

Changes for the Better

MITSUBISHI CNC

Programming Manual (Machining Center System)
M800/M80/E80/C80 Series

A grayscale image of the Earth from space, showing continents and clouds. Overlaid on the center of the Earth is the text "MITSUBISHI CNC" in a large, white, stylized font with a slight shadow effect. The text is partially obscured by a semi-transparent, curved band that follows the curvature of the globe.

**MITSUBISHI
CNC**

Introduction

This manual describes how to carry out MITSUBISHI CNC programming.
Supported models are as follows:

Supported models	Abbreviations in this manual
M800W series	M800 series, M800, M8
M800S series	
M80W series	M80 series, M80, M8
M80 series	
C80 series	C80
E80 Series	E80, M800/M80 Series, M8

This manual describes programming, therefore, read this manual thoroughly before using this NC system.
To ensure safe use of this NC system, thoroughly study the "Precautions for Safety" on the following page before using this NC system.

Be sure to always keep this manual on hand so that users can refer to it at any time.

Details described in this manual

The description concerning "Signals" in the main text refers to information transmission between a machine and PLC or between NC and PLC.

The method for controlling the signals (ON/OFF) differs depending on the machine. Refer to the manual issued by the machine tool builder (MTB).

Some parameters can be used by end-users and some parameters are set by the MTB according to the specifications.

End-users may not be able to set or change some of the parameters described as "... can be set with the parameter #XXXX" in the main text. Confirm the specifications for your machine with the manual issued by the MTB.

CAUTION

- ⚠ For items described as "Restrictions" or "Usable State" in this manual, the instruction manual issued by the machine tool builder (MTB) takes precedence over this manual.
- ⚠ Items not described in this manual must be interpreted as "not possible".
- ⚠ This manual is written on the assumption that all the applicable functions are included. Some of them, however, may not be available for your NC system. Refer to the specifications issued by the machine tool builder before use.
- ⚠ Refer to the Instruction Manual issued by the MTB for details regarding each machine tool.
- ⚠ Some screens and functions may differ depending on the NC system (or its version), and some functions may not be available. Please confirm the specifications before use.
- ⚠ Do not connect NC system to the Internet-connected network.
- ⚠ To maintain the safety of the NC system against unauthorized access from external devices via the network, take appropriate measures.

In this manual, the following abbreviations might be used.

L system: Lathe system

M system: Machining center system

MTB: Machine tool builder

Also refer to the manuals on "Manual List" as necessary.

Manual List

Manuals related to M800/M80/E80/C80 Series are listed as follows.

These manuals are written on the assumption that all optional functions are added to the targeted model.

Some functions or screens may not be available depending on the machine or specifications set by MTB. (Confirm the specifications before use.)

The manuals issued by MTB take precedence over these manuals.

Manual	IB No.	Purpose and Contents
M800/M80/E80 Series Instruction Manual	IB-1501274	<ul style="list-style-type: none"> ♦ Operation guide for NC ♦ Explanation for screen operation, etc.
C80 Series Instruction Manual	IB-1501453	<ul style="list-style-type: none"> ♦ Operation guide for NC ♦ Explanation for screen operation, etc.
M800/M80/E80/C80 Series Programming Manual (Lathe System) (1/2)	IB-1501275	<ul style="list-style-type: none"> ♦ G code programming for lathe system ♦ Basic functions, etc.
M800/M80/E80/C80 Series Programming Manual (Lathe System) (2/2)	IB-1501276	<ul style="list-style-type: none"> ♦ G code programming for lathe system ♦ Functions for multi-part system, high-accuracy function, etc.
M800/M80/E80/C80 Series Programming Manual (Machining Center System) (1/2)	IB-1501277	<ul style="list-style-type: none"> ♦ G code programming for machining center system ♦ Basic functions, etc.
M800/M80/E80/C80 Series Programming Manual (Machining Center System) (2/2)	IB-1501278	<ul style="list-style-type: none"> ♦ G code programming for machining center system ♦ Functions for multi-part system, high-accuracy function, etc.
M800/M80/E80/C80 Series Alarm/Parameter Manual	IB-1501279	<ul style="list-style-type: none"> ♦ Alarms ♦ Parameters

Manuals for MTBs (NC)

Manual	IB No.	Purpose and Contents
M800/M80/E80/C80 Series Specifications Manual (Function)	IB-1501505	<ul style="list-style-type: none"> ♦ Model selection ♦ Outline of various functions
M800/M80/E80/C80 Series Specifications Manual (Hardware)	IB-1501506	<ul style="list-style-type: none"> ♦ Model selection ♦ Specifications of hardware unit
M800W/M80W Series Connection and Setup Manual	IB-1501268	<ul style="list-style-type: none"> ♦ Detailed specifications of hardware unit ♦ Installation, connection, wiring, setup (startup/adjustment)
M800S/M80/E80 Series Connection and Setup Manual	IB-1501269	<ul style="list-style-type: none"> ♦ Detailed specifications of hardware unit ♦ Installation, connection, wiring, setup (startup/adjustment)
C80 Series Connection and Setup Manual	IB-1501452	<ul style="list-style-type: none"> ♦ Detailed specifications of hardware unit ♦ Installation, connection, wiring, setup (startup/adjustment)
M800/M80/E80 Series PLC Development Manual	IB-1501270	<ul style="list-style-type: none"> ♦ Electrical design ♦ I/O relation (assignment, setting, connection), field network ♦ Development environment (PLC on-board, peripheral development environment), etc.
M800/M80/E80 Series PLC Programming Manual	IB-1501271	<ul style="list-style-type: none"> ♦ Electrical design ♦ Sequence programming ♦ PLC support functions, etc.
M800/M80/E80/C80 Series PLC Interface Manual	IB-1501272	<ul style="list-style-type: none"> ♦ Electrical design ♦ Interface signals between NC and PLC
M800/M80/E80 Series Maintenance Manual	IB-1501273	<ul style="list-style-type: none"> ♦ Cleaning and replacement for each unit ♦ Other items related to maintenance
C80 Series Maintenance Manual	IB-1501454	<ul style="list-style-type: none"> ♦ Cleaning and replacement for each unit ♦ Other items related to maintenance

Manuals for MTBs (drive section)

Manual	IB No.	Contents
MDS-E/EH Series Specifications Manual	IB-1501226	<ul style="list-style-type: none"> ♦ Specifications for power supply regeneration type
MDS-E/EH Series Instruction Manual	IB-1501229	<ul style="list-style-type: none"> ♦ Instruction for power supply regeneration type
MDS-EJ/EJH Series Specifications Manual	IB-1501232	<ul style="list-style-type: none"> ♦ Specifications for regenerative resistor type
MDS-EJ/EJH Series Instruction Manual	IB-1501235	<ul style="list-style-type: none"> ♦ Instruction for regenerative resistor type
MDS-EM/EMH Series Specifications Manual	IB-1501238	<ul style="list-style-type: none"> ♦ Specifications for multi-hybrid, power supply regeneration type
MDS-EM/EMH Series Instruction Manual	IB-1501241	<ul style="list-style-type: none"> ♦ Instruction for multi-hybrid, power supply regeneration type
DATA BOOK	IB-1501252	<ul style="list-style-type: none"> ♦ Specifications of servo drive unit, spindle drive unit, motor, etc.

Manuals for MTBs (Others)

Manual	No.	Purpose and Contents
GOT2000 Series User's Manual (Hardware)	SH-081194	♦ Outline of hardware such as part names, external dimensions, installation, wiring, maintenance, etc. of GOTs
GOT2000 Series User's Manual (Utility)	SH-081195	♦ Outline of utilities such as screen display setting, operation method, etc. of GOTs
GOT2000 Series User's Manual (Monitor)	SH-081196	♦ Outline of each monitor function of GOTs
GOT2000 Series Connection Manual (Mitsubishi Electric Products)	SH-081197	♦ Outline of connection types and connection method between GOT and Mitsubishi Electric connection devices
GT Designer3 (GOT2000) Screen Design Manual	SH-081220	♦ Outline of screen design method using screen creation software GT Designer3

■ For M800/M80/E80 Series

Manual	No.	Purpose and Contents
GOT2000/GOT1000 Series CC-Link Communication Unit User's Manual	IB-0800351	♦ Explanation for handling CC-Link communication unit (for GOT2000 series/GOT1000 series)
GX Developer Version 8 Operating Manual (Startup)	SH-080372E	♦ Explanation for system configuration, installation, etc. of PLC development tool GX Developer
GX Developer Version 8 Operating Manual	SH-080373E	♦ Explanation for operations using PLC development tool GX Developer
GX Converter Version 1 Operating Manual	IB-0800004E	♦ Explanation for operations using data conversion tool GX Converter
MELSEC-Q CC-Link System Master/Local Module User's Manual	SH-080394E	♦ Explanation for system configuration, installation, wiring, etc. of master/local modules for CC-Link system
GOT2000 Series Connection Manual (Non-Mitsubishi Electric Products 1)	SH-081198ENG	♦ Explanation for connection types and connection method between GOT and other company's devices
GOT2000 Series Connection Manual (Non-Mitsubishi Electric Products 2)	SH-081199ENG	
GOT2000 Series Connection Manual (Microcomputers, MODBUS/Fieldbus Products, Peripherals)	SH-081200ENG	♦ Explanation for connection types and connection method between GOT and microcomputers, MODBUS/fieldbus products, peripherals
GT SoftGOT2000 Version1 Operating Manual	SH-081201ENG	♦ Explanation for system configuration, screen configuration and operation method of monitoring software GT SoftGOT2000

■ For C80 Series


Manual	No.	Purpose and Contents
MELSEC iQ-R Module Configuration Manual	SH-081262	♦ Outline of system configuration, specifications, installation, wiring, maintenance, etc.
MELSEC iQ-R CPU Module User's Manual (Startup)	SH-081263	♦ Outline of specifications, procedures before operation, troubleshooting, etc. for CPU module
MELSEC iQ-R CPU Module User's Manual (Application)	SH-081264	♦ Outline of memory, functions, devices, parameters, etc. for CPU module
QCPU User's Manual (Hardware Design, Maintenance and Inspection)	SH-080483	♦ Outline of specifications, necessary knowledge to configure the system and maintenance-related descriptions for Q series CPU module, etc.
GX Works3 Operating Manual	SH-081215	♦ Outline of functions, programming, etc.


Reference Manual for MTBs


Manual	No.	Purpose and Contents
M800/M80 Series Smart safety observation Specification manual	BNP-C3072-022	♦ Explanation for smart safety observation function
C80 Series Smart safety observation Specification manual	BNP-C3077-022	
M800/M80 Series CC-Link (Master/Local) Specification manual	BNP-C3072-089	♦ Explanation for CC-Link
M800/M80 Series PROFIBUS-DP Specification manual	BNP-C3072-118	♦ Explanation for PROFIBUS-DP communication function
M800/M80 Series Interactive cycle insertion (Customization) Specification manual	BNP-C3072-121-0003	♦ Explanation for interactive cycle insertion
M800/M80 Series EtherNet/IP Specifications manual	BNP-C3072-263	♦ Explanation for EtherNet/IP

Precautions for Safety



Always read the specifications issued by the machine tool builder, this manual, related manuals and attached documents before installation, operation, programming, maintenance or inspection to ensure correct use.
 Understand this numerical controller, safety items and cautions before using the unit.
 This manual ranks the safety precautions into "DANGER", "WARNING" and "CAUTION".



 **DANGER**
 When the user may be subject to imminent fatalities or major injuries if handling is mistaken.

 **WARNING**
 When the user may be subject to fatalities or major injuries if handling is mistaken.

 **CAUTION**
 When the user may be subject to injuries or when physical damage may occur if handling is mistaken.

Note that even items ranked as " CAUTION", may lead to major results depending on the situation. In any case, important information that must always be observed is described.
 The following signs indicate prohibition and compulsory.

 This sign indicates prohibited behavior (must not do).
 For example,  indicates "Keep fire away".

 This sign indicates a thing that is compulsory (must do).
 For example,  indicates "it must be grounded".

The meaning of each pictorial sign is as follows.

 CAUTION	 CAUTION rotated object	 CAUTION HOT	 Danger Electric shock risk	 Danger explosive
 Prohibited	 Disassembly is prohibited	 KEEP FIRE AWAY	 General instruction	 Earth ground

For Safe Use

Mitsubishi CNC is designed and manufactured solely for applications to machine tools to be used for industrial purposes.
 Do not use this product in any applications other than those specified above, especially those which are substantially influential on the public interest or which are expected to have significant influence on human lives or properties.

 **DANGER**

Not applicable in this manual.

⚠ WARNING

1. Items related to operation

- ⚠ If the operation start position is set in a block which is in the middle of the program and the program is started, the program before the set block is not executed. Please confirm that G and F modal and coordinate values are appropriate. If there are coordinate system shift commands or M, S, T and B commands before the block set as the start position, carry out the required commands using the MDI, etc. If the program is run from the set block without carrying out these operations, there is a danger of interference with the machine or of machine operation at an unexpected speed, which may result in breakage of tools or machine tool or may cause damage to the operators.
- ⚠ Under the constant surface speed control (during G96 modal), if the axis targeted for the constant surface speed control (normally X axis for a lathe) moves toward the spindle center, the spindle rotation speed will increase and may exceed the allowable speed of the workpiece or chuck, etc. In this case, the workpiece, etc. may jump out during machining, which may result in breakage of tools or machine tool or may cause damage to the operators.

⚠ CAUTION

1. Items related to product and manual

- ⚠ For items described as "Restrictions" or "Usable State" in this manual, the instruction manual issued by the machine tool builder takes precedence over this manual.
- ⚠ Items not described in this manual must be interpreted as "not possible".
- ⚠ This manual is written on the assumption that all the applicable functions are included. Some of them, however, may not be available for your NC system.
Refer to the specifications issued by the machine tool builder before use.
- ⚠ Refer to the Instruction Manual issued by each machine tool builder for details on each machine tool.
- ⚠ Some screens and functions may differ depending on the NC system (or its version), and some functions may not be possible. Please confirm the specifications before use.
- ⚠ Do not connect NC system to the Internet-connected network.
- ⚠ To maintain the safety of the NC system against unauthorized access from external devices via the network, take appropriate measures.

2. Items related to operation

- ⚠ Before starting actual machining, always carry out graphic check, dry run operation and single block operation to check the machining program, tool offset amount, workpiece compensation amount and etc.
- ⚠ If the workpiece coordinate system offset amount is changed during single block stop, the new setting will be valid from the next block.
- ⚠ Turn the mirror image ON and OFF at the mirror image center.
- ⚠ If the tool offset amount is changed during automatic operation (including during single block stop), it will be validated from the next block or blocks onwards.
- ⚠ Do not make the synchronized spindle rotation command OFF with one workpiece chucked by the reference spindle and synchronized spindle during the spindle synchronization.
Failure to observe this may cause the synchronized spindle stop, and hazardous situation.

3. Items related to programming

- ⚠ The commands with "no value after G" will be handled as "G00".
- ⚠ ";", "EOB" and "%" "EOR" are expressions used for explanation. The actual codes are: For ISO: "CR, LF", or "LF" and "%". Programs created on the Edit screen are stored in the NC memory in a "CR, LF" format, but programs created with external devices such as the FLD or RS-232C may be stored in an "LF" format.
The actual codes for EIA are: "EOB (End of Block)" and "EOR (End of Record)".
- ⚠ When creating the machining program, select the appropriate machining conditions, and make sure that the performance, capacity and limits of the machine and NC are not exceeded. The examples do not consider the machining conditions.
- ⊘ Do not change fixed cycle programs without the prior approval of the machine tool builder.
- ⚠ When programming the multi-part system, take special care to the movements of the programs for other part systems.

Disposal



(Note) This symbol mark is for EU countries only.
This symbol mark is according to the directive 2006/66/EC Article 20 Information for end-users and Annex II.

Your MITSUBISHI ELECTRIC product is designed and manufactured with high quality materials and components which can be recycled and/or reused.

This symbol means that batteries and accumulators, at their end-of-life, should be disposed of separately from your household waste.

If a chemical symbol is printed beneath the symbol shown above, this chemical symbol means that the battery or accumulator contains a heavy metal at a certain concentration. This will be indicated as follows:

Hg: mercury (0,0005%), Cd: cadmium (0,002%), Pb: lead (0,004%)

In the European Union there are separate collection systems for used batteries and accumulators. Please, dispose of batteries and accumulators correctly at your local community waste collection/recycling centre.

Please, help us to conserve the environment we live in!

Trademarks

MELDAS, MELSEC, EZSocket, EZMotion, iQ Platform, MELSEC iQ-R, MELSOFT, GOT, CC-Link, CC-Link/LT, CC-Link IE, CC-Link IE/field, EcoMonitorLight and SLMP are either trademarks or registered trademarks of Mitsubishi Electric Corporation in Japan and/or other countries.

Ethernet is a registered trademark of Xerox Corporation in the United States and/or other countries.

Microsoft®, Windows®, SQL Server® and Access® are either trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.

SD logo and SDHC logo are either registered trademarks or trademarks of LLC.

UNIX is a registered trademark of The Open Group in the United States and/or other countries.

Intel® and Pentium® are either trademarks or registered trademarks of Intel Corporation in the United States and/or other countries.

MODBUS® is either a trademark or a registered trademark of Schneider Electric USA, Inc. or the affiliated companies in Japan and/or other countries.

EtherNet/IP is a trademark of Open DeviceNet Vendor Association, Inc.

PROFIBUS-DP and PROFINET are either trademarks of Profibus International.

Oracle® is a registered trademark of Oracle Corporation, the subsidiaries, or the affiliated companies in the United States and /or other countries.

VNC is a registered trademark of RealVNC Ltd. in the United States and other countries.

Other company and product names that appear in this manual are trademarks or registered trademarks of the respective companies.

本製品の取扱いについて

(日本語/Japanese)

本製品は工業用(クラス A)電磁環境適合機器です。販売者あるいは使用者はこの点に注意し、住商業環境以外での使用をお願いいたします。

Handling of our product

(English)

This is a class A product. In a domestic environment this product may cause radio interference in which case the user may be required to take adequate measures.

본 제품의 취급에 대해서

(한국어/Korean)

이 기기는 업무용(A급) 전자파적합기기로서 판매자 또는 사용자는 이 점을 주의하시기 바라며 가정외의 지역에서 사용하는 것을 목적으로 합니다.

Contents

Chapter 1 - 14 : Refer to Programming Manual (Machining Center System) (1/2)

Chapter 15 and later : Refer to Programming Manual (Machining Center System) (2/2)

1 Control Axes	1
1.1 Coordinate Words and Control Axes	2
1.2 Coordinate Systems and Coordinate Zero Point Symbols	3
2 Minimum Command Unit	5
2.1 Input Setting Unit	6
2.2 Input Command Increment Tenfold	7
2.3 Indexing Increment	8
3 Program Formats	9
3.1 Program Format	10
3.2 File Format	14
3.3 Optional Block Skip	16
3.3.1 Optional Block Skip; /	16
3.3.2 Optional Block Skip Addition ; /n	18
3.4 G Code	20
3.4.1 Modal, Unmodal	20
3.4.2 G Code Lists	20
3.5 Precautions before Starting Machining	25
4 Pre-read Buffer	27
4.1 Pre-read Buffer	28
5 Position Commands	29
5.1 Position Command Methods ; G90,G91	30
5.2 Diameter Designation and Radius Designation	32
5.2.1 Diameter/Radius Designation Switch; G10.9	32
5.3 Inch/Metric Conversion; G20, G21	35
5.4 Decimal Point Input	37
6 Interpolation Functions	43
6.1 Positioning (Rapid Traverse); G00	44
6.2 Linear Interpolation; G01	47
6.3 Circular Interpolation; G02, G03	49
6.4 R Specification Circular Interpolation; G02, G03	55
6.5 Plane Selection; G17, G18, G19	58
6.6 Thread Cutting	60
6.6.1 Constant Lead Thread Cutting; G33	60
6.6.2 Inch Thread Cutting; G33	64
6.7 Helical Interpolation ; G17,G18,G19 and G02,G03	66
6.8 Unidirectional Positioning	72
6.8.1 Unidirectional Positioning; G60	72
6.8.2 Axis-based Unidirectional Positioning	74
6.9 Cylindrical Interpolation; G07.1	75
6.10 Circular Cutting; G12,G13	82
6.11 Polar Coordinate Interpolation; G12.1, G13.1/G112, G113	84
6.12 Exponential Interpolation; G02.3, G03.3	91
6.13 Polar Coordinate Command; G16	98
6.14 Spiral/Conical Interpolation; G02.1/G03.1 (Type 1), G02/G03 (Type 2)	105
6.15 3-dimensional Circular Interpolation; G02.4, G03.4	109
6.16 NURBS Interpolation; G06.2	114
6.17 Hypothetical Axis Interpolation; G07	120
6.18 Involute Interpolation; G02.2/G03.2	122
7 Feed Functions	131
7.1 Rapid Traverse Rate	132
7.1.1 Rapid Traverse Rate	132
7.1.2 G00 Feedrate Command (,F Command)	133

7.2 Cutting Feedrate	137
7.3 F1-digit Feed.....	138
7.4 Feed Per Minute/Feed Per Revolution (Asynchronous Feed/Synchronous Feed); G94,G95	140
7.5 Inverse Time Feed; G93	143
7.6 Feedrate Designation and Effects on Control Axes.....	148
7.7 Rapid Traverse Constant Inclination Acceleration/Deceleration.....	152
7.8 Rapid Traverse Constant Inclination Multi-step Acceleration/Deceleration	157
7.9 Cutting Feed Constant Inclination Acceleration/Deceleration.....	165
7.10 Exact Stop Check; G09	171
7.11 Exact Stop Check Mode; G61	175
7.12 Deceleration Check	176
7.12.1 Deceleration Check.....	176
7.12.2 Deceleration Check When Movement in the Opposite Direction Is Reversed	184
7.13 Rapid Traverse Block Overlap; G0.5 P1.....	186
7.13.1 Rapid Traverse Block Overlap for G00; G0.5	188
7.13.2 Rapid Traverse Block Overlap for G28	196
7.14 Automatic Corner Override.....	198
7.14.1 Automatic Corner Override ; G62.....	205
7.14.2 Inner Arc Override.....	206
7.15 Tapping Mode; G63	207
7.16 Cutting Mode; G64.....	208
8 Dwell.....	209
8.1 Dwell (Time-based Designation); G04	210
9 Miscellaneous Functions	213
9.1 Miscellaneous Functions (M8-digits)	214
9.2 Second Miscellaneous Functions (A8-digits, B8-digits or C8-digits)	216
9.3 Index Table Indexing	217
10 Spindle Functions	223
10.1 Spindle Functions	224
10.2 Constant Surface Speed Control; G96, G97	225
10.3 Spindle Clamp Speed Setting; G92	232
10.4 Spindle Position Control (Spindle/C Axis Control)	234
10.5 Spindle Speed Fluctuation Detection; G162/G163	242
11 Tool Functions (T command).....	249
11.1 Tool Functions (T8-digit BCD)	250
12 Tool Compensation Functions	251
12.1 Tool Compensation.....	252
12.1.1 Tool Compensation	252
12.1.2 Number of Tool Offset Sets Allocation to Part Systems.....	256
12.2 Tool Length Compensation/Cancel; G43, G44 / G49	258
12.3 Tool Radius Compensation; G38,G39/G40/G41,G42	266
12.3.1 Tool Radius Compensation Operation	267
12.3.2 Other Commands and Operations during Tool Radius Compensation.....	276
12.3.3 G41/G42 Commands and I, J, K Designation	286
12.3.4 Interrupts during Tool Radius Compensation.....	292
12.3.5 General Precautions for Tool Radius Compensation.....	294
12.3.6 Changing of Compensation No. during Compensation Mode.....	295
12.3.7 Start of Tool Radius Compensation and Z Axis Cut in Operation	298
12.3.8 Interference Check.....	300
12.3.9 Diameter Designation of Compensation Amount.....	310
12.3.10 Workpiece Coordinate Changing during Radius Compensation.....	312
12.4 Tool Nose Radius Compensation (for Machining Center System)	314
12.5 3-dimensional Tool Radius Compensation; G40/G41, G42.....	317
12.6 Tool Position Offset; G45 to G48	329
13 Fixed Cycle	337
13.1 Fixed Cycles	338
13.1.1 Drilling, Spot Drilling; G81	342
13.1.2 Drilling, Counter Boring; G82	343
13.1.3 Deep Hole Drilling Cycle; G83	344

13.1.3.1 Deep Hole Drilling Cycle	344
13.1.3.2 Small Diameter Deep Hole Drilling Cycle	346
13.1.4 Tapping Cycle; G84	349
13.1.5 Boring; G85	363
13.1.6 Boring; G86	364
13.1.7 Back Boring; G87	365
13.1.8 Boring; G88	368
13.1.9 Boring; G89	369
13.1.10 Stepping Cycle; G73	370
13.1.11 Reverse Tapping Cycle; G74	372
13.1.12 Circular Cutting; G75	374
13.1.13 Fine Boring; G76	376
13.1.14 Thread Milling Cycle; G187	377
13.1.15 Precautions for Using a Fixed Cycle	381
13.1.16 Initial Point and R Point Level Return; G98, G99	383
13.1.17 Setting of Workpiece Coordinates in Fixed Cycle Mode	384
13.1.18 Drilling Cycle High-Speed Retract	385
13.1.19 Acceleration/Deceleration Mode Change in Hole Drilling Cycle	389
13.2 Special Fixed Cycle	391
13.2.1 Bolt Hole Cycle; G34	392
13.2.2 Line at Angle; G35	393
13.2.3 Arc; G36	394
13.2.4 Grid; G37.1	395
13.3 Fixed Cycle for Turning Machining	397
13.3.1 Longitudinal Cutting Cycle; G174	398
13.3.2 Thread Cutting Cycle; G175	401
13.3.3 Face Cutting Cycle; G176	405
14 Macro Functions	409
14.1 Subprogram Control; M98, M99, M198	410
14.1.1 Subprogram Call; M98, M99	410
14.1.2 Subprogram Call; M198	416
14.1.3 Figure Rotation; M98 I_J_K_	417
14.2 Variable Commands	420
14.3 User Macro	425
14.4 Macro Call Instructions	426
14.4.1 Simple Macro Calls; G65	426
14.4.2 Modal Call A (Movement Command Call) ; G66	430
14.4.3 Modal Call B (for Each Block); G66.1	432
14.4.4 G Code Macro Call	434
14.4.5 Miscellaneous Command Macro Call (for M, S, T, B Code Macro Call)	435
14.4.6 Detailed Description for Macro Call Instruction	437
14.4.7 ASCII Code Macro	439
14.5 Variables Used in User Macros	443
14.5.1 Common Variables	445
14.5.2 Local Variables (#1 to #33)	446
14.5.3 System Variables	449
14.6 User Macro Commands	450
14.6.1 Operation Commands	450
14.6.2 Control Commands	455
14.6.3 External Output Commands; POPEN, PCLOS, DPRNT	461
14.6.4 Precautions	465
14.6.5 Actual Examples of Using User Macros	467
14.7 Macro Interruption; M96, M97	471
15 Program Support Functions	481
15.1 Corner Chamfering I/Corner Rounding I	482
15.1.1 Corner Chamfering I ; G01 X_Y_ ,C	482
15.1.2 Corner Rounding I ; G01 X_Y_ ,R_	484
15.1.3 Corner Chamfering Expansion/Corner Rounding Expansion	486
15.1.4 Interrupt during Corner Chamfering/Interrupt during Corner Rounding	488
15.2 Corner Chamfering II/Corner Rounding II	489
15.2.1 Corner Chamfering II ; G01/G02/G03 X_Y_ ,C_	489
15.2.2 Corner Rounding II ; G01/G02/G03 X_Y_ ,R_	491
15.2.3 Corner Chamfering Expansion/Corner Rounding Expansion	492
15.2.4 Interrupt during Corner Chamfering/Interrupt during Corner Rounding	492

15.3 Linear Angle Command; G01 X/Y/A_/A_.....	493
15.4 Geometric; G01 A_.....	494
15.5 Geometric IB.....	496
15.5.1 Geometric IB (Automatic Calculation of Two-arc Contact) ; G02/G03 P_Q_/R_.....	497
15.5.2 Geometric IB (Automatic Calculation of Linear - Arc Intersection) ; G01 A_ , G02/G03 P_Q_H_.....	499
15.5.3 Geometric IB (Automatic Calculation of Linear - Arc Intersection) ; G01 A_ , G02/G03 R_H_.....	502
15.6 Mirror Image by G code ; G50.1,G51.1.....	504
15.7 Normal Line Control; G40.1/G41.1/G42.1 (G150/G151/G152).....	508
15.8 Manual Arbitrary Reverse Run Prohibition ; G127.....	528
15.9 Data Input by Program.....	534
15.9.1 Parameter Input by Program; G10 L70/L100, G11.....	534
15.9.2 Compensation Data Input by Program ; G10 L2/L10/L11/L12/L13/L20, G11.....	537
15.9.3 Compensation Data Input by Program (Turning Tool) ; G10 L12/L13, G11.....	543
15.9.4 Tool Shape Input by Program; G10 L100, G11.....	545
15.9.5 R-Navi Data Input by Program; G10 L110/L111, G11, G68.2, G69.....	548
15.10 Inputting the Tool Life Management Data; G10,G11.....	552
15.10.1 Inputting the Tool Life Management Data by G10 L3 Command; G10 L3,G11.....	552
15.10.2 Inputting the Tool Life Management Data by G10 L30 Command; G10 L30,G11.....	555
15.10.3 Precautions for Inputting the Tool Life Management Data.....	558
15.10.4 Allocation of the Number of Tool Life Management Sets to Part Systems.....	559
15.11 Interactive Cycle Insertion; G180.....	561
15.11.1 Interactive Cycle Insertion.....	561
15.11.2 Interactive Macro.....	564
15.12 Axis Name Extension.....	565
15.13 Machining Interruption [C80]; G26.....	572
16 Multi-part System Control	587
16.1 Timing Synchronization Operation.....	588
16.1.1 Timing Synchronization Operation (! code) !n (!m ...) L.....	588
16.1.2 Timing Synchronization Operation with Start Point Designated (Type 1) ; G115.....	591
16.1.3 Timing Synchronization Operation with Start Point Designated (Type 2) ; G116.....	594
16.1.4 Timing Synchronization Operation Function Using M codes ; M***.....	597
16.1.5 Timing Synchronization When Timing Synchronization Ignore Is Set.....	601
16.2 Mixed Control.....	604
16.2.1 Arbitrary Axis Exchange ; G140, G141, G142.....	604
16.3 Sub Part System Control.....	607
16.3.1 Sub Part System Control I; G122.....	607
17 High-speed High-accuracy Control	623
17.1 High-speed Machining Mode.....	624
17.1.1 High-speed Machining Mode I, II ; G05 P1, G05 P2.....	624
17.2 High-accuracy Control.....	633
17.2.1 High-accuracy Control ; G61.1, G08.....	633
17.2.2 SSS Control.....	653
17.2.3 Tolerance Control.....	657
17.2.4 Variable-acceleration Pre-interpolation Acceleration/Deceleration.....	661
17.2.5 Initial High-accuracy Control.....	664
17.2.6 Multi-part System Simultaneous High-accuracy.....	665
17.3 High-speed High-accuracy Control.....	667
17.3.1 High-speed High-accuracy Control I, II, III ; G05.1 Q1/Q0, G05 P10000/P0, G05 P20000/P0.....	667
17.3.2 Fairing.....	683
17.3.3 Smooth Fairing.....	684
17.3.4 Acceleration Clamp Speed.....	693
17.3.5 High-speed Mode Corner Deceleration.....	694
17.3.6 Precautions on High-speed High-accuracy Control.....	695
17.4 Spline Interpolation ; G05.1 Q2/Q0.....	698
17.5 Spline Interpolation 2; G61.4.....	706
17.6 High-accuracy Spline Interpolation ; G61.2.....	715
17.7 Machining Condition Selection I ; G120.1,G121.....	717
18 Advanced Multi-Spindle Control Function	721
18.1 Spindle Synchronization.....	722
18.1.1 Spindle Synchronization I; G114.1.....	723
18.1.2 Spindle Position Control (Spindle/C Axis Control) under Spindle Synchronization Control.....	734
18.2 Multiple Spindle Synchronization Set Control [C80].....	739

19 Advanced Machining Control	743
19.1 Tool Position Compensation; G43.7/G49	744
19.2 Tool Length Compensation Along the Tool Axis; G43.1/G49	751
19.3 Tool Center Point Control; G43.4, G43.5/G49	758
19.3.1 Circular Command in Tool Center Point Control (G43.4/G43.5)	785
19.4 Inclined Surface Machining; G68.2, G68.3/G69	789
19.4.1 How to Define Feature Coordinate System Using Euler Angles	790
19.4.2 How to Define Feature Coordinate System Using Roll-Pitch-Yaw Angles	792
19.4.3 How to Define Feature Coordinate System Using Three Points in a Plane	794
19.4.4 How to Define Feature Coordinate System Using Two Vectors	796
19.4.5 How to Define Feature Coordinate System Using Projection Angles	798
19.4.6 Define by Selecting the Registered Machining Surface	800
19.4.7 How to Define Feature Coordinate System Using Tool Axis Direction	801
19.4.8 Tool Axis Direction Control; G53.1/G53.6	803
19.4.9 Details of Inclined Surface Machining Operation	811
19.4.10 Rotary Axis Basic Position Selection	816
19.4.11 Relationship between Inclined Surface Machining and Other Functions	822
19.4.12 Precautions for Inclined Surface Machining	826
19.5 3-dimensional Tool Radius Compensation (Tool's Vertical-direction Compensation) ; G40/G41.2,G42.2	830
19.6 Workpiece Installation Error Compensation; G54.4	841
20 Coordinate System Setting Functions	855
20.1 Coordinate Words and Control Axes	856
20.2 Types of Coordinate Systems	857
20.2.1 Basic Machine, Workpiece and Local Coordinate Systems	857
20.2.2 Machine Zero Point and 2nd, 3rd, 4th Reference Position (Zero Point)	858
20.2.3 Automatic Coordinate System Setting	859
20.2.4 Coordinate System for Rotary Axis	860
20.3 Basic Machine Coordinate System Selection; G53	863
20.4 Coordinate System Setting; G92	866
20.5 Local Coordinate System Setting; G52	868
20.6 Workpiece Coordinate System Setting and Offset; G54 to G59 (G54.1)	872
20.7 Workpiece Coordinate System Preset; G92.1	882
20.8 3-dimensional Coordinate Conversion; G68/G69	887
20.9 Coordinate Rotation by Program; G68/G69	905
20.10 Coordinate Rotation Input by Parameter; G10 I_ J_ /K_	912
20.11 Scaling; G50/G51	929
20.12 Reference Position (Zero Point) Return; G28, G29	933
20.13 2nd, 3rd, and 4th Reference Position (Zero Point) Return ; G30	937
20.14 Tool Change Position Return; G30.1 - G30.6	940
20.15 Reference Position Check; G27	943
21 Protection Function	945
21.1 Stroke Check before Travel; G22/G23	946
21.1.1 Stroke Check before Travel in Stored Stroke Limit Area	948
21.2 Enable Interfering Object Selection Data; G186	950
22 Measurement Support Functions	953
22.1 Automatic Tool Length Measurement; G37	954
22.2 Skip Function; G31	958
22.3 Multi-step Skip Function 1; G31.n, G04	964
22.4 Multi-step Skip Function 2; G31 P	966
22.5 Speed Change Skip; G31 Fn	968
22.6 Torque Limitation Skip; G160	972
22.7 Programmable Current Limitation; G10 L14	976
23 System Variables	977
23.1 System Variables List	978
23.2 System Variables (G Command Modal)	980
23.3 System Variables (Non-G Command Modal)	981
23.4 System Variables (Modal Information at Macro Interruption)	982
23.5 System Variables (Tool Information)	984
23.6 System Variables (Tool Compensation)	992
23.7 System Variables (Tool Life Management)	993
23.8 System Variables (Workpiece Coordinate Offset)	998

23.9 System Variables (Extended Workpiece Coordinate Offset)	999
23.10 System Variables (External Workpiece Coordinate Offset)	1000
23.11 System Variables (Position Information).....	1001
23.12 System Variables (Alarm).....	1005
23.13 System Variables (Message Display and Stop).....	1006
23.14 System Variables (Cumulative Time)	1006
23.15 System Variables (Time Read Variables).....	1007
23.16 System Variables (Machining Information).....	1009
23.17 System Variables (Reverse Run Information)	1010
23.18 System Variables (Number of Workpiece Machining Times)	1010
23.19 System Variables (Mirror Image).....	1010
23.20 System Variables (Coordinate Rotation Parameter).....	1011
23.21 System Variables (Rotary Axis Configuration Parameter).....	1012
23.22 System Variables (Normal Line Control Parameter).....	1013
23.23 System Variables (Parameter Reading)	1014
23.24 System Variables (Workpiece Installation Error Compensation Amount).....	1018
23.25 System Variables (Macro Interface Input (PLC -> NC)).....	1019
23.26 System Variables (Macro Interface Output (NC -> PLC)).....	1025
23.27 System Variables (R Device Access Variables)	1031
23.28 System Variables (PLC Data Reading)	1037
23.29 System Variables (Interfering Object Selection)	1041
23.30 System Variables (ZR Device Access Variables) [C80]	1044
23.31 System Variables (NC Data Reading/Writing with API Section and Sub-section Nos. Input/Output by Program)....	1046
24 Appx.1: Fixed Cycles	1049
25 Appx.2: Command Value Range Lists	1053



Control Axes

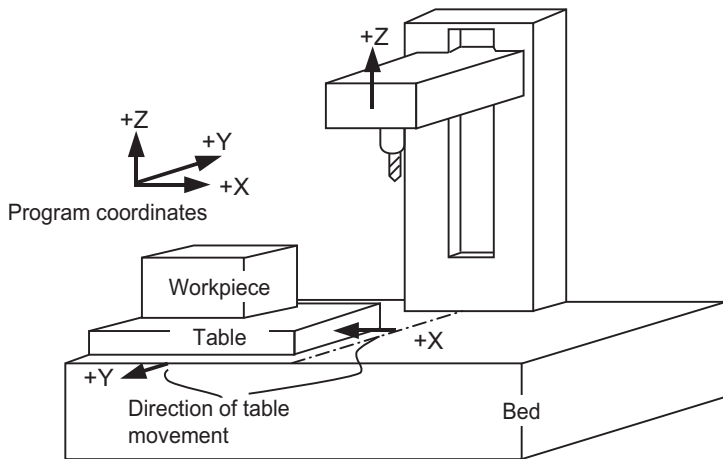
1.1 Coordinate Words and Control Axes



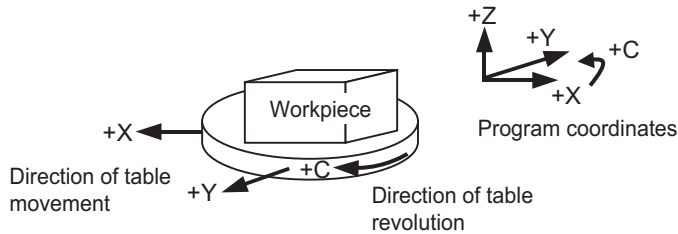
Function and purpose

The number of control axes is set to "3" in the standard specifications; however, up to eight axes can be controlled if an additional axis is added. To specify each machining direction, use alphabetical coordinate words that are pre-defined appropriately.

X-Y table



X-Y and rotating table



1.2 Coordinate Systems and Coordinate Zero Point Symbols



Reference position:
A specific position to establish coordinate systems and change tools



Basic machine coordinate zero point:
A position specific to machine



Workpiece coordinate zero points (G54 to G59)
A coordinate zero point used for workpiece machining

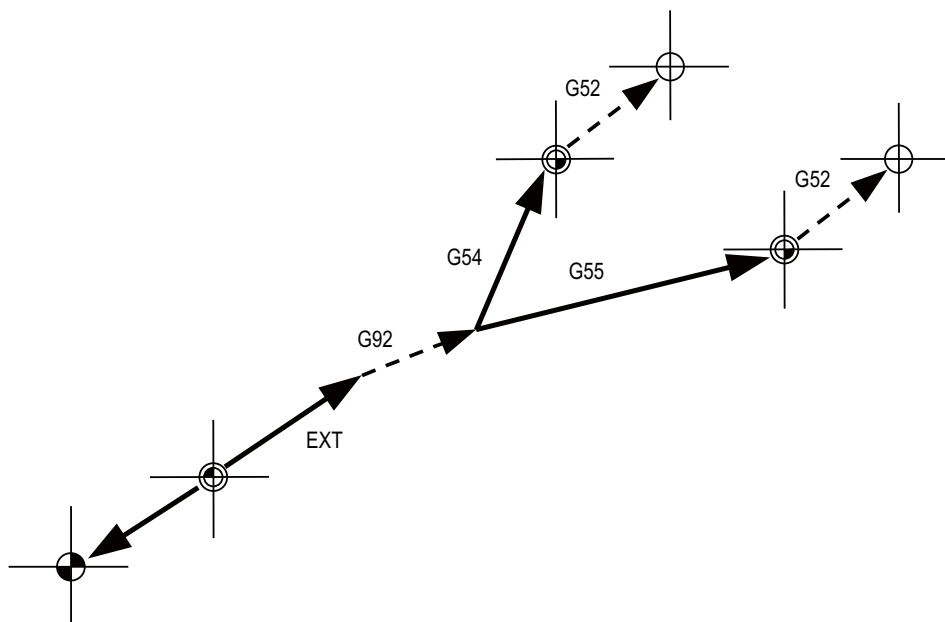
The basic machine coordinate system is the coordinate system that expresses the position (tool change position, stroke end position, etc.) that is specific to the machine.







Workpiece coordinate systems are used for workpiece machining.

Upon completion of the dog-type reference position return, the parameters are referred and the basic machine coordinate system and workpiece coordinate systems (G54 to G59) are automatically set.

The offset of the basic machine coordinate zero point and reference position is set by a parameter. (Normally, set by MTB)

Workpiece coordinate systems can be set with coordinate systems setting functions, workpiece coordinate offset measurement (additional specification), and etc.



-  Reference position
-  Basic machine coordinate zero point
-  Workpiece coordinate zero points
-  Local coordinate zero point
-  Offset set by a parameter
-  Offset set by a program ("0" is set when turning the power ON)
- G52 Local coordinate system offset (*1)
- G54 Workpiece coordinate (G54) system offset (*1)
- G55 Workpiece coordinate (G55) system offset
- G92 G92 Coordinate system shift
- EXT External workpiece coordinate offset

(*1) G52 offset is independently possessed by G 54 to G59 respectively.

The local coordinate systems (G52) are valid on the coordinate systems designated by workpiece coordinate systems 1 to 6.

Using the G92 command, the basic machine coordinate system can be shifted and made into a hypothetical machine coordinate system. At the same time, workpiece coordinate systems 1 to 6 are also shifted.

Minimum Command Unit

2.1 Input Setting Unit



Function and purpose

The input setting units are the units of setting data including tool compensation amounts and workpiece coordinates compensation.

The program command units are the units of movement amounts in programs.

These are expressed with mm, inch or degree (°).



Detailed description

Program command units for each axis and input setting units, common for all axes, are determined by the setting of parameters as follows. (This depends on the MTB specifications.)

	Parameter		Linear axis		Rotary axis (°)
			Metric	Inch	
Input setting unit	#1003 iunit	= B	0.001	0.0001	0.001
		= C	0.0001	0.00001	0.0001
		= D	0.00001	0.000001	0.00001
		= E	0.000001	0.0000001	0.000001
Program command unit	#1015 cunit	= 0	Follow #1003 iunit		
		= 1	0.0001	0.00001	0.0001
		= 10	0.001	0.0001	0.001
		= 100	0.01	0.001	0.01
		= 1000	0.1	0.01	0.1
		= 10000	1.0	0.1	1.0



Precautions

(1) Inch/metric changeover can be handled by either a parameter screen (#1041 I_inch: valid only when the power is turned ON) or G commands (G20 or G21).

However, the changeover by a G command applies only to the program command units, and not to the input setting units. Consequently, the tool offset amounts and other compensation amounts as well as the variable data should be preset in order to correspond to input setting units.

(2) The millimeter and inch systems cannot be used together.

(3) When performing a circular interpolation between the axes whose program command units are different, the center command (I, J, K) and the radius command (R) are designated by the input setting units. (Use a decimal point to avoid confusion.)

2.2 Input Command Increment Tenfold



Function and purpose

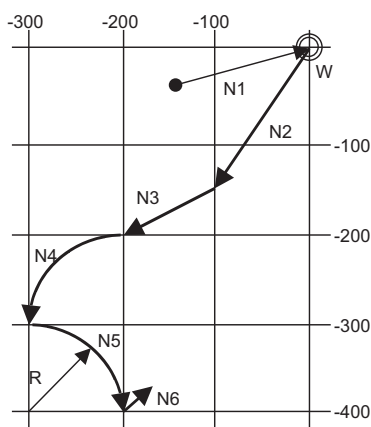
The program's command increment can be multiplied by an arbitrary scale with the parameter designation. This function is valid when a decimal point is not used for the command increment. The scale is set with the parameter "#8044 UNIT*10".



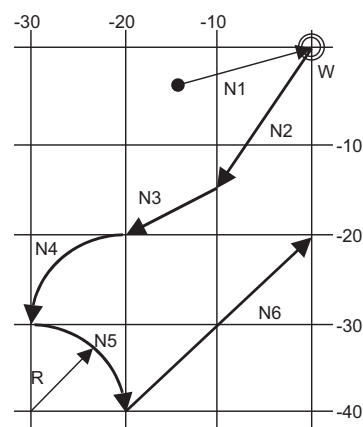
Detailed description

- (1) When running a machining program already created with a 10 μ m input command increment with a CNC unit for which the command increment is set to 1 μ m and this function's parameter value is set to "10", this function enables the same machining as the original program.
- (2) When running a machining program already created with a 1 μ m input command increment with a CNC unit for which the command increment is set to 0.1 μ m and this function's parameter value is set to "10", this function enables the same machining as the original program.
- (3) This function cannot be used for the dwell function G04_X_(P_);.
- (4) This function cannot be used for the compensation amount of the tool compensation input.
- (5) This function can be used when decimal point type I is valid, but cannot be used when decimal point type II is valid.
- (6) This function cannot be used for a tool shape setting command (in G10L100 format).

Program example (Machining program : programmed with 1=10 μ m) (CNC unit is 1=1 μ m system)	"UNIT*10" parameter			
	10		1	
	X	Y	X	Y
N1 G90 G00 X0 Y0;	0	0	0	0
N2 G91 X-10000 Y-15000;	-100.000	-150.000	-10.000	-15.000
N3 G01 X-10000 Y-5000 F500;	-200.000	-200.000	-20.000	-20.000
N4 G03 X-10000 Y-10000 J-10000;	-300.000	-300.000	-30.000	-30.000
N5 X10000 Y-10000 R10000;	-200.000	-400.000	-20.000	-40.000
N6 G01 X20.000 Y20.000	-180.000	-380.000	0.000	-20.000



UNIT*10 ON



UNIT*10 OFF

2.3 Indexing Increment



Function and purpose

This function limits the command value for the rotary axis.

This can be used for indexing the rotary table, etc. It is possible to cause a program error with a program command other than an indexing increment (parameter setting value).



Detailed description

When the indexing increment (parameter) which limits the command value is set, the rotary axis can only be positioned with that indexing increment. If a program other than the indexing increment setting value is commanded, a program error (P20) will occur.

The indexing position will not be checked when the parameter is set to 0.

(Example) When the indexing increment setting value is 2 degrees, the machine coordinate position at the end point can only be commanded with the 2-degree increment.

G90 G01 C102.000 ; Moves to the 102 degree angle.

G90 G01 C101.000 ; Program error

G90 G01 C102 ; Moves to the 102 degree angle. (Decimal point type II)

The following axis specification parameter is used. (This depends on the MTB specifications.)

#	Item		Details	Setting range (unit)
2106	Index unit	Indexing increment	Set the indexing increment with which the rotary axis can be positioned.	0 to 360(°)



Precautions

- (1) When the indexing increment is set, positioning will be conducted in degree unit.
- (2) The indexing position is checked with the rotary axis, and is not checked with other axes.
- (3) When the indexing increment is set to 2 degrees, the rotary axis is set to the B axis, and the B axis is moved with JOG to the 1.234 position, an indexing error will occur if "G90B5." or "G91B2." is commanded.

Program Formats

3.1 Program Format

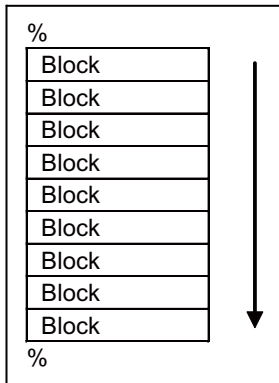
A collection of commands assigned to an NC to move a machine is called "program".

A program is a collection of units called "block" which specifies a sequence of machine tool operations.

Blocks are written in the order of the actual movement of a tool.

A block is a collection of units called "word" which constitutes a command to an operation.

A word is a collection of characters (alphabets, numerals, signs) arranged in a specific sequence.

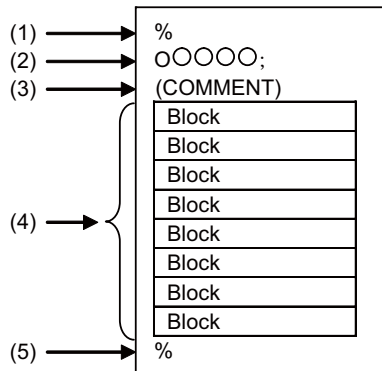




Detailed description

Program

A program format looks as follows.



(1) Program start

Input an End Of Record (EOR, %) at the head of a program.

It is automatically added when writing a program on an NC. When using an external device, do not forget to input it at the head of a program. For details, refer to the description of the file format.

(2) Program No.

Program Nos. are used to classify programs by main program unit or subprogram unit. They are designated by the address "O" followed by numbers of up to 8 digits. Program Nos. must be written at the head of programs.

A setting is available to prohibit O8000s and O9000s from editing (edit lock). Refer to the instruction manual for the edit lock.

(3) Comment

Data between control out "(" and control in ")" is ignored.

Information including program names and comments can be written in.

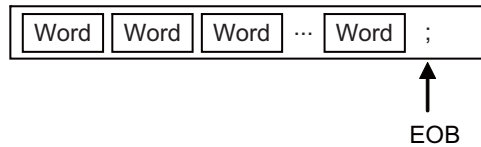
(4) Program section

A program is a collection of several blocks.

(5) Program end

Input an end of record (EOR, %) at the end of a program.

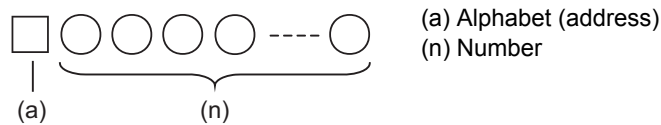
It is automatically added when writing a program on an NC.

Block and word**[Block]**

A block is a least command increment, consisting of words.

It contains the information which is required for a machine tool to execute a specific operation. One block unit constitutes a complete command.

The end of each block is marked with an End of Block (EOB, expressed as ";") for the sake of convenience).

[Word]

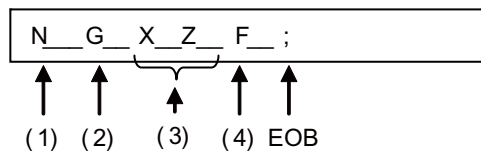
A word consists of a set of an alphabet, which is called an address, and numerals (numerical information).

Meanings of the numerical information and the number of significant digits of words differ according to an address.

Note

- (1) Leading zeros can be omitted from numerals.

The major contents of a word are described below.



- (1) Sequence No.

"Sequence No." consists of the address N followed by numbers of up to 8 digits. It is used as an index when searching a necessary block in a program (as branch destination and etc.). It does not affect the operation of a tool machine.

- (2) Preparatory function (G code, G function)

"Preparatory function (G code, G function)" consists of the address G followed by numbers of 2 or 3 digits (it may include 1 digit after the decimal point). G codes are mainly used to designate functions, such as axis movements and setting of coordinate systems. For example, G00 executes a positioning and G01 executes a linear interpolation.

- (3) Coordinate words

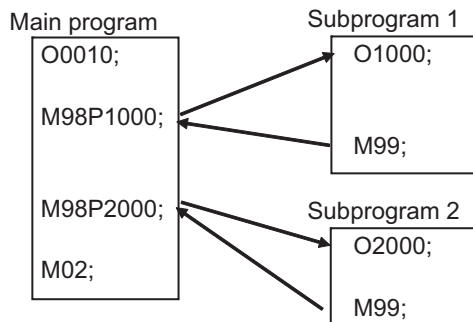
"Coordinate words" specify the coordinate positions and movement amounts of machine tool axes. They consist of an address which indicates each axis of a tool machine followed by numerical information (+ or - signs and numerals).

X, Y, Z, U, V, W, A, B and C are used as address. Coordinate positions and movement amounts are specified by either "incremental value commands" or "absolute value commands".

The axis name can be expanded to two letters depending on the MTB specifications. For details, refer to "15.12 Axis Name Extension".

- (4) Feed functions (F functions)

"Feed Functions (F functions)" designate the speed of a tool relative to a workpiece. They consist of the address F followed by numbers.

Main program and subprograms

Fixed sequences or repeatedly used parameters can be stored in the memory as subprograms which can then be called from the main program when required.

If a command is issued to call a subprogram while a main program is being executed, the subprogram will be executed. And when the subprogram is completed, the main program will be resumed.

Refer to "14.1 Subprogram Control; M98, M99, M198" for details of subprogram execution.

3.2 File Format



Function and purpose

Program file can be created using NC edit screen and PC.

It can be input/output between NC memory and an external I/O device. Hard discs stored in NC unit are regarded as an external I/O device. For the details of input/output method, refer to the instruction manual.

Program file format differs depending on the device which creates the program.



Detailed description

Devices available for input/output

Devices which can input/output program files are as follows.

External data input/output interface	M800W	M800S	M80	C80
NC memory	○	○	○	○
Serial	○	○	○	-
SD card in control unit	○	-	-	-
Front-side SD card	○	○	○	○ (*1)
Ethernet	○	○	○	○
Display unit-side data server	○	○	○	-
Front-side USB memory	○	○	○	○ (*2)

(*1) GOT back-side SD card

(*2) GOT front-side USB memory

Program file format

The file format for each external I/O device is as follows:

(1) NC memory (Creates program on NC)

```
(COMMENT) ;
G28XYZ ;
:
:
M02 ;
%
```

End of record (EOR, %)	The end of record (EOR, %) is automatically added. It does not need to be input purposely.
Program No. (O No.)	Not necessary.
File transfer	When multiple programs within the NC memory are transferred to an external device as serial, they will be integrated into one file in the external device. When a file containing multiple programs in an external device is transferred to NC memory as serial, it will be divided into one file per one program.

3 Program Formats

(2) External device (except for serials such as SD card and USB memory)

[Single program]	[Multiple programs]
<pre> CRLF (COMMENT) CRLF G28 XYZ CRLF : : M02 CRLF % ^Z </pre>	<pre> CRLF O100(COMMENT) CRLF G28 XYZ CRLF : : M02 CRLF O101(COMMENT1) CRLF : M02 CRLF % ^Z </pre>
End of record (EOR, %)	The first line (from % to LF, or CR LF) will be skipped. Also, the content after the second % will not be transferred. "% " must be included in the first line because if not, the necessary information when transferring a file to an NC memory cannot be transferred.
Program No. (O No.)	O No. before (COMMENT) will be ignored and the file name will be given the priority.
File transfer	Multiple programs cannot be transferred or collated between the serial-connected device and the external devices except for the serial connection. When a file containing multiple programs in an external device is transferred to NC memory as serial, it will be divided into one file per one program. When transferring divided programs one by one from an external device, which is not serial, (multiple programs) to an NC memory, the head program name can be omitted like "(COMMENT)" only when the transferring destination file name is designated to the file name field of device B.
Program name	Program name should be designated with up to 32 alphanumeric characters (29 characters for a multi-part system program).
End of block (EOB, ;)	When the I/O parameter "CR output" is set to "1", EOB becomes CRLF.

(3) External device (serial)

<pre> % LF O100(COMMENT) LF G28 XYZ LF : : M02 LF % </pre>	
End of record (EOR, %)	The first line (from % to LF, or CR LF) will be skipped. Also, the content after the second % will not be transferred. "% " must be included in the first line because if not, the necessary information when transferring a file to an NC memory cannot be transferred.
File transfer	Multiple programs cannot be transferred or collated between the serial-connected device and the external devices except for the serial connection. When transferring a file as serial, the head program name can be omitted like "(COMMENT)" only when the transferring destination file name is designated to the file name field of device B.
Program name	Program name should be designated with up to 32 alphanumeric characters (29 characters for a multi-part system program).
End of block (EOB, ;)	When the I/O parameter "CR output" is set to "1", EOB becomes CRLF.

3.3 Optional Block Skip

3.3.1 Optional Block Skip; /



Function and purpose

This function selectively ignores a section of a machining program from a "/" (slash code) to the end of the block.



Detailed description

Provided that the optional block skip switch is ON, a section of a machining program from a "/" to the end of the block are ignored. They are executed if the switch is OFF.

Parity check is valid regardless of whether the optional block skip switch is ON or OFF.

When, for instance, all blocks are to be executed for one workpiece but specific blocks are not to be executed for another workpiece, one machining program can be used to machine different parts by inserting the "/" into those specific blocks.



Program example

- (1) When the parameter "#1274 ext10/bit4" is set to "0" and the parameter "#1226 aux10/bit1" is set to "0":

A "/" placed in the middle of a block is always interpreted as a division instruction regardless of whether or not the optional block skip signal state is ON or OFF.

```
G00 X0. Z0.;
#101 = [ 100. / 4 ];           Sets "25." to #101. (As the result of execution of a division instruction)
G00 Z[ 100. / 4 ];           Moves Z axis to "25.". (As the result of execution of a division instruction)
#102 = 100. / #101 ;         Sets "4." to #102. (As the result of execution of a division instruction)
M30 ;
```

- (2) When the parameter "#1274 ext10/bit4" is set to "0" and the parameter "#1226 aux10/bit1" is set to "1":

A "/" placed in a bracketed ("[]") expression is interpreted as a division instruction.

As for a "/" that appears in any other contexts, the section of the block following the "/" will be skipped if the optional skip signal is ON, and the "/" itself will be ignored if the optional skip signal is OFF.

<Operation example of a case when optional block skip signal is ON>

```
G00 X0. Z0.;
#101 = [ 100. / 4 ];           Sets "25." to #101. (As the result of execution of a division instruction)
G00 X100. / Z200. ;           Moves X axis to "100. No Z axis movements made. (As the result of skipping the section of the block after "/")
G00 Z[ 100. / 4 ];           Moves Z axis to "25.". (As the result of execution of a division instruction)
#102 = 100. / #101 ;         Sets "100." to #102. (As the result of skipping the section of the block after "/")
M30 ;
```

<Operation example of a case when optional block skip signal is OFF>

```
G00 X0. Z0.;
#101 = [ 100. / 4 ];           Sets "25." to #101. (As the result of execution of a division instruction)
G00 X100. / Z200. ;           Moves X axis to "100." and Z axis to "200.". (As the result of ignoring "/")
G00 Z[ 100. / 4 ];           Moves Z axis to "25.". (As the result of execution of a division instruction)
#102 = 100. / #101 ;         Program error (P242) occurs. (As the result of ignoring "/")
M30 ;
```

- (3) When the parameter "#1274 ext10/bit4" is set to "1" :

When a "/" is placed in a bracketed expression or when an expression that includes a "/" is on the right side of an equation, the "/" is interpreted as a division instruction.

