signal	SF	○	○	8.1
F007#3	Tool function strobe signal	TF	○	○	8.1

Address	Signal name	Symbol	T series	M series	Refer-ence Item
F007#4	2nd auxiliary function strobe signal	BF	○	-	8.1
F007#7		BF	-	○	8.1
F008#0	External operation signal	EF	-	○	11.8
F008#4	2nd M function strobe signal	MF2	○	○	8.3
F008#5	3rd M function strobe signal	MF3	○	○	8.3
F009#4	Decode M signal	DM30	○	○	8.1
F009#5		DM02	○	○	8.1
F009#6		DM01	○	○	8.1
F009#7		DM00	○	○	8.1
F010 to F013	Miscellaneous function code signal	M00 to M31	○	○	8.1
F014 to F015	2nd M function code signal	M200 to M215	○	○	8.3
F016 to F017	3rd M function code signal	M300 to M315	○	○	8.3
F022 to F025	Spindle speed code signal	S00 to S31	○	○	8.1
F026 to F029	Tool function code signal	T00 to T31	○	○	8.1
F030 to F033	2nd miscellaneous function code signal	B00 to B31	○	○	8.1
F034#0 to #2	Gear selection signal (output)	GR1O,GR2O, GR3O	-	○	9.3
F035#0	Spindle fluctuation detection alarm signal	SPAL	○	○	9.6
F036#0 to F037#3S	12−bit code signal	R01O to R12O	○	○	9.3
F038#0	Spindle clamp signal	SCLP	○	-	9.8
F038#1	Spindle unclamp signal	SUCLP	○	-	9.8
F038#2	Spindle enable signal	ENB2	○	-	9.10
F038#3		ENB3	○	-	9.10
F040,F041	Actual spindle speed signal	AR0 to AR15	○	-	9.7
F044#1	Cs contour control switch completion signal	FSCSL	○	○	9.9
F044#2	Spindle synchronous speed control completion signal	FSPSY	○	○	9.12
F044#3	Spindle phase synchronous control completion signal	FSPPH	○	○	9.12
F044#4	Spindle synchronous control alarm signal	SYCAL	○	○	9.12
F045#0	Alarm signal (serial spindle)	ALMA	○	○	9.2
F045#1	Speed zero signal (serial spindle)	SSTA	○	○	9.2
F045#2	Speed detection signal (serial spindle)	SDTA	○	○	9.2
F045#3	Speed arrival signal (serial spindle)	SARA	○	○	9.2
F045#4	Load detection signal 1 (serial spindle)	LDT1A	○	○	9.2
F045#5	Load detection signal 2 (serial spindle)	LDT2A	○	○	9.2
F045#6	Torque limit signal (serial spindle)	TLMA	○	○	9.2
F045#7	Orientation completion signal (serial spindle)	ORARA	○	○	9.2
F046#0	Power line switch signal (serial spindle)	CHPA	○	○	9.2
F046#1	Spindle switch completion signal (serial spindle)	CFINA	○	○	9.2
F046#2	Output switch signal (serial spindle)	RCHPA	○	○	9.2