As for a "/" that appears in any other contexts, the section of the block following the "/" will be skipped if the optional skip signal is ON, and the "/" itself will be ignored if the optional skip signal is OFF.

<Operation example of a case when optional block skip signal is ON>

G00 X0. Z0.;

#101 = [100. / 4] ; Sets "25." to #101. (As the result of execution of a division instruction)

G00 X100. / Z200. ; Moves X axis to "100. No Z axis movements made. (As the result of skipping the section of the block after "/")

G00 Z[100. / 4] ; Moves Z axis to "25.". (As the result of execution of a division instruction)

#102 = 100. / #101 ; Sets "4." to #102. (As the result of execution of a division instruction)

M30 ;

<Operation example of a case when optional block skip signal is OFF>

G00 X0. Z0.;

#101 = [100. / 4] ; Sets "25." to #101. (As the result of execution of a division instruction)

G00 X100. / Z200. ; Moves X axis to "100." and Z axis to "200.". (As the result of ignoring "/")

G00 Z[100. / 4] ; Moves Z axis to "25.". (As the result of execution of a division instruction)

#102 = 100. / #101 ; Sets "4." to #102. (As the result of execution of a division instruction)

M30 ;



Precautions

- (1) When the parameter "#1274 ext10/bit4" is set to "0" and parameter "#1226 aux10/bit1" is set to "0", put the "/" code for optional block skip at the beginning of a block. If it is placed inside the block, it is assumed as a user macro, a division instruction.

(Example)

N20 G01 X25. /Z25. ; NG (User macro, a division instruction; a program error results.)

/N20 G01 X25. Z25. ; OK

When parameter "#1274 ext10/bit4" = "0" and parameter "#1226 aux10/bit1" = "1", a "/" placed in the middle of a block functions as a starting point of the optional skip.

To use a "/" as a division instruction, bracket (enclose in square brackets) the formula containing a slash code.

- (2) A space immediately followed by a "/" at the very beginning of a block is always regarded as equal to a "/" at the head of a block regardless of the value set in parameter "#1226 aux10/bit1".
- (3) The optional block skip is processed immediately before the pre-read buffer. Consequently, it is not possible to skip up to the block which has been read into the pre-read buffer.
- (4) This function is valid even during a sequence number search.
- (5) All blocks with the "/" code are also input and output during tape storage and tape output, regardless of the position of the optional block skip switch.

3.3.2 Optional Block Skip Addition ; /n



Function and purpose

Whether the block with "/n (n:1 to 9)" (slash) is executed during automatic operation and searching is selected. By using the machining program with "/n" code, different parts can be machined by the same program.



Detailed description

The block with "/n" (slash) code is skipped when the "/n" is programmed to the head of the block and the optional block skip n signal is turned ON. For a block with the "/n" code inside the block (not at the head of the block), the program is operated according to the value of the parameter "#1226 aux10/bit1" setting. When the optional block skip n signal is OFF, the block with "/n" is executed.



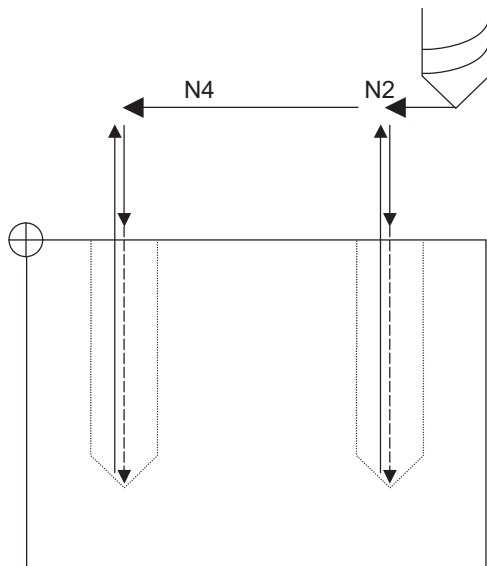
Program example

(1) When the 2 parts like the figure below are machined, the following program is used. When the optional block skip 5 signal is ON, the part 1 is created. When the optional block skip 5 signal is OFF, the part 2 is created.

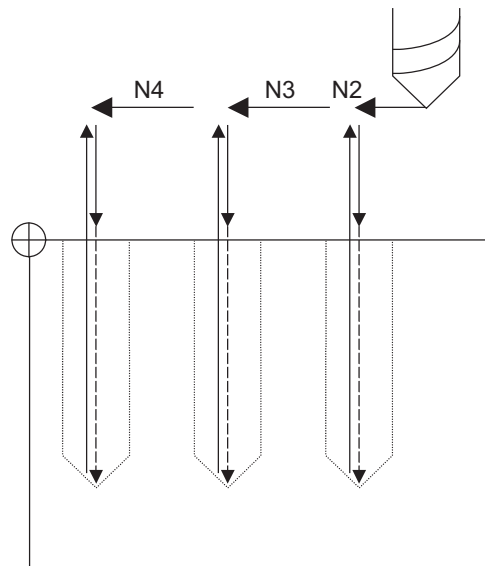
```

N1 G54 ;
N2 G90 G81 X50. Z-20. R3. F100 ;
/5 N3 X30. ;
N4 X10. ;
N5 G80 ;
M02 ;
    
```

Part 1
Optional block skip 5 signal ON



Part 2
Optional block skip 5 signal OFF



- (2) When two or more "/n" codes are commanded at the head of the same block, the block will be ignored if either of the optional block skip n signals corresponding to the command is ON.

N01 G90 Z3. M03 S1000 ;	(a) Optional block skip 1 signal ON (Optional block skip 2.3 signal OFF)
/1/2 N02 G00 X50. ;	N01 -> N08 -> N09 -> N10 -> N11 -> N12
/1/2 N03 G01 Z-20. F100 ;	
/1/2 N04 G00 Z3. ;	
/1 /3 N05 G00 X30. ;	(b) Optional block skip 2 signal ON (Optional block skip 1.3 signal OFF)
/1 /3 N06 G01 Z-20. F100 ;	N01 -> N05 -> N06 -> N07 -> N11 -> N12
/1 /3 N07 G00 Z3. ;	
/2/3 N08 G00 X10. ;	(c) Optional block skip 3 signal ON (Optional block skip 1.2 signal OFF)
/2/3 N09 G01 Z-20. F100 ;	N01 -> N02 -> N03 -> N04 -> N11 -> N12
/2/3 N10 G00 Z3. ;	
N11 G28 X0 M05 ;	
N12 M02 ;	

- (3) When the parameter "#1226 aux10/bit1" is "1" and two or more "/n" are commanded inside the same block, the commands following "/n" in the block are ignored if either of the optional block skip n signals corresponding to the command is ON.

N01 G91 G28 X0.Y0.Z0.;	N03 block will operate as follows.
N02 G01 F1000;	(a) Optional block skip 1 signal ON Optional block skip 2 signal OFF
N03 X1. /1 Y1. /2 Z1.;	"Y1. Z1." is ignored.
N04 M30;	(b) Optional block skip 1 signal OFF Optional block skip 2 signal ON
	"Z1." is ignored.

3.4 G Code

3.4.1 Modal, Unmodal

G codes define the operation modes of each block in programs.

G codes can be modal or unmodal command.

Modal commands always designate one of the G codes in the group as the NC operation mode. The operation mode is maintained until a cancel command is issued or other G code among the same group is commanded.

An unmodal command designates the NC operation mode only when it is issued. It is invalid for the next block.

3.4.2 G Code Lists

G code	Group	Function	Section
Δ 00	01	Positioning	6.1
0.5	28	Rapid traverse block overlap	7.13.1
Δ 01	01	Linear interpolation	6.2
02	01	Circular interpolation CW	6.3
		R-specified circular interpolation CW	6.4
		Helical interpolation CW	6.7
		Spiral/Conical interpolation CW (type2)	6.14
03	01	Circular interpolation CCW	6.3
		R-specified circular interpolation CCW	6.4
		Helical interpolation CCW	6.7
		Spiral/Conical interpolation CCW (type2)	6.14
02.1	01	Spiral/Conical interpolation CW (type1)	6.14
03.1	01	Spiral/Conical interpolation CCW (type1)	6.14
02.2	01	Involute interpolation CW	6.18
03.2	01	Involute interpolation CCW	6.18
02.3	01	Exponential function interpolation positive rotation	6.12
03.3	01	Exponential function interpolation negative rotation	6.12
02.4	01	3-dimensional circular interpolation CW	6.15
03.4	01	3-dimensional circular interpolation CCW	6.15
04	00	Dwell (Time-based designation)	8.1
05	00	High-speed machining mode	17.1
		High-speed high-accuracy control II/III	17.3
05.1	00	High-speed high-accuracy control I	17.3
		Spline interpolation	17.4
06.2	01	NURBS interpolation	6.16
07	00	Hypothetical axis interpolation	6.17
07.1 107	19	Cylindrical interpolation	6.9
08	00	High-accuracy control	17.2
09	00	Exact stop check	7.10
10	00	Data input by program (Parameter input, Compensation input, Tool shape input, R-Navi data input)	15.9
		Tool life management data input	15.10
		Coordinate rotation input by parameter	20.10
11	00	Data input by program cancel (Parameter input, Compensation input, Tool shape input, R-Navi data input)	15.9
		Tool life management data input	15.10
10.9	00	Diameter/Radius designation switch	5.2.1
12	00	Circular cutting CW	6.10

3 Program Formats

G code	Group	Function	Section
13	00	Circular cutting CCW	6.10
12.1 112	21	Polar coordinate interpolation ON	6.11
* 13.1 113	21	Polar coordinate interpolation cancel	6.11
14			
* 15	18	Polar coordinate command OFF	6.13
16	18	Polar coordinate command ON	6.13
Δ 17	02	X-Y plane selection	6.5
Δ 18	02	Z-X plane selection	6.5
Δ 19	02	Y-Z plane selection	6.5
Δ 20	06	Inch command	5.3
Δ 21	06	Metric command	5.3
22	04	Stroke check before travel ON	21.1
23	04	Stroke check before travel cancel	21.1
24			
25			
26	00	Return to the selected point/Tapping retract	15.13
27	00	Reference position check	20.15
28	00	Reference position return	20.12
29	00	Start position return	20.12
30	00	2nd to 4th reference position return	20.13
30.1	00	Tool change position return 1	20.14
30.2	00	Tool change position return 2	20.14
30.3	00	Tool change position return 3	20.14
30.4	00	Tool change position return 4	20.14
30.5	00	Tool change position return 5	20.14
30.6	00	Tool change position return 6	20.14
31	00	Skip/Speed change skip	22.2
		Multi-step skip 2	22.4
31.1	00	Multi-step skip 1-1	22.3
31.2	00	Multi-step skip 1-2	22.3
31.3	00	Multi-step skip 1-3	22.3
32			
33	01	Thread cutting	6.6
34	00	Special fixed cycle (bolt hole circle)	13.2.1
35	00	Special fixed cycle (line at angle)	13.2.2
36	00	Special fixed cycle (arc)	13.2.3
37	00	Automatic tool length measurement	22.1
37.1	00	Special fixed cycle (grid)	13.2.4
38	00	Tool radius compensation vector designation	12.3
39	00	Tool radius compensation corner arc	12.3
* 40	07	Tool radius compensation (tool nose radius compensation) cancel	12.3
		3-dimensional tool radius compensation cancel	12.5
		3-dimensional tool radius compensation (Tool's vertical-direction compensation) cancel	19.5
41	07	Tool radius compensation (tool nose radius compensation) left	12.3
		3-dimensional tool radius compensation left	12.5
42	07	Tool radius compensation (tool nose radius compensation) right	12.3
		3-dimensional tool radius compensation right	12.5

3 Program Formats

G code	Group	Function	Section
* 40.1 150	15	Normal line control cancel	15.7
41.1 151	15	Normal line control left ON	15.7
42.1 152	15	Normal line control right ON	15.7
41.2	07	3-dimensional tool radius compensation (Tool's vertical-direction compensation) (left)	19.5
42.2	07	3-dimensional tool radius compensation (Tool's vertical-direction compensation) (right)	19.5
43	08	Tool length compensation (+)	12.2
44	08	Tool length compensation (-)	12.2
43.1	08	Tool length compensation along the tool axis ON	19.2
43.4	08	Tool center point control type1 ON	19.3
43.5	08	Tool center point control type2 ON	19.3
43.7	08	Tool position compensation start	19.1
45	00	Tool position offset (extension)	12.6
46	00	Tool position offset (reduction)	12.6
47	00	Tool position offset (double elongation)	12.6
48	00	Tool position offset (double contraction)	12.6
* 49	08	Tool length compensation cancel	12.2
		Tool length compensation along the tool axis	19.2
		Tool center point control cancel	19.3
		Tool position compensation cancel	19.1
* 50	11	Scaling cancel	20.11
51	11	Scaling ON	20.11
* 50.1	19	Mirror image by G code cancel	15.6
51.1	19	Mirror image by G code ON	15.6
52	00	Local coordinate system setting	20.5
53	00	Basic machine coordinate system selection	20.3
53.1	00	Tool axis direction control (type 1)	19.4.8
53.6	00	Tool axis direction control (type 2)	19.4.8
* 54	12	Workpiece coordinate system 1 selection	20.6
55	12	Workpiece coordinate system 2 selection	20.6
56	12	Workpiece coordinate system 3 selection	20.6
57	12	Workpiece coordinate system 4 selection	20.6
58	12	Workpiece coordinate system 5 selection	20.6
59	12	Workpiece coordinate system 6 selection	20.6
54.1	12	Extended workpiece coordinate system selection	20.6
54.4	27	Workpiece installation error compensation	-
60	00(01)	Unidirectional positioning	6.8.1
61	13	Exact stop check mode	7.11
61.1	13	High-accuracy control ON	17.2
61.2	13	High-accuracy spline	17.6
61.4	13	Spline interpolation 2	17.5
62	13	Automatic corner override	7.14.1
63	13	Tapping mode	7.15
* 64	13	Cutting mode	7.16
65	00	User macro simple call	14.4.1
66	14	User macro modal call A	14.4.2
66.1	14	User macro modal call B	14.4.3

3 Program Formats

G code	Group	Function	Section
* 67	14	User macro modal call cancel	14.3
68	16	Coordinate rotation by program ON	20.9
		3-dimensional coordinate conversion mode ON	20.8
68.2	16	Inclined surface machining command	19.4
		R-Navi data input (Selecting the registered machining surface)	15.9.5
68.3	16	Inclined surface machining command (Based on tool axis direction)	19.4
* 69	16	Coordinate rotation by program cancel	20.9
		3-dimensional coordinate conversion mode OFF	20.8
		Inclined surface machining cancel	19.4
		R-Navi data input (Canceling the selected machining surface)	15.9.5
70	09	User fixed cycle	
71	09	User fixed cycle	
72	09	User fixed cycle	
73	09	Fixed cycle (step)	13.1.10
74	09	Fixed cycle (reverse tap)	13.1.11
75	09	Fixed cycle (circle cutting cycle)	13.1.12
76	09	Fixed cycle (Fine boring)	13.1.13
77	09	User fixed cycle	
78	09	User fixed cycle	
79	09	User fixed cycle	
* 80	09	Fixed cycle cancel	13.1
81	09	Fixed cycle (drill/spot drill)	13.1.1
82	09	Fixed cycle (drill/counter boring)	13.1.2
83	09	Fixed cycle (deep drilling/small-diameter deep-hole drilling)	13.1.3
84	09	Fixed cycle (tapping)	13.1.4
85	09	Fixed cycle (boring)	13.1.5
86	09	Fixed cycle (boring)	13.1.6
87	09	Fixed cycle (back boring)	13.1.7
88	09	Fixed cycle (boring)	13.1.8
89	09	Fixed cycle (boring)	13.1.9
Δ 90	03	Absolute value command	5.1
Δ 91	03	Incremental value command	5.1
92	00	Coordinate system setting	20.4
		Spindle clamp speed setting	10.3
92.1	00	Workpiece coordinate system preset	20.7
93	05	Inverse time feed	7.5
Δ 94	05	Feed per minute (asynchronous feed)	7.4
Δ 95	05	Feed per revolution (synchronous feed)	7.4
Δ 96	17	Constant surface speed control ON	10.2
Δ 97	17	Constant surface speed control OFF	10.2
* 98	10	Fixed cycle Initial level return	13.1.16
99	10	Fixed cycle (R point level return)	13.1.16
100-225	00	User macro (G code call) Max. 10	14.4.4
113.1	00	Spindle synchronization I cancel	18.1.1
114.1	00	Spindle synchronization I	18.1.1
120.1	00	Machining condition selection I	17.7
121	00	Machining condition selection I cancel	17.7
122	00	Activate sub part system I	16.3.1
127	00	Prohibits the arbitrary reverse run in all part systems	15.8
140	00	Arbitrary axis exchange command	16.2.1


G code	Group	Function	Section
141	00	Arbitrary axis exchange return command	16.2.1
142	00	Arbitrary axis exchange/reference axis arrange return command	16.2.1
145	00	Cancel sub part systems	16.3
160	00	Torque limitation skip	22.6
162	00	Spindle speed fluctuation detection	10.5
163	00	Spindle speed fluctuation detection cancel	10.5
180	00	Interactive cycle insertion program	15.11
186	00	Interference check III interfering object data enable command	21.2
187	09	Thread milling cycle	13.1.14



Precautions



- (1) Codes marked with * are codes that must be or are selected in the initial state.
The codes marked with Δ are codes that should be or are selected in the initial state by the parameters.
- (2) If two or more G codes from the same code are commanded, the latter G code will be valid.
- (3) This G code list is a list of conventional G codes. Depending on the machine, movements that differ from the conventional G commands may be included when called by the G code macro. Refer to the Instruction Manual issued by the MTB.
- (4) Whether the modal is initialized or not depends on each reset input.
 - ♦"Reset 1"
The modal is initialized when the reset initialization parameter (#1151 rstinit) is ON. (This depends on the MTB specifications.)
 - ♦"Reset 2" and "Reset & rewind"
The modal is initialized when the signal is input.
 - ♦Reset at emergency stop release
Conforms to "Reset 1".
 - ♦When modal is automatically reset at the start of individual functions such as reference position return
Conforms to "Reset & rewind".

CAUTION

 The commands with "no value after G" will be handled as "G00".

3.5 Precautions before Starting Machining

 **CAUTION**

-  When creating the machining program, select the appropriate machining conditions, and make sure that the performance, capacity and limits of the machine and NC are not exceeded. The examples do not take into account the machining conditions.
-  Before starting actual machining, always carry out a graphic check, a dry run operation, and a single block operation to check the machining program, tool offset amount, workpiece offset amount, etc.

Pre-read Buffer

4.1 Pre-read Buffer



Function and purpose

During automatic operation, the contents of one block ahead are normally pre-read so that program analysis processing is conducted smoothly. However, during tool radius compensation, a maximum of 5 blocks are pre-read for the intersection point calculation including interference check.



Detailed description

The specifications of pre-read buffers in 1 block are as follows:

- (1) The data of 1 block is stored in this buffer.
- (2) When comments and the optional block skip function is ON, the data extending from the "/" (slash) code up to the EOB code are not read into the pre-read buffer.
- (3) The pre-read buffer contents are cleared with resetting.
- (4) When the single block function is ON during continuous operation, the pre-read buffer stores the next block's data and then stops operation.
- (5) The way to prohibit the M command which operates the external controls from pre-reading, and to make it to recalculate, is as follows:
Identify the M command which operates the external controls by a PLC, and turn on the "recalculation request" on PLC output signal. (When the "recalculation request" is turned ON, the block that has been pre-read is recalculated.)
These operations depend on the MTB specifications.



Precautions

- (1) Depending on whether the program is executed continuously or by single blocks, the timing of the validation/invalidation of the PLC signals, including optional block skip, will differ.
- (2) If the PLC signal such as optional block skip is turned ON/OFF with the M command, the PLC signal operation will not be effective for the program pre-read with the buffer register.

Position Commands

5.1 Position Command Methods ; G90,G91



Function and purpose

By using the G90 and G91 commands, it is possible to execute the next coordinate commands using absolute values or incremental values.

The R-designated circle radius and the center of the circle determined by I, J, K are always incremental value commands.



Command format

G90/G91 X__ Y__ Z__ α__ ;

G90	Absolute command
G91	Incremental command
X,Y,Z,α	Coordinate values (α is the additional axis.)



Detailed description

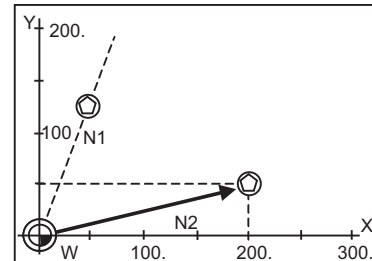
- (1) Regardless of the current position, in the absolute value mode, it is possible to move to the position of the workpiece coordinate system that was designated in the program.

N1 G90 G00 X0 Y0 ;

In the incremental value mode, the current position is the start point (0), and the movement is made only the value determined by the program, and is expressed as an incremental value.

N2 G90 G01 X200. Y50. F100 ;
N2 G91 G01 X200. Y50. F100 ;

Using the command from the 0 point in the workpiece coordinate system, it becomes the same coordinate command value in either the absolute value mode or the incremental value mode.



 Tool

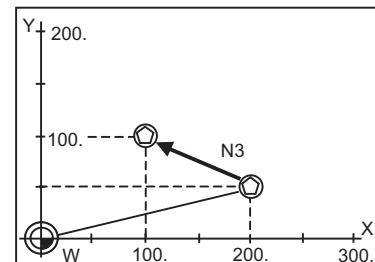
- (2) For the next block, the last G90/G91 command that was given becomes the modal.

(G90) N3 X100. Y100. ;

The axis moves to the workpiece coordinate system X = 100. mm and Y = 100. mm position.

(G91) N3 X-100. Y50. ;

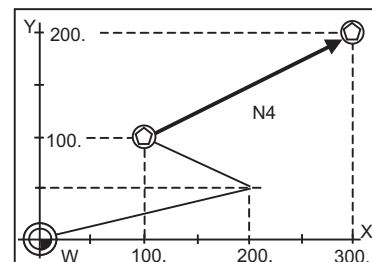
The X axis moves to -100. mm and the Y axis to +50.0 mm as an incremental value, and as a result X moves to 100. mm and Y to 100. mm.



- (3) Since multiple commands can be issued in the same block, it is possible to command specific addresses as either absolute values or incremental values.

N4 G90 X300. G91 Y100. ;

The X axis is treated in the absolute value mode, and with G90 is moved to the workpiece coordinate system 300. mm position. The Y axis is moved +100. mm with G91. As a result, Y moves to the 200. mm position. In terms of the next block, G91 remains as the modal and becomes the incremental value mode.



- (4) When the power is turned ON, it is possible to select whether you want absolute value commands or incremental value commands with the #1073 I_Absm parameter.

- (5) Even when commanding with the manual data input (MDI), it will be treated as a modal from that block.

5.2 Diameter Designation and Radius Designation

5.2.1 Diameter/Radius Designation Switch; G10.9



Function and purpose

The method of commanding a travel distance (command with a diameter dimension/command with a radius dimension (as-is distance)) in a program is defined individually for each axis depending on MTB specifications (parameter "#1019 dia").

Diameter/Radius designation switch function, however, enables you to switch the diameter/radius designation of each axis using a G code at your desired timing. When you use the function to switch diameter/radius designation, it helps you create a program more flexibly according to each machining situation.

Term

Diameter/Radius being switched	This refers to a condition where the diameter/radius selection of an axis is different from the power-ON state.
--------------------------------	---



Command format

Diameter/Radius designation switch enables you to select any desired NC axis, excluding rotary axes, and switch the diameter/radius designation of the axis.

Diameter/Radius designation switch

G10.9 Axis address 1__ Axis address 2__ ... Axis address n__ ;

Axis address n	Axis address for which diameter/radius designation is switched. Select radius or diameter designation with a value that follows the axis address. 0: Radius designation 1: Diameter designation If you select no axis address, the diameter/radius designation statuses of all the axes of the part system are returned to the initial power-ON state.
----------------	--



Detailed description

- (1) G10.9 is a non-modal command that belongs to Group 0.
- (2) G10.9 is effective for all the G code lists.
- (3) If G10.9 is commanded together with any other G code in a block, the program error (P33) occurs.
- (4) For the axis address, specify the address set in "#1013 axname" or "#1014 incax" (*1). If you specify a non-existent axis address, the program error (P33) occurs.
 - (*1) The axis address set in "#1014 incax" can be specified only when "#1076 AbsInc" is "1".
These settings depend on the MTB specifications.
- (5) If a rotary axis is specified with "Axis address", the program error (P32) occurs.
- (6) If a number with a decimal point is specified with "Axis address", the fraction is ignored. If any value other than "0" or "1" is specified, the program error (P35) occurs. If specified with a variable, the fraction is rounded to the closest whole number.
- (7) G10.9 switches the diameter/radius designation for programmed coordinates command and coordinates of counter in Monitor screen. However, diameter/radius designation is not switched for data including parameters, workpiece offsets, tool data and tool offsets.
- (8) The axis specified for diameter/radius designation switch by G10.9 is effective until G10.9 is given again for switching or until Reset or Emergency stop is input. After Reset or Emergency stop, the diameter/radius designation before G10.9 is restored.
- (9) If diameter/radius designation switch is not performed for an axis in the part system of G10.9 command, the axis behaves as follows:
 - ♦The axis behaves according to the setting of "#1019 dia" for L system. This setting depends on the MTB specifications.
 - ♦The axis acts as radius-designation axis in the M system.
- (10) Irrespective of the "#1019 dia" and "#1077 radius" settings, the designation selected by G10.9 has priority.
- (11) If you use G10.9 to switch diameter/radius designation for two axes (X axis and Z axis) at a time, program the command as follows:
G10.9 X1 Z0;
In the example above, X axis is switched to diameter designation, and Z axis to radius designation.



Relationship with other functions

- (1) Restart search
If you perform restart search after a G10.9 block, the diameter/radius designation switched by G10.9 is applied.
- (2) Arbitrary reverse run
If you perform reverse run for a G10.9 block, the reverse run stops at the G10.9 block, and is disabled for the prior blocks before the G10.9 block.
- (3) Manual arbitrary reverse run
If you perform forward run for G10.9 block, the diameter/radius designation switched by G10.9 is applied. If you perform reverse run for a G10.9 block, the reverse run stops at the G10.9 block, and is disabled for the prior blocks before the G10.9 block.
- (4) Chopping
G10.9 is unable to switch diameter/radius designation of a chopping axis. If G10.9 is given to an axis that is in chopping mode, or if you enable chopping mode for an axis for which diameter/radius designation is being switched by G10.9, the operation error (M01 0095) occurs.
If the alarm above has been caused by giving G10.9 to an axis that is in chopping mode, the chopping axis keeps moving.
Use the corresponding PLC signal or NC reset to stop the axis.
- (5) Synchronous control
For a master or slave axis that is in synchronous operation, you can switch diameter/radius designation using G10.9.
Even when the diameter/radius designation status is different between master and slave axes, their synchronization will not fail. The coordinates displayed on Monitor screen are dependent on the diameter/radius designation of each axis.

(6) Arbitrary axis exchange control

The diameter/radius command switching for an axis in arbitrary axis exchange control mode depends on the MTB specifications (*1).

(*1) Parameters of the command address: "#12071 adr_abs[1]" - "#12078 adr_abs[8]", "#12079 adr_inc[1]" - "#12086 adr_inc[8]"

If an arbitrary axis exchange-related command (G140/G141/G142) has given in a part system, all the axes of the part system are returned to the initial status under the exchanged axis configuration after the axis exchange. For an axis that moved to another part system due to an arbitrary axis exchange-related command, the switched diameter/radius designation is retained.

Also for an axis that moved to another part system due to Reset, the switched diameter/radius designation is retained.

(7) Program format switch

In M system, the setting of "#1019 dia" is disabled, thus all the axes are treated as radius-designation axis. However, G10.9 enables a diameter-value command.

When you switch the program format, the diameter/radius designation is returned to the default state for all the axes of the part system.

(8) Axis name extension

Diameter/Radius designation switch with G10.9 is disabled for a name-extended axis.

(9) Manual arbitrary feed

If a position command in manual arbitrary feed is given, the diameter/radius axis setting of each axis at power ON is applied. Therefore, even if a manual arbitrary feed command is given to an axis in diameter/radius designation switch mode using the G10.9 command, the diameter/radius switch status is ignored.

(10) Cylindrical interpolation/Polar coordinate interpolation

The diameter/radius designation of each axis at which these functions are being executed follows the setting of "#8111 Milling Radius" without being affected by the G10.9 command.

When these functions are canceled, the setting returns to the diameter/radius axis designation that was valid before the functions are executed.

If G10.9 is commanded while these functions are being executed, a program error (P481) will occur.

(11) Graphic check (2D/3D), Graphic trace

During graphic check or graphic trace, the diameter/radius designation status switched by G10.9 is reflected both in the drawing image and the counter of coordinate values.



Precautions

- (1) If diameter/radius designation is switched, the travel distance changes even though the command value is unchanged. Thus special care must be taken when creating or executing a machining program.
- (2) Command the feedrate with the radius value regardless of whether the diameter designation or radius designation is selected. (This is applied to both the movement amount per rotation and that per minute.)
- (3) Diameter/Radius designation is not switched for a value that is read or written using PLC window or system variables. For M system, it is treated as a radius value. For L system, the value follows the setting of "#1019 dia".
- (4) When manual handle feed or incremental feed is performed at manual interruption, the switched diameter/radius designation is applied.
- (5) In a G10.9 command block, the control confirms that all the axes of the part system are decelerated to a stop before switching the diameter/radius designation. If G10.9 is given between cutting blocks, it causes acceleration and deceleration, which may damage the workpiece. Thus make sure that the tool is away from the workpiece before giving G10.9.
- (6) If a G10.9 command which has the same diameter/radius designation as the existing operation is given, the G10.9 command fails to be enabled. Thus if a G code that causes an error when combined with G10.9 is given together, no error occurs.
- (7) Do not give "Start point designation timing synchronization" or "M code output during axis traveling" command to an axis for which diameter/radius designation is being switched. If commanded, it cannot be assured that the timing synchronization or the M code output is performed in the correct position.

5.3 Inch/Metric Conversion; G20, G21



Function and purpose

The commands can be changed between inch and metric with the G20/G21 command.



Command format

Inch command

G20;

Metric command

G21;



Detailed description

The G20 and G21 commands merely select the command units. They do not select the Input units. G20 and G21 selection is meaningful only for linear axes. It is invalid for rotation axes.

Output unit, command unit and setting unit

The counter, parameter setting and display unit are determined by parameter "#1041 I_inch". The movement/speed command will be displayed as metric units when "#1041 I_inch" is ON during the G21 command mode. The internal unit metric data of the movement/speed command will be converted into an inch unit and displayed when "#1041 I_inch" is OFF during the G20 command mode. The command unit for when the power is turned ON and reset is decided by combining the parameters "#1041 I_inch", "#1151 rstint" and "#1210 RstGmd/bit5".

These parameter settings depend on the MTB specifications.

NC axis

Item	Initial inch OFF (metric internal unit) #1041 I_inch=0		Initial inch ON (inch internal unit) #1041 I_inch=1	
	G21	G20	G21	G20
Movement/speed command	Metric	Inch	Metric	Inch
Counter display	Metric	Metric	Inch	Inch
Speed display	Metric	Metric	Inch	Inch
User parameter setting/display	Metric	Metric	Inch	Inch
Workpiece/tool offset setting/display	Metric	Metric	Inch	Inch
Handle feed command	Metric	Metric	Inch	Inch

PLC axis

Item	#1042 pcinch=0 (metric)	#1042 pcinch=1 (inch)
Movement/speed command	Metric	Inch
Counter display	Metric	Inch
User parameter setting/display	Metric	Inch



Precautions

- (1) The parameter and tool data will be input/output with the unit set by "#1041 I_inch".
If "#1041 I_inch" is not found in the parameter input data, the unit will follow the unit currently set to NC.
- (2) The unit of read/write used in PLC window is fixed to metric unit regardless of a parameter and G20/G21 command modal.
- (3) A program error (P33) will occur if G20/G21 command is issued in the same block as following G codes. Command in a separate block.
 - G05 (High-speed machining mode)
 - G7.1 (Cylindrical Interpolation)
 - G12.1 (Polar coordinate interpolation)

5.4 Decimal Point Input



Function and purpose

This function enables to input decimal points. It assigns the decimal point in millimeter or inch units for the machining program input information that defines the tool paths, distances and speeds.

Whether to apply minimum input command increment (type I) or zero point (type II) to the least significant digit of data without a decimal point depends on the MTB specifications (parameter "#1078 Decpt2").



Detailed description

- (1) The decimal point command is valid for the distances, angles, times, speeds and scaling rate, in machining programs. (Note, only after G51)
- (2) In decimal point input type I and type II, the values of the data commands without the decimal points are shown in the table below.

Command	Command unit	Type I	Type II
X1;	cunit=10000	1000 (μm , 10^{-4}inch , 10^{-3}°)	1 (mm, inch, °)
	cunit= 1000	100	1
	cunit= 100	10	1
	cunit= 10	1	1

- (3) The valid addresses for the decimal points are X, Y, Z, U, V, W, A, B, C, I, J, K, E, F, P, Q, and R. However, P is valid only during scaling. For details, refer to the list.
- (4) In decimal point command, the valid range of command value is as shown below. (for input command increment cunit=10)

	Movement command (linear)	Movement command (rotary)	Feedrate	Dwell
Input unit [mm]	-99999.999 to 99999.999	-99999.999 to 99999.999	0.001 to 1000000.000	0 to 99999.999
Input unit [inch]	-9999.9999 to 9999.9999		0.0001 to 1000000.0000	

- (5) The decimal point command is valid even for commands defining the variable data used in subprograms.
- (6) While the smallest decimal point command is validated, the smallest unit for a command without a decimal point designation is the smallest command input unit set in the specifications ($1\mu\text{m}$, $10\mu\text{m}$, etc.) or mm can be selected. This selection can be made with parameter "#1078 Decpt2".
- (7) Decimal point commands for decimal point invalid addresses are processed as integer data only and everything after the decimal point is ignored. Decimal point invalid addresses include the followings; D,H,L,M,N,O,P,S,T. All variable commands, however, are treated as data with decimal points.
- (8) "Input command increment tenfold" is applied in the decimal point type I mode, but not in the decimal point type II mode.

Decimal point input I, II and decimal point command validity

Decimal point input I and II will result as follows when decimal points are not used in an address which a decimal point command is valid.

Both decimal point input I and II will produce the same result when a command uses a decimal point.

(1) Decimal point input I

The least significant digit of command data matches the command unit.

(Example) When "X1" is commanded in 1 μ m system, the same result occurs as for an "X0.001" command.

(2) Decimal point input II

The least significant digit of command data matches the command unit.

(Example) When "X1" is commanded in 1 μ m system, the same result occurs as for an "X1." command.

[Addresses used, validity of decimal point commands]

Address	Decimal point command	Usage	Remarks
A	Valid	Coordinate position data	
	Invalid	Revolving table	
	Invalid	Miscellaneous function codes	
	Valid	Angle data	
	Invalid	Data settings, axis numbers (G10)	
	Valid	Spindle synchronization: Acceleration/deceleration time constant	
	Invalid	Program No.	
	Invalid	R-Navi data input by program: Coordinate axis selection	
B	Valid	Coordinate position data	
	Invalid	Revolving table	
	Invalid	Miscellaneous function codes	
	Invalid	Sub part system I: Identification No.	
	Invalid	R-Navi data input by program: Coordinate axis direction setting	
C	Valid	Coordinate position data	
	Invalid	Revolving table	
	Invalid	Miscellaneous function codes	
	Valid	Corner chamfering amount	,C
	Invalid	Target C axis in C axis mode for spindle position control (spindle/C axis control)	
	Invalid	Tool shape input by program: Tool color	
	Invalid	R-Navi data input by program: Basic coordinate system	
D	Invalid	R-Navi data input by program: Coordinate axis direction setting	
	Invalid	Compensation numbers (tool position, tool radius)	
	Valid	Automatic tool length measurement: Deceleration distance d	
	Invalid	Parameter input by program: Byte type data	
	Invalid	Spindle synchronization: Designation of synchronous spindle	
	Invalid	Subprogram storing device number	,D
	Invalid	Sub part system I: Synchronous control designation	
	Valid	Droop skip value	
	Valid	Tool shape input by program: Shape data 1	
	Invalid	R-Navi data input by program: Machining registration No.	
	Invalid	Rotation direction	
Invalid	Subprogram device No.	,D	
E	Valid	Inch thread: number of ridges, precision thread: lead	
	Invalid	R-Navi data input by program: Coordinate axis direction setting	
	Valid	Synchronous tap: Cutting feedrate (Number of screw threads)	

5 Position Commands

Address	Decimal point command	Usage	Remarks
F	Valid	Cutting feedrate, automatic tool length measurement speed	
	Valid	Thread lead	
	Valid	Number of Z axis pitch in synchronous tap	
	Valid	Rapid traverse rate	,F
	Invalid	R-Navi data input by program: Workpiece shape	
	Invalid	R-Navi data input by program: Coordinate axis direction setting	
	Valid	Involute interpolation: Feedrate (in involute curve tangent direction)	
G	Valid	Preparatory function code	
H	Invalid	Tool length compensation No.	
	Invalid	Sequence Nos. in subprograms	
	Invalid	Parameter input by program: Bit type data	
	Invalid	Reference spindle selection	
	Invalid	Sub part system I: Reset type	
	Valid	Tool shape input by program: Shape data 2	
	Invalid	R-Navi data input by program: Coordinate axis direction setting	
	Invalid	Tool position compensation: Compensation No.	
I	Valid	Coordinates for arc center and center of figure rotation	
	Valid	Tool radius compensation vector components	
	Valid	Hole pitch in the special fixed cycle	
	Valid	Circle radius of cut circle (increase amount)	
	Valid	G0/G1 in-position width, Hole drilling cycle: G0 in-position width	,I
	Valid	Stroke check before travel: Lower limit coordinates	
	Valid	Tool shape input by program: Shape data 3	
	Valid	R-Navi data input by program: Workpiece shift	
	Invalid	R-Navi data input by program: Coordinate axis direction setting	
	Valid	Arc radius and approach direction	
	Invalid	Allowable fluctuation range of spindle speed	
	Valid	Involute interpolation: Distance to the center of base circle	
	Valid	Inclined surface machining: Rotation angle about the X axis (roll angle)	
J	Invalid	Coordinates for arc center and center of figure rotation	
	Valid	Tool radius compensation vector components	
	Valid	Special fixed cycle's hole pitch or angle	
	Valid	G0/G1 in-position width, Hole drilling cycle: G1 in-position width	,J
	Valid	Stroke check before travel: Lower limit coordinates	
	Valid	Tool shape input by program: Shape data 4	
	Valid	R-Navi data input by program: Workpiece shift	
	Valid	Arc radius and approach direction	
	Valid	Involute interpolation: Distance to the center of base circle	
	Valid	Inclined surface machining: Rotation angle about the Y axis (pitch angle)	
K	Valid	Coordinates for arc center and center of figure rotation	
	Valid	Tool radius compensation vector components	
	Invalid	Number of holes of the special fixed cycle	
	Invalid	Hole drilling cycle, sub part system I: Number of repetitions	
	Valid	Stroke check before travel: Lower limit coordinates	
	Valid	Spline interpolation 2: Tolerance (Linear axis)	,K
	Invalid	Tool shape input by program: Tool type	
	Valid	R-Navi data input by program: Workpiece shift	
	Valid	Involute interpolation: Distance to the center of base circle	
	Valid	Inclined surface machining: Rotation angle about the Z axis (yaw angle)	

5 Position Commands

Address	Decimal point command	Usage	Remarks
L	Invalid	Number of fixed cycle and subprogram repetitions	
	Invalid	Tool compensation data input by program/workpiece offset input: type selection	L2,L20,L10, L11,L12, L13
	Invalid	Parameter input by program: data setting selection	L70
	Invalid	Parameter input by program: 2-word type data	4 bytes
	Invalid	R-Navi data input by program: Start setting workpiece data	L110
	Invalid	R-Navi data input by program: Start setting machining surface data	L111
	Invalid	Timing synchronization number	
	Invalid	Tool life data	
M	Invalid	Miscellaneous function codes	
	Invalid	R-Navi data input by program: Coordinate axis direction designation method	
N	Invalid	Sequence No.	
	Invalid	Parameter input by program: data numbers	
O	Invalid	Program No.	
P	Invalid/Valid	Dwell time	Parameters
	Invalid	Subprogram program call: program No.	
	Invalid/Valid	Dwell at tap cycle hole base	Parameters
	Invalid	Number of holes of the special fixed cycle	
	Invalid	Amount of helical pitch	
	Valid	Thread milling cycle: Pitch amount	
	Invalid	Offset number (G10)	
	Invalid	Constant surface speed control axis number	
	Invalid	Parameter input by program: Broad classification number	
	Invalid	Tool compensation data input by program/workpiece offset input: Compensation No.	
	Invalid	Tool shape input by program: Data numbers	
	Invalid	Multi-step skip function 2 signal command	
	Invalid	Subprogram return destination sequence No.	
	Invalid	2nd, 3rd, 4th reference position return number	
	Valid	Scaling factor	
	Invalid	High-speed mode type	
	Invalid	High-accuracy control mode: Start/End	
	Invalid	Extended workpiece coordinate system No, external workpiece coordinate system offset compensation No.	
	Invalid	Tool life data: Group No.	
	Invalid	Machining purpose	
	Invalid	Sub part system I: Start sequence No.	
	Invalid	R-Navi data input by program: Machining surface registration	
	Invalid	R-Navi data input by program: Coordinate axis direction axis designation	
	Valid	Spindle speed fluctuation detection: Start delay time	
Invalid	Interactive cycle insertion: Cycle information identification number		

5 Position Commands

Address	Decimal point command	Usage	Remarks
Q	Valid	Cut amount of deep hole drill cycle	
	Valid	Shift amount of back boring	
	Valid	Shift amount of fine boring	
	Invalid	Minimum spindle clamp speed	
	Valid	Thread cutting start shift angle	
	Invalid	Tool life data management method	
	Invalid	Machining condition	
	Invalid	Sub part system I: End sequence No.	
	Invalid	Droop skip value	
	Invalid	R-Navi data input by program: Workpiece registration No.	
	Valid	Spindle up-to-speed detection width	
	Valid	Thread milling cycle: Dwell time	
	Invalid	Inclined surface machining: Rotation order	
	Invalid	Machining interruption: Sequence No. [C80]	,Q
R	Valid	R-point in the fixed cycle	
	Valid	R-specified arc radius	
	Valid	Corner R arc radius	,R
	Valid	Offset amount (G10)	
	Invalid	Synchronous tap/asynchronous tap changeover	,R
	Valid	Synchronous tap: Designation of R point position (absolute or incremental value)	
	Valid	Synchronized spindle phase shift amount	
	Valid	Automatic tool length measurement: deceleration distance r	
	Valid	Rotation angle	
	Invalid	Skip acceleration/deceleration time constant	
	Valid	Spline interpolation 2: Tolerance (Rotary axis)	,R
	Valid	Tool compensation data input by program/workpiece offset input: Compensation amount	
	Invalid	R-Navi data input by program: Marked point No.	
	Valid	Involute interpolation: Radius of base circle	
	Invalid	Allowable fluctuation rate of spindle speed	
S	Invalid	Spindle function codes	
	Invalid	Maximum spindle clamp speed	
	Invalid	Constant surface speed control or constant surface speed cancel: Surface speed	
	Invalid	Parameter input by program: word type data	2 bytes
	Valid	Synchronous tap: Designation of spindle rotation speed at the return	
	Invalid	Spindle designation	
T	Invalid	Tool function codes	
U	Valid	Coordinate position data	
V	Valid	Coordinate position data	
W	Valid	Coordinate position data	
X	Valid	Coordinate position data	
	Valid	Dwell time	
	Valid	R-Navi data input by program: Workpiece size	
	Valid	R-Navi data input by program/Inclined surface machining: Feature coordinate zero point	

Address	Decimal point command	Usage	Remarks
Y	Valid	Coordinate position data	
	Valid	R-Navi data input by program: Workpiece size	
	Valid	R-Navi data input by program/Inclined surface machining: Feature coordinate zero point	
Z	Valid	Coordinate position data	
	Valid	R-Navi data input by program: Workpiece size	
	Valid	R-Navi data input by program / Inclined surface machining: Feature coordinate zero point	

Note

- (1) Decimal points are all valid in user macro arguments.



Program example

- (1) Program example of decimal point valid address

Program example	Decimal point command 1		Decimal point command 2 1 = 1 mm
	When 1 = 1 μ m	When 1 = 10 μ m	
G00 X123.45 (decimal points are all mm points)	X123.450 mm	X123.450 mm	X123.450 mm
G00 X12345	X12.345 mm (last digit is 1 μ m unit)	X123.450 mm	X12345.000 mm
#111=123 #112=5.55 X#111 Y#112	X123.000 mm Y5.550 mm	X123.000 mm Y5.550 mm	X123.000 mm Y5.550 mm
#113=#111+#112 (addition)	#113=128.550	#113=128.550	#113=128.550
#114=#111-#112 (subtraction)	#114=117.450	#114=117.450	#114=117.450
#115=#111*#112 (multiplication)	#115=682.650	#115=682.650	#115=682.650
#116=#111/#112 #117=#112/#111 (division)	#116=22.162 #117=0.045	#116=22.162 #117=0.045	#116=22.162 #117=0.045



Precautions

- (1) If an arithmetic operator is inserted, the data will be handled as data with a decimal point.
(Example 1) G00 X123+0 ;
This is the X axis 123mm command. It will not be 123 μ m.

Interpolation Functions

6.1 Positioning (Rapid Traverse); G00



Function and purpose

This command is accompanied by coordinate words and performs high-speed positioning of a tool, from the present point (start point) to the end point specified by the coordinate words.



Command format

Positioning (Rapid Traverse)

```
G00 X__ Y__ Z__ α__ ,I__ ,F__;
```

X, Y, Z, α	Coordinate values. (α is the additional axis.) An absolute position or incremental position is indicated based on the state of G90/G91 at that time.
,I	In-position width. (1 to 999999) This address is valid only in the commanded block. A block that does not contain this address will follow the parameter "#1193 inpos" settings. For details, refer to "7.12 Deceleration Check".
,F	Specifies the rapid traverse rate of the movement initiated by a G00 command, the movement in the G00 mode, and the movement during the fixed cycle for drilling. The range is equal to the range of the feed per minute F command (mm/min, inch/min) in the G01 mode. Switching inch/mm is invalid for rotary axes. For details, refer to "7.1.2 G00 Feedrate Command (,F Command)".

The command addresses are valid for all additional axes.



Detailed description

- (1) The rapid traverse speed varies depending on the MTB specifications (parameter "#2001 rapid").
When the "G00 feedrate designation (,F command)" function is enabled and an ",F" command is included in the same block as for the G00 command, positioning is carried out at the feedrate specified by the ",F" command. If this function is invalid or an ",F" command is not designated, positioning is carried out at the feedrate specified in parameter "#2001 rapid".
- (2) G00 command belongs to the 01 group and is modal. When G00 command is successively issued, the following blocks can be specified only by the coordinate words.
- (3) In the G00 mode, acceleration and deceleration are always carried out at the start point and end point of the block. Before advancing to the next block, a commanded deceleration or an in-position check is conducted at the end point to confirm that the movement is completed for all the moving axes in each part system.
- (4) G functions (G72 to G89) in the 09 group are canceled (G80) by the G00 command.

CAUTION

The commands with "no value after G" will be handled as "G00".

Tool path

Whether the tool moves along a linear or non-linear path varies depending on the MTB specifications (parameter "#1086 G0Intp").

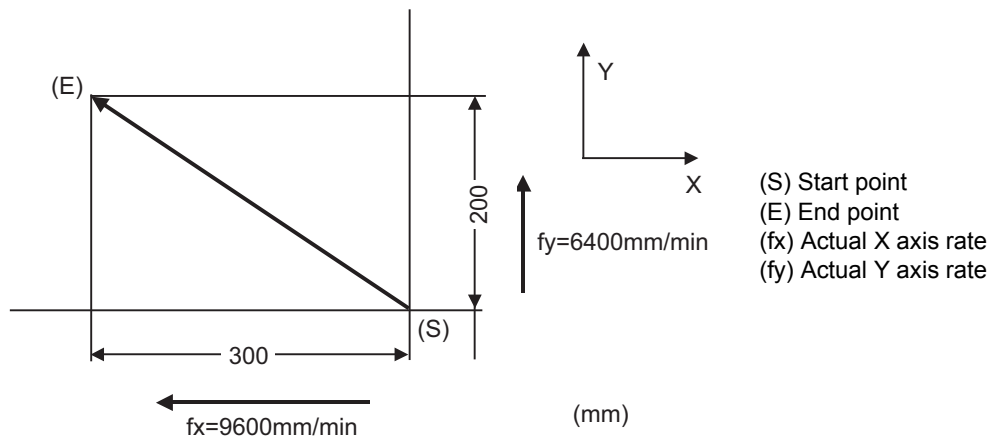
The positioning time does not change according to the path.

(1) Linear path (When parameter "#1086 G0Intp" is set to "0")

In the positioning process, a tool follows the shortest path that connects the start point and the end point. The positioning speed is automatically calculated so that the shortest distribution time is obtained in order that the commanded speeds for each axis do not exceed the rapid traverse rate.

When, for instance, the X axis and Y axis rapid traverse rates are both 9600 mm/min and when programmed as follows, the tool will follow the path shown in the figure below.

G91 G00 X-300000 Y200000; (With an input setting unit of 0.001 mm)

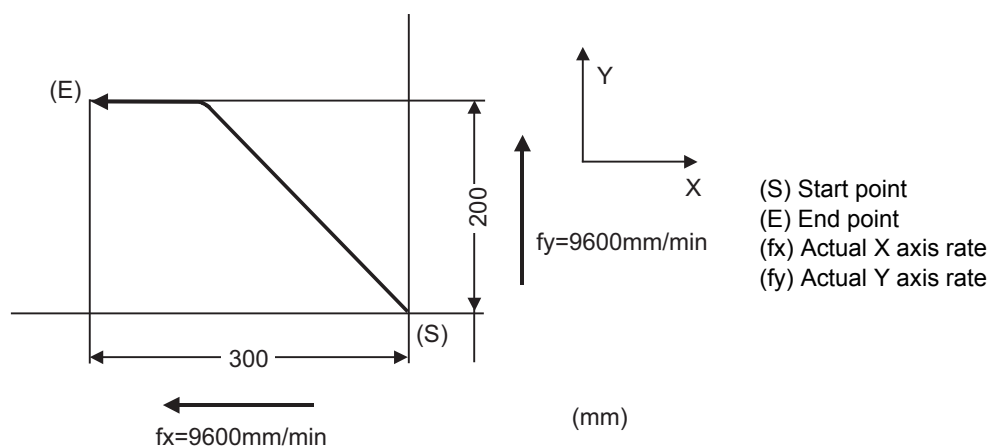


(2) Non-linear path (When parameter "#1086 G0Intp" is set to "1")

In positioning, the tool will move along the path from the start point to the end point at the rapid traverse rate of each axis.

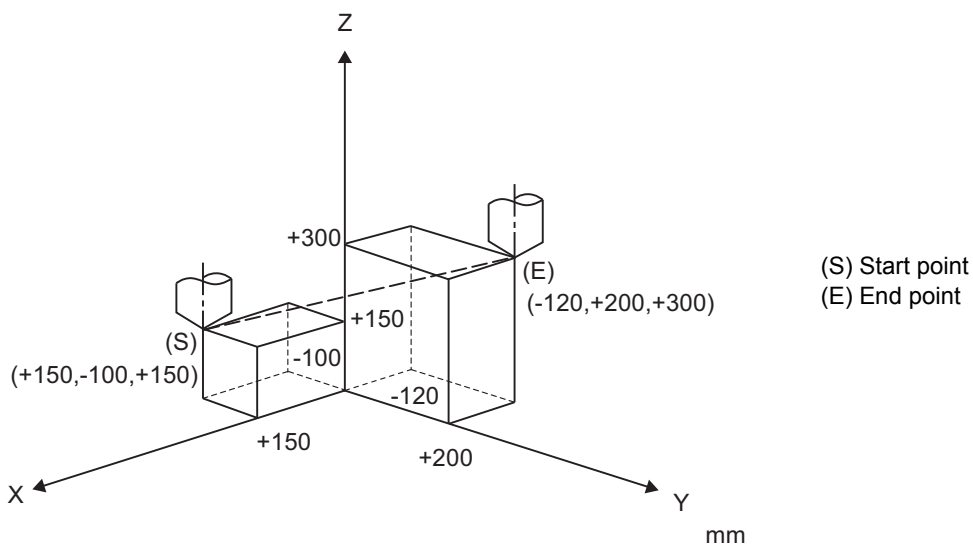
When, for instance, the X axis and Y axis rapid traverse rates are both 9600 mm/min and when programmed as follows, the tool will follow the path shown in the figure below.

G91 G00 X-300000 Y200000; (With an input setting unit of 0.001 mm)





Program example



```
G91 G00 X-270. Y300. Z150. ;
```



Precautions for deceleration check

There are three methods of carrying out a deceleration check: the command deceleration check method, the smoothing check method, and the in-position check method. The method used for rapid traverse or cutting feed varies depends on the MTB specification (combination of parameters "#1306 InpsTyp", "#1389 G1SmthChk", "#1223 aux07/bit1", and "#1193 inpos").

A block with an in-position width command performs an in-position check with a temporarily changed in-position width. (Programmable in-position width command)

A block without an in-position width command is processed using the deceleration check method based on the MTB specifications (parameter "#1193 inpos").

During cutting feed and when the error detection is ON, the in-position check is forcibly carried out.

Rapid traverse (G00)		#1193 inpos	
		0	1
,I command	No	Commanded deceleration method (Commanded deceleration check that varies according to the type of acceleration/deceleration, set in "#2003 smgst" bit3-0)	In-position check method (In-position check by "#2077 G0inps", "#2224 SV024")
	Yes	In-position check method (In-position check by ",I", "#2077 G0inps", "#2224 SV024")	
Cutting feedrate (G01)		#1193 inpos	
		0	1
,I command	No	Commanded deceleration method (Commanded deceleration check that varies according to the type of acceleration/deceleration, set in "#2003 smgst" bit7-4)	In-position check method (In-position check by "#2078 G1inps", "#2224 SV024")
	Yes	In-position check method (In-position check by ",I", "#2078 G1inps", "#2224 SV024")	

Refer to "7.12 Deceleration Check" for the deceleration check method.

6.2 Linear Interpolation; G01



Function and purpose

This command is accompanied by coordinate words and a feedrate command. It makes the tool move (interpolate) linearly from its current position to the end point specified by the coordinate words at the speed specified by address F. In this case, the feedrate specified by address F always acts as a linear speed in the tool nose center advance direction.



Command format

Linear interpolation

```
G01 X_ Y_ Z_ α_ F_ ,I_ ;
```

X,Y,Z,α	Coordinate values. (α is the additional axis.) An absolute position or incremental position is indicated based on the state of G90/G91 at that time.
F	Feedrate (mm/min or °/min)
,I	In-position width. (1 to 999999) This address is valid only in the commanded block. A block that does not contain this address will follow the parameter "#1193 inpos" settings.



Detailed description

- (1) G01 command is a modal command in the 01 group. When G01 command is successively issued, the following blocks can be specified only by the coordinate words. If there is no command, a program error (P62) will occur.
- (2) The feedrate for a rotary axis is commanded by °/min (decimal point position unit). (F300=300°/min)
- (3) The G functions (G72 to G89) in the 09 group are cancelled (G80) by the G01 command.

Programmable in-position width command for linear interpolation

This command commands the in-position width for the linear interpolation command from the machining program.

```
G01 X_ Y_ Z_ F_ ,I_ ;
```

X,Y,Z	Linear interpolation coordinate value of each axis
F	Feedrate
,I	In-position width

The commanded in-position width is valid in the linear interpolation command only when carrying out deceleration check.

- When the error detection switch is ON.
- When G09 (exact stop check) is commanded in the same block.
- When G61 (exact stop check mode) is selected.

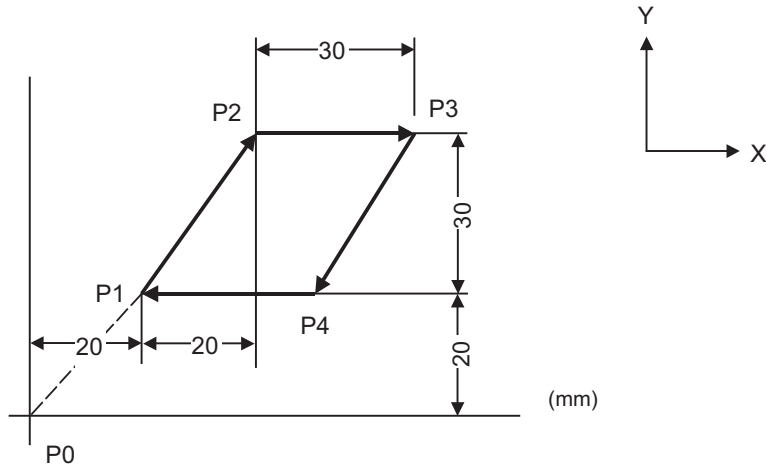
Note

- (1) Refer to section "6.1 Positioning (Rapid Traverse); G00" for details on the in-position check operation.



Program example

(Example) Cutting in the sequence of P1 -> P2 -> P3 -> P4 -> P1 at 300mm/min feedrate.
 However, P0 -> P1 is for tool positioning.



G91 G00 X20. Y20. ;	P0 -> P1
G01 X20. Y30. F300 ;	P1 -> P2
X30. ;	P2 -> P3
X-20. Y-30. ;	P3 -> P4
X-30. ;	P4 -> P1

6.3 Circular Interpolation; G02, G03



Function and purpose

These commands serve to move the tool along a circular.



Command format

Circular interpolation : Clockwise (CW)

```
G02 X__Y__I__J__F__;
```

Circular interpolation : Counterclockwise (CCW)

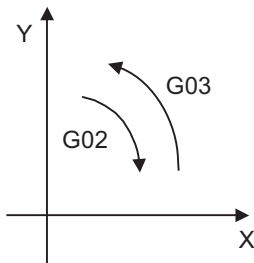
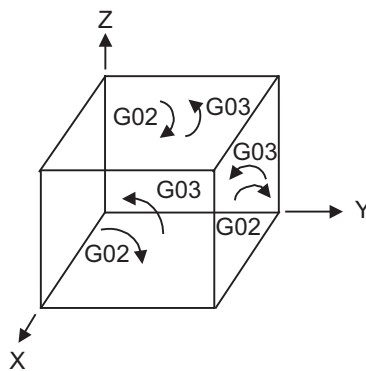
```
G03 X__Y__I__J__F__;
```

X,Y	Arc end point coordinates
I,J	Arc center coordinates
F	Feedrate

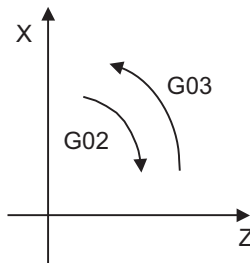


Detailed description

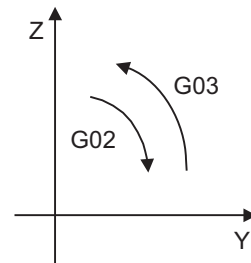
- (1) For the arc command, the arc end point coordinates are assigned with addresses X, Y (or Z, or parallel axis X, Y, Z), and the arc center coordinate value is assigned with addresses I, J (or K).
 Either an absolute value or incremental value can be used for the arc end point coordinate value command, but the arc center coordinate value must always be commanded with an incremental value from the start point.
 The arc center coordinate must be commanded in the input setting unit. Caution is required for the arc command of an axis for which the input command unit differs. Command with a decimal point to avoid confusion.
- (2) G02 (G03) is a modal command of the 01 group. When G02 (G03) command is issued continuously, the next block and after can be commanded with only coordinate words.
 The circular rotation direction is distinguished by G02 and G03.
 G02 CW (Clockwise)
 G03 CCW (Counterclockwise)
- (3) Select the XY plane, ZX plane or YZ plane to draw an arc on it, using the plane selection G code.



G17(X-Y) plane



G18(Z-X) plane



G19(Y-Z) plane

- (4) An arc which extends for more than one quadrant can be executed with a single block command.
- (5) The following information is needed for circular interpolation.

(a) Plane selection	Is there an arc parallel to one of the XY, ZX or YZ planes?
(b) Rotation direction	Clockwise (G02) or counterclockwise (G03)
(c) Arc end point coordinates	Set by addresses X, Y, Z.
(d) Arc center coordinates	Set by addresses I, J, K. (incremental value commands)
(e) Feedrate	Set by address F

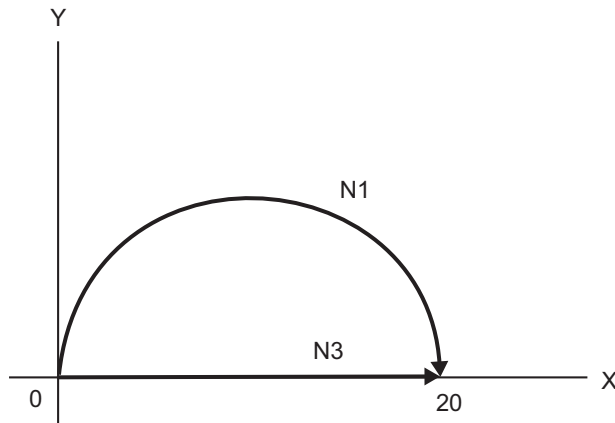
- (6) If an R specification and I, K specification are given at the same time in the same block, the circular command with the R specification takes precedence.

Change into linear interpolation command

Program error (P33) will occur in general use when the center and radius are not designated at circular command. Depending on the MTB specifications, the linear interpolation can be carried out up to the end point coordinates only in a block with no center coordinates or radius specified (parameter "#11029 Arc to G1 no Cent"). However, a modal is the circular modal.

This function is not applied to a circular command by a geometric function.

(Example) #11029 = "1"



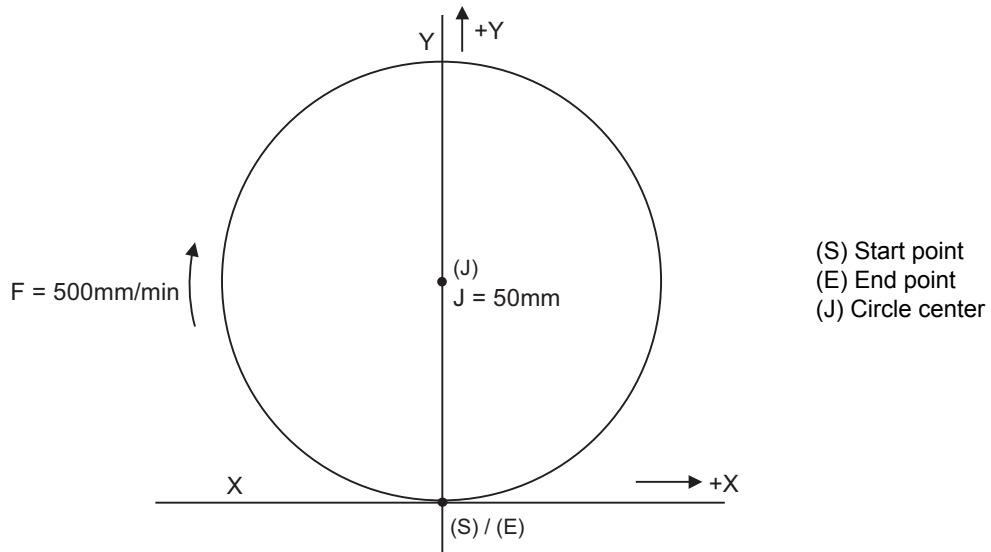
```
G90 X0 Y0 ;
N1 G02 X20. I10. F500 ; ... (a)
N2 G00 X0 ;
N3 G02 X20. F500 ; ... (b)
M02 ;
```

- (a) The circular interpolation (G02) is executed because there is a center command.
 (b) The linear interpolation (G01) is executed because there is no center and radius command.



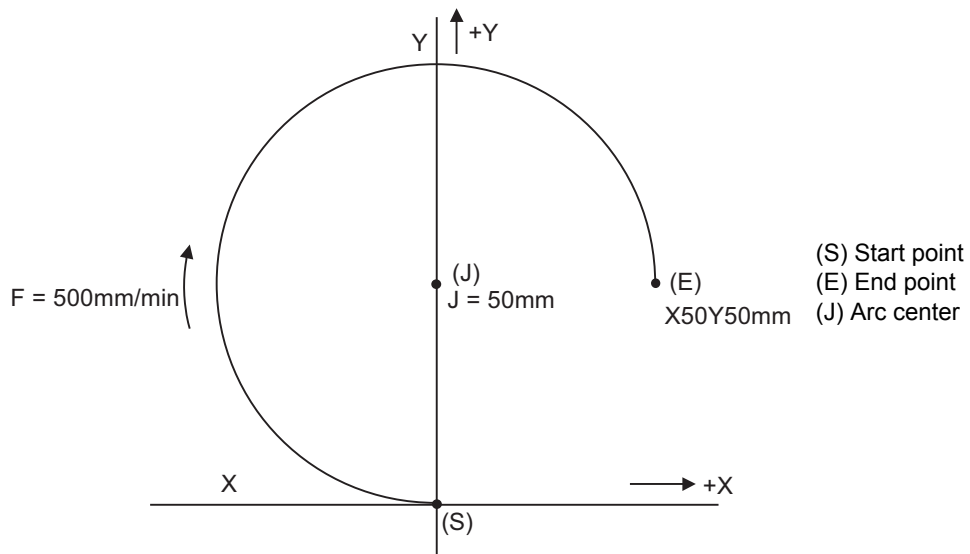
Program example

(Example 1)



G02 J50. F500;	Circle command
----------------	----------------

(Example 2)



G91 G02 X50. Y50. J50. F500;	3/4 command
------------------------------	-------------

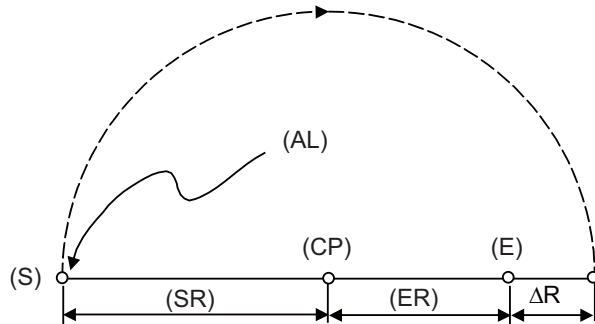


Precautions

- (1) The terms "clockwise" (G02) and "counterclockwise" (G03) used for circular operations are defined as a case where, in a right-hand coordinate system, the negative direction is viewed from the positive direction of the coordinate axis which is at right angles to the plane in question.
- (2) If all the end point coordinates are omitted or the end point is at the same position as the start point, commanding the center using I, J and K is the same as commanding a 360°arc (perfect circle).
- (3) The following occurs when the start and end point radius do not match in a circular command :
 - (a) Program error (P70) results at the circular start point when error ΔR is greater than parameter "#1084 RadErr".

(G91) G02 X9.899 I5. ;

#1084 RadErr parameter value 0.100
 Start point radius=5.000
 End point radius=4.899
 Error ΔR =0.101

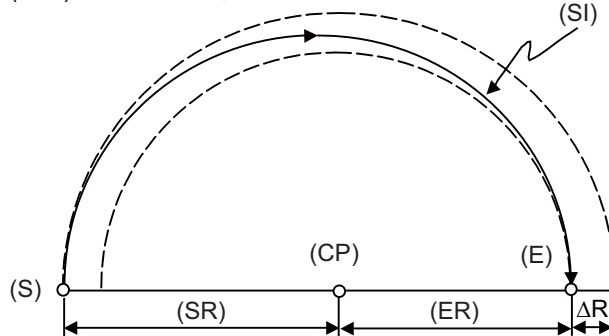


(S) Start point
 (CP) Center point
 (E) End point
 (SR) Start point radius
 (ER) End point radius
 (AL) Alarm stop

- (b) Spiral interpolation in the direction of the commanded end point will be conducted when error ΔR is less than the parameter value.

(G91) G02 X9.9 I5. ;

#1084 RadErr parameter value 0.100
 Start point radius=5.000
 End point radius=4.900
 Error ΔR =0.100



(S) Start point
 (CP) Center point
 (E) End point
 (SR) Start point radius
 (ER) End point radius
 (SI) Spiral interpolation

Also, if "#1084 RadErr" is set to "0", "0.1" is assumed to set.

6 Interpolation Functions

(c) If the start point radius differs from the end point radius but if the start point angle does not differ from the end point angle, the linear interpolation or spiral interpolation is selected depending on the MTB specifications (parameter "#1278 ext14/bit7").

#1278 ext14/bit = 0 Linear interpolation	#1278 ext14/bit7 = 1 Spiral interpolation
G90 G00 X10. Y0.; G02 X10.01 Y0. I-10.01;	G90 G00 X10. Y0.; G02 X10.01 Y0. I-10.01;
(CP) Center point (S) Start point (E) End point	(SR) Start point radius (ER) End point radius (LI) Linear interpolation (SI) Spiral interpolation

6.4 R Specification Circular Interpolation; G02, G03



Function and purpose

Along with the conventional circular interpolation commands based on the circular center coordinate (I, J, K) designation, these commands can also be issued by directly designating the circular radius R.



Command format

R specification circular interpolation Clockwise (CW)

```
G02 X__Y__R__F__;
```

R specification circular interpolation Counterclockwise (CCW)

```
G03 X__Y__R__F__;
```

X	X axis end point coordinate
Y	Y axis end point coordinate
R	Circular radius
F	Feedrate

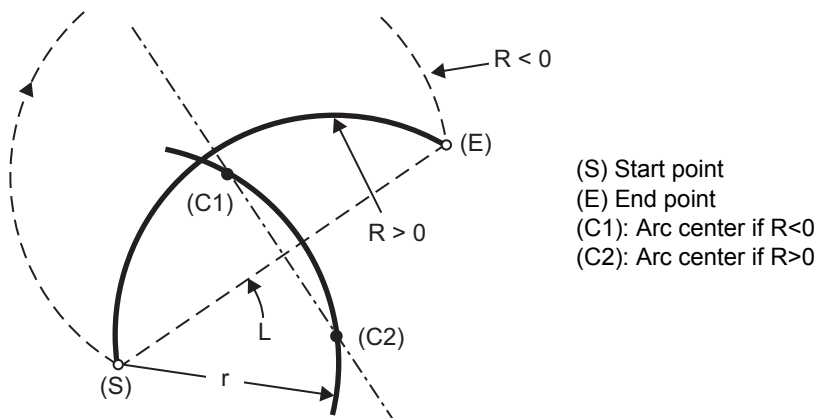
The arc radius must be commanded in the input setting unit. Caution is required for the arc command of an axis for which the input command unit (#1015 cunit) differs. Command with a decimal point to avoid confusion. A maximum of 6 digits before decimal point can be specified for the radius.



Detailed description

The circular center is on the bisector line which is perpendicular to the line connecting the start and end points of the circular. The point, where the circular with the specified radius whose start point is the center intersects the perpendicular bisector line, serves as the center coordinates of the circular command.

If the R sign of the commanded program is plus, the circular is smaller than a semicircular; if it is minus, the circular is larger than a semicircular.



6 Interpolation Functions

The following condition must be met with an R-specified arc interpolation command:

$$\frac{L}{2 \cdot r} \leq 1 \quad \text{When } (L/2 - r) > (\text{parameter} : \#1084 \text{ RadErr}), \text{ an alarm will occur.}$$

Where L is the line from the start point to the end point. If an R specification and I, J, (K) specification are given at the same time in the same block, the circular command with the R specification takes precedence. In the case of a full-circle command (where the start and end points coincide), an R specification circular command will be completed immediately even if it is issued and no operation will be executed. An I, J, (K) specification circular command should therefore be used in such a case.

The plane selection command is the same as the I, J, or K specification circular command.

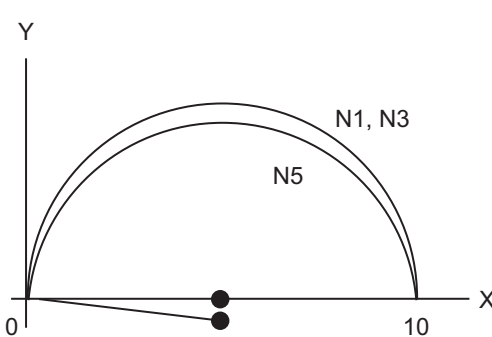
Circular center coordinate compensation

When "the error margin between the segment connecting the start and end points" and "the commanded radius × 2" is less than the setting value because the required semicircle is not obtained by calculation error in R specification circular interpolation, "the midpoint of the segment connecting the start and end points" is compensated for as the circular center.

The setting value depends on the MTB specifications (parameter "#11028 Tolerance Arc Cent" (Tolerable correction value of arc center error)).

(Example) #11028 = "0.000 (mm)"

Setting value	Tolerance value
Setting value < 0	0 (Center error will not be interpolated)
Setting value = 0	2×minimum setting increment
Setting value > 0	Setting value



```
G90 X0 Y0 ;
N1 G02 X10. R5.000;
N2 G00 X0;
N3 G02 X10. R5.001; ... (a)
N4 G00 X0;
N5 G02 X10. R5.002; ... (b)
N6 G00 X0;
M02 ;
```

- (a) Compensate the center coordinate: Same as N1 path
- (b) Do not compensate the center coordinate: Inside path a little than N1

Calculation error margin compensation allowance value: 0.002 mm
 Segment connecting the start and end points: 10.000
 N3: Radius × 2 = 10.002 "Error 0.002 -> Compensate"
 N5: Radius × 2 = 10.004 "Error 0.004 -> Do not compensate"



Program example

(Example 1)

G02 Xx1 Yy1 Rr1 Ff1 ;	XY plane R-specified arc
-----------------------	--------------------------

(Example 2)

G03 Zz1 Xx1 Rr1 Ff1 ;	R specification circular on Z-X plane
-----------------------	---------------------------------------

(Example 3)

G02 Xx1 Yy1 Ii1 Jj1 Rr1 Ff1 ;	XY plane R-specified arc (When the R specification and I, J, (K) specification are contained in the same block, the circular command with the R specification takes precedence.)
-------------------------------	---

(Example 4)

G17 G02 Ii1 Jj1 Rr1 Ff1 ;	XY plane This is an R-specified arc, but as this is a circle command, it will be completed immediately.
---------------------------	---



Precautions

- (1) In the case of a full-circle command (where the start and end points coincide), an R specification circular command will be completed immediately even if it is issued and no operation will be executed. An I, J, K specification circular command should therefore be used in such a case.
- (2) If an R specification and I, K specification are given at the same time in the same block, the circular command with the R specification takes precedence.

6.5 Plane Selection; G17, G18, G19



Function and purpose

The plane to which the movement of the tool during the circle interpolation (including helical cutting) and tool radius compensation command belongs is selected.

If the 3 basic axes and the parallel axes corresponding to these basic axes are entered as parameters, the commands can select the plane composed of any 2 axes which are not parallel axes. If a rotary axis is entered as a parallel axis, the commands can select the plane containing the rotary axis.

These commands are used to select following planes:

- Plane that executes circular interpolation (including helical cutting)
- Plane that executes tool radius compensation
- Used to select a plane that executes fixed cycle positioning.



Command format

G17 ; X-Y plane selection

G18 ; Z-X plane selection

G19 ; Y-Z plane selection

X, Y and Z indicate each coordinate axis or the parallel axis.



Detailed description

Parameter entry

	#1026-1028 Basic axis I, J, K	#1029-1031 Flat axis I, J, K
I	X	U
J	Y	
K	Z	V

Table 1 Examples of plane selection parameter entry

As shown in the above example, the basic axis and its parallel axis can be registered.

The basic axis can be an axis other than X, Y and Z.

Axes that are not registered are irrelevant to the plane selection.

Plane selection system

In Table 1, characters I, J, K represent the following axes.

I: Horizontal axis for the G17 plane or the vertical axis for the G18 plane

J: Vertical axis for the G17 plane or horizontal axis for the G19 plane

K: Horizontal axis for the G18 plane or vertical axis for the G19 plane

Therefore, the G17, G18 and G19 commands select the following planes.

G17: I-J plane

G18: K-I plane

G19: J-K plane

- (1) Axis addresses assigned in the same block as the plane selection (G17, G18, G19) command determine which of the basic axes or parallel axes are to be in the actual plane selected.

For the parameter entry example in Table 1

G17 X__ Y__ ; X-Y plane

G18 X__ V__ ; V-X plane

G18 U__ V__ ; V-U plane

G19 Y__ Z__ ; Y-Z plane

G19 Y__ V__ ; Y-V plane

- (2) Plane selection is not performed with blocks in which the plane selection G code (G17, G18, G19) is not assigned.

G17 X__ Y__ ; X-Y plane

Y__ Z__ ; X-Y plane (No plane change)

- (3) If the axis address is omitted in the block where the plane selection G code (G17, G18, G19) is commanded, it is assumed that the axis addresses of the 3 basic axes have been omitted.

For the parameter entry example in Table 1

G17 ; X-Y plane

G17 U__ ; U-Y plane

G18 U__ ; Z-U plane

G18 V__ ; V-X plane

G19 Y__ ; Y-Z plane

G19 V__ ; Y-V plane

- (4) When the axis addresses are commanded to the same block as the plane selection G code (G17, G18, G19), the commanded axes will travel.

- (5) The axis command that does not exist in the plane determined by the plane selection G code (G17, G18, G19) is irrelevant to the plane selection.

For the parameter entry example in Table 1, if the command is issued as below, the U-Y plane will be selected and "Z" will move regardless of the plane.

G17 U__ Z__ ;

- (6) When the basic axes or their parallel axes are duplicated and assigned in the same block as the plane selection G code (G17, G18, G19), the plane is determined in the order of basic axes, and then parallel axes.

For the parameter entry example in Table 1, if the command is issued as below, the U-Y plane will be selected and "W" will move regardless of the plane.

G17 U__ Y__ W__ ;

Note

- When the power is turned ON or when the system is reset, the plane set by the parameter "#1025 Initial plane selection" is selected.

6.6 Thread Cutting

6.6.1 Constant Lead Thread Cutting; G33



Function and purpose

The G33 command exercises feed control over the tool which is synchronized with the spindle rotation and so this makes it possible to conduct constant-lead straight thread-cutting, and tapered thread-cutting. Multiple thread screws, etc., can also be machined by designating the thread cutting angle.



Command format

Normal lead thread cutting

G33 Z_(X_ Y_ α_) F_ Q_ ;

Z (X Y α)	End point of thread cutting
F	Lead of long axis (axis which moves most) direction
Q	Thread cutting start shift angle (0.001 - 360.000°)

Precision lead thread cutting

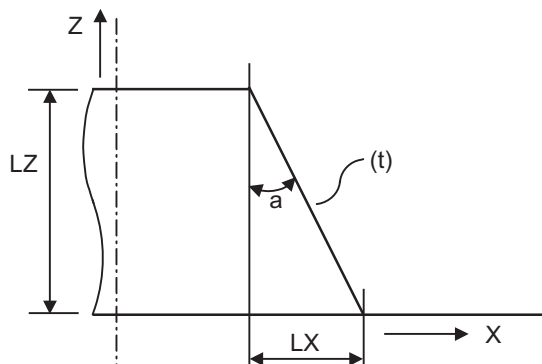
G33 Z_(X_ Y_ α_) E_ Q_ ;

Z (X Y α)	End point of thread cutting
E	Lead of long axis (axis which moves most) direction
Q	Thread cutting start shift angle (0.001 - 360.000°)



Detailed description

- (1) The E command is also used for the number of ridges in inch thread cutting, and whether the number of ridges or precision lead is to be designated can be selected by parameter setting. (Parameter "#8156 Fine thread cut E" is set to "1" for precision lead designation.)
- (2) The lead in the long axis direction is commanded for the taper thread lead.



(t) Tapered thread section

When $a < 45^\circ$, lead is in LZ direction.

When $a > 45^\circ$, lead is in LX direction.

When $a = 45^\circ$, lead can be in either LZ or LX direction.

Thread cutting metric input

Input Setting unit	B (0.001 mm)			C (0.0001 mm)		
	Command address	F (mm/rev)	E (mm/rev)	E (ridges/inch)	F (mm/rev)	E (mm/rev)
Minimum command unit	1 (=0.001) (1.=1.000)	1 (=0.0001) (1.=1.0000)	1 (=1.00) (1.=1.00)	1 (=0.0001) (1.=1.0000)	1 (=0.00001) (1.=1.00000)	1 (=1.000) (1.=1.000)
Range	0.001 to 999.999	0.0001 to 999.9999	0.03 to 999.99	0.0001 to 999.9999	0.00001 to 999.99999	0.026 to 222807.017
Input Setting unit	D (0.00001 mm)			E (0.000001 mm)		
	Command address	F (mm/rev)	E (mm/rev)	E (ridges/inch)	F (mm/rev)	E (mm/rev)
Minimum command unit	1 (=0.00001) (1.=1.00000)	1 (=0.000001) (1.=1.000000)	1 (=1.0000) (1.=1.0000)	1 (=0.000001) (1.=1.000000)	1 (=0.0000001) (1.=1.0000000)	1 (=1.00000) (1.=1.00000)
Range	0.00001 to 999.99999	0.000001 to 999.999999	0.0255 to 224580.0000	0.000001 to 999.999999	0.0000001 to 999.9999999	0.02541 to 224719.00000

Thread cutting inch input

Input Setting unit	B (0.0001inch)			C (0.00001inch)		
	Command address	F (inch/rev)	E (inch/rev)	E (ridges/inch)	F (inch/rev)	E (inch/rev)
Minimum command unit	1 (=0.0001) (1.=1.0000)	1 (=0.00001) (1.=1.00000)	1 (=1.000) (1.=1.000)	1 (=0.00001) (1.=1.00000)	1 (=0.000001) (1.=1.000000)	1 (=1.0000) (1.=1.0000)
Range	0.0001 to 39.3700	0.00001 to 39.37007	0.025 to 9999.999	0.00001 to 39.37007	0.000001 to 39.370078	0.0255 to 9999.9999
Input Setting unit	D (0.000001 inch)			E (0.0000001 inch)		
	Command address	F (inch/rev)	E (inch/rev)	E (ridges/inch)	F (inch/rev)	E (inch/rev)
Minimum command unit	1 (=0.000001) (1.=1.000000)	1 (=0.0000001) (1.=1.0000000)	1 (=1.00000) (1.=1.00000)	1 (=0.0000001) (1.=1.0000000)	1 (=0.00000001) (1.=1.00000000)	1 (=1.000000) (1.=1.000000)
Range	0.000001 to 39.370078	0.0000001 to 39.3700787	0.02541 to 9999.99999	0.0000001 to 39.3700787	0.00000001 to 39.37007873	0.025401 to 9999.999999

<Note>

•It is not possible to assign a lead where the feedrate as converted into feed per minute exceeds the maximum cutting feedrate.

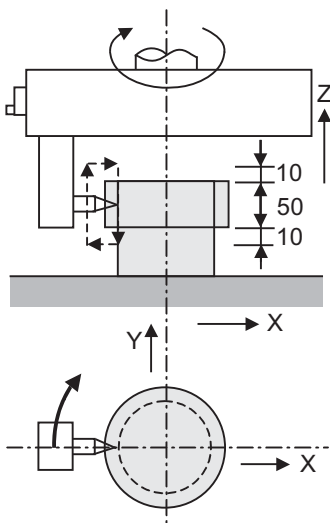
- (3) The constant surface speed control function should not be used for taper thread cutting commands or scrolled thread cutting commands.
- (4) The spindle rotation speed should be kept constant throughout from the rough cutting until the finishing.
- (5) If the feed hold function is employed during thread cutting to stop the feed, the thread ridges will lose their shape. For this reason, feed hold does not function during thread cutting.

Note that this is valid from the time the thread cutting command is executed to the time the axis moves. If the feed hold switch is pressed during thread cutting, block stop will occur at the end point of the block following the block in which thread cutting is completed (no longer G33 mode).

- (6) The converted cutting feedrate is compared with the cutting feed clamp rate when thread cutting starts, and if it is found to exceed the clamp rate, an operation error will occur.
- (7) In order to protect the lead during thread cutting, a cutting feedrate which has been converted may sometimes exceed the cutting feed clamp rate.
- (8) An illegal lead is normally produced at the start of the thread and at the end of the cutting because of servo system delay and other such factors.
Therefore, it is necessary to command a thread length which is determined by adding the illegal lead lengths to the required thread length.
- (9) The spindle rotation speed is subject to the following restriction:
 $1 \leq R \leq \text{Maximum feedrate/Thread lead}$
Where $R \leq \text{Tolerable speed of encoder (r/min)}$
R: Spindle rotation speed (r/min)
Thread lead = mm or inches
Maximum feedrate= mm/min or inch/min (this is subject to the restrictions imposed by the machine specifications.)
- (10) A program error (P93) may occur when the result of the expression (9) is $R < 1$ because the thread lead is very large to the highest cutting feedrate.
- (11) Dry run is valid for thread cutting but the feedrate based on dry run is not synchronized with the spindle rotation. The dry run signal is checked at the start of thread cutting and any switching during thread cutting is ignored.
- (12) Synchronous feed applies for the thread cutting commands even with an asynchronous feed command (G94).
- (13) Spindle override and cutting feed override are invalid and the speeds are fixed to 100% during thread cutting.
- (14) When a thread cutting is commanded during tool radius compensation, the compensation is temporarily canceled and the thread cutting is executed.
- (15) When the mode is switched to another automatic mode while G33 is executed, the following block which does not contain a thread cutting command is first executed and then the automatic operation stops.
- (16) When the mode is switched to the manual mode while G33 is executed, the following block which does not contain a thread cutting command is first executed and then the automatic operation stops. In the case of a single block, the following block which does not contain a thread cutting command (G33 mode is canceled) is first executed and then the automatic operation stops. Note that automatic operation is stopped until the G33 command axis starts moving.
- (17) The thread cutting command waits for the single rotation synchronization signal of the rotary encoder and starts movement.
Make sure to carry out timing synchronization operation between part systems before issuing a thread cutting command with multiple part systems. For example, when using the 1-spindle specifications with two part systems, if one part system issues a thread cutting command during ongoing thread cutting by another part system, the movement will start without waiting for the rotary encoder single rotation synchronization signal causing an illegal operation.
- (18) The thread cutting start shift angle is not modal. If there is no Q command with G33, this will be handled as "Q0".
- (19) The automatic handle interrupt/interruption is valid during thread cutting.
- (20) If a value exceeding 360.000 is command in G33 Q, a program error (P35) will occur.
- (21) G33 cuts one row with one cycle. To cut two rows, change the Q value, and issue the same command.



Program example



N110 G90 G00 X-200. Y-200. S50 M3 ;	The spindle center is positioned to the workpiece center, and the spindle rotates in the forward direction.
N111 Z110 ;	
N112 G33; Z40 F6.0;	The first thread cutting is executed. Thread lead = 6.0mm
N113 M19 ;	Spindle orientation is executed with the M19 command.
N114 G00 X-210.;	The tool is evaded in the X axis direction.
N115 Z110. M0 ;	The tool rises to the top of the workpiece, and the program stops with M00. Adjust the tool if required.
N116 X-200. ; M3 ;	Preparation for second thread cutting is done.
N117 G04 X5.0 ;	Command dwell to stabilize the spindle rotation if necessary.
N118 G33 Z40.;	The second thread cutting is executed.

6.6.2 Inch Thread Cutting; G33

**Function and purpose**

If the number of ridges per inch in the long axis direction is assigned in the G33 command, the feed of the tool synchronized with the spindle rotation will be controlled, which means that constant-lead straight thread-cutting and tapered thread-cutting can be performed.

**Command format**

G33 Z_ (X_ Y_ α_) E_ Q_ ; ... Inch thread cutting
--

Z (X Y α)	End point of thread cutting
E	Number of ridges per inch in direction of long axis (axis which moves most) (decimal point command can also be assigned)
Q	Thread cutting start shift angle, 0 to 360°

**Detailed description**

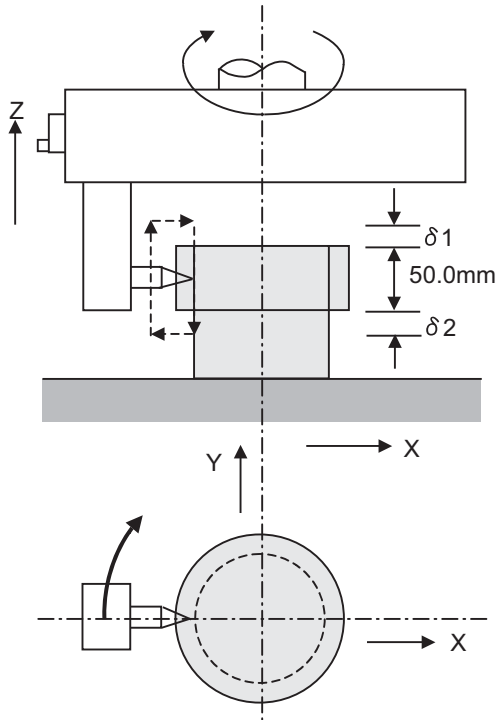
- (1)The number of ridges in the long axis direction is assigned as the number of ridges per inch.
- (2)The E code is also used to assign the precision lead length, and whether the number of ridges or precision lead length is to be designated can be selected by parameter setting. (The number of ridges is designated by setting the parameter "#8156 Fine thread cut E" to "0".)
- (3)The E command value should be set within the lead value range when converted to lead.
- (4)See Section "Constant lead thread cutting" for other details.



Program example

Thread lead 3 threads/inch (= 8.46666 ...)

When programmed with $\delta 1 = 10$ mm, $\delta 2 = 10$ mm using metric input



N210 G90 G00 X-200. Y-200. S50 M3 ;	
N211 Z110. ;	
N212 G91 G33 Z-70. E3.0 ;	(First thread cutting)
N213 M19;	
N214 G90 G00 X-210. ;	
N215 Z110. M0 ;	
N216 X-200. ; M3 ;	
N217 G04 X2.0 ;	
N218 G91 G33 Z-70. ;	(Second thread cutting)

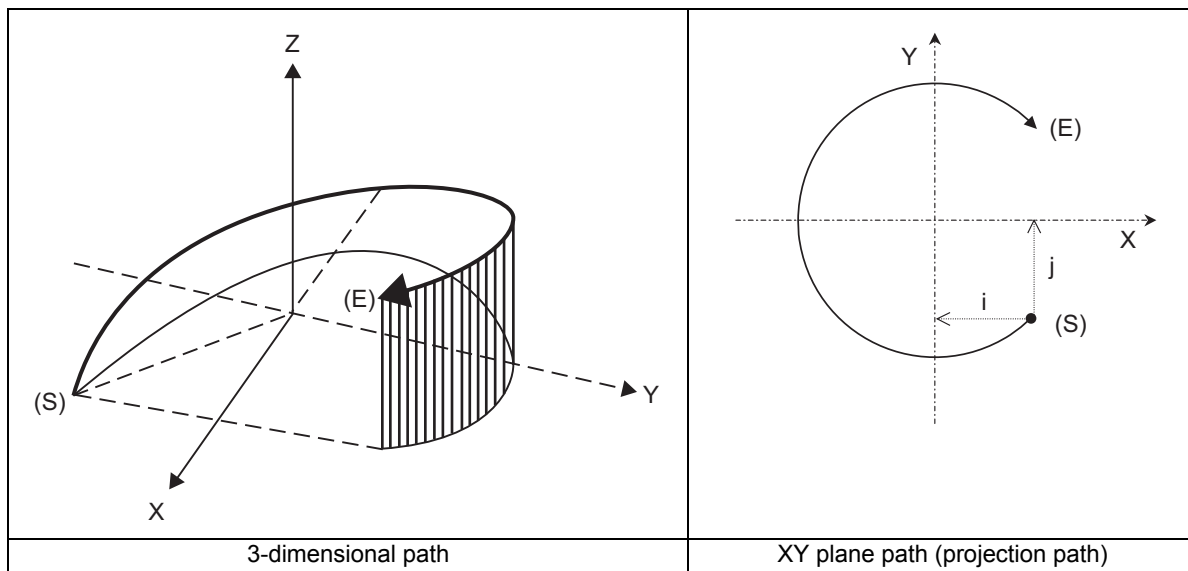
6.7 Helical Interpolation ; G17,G18,G19 and G02,G03



Function and purpose

When this interpolation is performed with 3 orthogonal axes, the tool will travel helically when circular interpolation is executed for any 2 axes and, at the same time, when another 1 axis is synchronized with the rotation of the circular and linear interpolation is executed synchronously with the rotation of the circular arc.

Large diameter threads and 3-dimensional cams can be machined.



(S) Start point

(E) End point

———— Program command path

———— XY plane projection path in command program



Command format

Helical interpolation command (Specify arc center)

G17/G18/G19 G02/G03 X__ Y__ Z__ I__ J__ P__ F__ ;

Helical interpolation command (Specify radius (R))

G17/G18/G19 G02/G03 X__ Y__ Z__ R__ F__ ;

G17/G18/G19	Arc plane (G17: XY plane, G18: ZX plane, G19: YZ plane)
G02/G03	Arc rotation direction (G02: clockwise, G03: counterclockwise)
X, Y	Arc end point coordinates
Z	Linear axis end point coordinates
I, J	Arc center coordinates
P	Number of pitches
R	Circular radius
F	Feedrate

Either an absolute value or incremental value can be used for the arc end point coordinate value command and the linear axis end point coordinate value command, but the arc center coordinate value must always be commanded with an incremental value from the start point.

The arc center coordinate value and arc radius value must be commanded in the input setting unit. Caution is required for the helical interpolation command of an axis for which the input command unit differs.

Command with a decimal point to avoid confusion.

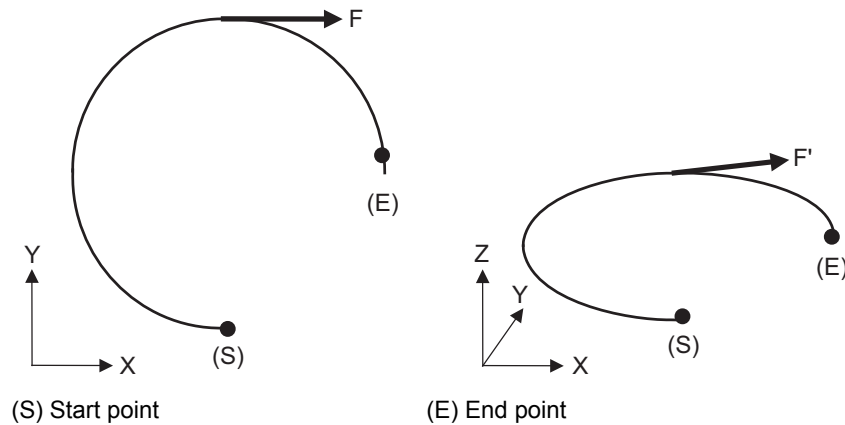


Detailed description

Speed designation during the helical interpolation

Normally, the helical interpolation speed is designated with the tangent speed F' including the 3rd axis interpolation element as shown in the lower drawing. However, when designating the arc plane element speed, the tangent speed F on the arc plane is commanded as shown in the upper drawing.

The NC automatically calculates the helical interpolation tangent speed F' so that the tangent speed on the arc plane is F .

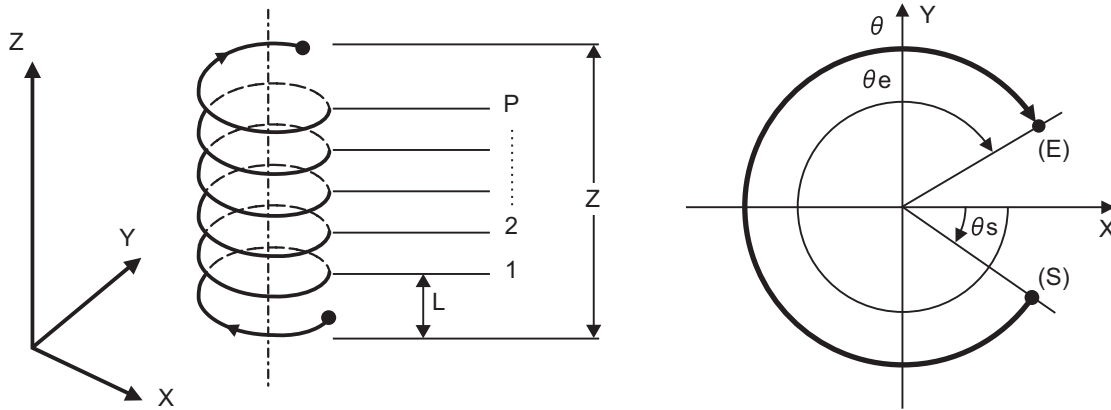


Whether the arc plane element speed designation or the normal speed designation is valid depends on the MTB specifications.

#1235 set07/bit0	Meaning
1	Arc plane element speed designation is selected.
0	Normal speed designation is selected.

Command the feedrate F as the composite speed for each axis.

Normal speed designation



(S) Start point (E) End point

- (1) This command should be issued with a linear axis (multiple axes can be commanded) that does not contain a circular axis in the circular interpolation command combined.
- (2) For feedrate F, command the X, Y and Z axis composite element directions speed.
- (3) Pitch L is obtained with the following expression.

$$L = \frac{Z}{(2\pi \cdot P + \theta) / 2\pi}$$

$$\theta = \theta_e - \theta_s = \tan^{-1} \frac{y_e}{x_e} - \tan^{-1} \frac{y_s}{x_s} \quad (0 \leq \theta < 2\pi)$$

x_s, y_s : Start point coordinates from the arc center x_e, y_e : End point coordinates from the arc center

- (4) If pitch No. is 0, address P can be omitted.

<Note>

- ♦The pitch No. P command range is 0 to 9999.

The pitch No. designation (P command) cannot be made with the R-specified arc.

- (5) Plane selection

The helical interpolation arc plane selection is determined with the plane selection mode and axis address as for the circular interpolation. For the helical interpolation command, the plane where circular interpolation is executed is commanded with the plane selection G code (G17, G18, G19), and the 2 circular interpolation axes and linear interpolation axis (axis that intersects with circular plane) 3 axis addresses are commanded.

XY plane circular, Z axis linear

Command the X, Y and Z axis addresses in the G02 (G03) and G17 (plane selection G code) mode.

ZX plane circular, Y axis linear

Command the X, Y and Z axis addresses in the G02 (G03) and G18 (plane selection G code) mode.

YZ plane circular, X axis linear

Command the X, Y and Z axis addresses in the G02 (G03) and G19 (plane selection G code) mode.

The plane for an additional axis can be selected as with circular interpolation.

UY plane circular, Z axis linear

Command the U, Y and Z axis addresses in the G02 (G03) and G17 (plane selection G code) mode.

In addition to the basic command methods above, the command methods described in the following program example can be used. Refer to the section "6.5 Plane Selection; G17, G18, G19" for the arc planes selected with these command methods.

Arc plane element speed designation selection

If arc plane element speed designation is selected, the F command will be handled as modal data in the same manner as the normal F command. This will also apply to the following G01, G02 and G03 commands.

(Example)

G17 G91 G02 X10. Y10. Z-4. I10. F100 ;	Helical interpolation at speed at which arc plane element is F100
G01 X20 ;	Linear interpolation at F100
G02 X10. Y-10. Z4. J-10. ;	Helical interpolation at speed at which arc plane element is F100
G01 Y-40. F120;	Linear interpolation at F120
G02 X-10. Y-10. Z-4. I-10. ;	Helical interpolation at speed at which arc plane element is F120
G01 X-20 ;	Linear interpolation at F120

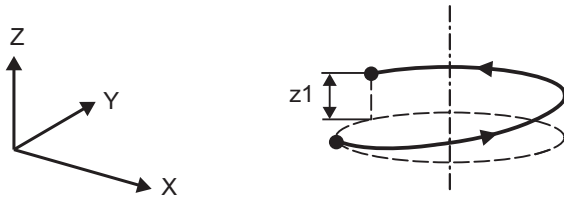
When the arc plane element speed designation is selected, only the helical interpolation speed command is converted to the speed commanded with the arc plane element and operates. The other linear and arc commands operate as normal speed commands.

- (1) The actual feedrate display (Fc) indicates the tangent element of the helical interpolation.
- (2) The modal value speed display (FA) indicates the command speed.
- (3) The speed data acquired with API functions follows the Fc and FA display.
- (4) This function is valid only when feed per minute (asynchronous feed: G94) is selected. If feed per revolution (synchronous feed: G95) is selected, the arc plane element speed will not be designated.



Program example

(Example 1)

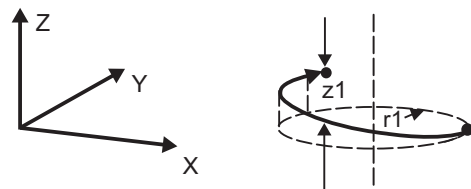


G17;	XY plane
G03 Xx1 Yy1 Zz1 Ii1 Jj1 P0 Ff1;	XY plane arc, Z axis linear

Note

(1) If pitch No. is 0, address P can be omitted.

(Example 2)

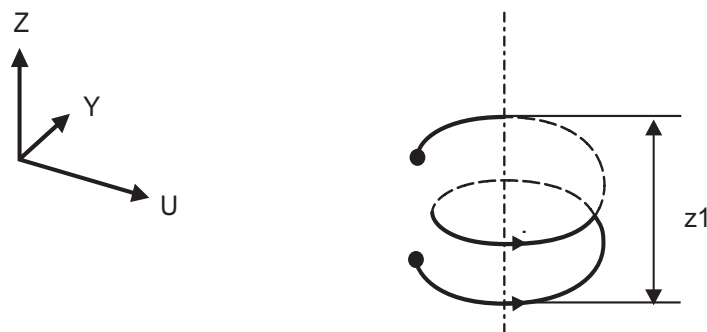


G17;	XY plane
G02 Xx1 Yy1 Zz1 Rr1 Ff1;	XY plane arc, Z axis linear

Note

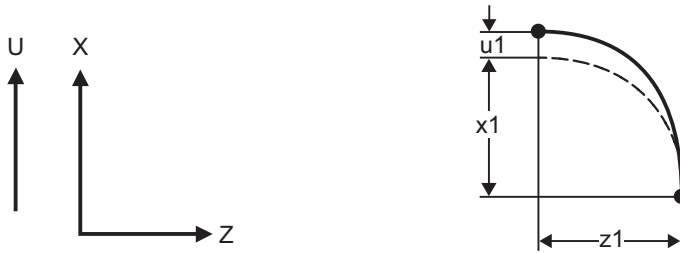
(1) The number of pitches is ignored even when it is commanded.

(Example 3)



G17 G03 Uu1 Yy1 Zz1 Ii1 Jj1 P2 Ff1;	UY plane arc, Z axis linear
-------------------------------------	-----------------------------

(Example 4)



G18 G03 Xx1 Uu1 Zz1 Ii1 Kk1 Ff1;	ZX plane arc, U axis linear
----------------------------------	-----------------------------

Note

(1) If the same system is used, the standard axis will perform circular interpolation and the additional axis will perform linear interpolation.

(Example 5)

G18 G02 Xx1 Uu1 Yy1 Zz1 Ii1 Jj1 Kk1 Ff1;	ZX plane arc, U axis, Y axis linear (The J command is ignored.)
--	--

Note

(1) Two or more axes can be designated for the linear interpolation axis.

6.8 Unidirectional Positioning

The unidirectional positioning function positions the tool at a high degree of precision without backlash error by locating the final tool position from a constant direction.

There are two types of positioning methods: G command method (G60 use method) and axis-based unidirectional positioning method (in which the axis specified by the MTB is always targeted for unidirectional positioning).

For the specifications of the machine you are using, see the specifications or Instruction Manual issued by the MTB.

6.8.1 Unidirectional Positioning; G60



Function and purpose

The G60 command can position the tool at a high degree of precision without backlash error by locating the final tool position from a constant direction.



Command format

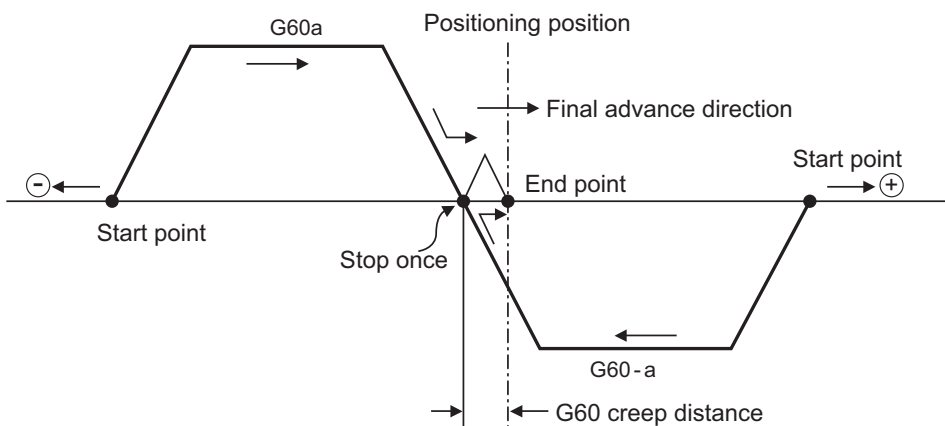
G60 X__ Y__ Z__ α__ ; ... Unidirectional positioning

α	Additional axis
---	-----------------



Detailed description

- (1) The creep distance for the final positioning as well as the final positioning direction is set by parameter.
- (2) After the tool has moved at the rapid traverse rate to the position separated from the final position by an amount equivalent to the creep distance, it moves to the final position in accordance with the rapid traverse setting where its positioning is completed.



- (3) The above positioning operation is performed even when Z axis commands have been assigned for Z axis cancel and machine lock. (Display only)
- (4) When the mirror image function is ON, the tool will move in the opposite direction as far as the intermediate position due to the mirror image function but the operation within the creep distance during its final advance will not be affected by a mirror image.
- (5) The tool moves to the end point at the dry run speed during dry run when the G00 dry run function is valid.

- (6) Feed during creep distance movement with final positioning can be stopped by resetting, emergency stop, interlock, feed hold and rapid traverse override zero.
The tool moves over the creep distance at the rapid traverse setting. Rapid traverse override is valid.
- (7) Unidirectional positioning is not performed for the drilling axis during fixed cycle for drilling.
- (8) Unidirectional positioning is not performed for shift amount movements during the fine boring or back boring fixed cycle.
- (9) Normal positioning is performed for axes whose creep distance has not been set by parameter.
- (10) Unidirectional positioning is always a non-interpolation type of positioning.
- (11) When the same position (movement amount of zero) has been commanded, the tool moves back and forth over the creep distance and is positioned at its original position from the final advance direction.
- (12) Program error (P61) will occur when the G60 command is assigned with an NC system which has not been provided with this particular specification.
- (13) The G60 command is assigned to group 00 (unmodal) in the previous versions; however, it can be operated as the modal G code of group 01 depending on the MTB specifications (parameter "#1271 ext07/bit3").
This omits a step to command G60 for each block.
This G60 command is the same as the previous unmodal G60 command, except it handles the G60 command as a modal.
- (14) If the G code of group 01 is commanded in the same block when the G60 command is handled as a modal, the G code commanded next becomes valid.

6.8.2 Axis-based Unidirectional Positioning



Function and purpose

This function carries out unidirectional positioning for each axis for G00 positioning.

The target axis is determined in the MTB specifications (parameter "#2084 G60_ax").

When the unidirectional positioning is commanded, set the last positioning direction and distance to parameter "#8209 G60 shift amount" for each axis.

The example below shows a case in which axis B is set as the unidirectional positioning axis.

G00 X100.;	Normal positioning
G00 B100.;	Unidirectional positioning
G00X200.B200.;	Axis X: Normal positioning, Axis B: Unidirectional positioning

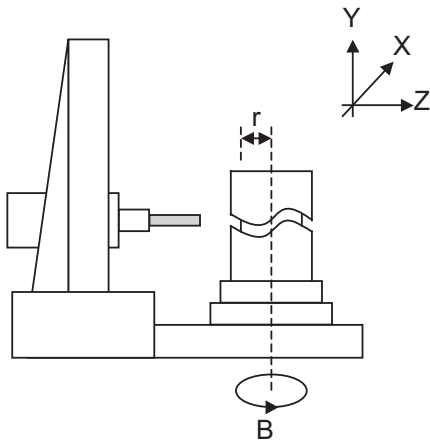
The axis-based unidirectional positioning is the same as for the G60 command. Refer to "6.8.1 Unidirectional Positioning; G60".

6.9 Cylindrical Interpolation; G07.1

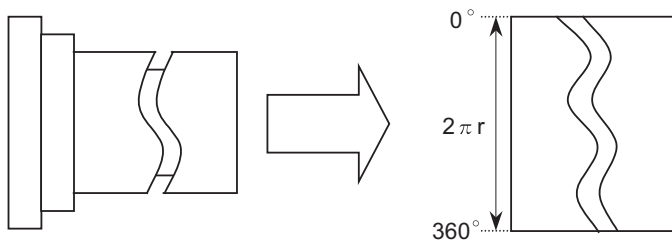


Function and purpose

This function develops a shape on the side of a cylinder (shape in a cylindrical coordinate system) into a plane. When the developed shape is programmed as the plane coordinates, it will be converted into a linear axis movement and rotation axis (temporarily, "B axis") movement in the original cylindrical coordinates to conduct contour control when machining.



As programming can be carried out to the developed shape of the side of the cylinder, this is effective for machining cylindrical cams, etc. When programmed with the rotary axis and its orthogonal axis, grooves and other shapes can be machined on the side of the cylinder.



Command format

Cylindrical interpolation mode start

G07.1 Rotary axis name, rotation radius value;

G107 Rotary axis name, rotation radius value;

Rotary axis name	Axis name assigned to rotary axis
Rotation radius value	Command a value other than "0". When a value other than "0" is commanded, the cylindrical interpolation mode starts.

Cylindrical interpolation mode cancel

G07.1 Rotary axis name 0;

G107 Rotary axis name 0;



Detailed description

- (1) The cylindrical interpolation is carried out between the rotary axis designated in the G07.1 block and another linear axis. (The following example shows a case in which the rotary axis name is set to "C".)

G19 ;	Plane selection
G07.1 C20. ;	Cylindrical interpolation mode start (Cylindrical interpolation will start.)
:	(The coordinate commands in this interval will be the cylindrical coordinate system)
G07.1 C0 ;	Cylindrical interpolation mode cancel (Cylindrical interpolation will be canceled.)

- (2) G107 can be used instead of G07.1.
- (3) Command G07.1 is an independent block. If it is commanded in the same block with other G code, a program error (P33) will occur.
- (4) The cylindrical interpolation mode is canceled when the power is turned ON or at resetting.
- (5) Linear interpolation or circular interpolation can be commanded during the cylindrical interpolation mode. Note that the plane selection command must be issued just before or after the G07.1 block.
- (6) The coordinate commands can be both an absolute command or incremental command.
- (7) Tool radius compensation can be applied on the program command. Cylindrical interpolation will be executed to the path after it has gone through a tool radius compensation.
- (8) Command the tangent speed on the developed cylinder by F. F is in mm/min or inch/min unit.
- (9) A program error (P484) will occur if any axis commanded during cylindrical interpolation has not completed the reference position return.
- (10) The deceleration check is made for the cylindrical interpolation start command block.

Cylindrical interpolation accuracy

In the cylindrical interpolation mode, the movement amount of the rotary axis commanded with an angle is converted into distance on a circle periphery, and after calculating the linear and circular interpolation between the other axes, the amount is converted into an angle again.

Thus, the actual movement amount may differ from the commanded value such as when the cylinder radius is small. Note that the gap generated by this will not be cumulated.

Cylindrical interpolation mode cancel

- (1) To cancel the cylindrical interpolation mode, the following condition must be satisfied.
- ◆Tool radius compensation is canceled.
- (2) When the cylindrical interpolation mode is canceled, the plane selected before the cylindrical interpolation will be restored.
- (3) The deceleration check is made for the cylindrical interpolation cancel command block.

Plane selection

The axis used for cylindrical interpolation must be set with the plane selection command.

Use parameters (#1029, #1030 and #1031) to set which parallel axis corresponds to the rotary axis.

The circular interpolation and tool radius compensation, etc., can be designated on that plane.

The plane selection command is set immediately before or after the G07.1 command. If a movement command is issued without this command, a program error (P485) will occur.

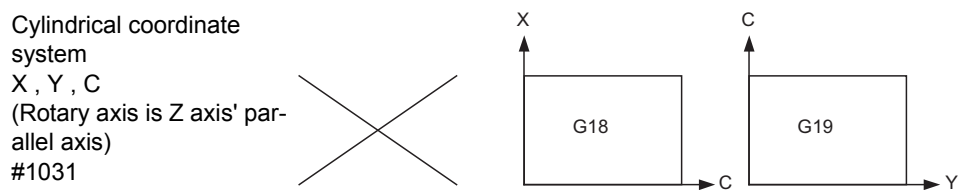
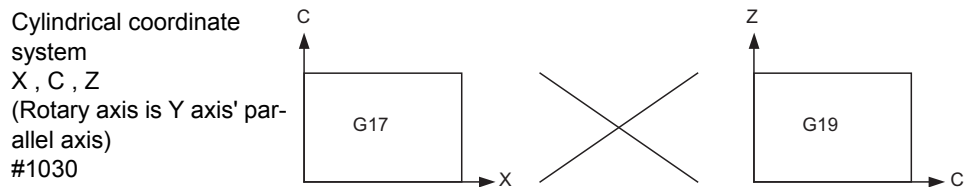
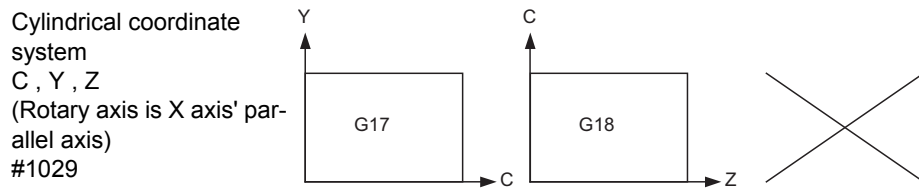
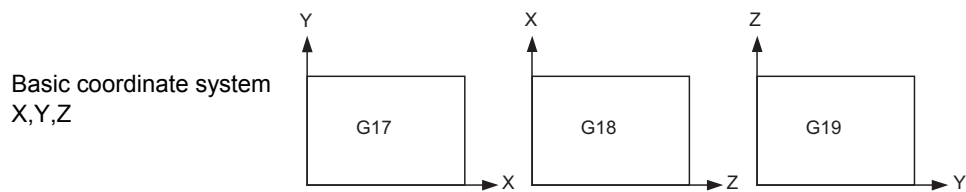
(Example)

G19 Z0. C0. ; Plane selection command for cylindrical interpolation, and 2-axis command of Z axis and C axis for interpolation

G07.1 C100. ; Cylindrical interpolation start

:

G07.1 C0 ; Cylindrical interpolation cancel





Program example

<Program>

```

N01 G28 XZC ;
N02 T0202 F500 ;
N03 G97 S100 M23 ;
N04 G00 X50. Z0. ;
N05 G94 G01 X40. F100. ;
N06 G19 C0 Z0 ;
N07 G07.1 C20. ;
N08 G41 ;
N09 G01 Z-10. C80. F150 ;
N10 Z-25. C90. ;
N11 Z-80. C225. ;
N12 G03 Z-75. C270. R55. ;
N13 G01 Z-25. ;
N14 G02 Z-20. C280. R80. ;
N15 G01 C360. ;
N16 G40 ;
N17 G07.1 C0 ;
N18 G01 X50. ;
N19 G00 X100. Z100. ;
N20 M25 ;
N21 M30 ;
    
```

Plane selection command for cylindrical interpolation and two axes command for interpolation

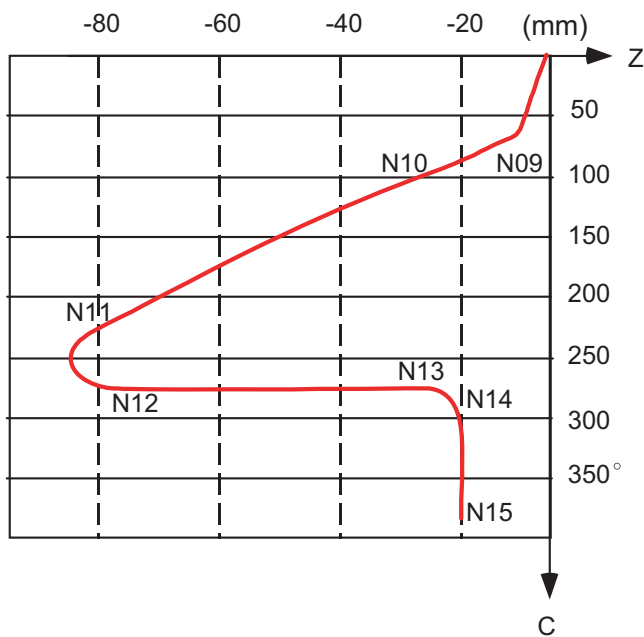
Cylindrical interpolation start

Cylindrical interpolation cancel

<Parameters>

```

#1029 aux_I
#1030 aux_J C
#1031 aux_K
    
```





Relationship with other functions

Circular interpolation

- (1) Circular interpolation between the rotary axis and linear axis is possible during the cylindrical interpolation mode.
- (2) Only the R specification command (mm/inch) is available for circular interpolation. (I, J and K cannot be designated.)

Tool radius compensation/tool nose radius compensation

The tool radius can be compensated during the cylindrical interpolation mode.

- (1) Command the plane selection in the same manner as circular interpolation.
When using tool radius compensation, start up/cancel the compensation in the cylindrical interpolation mode.
- (2) A program error (P485) will occur if G07.1 is commanded during tool radius compensation.
- (3) If the G07.1 command is issued with no movement command after the tool radius compensation has been canceled with the G40 independent command, the position of the axis in the G07.1 command block is interpreted as the position applied after the tool radius compensation has been canceled and the following operations are performed.

Miscellaneous functions

- (1) The miscellaneous functions (M) and 2nd miscellaneous functions (B) can be issued in the cylindrical interpolation mode.
- (2) The S command in the cylindrical interpolation mode specifies the rotary tool's rotation speed instead of the spindle rotation speed.

Tool length compensation

- (1) Program error (P481) will occur if tool length compensation is performed in the cylindrical interpolation mode.

:

```
G43 H12 ;           Tool length compensation before cylindrical interpolation -> Valid
G00 X100. Z0. ;
G19 Z C ;
G07.1 C100. ;
```

:

```
G43 H11 ;           Tool length compensation in cylindrical interpolation mode -> Program error
```

:

```
G07.1 C0 ;
```

- (2) Complete the tool compensation operation (movement of tool length and wear compensation amount) before executing the cylindrical interpolation.