Address	Signal name	Symbol	T series	M series	Refer-ence Item
F046#3	Output switch completion signal (serial spindle)	RCFNA	○	○	9.2
F049#0	Alarm signal (serial spindle)	ALMB	○	○	9.2
F049#1	Speed zero signal (serial spindle)	SSTB	○	○	9.2
F049#2	Speed detection signal (serial spindle)	SDTB	○	○	9.2
F049#3	Speed arrival signal (serial spindle)	SARB	○	○	9.2
F049#4	Load detection signal 1 (serial spindle)	LDT1B	○	○	9.2
F049#5	Load detection signal 2 (serial spindle)	LDT2B	○	○	9.2
F049#6	Torque limit signal (serial spindle)	TLMB	○	○	9.2
F049#7	Orientation completion signal (serial spindle)	ORARB	○	○	9.2
F050#0	Power line switch signal (serial spindle)	CHPB	○	○	9.2
F050#1	Spindle switch completion signal (serial spindle)	CFINB	○	○	9.2
F050#2	Output switch signal (serial spindle)	RCHPB	○	○	9.2
F050#3	Output switch completion signal (serial spindle)	RCFNB	○	○	9.2
F053#0	Key input disable signal	INHKY	○	○	15.5
F053#1	Program screen display mode signal	PRGDPL	○	○	15.5
F053#2	Read/punch busy signal	RPBSY	○	○	13.5
F053#3	Read/punch alarm signal	RPALM	○	○	13.5
F053#4	Background editing signal	BGEACT	○	○	13.5
F053#7	Key code read completion signal	EKENB	○	○	15.5
F054,F055	Output signal for custom macro	UO000 to UO015	○	○	11.6.1
F056 to F059		UO100 to UO131	○	○	11.6.1
F060#0	Read completion signal for external data input	EREND	○	○	15.2
F060#1	Search completion signal for external data input	ESEND	○	○	15.2
F061#0	B–axis unclamp signal	BUCLP	-	○	11.11
F061#1	B–axis clamp signal	BCLP	-	○	11.11
F062#7	Target part count reached signal	PRTSF	○	○	12.1.11
F063#0	Master axis not arrival signal	PSE1	○	-	6.10.2
F063#1	Polygon synchronous axis not arrival signal	PSE2	○	-	6.10.2
F063#2	Spindle polygon speed arrival signal	PSAR	○	-	6.10.2
F063#6	Waiting signal	WATO	●	●	8.5
F063#7	Polygon synchronization under way signal	PSYN	○	-	6.10.1
F064#0	Tool change signal	TLCH	○	○	10.3
F064#1	New tool select signal	TLNW	○	○	10.3
F064#2	Individual tool change signal	TLCHI	-	○	10.3
F064#5	Spindle command signal	COSP	●	-	9.4
F064#6	Tool post interference check signal	TICHK	●	-	2.3.5
F064#7	tool post interference alarm signal	TIALM	●	-	2.3.5

Address	Signal name	Symbol	T series	M series	Reference Item
F065#0	Spindle rotation direction signal	RGSPP	-	○	9.11
F065#1		RGSPM	-	○	9.11
F066#0	Lock–ahead control mode signal	G08MD	-	○	7.1.13
F066#5	Small–diameter peck drilling in progress signal	PECK2	-	○	11.17
F066#6	HPCC mode signal	MHPCC	-	○	7.1.14
F066#7	HPCC operation signal	EXHPCC	-	○	7.1.14
F069	Output signal for remote buffer	RMTDO0 to RMTDO7	○	○	13.2
F070#0 to F071#1	Position switch signal	PSW01 to PSW10	○	○	1.2.9
F072	Software operator's panelgeneral–purpose switch signal	OUT0 to OUT7	○	○	12.1.15
F073#0	Software operator's panel signal (MD1)	MD1O	○	○	12.1.15
F073#1	Software operator's panel signal (MD2)	MD2O	○	○	12.1.15
F073#2	Software operator's panel signal (MD4)	MD4O	○	○	12.1.15
F073#4	Software operator's panel signal (ZRN)	ZRNO	○	○	12.1.15
F075#2	Software operator's panel signal (BDT)	BDTO	○	○	12.1.15
F075#3	Software operator's panel signal (SBK)	SBKO	○	○	12.1.15
F075#4	Software operator's panel signal (MLK)	MLKO	○	○	12.1.15
F075#5	Software operator's panel signal (DRN)	DRNO	○	○	12.1.15
F075#6	Software operator's panel signal (KEY1 to KEY4)	KEYO	○	○	12.1.15
F075#7	Software operator's panel signal (*SP)	SPO	○	○	12.1.15
F076#0	Software operator's panel signal (MP1)	MP1O	○	○	12.1.15
F076#1	Software operator's panel signal (MP2)	MP2O	○	○	12.1.15
F076#3	Rigid tapping mode signal	RTAP	○	○	9.11
F076#4	Software operator's panel signal (ROV1)	ROV1O	○	○	12.1.15
F076#5	Software operator's panel signal (ROV2)	ROV2O	○	○	12.1.15
F077#0	Software operator's panel signal (HS1A)	HS1AO	○	○	12.1.15
F077#1	Software operator's panel signal (HS1B)	HS1BO	○	○	12.1.15
F077#2	Software operator's panel signal (HS1C)	HS1CO	○	○	12.1.15
F077#3	Software operator's panel signal (HS1D)	HS1DO	○	○	12.1.15
F077#6	Software operator's panel signal (RT)	RTO	○	○	12.1.15
F078	Software operator's panel signal (*FV0 to *FV7)	*FV0O to *FV7O	○	○	12.1.15
F079,F080	Software operator's panel signal (*JV0 to *JV15)	*JV0O to *JV15O	○	○	12.1.15
F081#0,#2, #4,#6	Software operator's panel signal (+J1 to +J4)	+J1O to +J4O	○	○	12.1.15
F081#1,#3, #5,#7	Software operator's panel signal (-J1 to -J4)	-J1O to -J4O	○	○	12.1.15
F082#2	Retrace–in–progress signal	RVSL	-	○	11.15
F090#1	Servo axis abnormal load detected signal	ABTQSV	○	○	2.10