If the tool compensation operation is not completed when the cylindrical interpolation start command is issued, the followings will occur:

The workpiece coordinate system shifts so that the relationship between the machine coordinate position and workpiece coordinate position matches the "positional relationship after the tool compensation has been completed" without actually moving the axis.

The workpiece coordinate system shifted here is not reset even if the cylindrical interpolation is canceled. The subsequent operations are performed, assuming that the tool compensation operation has been completed.

Feed mode and F command before and after cylindrical interpolation mode

The operation varies depending on whether the synchronous feed is valid or invalid during cylindrical interpolation. Whether the synchronous feed is valid or invalid depends on the MTB specifications (parameter "#1293 ext29/bit0).

(1) When the synchronous feed is invalid (#1293 ext29/bit0 = 0):

Only the asynchronous feed is valid during the cylindrical interpolation mode.

If the synchronous feed command (G95) is issued during the cylindrical interpolation mode, a program error (P481) will occur.

The operation to be performed when the cylindrical interpolation mode is started or canceled varies depending on the feed mode that is specified before the cylindrical interpolation mode is started.

Feed mode before start	Operation in cylindrical interpolation mode (At start/cancel)
Asynchronous feed (G94)	The previous feed mode (asynchronous feed) and the feedrate designated with the F command are inherited when the cylindrical interpolation mode is started or canceled.
Synchronous feed (G95)	When the cylindrical interpolation mode is started, the asynchronous mode is enabled forcibly, and the feedrate is canceled. After the cylindrical interpolation mode has been started, designate the feedrate using the F command. A program error (P62) will occur if the F command is not designated. When the cylindrical interpolation mode is canceled, the setting returns to the feed mode (synchronous feed) and feedrate before the cylindrical interpolation mode starts.

(2) When the synchronous feed is valid (#1293 ext29/bit0 = 1):

Both the synchronous feed and asynchronous feed are valid during the cylindrical interpolation mode.

The feed mode and feedrate remain unchanged and take over the previous state when the cylindrical interpolation mode is started or canceled.

Synchronous feed function valid/invalid (#1293 bit0)	Change of feed mode and feedrate		
	Before the cylindrical interpolation mode starts	When the cylindrical interpolation mode starts	When the cylindrical interpolation mode is canceled
Invalid	Asynchronous feed	Asynchronous feed Feedrate just before start	Asynchronous feed Feedrate just before cancel
	Synchronous feed	Asynchronous feed Feedrate = 0 (Cancel)	Synchronous feed Feedrate before cylindrical interpolation starts
Valid	Asynchronous feed	Asynchronous feed Feedrate just before start	Feed mode just before the mode is canceled Feedrate just before cancel
	Synchronous feed	Synchronous feed Feedrate just before start	

Cylindrical interpolation coordinate system

In the cylindrical coordinate system of the rotary axis for cylindrical interpolation, the coordinate positions depend on the MTB specifications (#1270 ext06/bit7).

Parameter setting (#1270 bit7)	Coordinate position of rotary axis in cylindrical coordinate system
0	In this coordinate system, the rotary axis position is set to "0" when the cylindrical interpolation start command is issued.
1	The workpiece coordinate positions just before the cylindrical interpolation starts are also continued to be used during cylindrical interpolation.

Cylindrical interpolation function: Combinations of G code commands

The following G code commands can be used during the cylindrical interpolation mode.

G code	Description
G00	Positioning
G01	Linear interpolation
G02	Circular interpolation (CW)
G03	Circular interpolation (CCW)
G04	Dwell
G09	Exact stop check
G40-42	Tool radius compensation
G61	Exact stop mode
G64	Cutting mode
G65	Macro call (simple call)
G66	Macro modal call (modal call)
G66.1	Macro modal call (block call per macro)
G67	Macro modal call cancel (modal call cancel)
G80-89	Fixed cycle for drilling
G90/91	Absolute/incremental value command
G94	Asynchronous feed
G98	Hole drilling cycle initial return
G99	Hole drilling cycle R point return

A program error will occur if a G code other than those listed above is commanded during cylindrical interpolation.

[Combination of cylindrical interpolation and high-accuracy control function]

To enable the G08P1 or G61.1 (high-accuracy control) mode during the G07.1 (cylindrical interpolation) or G12.1 (polar coordinate interpolation) mode or to enable the G07.1 (cylindrical interpolation) or G12.1 (polar coordinate interpolation) mode during the G08P1 or G61.1 (high-accuracy control) mode, you need to enable the tolerable acceleration control for each axis (optimum acceleration control) or variable-acceleration pre-interpolation acceleration/deceleration. (The validity of these functions depends on the MTB specifications.)

If the cylindrical interpolation or polar coordinate interpolation command is issued during the high-accuracy control mode while the functions above are invalid, a program error (P126) will occur.

Also, if the high-accuracy control command is issued during the cylindrical interpolation or polar coordinate interpolation mode, a program error (P481) will occur.

**Restrictions and precautions**

- (1) The cylindrical interpolation mode is canceled when the power is turned ON or reset.
- (2) Program cannot be restarted (program restart) when the block is in the cylindrical interpolation.
- (3) The cylindrical interpolation command cannot be issued in mirror image (parameter/external input ON). If the command is issued, a program error (P486) will occur.
- (4) A program error (P481) will occur if the cylindrical interpolation command (G07.1), the polar coordinate interpolation command (G12.1), or the milling interpolation command (G12.1) is issued again during the cylindrical interpolation mode.

6.10 Circular Cutting; G12,G13



Function and purpose

Circular cutting starts the tool from the center of the circle, and cuts the inner circumference of the circle. The tool continues cutting while drawing a circle and returns to the center position.



Command format

Circular cutting Clockwise (CW)

G12 I__ D__ F__ ;

Circular cutting Counterclockwise (CCW)

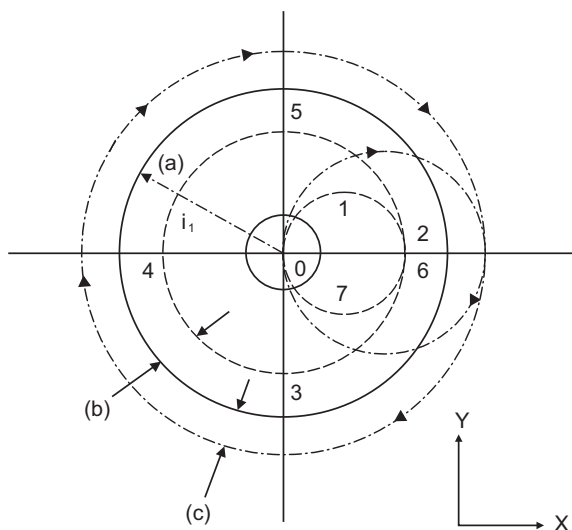
G13 I__ D__ F__ ;

I	Radius of circle (incremental value), the sign is ignored
D	Offset No. (The offset No. and offset data are not displayed on the setting and display unit.)
F	Feedrate



Detailed description

- (1) The sign + for the offset amount indicates reduction, and - indicates enlargement.
- (2) The circle cutting is executed on the plane G17, G18 or G19 currently selected.



- Compensation amount sign +
- Compensation amount sign -
- (a) Circle radius
- (b) d1 offset amount +
- (c) d1 offset amount -

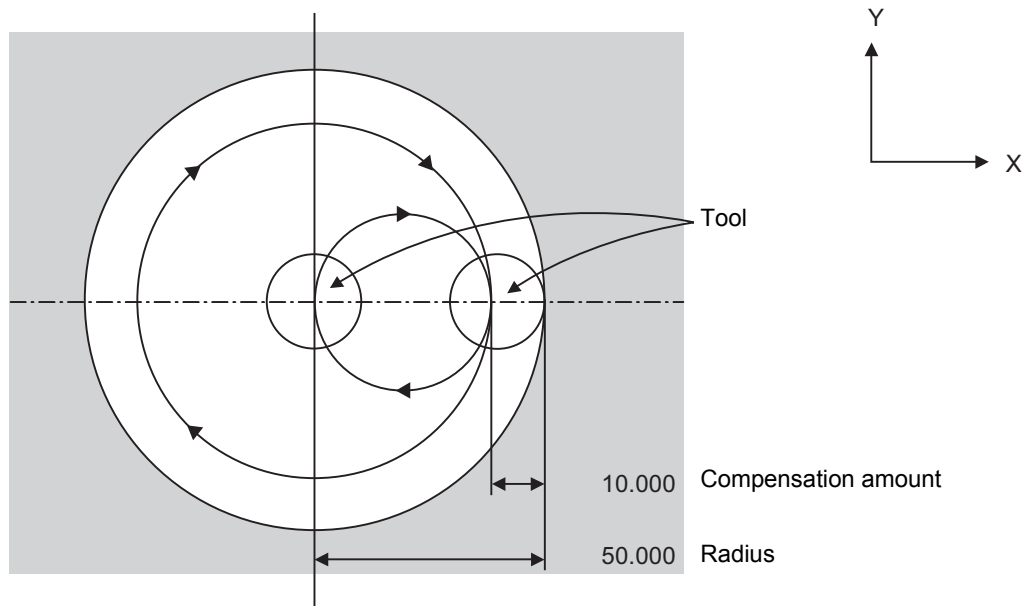
For G12 (tool center path) 0->1->2->3->4->5->6->7->0

For G13 (tool center path) 0->7->6->5->4->3->2->1->0



Program example

(Example 1) G12 I50.000 D01 F100 ; When compensation amount is +10.000mm



Precautions

- (1) If the offset No. "D" is not issued or if the offset No. is illegal, the program error (P170) will occur.
- (2) If [Radius (I) - offset amount] is 0 or negative, the program error (P223) will occur.
- (3) If G12 or G13 is commanded during radius compensation (G41, G42), the radius compensation will be validated on the path after compensated with the D, commanded with G12 or G13.
- (4) If an address not included in the format is commanded in the same block as G12 and G13, the program error (P32) will occur.
But when the parameter "#11034 Circular cutting command address check type" is set to "1", it operates as follows;
 - (a) Program error will not occur except for an "H" command.
 - (b) Only "D", "F", "I" and "M", "S", "T", "B" will be valid.

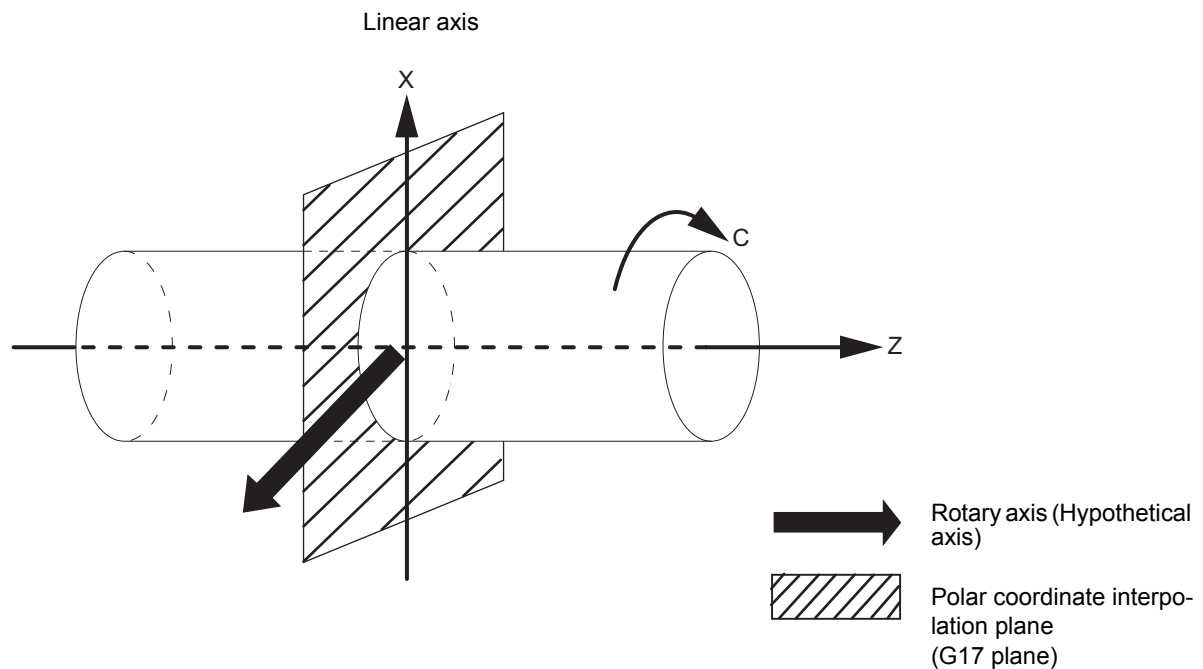
6.11 Polar Coordinate Interpolation; G12.1, G13.1/G112, G113



Function and purpose

This function converts the commands programmed with the orthogonal coordinate axis into linear axis movement (tool movement) and rotary axis movement (workpiece rotation), and controls the contour.

The plane that uses the linear axis as the plane's 1st orthogonal axis, and the intersecting hypothetical axis as the plane's 2nd axis (hereafter "polar coordinate interpolation plane") is selected. Polar coordinate interpolation is carried out on this plane. The workpiece coordinate system zero point is used as the coordinate system zero point during polar coordinate interpolation.



This is effective for cutting a notch in a linear line to the external diameter of the workpiece, for cutting cam shafts, etc.



Command format

Polar coordinate interpolation mode start

G12.1 ;

Polar coordinate interpolation mode cancel

G13.1 ;



Detailed description

- (1) The coordinate commands in the interval from the start to cancellation of the polar coordinate interpolation mode will be polar coordinate interpolation.
 - G12.1 ; Polar coordinate interpolation mode start (Polar coordinate interpolation will start.)
(The coordinate commands in this interval will be the polar coordinate interpolation.)
 - :
 - G13.1 ; Polar coordinate interpolation mode cancel (Polar coordinate interpolation is canceled.)
- (2) G112 and G113 can be used instead of G12.1 and G13.1.
- (3) Command G12.1,G13.1 in an independent block. If it is commanded in the same block with other G code, a program error (P33) will occur.
- (4) Linear interpolation or circular interpolation can be commanded during the polar coordinate interpolation mode.
- (5) The coordinate commands can be both an absolute command or incremental command.
- (6) Tool radius compensation can be applied on the program command. Polar coordinate interpolation will be executed to the path after it has gone through a tool radius compensation.
- (7) Command the tangent speed in the polar coordinate interpolation plane (orthogonal coordinate system) by F. F is in mm/min or inch/min unit.
- (8) When the G12.1/G13.1 command is issued, the deceleration check is executed.

Plane selection

The linear axis and rotary axis used for polar coordinate interpolation depend on the MTB specifications (parameter #1533).

- (1) Determine the deemed plane for carrying out polar coordinate interpolation with the parameter (#1533) of the linear axis used for polar coordinate interpolation.

Setting for #1533	Deemed plane
X	G17 (XY plane)
Y	G19 (YZ plane)
Z	G18 (ZX plane)
Blank (no setting)	G17 (XY plane)

- (2) A program error (P485) will occur if the plane selection command (G17 to G19) is issued during the polar coordinate interpolation mode.

<Note>

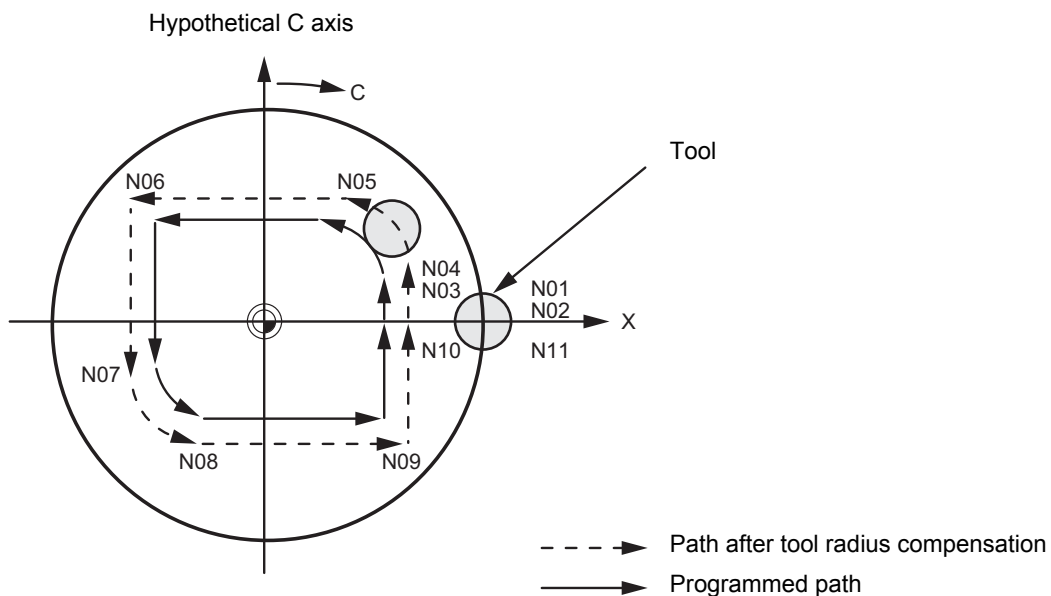
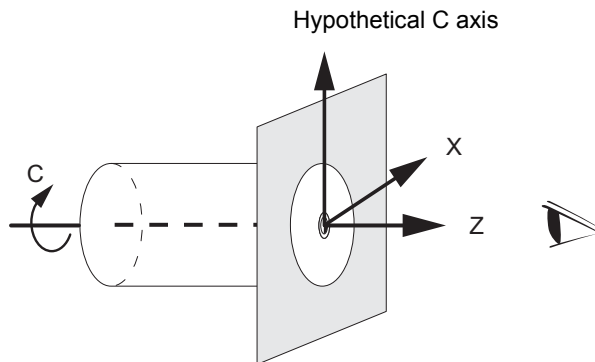
- ♦Depending on the model or version, parameter (#1533) may not be provided. In this case, the operation will be the same as when the parameter (#1533) is blank (no setting).

Related parameter

- #1516 mill_ax (Milling axis name)
- #1517 mill_c (Milling interpolation hypothetical axis name)
- #8111 Milling Radius
- #1533 mill_Pax (Polar coordinate linear axis name)



Program example



<Program>

```

:
N01 G17 G90 G00 X40.0 C0 Z0;           Setting of start position
N02 G12.1 ;                           Polar coordinate interpolation mode: Start
N03 G01 G42 X20.0 F2000;               Actual machining start
N04 C10.0;
N05 G03 X10.0 C20.0 R10.0;
N06 G01 X-20.0;                         Shape program
N07 C-10.0;
N08 G03 X-10.0 C-20.0 I10.0 J0;        (Follows orthogonal coordinate positions on X-C hypothetical axis
                                        plane.)
N09 G01 X20.0;
N10 C00;
N11 G40 X40.0;
N12 G13.1 ;                             Polar coordinate interpolation mode: Cancel
:
M30 ;
    
```



Relationship with Other Functions

Program commands during polar coordinate interpolation

- (1) The program commands in the polar coordinate interpolation mode are issued by the orthogonal coordinate value of the linear axis and rotary axis (hypothetical axis) on the polar coordinate interpolation plane.
The axis address of the rotary axis (C) is specified as the axis address for the plane's 2nd axis (hypothetical axis) command.
The command unit is not deg (degree). The same unit (mm or inch) as used for the command by the axis address of the plane's 1st axis (linear axis) will be used.
- (2) The hypothetical axis coordinate value will be set to "0" when G12.1 is commanded. In other words, the position where G12.1 is commanded will be interpreted as angle = 0, and the polar coordinate interpolation will start.

Circular interpolation on polar coordinate plane

The arc radius address for carrying out circular interpolation during the polar coordinate interpolation mode is determined with the linear axis parameter (#1533).

Setting for #1533	Center designation command
X	I, J (polar coordinate plane is interpreted as XY plane)
Y	J, K (polar coordinate plane is interpreted as YZ plane)
Z	K, I (polar coordinate plane is interpreted as ZX plane)
Blank (no setting)	I, J (polar coordinate plane is interpreted as XY plane)

The arc radius can also be designated with the R command.

Note

- (1) Depending on the model or version, parameter (#1533) may not be provided. In this case, the operation will be the same as when the parameter (#1533) is blank (no setting).

Tool radius compensation

The tool radius can be compensated during the cylindrical interpolation mode.

- (1) Command the plane selection in the same manner as polar coordinate interpolation.
When conducting tool radius compensation, it must be started up and canceled during the polar coordinate interpolation mode.
- (2) A program error (P485) will occur if polar coordinate interpolation is executed during tool radius compensation.
- (3) If the G12.1 and G13.1 commands are issued with no movement command after the tool radius compensation is canceled, the position of the axis in the G12.1 and G13.1 commands block is interpreted as the position applied after the tool radius compensation is canceled and the following operations are performed.

Cutting asynchronous feed

- (1) The asynchronous mode is forcibly set when the polar coordinate interpolation mode is started.
- (2) When the polar coordinate interpolation mode is canceled, the synchronous mode will return to the state before the polar coordinate interpolation mode was started.
- (3) A program error (P485) will occur if G12.1 is commanded in the constant surface speed control mode (G96).

Miscellaneous function

- (1) The miscellaneous function (M) and 2nd miscellaneous function can be issued in the polar coordinate interpolation mode.
- (2) The S command in the polar coordinate interpolation mode specifies the rotary tool's rotation speed instead of the spindle rotation speed.

Tool length compensation

- (1) Program error (P481) will occur if tool length compensation is performed in the polar coordinate interpolation mode.

```

:
G43 H12 ;           Tool length compensation before polar coordinate interpolation -> Valid
G00 X100. Z0. ;
G12.1 ;
:
G43 H11 ;           Tool length compensation in polar coordinate interpolation mode -> Pro-
                    program error
:
G13.1 ;

```

- (2) Complete the tool compensation operation (movement of tool length and wear compensation amount) before executing the polar coordinate interpolation.

If the tool compensation operation is not completed when the polar coordinate interpolation start command has been issued, the followings will occur:

- Machine coordinate will not change even if G12.1 is executed.
- When G12.1 is executed, the workpiece coordinate will change to that of the post tool length compensation.

(Even if polar coordinate interpolation is canceled, this workpiece coordinate will not be canceled.)

F command during polar coordinate interpolation

As for the F command during polar coordinate interpolation mode, whether to use the previous F command depends on the previous mode of the feed per minute command (G94/G98) or feed per rotation command (G95/G99).

- (1) When G94 (G98) is commanded just before G12.1

If there is no F command in the polar coordinate interpolation, the previous F command feedrate will be used. After the polar coordinate interpolation mode is canceled, the F command feedrate set at the start of the polar coordinate interpolation mode or the last F command feedrate set during polar coordinate interpolation will continue to be the feedrate.

- (2) When G95 (G99) is commanded just before G12.1

The previous F command feedrate cannot be used during polar coordinate interpolation. A new F command must be issued.

The feedrate after the polar coordinate interpolation mode is canceled will return to the state before the polar coordinate interpolation mode was started.

[When there is no F command in G12.1]

Previous mode	No F command	After G13.1
G94 (G98)	Previous F is used	(Same as on the left)
G95 (G99)	Program error (P62)	F just before G12.1 is used

[When F is commanded in G12.1]

Previous mode	With F command	After G13.1
G94 (G98)	Commanded F is used	(Same as on the left)
G95 (G99)	Commanded F is used (*1)	F just before G12.1 is used

(*1) Moves with the feed per minute command during G12.1.

Hole drilling axis in the fixed cycle for drilling command

Hole drilling axis in the fixed cycle for drilling command during the polar coordinate interpolation is determined with the linear axis parameter (# 1533).

Setting for #1533	Hole drilling axis
X	Z (polar coordinate plane is interpreted as XY plane)
Y	X (polar coordinate plane is interpreted as YZ plane)
Z	Y (polar coordinate plane is interpreted as ZX plane)
Blank (no setting)	Z (polar coordinate plane is interpreted as XY plane)

Shift amount in the G76 (fine boring) or G87 (back boring) command

Shift amount in the G76 (fine boring) or G87 (back boring) command during the polar coordinate interpolation is determined with the linear axis parameter (#1533).

Setting for #1533	Center designation command
X	I, J (polar coordinate plane is interpreted as XY plane)
Y	J, K (polar coordinate plane is interpreted as YZ plane)
Z	K, I (polar coordinate plane is interpreted as ZX plane)
Blank (no setting)	I, J (polar coordinate plane is interpreted as XY plane)



Restrictions and precautions

- (1) The following G code commands can be used during the polar coordinate interpolation mode.

G code	Details
G00	Positioning
G01	Linear interpolation
G02	Circular interpolation (CW)
G03	Circular interpolation (CCW)
G04	Dwell
G09	Exact stop check
G40-42	Tool radius compensation
G61	Exact stop mode
G64	Cutting mode
G65	Macro call (simple call)
G66	Macro modal call (modal call)
G66.1	Macro modal call (block call per macro)
G67	Macro modal call cancel (modal call cancel)
G80-89	Fixed cycle for drilling
G90/91	Absolute/incremental value command
G94	Asynchronous feed
G98	Hole drilling cycle initial return
G99	Hole drilling cycle R point return

A program error (P481) may occur if a G code other than those listed above is commanded during polar coordinate interpolation.

- (2) Program cannot be restarted (program restart) when the block is in the polar coordinate interpolation.
- (3) Before commanding polar coordinate interpolation, set the workpiece coordinate system so that the center of the rotary axis is at the coordinate system zero point. Do not change the coordinate system during the polar coordinate interpolation mode. (G50, G52, G53, relative coordinate reset, G54 to G59, etc.)
- (4) The feedrate during polar coordinate interpolation will be the interpolation speed on the polar coordinate interpolation plane (orthogonal coordinate system).
(The relative speed with the tool will vary according to the polar coordinate conversion.)
When passing near the center of the rotary axis on the polar coordinate interpolation plane (orthogonal coordinate system), the rotary axis side feedrate after polar coordinate interpolation will be very high.
- (5) The axis movement command outside of the plane during polar coordinate interpolation will move unrelated to the polar coordinate interpolation.
- (6) The current position displays during polar coordinate interpolation will all indicate the actual coordinate value. However, the "remaining movement amount" indicates the movement amount on the polar coordinate input plane.
- (7) The polar coordinate interpolation mode is canceled when the power is turned ON or reset.
- (8) A program error (P484) will occur if any axis commanded during polar coordinate interpolation has not completed the reference position return.
- (9) Tool radius compensation must be canceled before canceling the polar coordinate interpolation mode.
- (10) When the polar coordinate interpolation mode is canceled and switched to the cutting mode, the plane selected before the polar coordinate interpolation will be restored.
- (11) A program error (P486) will occur if the polar coordinate interpolation command is issued during the mirror image.
- (12) A program error (P481) will occur if the cylindrical interpolation or the polar coordinate interpolation is commanded during the polar coordinate interpolation mode.
- (13) During polar coordinate interpolation, if X axis moveable range is controlled in the plus side, X axis has to be moved to the plus area that includes "0" and above before issuing the polar coordinate interpolation command. If X axis moveable range is controlled in the minus side, X axis has to be moved to the minus area that does not include "0" before issuing the polar coordinate interpolation command.

6.12 Exponential Interpolation; G02.3, G03.3



Function and purpose

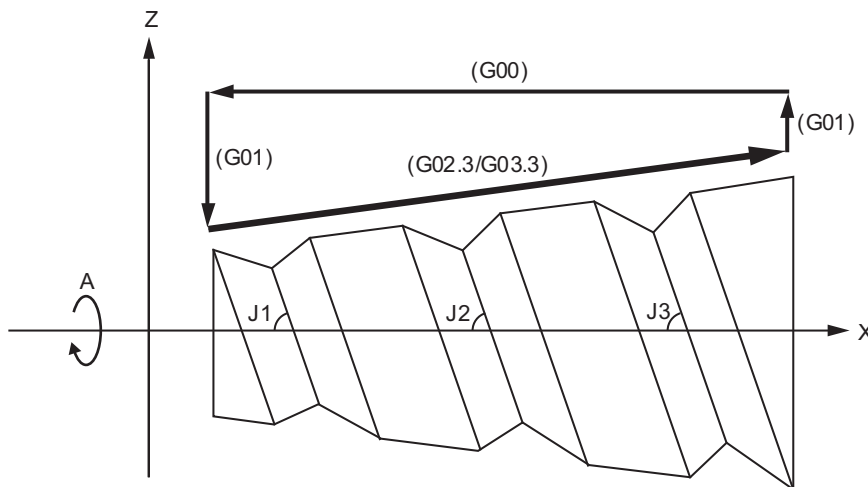
Exponential function interpolation changes the rotary axis into an exponential function shape in respect to the linear axis movement.

At this time, the other axes carry out linear interpolation between the linear axis.

This allows a machining of a taper groove with constant torsion angle (helix angle) (uniform helix machining of taper shape).

This function can be used for slotting or grinding a tool for use in an end mill, etc.

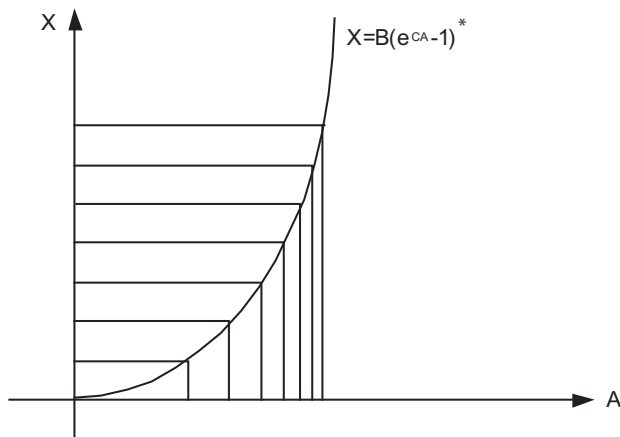
•Uniform helix machining of taper shape



A: A axis (rotation axis)
X: X axis (linear axis)

Torsion angle: $J1 = J2 = J3$

•Relation of linear axis and rotary axis



A: A axis (rotation axis)
X: X axis (linear axis)
* : {B, C... constant}



Command format

Forward rotation interpolation (Modal)

G02.3 Xx1 Yy1 Zz1 Ii1 Jj1 Rr1 Ff1 Qq1 Kk1 ;

Backward rotation interpolation (Modal)

G03.3 Xx1 Yy1 Zz1 Ii1 Jj1 Rr1 Ff1 Qq1 Kk1 ;

X	X axis end point (*1)
Y	Y axis end point (*1)
Z	Z axis end point (*1)
I	Angle i1 (*2)
J	Angle j1 (*2)
R	Constant value r1 (*3)
F	Initial feedrate (*4)
Q	Feedrate at end point (*5)
K	Command will be ignored.

(*1) Designate the end point of the linear axis specified by parameter "#1514 expLinax" and the axis that carries out linear interpolation between that axis.

If the end point on of the rotary axis designated with parameter "#1515 expRotax" is specified, linear interpolation without exponential function interpolation will take place. These parameter settings depend on the MTB specifications.

(*2) The command unit is as follows.

Setting unit	#1003=B	#1003=C	#1003=D	#1003=E
(Unit = °)	0.001	0.0001	0.00001	0.000001

The command range is -89 to +89°.

A program error (P33) will occur if there is no address I or J command.

A program error (P35) will occur if the address I or J command value is 0.

(*3) The command unit is as follows.

Setting unit	#1003=B	#1003=C	#1003=D	#1003=E	Unit
Millimeter system	0.001	0.0001	0.00001	0.000001	mm
Inch system	0.0001	0.00001	0.000001	0.0000001	inch

The command range is a positive value that does not include 0.

A program error (P33) will occur if there is no address R command.

A program error (P35) will occur if the address R command value is 0.

(*4) The command unit and command range is the same as the normal F code. (Command as per minute feed.)

Command the composite feedrate that includes the rotary axis.

The normal F modal value will not change by the address F command.

A program error (P33) will occur if there is no address F command.

A program error (P35) will occur if the address F command value is 0.

(*5) The command unit is as follows.

Setting unit	#1003 = B	#1003=C	#1003=D	#1003=E	Unit
Millimeter system	0.001	0.0001	0.00001	0.000001	mm
Inch system	0.0001	0.00001	0.000001	0.0000001	inch

The command unit and command range is the same as the normal F code.

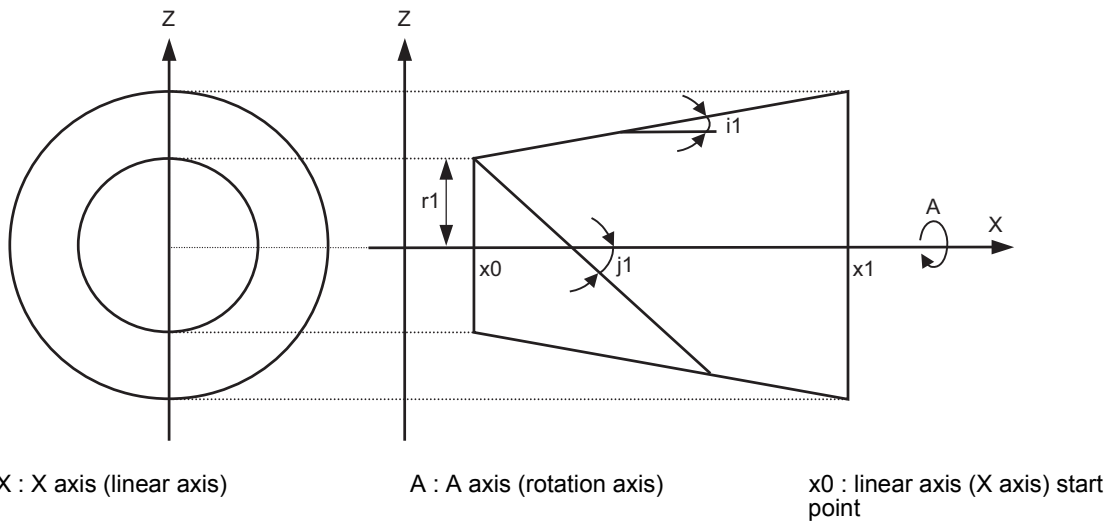
Command the composite feedrate that includes the rotary axis.

The normal F modal value will not change by the address Q command.

The axis will interpolate between the initial speed (F) and end speed (Q) in the CNC according to the linear axis. If there is no address Q command, interpolation will take place with the same value as the initial feedrate (address F command). (The start point and end point feedrates will be the same.)

A program error (P35) will occur if the address Q command value is 0.

[Example of uniform helix machining of taper shape]



Detailed description

Relational expression of exponential function

The exponential function relational expression of the linear axis (X) and rotary axis (A) in the G02.3/G03.3 command is defined in the following manner.

$$X(\theta) = r1 * (e^{\theta/D} - 1) / \tan(i1) \quad \text{Linear axis (X) movement (1)}$$

$$A(\theta) = (-1)^\omega * 360 * \theta / (2\pi) \quad \text{Rotary axis (A) movement}$$

where, "D" is as follows.

$$D = \tan(j1) / \tan(i1)$$

During forward rotation (G02.3): $\omega = 0$

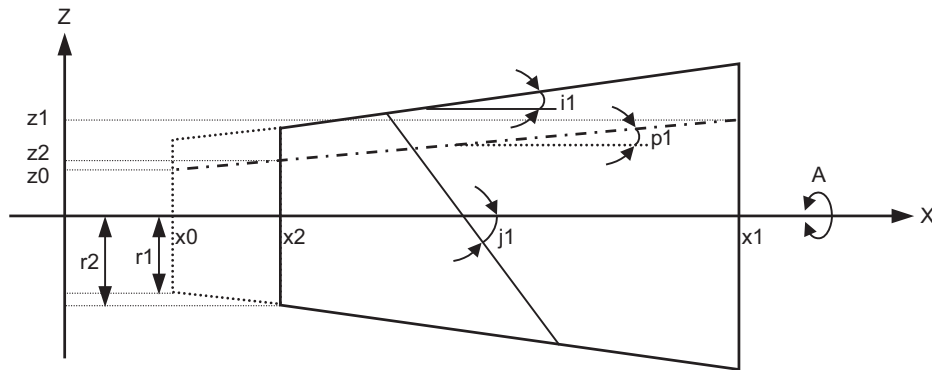
During backward rotation (G03.3): $\omega = 1$

θ is the rotation angle (radian) from the rotary axis' start point. The rotary axis' rotation angle (θ) is as follows according to expression (1).

$$\theta = D * 1n\{(X * \tan(i1) / r1) + 1\}$$

Machining example

[Uniform helix machining of taper shape]



<Relational expression of exponential function in machining example>

$$Z(\theta) = r1 * (e^{\theta/D} - 1) * \tan(p1) / \tan(i1) + z0 \quad (1)$$

$$X(\theta) = r1 * (e^{\theta/D} - 1) / \tan(i1) \quad (2)$$

$$A(\theta) = (-1)^\omega * 360 * \theta / (2\pi)$$

where, "D" is as follows.

$$D = \tan(j1) / \tan(i1)$$

Z(θ) Absolute value from zero point of Z axis (axis that linearly interpolates with linear axis (X axis))

X(θ) Absolute value from X axis (linear axis) start point

A(θ) Absolute value from A axis (rotary axis) start point

r1 Exponential function interpolation constant value (address R command)

r2 Workpiece left edge radius

x2 X axis (linear axis) position at the left edge of the workpiece

x1 X axis (linear axis) end point (address X command)

x0 X axis (linear axis) start point (Set as "x0 ≤ x1" so that workpiece does not interfere with the tool)

z1 End point of Z axis (axis that linearly interpolates between interval with linear axis (X axis)) (address Z command)

z0 Start point of Z axis (axis that linearly interpolates between interval with linear axis (X axis))

i1 Taper gradient angle (address I command)

p1 Slot base gradient angle

j1 Torsion angle (helix angle) (address J command)

ω Torsion direction (0: Forward rotation, 1: reverse direction)

θ Workpiece rotation angle (radian)

f1 Initial feedrate (address F command)

q1 Feedrate at end point (address Q command)

k1 Insignificant data (address K command)

According to expressions (1) and (2), expression (3) is obtained.

$$Z(\theta) = X(\theta) * \tan(p1) + z0 \dots(3)$$

According to expression (3), the slot base gradient angle (p1) is set from the X axis and Z axis end point positions (x1, z1).

The Z axis movement amount is determined by the slot base gradient angle (p1) and X axis position.

In the above diagram, the exponential function interpolation's constant value (r1) is determined with the following expression using the workpiece left edge radius (r2), X axis start point (x0), X axis position at workpiece left edge (x2) and taper gradient angle (i1).

$$r1 = r2 - \{(x2 - x0) * \tan(i1)\}$$

The taper gradient angle (i1) and torsion angle (j1) are set by the command address I and J, respectively.

Note that if the shape is a reverse taper shape, the taper gradient angle (i1) is issued as a negative value.

The torsion direction (ω) is changed by the G code. (Forward rotation when G02.3 is commanded, negative rotation when G03.3 is commanded)

The above settings allow uniform helix machining of a taper shape (or reverse taper shape).

Command and operation

(1) G2.3 (equivalent to G3.3 if $j1 < 0$)

In the conditional figure below, the upper side shows a command, and the lower side shows an operation.

X movement direction > 0		X movement direction < 0	
i1 > 0	i1 < 0	i1 > 0	i1 < 0
N10 G28XYZC; N20 G91G0 X100. Z100.; N30 G2.3 X100. Z100. I50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X100. Z200.; N30 G2.3 X100. Z-100. I- 50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X-100. Z100.; N30 G2.3 X-100. Z100. I50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X-100. Z200.; N30 G2.3 X-100. Z-100. I-50. J80. R105. F500.; N40 M30;

(S) Start point, (E) End point

(2) G3.3 (equivalent to G2.3 if $j1 < 0$)

In the conditional figure below, the upper side shows a command, and the lower side shows an operation.

X movement direction > 0		X movement direction < 0	
i1 > 0	i1 < 0	i1 > 0	i1 < 0
N10 G28XYZC; N20 G91G0 X100. Z100.; N30 G3.3 X100. Z100. I50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X100. Z200.; N30 G3.3 X100. Z-100. I- 50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X-100. Z100.; N30 G3.3 X-100. Z100. I50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X-100. Z200.; N30 G3.3 X-100. Z-100. I-50. J80. R105. F500.; N40 M30;

(S) Start point, (E) End point



Precautions

- (1) When G02.3/G03.3 is commanded, interpolation takes place with the exponential function relational expression using the start position of the linear axis and rotary axis as 0.
- (2) Linear interpolation will take place in the following cases, even if in the G02.3/G03.3 mode.
The feedrate for linear interpolation will be the F command in that block. (Note that the normal F modal is not updated.)
 - ♦The linear axis designated with the parameter (#1514 expLinax) is not commanded, or the movement amount for that axis is 0.
 - ♦The rotary axis designated with the parameter (#1515 expRotax) is commanded.
- (3) A program error will occur if the following commands are issued during the G02.3/G03.3 mode.
A program error will also occur if G02.3 or G03.3 command is issued in the following modes.
 - ♦Tool length compensation (A program error will occur only when the compensation starts at the same time as the movement by exponential function interpolation.)
 - ♦The tool length compensation will operate normally if it has started before the G02.3/G03.3 mode starts.)
 - ♦Tool radius compensation
 - ♦High-speed High-accuracy Control
 - ♦High-speed machining
 - ♦Scaling
 - ♦Tool length compensation along the tool axis
 - ♦Figure rotation
 - ♦Coordinate rotation by program
 - ♦Coordinate rotation by parameter
 - ♦3-dimensional coordinate conversion
- (4) A program error (P481) will occur if commands are issued during the pole coordinate interpolation, cylindrical interpolation or milling interpolation modes.
- (5) Program error (P612) will occur if commands are issued during the scaling or mirror image.
- (6) Program error (P34) will occur if commands are issued during the high-speed high-accuracy control II.
- (7) G02.3/G03.3 will function with asynchronous feed even during the synchronous feed mode, and the synchronous feed mode will be canceled.
- (8) If the parameter "#1515 expRota" setting is the same axis name as the initial C axis, the axis selected with the C axis selection signal will interpolate as the rotary axis.

6.13 Polar Coordinate Command; G16



Function and purpose

With this function, the end point coordinate value is commanded with the polar coordinate of the radius and angle.



Command format

G16 ; ... Polar coordinate command mode ON

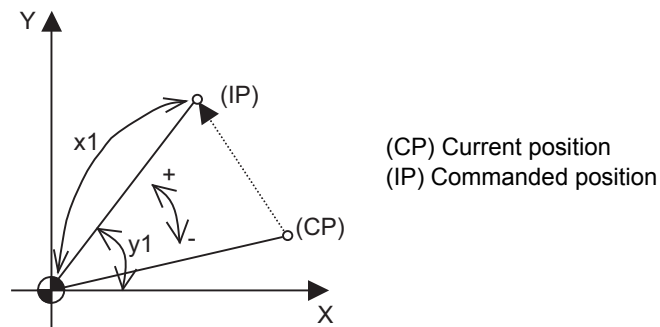
G15 ; ... Polar coordinate command mode OFF



Detailed description

- (1) The polar coordinate command is applied in the interval from turning ON to OFF of the polar coordinate command mode

G1x ;	Plane selection for polar coordinate command (G17/G18/G19)
G16 ;	Polar coordinate command mode ON
G9x G01 Xx1 Yy1 F2000 ;	Polar coordinate command
:	G9x: Center selection for polar coordinate command (G90/G91)
	G90: The workpiece coordinate system zero point is the polar coordinate center.
	G91: The current position is the polar coordinate center.
	x1: 1st axis for the plane: The radius of the polar coordinate commanded
	y1: 2nd axis for the plane: The angle of the polar coordinate commanded
G15 ;	Polar coordinate command mode OFF



For G90/G17 (X-Y plane)

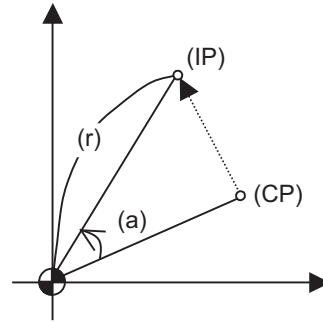
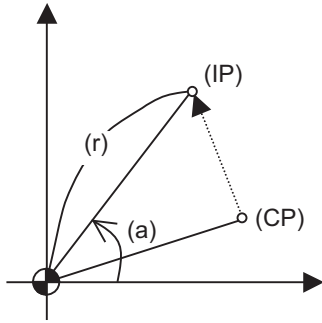
- (2) The plane selection during the polar coordinate command mode is carried out with G17, G18 and G19.
- (3) The polar coordinate command is a modal. The polar coordinate command mode when the power is turned ON is off (G15). Whether to initialize the modal at reset or not can be selected with the parameter (#1210 RstGmd/bit 11) setting.
- (4) During polar coordinate command mode, command the radius with the 1st axis for the selected plane, and the angle with the 2nd axis. For example, when the X-Y plane is selected, command the radius with the address "X", and the angle with the address "Y".
- (5) For the angle, the counterclockwise direction of the selected plane is positive and the clockwise direction is negative.
- (6) The radius and angle can be commanded with both the absolute value (G90) and incremental value (G91).
- (7) When the radius is commanded with the absolute value, command the distance from the zero point in the workpiece coordinate system (note that when the local coordinate system is set, command the distance in the local coordinate system).
- (8) When the radius is commanded with the incremental value command, considering the end point of the previous block as the polar coordinate center, command the incremental value from that end point. The angle is commanded with the incremental value of the angle from the previous block.
- (9) When the radius is commanded with the negative value, the same operation as the command that the radius command value is changed to the absolute value and 180° is added to the angle command value.

Commanded position

(1) When the zero point in the workpiece coordinate system is applied to the polar coordinate center

The zero point in the workpiece coordinate system is applied to the polar coordinate center by commanding the radius value with the absolute value.

Note that the zero point in the local coordinate system is applied to the polar coordinate center if the local coordinate system (G52) is used.

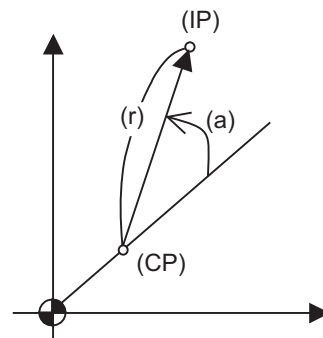
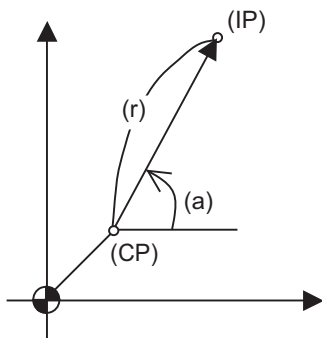


When the angle is the absolute value command

When the angle is the incremental value command

(2) When the present position is applied to the polar coordinate center

The present position is applied to the polar coordinate center by commanding the radius value with the incremental value.



When the angle is the absolute value command

When the angle is the incremental value command

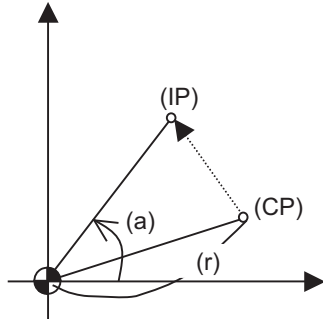
(CP) Current position (IP) Command position

(a) Angle (r) Radius

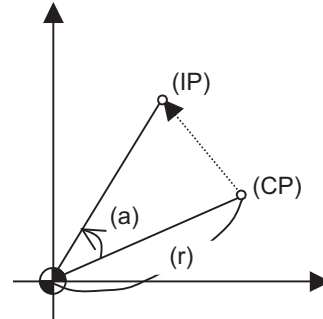
6 Interpolation Functions

(3) When the radius value command is omitted

When the radius value command is omitted, the zero point in the workpiece coordinate system is applied to the polar coordinate center, and the distance between the polar coordinate center and current position is regarded as the radius. Note that the zero point in the local coordinate system is applied to the polar coordinate center if the local coordinate system (G52) is used.



When the angle is the absolute value command



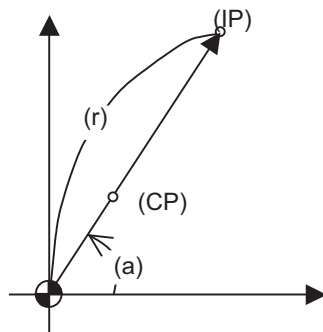
When the angle is the incremental value command

(4) When the angle command is omitted

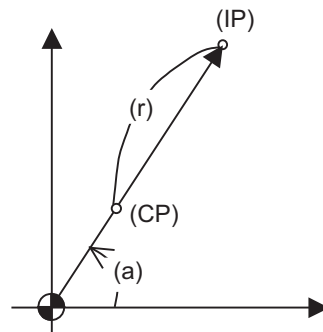
When the angle command is omitted, the angle of the present position in the workpiece coordinate system is applied to the angle command.

The zero point in the workpiece coordinate system is applied to the polar coordinate center by commanding the radius value with the absolute value. Note that the zero point in the local coordinate system is applied to the polar coordinate center if the local coordinate system (G52) is used.

If the radius value is commanded with the incremental value, the current position is applied to the polar coordinate center.



When the angle is the absolute value command



When the angle is the incremental value command

(CP) Current position (IP) Command position

(a) Angle

(r) Radius

Axis command not interpreted as polar coordinate command

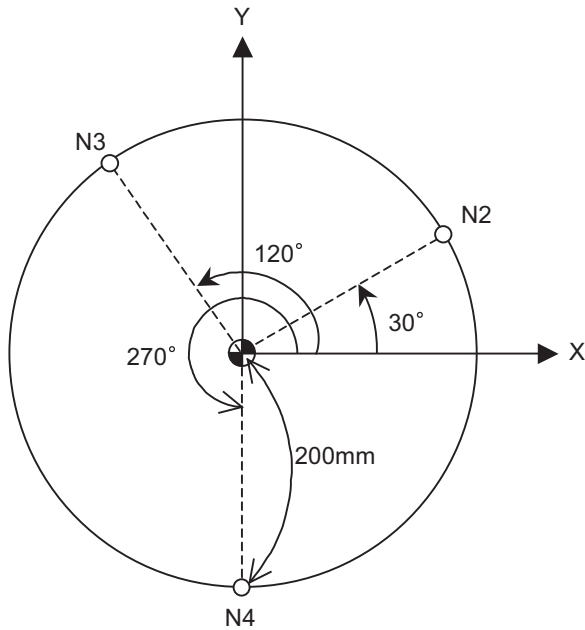
The axis command with the following command is not interpreted as the polar coordinate command during the polar coordinate command mode. The movement command that has no axes commands for the 1st axis and 2nd axis in the selected plane mode is also not interpreted as polar coordinate command during the polar coordinate command mode.

Function	G code
Dwell	G04
Parameter input by program/Compensation data input	G10
Local coordinate system setting	G52
Machine coordinate system setting	G92
Machine coordinate system selection	G53
Coordinate rotation by program	G68
Scaling	G51
G command mirror image	G51.1
Reference position check	G27
Reference position return	G28
Start position return	G29
2nd, 3rd, 4th reference position return	G30
Tool change position return 1	G30.1
Tool change position return 2	G30.2
Tool change position return 3	G30.3
Tool change position return 4	G30.4
Tool change position return 5	G30.5
Tool change position return 6	G30.6
Automatic tool length measurement	G37
Skip	G31
Multi-step skip 1-1	G31.1
Multi-step skip 1-2	G31.2
Multi-step skip 1-3	G31.3
Linear angle command	G01 Aa1



Program example

When the zero point in the workpiece coordinate system is the polar coordinate zero point



- The polar coordinate zero point is the zero point in the workpiece coordinate system.
- The plane is the X-Y plane.

(1) When the radius and angle are the absolute value command

N1 G17 G90 G16 ;

Polar coordinate command, X-Y plane selection

The polar coordinate zero point is the zero point in the workpiece coordinate system.

N2 G85 X200. Y30. Z-20. F200.

Radius 200mm, angle 30°

N3 Y120.

Radius 200mm, angle 120°

N4 Y270.

Radius 200mm, angle 270°

N5 G15 G80 ;

Polar coordinate command cancel

(2) When the radius is the absolute value command and the angle is the incremental value command

N1 G17 G90 G16 ;

Polar coordinate command, X-Y plane selection

The polar coordinate zero point is the zero point in the workpiece coordinate system.

N2 G85 X200. Y30. Z-20. F200.

Radius 200mm, angle 30°

N3 G91 Y90.

Radius 200mm, angle +90°

N4 Y150.

Radius 200mm, angle +150°

N5 G15 G80 ;

Polar coordinate command cancel



Precautions

- (1) If the high-accuracy commands are carried out during the polar coordinate command mode, or if the polar coordinate commands are carried out during the high-accuracy command mode, operations are performed depending on your machine's specifications.
Refer to "High-accuracy Control" and "High-speed High-accuracy Control" for details.
- (2) When the mirror image (G code/parameter/PLC signal) is canceled anywhere except at the mirror image center during the polar coordinate command mode, the absolute value and machine position will deviate. The mirror center is set with an absolute value, so if the mirror center is commanded again in this state, the center may be set to an unpredictable position. Cancel the mirror image above the mirror center or, after cancellation, assign a positioning command using absolute value command that the radius and angle of the polar coordinate command are designated.

6.14 Spiral/Conical Interpolation; G02.1/G03.1 (Type 1), G02/G03 (Type 2)



Function and purpose

This function carries out interpolation that smoothly joins the start and end points in a spiral. This interpolation is carried out for arc commands in which the start point and end point are not on the same circumference.

There are two types of command formats, and they can be switched with the parameters.



Command format

Spiral/conical interpolation (Type 1: #1272 ext08/bit2=0)

```
G17 G02.1/G03.1 X__Y__I__J__P__F__;
```

Spiral/conical interpolation (Type 2: #1272 ext08/bit2=1)

```
G17 G02/G03 X__Y__I__J__Q__L__K__F__;
```

G17	Arc plane
G02.1/G03.1 (Type 1)	Arc rotation direction (Type 1)
G02/G03 (Type 2)	Arc rotation direction (Type 2)
X Y	End point coordinates (Conical Interpolation when the axis other than arc plane axes is included.)
I J	Arc center
P (Type 1)	Number of pitches (number of spirals) (Type 1)
Q/L/K (Type 2)	Incremental-decremental amount of radius /Number of pitches (Number of spirals)/ Increment-decrement amount of height (Type 2)
F	Feedrate (tool path direction speed)

Circular interpolation operations are carried out at the f1 speed by the commands above.

The path is toward the end point, following a spiral arc path centered at the position designated by distance i (X axis direction) and distance j (Y axis direction) in respect to the start point.

(1) The arc plane is designated by G17, G18 and G19. (Common for type 1 and 2)

G17	XY plane
G18	ZX plane
G19	YZ plane

(2) The arc rotation direction is designated by G02.1 (G02) or G03.1 (G03). (Common for type 1 and 2)

G02.1/G02	Clockwise (CW)
G03.1/G03	Counterclockwise (CCW)

(3) The end point coordinates are designated with XYZ. (Common for type 1 and 2)

(Decimal point command is possible. Use mm (or inch) as the unit.)

When designation of arc plane axes is omitted, the coordinates of the start point are inherited.

If the axis other than arc plane axes is designated, conical interpolation is applied.

6 Interpolation Functions

- (4) The arc center is designated with IJK. (Common for type 1 and 2)
 (Decimal point command is possible. Use mm (or inch) as the unit.)
 I: Incremental designation in the X axis direction from the start point
 J: Incremental designation in the Y axis direction from the start point
 K: Incremental designation in the Z axis direction from the start point
 When either 1 axis of arc plane is omitted, the coordinates of the start point are inherited.

- (5) P designates the number of pitches (number of spirals). (Type 1)
 The number of pitches and rotations are as shown below.

Number of pitches (0 to 99)	Number of rotations
P0	Less than 1 rotation (Can be omitted.)
P1	1 or more rotation and less than 2 rotations
Pn	n or more rotation and less than (n+1) rotations

- (6) Q designates the increment/decrement amount of radius per spiral rotation. (Type 2)
 The number of spiral rotations when the radius increment/decrement amount is specified can be calculated with the following expression.

Number of rotations = | (arc end point radius - arc start point radius) / | increment or decrement amount of radius |

- (7) L designates the number of pitches (number of spirals). (Type 2) (range: 0 to 99)
 When omitted, L1 is designated.

The number of pitches and rotations are as shown below.

Number of pitches (0 to 99)	Number of rotations
L1	Less than 1 rotation
L2	1 or more rotation and less than 2 rotations
Ln	(n-1) or more rotations and less than n rotations

Q takes precedence over L if both Q and L have been designated at the same time.

- (8) K designates the increment or decrement amount of height per spiral rotation in conical interpolation. (Type 2)
 The increment or decrement amount of height is designated with I/J/K for the axis other than arc plane.
 The relation between increment or decrement amount of height and the rotation plane is as shown below.

Rotation plane	Increment or decrement amount of height
G18	J command
G19	I command
Other than G18/G19	K command

The number of rotations when the designation of increment or decrement amount of height is specified can be calculated with the following expression.

Number of rotations = Height / | Increment/decrement amount of height |

If Q, K and L have been designated at the same time, the order of precedence is Q>K>L.

Decimal point command is possible in the range of the increment or decrement amount of radius and height. Use mm (or inch) as the unit.



Detailed description

- (1) The arc rotation direction G02.1 is the same as G02, and G03.1 is the same as G03.
- (2) There are no R-designated arcs in spiral interpolation.
- (3) Conical cutting, tapered thread-cutting and other such machining operations can be conducted by changing the start point and end point radius and commanding the linear axis simultaneously.
- (4) Normally the spiral interpolation is automatically enabled with the arc commands (G02, G03) when the difference between the start point radius and the end point radius is less than the parameter setting value.
- (5) The axis combination that can be simultaneously commanded depends on the specifications. The combination within that range is arbitrary.
- (6) The feedrate is the constant tangent speed.
- (7) Simultaneous control by combining with tool radius compensation (G41, G42) is not possible.
- (8) The arc plane always follows G17, G18 and G19. The plane arc control is carried out by G17, G18 and G19, even if designated by two addresses that do not match the plane.
- (9) Conical interpolation
When an axis designation other than the spiral interpolation plane is simultaneously designated, other axes are also interpolated in synchronization with the spiral interpolation.
- (10) In the following cases, a program error will occur.

(a) Items common for type 1 and 2

Setting Items	Command range (unit)	Error
End point coordinates	Range of coordinate command (mm/inch) (Decimal point command is possible.)	<ul style="list-style-type: none"> ♦If a value exceeding the command range is issued, a program error (P35) will occur. ♦If an axis other than one which can be controlled with the command system is commanded, a program error (P33) will occur.
Arc center	Range of coordinate command (mm/inch) (Decimal point command is possible.)	<ul style="list-style-type: none"> ♦If a value exceeding the command range is issued, a program error (P35) will occur. ♦If an axis other than one which can be controlled with the command system is commanded, a program error (P33) will occur. ♦If rotation plane axis is not designated completely, a program error (P33) will occur.
Number of pitches	0 to 99	♦If a value exceeding the command range is issued, a program error (P35) will occur.
Feedrate	Range of speed command (mm/min, inch/min) (Decimal point command is possible.)	♦If a value exceeding the command range is issued, a program error (P35) will occur.

(b) Items for type 2 only

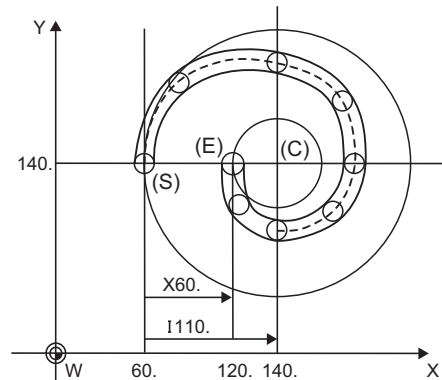
Setting Items	Command range (unit)	Error
Increment or decrement amount of radius	Range of coordinate command (mm/inch) (Decimal point command is possible.)	<ul style="list-style-type: none"> ♦If the sign of designated increment or decrement amount is opposite from that of the difference between the start point radius and the end point radius, a program error (P33) will occur. ♦If the end point position obtained from the speed and increment or decrement amount is larger than "SpiralEndErr (#8075)", a program error (P70) will occur.
Increment or decrement amount of height	Range of coordinate command (mm/inch) (Decimal point command is possible.)	<ul style="list-style-type: none"> ♦If the sign of designated increment or decrement amount is opposite from that of the movement direction of height, a program error (P33) will occur. ♦If the end point position obtained from the speed and increment or decrement amount is larger than "SpiralEndErr (#8075)", a program error (P70) will occur.
G02.1/G03.1		♦Program error (P34) will occur if G02.1/G03.1 are used during type 2.



Program example

(Example 1)

```
G91 G17 G01 X60. F500 ;
Y140. ;
G2.1 X60. Y0 I100. P1 F300 ;
G01 X-120. ;
G90
G17 G01 X60. F500 ;
Y140. ;
G2.1 X120. Y140. I100. P1 F300 ;
G01 X0 ;
```



(S) Start point

(E) End point

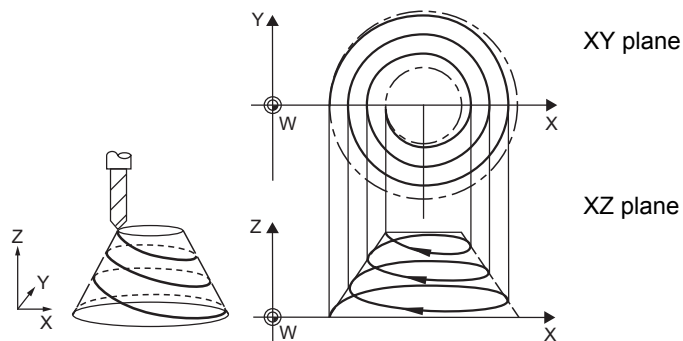
(C) Center

(Example 2)

```
G91 G17 G01 X60. F500 ;
Y140. ;
G02.1X60.0 Z100.0 I100. P1 F300 ; -> Because this is the G17 plane, arc control is not carried out by X-Z.
G01X-120 ; Arc control is carried out by X-Y.
```

(Example 3) In this example, the interpolation is truncated cone interpolation.

```
G17 G91 G02.1 X100.Z150. I150.P3 F500;
```



Relationship with other functions

(1) Items common for type 1 and 2

- ♦As the start point and end point are not on the same arc, a normal line control will not be applied correctly.
- ♦If there is no center command when geometric is valid, a program error (P33) will occur.

(2) Items for type 2 only

- ♦If the spiral interpolation command is issued during the mirror image, a program error (P34) will occur.
- ♦If the spiral interpolation command is issued during the scaling, a program error (P34) will occur.
- ♦If the spiral interpolation command is issued during the corner chamfering/corner rounding command, a program error (P33) will occur.

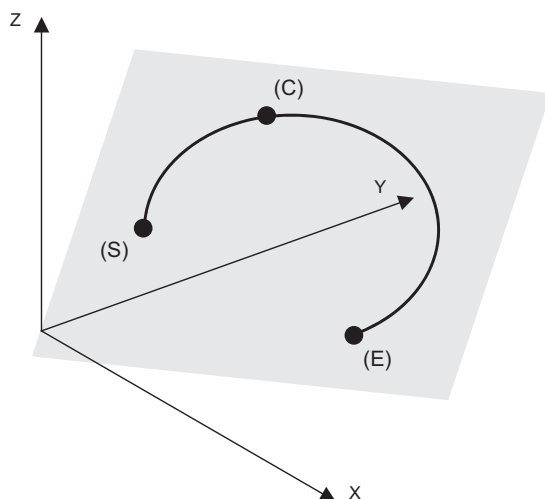
6.15 3-dimensional Circular Interpolation; G02.4, G03.4



Function and purpose

To issue a circular command over a 3-dimensional space, an arbitrary point (intermediate point) must be designated on the arc in addition to the start point (current position) and end point. By using the 3-dimensional circular interpolation command, an arc shape determined by the three points (start point, intermediate point, end point) designated on the 3-dimensional space can be machined.

The validity of this function depends on the MTB specifications. If this specification is invalid and the 3-dimensional circular interpolation command is issued, a program error (P39) will occur.



(S) Start point (Current position)
(E) End point
(C) Intermediate point



Command format

G02.4 (G03.4)	Xx1 Yy1 Zz1 αα1 ; Intermediate point designation (1st block)
	Xx2 Yy2 Zz2 αα2 ; End point designation (2nd block)

G02.4 (G03.4)	3-dimensional circular interpolation command (Cannot designate the rotation direction)
x1, y1, z1	Intermediate point coordinates
x2, y2, z2	End point coordinates
α	Arbitrary axis other than axis used as the reference (X,Y,Z) in 3-dimensional circular interpolation (Cannot be omitted.)

- (1) The G02.4 and G03.4. operations are the same. (Cannot designate the rotation direction)
- (2) The axes used as the reference in 3-dimensional circular interpolation are the three basic axes set with the parameters.
- (3) The X, Y, Z address in the block may be omitted. The intermediate point coordinates omitted in the 1st block become the start point coordinates, and the end point coordinates omitted in the 2nd block become the intermediate point coordinates.
- (4) When using the 3-dimensional circular interpolation command, an arbitrary axis can be commanded in addition to the orthogonal coordinate system (X, Y, Z) used as the reference. The arbitrary axis designated in the intermediate point designating block (1st block) will interpolate to the command point when moving from the start point to intermediate point. The arbitrary axis designated in the end point command block (2nd block) will interpolate to the command point when moving from the intermediate point to the end point. The number of arbitrary axes that can be commanded differs according to the number of simultaneous contour control axes. The total of the basic three axes used as the reference of the 3-dimensional circular interpolation and the arbitrary axes commanded simultaneously must be less than the number of simultaneous contour control axes.



Detailed description

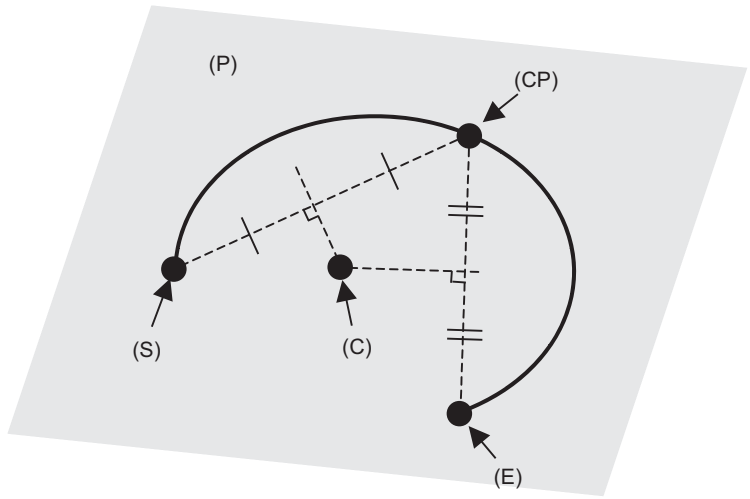
Designating intermediate point and end point

When using the 3-dimensional circular interpolation command, an arc that exists over the 3-dimensional space can be determined by designating the intermediate point and end point in addition to the start point (current position) (Refer to following figure). So according to the command format, it is necessary to designate an intermediate point in the 1st block and an end point in the 2nd block. If only one block is designated, a program error (P74) will occur.

Liner interpolation is applied when the end point match the start point in the 3-dimensional circular interpolation command (Refer to "When liner interpolation is applied"). Thus, a true circle (360-degree rotation) cannot be designated in the 3-dimensional circular interpolation.

In addition, designate the intermediate point in the middle of a start point and an end point. If the intermediate point is near the start point or the end point, arc accuracy may fall.

[Designation of arc in 3-dimensional space]



- (P) Plane including start point, intermediate point and end point
- (S) Start point (Current position)
- (E) End point
- (CP) Intermediate point
- (C) Center

As shown in the above figure, when three points (start point, intermediate point, end point) are specified on 3-dimensional space, arc center coordinates can be obtained. An arc center cannot be obtained if only two points are specified, and a liner interpolation is applied.

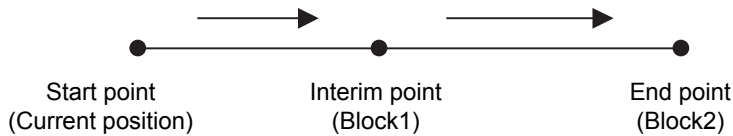
If the intermediate point is near the start point or the end point, an error may occur when calculating arc center.

When liner interpolation is applied

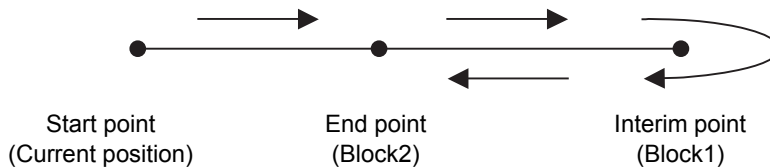
In the following case, liner interpolation is applied without executing 3-dimensional circular interpolation.

- (1) When the start point, intermediate point, and end point are on the same line (refer to the following figure)
(If the end point exists between the start point and intermediate point, axes move in the order of start point, intermediate point, and end point.)
- (2) When two of the start point, intermediate point and end point match
(Liner interpolation is applied even if the end point matches the start point to command true circle. When the start point matches the end point, axes move in order of the start point, an intermediate point, and an end point.)

[When liner interpolation is applied]



When the three points are on the same line, liner interpolation is applied.



Even if the end point exists between the start point and intermediate point, move in the order of start point, intermediate point, and end point.

Modal command

The 3-dimensional circular interpolation command G02.4 (G03.4) is a modal command belonging to 01 group. Therefore, the command will remain valid until the other G command in the 01 group is issued. When the 3-dimensional circular interpolation command is carried out continuously, the end point of the present command is the start point of the next command.



Relationship with other functions

Commands that cannot be used

G code command which leads to a program error during 3-dimensional circular interpolation modal

G code	Function name	Program error
G05 Pn	High-speed machining mode	P34
G05 P10000	High-speed high-accuracy control II	P34
G05.1 Q0/G05.1 Q1	High-speed high-accuracy control I	P34
G07.1	Cylindrical interpolation	P485
G12/G13	Circular cutting CW/CCW	P75
G12.1	Polar coordinate interpolation	P485
G16	Polar coordinate command	P75
G41/G42	Tool radius compensation	P75
G41/G42	3-dimensional tool radius compensation	P75
G41.1/G42.1	Normal line control	P75
G43/G44	Tool length compensation	P75
G51	Scaling	P75
G51.1	Mirror image	P75
G66/G66.1	User macro	P75
G67	User macro	P276
G68	Coordinate rotation by program	P75
G68	3-dimensional coordinate conversion	P921
G73/G74/G76/G81/G82 G83/G84/G85/G86/G87 G88/G89	Fixed cycle	P75

G code command which leads to a program error when 3-dimensional circular interpolation is commanded

G code modal	Function name	Program error
G05 Pn	High-speed machining mode	P34
G05 P10000	High-speed high-accuracy control II	P34
G05.1 Q1	High-speed high-accuracy control I	P34
G07.1	Cylindrical interpolation	P481
G12.1	Polar coordinate interpolation	P481
G16	Polar coordinate command	P75
G41/G42	Tool radius compensation	P75
G41/G42	3-dimensional tool radius compensation	P75
G41.1/G42.1	Normal line control	P75
G43/G44	Tool length compensation	P75
G51	Scaling	P75
G51.1	Mirror image	P75
G66/G66.1	User macro	P75
G68	Coordinate rotation by program	P75
G68	3-dimensional coordinate conversion	P922

Functions that cannot be used

If following functions are used in 3-dimensional circular interpolation, alarm will occur.

- Chopping
- Macro interruption
- Mirror image by parameter setting
- Mirror image by external input
- Corner chamfering / corner R

Restrictions may be added for other functions. Refer to explanation of each function.

**Precautions**

- (1) If single block is valid and this command is operated, a block stop is carried out at an intermediate point and the end point.
- (2) The speed command during 3-dimensional circular interpolation is the tangent speed on arc.
- (3) When 3-dimensional circular interpolation is commanded while incremental command is valid, the relative position of the intermediate point in respect to the start point is designated in the intermediate point designation block, and the relative position of the end point in respect to the intermediate point is designated in the intermediate point designation block.
- (4) The path of 3-dimensional circular interpolation during graphic check is drawn as linear at each range from start point to intermediate point and from intermediate point to end point.

6.16 NURBS Interpolation; G06.2



Function and purpose

This function realizes NURBS (Non-Uniform Rational B-Spline) curve machining by simply commanding NURBS curve parameters (stage, weight, knot, control point), which is used for the curved surface/line machining, without replacing the path with minute fine segments.

This function operates only in the high-speed high-accuracy control II/III mode, therefore, the high-speed high-accuracy control II/III function is also required as the specification.

Note

(1) High-speed high-accuracy control III functions as high-speed high-accuracy control II while NURBS interpolation is ON.

However, if the curvature is large, the speed is clamped so that the machine's tolerable acceleration rate is not exceeded.



Command format

NURBS interpolation start

```
G06.2 Pp Kk1 X1 Yy1 Zz1 Rr1 Ff;
      Kk2 Xx2 Yy2 Zz2 Rr2;
      Kk3 Xx3 Yy3 Zz3 Rr3;
      Kk4 Xx4 Yy4 Zz4 Rr4;
      :
      Kkn Xxn Yyn Zzn Rrn;
      Kkn+1;
      Kkn+2;
      Kkn+3;
      Kkn+4;
```

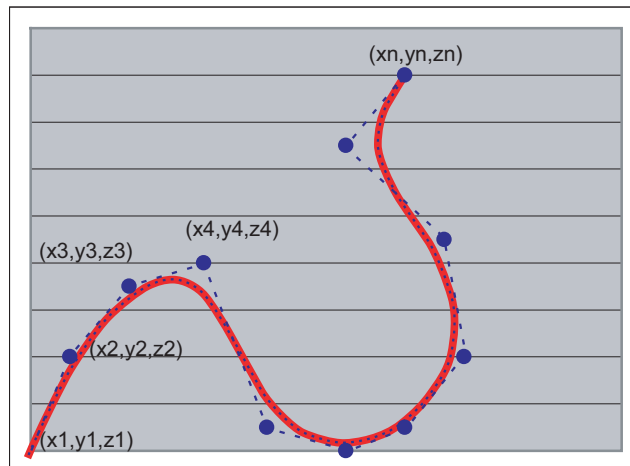
Pp	Set the stage of the NURBS curve. Designate in the same block as G06.2 command. The NURBS curve of the stage p will be (p-1)th curve. When omitted, Pp means the same as P4. (Example) P2: Primary curve (linear)
Kkn	Knot Set the knot for each NURBS interpolation block. Set the same value for the knot in the 1st block to the stage p block. NURBS interpolation is terminated if there is a block exclusively with knot.
Xxn Yyn Zzn	Control point coordinate value. Designate the same coordinate value for the 1st block control point as that designated right before NURBS interpolation.
Rrn	Control point weight. Set the weight of each NURBS interpolation control point.
Ff	Interpolation speed (Can be omitted)



Detailed description

- (1) Designate the stage P for the 1st block of NURBS interpolation.
- (2) Designate the same coordinate value for the 1st block control point of NURBS interpolation as that designated right before NURBS interpolation.
- (3) Designate all axes to be used in the subsequent NURBS interpolation blocks for 1st block of NURBS interpolation.
- (4) Set the same value for knot K from the 1st block of NURBS interpolation to setting value block of the stage P.
- (5) Command knot K exclusive block of the same number as the setting value of the stage P for terminating NURBS interpolation.

At this time, set the same value for knot K setting.



- Passes through control point
- NURBS interpolation curve

Note

- (1) If an exclusive knot is commanded immediately after NURBS interpolation, NURBS interpolation mode is active again.

An exclusive knot that is commanded immediately after NURBS interpolation is the same meaning as following command.

G06.2 Pp Km Xxn Yyn Zzn R1.0



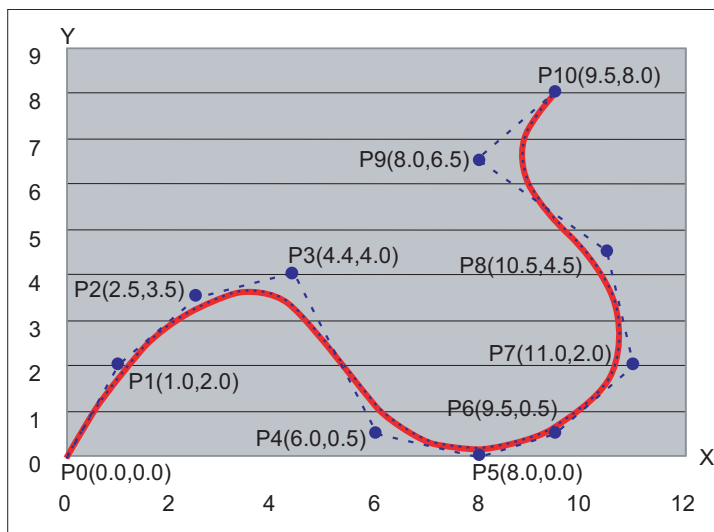
Program example

The example of program that has 4 stages (cubic curve) and 11 control points is shown below.

Control point	P0	P1	P2	P3	P4	P5	P6	P7	P8	P9	P10				
Knot	0.0	0.0	0.0	0.0	1.0	2.0	3.0	4.0	5.0	6.0	7.0	8.0	8.0	8.0	8.0

```

:
:
G05 P10000;                               High-speed high-accuracy control II mode ON
G90 G01 X0. Y0. Z0. F300 ;
G06.2 P4 X0. Y0. R1. K0 ;                 P0     NURBS interpolation ON
X1.0 Y2.0 R1. K0 ;                       P1
X2.5 Y3.5 R1. K0 ;                       P2
X4.4 Y4.0 R1. K0 ;                       P3
X6.0 Y0.5 R1. K1 ;                       P4
X8.0 Y0.0 R1. K2 ;                       P5
X9.5 Y0.5 R1. K3 ;                       P6
X11.0 Y2.0 R1. K4 ;                      P7
X10.5 Y4.5 R1. K5 ;                      P8
X8.0 Y6.5 R1. K6 ;                       P9
X9.5 Y8.0 R1. K7 ;                       P10
K8;
K8;
K8;
K8;                                       NURBS interpolation OFF
G05 P0;                                   High-speed high-accuracy control II mode OFF
:
:
    
```



-● Passes through control point
- NURBS interpolation curve



Relationship with other functions

G code/Feed/Miscellaneous functions

All the G code, feedrate and MSTB code cannot be set during NURBS interpolation.

However, when the fixed cycle G code is commanded in the same block where G06.2 is commanded, the fixed cycle G code is ignored.

If a command other than the axis address designated in the 1st block of NURBS interpolation, R and K is commanded, a program error will occur.

Data format

- (1) Optional block skip "/"
Cannot be set in the NURBS interpolation 2nd block or after.
- (2) Control IN ("and Control OUT ")
Cannot be set in the NURBS interpolation 2nd block or after.
- (3) Local variables and common variables
Can be referred but cannot be set in the NURBS interpolation. Setting the variables causes a program error (P29).
- (4) System variable
Cannot be referred nor set in the NURBS interpolation; a program error (P29) will occur.

Interruption/restart

The validity of program interruption/restart is shown below.

Type	During NURBS interpolation
Single block	Valid (*1)
Feed hold	Enabled
Resetting	Valid (*2)
Program stop	Disabled
Optional stop	Disabled
Manual interruption	Invalid (*3)
MDI interruption	Disabled
Restart search	Disabled
Macro interruption	Invalid (*4)
PLC interruption	Invalid (*5)

(*1) A single block stop is carried out at the last control points only.

The single block stop is not applied during NURBS interpolation.

(*2) NURBS interpolation mode is canceled with Reset (Reset1/Reset2/Reset&Rewind).

(*3) The operation differs according to the manual absolute signal status.

- ◆When the manual absolute signal OFF,
NURBS interpolation is carried out in the state where axis-coordinate system is shifted by the manual absolute movement amount.
- ◆When the manual absolute signal ON
Upon automatic start after manual interruption, a program error (P554) will occur after moving by the remaining distance.
Note that the operation can run continually by returning the axis to the original position after manual interruption.

(*4) "Macro interrupt" signal (UIT) is ignored.

(*5) "PLC interrupt" signal (PIT) is ignored.

Graphic check

NURBS interpolation cannot be applied during graphic check (continuous/step check).
Linear interpolation that connects the control points is applied during graphic check.

High-accuracy Control in 2 part Systems

With the high-accuracy control in 2 part systems specification, NURBS interpolation can be commanded by 1st and 2nd part systems.

**Precautions**

- (1) Target axes for NURBS interpolation are 3 basic axes.
- (2) Command the control point for all the axes for which NURBS interpolation is carried out in the 1st block (G06.2 block). A program error (P32) will occur if an axis which was not commanded in the 1st block is commanded in the 2nd block or after.
- (3) The first control point (G06.2 block coordinate value) should be commanded as the start point of the NURBS curve. Thus, the start point of the NURBS curve should be commanded to match the end point of the previous block. A program error will occur if the points do not match. (P552)
- (4) The command range of the weight is 0.0001 to 99.9999. If "1" is commanded, the resulting command will be equal to "1.0".
If more than 5 digits are commanded after the decimal point, a program error (P33) will occur.
- (5) The knot command cannot be omitted, and must be commanded in each block. A program error (P33) will occur if omitted.
- (6) As with knot, in the same manner as weight, up to 4 digits can be commanded after the decimal point. Even if the decimal point is omitted, the value will be handled as the one with a decimal point. If "1" is commanded, the result will be the same as "1.0".
If more than 5 digits are commanded after the decimal point, a program error (P33) will occur.
- (7) As with knot, command the same or greater value than the previous block. If a smaller value than previous block is set, a program error (P551) will occur.
- (8) NURBS interpolation cannot be applied during graphic check (continuous/step check).
Linear interpolation that connects the control points is applied during graphic check.
- (9) NURBS interpolation mode is canceled with Reset (Reset1/Reset2/Reset&Rewind).

(10) NURBS interpolation can be commanded in only the following modes. If NURBS interpolation is commanded in other than the following modes, the program error (P29) will occur.

Type	Mode in which NURBS interpolation can be commanded
G group 0	High-speed high-accuracy control II (G05 P10000) High-speed high-accuracy control III (G05 P20000)
G group 5	Asynchronous feed (G94)
G group 7	Tool radius compensation cancel (G40)
G group 8	Tool length offset +/- (G43/G44) Tool length offset cancel (G49)
G group 9	Fixed cycle cancel (G80)
G group 11	Scaling cancel (G50)
G group 13	High-accuracy control ON (G61.1) Cutting mode (G64)
G group 14	User macro modal call cancel (G67)
G group 15	Normal line control cancel (G40.1)
G group 16	Programmable coordinate rotation mode OFF /3-dimensional coordinate conversion mode OFF (G69)
G group 17	Constant surface speed control OFF (G97)
G group 18	Polar coordinate command OFF (G15)
G group 19	G command mirror image cancel (G50.1)
G group 21	Polar coordinate interpolation cancel (G13.1)
-	Not during the coordinate rotation by parameter
-	Not during the mirror image by parameter setting
-	Not during the mirror image by external input

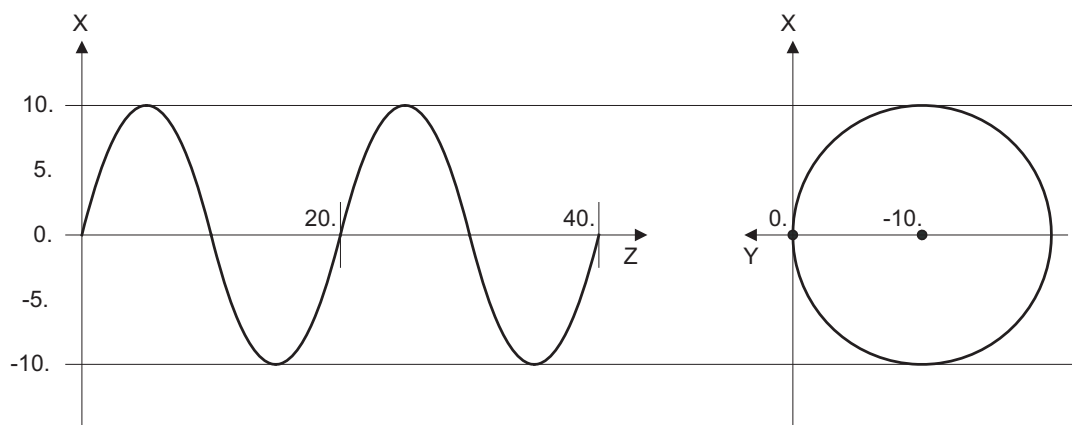
6.17 Hypothetical Axis Interpolation; G07



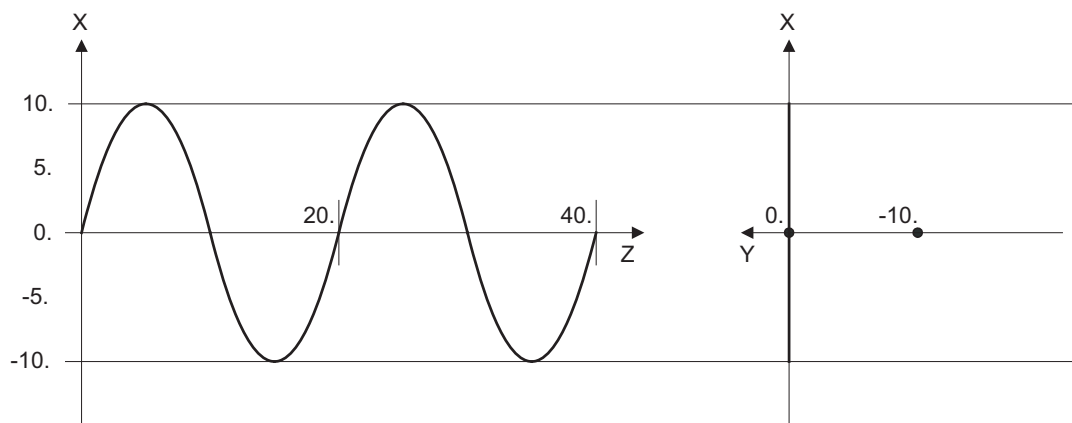
Function and purpose

Take one of the axes of the helical interpolation or spiral interpolation, including a linear axis, as a hypothetical axis (axis with no actual movement) and perform pulse distribution. With this procedure, an interpolation equivalent to the helical interpolation or spiral interpolation looked from the side (hypothetical axis), or SIN or COS interpolation, will be possible.

Normal helical interpolation



Helical interpolation in the hypothetical axis interpolation mode



To perform the SIN interpolation on Z-X plane, execute the helical interpolation (Y-X plane: G17 G02) with Y axis, which is designated as the hypothetical axis. The hypothetical axis does not make any actual movement.



Command format

G07 α0 ; ... Hypothetical axis interpolation mode ON

G07 α1 ; ... Hypothetical axis interpolation mode cancel

α	Axis name for which hypothetical axis interpolation is performed.
---	---



Detailed description

- (1) During "G07 α 0 ;" to "G07 α 1 ;", α axis will be the hypothetical axis.
- (2) Any axis among the NC axes can be designated as the hypothetical axis.
- (3) Multiple axes can be designated as the hypothetical axis.
- (4) The number other than "0" (hypothetical axis interpolation mode ON) or "1" (cancel) is commanded, it will be handled as "1" (cancel). However, when only the axis name is designated without a number, it will be handled as "0" (mode ON).



Program example

N01 G07 Y0 ;	Y axis is handled as hypothetical axis.
N02 G17 G02 X0. Y0. Z40. I0. J-10. P2 F50;	SIN interpolation is executed on X-Z plane.
N03 G07 Y1 ;	Y axis is returned to the actual axis.



Precautions

- (1) Interpolation functions that are used for hypothetical axis interpolation are helical interpolation and spiral interpolation.
- (2) Cancel the hypothetical axis interpolation before the high-speed high-accuracy control II (G05P10000) is commanded.
- (3) The hypothetical axis interpolation is valid only in the automatic operation. It is invalid in the manual operation mode. Handle interruption is valid even for the hypothetical axis, that is, axis will move by the interrupted amount.
- (4) Movement command for the hypothetical axis will be ignored. The feedrate will be distributed in the same manner as actual axis.
- (5) The protection functions such as interlock or stored stroke limit are valid for the hypothetical axis.
- (6) Even when the hypothetical axis is applied for the hypothetical axis again, no error will occur and the hypothetical mode will be continued.
- (7) When the hypothetical axis cancel is commanded to the actual axis, no error will occur and the axis remains as the actual axis.
- (8) The hypothetical axis will be canceled by carrying out the reset 2 or reset & rewind.

6.18 Involute Interpolation; G02.2/G03.2



Function and purpose

Involute interpolation moves a tool along an involute curve. This can be used for scroll machining of involute gears or compressors, and smooth accurate machining can be performed without stepping of path from the command by fine segment or without acceleration/deceleration by segment length.

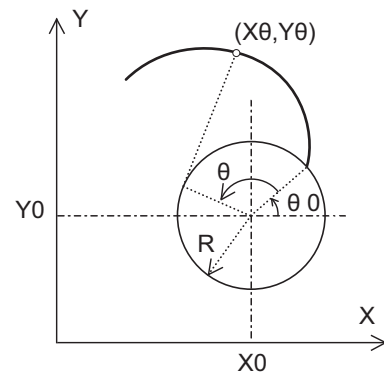
More accurate machining can be performed by using the automatic speed control function for the speed such as "involute interpolation override" and "acceleration clamping during involute interpolation".

Involute curve is obtained with the following expression:

$$X(\theta) = R\{\cos(\theta + \theta_0) + \theta \sin(\theta + \theta_0)\} + X_0$$

$$Y(\theta) = R\{\sin(\theta + \theta_0) - \theta \cos(\theta + \theta_0)\} + Y_0$$

Circle in the right figure is the base circle.



Command format

G02.2 (G03.2) X__ Y__ I__ J__ R__ F__; **G17 plane**

G02.2 (G03.2) Z__ X__ K__ I__ R__ F__; **G18 plane**

G02.2 (G03.2) Y__ Z__ J__ K__ R__ F__; **G19 plane**

G02.2/G03.2	Involute curve rotation direction (G02.2: clockwise; G03.2: counterclockwise)
X	End point of involute interpolation (X axis)
Y	End point of involute interpolation (Y axis)
Z	End point of involute interpolation (Z axis)
I	Distance between start point and center of base circle (X axis)
J	Distance between start point and center of base circle (Y axis)
K	Distance between start point and center of base circle (Z axis)
R	Base circle radius
F	Feedrate (in involute curve tangent direction)

The range of command value follows the input range of coordinate position data.



Detailed description

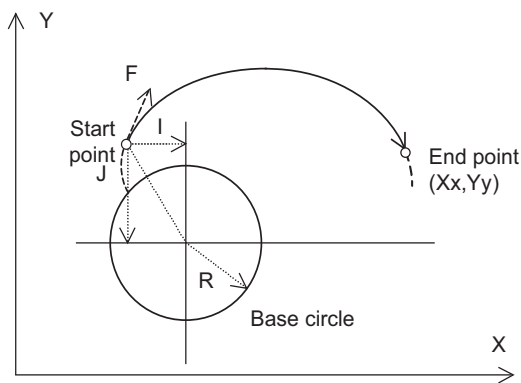
The following information is required to perform the involute interpolation:

- (1) Rotation direction
- (2) Start point of involute interpolation and end point coordinate
- (3) Base circle radius and center coordinate
- (4) Feedrate

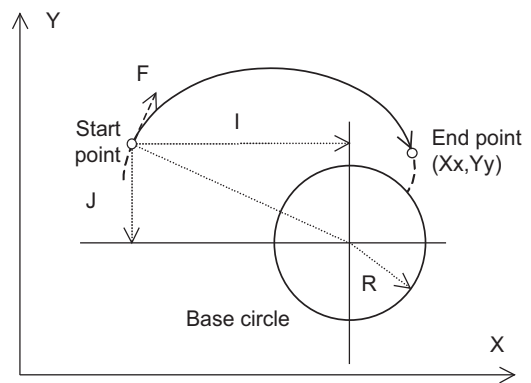
Details of operation

The involute interpolation is performed on the selected plane. Plane selection is same as the circular command.

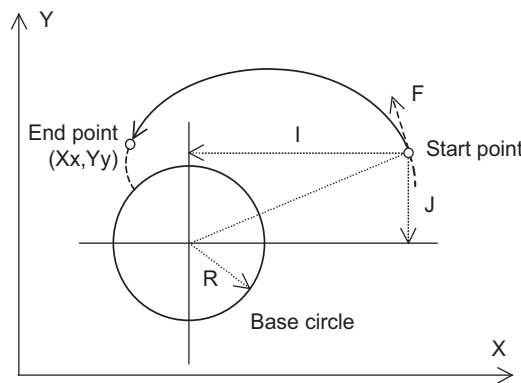
Operation example (G17 plane)



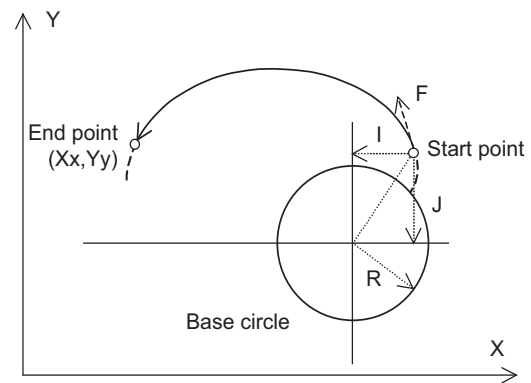
G02.2 (When going away from the base circle)



G02.2 (When closing to the base circle)



G03.2 (When closing to the base circle)

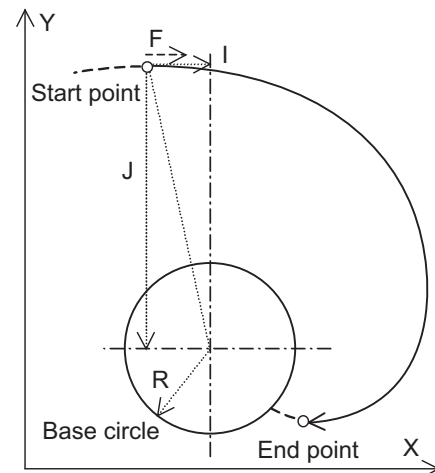


G03.2 (When going away from the base circle)

The program example of the absolute value command on G17 plane is as below:

G02.2 X223. Y34. I92. J-416. R100. F500;

Start point	(50,550)
End point	223, 34
Distance to the center of base circle (I, J)	92, -416
Base circle radius R	100
Feedrate (in curve tangent direction)	500



Rotation direction

Command the rotation direction with G02.2 (CW) or G03.2 (CCW). Plane selection (G17/G18/G19) and rotation direction (clockwise/counterclockwise) are the same as circular interpolation (G02/G03).

Start point and end point of involute interpolation

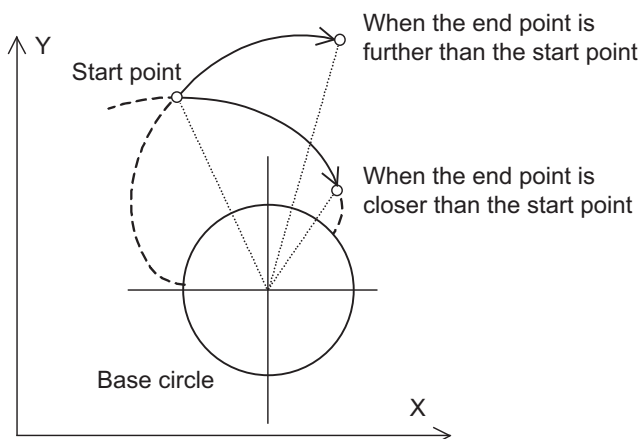
The start point of the involute interpolation is the current position.

The end point coordinates are commanded by X and Y (or Z). The command value is either absolute or incremental based on G90/G91.

There are two involute curves passing the start point, and one of them is selected as follows:

- When the end point is closer to the center of base circle than the start point, the involute curve moves toward the base circle.
- When the end point is further from the center of base circle than the start point, the involute curve moves away from the base circle.
- When the start point and the end point are at an even distance from the center of base circle, the program error (P71) occurs because the direction of involute curve cannot be decided.
- When the end point is not commanded, the program error (P33) occurs.
- When either the start point or the end point is inside the base circle, the program error occurs (P71) because the involute curve cannot be created.

The following figure shows the positional relationship between the start point and the end point:



Base circle's radius and center coordinate

Command the base circle radius with R. The command value is always issued with a positive value. When the command value is "0" or negative, the program error (P33) occurs. Command the center coordinate of base circle by I and J (or K). The command value is always issued with the incremental value from the start point regardless of G90/G91.

Command I and J (or K) with a sign depending on the direction of base circle center as seen from the start point. When I and J (or K) is not commanded or the both are "0" (equal to the start point), the program error (P33) occurs.

Feedrate

Command the feedrate with "F". The speed direction is the involute curve tangent direction.

Tool radius compensation

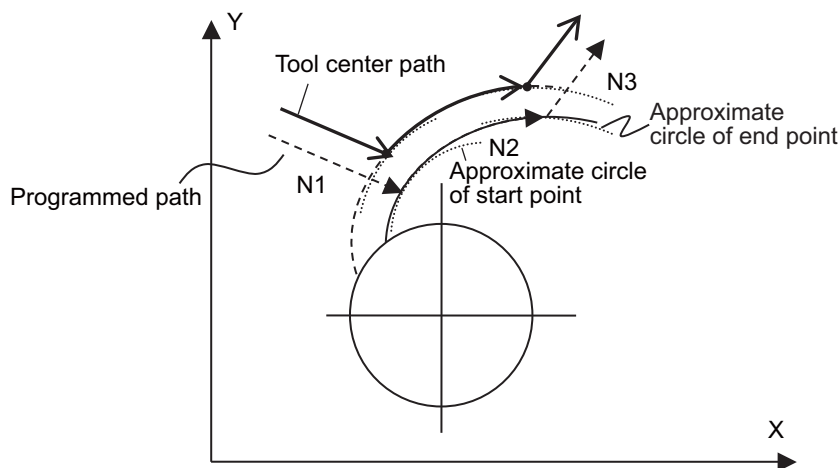
Tool radius compensation can be commanded even during the involute interpolation.

Command the compensation with G40/G41/G42 in the same way as in circular interpolation.

- G40: Cancel tool radius compensation.
- G41: Compensate the left side facing forward.
- G42: Compensate the right side facing forward.

For tool radius compensation of involute interpolation, the intersection points obtained using the approximated circular command are regarded as the start and end points, and the interpolation is performed on the involute curve that connects the obtained intersection points.

Command the feedrate with "F". The speed is in the involute curve tangent direction.



When the start point or the end point is inside the base circle as a result of compensation, the involute curve cannot be created. Therefore, even when the start point and the end point are outside the base circle before compensation, the program error (P71) occurs. The speed at the cutting point varies depending on the curvature radius even when the feedrate of the tool center path is constant because the curvature of involute curve is not constant. This tendency is more obvious near the base circle, because the curvature radius is smaller there. The speed at the cutting point can be constant using the involute interpolation override function described later. When the tool radius compensation is started or canceled in the involute interpolation modal, the program error (P151) occurs. During the involute interpolation, interference avoidance cannot be performed. Even when the parameter "#8102 COLL. ALM OFF" is "1", the program error (P153) occurs if an interference occurs.

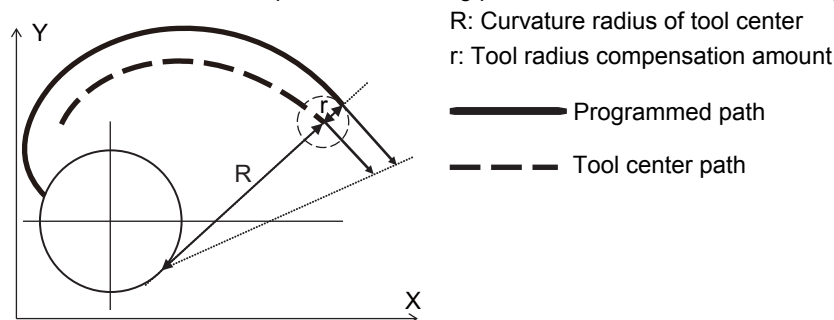
Automatic speed control

The following two types of automatic speed control functions can be used during the involute interpolation, which can improve machining accuracy.

[Involute interpolation override]

For the tool radius compensation, the feedrate is commanded as the speed on the tool center path. However, the curvature of involute curve is inconstant, thus even when the feedrate on the tool center path is consistent, the speed at the cutting point varies depending on the curvature radius. This tendency is more obvious near the base circle, because the curvature radius is smaller there.

"Involute interpolation override" function applies override to the speed on the tool center path according to the curvature radius so that the speed at the cutting point becomes the commanded speed.



Override value is calculated as follows:

Tool radius compensation	Override
Inside	$R / (R + r)$
Outside	$R / (R - r)$

- (1) When the tool radius compensation is inside, the override may be considerably small near the base circle. The lower limit value of override can be set by "#1558 IvOMin" and the tool center speed is controlled so as not to be below the lower limit value of override. (The parameter setting depends on the MTB specifications.)
- (2) When "0" is set in "#1558 IvOMin", the involute interpolation override function is invalid and the override during the tool radius compensation is always 100%.
- (3) Note that when the tool radius compensation is outside, the actual travel speed becomes larger than the commanded speed. Even in this case, the travel speed does not exceed the cutting feed clamp speed of each axis.

6 Interpolation Functions

[Involute interpolation acceleration clamp]

In the involute interpolation, even when the tool moves at a constant speed in the tangential direction, the acceleration increases because the curvature radius becomes smaller near the base circle. To prevent the excessive load on the machine, the speed in the tangent direction can be clamped to be at the acceleration set by the parameter or below according to the curvature radius in high-accuracy control mode.

The allowable acceleration is determined by the following expression:

$$(\text{Allowable acceleration}) = (\text{Maximum speed}) / (\text{Involute interpolation allowable acceleration})$$

The clamp speed in the tangent direction is obtained by the following expression:

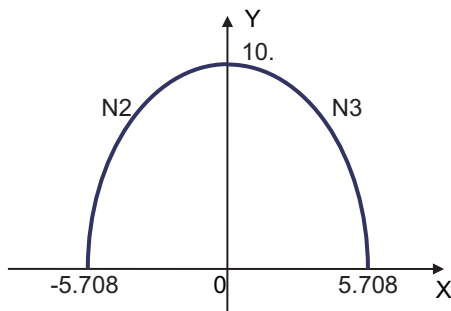
$$(\text{Clamp speed}) = \sqrt{(\text{Curvature radius}) \times (\text{Allowable acceleration})}$$

(Example) In the case of the following conditions:

Maximum speed "#1206 G1bF": 30000 (mm/min)

Involute interpolation allowable acceleration "#1559 IvAMax": 600 (ms)

Involute interpolation minimum feedrate "#1560 IvFMin": 1000 (mm/min)



```

:
G00 X-5.708 Y0.;
N1 G05P10000;
N2 G02.2 X0. Y10. I15.708 J10. R10. F3000;
N3 G02.2 X5.708 Y0. I-10. J0. R10.;
G05P0;
:

```

Allowable acceleration is as follows:

$$30000 (\text{mm/min}) / 600 (\text{ms}) \approx 833.333 (\text{mm/s}^2)$$

The clamp speed at the position of curvature radius 2 mm is as follows:

$$\sqrt{2 (\text{mm}) \times 833.333 (\text{mm/s}^2)} \approx 2449.489 (\text{mm/min})$$

- (1) When "0" is set in "#1560 IvFMin", the involute interpolation acceleration clamp function is invalid and the speed in the tangent direction is constant regardless of the curvature radius.
- (2) The clamp speed may be extremely small (or "0") near the base circle. The lower limit value of clamp speed can be set by "#1560 IvFMin" and the speed in the tangent direction is controlled so as not to be below the lower limit value of clamp speed.
- (3) Note that the speed is nearly "0" when "0" is set in "#1560 IvFMin" and the curvature radius is extremely small.



Relationship with other functions

Commands which cannot be issued in involute interpolation modal

When the following is commanded in involute interpolation modal, the program error occurs.

G code command	G group	Function
G05.1 Q2	0	Fine spline
G31	0	Skip
G31.1, G31.2, G31.3	0	Multi-step skip 1 to 3
G34, G35, G36, G37.1	0	Special fixed cycle
G37	0	Automatic tool length measurement
G38	0	Tool radius compensation vector designation
G39	0	Tool radius compensation corner arc
G45, G46, G47, G48	0	Tool position offset
G53.1	0	Tool axis direction control
G60	0	Unidirectional positioning
G40 (*1)	7	Tool radius compensation cancel
G41, G42	7	Tool radius compensation/3-dimensional tool radius compensation
G41.2, G42.2	7	3-dimensional tool radius compensation (Tool's vertical-direction compensation)
G43, G44	8	Tool length offset
G43.4, G43.5	8	Tool center point control
G49 (*2)	8	Tool length offset cancel
G51	11	Scaling ON
G63	13	Tapping mode
G63.1, G63.2	13	Synchronous tap mode
G41.1, G42.1 G151, G152	15	Normal line control
G68	16	Coordinate rotation ON 3-dimensional coordinate conversion ON
G68.2, G68.3	16	Inclined surface machining command
G69	16	Coordinate rotation cancel 3-dimensional coordinate conversion cancel
G96	17	Constant surface speed control ON
G16	18	Polar coordinate command ON
G51.1	19	Mirror image by G code ON
G7.1, G107	21	Cylindrical interpolation
G12.1, G112	21	Polar coordinate interpolation ON

When the following function is used in involute interpolation modal, the program error occurs.

- ♦Corner rounding
- ♦Corner chamfering

(*1) G40 in the tool radius compensation mode causes the program error, but does not cause the error in the radius compensation cancel mode.

(*2) G49 in the tool length offset mode causes the program error, but does not cause the error in the tool length offset cancel mode.

Mode in which involute interpolation cannot be commanded

When the involute interpolation (G02.2, G03.3) is commanded in the following modes, the program error occurs.

G code command	G group	Function
G05.1 Q2	0	Fine spline
G10	0	Parameter input by program/Compensation data input
G41.2, G42.2	7	3-dimensional tool radius compensation (Tool's vertical-direction compensation)
G43.4 G43.5	8	Tool center point control
G51	11	Scaling
G63	13	Tapping mode
G63.1, G63.2	13	Synchronous tap mode
G41.1, G42.1 G151, G152	15	Normal line control
G68	16	Coordinate rotation 3-dimensional coordinate conversion
G96	17	Constant surface speed control
G16	18	Polar coordinate command
G51.1	19	Mirror image by G code ON
G7.1, G107	21	Cylindrical interpolation
G12.1, G112	21	Polar coordinate interpolation

When the involute interpolation (G02.2, G03.3) is commanded while the functions below are used, the program error occurs.

- Mirror image by parameter setting



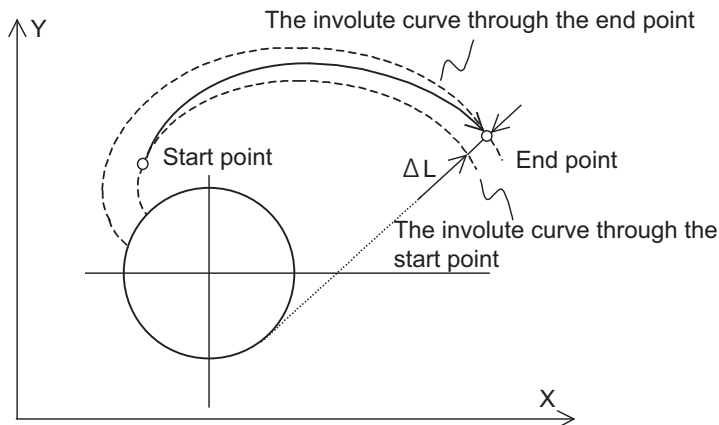
Precautions and restrictions

- (1) This function does not support SSS control. Even when "#8090 SSS ON" is ON, the involute block operates with SSS control OFF. It decelerates once before and after the involute block.
- (2) Even when "#1572 cirorp" is ON, the overlap is not performed between a linear or arc block and involute block.
- (3) Involute interpolation can be commanded in high-speed machining mode or high-speed high-accuracy mode, but does not support these modes. The fine segment capacity is the same as the speed in the normal mode.
- (4) This function does not support graphic check.

End point error

Note that if the end point is not on the involute curve that passes through the start point, the following operation results:

- ◆When the error ΔL is greater than "#8077 Involute error", the program error (P70) occurs at the start point of the involute interpolation.
- ◆When the error ΔL is the same or smaller than "#8077 Involute error", the curve is in the direction toward the commanded end point.



When the end point error is large, the actual speed may be different from the commanded feedrate.

Note

- ◆When the involute error is not set (setting value: 0.000), the allowable error is 0.1 (mm) as a default value.

Commanded value

If an axis given in the involute interpolation command is not in the selected plane, the program error (P33) occurs.

Number of revolutions

Set the number of revolutions within 100 from the start point of the curve at start/end point of involute interpolation. The number of revolutions is obtained by the angle indicated with θ in the calculation formula of the involute curve in the "Function and purpose" section. ($\theta \leq 360^\circ \times 100$)

When the number of revolutions exceeds 100, the program error (P35) occurs.



Feed Functions

7.1 Rapid Traverse Rate

7.1.1 Rapid Traverse Rate



Function and purpose

The rapid traverse rate can be set with parameters independently for each axis. The available speed ranges are from 1 mm/min to 10000000 mm/min. The upper limit is subject to the restrictions limited by the machine specifications.

Refer to the specifications manual of the machine for the rapid traverse rate settings.

The feedrate is valid for the G00, G27, G28, G29, G30 and G60 commands.

Two paths are available for positioning: the interpolation type where the area from the start point to the end point is linearly interpolated or the non-interpolation type where movement proceeds at the maximum speed of each axis.

The type is selected with parameter "#1086 G0Intp". The positioning time is the same for each type.

If the high-accuracy control mode's rapid traverse rate is set, the axis will move at that feedrate during high-accuracy control, high-speed high-accuracy control I/II/III, high-accuracy spline control or SSS control.

- If the value set for the high-accuracy control mode rapid traverse rate is 0, the axis will move at the rapid traverse rate.
- The high-accuracy control mode rapid traverse rate can be set independently for each axis.
- The high-accuracy control mode rapid traverse rate is effective for the following G commands: G00, G27, G28, G29, G30 and G60.
- Override can be applied on the high-accuracy control mode rapid traverse rate using the PLC signal supplied. (The operation of the PLC signal depends on the MTB specifications.)

Note

(1) Rapid traverse override

Override can be applied by a PLC input signal for both manual and automatic rapid traverse. There are 2 types which are determined by the PLC specifications.

Type1 : Override in 4 steps (1%, 25%, 50% and 100%).

Type2 : Override in 1% steps from 0% to 100%.

7.1.2 G00 Feedrate Command (,F Command)



Function and purpose

Use this function to specify G00 (positioning command) and an axis feedrate in G00 mode.

The speed of tool exchange, axis movement of gantry, etc. can be specified with the machining program so that the mechanical vibration can be suppressed.

Operations other than the feedrate follows the G00 specification.



Command format

Rapid traverse at a feedrate specified with the ",F" command

```
G00 X_ Z_ (Y_) ,F1000;
```

,F	Specifies the rapid traverse rates for G00, movement in G00 mode and the movement during the fixed cycle for drilling. The range is equal to the range of the feed per minute F command (mm/min, inch/min) in the G01 mode. Switching inch/mm is invalid for rotary axes.
----	---



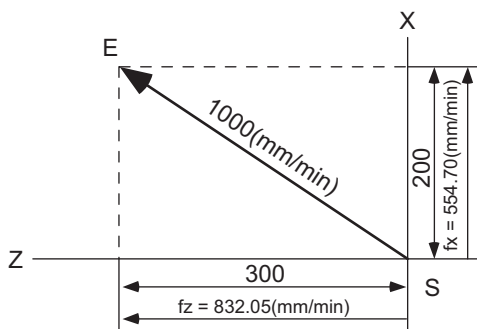
Detailed description

- (1) ",F" command is in effect only in the block in which it is commanded.
- (2) If ",F" is commanded in G00, G27 to G30, G60, G00 mode, a block other than the one that specifies the movement to the initial point of the hole position for the drilling cycle or a block that does not contain a movement command (axis address command), ",F" is ignored.
- (3) ",F" command in the feed per revolution (G95) mode will also be considered a feed per minute feedrate.
- (4) The motion of the ",F" command varies depending on the status of parameter "#1086 G0Intp".

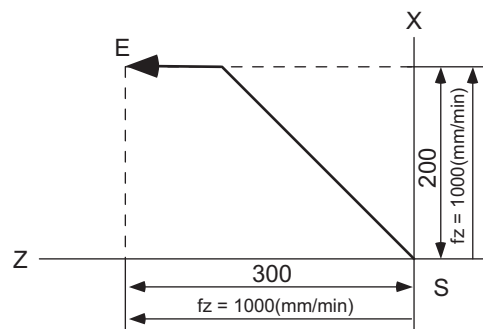
"#1086 G0Intp"	Handling of ",F"command
OFF (see figure shown at below left)	Handled as an interpolation speed.
ON (see figure shown at below right)	Handled as a commanded speed for each axis.

Feedrates when commanding G00 X200. Z300. ,F1000

When "G0 non-interpolation" is OFF



When "G0 non-interpolation" is ON



fx: Actual X axis rate
fz: Actual Z axis rate

- (5) When the ",F" command has not been issued, the rapid traverse rate set by the axis specification parameter will be valid. (*1)
- (6) The ",F" command is clamped by the rapid traverse rate set by the axis specification parameter. (*1)
Feedrate clamping depends on the setting of parameter "#1086 G0Intp".

"#1086 G0Intp"	Speed clamp
OFF	If it is found that, after converting ",F" command value (interpolation speed) into a speed for each axis, there is an axis for which the programmed feedrate exceeds the rapid traverse rate parameter, the interpolation speed is calculated so that it does not exceed the rapid traverse rate. (*1)
ON	An axis whose ",F" command value (per axis speed) exceeds the rapid traverse rate parameter is clamped to a speed specified by the parameter. (*1) For an axis that does not exceed the rapid traverse rate parameter, the commanded speed is applied.

(*1) The rapid traverse rate parameter depends on the MTB specifications.
Typically, parameter "#2001 rapid" is selected.



Program example

- (1) Feedrate command in G00 block and G00 mode (for G00 interpolation)

:	
G00 X100. Z100. ,F1000 ;	The tool moves at the combined feedrate, 1000 (mm/min), of XZ.
X200. Z200. ;	The X and Z axes interpolate at the fastest feedrate that does not exceed the rapid traverse rate parameter for each of these axes.
X300.Z300. ,F2500 ;	The tool moves at the combined feedrate, 2500 (mm/min), of XZ.
:	

- (2) Speed command for the movement to the initial point of the hole position for the drilling cycle (for a longitudinal tapping cycle)

:	
G88 X-20. Z30 R5. F1.D3 S500 ,R1 ,F2000 ;	The tool moves to the initial point (Z30.) of the hole position at 2000 (mm/min). Positioning (G00) during the drilling cycle moves at 2000 (mm/min).
X-20. Z35. R5. ;	The tool moves to the initial point (Z35.) of the hole position at the Z axis rapid traverse rate (parameter setting value).
X-20. Z40. R5.,F3000 ;	The tool moves to the initial point (Z40.) of the hole position at 3000 (mm/min). Positioning (G00) during the drilling cycle moves at 2000 (mm/min).
G80 ;	
:	



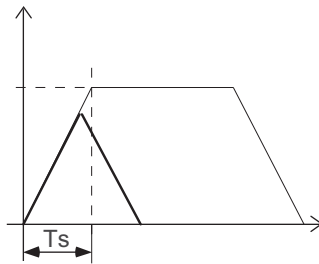
Relationship with Other Functions

Rapid Traverse Constant Inclination Acceleration/Deceleration

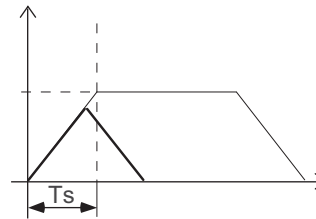
When ",F" is specified, constant inclination acceleration/deceleration control is applied to the feedrate specified by ",F".

The feedrate (vertical axis in the figure below) varies depending on whether or not the ",F" command has been issued.

(1) When the ",F" command has not been issued
Rapid traverse rate set by the parameter



(2) When the ",F" command has been issued
Feedrate specified by the ",F" command

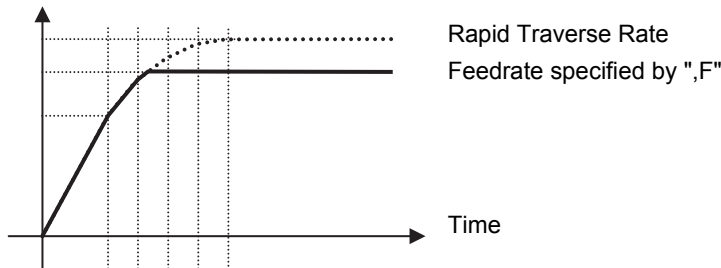


- Movement when rapid traverse constant inclination is enabled
- Movement when rapid traverse constant inclination is disabled

Rapid Traverse Constant Inclination Multi-step Acceleration/Deceleration

The feedrate specified by ",F" is applied according to acceleration pattern calculated from acceleration rate to "rapid traverse rate", "rated speed", "G00 time constant to rated speed" and "maximum acceleration".

Rated speed



Rapid traverse override

An override for ",F" command

Override cancel

The override cancel for the rapid traverse override is also invalid when ",F" is specified.

Dry run

Dry run is valid when the parameter "#1085 G00Drn" is ON and the rapid traverse is OFF. The axis will move at the manual feedrate that is set. If the manual override valid is turned ON, the cutting feed override also becomes valid.

External deceleration

It is also valid when ",F" is specified.

Programmable in-position check

It is also valid when ",F" is specified.

Tool center point control

",F" command is ineffective in tool center point control.

Special fixed cycle

When an ",F" command is specified in the fixed cycle for drilling, the movement between hole positions is carried out at the speed commanded with ",F".

",F" commands in the same block as for the special fixed cycle are ignored.

Unidirectional positioning

When an ",F" command is specified in the same block as G60 (unidirectional positioning), the feedrate specified by ",F" is assumed.

Reference position check, Start point return, Tool change position return

When an ",F" command is specified in the same block as G27 (reference position check), G29 (start point return), and/or G30.n (tool change position return), the feedrate specified by ",F" is assumed.

Reference position return, 2nd to 4th reference position return

When an ",F" command is specified in the same block as G28 (reference position return) and G30 (2nd to 4th reference position return), the feedrate specified by ",F" is assumed.

Axes not subject to high-speed reference position return are returned by the dog-type of in the same way as with the manual type. The feedrate depends on the MTB specifications (parameter "#2025 G28rap").