Address	Signal name	Symbol	T series	M series	Refer-ence Item
F090#1	First–spindle abnormal load detected signal	ABTSP1	○	○	2.10
F090#2	Second–spindle abnormal load detected signal	ABTSP2	○	○	2.10
F092#3	Tool retraction mode signal	TRACT	○	○	5.8
F092#5	Tool return completion signal	TRSPS	○	○	5.8
F094	Reference position return end signal	ZP1 to ZP8	○	○	4.1
	Spindle orientation completion signal	ZPX	○	-	9.8
F096	2nd reference position return end signal	ZP21 to ZP28	○	○	4.5
F098	3rd reference position return end signal	ZP31 to ZP38	○	○	4.5
F100	4th reference position return end signal	ZP41 to ZP48	○	○	4.5
F102	Axis moving signal	MV1 to MV8	○	○	1.2.5
F104	In–position signal	INP1 to INP8	○	○	7.2.6.1
F106	Axis moving direction signal	MVD1 to MVD8	○	○	1.2.5
F108	Mirror image check signal	MMI1 to MMI8	○	○	1.2.6
F110	Controlled axis detach status signal	MDTCH1 to MDTCH8	○	○	1.2.4
F112	Distribution completion signal (PMC axis control)	EADEN1 to EADEN8	○	○	15.1
F114	Torque limit reached signal	TRQL1 to TRQL8	○	-	14.3.4
F116	Floating reference position return end signal	FRP1 to FRP8	○	○	4.6
F118	Synchronous control under way signals	SYN1O to SYN8O	○	-	1.8
F118#0 to #6	Synchronous/composite/superimposed control under way signals	SYN1O to SYN7O	●	-	1.9
F120	Reference position establishment signal	ZRF1 to ZRF8	○	○	4.1
F122	High–speed skip status signal	HDO0 to HDO7	○	○	14.3.2
F129#5	Override 0% signal (PMC axis control)	EOV0	○	○	15.1
F129#7	Control axis selection status signal (PMC axis control)	*EAXSL	○	○	15.1
F130#0	In–position signal (PMC axis control)	EINPA	○	○	15.1
F130#1	Following zero checking signal (PMC axis control)	ECKZA	○	○	15.1
F130#2	Alarm signal (PMC axis control)	EIALA	○	○	15.1
F130#3	Auxiliary function executing signal (PMC axis control)	EDENA	○	○	15.1
F130#4	Axis moving signal (PMC axis control)	EGENA	○	○	15.1
F130#5	Positive–direction overtravel signal (PMC axis control)	EOTPA	○	○	15.1
F130#6	Negative–direction overtravel signal (PMC axis control)	EOTNA	○	○	15.1
F130#7	Axis control command read completion signal (PMC axis control)	EBSYA	○	○	15.1
F131#0	Auxiliary function strobe signal (PMC axis control)	EMFA	○	○	15.1
F131#1	Bufferful signal (PMC axis control)	EABUFA	○	○	15.1
F132,F142	Auxiliary function code signal (PMC axis control)	EM11A to EM48A	○	○	15.1