**Precautions**

- (1) If an ",F" command is specified when there is no specifications for the feedrate specified for G00, a program error (P39) will occur.
- (2) ",F" and "F" commands may be specified in the same block. The "F" command is assumed to the feedrate for cutting feed.
- (3) Depending on the MTB specifications (parameter "#1100 Tmove"), compensation may be performed on a block that does not contain a move command.
If an ",F" is specified in a tool compensation command (T command) block in which no move command is specified, compensation move is made at the feedrate specified by ",F" only in G00 mode.
- (4) If an ",F" is specified in a tool radius compensation cancel command (G40) block in which no move command is specified, tool radius compensation is canceled at the specified feedrate only in G00 mode.
This is the same as when using the tool nose radius compensation instead of tool radius compensation.

7.2 Cutting Feedrate



Function and purpose

The cutting feedrate is assigned with address F and numerals.

The cutting feedrate is valid for the G01, G02, G03, G02.1 and G03.1 commands.

If the cutting clamp feedrate for the high-accuracy control mode is set, the axis will move at that feedrate during high-accuracy control, high-speed high-accuracy control I/II/III, high-accuracy spline control or SSS control.

- If the value set for the high-accuracy control mode cutting clamp speed is "0", the axis will be clamped at the cutting feed clamp speed.
- The cutting feedrate is clamped with high-accuracy control mode cutting clamp speed in the parameter.

Examples Feed per minute (asynchronous feed)

	Feedrate	
G01 X100. Y100. F200 ;	200.0mm/min	F200 or F200.000 gives the same rate.
G01 X100. Y100. F123.4 ;	123.4mm/min	
G01 X100. Y100. F56.789 ;	56.789mm/min	

Speed range that can be commanded (when input setting unit is 1 μ m)

Command Mode	Feedrate command range	Remarks
mm/min	0.001 to 10000000	
inch/min	0.0001 to 1000000	
°/min	0.001 to 10000000	

Note

- (1) A program error (P62) will occur when there is no F command in the first cutting command (G01, G02, G03) after the power has been turned ON.

7.3 F1-digit Feed



Function and purpose

By setting the F1-digit feed parameter, the feedrate which has been set to correspond to the 1-digit number following the F address serves as the command value.

When F0 is assigned, the rapid traverse rate is established and the speed is the same as for G00. (G modal does not change, but the acceleration/deceleration method follows the rapid traverse setting.)

When F1 to F5 is assigned, the feedrate set to correspond to the command serves as the command value.

If F1-digit feedrate changing valid signal is turned ON when F1-digit feed is commanded, the feedrate specified by the parameter can be increased or decreased by operation of manual handle. For the changing of F1-digit feedrate with the handle feed, refer to the instruction manual.



Detailed description

- (1) To validate the F1-digit feed, the parameter "#8145 Validate F1 digit" or "#1079 f1digit" must be ON.
- (2) The feedrates that correspond to F1 to F5 depend on the MTB specifications (parameters "#1185 spd_F1" to "#1189 spd_F5").
The increase/reduction range is from "0" to the set value of the parameter "#1506 F1_FM".
An operation error (M01 0104) will occur when the feedrate is "0".
When F0 is commanded, the acceleration or deceleration method follows the rapid traverse setting. Note that the G modal is not changed.
- (3) Use of both the F1-digit command and normal cutting feedrate command is possible when the F1-digit is valid.
(Example 1) F0 Rapid traverse rate
 F1 to F5 F1 digit
 F6 or more Normal cutting feedrate command
- (4) The F1-digit command is valid in a G01, G02, G03, G02.1 or G03.1 modal.
- (5) The F1-digit command can also be used for fixed cycle.
- (6) The F1-digit feedrate command can also be used during high-speed high-accuracy control II.
However, a program error (P62) will occur when F0 command is issued.
- (7) The F1-digit command is modal.
- (8) The number of manual handle pulses is 1 pulse per scale unit regardless of the scaling factor.
- (9) During a F1-digit command, the F1-digit number and F1-digit command signal are output as the PLC signals.
(Based on the MTB specifications.)



Precautions

- (1) F1 to F5 are invalid in the G00 mode and the rapid traverse rate is established instead.
- (2) If F0 is used in the G02, G03, G02.1 or G03.1 mode, the program error (P121) will occur. The error will be eliminated if the F0 command is rewritten.
- (3) When F1. to F5. (with decimal point) are assigned, the 1mm/min to 5mm/min (direct numerical value command) are established instead of the F1-digit feed command.
- (4) When the commands are used with inch units, one-tenth of the feedrate set correspond to F1 to F5 serves at the assigned speed inch/min.
- (5) When the commands are used with the millimeter or degree units, the feedrate set to correspond to F1 to F5 serves as the assigned speed mm (°)/min.
- (6) Even if the F1-digit feed is commanded during feed per revolution (G95), it is executed as a normal F command (direct numerical value command).
- (7) When both the F1-digit feed command and inverse time feed command are present, the inverse time feed command will have priority.
(The inverse time feed function is available only for a machining center system.)
- (8) When both the F1-digit feedrate changing and the manual speed command are present, the manual speed command will have the priority.
- (9) In the synchronous tapping command, the speed cannot be changed with the handle.

7.4 Feed Per Minute/Feed Per Revolution (Asynchronous Feed/ Synchronous Feed); G94,G95



Function and purpose

Feed per minute (asynchronous feed)

By issuing the G94 command, the commands from that block are issued directly by the numerical value following F as the feedrate per minute (mm/min, inch/min).

Feed per revolution (synchronous feed)

By issuing the G95 command, the commands from that block are issued directly by the numerical value following F as the feedrate per spindle revolution (mm/rev, inch/rev).

When this command is used, the rotary encoder must be attached to the spindle.



Command format

G94; ... Feed per minute (mm/min) (asynchronous feed)

G95; ... Feed per revolution (mm/rev) (synchronous feed)



Detailed description

G94/G95 commands are modal commands.

(Ex.) After the G95 command is assigned, the G95 command is valid until the G94 command or G93 command (inverse time feed) is next assigned.

(1) The F code command range is as follows.

Metric input

Input Setting unit	B(0.001mm)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(mm/min)	F(mm/rev)
Minimum command unit	1 (=1.000) (1.=1.000)	1 (=0.001) (1.=1.000)
Command range	0.001 - 1000000.000	0.001 - 999.999
Input Setting unit	C(0.0001mm)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(mm/min)	F(mm/rev)
Minimum command unit	1 (=1.0000) (1.=1.0000)	1 (=0.0001) (1.=1.0000)
Command range	0.0001 - 1000000.0000	0.0001 - 999.9999
Input Setting unit	D(0.00001mm)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(mm/min)	F(mm/rev)
Minimum command unit	1 (=1.00000) (1.=1.00000)	1 (=0.00001) (1.=1.00000)
Command range	0.00001 - 1000000.00000	0.00001 - 999.99999

Input Setting unit	E(0.000001mm)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(mm/min)	F(mm/rev)
Minimum command unit	1 (=1.000000) (1.=1.000000)	1 (=0.000001) (1.=1.000000)
Command range	0.000001 - 1000000.000000	0.000001 - 999.999999

Inch input

Input Setting unit	B(0.0001inch)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(inch/min)	F(inch/rev)
Minimum command unit	1 (=1.0000) (1.=1.0000)	1 (=0.0001) (1.=1.0000)
Command range	0.0001 - 100000.0000	0.0001 - 999.9999

Input Setting unit	C(0.00001inch)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(inch/min)	F(inch/rev)
Minimum command unit	1 (=1.00000) (1.=1.00000)	1 (=0.00001) (1.=1.00000)
Command range	0.00001 - 100000.00000	0.00001 - 999.99999

Input Setting unit	D(0.000001inch)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(inch/min)	F(inch/rev)
Minimum command unit	1 (=1.000000) (1.=1.000000)	1 (=0.000001) (1.=1.000000)
Command range	0.000001 - 100000.000000	0.000001 - 999.999999

Input Setting unit	E(0.0000001inch)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(inch/min)	F(inch/rev)
Minimum command unit	1 (=1.0000000) (1.=1.0000000)	1 (=0.0000001) (1.=1.0000000)
Command range	0.0000001 - 100000.000000	0.0000001 - 999.9999999

(2) The effective rate (actual movement speed of machine) under per-revolution feed conditions is given in the following formula (Formula 1).

$$FC = F \times N \times OVR \quad \text{..... (Formula 1)}$$

FC : Effective rate (mm/min, inch/min)

F : Commanded feedrate (mm/rev, inch/rev)

N : Spindle rotation speed (r/min)

OVR: Cutting feed override

When a multiple number of axes have been commanded at the same time, the effective rate FC in formula 1 applies in the vector direction of the command.



Precautions

- (1) The effective rate (mm/min or inch/min), which is produced by converting the commanded speed, the spindle rotation speed and the cutting feed override into the per-minute speed, appears as the FC on the monitor 1. Screen of the setting and display unit.
- (2) When the above effective rate exceeds the cutting feed clamp rate, it is clamped at that clamp rate.
- (3) If the spindle rotation speed is zero when feed per revolution is executed, the error "M01 operation error 0105" occurs.
- (4) Feedrate in the machine lock status is the command speed.
- (5) Under dry run conditions, feed per minute applies and movement results at the manual feedrate (mm/min or inch/min).
- (6) The fixed cycle G84 (tapping cycle) and G74 (reverse tapping cycle) are executed according to the feed mode that is already designated.
- (7) Whether feed per minute (G94) or feed per revolution (G95) is to be established when the power is turned ON or when M02 or M30 is executed can be selected by setting parameter "#1074 I_Sync".

7.5 Inverse Time Feed; G93



Function and purpose

During inside cutting when machining curved shapes with radius compensation applied, the machining speed on the cutting surface becomes faster than the tool center feedrate. Therefore, problems such as reduced accuracy may occur.

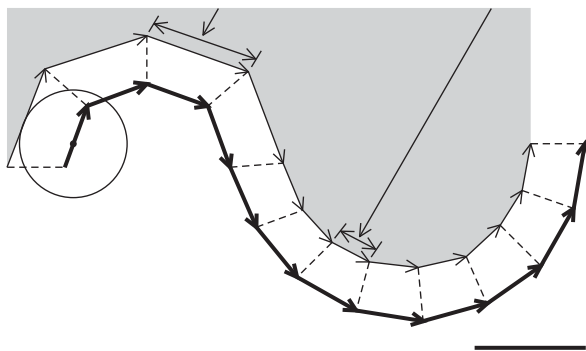
This reduced accuracy can be prevented with inverse time feed. This function can, in place of normal feed commands, issue one block of machining time (inverse) in F commands. The machining speed on the cutting surface is constantly controlled, even if radius compensation is applied to the machining program that expresses the free curve surface with fine segment lines.

Note that when the calculated machining time exceeds the cutting feed clamp speed, the F command value in the inverse time feed follows the cutting feed clamp speed.

Regular F command

Actual machining speed: High

Actual machining speed: Low

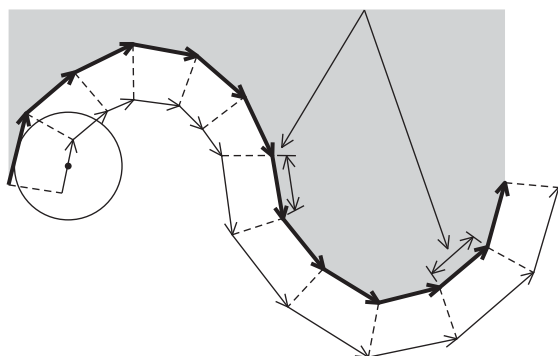


The speed of tool center is commanded, thus the actual speed at the cutting surface may become larger or smaller.

————— F command

Inverse time feed

The actual machining speed is constant.



The actual speed at the cutting surface is commanded, thus, the speed will be constant and machining speed can be maintained as commanded regardless of the tool radius.

————— F command



Command format

Inverse time feed

G93;

Inverse time feed (G93) is a modal command. Once commanded, it will be valid until feed per minute or feed per revolution is commanded.

G00 Xx1 Yy1;	
G93;	-> Inverse time feed mode ON
G01 Xx2 Yy2 Ff2;	-> In inverse time feed mode
G02 Xx3 Yy3 Ii3 Jj3 Ff3;	:
G94 (G95);	-> Inverse time feed mode OFF

In movement blocks, since processing time is commanded to a line segment, command the feedrate "F" each time.



Detailed description

- (1) Inverse time feed (G93) is a modal command. Once commanded, it is valid until feed per minute (G94) or feed per revolution (G95) is commanded, or until a reset (M02, M30, etc.) is executed.
- (2) Command method of F command values in inverse time feed

		Metric command (G21)	Inch command (G20)
In linear mode (G01)		Cutting point feedrate (mm/min)	Cutting point feedrate (inch/min)
		Line segment length (mm)	Line segment length (inch)
In arc mode (G02, G03) (G02.1, G03.1)		Cutting point feedrate (mm/min)	Cutting point feedrate (inch/min)
		Start point arc radius (mm)	Start point arc radius (inch)
Command range	B	0.001 to 999999.999 (1/min)	
	C	0.0001 to 999999.9999 (1/min)	
	D	0.00001 to 999999.99999 (1/min)	
	E	0.000001 to 999999.999999 (1/min)	

- (3) The initial modal after a restart is G94 (feed per minute) or G95 (feed per revolution).
- (4) The feedrate of the block inserted in tool radius compensation and corner R/C is the same speed as the feedrate of the block immediately before it.
- (5) The feedrate of the block inserted in C axis normal line control (normal line control type II) is the same speed as the feedrate of the movement block after turning.



Program example

When using inverse time feed during tool radius compensation

Feed per minute	
N01 G90 G00 X80. Y-80. ;	
N02 G01 G41 X80. Y-80. D11 F500 ;	
N03 X180. ;	
N04 G02 Y-280. R100. ;	
N05 G03 Y-480. R100. ;	
N06 G02 Y-680. R100. ;	
N07 G01 X80. F500 ;	
N08 Y-80. ;	
N09 G04 X80. Y-80. ;	
N10 M02 ;	
Inverse time feed	
N01 G90 G00 X80. Y-80. ;	
N02 G01 G41 X80. Y-80. D11 F500 ;	
N03 X180. ;	
N04 G93 G02 Y-280. R100. F5 ;	
N05 G03 Y-480. R100. F5 ;	
N06 G02 Y-680. R100. F5 ;	
N07 G94 G01 X80. F500 ;	
N08 Y-80. ;	
N09 G04 X80. Y-80. ;	
N10 M02 ;	

Comparison between feed per minute and inverse time feed (Assuming that tool radius is 10. [mm]) (Unit: mm/min)

Sequence No.	Feed per minute		Inverse time feed	
	Feedrate of tool center	Feedrate of cutting point	Feedrate of tool center	Feedrate of cutting point
N04	F500	F450	F550	F500
N05	F500	F550	F450	F500
N06	F500	F450	F550	F500
		↓		↓
		The block seam protrudes due to the cutting speed change at the block seam.		The feedrate follows the command regardless of the tool radius.



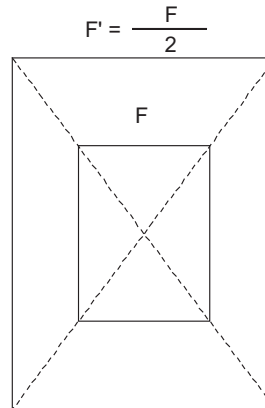
Relationship with other functions

(1) Scaling (G51)

When using a scaling function, issue a F command for the shape after scaling. For example, if a doublesize scaling is carried out, the machining distance will be twice.

Thus, if executing a cutting at the same speed as that of before scaling, command the value (F') calculated by dividing F value by the multiples of scaling.

$$F = \text{Feedrate (mm/min)} / \text{Distance (mm)}$$



Shape after scaling (Double size)

(2) High-speed machining mode II (G05P2)

With the inverse time feed (G93) modal, high-speed machining mode II (G05P2) is operated in the inverse time feed mode, instead of high-speed machining mode. High-speed machining mode will be valid when the inverse time feed mode is canceled.

- (3) If the speed calculated in the G93 mode exceeds the speed range at the feed per minute, clamping is performed at the clamp speed set with parameters.
- (4) The program error (P125) will occur when the commands below are issued in the inverse time feed (G93) mode.

G code	Function
G02.3, G03.3	Exponential interpolation
G06.2	NURBS interpolation
G12	Circular cutting CW
G13	Circular cutting CCW
G31 to G31.3	Skip
G33	Thread cutting
G34 to G36, G37.1	Special fixed cycle
G37	Automatic tool length measurement
G73 to G89	Fixed cycle
G96	Constant surface speed control ON

- (5) The program error (P125) will occur if inverse time feed (G93) is commanded in the following modes.

G code	Function
G02.3, G03.3	Exponential interpolation
G33	Thread cutting
G73 to G89	Fixed cycle
G96	Constant surface speed control ON



Precautions

- (1) The initial modal after a restart is G94 (feed per minute) or G95 (feed per revolution).
- (2) The F command in G93 modal is unmodal. Issue an F command for each block. The program error (P62) will occur in blocks with no F command.
- (3) The program error (P62) will occur when F0 is commanded.
- (4) An F command is necessary when changing from G93 to G94 or G95. The program error (P62) will occur if there is no F command.
- (5) The feed function is clamped at the maximum cutting speed. Consequently, the feed may be slower than the commanded speed.
- (6) If an extremely slow speed such as F0.001 is designated, an error will occur in the machining time.

7.6 Feedrate Designation and Effects on Control Axes



Function and purpose

It has already been mentioned that a machine has a number of control axes. These control axes can be divided into linear axes which control linear movement and rotary axes which control rotary movement. The feedrate is designed to assign the displacement speed of these axes, and the effect exerted on the tool movement speed which poses problems during cutting differs according to when control is exercised over the linear axes or when it is exercised over the rotary axes.

The displacement amount for each axis is assigned separately for each axis by a value corresponding to the respective axis. The feedrate is not assigned for each axis but assigned as a single value. Therefore, when two or more axes are to be controlled simultaneously, it is necessary to understand how this will work for each of the axes involved.

The assignment of the feedrate is described with the following related items.

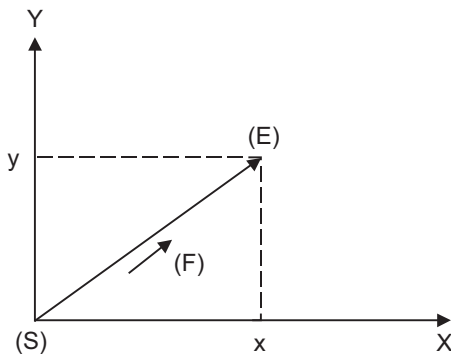


Detailed description

When controlling linear axes

Even when only one machine axis is to be controlled or there are two or more axes to be controlled simultaneously, the feedrate which is assigned by the F code functions as a linear speed in the tool advance direction.

(Example) When the feedrate is designated as "f" and linear axes (X and Y) are to be controlled:



$$f_x = f \times \frac{x}{\sqrt{x^2 + y^2}} \dots \text{Feedrate for X axis}$$

$$f_y = f \times \frac{y}{\sqrt{x^2 + y^2}} \dots \text{Feedrate for Y axis}$$

(S) Tool start point

(E) Tool end point

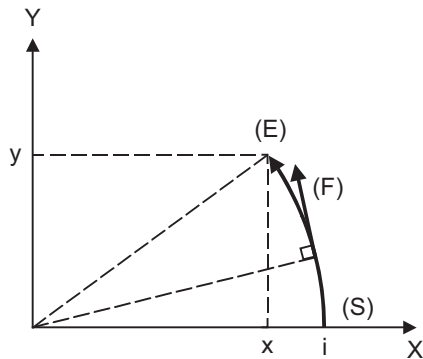
(F) Speed in this direction is "f".

When only linear axes are to be controlled, it is sufficient to designate the cutting feed in the program.

The feedrate for each axis is such that the designated rate is broken down into the components corresponding to the movement amounts.

(Example) When the feedrate is designated as "f" and the linear axes (X and Y) are to be controlled using the circular interpolation function:

The rate in the tool advance direction, or in other words the tangential direction, will be the feedrate designated in the program.



- (S) Tool start point
- (E) Tool end point
- (F) Speed in this direction is "f".

In this case, the feedrate of the X and Y axes will change along with the tool movement. However, the combined speed will always be maintained at the constant value "f".

When controlling rotary axes

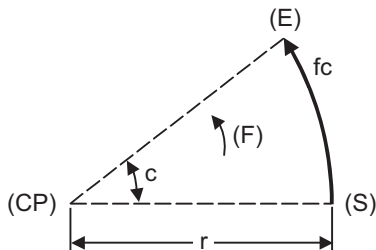
When rotary axes are to be controlled, the designated feedrate functions as the rotary speed of the rotary axes or, in other words, as an angular speed.

Consequently, the cutting feed in the tool advance direction, or in other words the linear speed, varies according to the distance between the center of rotation and the tool.

This distance must be borne in mind when designating the feedrate in the program.

(Example) When the feedrate is designated as "f" and the rotary axis (C axis) is to be controlled:

("f" units = °/min)



- (S) Tool start point
- (E) Tool end point
- (CP) Center of rotation
- (F) Angular speed is "f".

In this case, the cutting feed (linear feed) in the tool advance direction "fc" is obtained as follows:

$$f_c = f \times \frac{\pi \cdot r}{180}$$

Therefore, the feedrate to be designated in the program must be as follows:

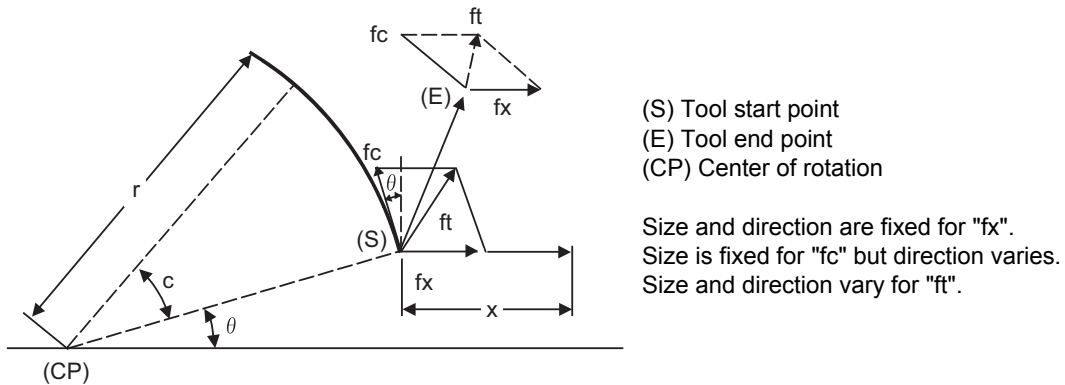
$$f = f_c \times \frac{180}{\pi \cdot r}$$

When linear and rotary axes are to be controlled at the same time

The controller proceeds in exactly the same way whether linear or rotary axes are to be controlled. When a rotary axis is to be controlled, the numerical value assigned by the coordinate word (A,B,C) is the angle and the numerical values assigned by the feedrate (F) are all handled as linear speeds. In other words, 1° of the rotary axis is treated as being equivalent to 1mm of the linear axis. Consequently, when both linear and rotary axes are to be controlled simultaneously, in the components for each axis of the numerical values assigned by F will be the same as previously described "When controlling linear axes". However, although in this case both the size and direction of the speed components based on linear axis control do not vary, the direction of the speed components based on rotary axis control will change along with the tool movement (their size will not change). This means, as a result, that the combined tool advance direction feedrate will vary along with the tool movement.

(Example) When the feedrate is designated as "f" and linear (X) and rotary (C) axes are to be controlled simultaneously:

In the figure below, the X axis incremental command value is "x" and the C axis incremental command value is "c".



X axis feedrate (linear speed) "fx" and C axis feedrate (angular speed) "ω" are expressed as:

$$f_x = f \times \frac{x}{\sqrt{x^2 + c^2}} \dots\dots (1)$$

$$\omega = f \times \frac{c}{\sqrt{x^2 + c^2}} \dots\dots (2)$$

Linear speed "fc" based on C axis control is expressed as:

$$f_c = \omega \times \frac{\pi \times r}{180} \dots\dots (3)$$

If the speed in the tool advance direction at start point (S) is "ft" and the component speeds in the X axis and Y axis directions are "ftx" and "fty", respectively, then these can be expressed as:

$$f_{tx} = -r \sin \left(\frac{\pi}{180} \theta \right) \times \frac{\pi}{180} \omega + f_x \dots\dots (4)$$

$$f_{ty} = -r \cos \left(\frac{\pi}{180} \theta \right) \times \frac{\pi}{180} \omega \dots\dots (5)$$

Where "r" is the distance (in millimeters) between the center of rotation and the tool and "θ" is the angle (in degrees) between the (S) point and the X axis at the center of rotation.

The combined speed "ft" according to (1), (2), (3), (4) and (5) is as follows:

$$\begin{aligned}
 ft &= \sqrt{ft_x^2 + ft_y^2} \\
 &= f \times \frac{\sqrt{x^2 - x \times c \times r \sin\left(\frac{\pi}{180} \theta\right) \frac{\pi}{90} + \left(\frac{\pi \times r \times c}{180}\right)^2}}{x^2 + c^2} \dots\dots (6)
 \end{aligned}$$

Consequently, feedrate "f" designated by the program must be as follows:

$$f = ft \times \frac{x^2 + c^2}{\sqrt{x^2 - x \times c \times r \sin\left(\frac{\pi}{180} \theta\right) \frac{\pi}{90} + \left(\frac{\pi \times r \times c}{180}\right)^2}} \dots\dots (7)$$

"ft" in formula (6) is the speed at the (S) point and the value of θ changes as the C axis rotates, which means that the value of "ft" will also change. Consequently, in order to keep the cutting feed "ft" as constant as possible the angle of rotation which is designated in one block must be reduced to as low as possible and the extent of the change in the θ value must be minimized.

7.7 Rapid Traverse Constant Inclination Acceleration/Deceleration



Function and purpose

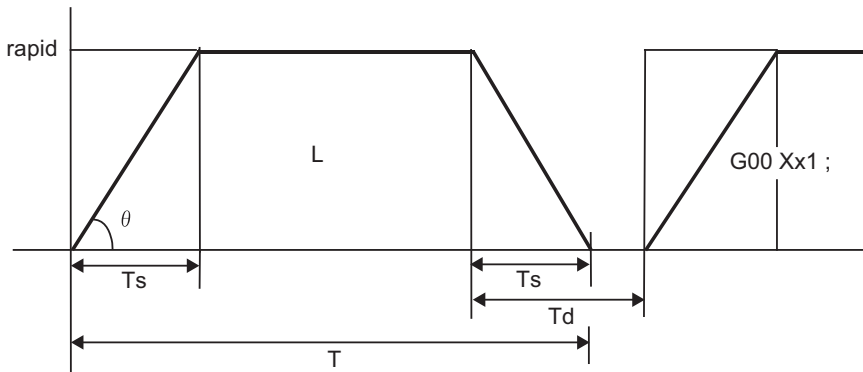
This function performs acceleration and deceleration at a constant inclination during linear acceleration/deceleration in the rapid traverse mode. Compared to the method of acceleration /deceleration after interpolation, the constant inclination acceleration/deceleration method makes for improved cycle time.



Detailed description

- (1) Rapid traverse constant inclination acceleration/deceleration are valid only for a rapid traverse command. Also, this function is effective only when the rapid traverse command acceleration/deceleration mode is linear acceleration and linear deceleration.
- (2) The acceleration/deceleration patterns in the case where rapid traverse constant inclination acceleration/deceleration are performed are as follows.

[When the interpolation distance is long enough for the rapid traverse rate to be achieved]



$$T = \frac{L}{\text{rapid}} + T_s$$

$$T_d = T_s + (0 \sim 14\text{ms})$$

$$\theta = \tan^{-1} \left(\frac{\text{rapid}}{T_s} \right)$$

rapid : Rapid Traverse Rate

Ts : Acceleration/deceleration time constant

Td : Command deceleration check time

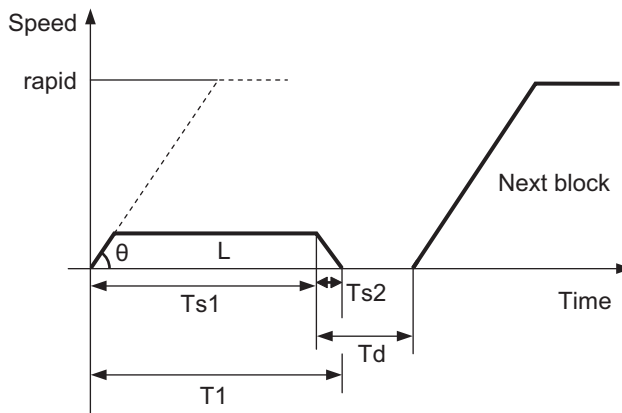
θ : Acceleration/deceleration inclination

T : Interpolation time

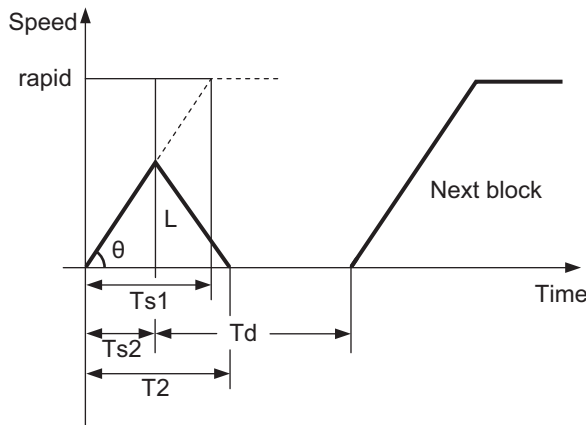
L : Interpolation distance

[When the interpolation distance is smaller than the acceleration/deceleration distance]

In case of time-constant acceleration/deceleration:



In case of inclination-constant acceleration/deceleration:



$$T1 = Ts1 + Ts2$$

$$T2 = 2 \times \sqrt{Ts1 \times L / \text{rapid}}$$

$$Td = \frac{T2}{2} + (0 \sim 14 \text{ ms})$$

$$\theta = \frac{\tan^{-1}(\text{rapid})}{Ts1}$$

rapid : Rapid traverse speed (Axis specification parameter #2001 rapid)

Ts1 : Acceleration/deceleration time (Axis specification parameter #2004 G0tL)

Ts2 : Acceleration/deceleration time to reach the maximum speed

Td : Command deceleration check time

θ : Acceleration/deceleration inclination

T1 : Interpolation time (Time-constant acceleration/deceleration)

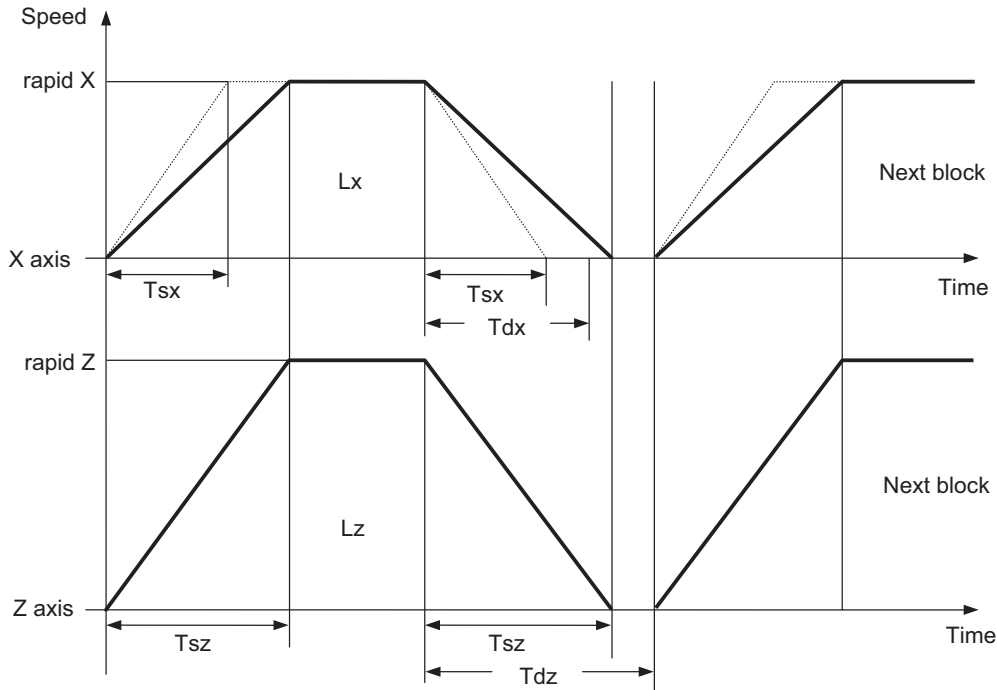
T2 : Interpolation time (Inclination-constant acceleration/deceleration)

L : Interpolation distance

7 Feed Functions

- (3) When 2-axis simultaneous interpolation (linear interpolations) is performed during rapid traverse constant inclination acceleration and deceleration, the acceleration (deceleration) time is the longest value of the acceleration (deceleration) times determined for each axis by the rapid traverse rate of commands executed simultaneously, the rapid traverse acceleration and deceleration time constant, and the interpolation distance, respectively. Consequently, linear interpolation is performed even when the axes have different acceleration and deceleration time constants.

[2-axis simultaneous interpolation (When linear interpolation is used, $T_{sx} < T_{sz}$, $L_x \neq L_z$)]

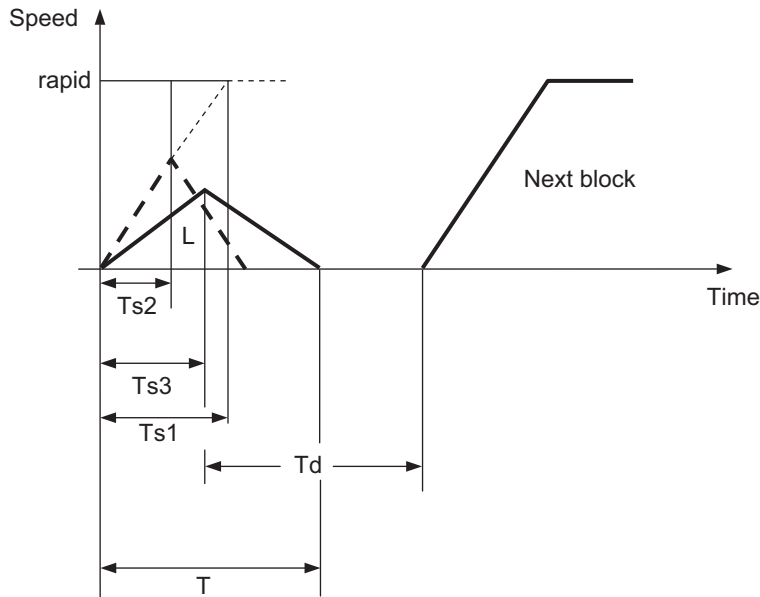


When T_{sz} is greater than T_{sx} ,
 T_{dz} is also greater than T_{dx} , and
 $T_d = T_{dz}$ in this block.

- T_{sx} : X axis acceleration/deceleration time
- T_{sz} : Z axis acceleration/deceleration time
- T_{dx} : X axis commanded deceleration check time
- T_{dz} : Z axis commanded deceleration check time
- L_x : X axis interpolation distance
- L_z : Z axis interpolation distance

- (4) If a minimum time constant for inclination-constant acceleration/deceleration has been set by the parameter, acceleration/deceleration speed is adjusted to prevent the acceleration/deceleration time calculated by interpolation distance from going below the minimum time constant.

[When the interpolation distance is so short that the acceleration/deceleration time is shorter than the minimum time constant for constant inclination acceleration/deceleration]



$$T = 2 \times Ts2$$

$$Td = \frac{T}{2} + (0 \sim 14 \text{ ms})$$

rapid : Rapid traverse speed (Axis specification parameter #2001 rapid)

Ts1 : Acceleration/deceleration time (Axis specification parameter #2004 G0tL)

Ts2 : Acceleration/deceleration time to reach the maximum speed

Ts3 : Minimum time for inclination-constant acceleration/deceleration (Axis specification parameter #2198 G0tMin)

Td : Command deceleration check time

T : Interpolation time

L : Interpolation distance

- (5) Use the rapid traverse time constant changeover request signal to switch the rapid traverse time constant. The operations via PLC signals and the settings of related parameters depend on the MTB specifications. The time constant is switched in the block next to where the rapid traverse time constant changeover request signal is turned ON/OFF.

	Basic rapid traverse time constant (signal OFF)	Rapid traverse time constant for switching (signal ON)
Rapid traverse time constant	#2004 G0tL	#2598 G0tL_2
Rapid traverse time constant (primary delay) / 2nd step time constant of soft acceleration/deceleration	#2005 G0t1	#2599 G0t1_2

<Note>

◆When #2598 is "0", #2004 is used. When #2599 is "0", #2005 is used.

- (6) The program format of G00 (rapid traverse command) when rapid traverse constant inclination acceleration/deceleration are executed is the same as when this function is invalid (time constant acceleration/deceleration).
- (7) This function is valid only for G00 (rapid traverse).



Precautions

- (1) When "#2003 smgst" (acceleration/deceleration mode) is set to the soft acceleration/deceleration, and "#1219 aux03 bit7" (time constant setting changeover for soft acceleration/deceleration) is set to "1", the acceleration/deceleration speed is adjusted to prevent the sum of the 1st step and 2nd step acceleration/deceleration times from going below the minimum time constant for inclination-constant acceleration/deceleration. In this case, the acceleration time will be "G0tL+G0t1" or "G1tL+G1t1".
- (2) When "#2003 smgst" (acceleration/deceleration mode) is set to the soft acceleration/deceleration, if the acceleration/deceleration is shorter than G0tL (or G1tL), the 2nd step time constant will be reduced by the same rate as the 1st step time constant.
- (3) If a commanded travel distance in a block is small, acceleration/deceleration time becomes quite short when the constant-inclination acceleration/deceleration method is enabled. Although this does contribute to reduce the cycle time, this can also be a cause of machine vibrations. In these cases, if the minimum time constant for inclination-constant acceleration/deceleration is set in parameter "#2198 G0tMin", it is possible to perform acceleration/deceleration to prevent the acceleration/deceleration time from being below this setting value. This parameter depends on the MTB specifications.

7.8 Rapid Traverse Constant Inclination Multi-step Acceleration/Deceleration



Function and purpose

This function carries out the acceleration/deceleration according to the torque characteristic of the motor in the rapid traverse mode during automatic operation. (This function is not available in manual operation.) The rapid traverse constant inclination multi-step acceleration/deceleration method makes for improved cycle time because the positioning time is shortened by using the motor ability to its maximum.

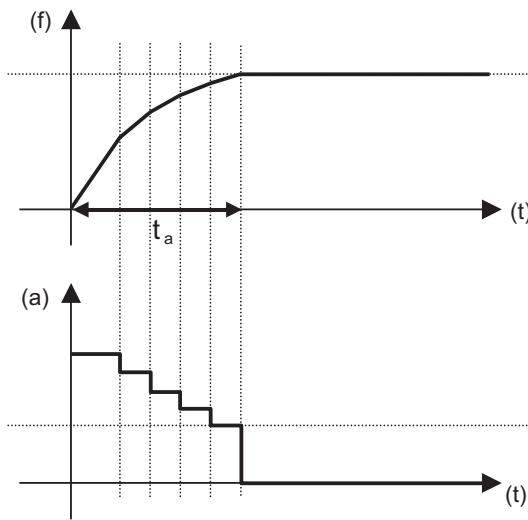
In general, the servomotor has the characteristic that the torque falls in the high-speed rotation range.

In the rapid traverse constant inclination acceleration/deceleration method, the acceleration is treated as constant because this torque characteristic is not considered. So, It is necessary to use a minimum acceleration within the used speed range. Therefore, the margin of acceleration must be had in a low-speed range. Or if the acceleration is used to its maximum, the upper limit of the rotation speed must be slowed.

Then, to use the servomotor ability to its maximum, acceleration/deceleration to which the torque characteristic is considered is carried out by the rapid traverse constant inclination multi-step acceleration/deceleration method.

The acceleration/deceleration patterns in the case where rapid traverse constant inclination multi-step acceleration/deceleration are performed are as follows.

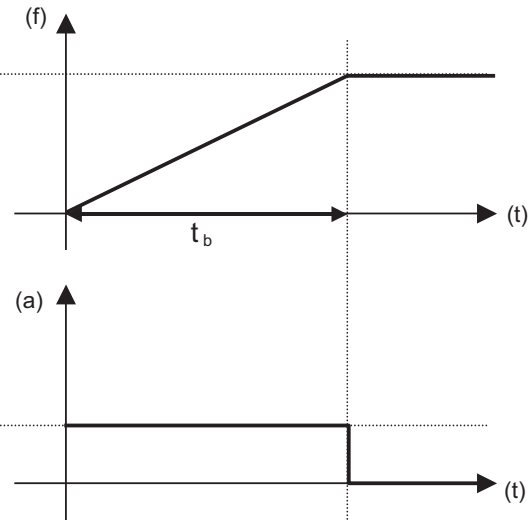
[Rapid traverse constant inclination multi-step acceleration/deceleration]



Number of steps is automatically adjusted by parameter setting.

(f) Speed (t) Time

[Rapid traverse constant inclination acceleration/deceleration]



It was necessary to slow down the acceleration for high speed rotation.

(a) Acceleration



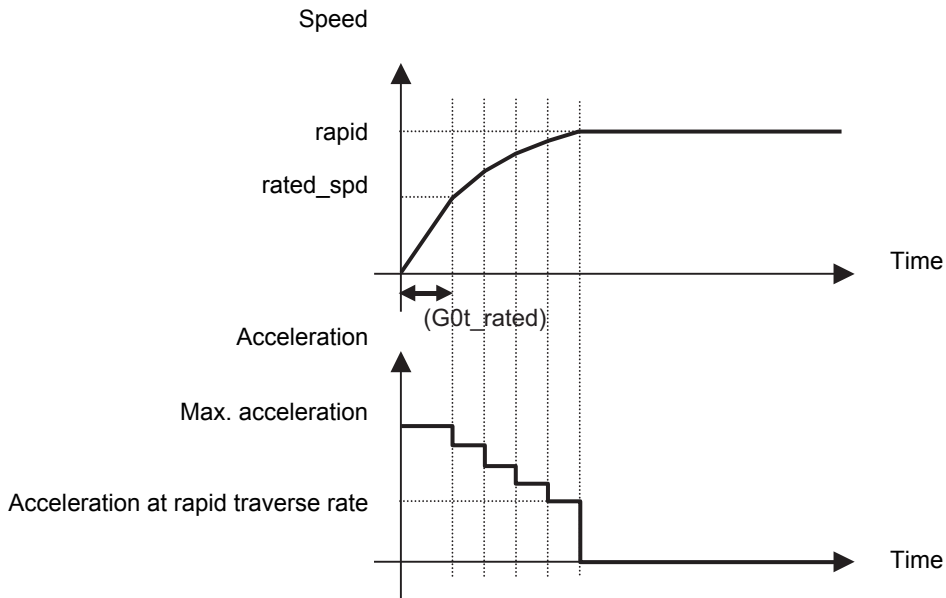
Detailed description

Use conditions

- (1) The validity of this function depends on the MTB specifications. (Parameter "#1205 G0bdcc")
 However, note the following conditions.
 - (a) "2" cannot be set to parameter "#1205 G0bdcc" except the 1st part system. When "2" is set to other than 1st part system, an MCP alarm (Y51 0017) will occur.
 - (b) When there is no specification for the rapid traverse constant inclination acceleration/deceleration, "2" cannot be set to parameter "#1205 G0bdcc". Even if the parameter is set to "2", this function is invalid. A normal time constant acceleration/deceleration (acceleration/deceleration after interpolation) is applied.
 - (c) Even if "2" is set to "#1205 G0bdcc" when G00 non-interpolation type ("#1086 G00Intp" = "1"), this function is invalid. In this case, a normal time constant acceleration/deceleration (acceleration/deceleration after interpolation) is applied.

(2) To use this function, the following parameters must be set for each axis.

#2001 rapid	Rapid traverse [mm/min]
#2151 rated_spd	Rated speed [mm/min]
#2153 G0t_rated	Acceleration time to rated speed [ms]
#2152 acc_rate	Acceleration at rapid traverse in ratio to the maximum acceleration [%]



Acceleration rate in proportion to the maximum acceleration rate = Acceleration at rapid traverse rate / Max. acceleration

- (3) When either of the following conditions applies, this function is invalid and operates as "rapid traverse constant inclination acceleration/deceleration". For the axis for which the rapid traverse constant inclination multi-step acceleration/deceleration is not necessary, set "0" to "#2151 rated_spd", "#2152 acc_rate" and "#2153 G0t_rated". However, these parameters depend on the MTB specifications.
 - (a) When "#2151 rated_spd" (rated speed) is "0" or larger than "#2001 rapid" (rapid traverse)
 - (b) When "#2152 acc_rate" (Acceleration rate in proportion to the maximum acceleration rate) is "0" or "100"
 - (c) Even if "2" is set to "#1205 G0bdcc" when G00 non-interpolation type ("#1086 G00Intp" = "1"), this function is invalid. In this case, a normal time constant acceleration/deceleration (acceleration/deceleration after interpolation) is applied.

(4) The comparison of the acceleration/deceleration patterns by the parameter setting is in the table below.

Mode	Rapid traverse constant inclination multi-step acceleration/deceleration	#1086 G00Intp	#1205 G0bdcc	Operation
G00 command	ON	0	0	Time constant acceleration/deceleration (interpolation type)
			1	Constant inclination acceleration/deceleration (acceleration/deceleration before interpolation)
			2	Constant inclination multi-step acceleration/deceleration
		1	Arbitrary	Time constant acceleration/deceleration (non-interpolation type)
	OFF	0	0	Time constant acceleration/deceleration (interpolation type)
			1	Constant inclination acceleration/deceleration (acceleration/deceleration before interpolation)
			2	Time constant acceleration/deceleration (interpolation type)
		1	Arbitrary	Time constant acceleration/deceleration (non-interpolation type)
Manual rapid traverse	Arbitrary	Arbitrary	Arbitrary	Time constant acceleration/deceleration (non-interpolation type)

Decision method of steps

For rapid traverse constant inclination multi-step acceleration/deceleration, the number of steps is automatically adjusted by set parameter.

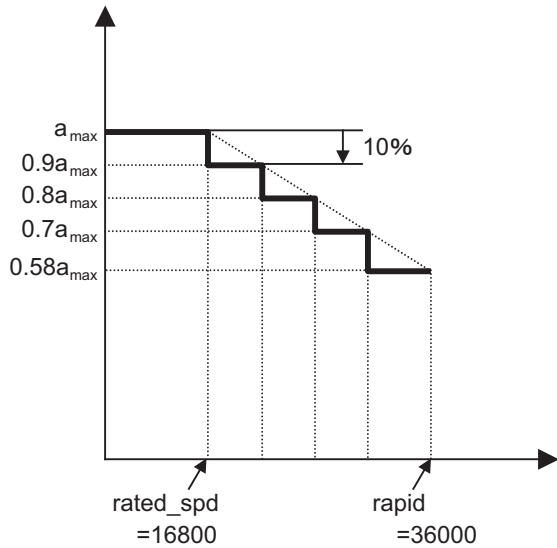
The acceleration per step is assumed to be a decrease by 10% of the maximum acceleration per step. Therefore, the number of steps is decided as follows.

$\text{"Step"} = (100 - \text{"\#2152 acc_rate"}) / 10 + 1 \text{ (Discard fractions less than 1)}$
--

The acceleration/deceleration pattern when the parameter setting value is shown below.

No.	Item		Setting value
2001	rapid	Rapid traverse rate	36000 [mm/min]
2151	rated_spd	Rated speed	16800 [mm/min]
2152	acc_rate	Acceleration rate in proportion to the maximum acceleration rate	58 [%]

Acceleration



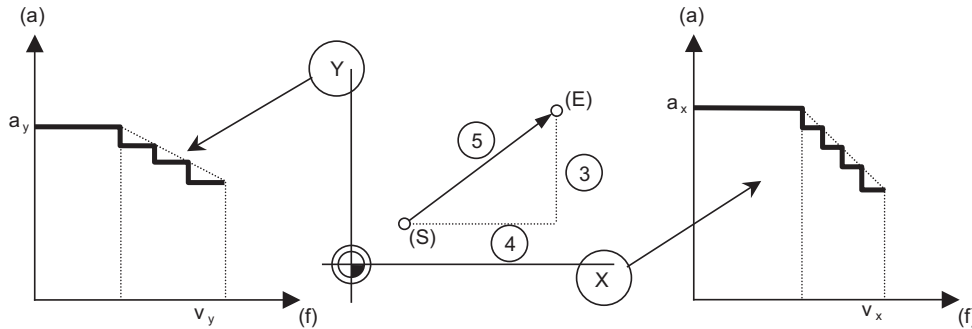
f: Speed

Acceleration pattern at two or more axis interpolation

When there are two or more rapid traverse axes with a different acceleration pattern, there are the following two operation methods.

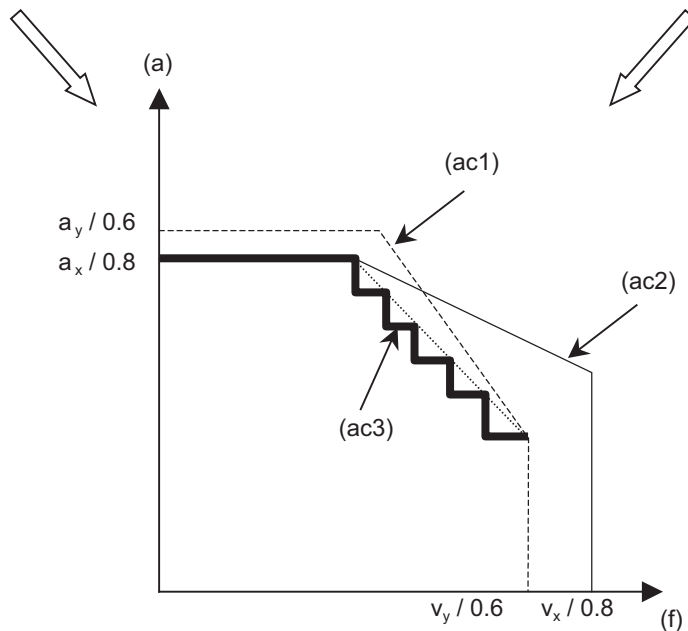
- Interpolation type (#1086 G0Intp = 0) : Moves from the start point to the end point by straight line
- Non-interpolation type (#1086 G0Intp = 1) : Each axis moves separately at the speed of the parameter

Rapid traverse constant inclination multi-step acceleration/deceleration are valid only for an interpolation type. For the interpolation type, the acceleration pattern operates to the maximum acceleration within the range where tolerable acceleration of each axis is not exceeded.



[Acceleration pattern of Y axis independently]

[Acceleration pattern of X axis independently]



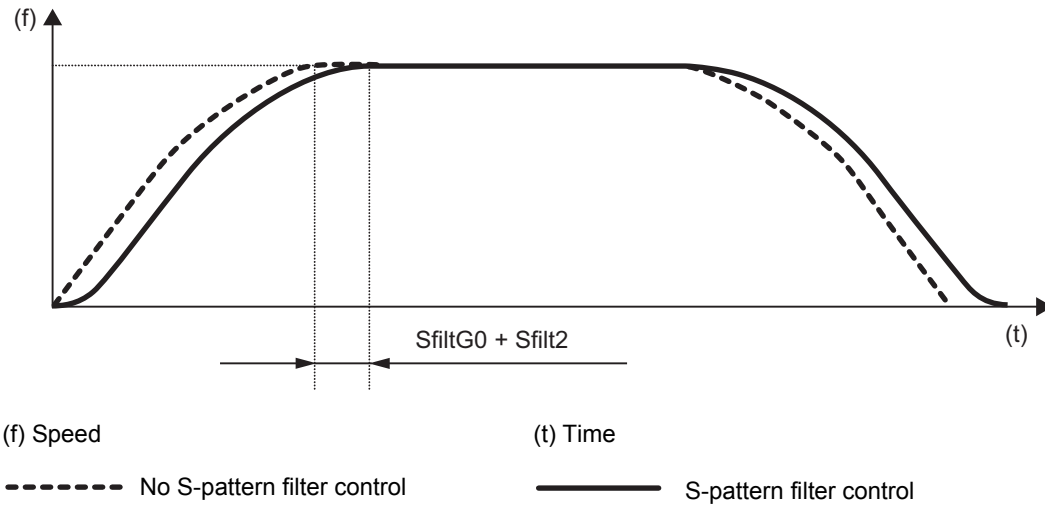
[Acceleration pattern of composite direction]

- (a) Acceleration
- (S) Start point
- (ac1) Acceleration pattern when the axis moved to composite direction at Y axis rapid traverse rate
- (ac2) Acceleration pattern when the axis moved to composite direction at X axis rapid traverse rate
- (ac3) Acceleration pattern of composite direction
- (f) Speed
- (E) End point

S-pattern filter control

With S-pattern filter control, this enables the rapid traverse inclination multi-step acceleration/deceleration fluctuation to further smoothen.

This can be set in the range of 0 to 200 (ms) with the base specification parameter "#1569 SfiltG0" (G00 soft acceleration/deceleration filter). With "#1570 Sfilt2" (Soft acceleration/deceleration filter 2), this also enables the acceleration/deceleration fluctuation to further smoothen.



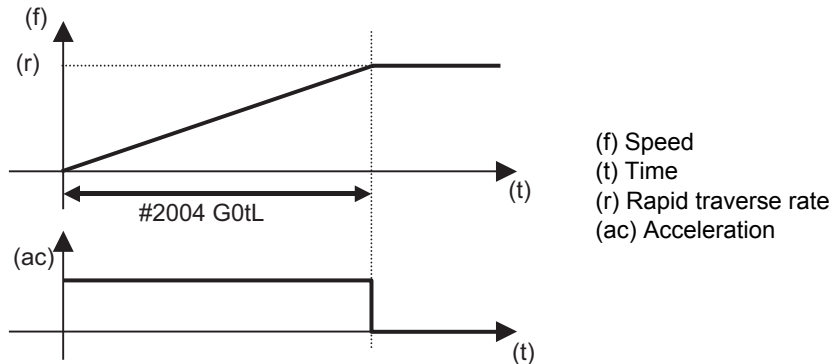
The high-accuracy control mode rapid traverse rate

During high-accuracy control, high-speed high-accuracy control I/II/III or high-accuracy spline control, the high-accuracy control mode's rapid traverse rate ("#2109 Rapid (H-precision)") can be set besides rapid traverse rate ("#2001 rapid").

Operation when the value is set at the high-accuracy control mode's rapid traverse rate is as follows.

(1) When "The high-accuracy control mode rapid traverse rate" > "rapid traverse rate"

This function is invalid and operates as "rapid traverse constant inclination acceleration/deceleration".

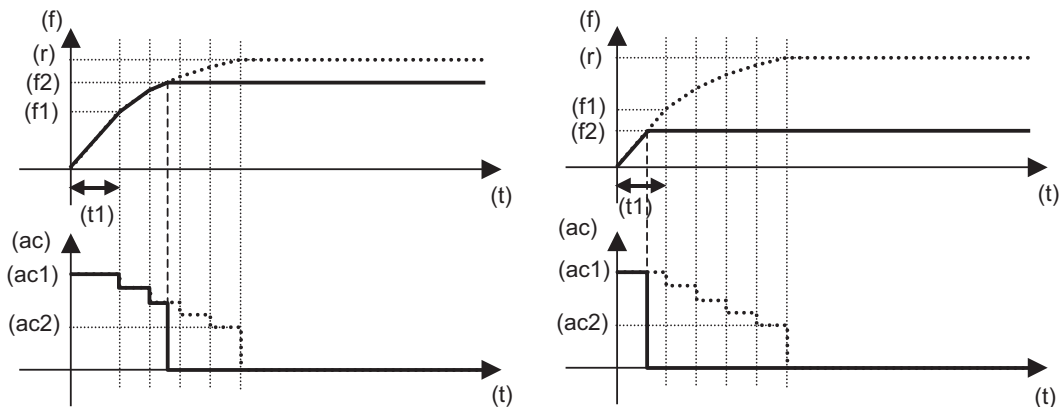


(2) When "The high-accuracy control mode rapid traverse rate" < "rapid traverse rate"

"The high-accuracy control mode rapid traverse rate" is applied according to acceleration pattern calculated from acceleration rate to "rapid traverse", "rated speed", "G00 time constant to rated speed" and "maximum acceleration".

Larger than the rated speed

Smaller than the rated speed



- (f) Speed
- (f1) Rated speed
- (f2) The high-accuracy control mode rapid traverse rate
- (t) Time
- (t1) Acceleration time to rated speed
- (ac) Acceleration
- (ac1) Max. Acceleration
- (ac2) Acceleration at rapid traverse rate
- (r) Rapid traverse rate



Precautions

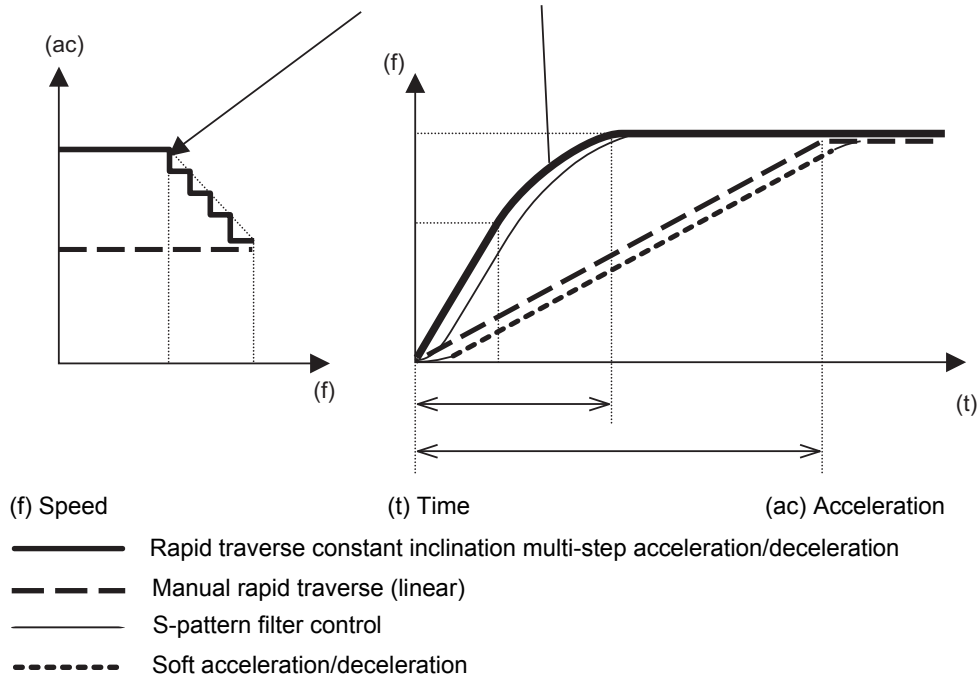
- (1) Rapid traverse constant inclination multi-step acceleration/deceleration are valid only for a rapid traverse command. Note that when the manual rapid traverse, rapid traverse constant inclination multi-step acceleration/deceleration cannot be used.

In this case, a time constant acceleration/deceleration (acceleration/deceleration after interpolation) is applied. So, acceleration/deceleration is decided by the following parameters.

- #2001 rapid : Rapid traverse rate
- #2003 smgst : Acceleration/deceleration mode
- #2004 G0tL: G00 time constant (linear)
- #2005 G0t1: G00 time constant (primary delay)

The acceleration time (time constant) is different between the rapid traverse constant inclination multi-step acceleration/deceleration and the manual rapid traverse as shown in figure.

Rapid traverse constant inclination multi-step acceleration/deceleration



- (2) Rapid traverse constant inclination multi-step acceleration/deceleration cannot be used in part system excluding 1st part system. However, even if two or more part systems are used, it is possible to use this function in case of the 1st part system.
- (3) When there is no specification for the rapid traverse constant inclination acceleration/deceleration, this function is invalid even if "2" is set to the parameter "#1205 G0bdcc". In this case, a normal time constant acceleration/deceleration (acceleration/deceleration after interpolation) is applied.
- (4) When G00 non-interpolation type ("#1086 G0Intp" = "1"), rapid traverse constant inclination multi-step acceleration/deceleration cannot be used. It is valid in interpolation mode only.
- (5) When the rapid traverse constant inclination multi-step acceleration/deceleration is applied, rapid traverse acceleration/deceleration types ("#2003 smgst" bit0 to bit3) are ignored.
- (6) When the rapid traverse constant inclination multi-step acceleration/deceleration is valid, G0 constant inclination ("#1200 G0_acc") cannot be used. Even if G0 constant inclination is valid ("#1200 G0_acc" = "1"), the setting is ignored.
- (7) When the rapid traverse constant inclination multi-step acceleration/deceleration is valid, programmable in-position check cannot be used. The in-position width will be ignored even if commanded.
- (8) This function cannot be used during the tool center point control.

7.9 Cutting Feed Constant Inclination Acceleration/Deceleration



Function and purpose

This function performs linear acceleration/deceleration at a constant inclination in the cutting feed mode. The inclination-constant acceleration/deceleration method can be more beneficial in reducing cycle time in comparison to the time-constant acceleration/deceleration method.



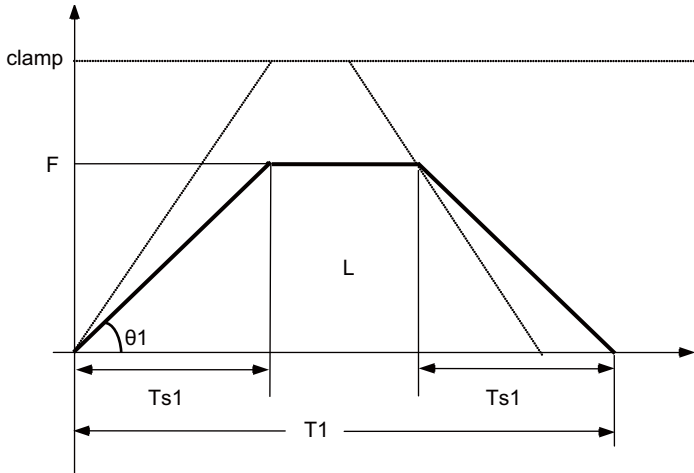
Detailed description

- (1) Cutting feed constant inclination acceleration/deceleration function is effective only when the commanded cutting feed acceleration/deceleration mode is linear method or soft method in linear interpolation (G01) command.
- (2) The program format of linear interpolation when cutting feed constant inclination acceleration/deceleration is executed is the same as when this function is invalid (time constant acceleration/deceleration).

(3) The acceleration/deceleration patterns in the case where cutting feed constant inclination acceleration/deceleration is performed are as follows.

[When the interpolation distance is long enough for the cutting feed rate to be achieved]

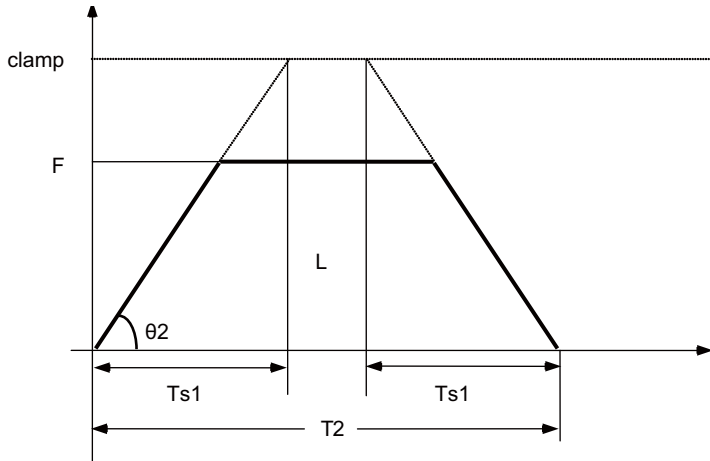
In case of time-constant acceleration/deceleration:



$$T1 = \frac{L}{F} + Ts1 \times 2$$

$$\theta1 = \tan^{-1} \left(\frac{F}{Ts1} \right)$$

In case of inclination-constant acceleration/deceleration:



$$T2 = \frac{L}{F} + \frac{Ts1 \times F}{clamp}$$

$$\theta2 = \tan^{-1} \left(\frac{clamp}{Ts1} \right)$$

clamp : Maximum cutting feed rate (Axis specification parameter "#2002 clamp")

F: Cutting feed rate

Ts1 : Acceleration/deceleration time (Axis specification parameter "#2007 G1tL")

θ1 : Acceleration/deceleration inclination (time-constant acceleration/deceleration)

θ2 : Acceleration/deceleration inclination (inclination-constant acceleration/deceleration)

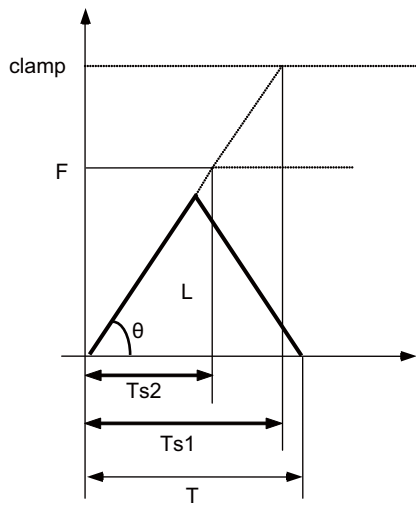
T1 : Interpolation time (time-constant acceleration/deceleration)

T2 : Interpolation time (inclination-constant acceleration/deceleration)

L: Interpolation distance

In the case of time-constant acceleration/deceleration, the acceleration/deceleration inclination is determined by the cutting feed rate. In the case of inclination-constant acceleration/deceleration, it's determined by the maximum cutting feed rate; therefore, the cycle time will be shorter than in the former case.

[When the interpolation distance is so short that the cutting feed rate is not achieved]



$$T = 2 \times \sqrt{Ts2 \times L / \text{clamp}}$$

$$\theta = \tan^{-1} \left(\frac{\text{clamp}}{Ts1} \right)$$

clamp : Maximum cutting feed rate (Axis specification parameter "#2002 clamp")

F : Cutting feed rate

Ts1 : Acceleration/deceleration time (Axis specification parameter "#2007 G1tL")

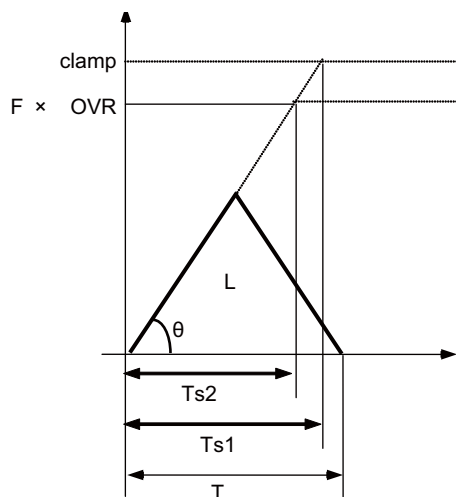
Ts2 : Acceleration/deceleration time to reach the cutting feed rate

θ : Acceleration/deceleration inclination

T : Interpolation time

L : Interpolation distance

[When the interpolation distance is so short that the maximum cutting feed rate is not achieved and the override for cutting feed inclination-constant acceleration/deceleration is activated]



$$T = 2 \times \sqrt{Ts2 \times L \times \text{OVR} / \text{clamp}}$$

$$\theta = \tan^{-1} \left(\frac{\text{clamp}}{Ts1} \right)$$

clamp : Maximum cutting feed rate (Axis specification parameter "#2002 clamp")

F : Cutting feed rate

OVR : Maximum override value for cutting feed inclination-constant acceleration/deceleration (Base Specifications Parameter "#1367 G1AccOVRMax")

Ts1 : Acceleration/deceleration time (Axis specification parameter "#2007 G1tL")

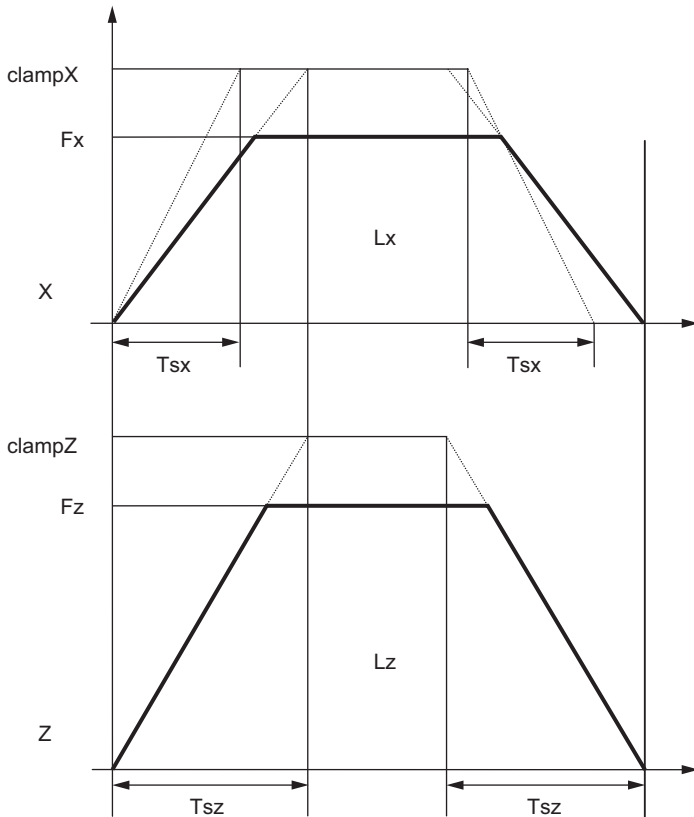
Ts2 : Acceleration/deceleration time to reach the cutting feed rate

θ : Acceleration/deceleration inclination

[2-axis simultaneous interpolation (When $T_{sx} < T_{sz}$, $L_x \neq L_z$)]

When 2-axis simultaneous interpolation is performed during linear interpolation inclination-constant acceleration/ deceleration, the acceleration/deceleration time of each axis will be set to the longest one among the acceleration/ deceleration times determined by maximum cutting feed rates (axis specification parameter "#2002 clamp"), cutting feed acceleration/deceleration time constant (axis specification parameter "#2007 G1tL"), cutting feed rates (F) and interpolation distances (L) of the simultaneously commanded axes.

Should the acceleration/deceleration time constant of an axis be greater than the cutting feed acceleration/ deceleration time constant (axis specification parameter "#2007 G1tL"), that axis accelerates/decelerates in accordance with the cutting feed acceleration/deceleration time constant (axis specification parameter "#2007 G1tL").

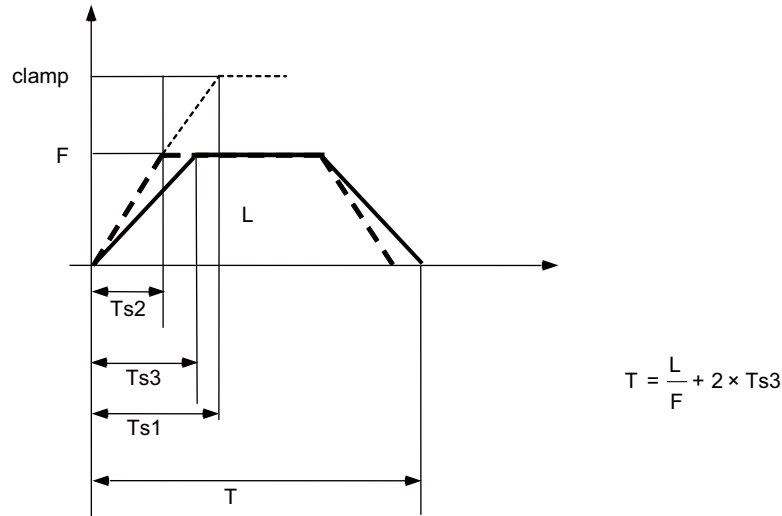


- Tsx : X axis acceleration/deceleration time
- Tsz : Z axis acceleration/deceleration time
- Lx : X axis interpolation distance
- Lz : Z axis interpolation distance
- Fx : X axis feed rate
- Fz : Z axis feed rate

When $T_{sx} < T_{sz}$, the acceleration/deceleration time of the block (T_s) will be equal to the acceleration/deceleration time T_{sz} (Z axis acceleration/deceleration time).

[When the feed rate is so low that the acceleration/deceleration time is shorter than the minimum time constant for inclination-constant acceleration/deceleration]

Acceleration/deceleration speed is adjusted to prevent the acceleration/deceleration time calculated by the cutting feed rate from going below the minimum time constant.



clamp : Maximum cutting feed rate (Axis specification parameter "#2002 clamp")

F: Cutting feed rate

Ts1 : Acceleration/deceleration time (Axis specification parameter "#2007 G1tL")

Ts2 : Acceleration/deceleration time to reach the cutting feed rate

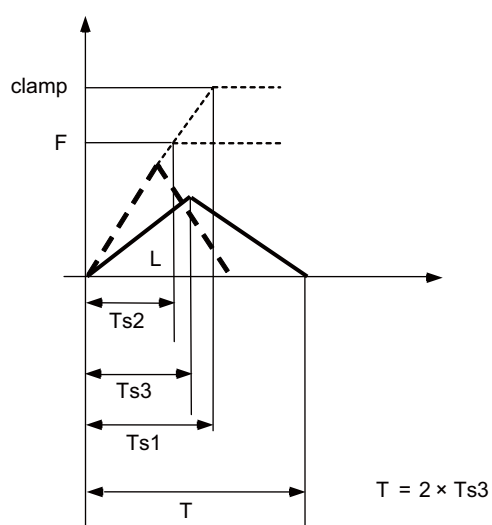
Ts3 : Minimum time for inclination-constant acceleration/deceleration (Axis specification parameter "#2199 G1tMin")

T : Interpolation time

L: Interpolation distance

[When the interpolation distance is so short that the acceleration/deceleration time is shorter than the minimum time constant for inclination-constant acceleration/deceleration]

Acceleration/deceleration speed is adjusted to prevent the acceleration/deceleration time calculated by interpolation distance from going below the minimum time constant.



clamp : Maximum cutting feed rate (Axis specification parameter "#2002 clamp")

F: Cutting feed rate

Ts1 : Acceleration/deceleration time (Axis specification parameter "#2007 G1tL")

Ts2 : Acceleration/deceleration time to reach the cutting feed rate

Ts3 : Minimum time for inclination-constant acceleration/deceleration (Axis specification parameter "#2199 G1tMin")

T : Interpolation time

L: Interpolation distance



Precautions

- (1) If a value greater than 100 (%) is designated for cutting feed override under cutting feed inclination constant acceleration/deceleration control, the acceleration/deceleration inclination becomes steeper as the feedrate increases.
To use the cutting feed override function at a rate higher than 100%, set the parameter "#1367 G1AccOVRMax" accordingly. (This parameter depends on the MTB specifications.) When the setting of this parameter is between 0 and 99 for "#1367 G1AccOVRMax", the override value is handled as 100% even if the specified cutting feed override is greater than 100%.
- (2) If there are one or more NC control axes that are set to soft acceleration/deceleration for G1, the parameter "#1367 G1AccOVRMax" setting will be ignored and the cutting feed override value is handled as 100%.
- (3) When "#2003 smgst" (acceleration/deceleration mode) is set to the soft acceleration/deceleration, and "#1219 aux03 bit7: Time constant setting changeover for soft acceleration/deceleration" is set to "1": Acceleration time is obtained by $G0tL+G0t1 (G1tL+G1t1)$, acceleration/deceleration speed is adjusted to prevent the sum of the 1st step and 2nd step acceleration/deceleration times from going below the minimum time constant for inclination-constant acceleration/deceleration.
- (4) When "#2003 smgst" (acceleration/deceleration mode) is set to the soft acceleration/deceleration, if the acceleration/deceleration is shorter than $G0tL$ (or $G1tL$), the 2nd step time constant will be reduced by the same rate as the 1st step time constant.
- (5) If the commanded travel distance in the block is small or the commanded linear interpolation (G01) feed rate is low, acceleration/deceleration time becomes quite short when the inclination-constant acceleration/deceleration method is enabled. Although this does contribute to reduce the cycle time, this can also be a cause of machine vibrations. In these cases, if the minimum time constant for inclination-constant acceleration/deceleration is set in parameter "#2199 G1tMin", it is possible to perform acceleration/deceleration to prevent the acceleration/deceleration time from being below this setting value.
This parameter depends on the MTB specifications.

7.10 Exact Stop Check; G09



Function and purpose

In order to prevent roundness during corner cutting and machine shock when the tool feedrate changes suddenly, there are times when it is desirable to start the commands in the following block once the in-position state after the machine has decelerated and stopped or the elapsing of the deceleration check time has been checked. The exact stop check function is designed to accomplish this purpose.

A deceleration check is performed when the G09 (exact stop check) command has been designated in the same block. The G09 command is unmodal.

Either the deceleration check time or in-position state is based on the parameter settings specified by the MTB. (Refer to section "7.12 Deceleration Check".)

The in-position width is set in servo parameter "#2224 sv024", "#2077 G0inps" or "#2078 G1inps". This parameter also depends on the MTB specifications.



Command format

Exact stop check

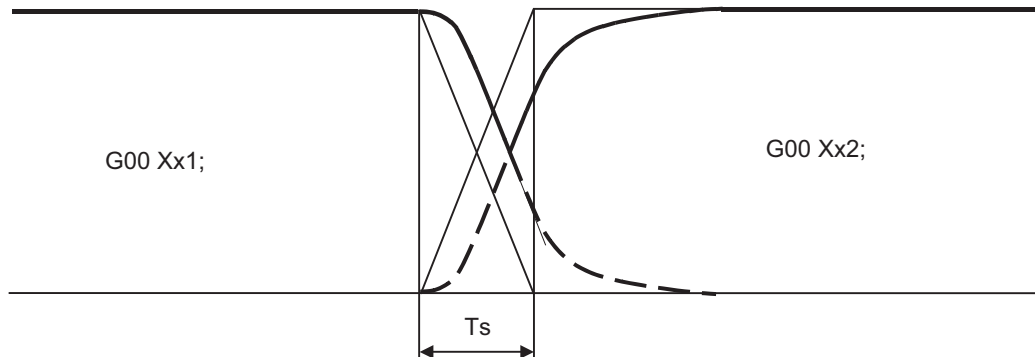
```
G09 G01 (G02, G03);
```

The exact stop check command G09 has an effect only with the cutting command (G01 - G03) in its particular block.

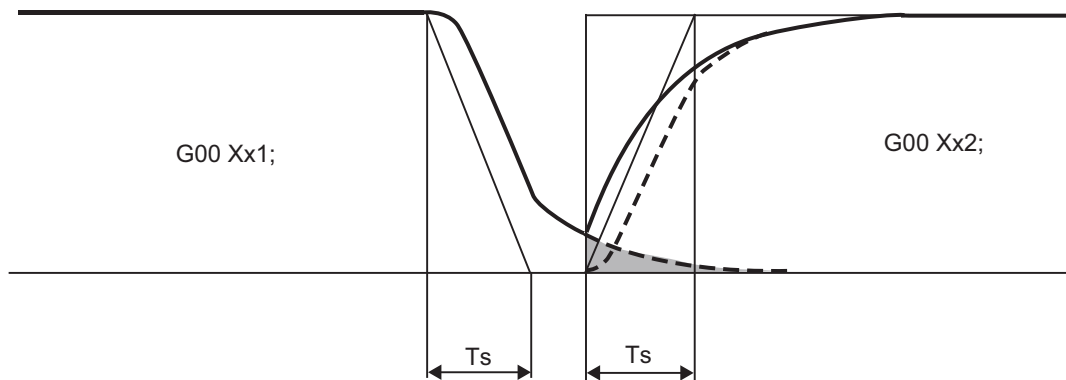


Detailed description

[With continuous cutting feed]



[With cutting feed in-position check]

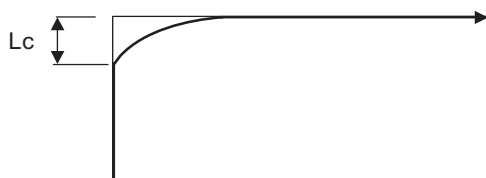


Ts : Cutting feed acceleration/deceleration time constant

In-position width

The in-position width, as shown in the figure above, is the remaining distance (shaded area in the above figure) of the previous block when the next block is started is set in the servo parameter "#2224 sv024". (This depends on the MTB specifications.)

The in-position width is designed to reduce the roundness at the workpiece corners to below the constant value.

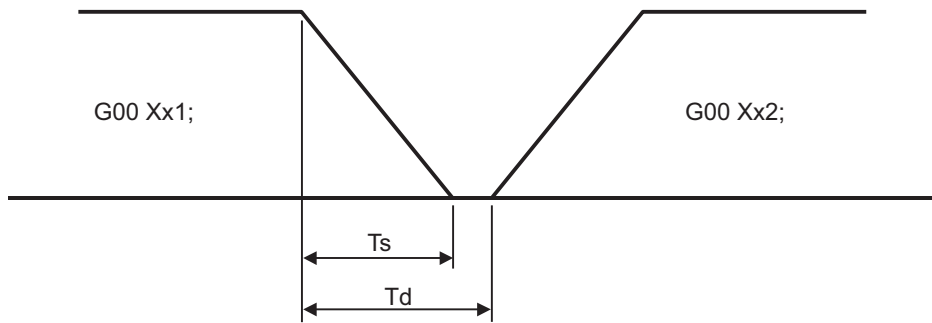


To eliminate corner roundness, set the value as small as possible to servo parameter "#2224 sv024" and perform an in-position check or assign the dwell command (G04) between blocks.

(The parameter setting depends on the MTB specifications.)

With deceleration check

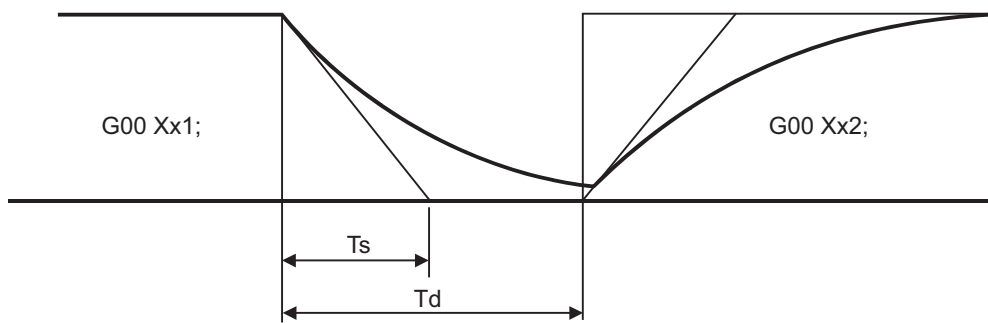
(1) With linear acceleration/deceleration



Ts: Acceleration/deceleration time constant

Td: Deceleration check time $Td = Ts + \alpha$ (0 to 10ms)

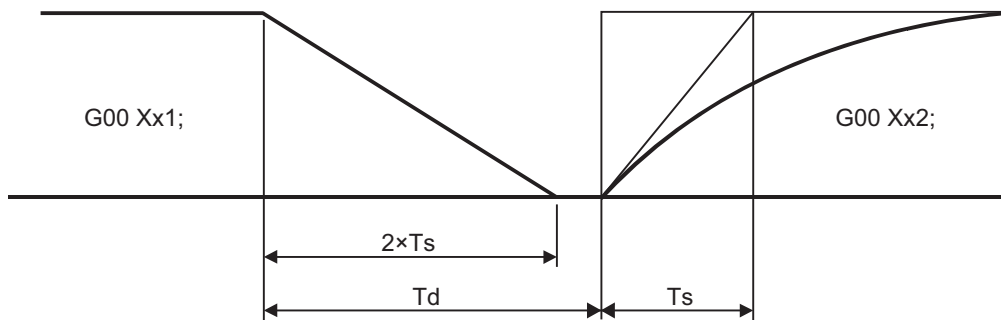
(2) With exponential acceleration/deceleration



Ts: Acceleration/deceleration time constant

Td: Deceleration check time $Td = 2 \times Ts + \alpha$ (0 to 10ms)

(3) With exponential acceleration/linear deceleration



Ts: Acceleration/deceleration time constant

Td: Deceleration check time $Td = 2 \times Ts + \alpha$ (0 to 10ms)

The time required for the deceleration check during cutting feed is the longest among the cutting feed deceleration check times of each axis determined by the cutting feed acceleration/deceleration time constants and by the cutting feed acceleration/ deceleration mode of the axes commanded simultaneously.

Note

(1) To execute exact stop check in a fixed cycle cutting block, insert command G09 into the fixed cycle subprogram.

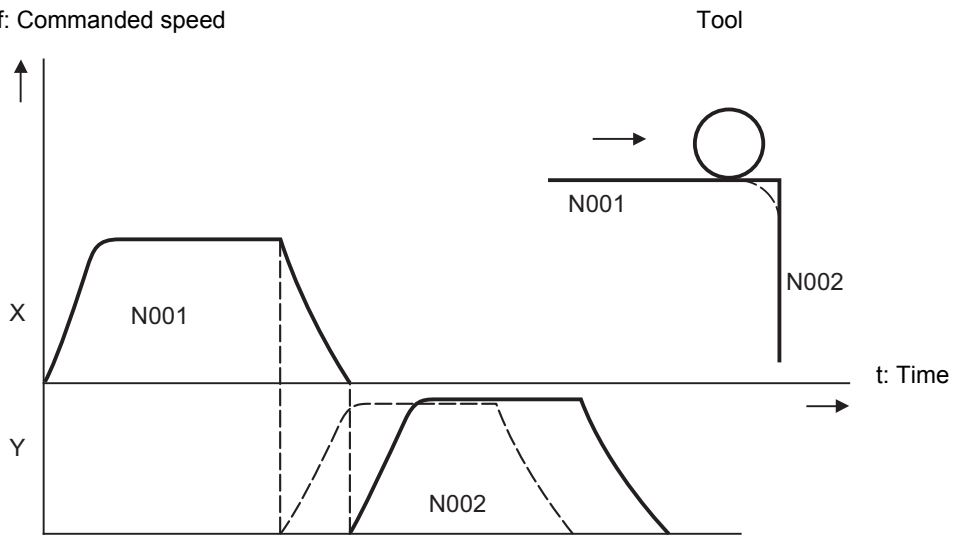


Program example

N001 G09 G01 X100.000 F150 ;	The commands in the following block are started once the deceleration check time or in-position state has been checked after the machine has decelerated and stopped.
N002 Y100.000 ;	

[Exact stop check result]

f: Commanded speed



Solid line indicates speed pattern with G09 command

Broken line indicates speed pattern without G09 command

7.11 Exact Stop Check Mode; G61



Function and purpose

Whereas the G09 exact stop check command checks the in-position status only for the block in which the command has been assigned, the G61 command functions as a modal. This means that deceleration will apply at the end points of each block to all the cutting commands (G01 to G03) subsequent to G61 and that the in-position status will be checked.

The modal command is released by the following commands.

G61.1..... High-accuracy control mode

G62 Automatic corner override

G63 Tapping mode

G64 Cutting mode



Command format

G61 ; ... Exact stop check mode

In-position check is executed when the G61 command has been selected, and thereafter, the in-position check is executed at the end of the cutting command block until the check mode is canceled.

7.12 Deceleration Check

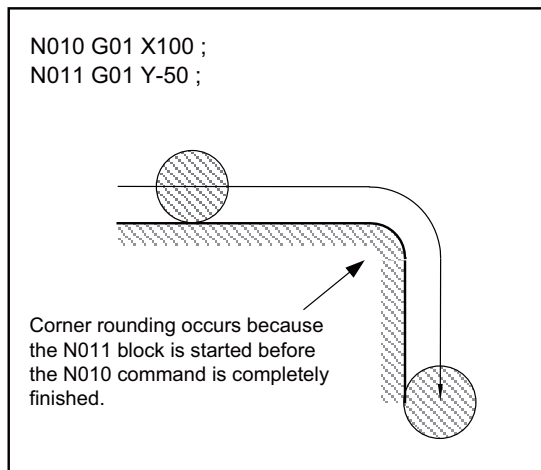
7.12.1 Deceleration Check



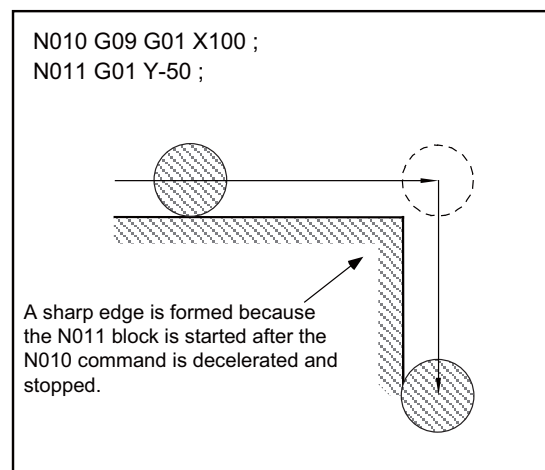
Function and purpose

The deceleration check reduces the machine shock that occurs when the control axis feedrate is suddenly changed and prevents corners from becoming rounded. This is accomplished by decelerating the motor to a stop at axis movement block joints before the next block is executed.

Without deceleration check



With deceleration check



The conditions for executing a deceleration check are described below.

- (1) Deceleration check in the rapid traverse mode
In the rapid traverse mode, the deceleration check is always performed when block movement is completed before executing the next block.
- (2) Deceleration check in the cutting feed mode
In the cutting feed mode, the deceleration check is performed and the program starts moving the next block when one of the following conditions is satisfied.
 - (a) When G61 (Exact stop check mode) is selected
 - (b) When the G09 (Exact stop check) command has been designated in the same block
 - (c) When the error detect switch (PLC signal) is ON

There are three methods for deceleration check: command deceleration check method, smoothing check method, and in-position check method.

The method that is selected for rapid traverse or cutting feed depends on the MTB specifications (combination of parameters "#1306 InpsTyp", "#1389 G1SmthChk", "#1223 aux07/bit1", and "#1193 inpos").

Depending on the MTB specifications, different deceleration check methods may be used for each feed command during rapid traverse command and cutting feed command (parameter "#1306 InpsTyp").



Detailed description

Behavior for each combination of movement commands

Next block	Current block		
	G00	G01	G00/G01 without movement
G00	○	(○) (*1)(*2)	×
G01	○	(○) (*1)(*3)	×
Others	○	(○) (*1)	×

○ : Deceleration check is performed.

(○): (*1) A deceleration check is performed when the error detection signal is ON or when G09 or G61 is enabled.

(*2) A command deceleration check is performed when G01 => G00 block is specified, "#1502 G0lpfg" is ON or the movement reverses to the opposite direction.

(*3) A command deceleration check is performed when G01 => G01 block is specified, "#1503 G1lpfg" is ON or the movement reverses to the opposite direction.

For the deceleration check when movement in the opposite direction is reversed, refer to "7.12.2 Deceleration Check When Movement in the Opposite Direction Is Reversed".

A deceleration check is not performed if the above conditions are not satisfied.

× : Deceleration check is not performed.

Types of deceleration check

<p>(1) Command deceleration check method Deceleration is completed after the deceleration check time (*1) has passed after the interpolation.</p>	
<p>(2) Smoothing check method Deceleration is completed after the deceleration check time (*1) has passed after the interpolation and all axis smoothing has become zero.</p>	
<p>(3) In-position check method The deceleration is completed after the deceleration check time (*1) had passed after the interpolation, all axis smoothing has become zero, and all axes have become in-position.</p>	

(*1) Deceleration check time is automatically calculated from the acceleration/deceleration mode and time constant.

Selecting deceleration checks (MTB specifications)

(1) When a rapid traverse command (G00/G53) block is to be executed

Parameters		Deceleration check method	Conditions of deceleration check
#1193 inpos			
0		Command deceleration check method	Deceleration check time has elapsed.
1		In-position check method	Deceleration check time has elapsed, all axis smoothing has become zero, and all axes have become in-position.
2		Smoothing check method	Deceleration check time has elapsed, and smoothing zero for all axes.

(2) When a cutting command (G01/G02/G03) block is to be executed

When parameter "#1306 InpsTyp" is "0", the following occurs (MTB specifications).

Parameters		Deceleration check method	Conditions of deceleration check
#1389 G1SmthChk	#1223 aux07/bit1		
0	0	Command deceleration check method	Deceleration check time has elapsed.
	1	In-position check method	Deceleration check time has elapsed, all axis smoothing has become zero, and all axes have become in-position.
1	-	Smoothing check method	Deceleration check time has elapsed, and smoothing zero for all axes.

When parameter "#1306 InpsTyp" is "1", the same method as for rapid traverse in (1) is used regardless of the value of parameter "#1389 G1SmthChk".

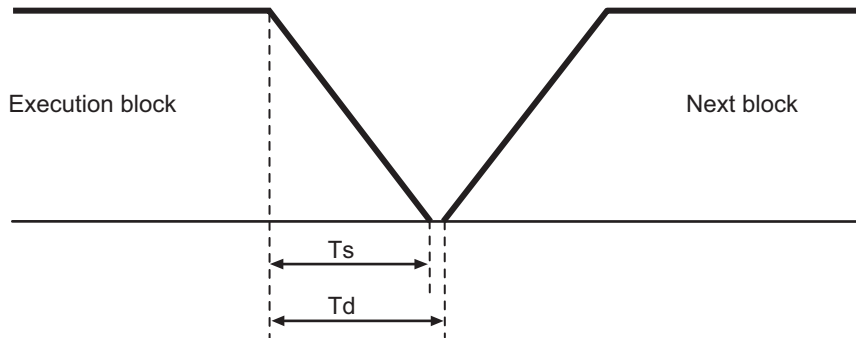
Command deceleration check method

Execution of the next block starts after confirming that the deceleration of the command system is completed upon completion of interpolation for one block.

The following explains an example of transition from the current block (rapid traverse) to the next block.

The time required for the deceleration check is the longest among the deceleration check times of each axis determined by the acceleration/deceleration mode and time constants of the axes commanded simultaneously.

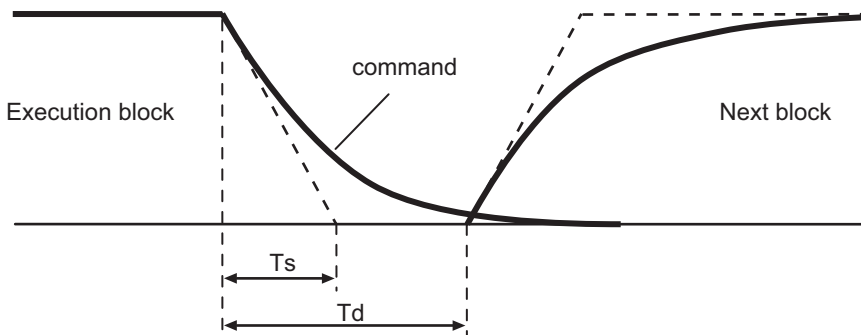
(a) For linear acceleration/deceleration



(Ts) Linear acceleration/deceleration time constant

(Td) Deceleration check time: $T_d = T_s + \alpha$ (0 to 10 ms)

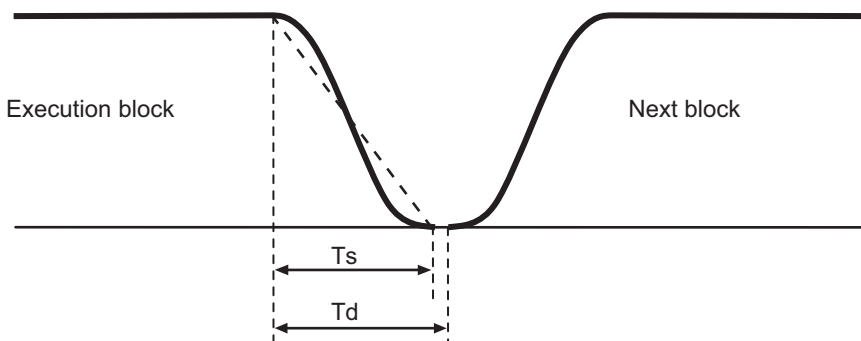
(b) For exponential acceleration/deceleration



(Ts) Exponential acceleration/deceleration time constant

(Td) Deceleration check time: $T_d = 2 \times T_s + \alpha$ (0 to 10 ms)

(c) For soft acceleration/deceleration



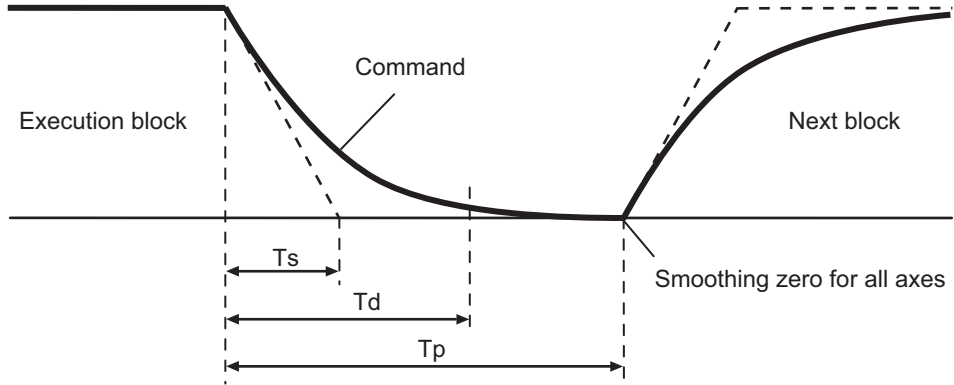
(Ts) Soft acceleration/deceleration time constant

(Td) Deceleration check time: $T_d = 2 \times T_s + \alpha$ (0 to 10 ms)

Smoothing check method

Execution of the next block starts after the command deceleration check is performed and after confirming that the smoothing for all axes in the part system has reached zero.

For exponential acceleration/deceleration



- (T_s) Exponential acceleration/deceleration time constant
- (T_d) Deceleration check time
- (T_p) Waiting time for a block to complete

In-position check method

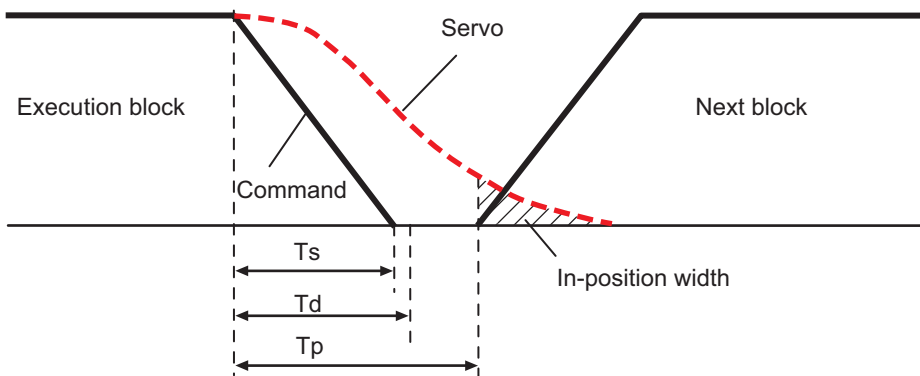
Execution of the next block starts after the command deceleration check is performed and after confirming that the remaining distances for all axes in the part system are below certain values.

The confirmation of the remaining distance should be done with the imposition width.

The bigger one of the servo parameter "#2224 SV024" or G0 in-position width "#2077 G0inps" (For G01, in-position width "#2078 G1inps"), will be adapted as the in-position width.

(For a rotary axis, the setting value of spindle parameter "#13024 SP024" is assumed to be the in-position width.)

With linear acceleration/deceleration



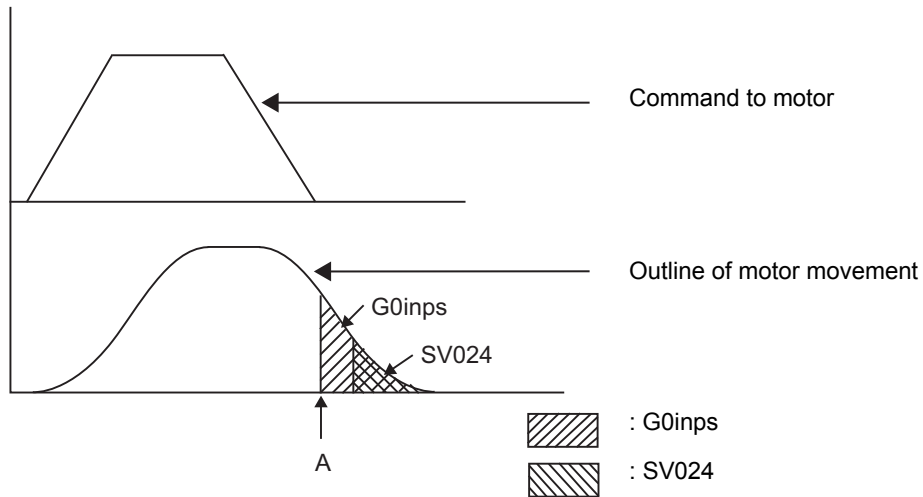
- (T_s) Linear acceleration/deceleration time constant
- (T_d) Deceleration check time
- (T_p) Waiting time for a block to complete

As shown in the figure above, the in-position width is the remaining distance from the previous block at the start of the next block. (Shaded area of the figure above).

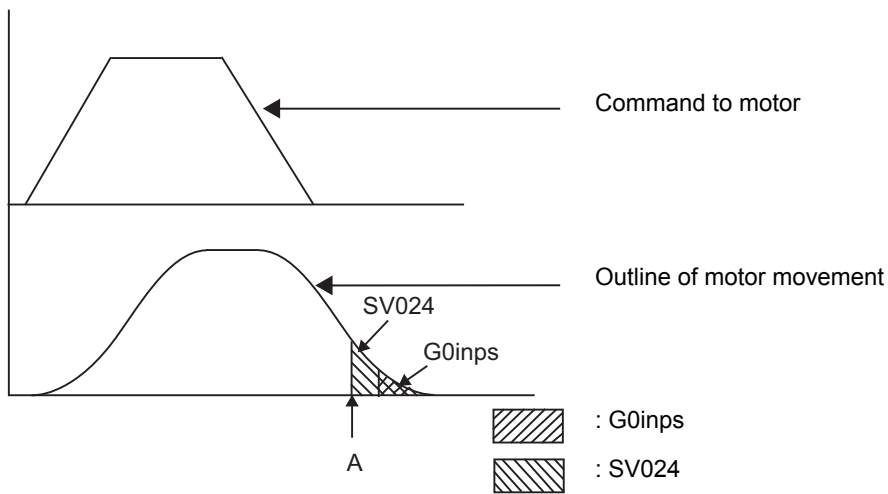
The purpose of the deceleration check is to minimize the positioning time. The bigger the setting value for the in-position width, the shorter the time is, but the remaining distance of the previous block at the start of the next block also becomes larger, and this could become an obstacle in the actual processing work.

The check for the remaining distance is done at set intervals. Accordingly, it may not be possible to get the effect of time reduction for positioning as in-position width setting value.

(1) In-position check by the G0inps: When $SV024 < G0inps$ (Stop is judged at A in the figure.)



(2) In-position check by the SV024: When $G0inps < SV024$ (Stop is judged at A in the figure.)



Programmable in-position width command

This command commands the in-position width for the positioning command from the machining program.

```
G00 X_ Z_(Y_) ,I_ ;
```

X,Z,(Y_)	Positioning coordinate value of each axis
,I	In-position width (setting range: 1 to 999999)

Execution of the next block starts after confirming that the position error amount in the block in which the deceleration check is to be performed is less than the in-position width.

The in-position width specified by parameter (SV024, G0inps (G1inps for G01)) or the one specified in the program, whichever is greater, will be adapted as the in-position width.

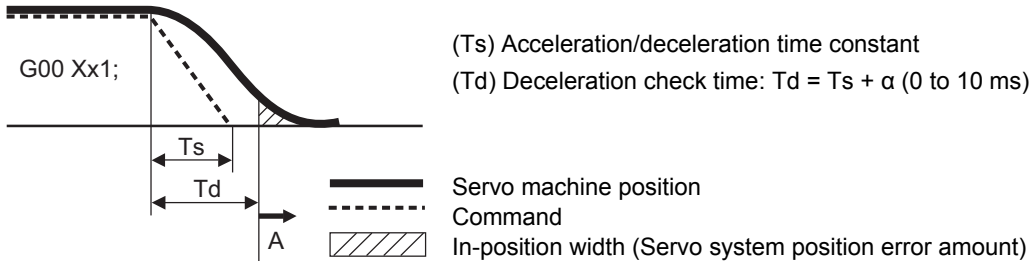
When there are several movement axes, the system confirms that the position error amount of each movement axis in each part system is less than the in-position width issued in this command before executing the next block. For ",I" command, also refer to "6.1 Positioning (Rapid Traverse); G00".

The differences of In-position check

The differences between the in-position check with parameter and with programmable command are as follows:

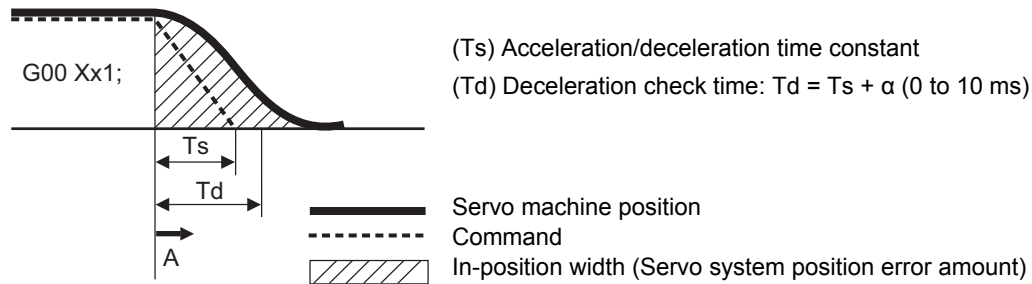
(1) In-position check with parameter

After completing deceleration of the command system ("A" in the figure), the servo system's position error amount and the parameter setting value (in-position width) are compared.



(2) In-position check with programmable command (",I" address command)

After starting deceleration of the command system ("A" in the figure), the position error amount and commanded in-position width are compared.





Relationship with Other Functions

Tool compensation

The deceleration check acts on the compensated block when tool compensation is performed.

Automatic error detection

The automatic error detection is disabled in a block in which deceleration check is enabled.

High-speed machining mode

When the axis movement reverses to the opposite direction in a G01 → G01 successive block during the high-speed machining mode other than high-speed machining mode I (G05 P1) the commanded deceleration will not take place even if parameter "#1503 G1lpfg" is set to 1.

Note that the G0lpfg setting will be followed if the axis direction reverses to the opposite direction in a G01 → G00 successive block.

High-speed simple program check

A deceleration check is performed even when high-speed simple program check is running. During high-speed simple program checking, the deceleration check time is reduced according to the time reduce coefficient.



Precautions

- (1) When the in-position check is valid, the parameter for the in-position width "#2224 SV024" must be set. (Based on the MTB specifications.)
- (2) This function is disabled for an axis to which automatic machine lock is applied.
- (3) If MSTB is commanded in the block that follows a cutting command, the MSTB code is output before deceleration is completed in the cutting command. If an MSTB command must be executed after the completion of axis movement, check the PLC signals (DEN) before executing it. (The behavior depends on the MTB specifications.)
- (4) If there is an axis in control axis synchronization/superimposition in the part system for which the in-position check method is specified, deceleration is considered to be completed when all axis smoothing has become zero. (Equivalent to smoothing check method)
- (5) If thread cutting commands are specified in succession, a deceleration check is not carried out at block joints.
- (6) If the parameter "#1205 G0bdcc" is set to "1", the value set with the parameter "#2224 SV024" (in-position detection width) will be used as the in-position width.
The setting of the parameter "#2077 G0inps" (G0 in-position width) and the programmable in-position check with ",I" address are disabled. These parameters depend on the MTB specifications.

7.12.2 Deceleration Check When Movement in the Opposite Direction Is Reversed

**Function and purpose**

A deceleration check cannot be designated for G01 → G00 or G01 → G01, but it can be designated in the following manner only when the movement reverses to the opposite direction in successive blocks.

A deceleration check can also be executed if even one axis is moving in the opposite direction while several axes are interpolating.

For the relation with other functions and precautions, refer to "Deceleration Check".

**Detailed description****Designating deceleration check for G01 → G00 opposite direction movement reversal**

If the axis movement reverses to the opposite direction in a G01 to G00 successive block, the deceleration check for the movement in the opposite direction can be changed with the MTB specifications (parameter "#1502 G01pfg").

	Same direction	Opposite direction
G01pfg:0		<p>The acceleration is excessive due to the G01 and G00 composite speed.</p>
G01pfg:1		<p>Command deceleration</p>

Example of program: When there is a deceleration check in the movement of several axes

(1)

G91 G01 X100. Y100. F4000 ; G00 X-100. Y120. ;	A deceleration check is carried out, because the X axis moves in the reverse direction in the program.
---	--

(2)

G91 G01 X100. Y-100. F4000 ; G00 X80. Y100. ;	A deceleration check is carried out, because the Y axis moves in the reverse direction in the program.
--	--

(3)

G90 G01 X100. Y100. F4000 ; G00 X80. Y120. ;	A deceleration check is carried out, because the X axis moves in the reverse direction in the program. (When the program start position is X0 Y0)
---	--

(4)

G91 G01 X100. Y100. F4000 ; G00 X100. Y100. ;	A deceleration check is not carried out, because both the X axis and the Y axis move in the same direction in the program.
--	--

(5)

G91 G01 X100. Y80. F4000 ; G00 X80. ;	A deceleration check is not carried out, because the X axis moves in the same direction, and there is no Y axis movement command in the program.
--	--

Designating deceleration check for G01 -> G01 opposite direction movement reversal

If the axis movement reverses to the opposite direction in a G01 to G01 successive block, the deceleration check for the movement in the opposite direction can be changed with the MTB specifications (parameter "#1503 G1lpfg").

	Same direction	Opposite direction
G1lpfg:0		
G1lpfg:1		

Example of program: When there is a deceleration check in the movement of several axes

(1)

G91 G01 X100. Y100. F4000 ; G01 X-100. Y120. ;	A deceleration check is carried out, because the X axis moves in the reverse direction in the program.
---	--

(2)

G91 G01 X100. Y-100. F4000 ; G01 X80. Y100. ;	A deceleration check is carried out, because the Y axis moves in the reverse direction in the program.
--	--

(3)

G90 G01 X100. Y100. F4000 ; G01 X80. Y120. ;	A deceleration check is carried out, because the X axis moves in the reverse direction in the program. (When the program start position is X0 Y0)
---	--

(4)

G91 G01 X100. Y100. F4000 ; G01 X100. Y100. ;	A deceleration check is not carried out, because both the X axis and the Y axis move in the same direction in the program.
--	--

(5)

G91 G01 X100. Y80. F4000 ; G01 X80. ;	A deceleration check is not carried out, because the X axis moves in the same direction, and there is no Y axis movement command in the program.
--	--

7.13 Rapid Traverse Block Overlap; G0.5 P1

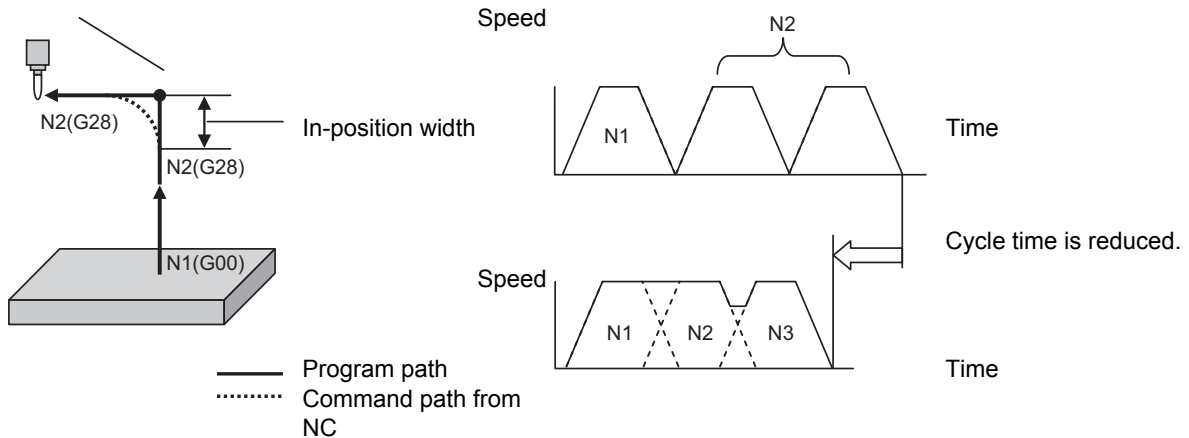


Function and purpose

This function enables the next block to start (overlap) without waiting for positioning (G00) or reference position return (G28/G30).
 Consequently, cycle time of machining including operation of positioning (G00) or reference position return (G28/G30) can be reduced.
 Adjust the overlap amount according to the command issued by the machining program or with the parameter, and specify it as in-position width for rapid traverse block overlap.
 Also, the operation does not decelerate between blocks if the movement command continues in same direction.
 The overlap is also valid when G00 is followed by a G01 block, rather than G00 or G28/G30.
 It is not invalid when G28 is followed by G00 or G28/G30.
 The validity of this function depends on the MTB specifications.

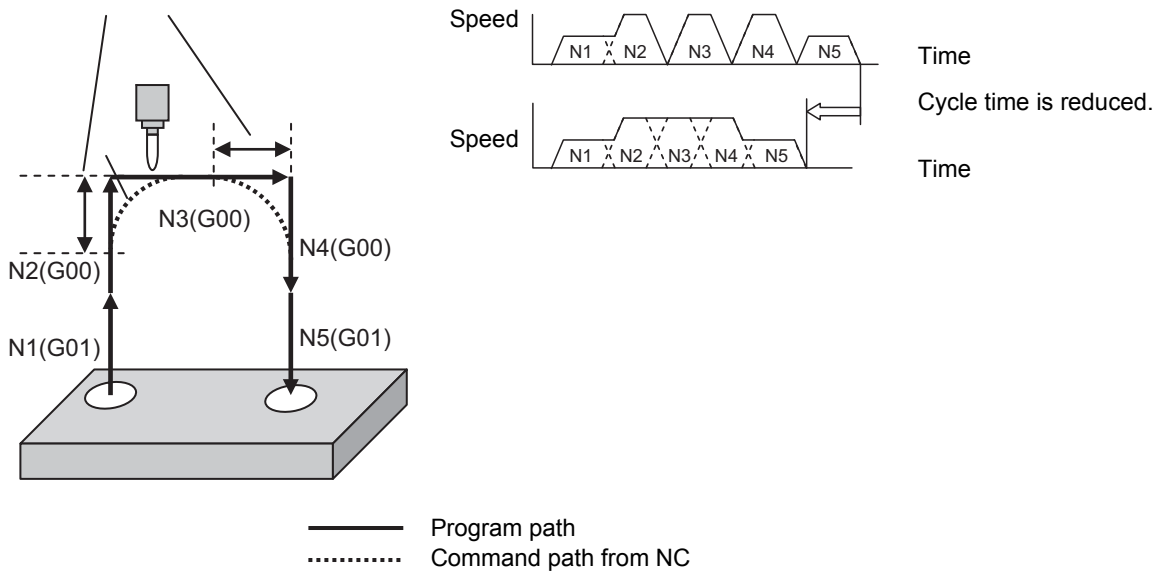
Example of behavior and velocity waveform 1 (example of application of rapid traverse block overlap in tool change motion)

Intermediate point



Example of behavior and velocity waveform 2 (example of application of rapid traverse block overlap in continuous drilling motion)

In-position width



Deceleration check method using in-position width

For a deceleration check method that uses the in-position width for rapid traverse (G00) or reference position return (G28/G30), a function with a higher priority that is enabled will be applied.

If none of the functions is enabled, the command deceleration is carried out.

Function (Deceleration check method)	Enabling conditions	Deceleration Check Enabled behavior	Priority
Programmable in-position	Valid when the in-position width is designated with address "I" in the same block as G00. (It is valid only for a block in which address "I" is specified.) (For details, refer to "6.1 Positioning (Rapid Traverse); G00" and "7.12 Deceleration Check".)	G00	1
Rapid traverse block overlap (this function)	(1) For G00 Parameter "#1442 G0ol" must be "1" and G00 rapid traverse block overlap must be valid modal code (G0.5P1). (2) For G28/G30 Parameter "#1443 G28ol" must be "1".	G00/G28/G30	2
In-position check by parameter settings	Parameter "#1193 inpos" must be "1". (For details, refer to "7.12 Deceleration Check".)	G00	3

For G00 overlap, refer to "7.13.1 Rapid Traverse Block Overlap for G00; G0.5". For G28/G30 overlap, refer to "7.13.2 Rapid Traverse Block Overlap for G28".

7.13.1 Rapid Traverse Block Overlap for G00; G0.5

**Function and purpose**

This function enables the next block to start (overlap) without waiting for positioning (G00) or reference position return (G28/G30).

For the rapid block overlap function, also refer to "7.13 Rapid Traverse Block Overlap; G0.5 P1".

G28/G30 can be overlapped when the rapid traverse block overlap for G28 is enabled. For details, refer to "7.13.2 Rapid Traverse Block Overlap for G28".

**Command format****Starting rapid traverse block overlap for G00**

```
G0.5 P1 J_ K_;
```

P	Starting or canceling the rapid traverse block overlap function (0: Cancel, 1: Start)
J	Liner axis in-position width (0.000 to 1000.000 (mm))
K	Rotary axis in-position width (0.000 to 1000.000 (°))

Canceling rapid traverse block overlap for G00

```
G0.5 P0;
```

Note

- (1) A program error (P35) will occur unless this block is independently specified.
- (2) This block can be specified simultaneously with an N code (sequence number).
- (3) The in-position width at joints between two blocks containing G28/G30 cannot be changed with G0.5P1 command.
- (4) G0.5P1 and G0.5P0 are modal.
- (5) Address J in G20 must be programmed in inches.
- (6) If an address is omitted, the width determined by the MTB specifications becomes valid. (Parameters "#2224 SV024" and "#13024 SP024")
If a value less than the width determined by the MTB is specified, that width becomes valid.
- (7) If address J or K is set to "0", the conventional deceleration check is performed.



Detailed description

Enabling conditions

The rapid traverse block overlap function for G00 becomes effective when all of the following conditions are satisfied.

- (1) The rapid traverse block overlap for G00 must be enabled.

Refer to the MTB specifications (parameter "#1442 G0oI").

- (2) G0.5P1 modal must be active.

To make G0.5P1 modal active:

- Specify a G code (G0.5P1) in which rapid traverse block overlap is enabled in the machining program.
- Set parameter "#12056 I_G0oL G00" to "1" (valid).

N1 G0.5 P1;	Rapid traverse block overlap function: Enabled
N2 G91 G00 X10.;	
N3 G00 X20.;	
N4 G0.5 P0;	Rapid traverse block overlap function: Disabled
:	

Motion subject to rapid traverse block overlap

- (1) When the rapid traverse block overlap for G00 is enabled, a G code (positioning (G00) or linear interpolation (G01)) following positioning (G00) may not be subject to rapid traverse block overlap depending on the current control mode or parameter settings that are specified by the MTB. (Parameters "#1086 G0intp" and "#1205 G0bdcc")

For details, refer to the table below.