Address	Signal name	Symbol	T series	M series	Refer-ence Item
F133#0	In–position signal (PMC axis control)	EINPB	○	○	15.1
F133#1	Following zero checking signal (PMC axis control)	ECKZB	○	○	15.1
F133#2	Alarm signal (PMC axis control)	EIALB	○	○	15.1
F133#3	Auxiliary function executing signal (PMC axis control)	EDENB	○	○	15.1
F133#4	Axis moving signal (PMC axis control)	EGENB	○	○	15.1
F133#5	Positive–direction overtravel signal (PMC axis control)	EOTPB	○	○	15.1
F133#6	Negative–direction overtravel signal (PMC axis control)	EOTNB	○	○	15.1
F133#7	Axis control command read completion signal (PMC axis control)	EBSYB	○	○	15.1
F134#0	Auxiliary function strobe signal (PMC axis control)	EMFB	○	○	15.1
F134#1	Bufferful signal (PMC axis control)	EABUFB	○	○	15.1
F135,F145	Auxiliary function code signal (PMC axis control)	EM11B to EM48B	○	○	15.1
F136#0	In–position signal (PMC axis control)	EINPC	○	○	15.1
F136#1	Following zero checking signal (PMC axis control)	ECKZC	○	○	15.1
F136#2	Alarm signal (PMC axis control)	EIALC	○	○	15.1
F136#3	Auxiliary function executing signal (PMC axis control)	EDENC	○	○	15.1
F136#4	Axis moving signal (PMC axis control)	EGENC	○	○	15.1
F136#5	Positive–direction overtravel signal (PMC axis control)	EOTPC	○	○	15.1
F136#6	Negative–direction overtravel signal (PMC axis control)	EOTNC	○	○	15.1
F136#7	Axis control command read completion signal (PMC axis control)	EBSYC	○	○	15.1
F137#0	Auxiliary function strobe signal (PMC axis control)	EMFC	○	○	15.1
F137#1	Buffer full signal (PMC axis control)	EABUFC	○	○	15.1
F138,F148	Auxiliary function code signal (PMC axis control)	EM11C to EM48C	○	○	15.1
F139#0	In–position signal (PMC axis control)	EINPD	○	○	15.1
F139#1	Following zero checking signal (PMC axis control)	ECKZD	○	○	15.1
F139#2	Alarm signal (PMC axis control)	EIALD	○	○	15.1
F139#3	Auxiliary function executing signal (PMC axis control)	EDEND	○	○	15.1
F139#4	Axis moving signal (PMC axis control)	EGEND	○	○	15.1
F139#5	Positive–direction overtravel signal (PMC axis control)	EOTPD	○	○	15.1
F139#6	Negative–direction overtravel signal (PMC axis control)	EOTND	○	○	15.1
F139#7	Axis control command read completion signal (PMC axis control)	EBSYD	○	○	15.1
F140#0	Auxiliary function strobe signal (PMC axis control)	EMFD	○	○	15.1
F140#1	Buffer full signal (PMC axis control)	EABUFD	○	○	15.1
F141,F151	Auxiliary function code signal (PMC axis control)	EM11D to EM48D	○	○	15.1
F182	Controlling signal (PMC axis control)	EACNT1 to EACNT8	○	○	15.1

Index

– Number –

2nd reference position return/3rd, 4th reference position return, 239

– A –

Abnormal load detection, 184
Absolute position detection, 46
Acceleration/deceleration control, 416
Actual spindle speed output, 527
Alarm signal, 165
Automatic acceleration/deceleration, 416
Automatic corner deceleration, 368
Automatic corner override (M series), 359
Automatic feedrate control function , 397
Automatic operation, 245
Automatic tool length measurement (M series) / automatic tool offset (T series), 845
Auxiliary function, 442
Auxiliary function lock, 454
Axis control, 1

– B –

Background editing, 809
Backlash compensation, 35
Balance cut (2–path control for T system), 279
Bell shaped acceleration/deceleration after cutting feed , 428
Bell–shaped acceleration/deceleration for rapid traverse, 421

– C –

Canned cycle (M series) / canned cycle for hole machining, 705
Canned cycle (T series)/multiple repetitive canned cycle (T series), 717
Chuck/tailstock barrier, 154
Circular interpolation, 290
CNC ready signal, 139
Command format, 586
Composite control, 93
Connection among spindle, spindle motor, and position coder, 576
Constant surface speed control, 517
Controlled axes, 2
Controlled axes detach, 12
Conversational programming with graphic function, 811
Coordinate system rotation, 738
Corner control, 436
Cs contour control, 548
Custom macro variables common to two–path control (T series (two–path control)), 703
Cutter compensation, 661
Cutter compensation B, C (M series), 661
Cutting feedrate clamp, 342
Cycle start/feed hold, 246
Cylindrical interpolation, 309

– D –

Decimal point programming/pocket calculator type decimal point programming, 669
Direct input of workpiece zero point offset value measured, 878
Direct operation by PMC/MMC, 942
Display/set/edit, 755
Display data on the diagnosis screen , 582
Display of hardware and software configuration, 766
Display/set, 756
Displaying alarm history, 760
Displaying operating monitor, 781
Displaying operation history, 757
DNC operation, 281
DNC1 interface, 821
DNC2 interface, 829
Dry run, 258

– E –

Edit, 803
Emergency stop, 136
Entering compensation values, 865
Error compensation, 27
Error detect (T series), 439
Exact stop/exact stop mode/tapping mode/cutting mode, 277
External data input, 918
External deceleration, 363
External I/O device control, 830
External key input, 937
External motion function (M series), 715
External operator message logging and display, 802
External program input, 839
External workpiece number search, 927

– F –

F1-digit feed (M series), 349
Feed forward in rapid traverse, 441
Feed per minute, 344
Feed per revolution/manual feed per revolution, 347
Feed stop function, 365
Feedrate clamping by arc radius, 366
Feedrate control, 339

Feedrate control/acceleration and deceleration control, 338
Feedrate inverse time specification (M series), 352
Feedrate override, 356
Floating reference position return, 242
Follow–up, 20
Forbidden area overrapping, 150

– G –

G code system, 671
Graphic display/dynamic graphic display, 774

– H –

Helical interpolation, 304
Help function, 759
High speed cycle cutting, 683
High–precision contour control by risc (M series), 384
High–speed M/S/T/B interface, 459
High–speed skip signal, 854

– I –

In–position check, 436
In–position check independently of feed/rapid traverse, 437
Inch/metric conversion, 678
Increment system, 6
Index table indexing function (M Series), 726
Input of offset value measured A (T series), 865
Input of tool offset value measured B (T series), 867
Input/output of data, 812
Interface between CNC and PMC, 945
Interpolation function, 284
Interruption type custom macro, 700
Involute interpolation (M series), 306

– J –

Jog feed/incremental feed, 195

– L –

Linear acceleration/deceleration after interpolation for cutting feed, 424
Linear acceleration/deceleration of cutting feed before interpolation, 431
Linear interpolation, 287
List of addresses, 946
List of addresses (One–path control), 946
List of addresses (Two–path control), 964
List of signals, 1001
List of signals (In order of functions), 1001
List of signals (In order of addresses), 1032
List of signals (In order of symbols), 1017
Look–ahead acceleration/deceleration before interpolation , 386
Look–ahead control, 376

– M –

M code loop check function, 466
M29 and G84 (G74) are specified in the same block, 604
Machine coordinate system, 47
Machine lock, 255
Macro compiler/executer, 747
Manual absolute ON/OFF, 263
Manual handle feed, 202
Manual handle interruption, 208
Manual intervention and return, 283
Manual operation, 194
Manual reference position return, 215
Measurement, 843
Memory protection key, 805
Mirror image, 17
Mirror image for double turrets, 724
Miscellaneous function/2nd auxiliary function, 443
Mode selection, 171
Multi–language display, 795
Multi–step skip (T series), 858
Multi-spindle control (T series), 563
Multiple M commands in a single block, 456

– N –

Name of axes, 4
No. of registered programs, 804
Normal direction control (M series), 334
Notes on interface with the PMC , 596

– O –

Optional block skip/addition of optional block skip, 266
Outputting the movement state of an axis , 15
Override, 353
Override cancel, 358
Overtravel check, 141
Overtravel signal, 141

– P –

Parameters related to servo, 41

Part program storage length, 803

Password function, 807

Path selection/display of optional path names, 178

Playback, 810

PMC control function, 879

Polar coordinate interpolation, 307

Polygonal turning, 313

Polygonal turning (T series), 312

Polygonal turning with two spindles, 318

Position display neglect, 767

Position switch, 24

Positioning, 285

Preparations for operation, 135

Program command, 668

Program configuration, 675

Program restart, 270

– R –

Rapid traverse override, 353

Rapid traverse rate, 339

Reader/puncher interface, 813

Reference position establishment, 214

Reference position return, 237

Reference position shift, 234

Remote buffer, 820

Remote diagnosis, 796

Reset/rewind, 251

Retrace (M series), 743

Rigid tapping, 574

Rigid tapping specification, 581

Rotary axis roll over, 54

Run hour and parts count display, 769

– S –

Scaling (M series), 734

Scaling of each axis, programmable mirror image (Negative magnification), 735

Scaling up or down along all axes at the same rate of magnification, 734

Second feedrate override, 357

Self–diagnosis, 765

Sequence number comparison and stop, 269

Servo off (Mechanical handle), 22

Servo tuning screen, 761

Setting each axis, 4

Setting the reference position without dogs, 227

Settings related to servo–controlled axes, 41

Settings related with coordinate systems, 47

Signals for the rigid tapping function, 590

Signals related to gear switching, 592

Signals related to S code output, 591

Signals related to second spindle rigid tapping (T series only), 594

Simple synchronous control, 56

Simultaneous input and output operations (M series), 837

Single block, 261

Single direction positioning (M series), 302

Skip function, 851

Small hole peck drilling cycle (M series), 748

Software operator’s panel, 785

Specifying G84 (G74) for rigid tapping by parameters, 608

Specifying the rotation axis, 9

Spindle orientation, 638

Spindle output control by the PMC, 930

Spindle output switching, 640

Spindle positioning (T series), 528

Spindle serial output/spindle analog output, 472

Spindle speed control, 480

Spindle speed control for two–path lathe, 507

Spindle speed fluctuation detection, 522

Spindle speed function (S code output), 471

Spindle synchronous control, 634

Spindle tuning screen, 762

Stamping the machining time, 783

Start lock/ interlock, 167

Status output signal, 180

Stored pitch error compensation, 27

Stored stroke limit 1, 144

Straightness compensation, 37

Superimposed control, 96

Synchronous control, 73, 86

Synchronous control and composite control (T series (Two–path control)), 83

– T –

Tandem control (M series), 64

Testing a program, 255

Thread cutting, 295

Thread cutting cycle retract (T series), 300

Three–dimensional coordinate conversion (M series), 741

Timing charts for rigid tapping specification, 599

Timing to cancel rigid tapping mode, 612

Tool axis direction handle feed function (M system), 210

Tool compensation value tool compensation number tool compensation memory, 646

Tool function, 643

Tool functions, 642

Tool length measurement (M series), 844

Tool life management, 653

Tool nose radius compensation (T series), 665

Tool post interference check (T series, two–path control), 160

Tool retraction and return (M series), 273

Torque limit skip (T series), 862

– V –

Vrdy off alarm ignore signal, 182

– W –

Waiting M code (T series (Two–path control), 464

Waveform diagnosis display, 763

When M29 is specified before G84 (G74), 600

Workpiece coordinate system/addition of workpiece coordinate system pair, 48

Revision Record

FANUC Series 16/18/160/180–MODEL B FOR CONNECTION MANUAL(Function) (B–62443E–1)

Edition	Date	Contents	Edition	Date	Contents
03	Jan., '95	Following functions are added. • Straightness compensation • Synchronous control (T series) • Synchronous control and composite control (T series (2–path control)) • Abnormal load detection • Small hole peck drilling cycle (M series) • External operator message logging and display			
02	Nov., '94	• Addition of Series 18–MODEL B • Addition of Series 160–MODEL B • Addition of Series 180–MODEL B • Correction of errors			
01	Jun., '94				