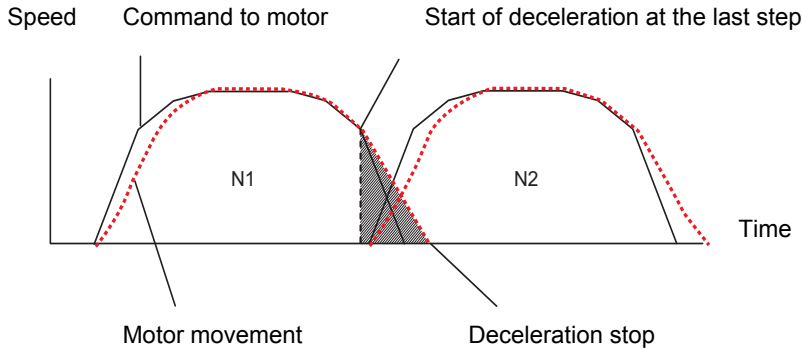
Control mode	Parameters		G code following positioning (G00)	
	#1086	#1205	G00	G01
High-accuracy mode OFF	0	0	○	○
		1	○	×
		2	○	×
	1	0/1/2	○	○
ON	0	0	○	×
		1	○	×
		2	○	×
	1	0/1/2	○	×

○ : Motion subject to rapid traverse block overlap for G00

× : Motion not subject to rapid traverse block overlap for G00

(2) When executing a rapid traverse block overlap in G00 multi-step acceleration/deceleration, the next block (N2 in the following program) will be started after the deceleration at the last step in the execution block (N1) has started. The hatched area in the figure below is an area in which the in-position width can be specified.

```
N1 G91 G00 X10.;
N2 X10.;
```



(3) When the rapid traverse block overlap for G00 is enabled, this function is valid if positioning (G00) is followed by a fixed cycle, subprogram or macro call command block. In addition, this function is valid if a fixed cycle, subprogram or macro program contains consecutive move commands to which this function is applied. (If the in-position width is specified in a fixed cycle command, that value is given priority.)

Adjustment of start position of overlap

The start position of overlap when a rapid traverse block overlap for G00 is executed can be adjusted with the in-position width. The next block is started when the remaining distances of all movement axes in the current movement block are smaller than the in-position width. (Refer to following figure.)

When setting the in-position width with J and K commands, set a value for each linear and rotary axis. Setting the in-position width for axes with parameter settings depend on the MTB specifications (parameter "#2631 G00inps").

The start position of the next block based on the remaining distance and in-position width for each movement axis is shown below.

This shows an example of when the X axis in-position width is set to 0.5 mm and the Y axis in-position width to 1 mm.

(a) For X axis

Program example

```
N1 G91 G00 X50.;
N2 Y50.;
```

Operation
Start position of N2 block

X axis in-position width = 0.5mm

(b) For Y axis

Program example

```
N1 G91 G00 Y50.;
N2 X-50.;
```

Operation
Y axis in-position width = 1mm

Start position of N2 block

(c) For X and Y axes

Program example

```

N1 G91 G00 X50. Y50.;
N2 X-50. Y50.;
        
```

Operation

If the position error amount is smaller than the in-position width for both of the X and Y axes, the next block is started.

If the Y axis position error amount is smaller than the in-position width, but the X axis position error amount is larger than that width, the next block is not started.

X axis in-position width = 0.5mm

Y axis in-position width = 1mm

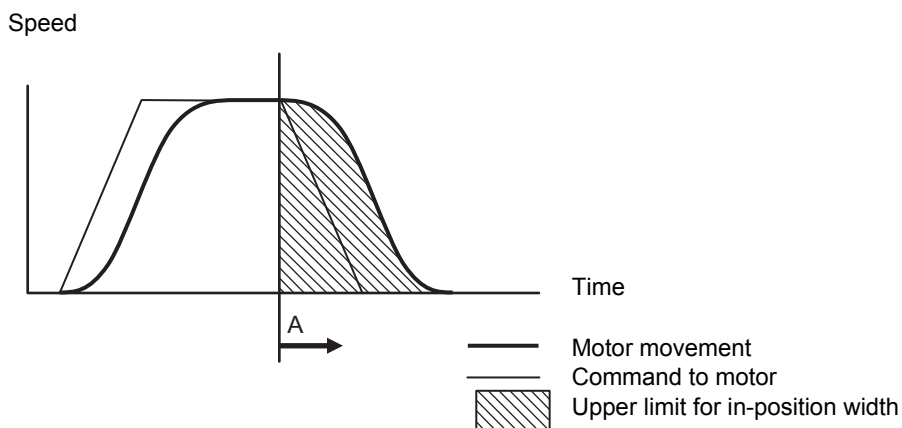
Start position of N2 block

The in-position width is determined by the G code address or parameter value.

- (1) When specifying the in-position width with a G code, the one specified with address J/K becomes effective. Note that if address J or K is set to "0", the rapid traverse overlap is disabled.
- (2) If a command with address J/K is omitted, the in-position width determined for each of positioning and cutting feed by the MTB specifications becomes effective. (Parameters "#2631 G0olimps" and "#2632 G1olimps")
 - (a) Positioning (G00) - Positioning (G00): Parameter "#2631 G0olimps"
 - (b) Positioning (G00) - Cutting feed (G01)(high-accuracy mode is OFF): Parameter "#2632 G1olimps"

Upper and lower limits for in-position width

- (1) Upper limit for in-position width
 When rapid traverse block overlap is enabled, the in-position check is performed after starting deceleration specified in the speed command ("A" in the figure). Thus, the distance from the servo machine position after starting the command deceleration to the commanded position (hatched area in the figure below) is the upper limit for the actual in-position width.



- (2) Lower limit for in-position width
 The lower limit for the in-position width depends on the MTB specifications (parameters "#2224 SV024" or "#13024 SP024").
 The value of this parameter is applied even if a value less than or equal to this parameter is specified as an in-position width.

Compensation for in-position width based on the path

The conventional deceleration check (in-position check method) applies the same in-position width regardless of the path (corner angle). Therefore, an extra speed change occurs and cycle time is increased even though path direction stays almost the same. The rapid traverse block overlap automatically compensates for the in-position width based on the path (corner angle).

However, the in-position width is not compensated for based on the path if a block without a movement command is inserted between the movement commands to be overlapped.

- (1) If the angle is greater than 90°, the rapid traverse overlap function is temporarily canceled.
- (2) If the angle is less than 90°, the in-position width is compensated for so that it matches the amount of droop at a corner when the corner angle is 90°.



Program example

When the in-position width is specified with address J (G0.5P1 J_)

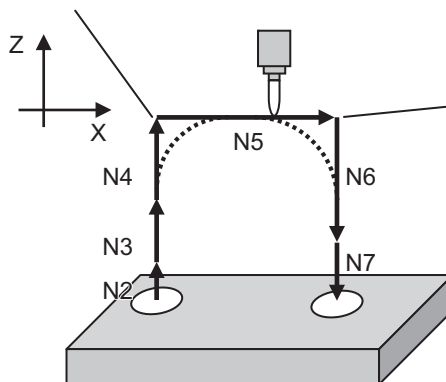
The following are examples of using G00 rapid traverse block overlap in combination with G00 (rapid traverse) and G01 (cutting feed). (When the high-accuracy control mode is OFF)

Parameter setting	X axis	Z axis
#2631(G0olnps)	2mm	1.5mm
#2632(G1olnps)	1 mm	0.5 mm

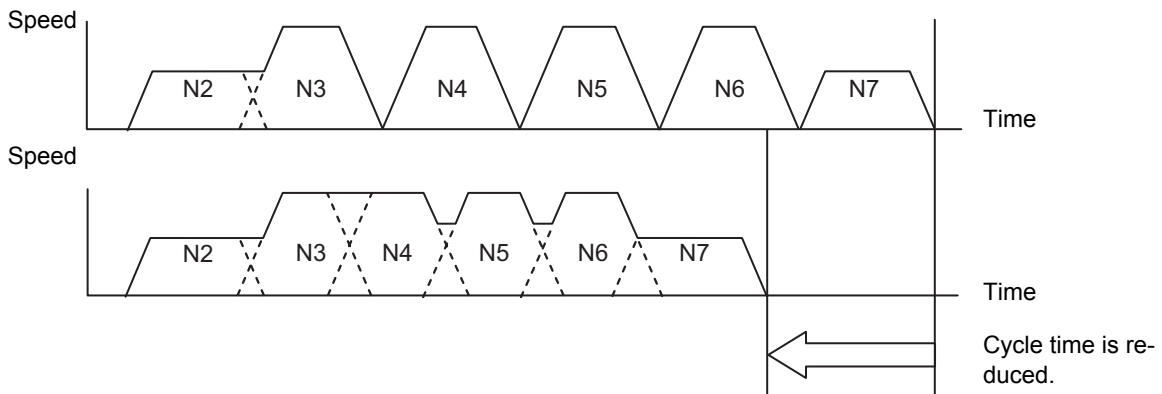
```

N1 G0.5P1 J1.0;
N2 G91 G01 Z25.;
N3 G00 Z25.;
N4 G00 Z50.;
N5 G00 X125.;
N6 G00 Z-75.;
N7 G01 Z-25. F1000.;
    
```

Z axis in-position width is set to 1 mm with the J address command.



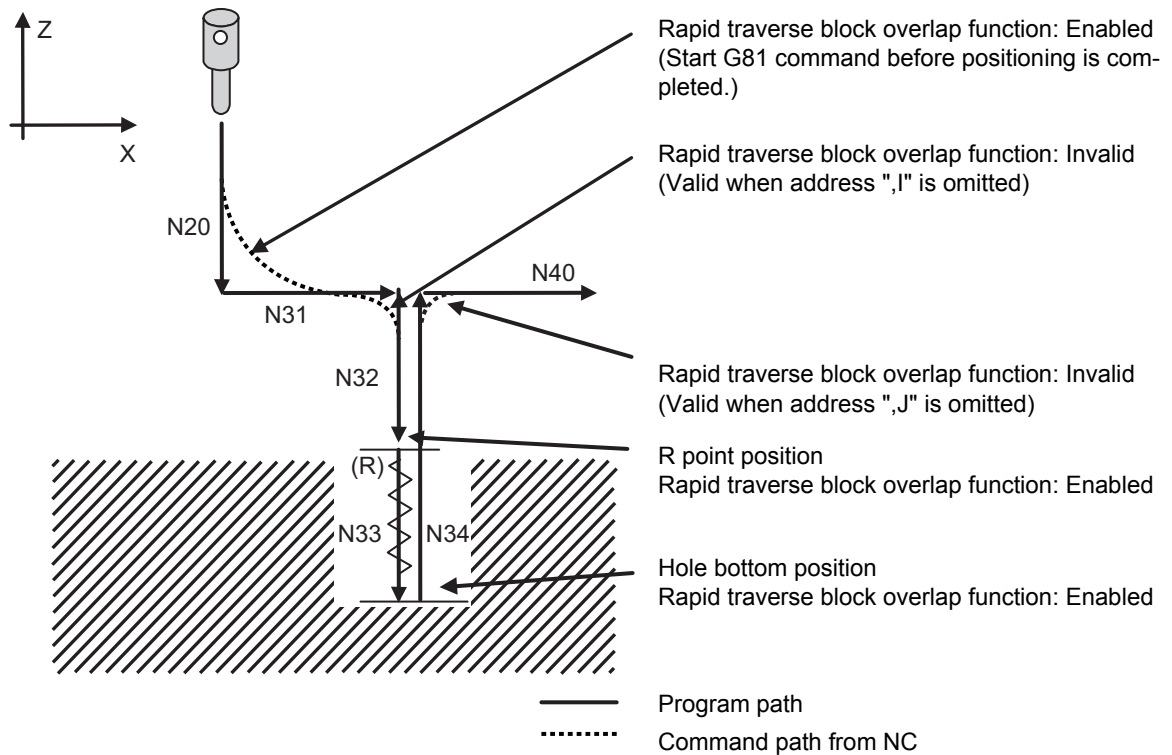
X axis in-position width is set to 1 mm with the J address command.



Example behavior in fixed cycle

When specifying G00 (positioning) -> G81 (drilling)

(Main program)	(G81 program)
N10 G0.5 P1 J0.5;	N31 G00 X50. Y0.;
N20 G91 G98 G64 G00 X50.;	N32 G00 Z-25.;
N30 G81 X50. Y0. Z-25. R-25. F1000. L1. ,I2.0 ,J1.0;	N33 G01 Z-25. F1000.;
N40 G00 X50. ;	N34 G00 Z50.;





Relationship with Other Functions

Programmable in-position

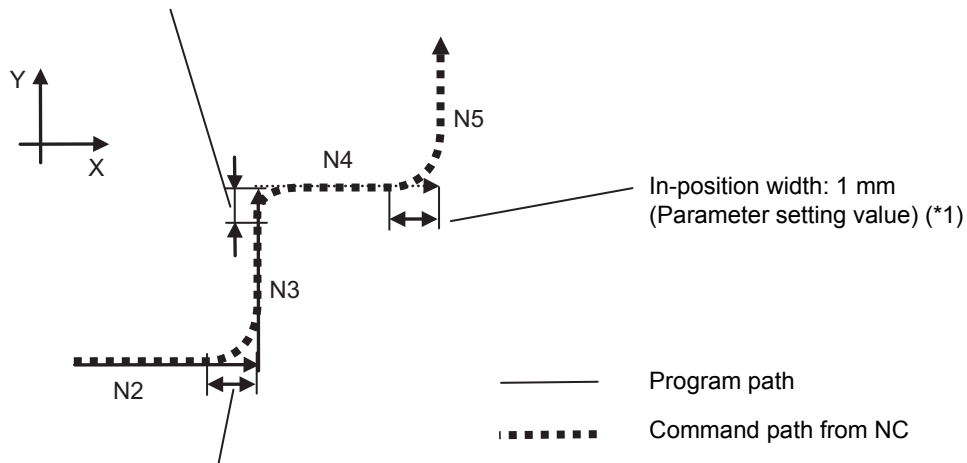
If an ",I" address command is used to specify the in-position width from the program when the rapid traverse block overlap is enabled, the in-position width of programmable in-position is given priority.

Because the programmable in-position is an unmodal command, the in-position width specified with the rapid traverse block overlap enabled is assumed for commands following ",I" address.

This shows an example of when the X and Y axis in-position widths for G00 are set to 1 mm by parameters.

N1 G0.5 P1;	G0.5 command (for G00)
N2 G91 G00 X50.;	Rapid traverse block overlap for G00: Valid
N3 Y50. ,I1.5;	",I" address command is valid
N4 X50.;	Rapid traverse block overlap for G00: Valid
N5 Y50.;	
:	

In-position width: 1.5 mm
(",I" address command is given priority.)



In-position width: 1 mm
(Parameter setting value) (*1)

(*1) The in-position width is the parameter setting value because the programmable in-position is an unmodal command.

Note

- (1) The programmable in-position ("I" command) for G00 pre-interpolation acceleration/deceleration can only be used when the rapid traverse block overlap is enabled.
- (2) When G00 is followed by a block without a movement command, a command of address ",I", if specified for G00, is handled as a command specifying a rapid traverse overlap. Therefore, the overlap takes place only when overlapped movements are executed.

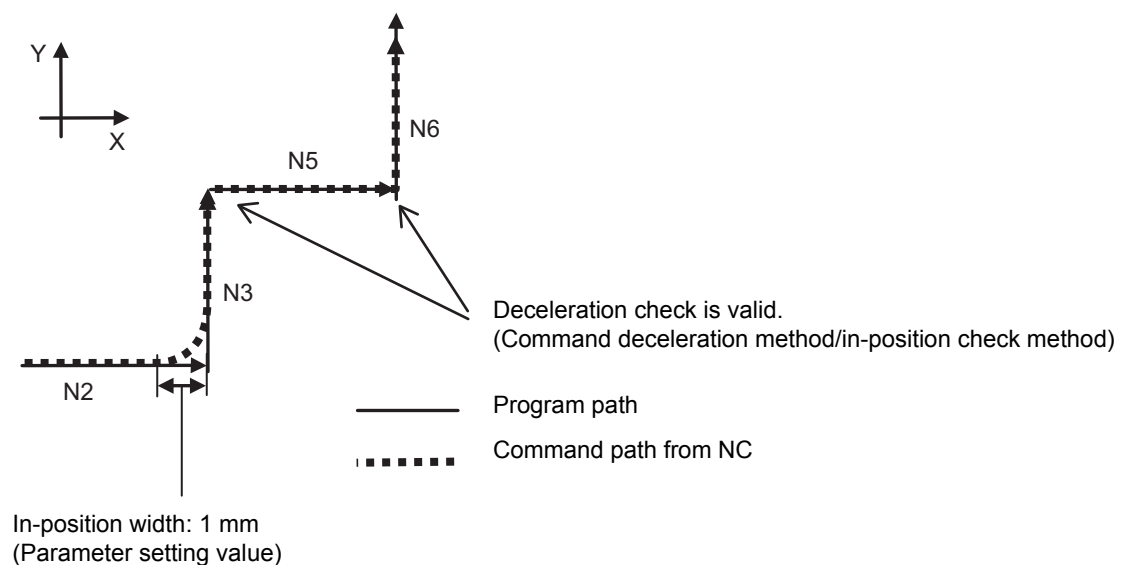
Deceleration Check

When the rapid traverse block overlap is enabled, the conventional deceleration check is disabled for the behavior subject to this function.

When the rapid traverse block overlap is disabled, the conventional deceleration check is enabled.

This shows an example of when the X and Y axis in-position widths for G00 are set to 1 mm by parameters.

N1 G0.5 P1;	G0.5 command (for G00)
N2 G91 G00 X50.;	Rapid traverse block overlap for G00 is valid
N3 Y50.;	Deceleration check is valid.
N4 G0.5 P0;	
N5 X50.;	Deceleration check is valid.
N6 Y50.;	Deceleration check is valid.
:	



Precautions

- (1) When a block without a movement command is inserted between blocks that are subject to the rapid traverse block overlap, blocks are overlapped if the high-accuracy mode is OFF (they are not overlapped if the mode is ON).
If the high-accuracy mode is OFF, a block without movement that is inserted between a G00 command and G28/30 block is not overlapped when the rapid traverse block overlap for G00 is disabled ("#1442 G00I" is "0") and rapid traverse block overlap for G28 is enabled ("#1443 G28O" is "1").
- (2) When a block without a movement command is inserted between blocks that are subject to the rapid traverse block overlap, the in-position width is not compensated for based on the path.
- (3) When the high-accuracy control mode is selected or the parameter #1205 is set to "1" or "2", the next block will not be performed until the speed is reduced below the rapid speed (#2001) if the speed at the completion of in-position check is higher than the rapid speed (parameter #2001) of the next block.
- (4) Even when the overlap process blocks continue, if one or more axes are moved in reversed direction, the overlap function is temporarily canceled.
- (5) (Only for C80 series) The rapid traverse block overlap function is temporarily canceled in the following cases: when the high-accuracy control mode is selected; when the parameter #1205 is set to "1" or "2" and the parameters #1569 and #1570 are set to "0".

7.13.2 Rapid Traverse Block Overlap for G28

**Function and purpose**

This function enables the next block to start (overlap) without waiting for positioning (G00) or reference position return (G28/G30).

For the rapid block overlap function, also refer to "7.13 Rapid Traverse Block Overlap; G0.5 P1".

G00 can be overlapped when the rapid traverse block overlap for G00 is enabled. For details, refer to "7.13.1 Rapid Traverse Block Overlap for G00; G0.5".

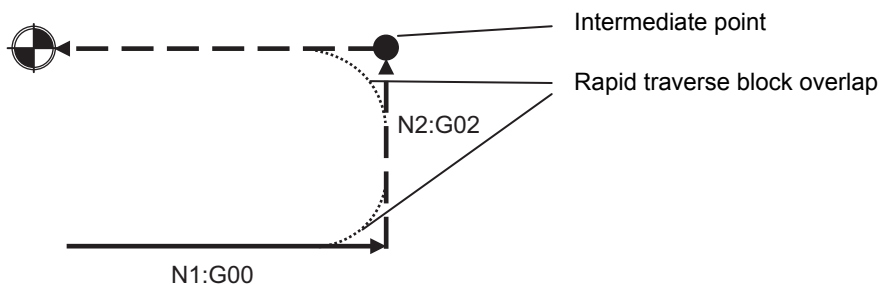
**Detailed description****Enabling conditions**

The rapid traverse block overlap function for G28 becomes effective when all of the following conditions are satisfied.

- (1) The rapid traverse block overlap for G28 is enabled.
(Refer to the MTB specifications. "#1443 G28ol")
- (2) High-speed reference position return is active. (Dog-type is not subject to this.)
- (3) When the rapid traverse block overlap for G00 is enabled, a G00 command is followed by G28 or G30 positioning command.

Note

- ♦For G28/G30, whether or not the appropriate block, if its movement is made via an intermediate point, is overlapped depends on the MTB specifications (parameters "#1205 G0bdcc" and "#1086 G0intp").
- ♦If G28/G30 command is followed by another G28/G30, blocks are not overlapped in rapid traverse. (Blocks are not overlapped.)

**Adjustment of start position of overlap**

The start position of overlap when a rapid traverse block overlap for G28 is executed can be adjusted with the in-position width. The next block is started when the remaining distances of all movement axes in the current movement block are smaller than the in-position width.

The in-position width depends on the MTB specifications (parameter "#2633 G28olinps").

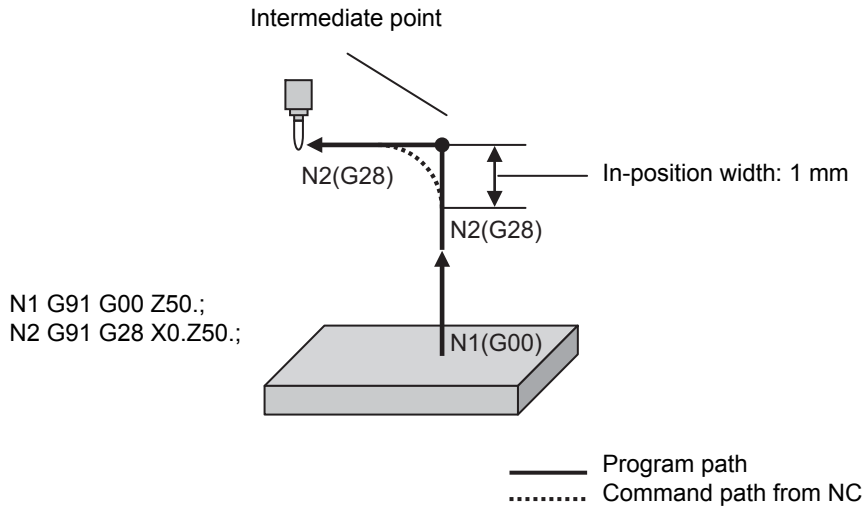


Program example

The following are examples of using rapid traverse block overlap for G28 in combination with G28/G30 (reference position return) and G00 (rapid traverse).

Parameter setting	X axis	Z axis
#2633 G28olnps	0.5 mm	1 mm

Initial position of axes: X axis = -50 mm; Z axis = -100 mm



Relationship with Other Functions

Refer to "7.13 Rapid Traverse Block Overlap; G0.5 P1".



Precautions

Refer to "7.13 Rapid Traverse Block Overlap; G0.5 P1".

7.14 Automatic Corner Override



Function and purpose

With tool radius compensation, this function reduces the load during inside cutting of automatic corner R, or during inside corner cutting, by automatically applying override to the feedrate.

There are two types of automatic corner override: Automatic corner override (G62) and inner arc override.

Automatic corner override (G62) is valid until the tool radius compensation cancel (G40), exact stop check mode (G61), high-accuracy control mode (G61.1), tapping mode (G63), or cutting mode (G64) command is issued.

The inner arc override is valid whenever the machine is in the tool radius compensation mode (G41/G42), regardless of the automatic corner override (G62) mode.

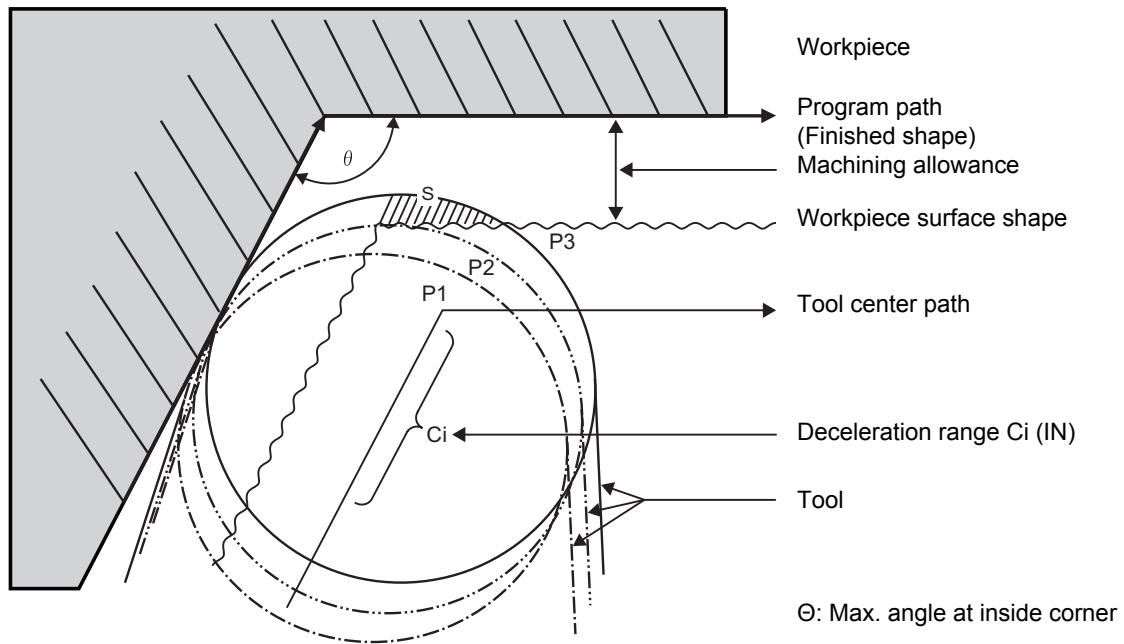


Detailed description

Machining inside corners

When cutting an inside corner, as shown in the figure below, the machining allowance amount increases and a greater load is applied to the tool. To remedy this, override is applied automatically within the corner set range, the feedrate is reduced, the increase in the load is reduced and cutting is performed effectively.

However, this function is valid only when finished shapes are programmed.



[Operation]

(1) If there is no G62 command:

When the tool moves in the order of P1 -> P2 -> P3 in the above figure, the machining allowance at P3 increase by an amount equivalent to the area of shaded section S compared to P2 and so that tool load increases.

(2) If there is G62 command:

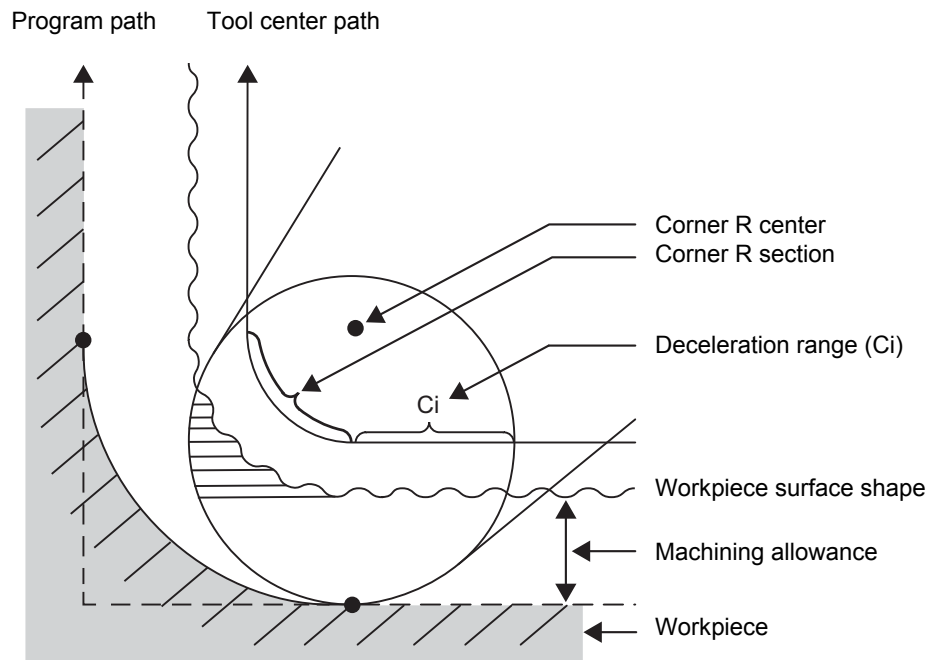
When the inside corner angle θ in the above figure is less than the angle set in the parameter, the override set into the parameter is automatically applied in the deceleration range Ci.

[Parameter setting]

The following parameters are set into the machining parameters. Refer to the Instruction Manual for details on the setting method.

#	Parameters	Setting range
#8007	Override	0 to 100 (%)
#8008	MAX ANGLE	0 to 180 [°]
#8009	DSC.ZONE	0 to 99999.999 [mm] or 0 to 3937.000 [inches]

Automatic corner R



(1) The override set in the parameter is automatically applied at the deceleration range Ci and corner R section for inside offset with automatic corner R. (There is no angle check.)

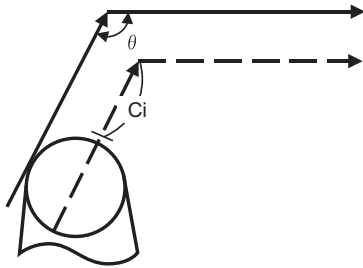


Application example

The lines in the figure denote:

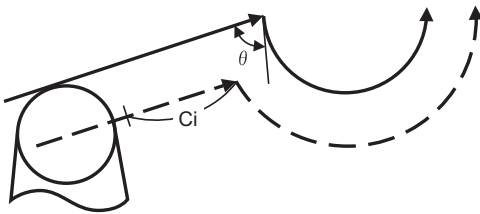
- Programmed path
- - - - - Tool center
- Arc (inside offset)

(1) Linear - linear corner



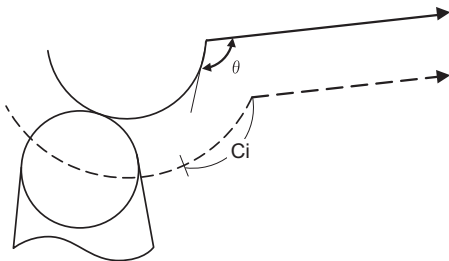
The override set in the parameter (#8007) is applied in the deceleration range Ci.

(2) Linear - arc (outside offset) corner



The override set in the parameter (#8007) is applied in the deceleration range Ci.

(3) Arc (outside offset) - linear corner



<Note>

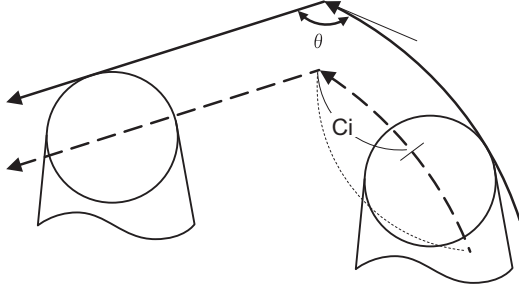
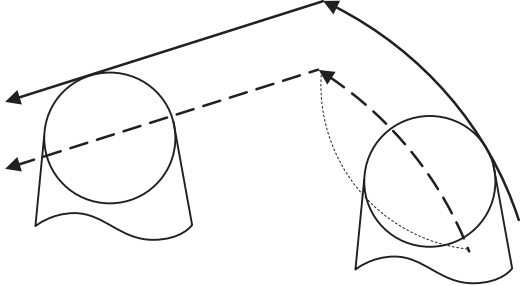
- ♦The deceleration range Ci where the override is applied is the length of the arc with an arc command.

(4) Linear - arc (inside offset) corner

(a) If there is a G62 command:	(b) If there is no G62 command:
<p>For straight lines, the override set in the parameter (#8007) is applied in the deceleration range Ci.</p>	

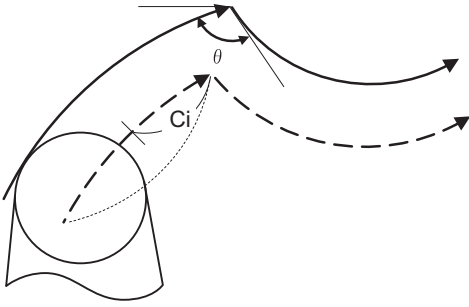
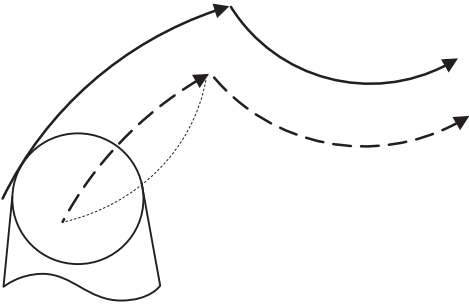
During cutting of arc (inside offset), an inner arc override is applied.

(5) Arc (inside offset) - linear corner

(a) If there is a G62 command:	(b) If there is no G62 command:
	
<p>In addition to the inner arc override, the override set in the parameter (#8007) is applied in the deceleration range Ci. (*1) F (speed command value) x (inner arc override) x (setting for #8007)</p>	

(*1) The deceleration range Ci where the override is applied is the length of the arc with an arc command. During cutting of arc (inside offset), an inner arc override is applied. Automatic corner override will not be applied to straight lines.

(6) Arc (inside offset) - arc (outside offset) corner

(a) If there is a G62 command:	(b) If there is no G62 command:
	
<p>In addition to the inner arc override, the override set in the parameter (#8007) is applied in the deceleration range Ci. (*1) F (speed command value) x (inner arc override) x (setting for #8007)</p>	

(*1) The deceleration range Ci where the override is applied is the length of the arc with an arc command. During cutting of arc (inside offset), an inner arc override is applied. Automatic corner override will not be applied to straight lines.



Relationship with other functions

Function	Operation of automatic corner override (G62)	Operation of inner arc override
F1-digit Feed	Automatic corner override is applied to the F1-digit speed.	Inner arc override is applied to the F1-digit speed.
Cutting feed override	Cutting feed override is applied to automatic corner override.	Cutting feed override is applied to the feedrate to which inner arc override has been applied.
Override cancel	Automatic corner override will not be canceled by override cancel.	Inner arc override is not canceled by override cancel.
External deceleration	External deceleration speed will be applied after automatic corner override is applied to the cutting feedrate.	External deceleration is applied to the feedrate to which the inner arc override has been applied.
Speed clamp	Clamp speed will be applied after automatic corner override is applied to the cutting feedrate.	Clamp speed is applied to the feedrate to which the inner arc override has been applied.
Dry run	Automatic corner override will not be applied.	Inner arc override is not applied.
Synchronous feed	Automatic corner override is applied to the synchronous feedrate.	Inner arc override is applied to the synchronous feedrate.
Thread cutting	Automatic corner override will not be applied.	Inner arc override is not applied.
G31 Skip	Program error occurs with G31 command during tool radius compensation.	Same as on the left.
Machine lock	Automatic corner override is applied even in the machine lock state.	Inner arc override is applied even in the machine lock state.
Positioning (G00)	Automatic corner override is not applied to the positioning command.	Inner arc override is not applied to the positioning command.
Linear interpolation (G01)	Automatic corner override is applied to linear interpolation.	Inner arc override is not applied to linear interpolation.
Circular Interpolation (G02, G03)	Automatic corner override is applied to circular interpolation.	Inner arc override is applied to circular interpolation.
Spiral/conical interpolation (G02.1, G03.1)	Automatic corner override is applied to spiral/conical interpolation.	Inner arc override is applied to spiral/conical interpolation.
Involute Interpolation (G02.2, G03.2)	Automatic corner override is applied to involute interpolation.	Inner arc override is not applied to involute interpolation (*1).
Tool radius compensation cancel (G40)	Automatic corner override will not be applied while the tool radius compensation is being canceled.	Inner arc override is not applied while the tool radius compensation is canceled.
3-dimensional tool radius compensation (Compensation vector designation type) (G41, G42)	Automatic corner override is not applied during 3-dimensional tool radius compensation (compensation vector designation type).	Inner arc override is applied during 3-dimensional tool radius compensation (compensation vector designation type).
3-dimensional tool radius compensation (Tool's vertical-direction compensation type) (G41.2, G42.2)	If G62 is commanded during 3-dimensional tool radius compensation (tool's vertical-direction compensation type), a program error will occur.	The 3-dimensional tool radius compensation (tool's vertical-direction compensation type) cannot be combined with circular interpolation or circular cutting; therefore, a program error will occur.
Nose radius compensation (G41, G42, G46)	Automatic corner override is applied during the nose radius compensation.	Inner arc override is not applied during the nose radius compensation.
Circular cut (G12, G13)	Automatic corner override will not be applied during circular cutting.	The operation is switched by the parameter "#19421 Arc inside min ovr type". For details, refer to "7.14.2 Inner Arc Override".

Function	Operation of automatic corner override (G62)	Operation of inner arc override
Circular cutting cycle (G75)	Automatic corner override will not be applied during the circular cutting cycle.	Inner arc override is not applied during the circular cutting cycle.
High-speed machining mode I/II (G05P1, G05P2)	Automatic corner override is applied during high-speed machining mode.	Inner arc override is applied during high-speed machining mode.
High-speed high-accuracy control I, II (G05.1Q1, G05P10000)	Automatic corner override is applied during high-speed high-accuracy control I/II.	Inner arc override is applied during high-speed high-accuracy control I or II.
High-accuracy control (G08P1)	A program error will occur if the G62 command is issued during high-accuracy control (G08P1).	Inner arc override is not applied during high-accuracy control mode (G08P1).
High-accuracy control (G61.1)	Both high-accuracy control (G61.1) and automatic corner override are functions of G code group 13; therefore, they cannot be combined.	Inner arc override is applied during high-accuracy control mode (G61.1).
High-accuracy spline interpolation 1,2 (G61.2, G61.3)	Both high-accuracy spline interpolation 1/2 and automatic corner override are functions of G code group 13; therefore, they cannot be combined.	High-accuracy spline interpolation 1/2 is enabled when G code group 01 is set to linear interpolation (G01); therefore, it cannot be combined with inner arc override.
SSS Control	Automatic corner override is applied to SSS control.	Inner arc override is applied to SSS control.
Corner rounding	Automatic corner override is applied to corner rounding.	Inner arc override is not applied to the corner R.
Feedrate override OFF (#3004 bit1 = ON)	Automatic corner override will not be applied while the feedrate override is invalid.	Inner arc override is applied while the feedrate override is invalid.

(* 1) Involute interpolation provides involute interpolation override (equivalent to inner arc override) to adjust the speed so that the tool center speed does not exceed the lower limit (#1558 lvOMin) of override when tool radius compensation inside is set.

To designate the minimum override value for inner arc cutting in involute interpolation, set the parameter above. (This parameter setting depends on the MTB specifications.)



Precautions

- (1) Automatic corner override (G62) is valid only in the G01, G02, and G03 modes; it is not effective in the G00 mode. When switching from the G00 mode to the G01 (or G02 or G03) mode at a corner (or vice versa), automatic corner override will not be applied at that corner in the G00 block.
- (2) Even if the automatic corner override mode is entered, the automatic corner override will not be applied until the tool radius compensation mode is entered.
- (3) Automatic corner override will not be applied on a corner where the tool radius compensation is started or canceled.
- (4) Automatic corner override will not be applied on a corner where the tool radius compensation I, K vector command is issued.
- (5) Automatic corner override will not be applied when intersection calculation cannot be executed. Intersection calculation cannot be executed in the following case.
 - ♦When the movement command block does not continue for four or more times.
- (6) The deceleration range with an arc command is the length of the arc.
- (7) The inside corner angle, as set by parameter, is the angle on the programmed path.
- (8) When the parameters are set as shown below, the automatic corner override (G62) or inner arc override is disabled.
 - (a) Conditions that disable the automatic corner override (G62)
 - ♦#8007 (override) is set to 0 or 100
 - ♦#8008 (max. angle) is set to 0 or 180
 - ♦#8009 (DSC.ZONE) is set to 0
 - (b) Condition that disables the inner arc override
 - ♦#19418 (minimum OVR for inner arc) is set to 0 or 100
- (9) The inclined surface machining command is not available during automatic corner override modal. To perform inclined surface machining, command G64 (cutting mode) in advance, then cancel the modal mode.

7.14.1 Automatic Corner Override ; G62



Command format**G62 ; ... Automatic Corner Override**

Automatic corner override (G62) is valid until the nose R compensation cancel (G40), exact stop check mode (G61), high-accuracy control mode (G61.1), tapping mode (G63), or cutting mode (G64) command is issued.

For detailed description, execution example, the relationship with other functions and precautions, refer to "7.15 Automatic Corner Override".

7.14.2 Inner Arc Override

**Detailed description**

When cutting an arc (inside offset), applying the override given by the following expression to the commanded feedrate (F) causes that feedrate to become the F value for which the feedrate of the programmed path is commanded.

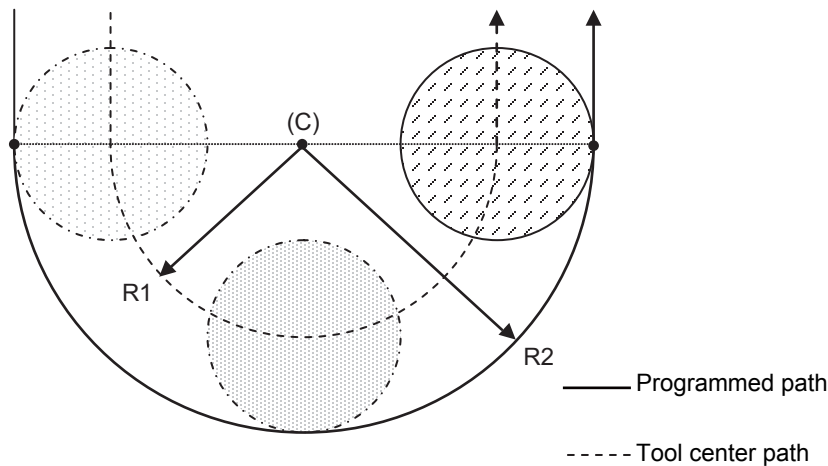
$$F \times \frac{R1}{R2}$$

R1: Radius of tool center path
R2: Radius of program path

The inner arc override is valid whenever the machine is in the tool radius compensation mode (G41/G42), regardless of the automatic corner override (G62) mode.

Inner arc override will not be applied when the machine is in automatic corner R.

Inner arc override can be enabled by the parameter "#19420 Arc inside ovr ON" while the tool radius compensation mode (G41/G42) and automatic corner override mode (G62) are ON.



(C) Arc center

R1: Radius of tool center path

R2: Radius of program path

If the radius (R1) of tool center path is very small compared to the radius (R2) of program path, R1/R2 is nearly equal to 0, causing the tool feed to stop.

To prevent the tool feed from being stopped, set the parameter "#19418 Arc inside min ovr".

When the value of inner arc override is the same or smaller than the parameter setting (*1), the tool feed speed is as follows:

$$F \times \text{Parameter setting value} / 100$$

(*1) $R1/R2 \leq \#19418$

The operation of inner arc override during the circular cutting (G12/G13) can be switched by the parameter "#19421 Arc inside ovr typ".

♦When #19421 is type 1, the inner arc override during the circular cutting is invalid.

♦When #19421 is type 2, the inner arc override during the circular cutting is valid.

Compensation by D address (compensation number) of the circular cutting is not added to the value of inner arc override. The value of inner arc override is calculated using the path after the compensation by D address as the program path and the path with tool radius compensation added to the path as the tool center path.

For detailed description, execution example, the relationship with other functions and precautions, refer to "7.14 Automatic Corner Override".

7.15 Tapping Mode; G63



Function and purpose

The G63 command allows the control mode best suited for tapping to be entered, as indicated below:

- (1) Cutting override is fixed at 100%.
- (2) Deceleration commands at joints between blocks are invalid.
- (3) Feed hold is invalid.
- (4) Single block is invalid.
- (5) In-tapping mode signal is output.

G63 is released by the exact stop check mode (G61), high-accuracy control mode (G61.1), automatic corner override (G62), or cutting mode (G64) command.

The machine is in the cutting mode status when its power is turned ON.



Command format

G63; ... Tapping mode

7.16 Cutting Mode; G64



Function and purpose

The G64 command allows the cutting mode in which smooth cutting surfaces are obtained to be established. Unlike the exact stop check mode (G61), the next block is executed continuously with the machine not decelerating and stopping between cutting feed blocks in this mode.

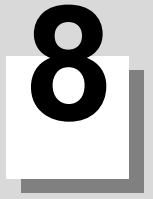
G64 is released by the exact stop check mode (G61), high-accuracy control mode (G61.1), automatic corner override (G62), or tapping mode (G63).

The machine is in the cutting mode status when its power is turned ON.



Command format

G64; ... Cutting mode



Dwell

8.1 Dwell (Time-based Designation); G04



Function and purpose

The machine movement is temporarily stopped by the program command to make the waiting time state. Therefore, the start of the next block can be delayed. The waiting time state can be canceled by inputting the skip signal.



Command format

Dwell (Time-based designation)

G94 G04 X__ /P__ ;

X/P	Dwell time
-----	------------

The input command unit for the dwell time depends on the parameter.

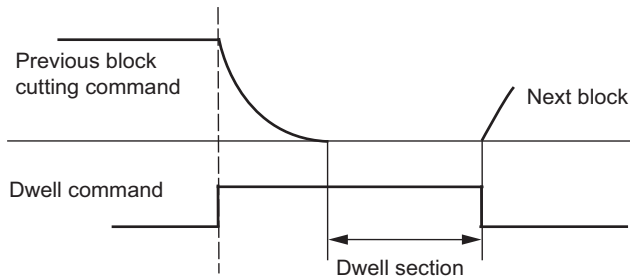


Detailed description

- (1) When designating the dwell time with X, the decimal point command is valid.
- (2) When designating the dwell time with P, the availability of the decimal point command can be selected with the parameter (#8112). When the decimal point command is set to be invalid, the command value below the decimal point with P is ignored.
- (3) When the decimal point command is valid or invalid, the dwell time command range is as follows.

Command range when the decimal point command is valid	Command range when the decimal point command is invalid
0 to 99999.999(s)	0 to 99999999 (ms)

- (4) The dwell time setting unit applied when there is no decimal point can be made 1s by setting 1 in the parameter "#1078 Decpt2". This is effective only for X and P for which the decimal command is valid.
- (5) When a cutting command is in the previous block, the dwell command starts calculating the dwell time after the machine has decelerated and stopped. When it is commanded in the same block as an M, S, T or B command, the calculation starts simultaneously.
- (6) The dwell is valid during the interlock.
- (7) The dwell is valid even for the machine lock.
- (8) Dwell can be canceled depending on the MTB specifications (parameter "#1173 dwlskp"). If the set skip signal is input during the dwell time, the remaining time is discarded, and the following block will be executed.





Program example

Command	Dwell time [s]			
	#1078 Decpt2 = 0		#1078 Decpt2 = 1	
	G04P DECIMAL PNT-N	G04P DECIMAL PNT-P	G04P DECIMAL PNT-N	G04P DECIMAL PNT-P
G04 X500 ;	0.5		500	
G04 X5000 ;	5		5000	
G04 X5 ;	5		5	
G04 X#100 ;	1000		1000	
G04 P5000 ;	5		5	5000
G04 P12.345 ;	0.012	12.345	0.012	12.345
G04 P#100 ;	1	1000	1	1000

Note

- (1) The above examples are the results under the following conditions.
 - ◆ Input setting unit 0.001mm or 0.0001inch
 - ◆ #100 = 1000 ;
- (2) "G04P DECIMAL PNT-P" is a control parameter (#8112).
- (3) If the input setting unit is 0.0001inch, the X before G04 will be multiplied by 10. For example for "X5. G04;", the dwell time will be 50 seconds.



Precautions and restrictions

- (1) When using this function, command X after G04 in order to make sure that the dwell is based on X.

Miscellaneous Functions

9.1 Miscellaneous Functions (M8-digits)



Function and purpose

The miscellaneous functions are also known as M functions, and they command auxiliary functions, such as spindle forward and reverse rotation, operation stop and coolant ON/OFF.



Detailed description

These functions are designated by an 8-digit number (0 to 99999999) following the address M, and up to 4 groups can be commanded in a single block. The number of M commands that can be issued within the same block depends on the MTB specifications (parameter "#12005 Mfig").

(Example) G00 Xx Mm1 Mm2 Mm3 Mm4;

When five or more commands are issued in a block, only the last four will be valid.

Whether to BCD output or binary output the 2nd miscellaneous function can be selected by a parameter.

The eight commands of M00, M01, M02, M30, M96, M97, M98 and M99 are used as auxiliary commands for specific objectives and so they cannot be used as general auxiliary commands. Therefore, 92 miscellaneous functions are available.

Reference should be made to the instructions issued by the MTB for the actual correspondence between the functions and numerical values.

When the M00, M01, M02, and M30 functions are used, the next block is not read into the pre-read buffer due to pre-read inhibiting.

If the M function is designated in the same block as a movement command, the commands may be executed in either of the following two orders. The machine specifications determine which sequence applies.

- (1) The M function is executed after the movement command.
- (2) The M function is executed at the same time as the movement command.

Processing and completion sequences are required in each case for all M commands except M96, M97, M98 and M99.

Program stop: M00

When the NC has read this function, it stops reading the next block. As far as the NC system's functions are concerned, it only stops reading the next block. Whether machine functions such as the spindle rotation and coolant supply are stopped or not differs according to the machine in question.

Re-start is enabled by pressing the automatic start button on the machine operation board.

Whether resetting can be initiated by M00 depends on the machine specifications.

Optional stop: M01

If the M01 command is read when the optional stop switch on the machine operation board is ON, it will stop reading the next block and perform the same operation as the M00.

If the optional stop switch is OFF, the M01 command is ignored.

(Example)

N10 G00 X1000 ;	The state and operation of optional stop switch
N11 M01 ;	Stops at N11 when the switch is ON
N12 G01 X2000 Z3000 F600 ;	Next command (N12) is executed without stopping at N11 when the switch is OFF
:	

Program end: M02 or M30

This command is normally used in the final block for completing the machining, and so it is primarily used for cueing up the machining program. Whether the program is actually cued up or not depends on the machine specifications. Depending on the machine specifications, the system is reset by the M02 or M30 command upon completion of cueing up the program and any other commands issued in the same block.

(Although the contents of the command position display counter are not cleared by this reset action, the modal commands and compensation amounts are canceled.)

The next operation stops when the cueing up operation is completed (the in-automatic operation lamp goes off).

To restart the unit, the automatic start button must be pressed or similar steps must be taken. When the program is restarted after M02 and M30 are completed, if the first movement command is designated only with a coordinate word, the interpolation mode will function when the program ends. It is recommended that a G function always be designated for the movement command designated first.

Note

- (1) Independent signals are also output respectively for the M00, M01, M02 and M30 commands and these outputs are each reset by pressing the reset key.
- (2) M02 or M30 can be assigned by manual data input (MDI).
At this time, commands can be issued simultaneously with other commands.

Macro interruption; M96, M97

M96 and M97 are M codes for user macro interrupt control.

The M code for user macro interrupt control is processed internally, and is not output externally.

To use M96 and M97 as miscellaneous functions, change to another M code with the parameter (#1109 subs_M, #1110 M96_M and #1111 M97_M).

Subprogram call/completion: M98, M99

These commands are used as the return instructions from branch destination subprograms and branches to subprograms.

M98 and M99 are processed internally and M code signals and strobe signals are not output.

Internal processing with M00/M01/M02/M30 commands

Internal processing suspends pre-reading when the M00, M01, M02 or M30 command has been read. Other tape rewinding operations and the initialization of modals by resetting differ according the machine specifications.

9.2 Second Miscellaneous Functions (A8-digits, B8-digits or C8-digits)



Function and purpose

These serve to assign the indexing table positioning and etc. In this controller, they are assigned by an 8-digit number from 0 to 99999999 following address A, B or C. The MTB determines which codes correspond to which positions.



Detailed description

The address that is used for the 2nd miscellaneous function (A, B, or C) depends on the MTB specifications (parameter "#1170 M2name"). (Except the address that is used for the axis name.) The 2nd miscellaneous function can be issued up to 4 sets in a block.

The number of M commands to be issued within a same block is determined by the parameter setting (#12011 Bfig). Whether to BCD output or binary output the 2nd miscellaneous function can be selected by a parameter.

If the A, B or C function is designated in the same block as a movement command, the commands may be executed in either of the following two orders. The machine specifications determine which sequence applies.

- (1) The A, B or C function is executed after the movement command.
- (2) The A, B or C function is executed simultaneously with the movement command.

Processing and completion sequences are required for all 2nd miscellaneous functions.

The table below gives address combinations. It is not possible to use an address that is the same for the axis name of an additional axis and 2nd miscellaneous function.

		Additional axis name		
		A	B	C
2nd miscellaneous function	A	-	○	○
	B	○	-	○
	C	○	○	-



Precautions

- (1) When A has been assigned as the 2nd miscellaneous function address, the following commands cannot be used.
 - Linear angle commands (,A can be used.)
 - Geometric command
- (2) When the sub part system control I function is valid, the independent command of address B is ignored.

9.3 Index Table Indexing



Function and purpose

Index table indexing can be carried out by setting the index axis.

The indexing command only requires specifying the indexing angle to the axis set for indexing. It is not necessary to command special M codes for table clamping and unclamping, thus simplifying the program.

There are the following two types for this function. Which type is valid and which axis is set as the indexing axis depend on the MTB specifications (parameters "#1282 ext18/bit3" and "#2076 index_x").

- ♦Type A: When the unclamp command signal is turned OFF, the clamp operation is performed.
- ♦Type B: When the clamp command signal is turned ON, the clamp operation is performed.

The PLC operation and each signal input/output depend on the MTB specifications.



Command format

G00 B90 ;

B	Index table indexing axis (designated with parameter "#2076 index_x")
---	---

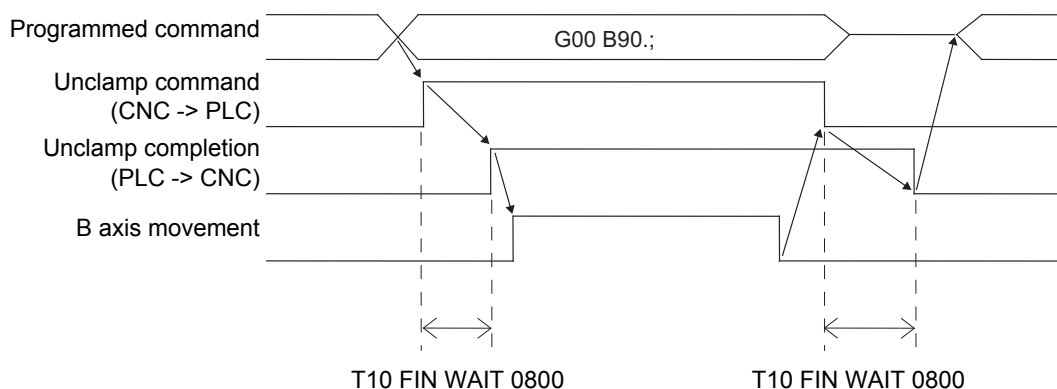


Detailed description

Type A operations

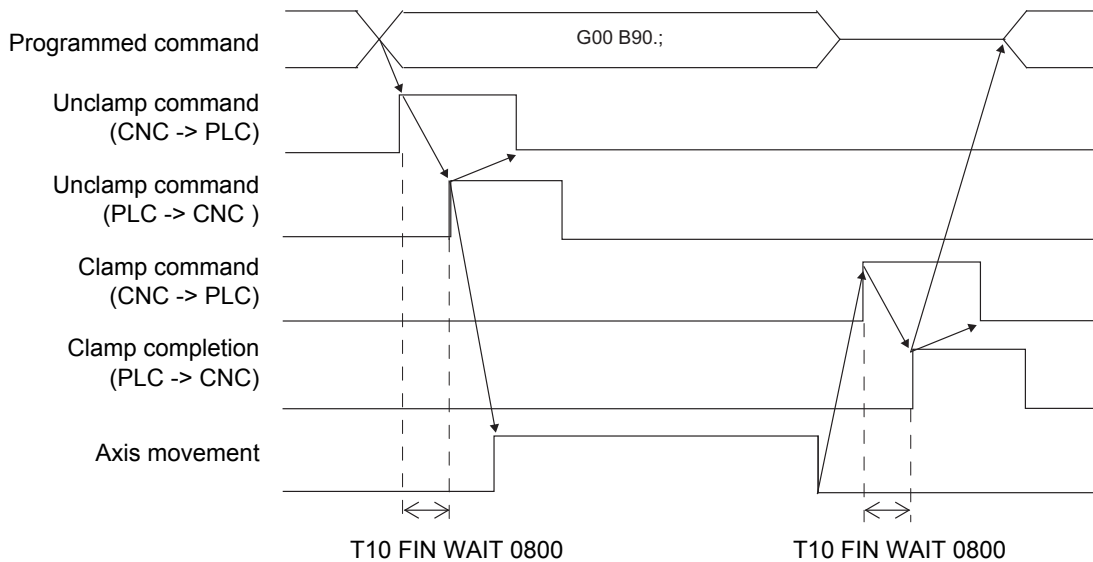
- (1) The movement command (either absolute or incremental) for the selected axis is executed with the program command.
- (2) The unclamp command signal is now output prior to the axis movement.
- (3) When the axes are unclamped, the unclamp completion signal is turned ON by the PLC.
(Turn the signal ON after performing required process such as servo ON or the unclamp process.)
- (4) After checking the unclamp completion signal, the designated axis starts moving.
- (5) Upon completion of the movement, the unclamp command signal is turned OFF.
- (6) Clamp the axes and turn the unclamp completion signal OFF with the PLC.
(Turn the signal OFF after performing required process such as in-position check, servo OFF or the clamp process.)
- (7) After checking that the unclamp completion signal is OFF, processing of the next block is initiated.

[Operation time chart]



Type B operations

- (1) The movement command (either absolute or incremental) for the selected axis is executed with the program command.
- (2) The unclamp command signal is now output prior to the axis movement.
- (3) When the axes are unclamped, the unclamp completion signal is turned ON by the PLC.
(Turn the signal ON after performing required process such as servo ON or the unclamp process.)
- (4) After checking the unclamp completion signal, turn the unclamp command signal OFF and the designated axis starts moving.
- (5) Turn the unclamp completion signal OFF with the PLC.
- (6) Upon completion of the movement, the clamp command signal is turned ON.
- (7) Clamp the axes and turn the clamp completion signal ON with the PLC.
(Turn the signal OFF after performing required process such as in-position check, servo OFF or the clamp process.)
- (8) After checking that the clamp completion signal is ON, turn the clamp command signal OFF and processing of the next block is initiated.
- (9) Turn the clamp completion signal OFF with the PLC.



Cutting feed prohibit of index table indexing axes

When the cutting feed of index table indexing axes is prohibited, the cutting feed can be prohibited by issuing a program error (P20) if all of the following conditions are satisfied during automatic operation.

- The indexing axis movement command is issued. (*1)
- The modal of G code group 1 is other than "G00" or "G60".

(*1) If a cutting feed command without axis movement (such as "G01 B0;" during incremental command) is issued, the program error does not occur. Also, the unclamp command is not output.

The cutting feed prohibit function is valid for both type A and type B, and the parameter settings depend on the MTB specifications (Parameter "#2580 index_Gcmd").



Relationship with other functions

Index table indexing and other functions

Function	Details
Machine coordinate system selection (G53)	Possible.
Unidirectional positioning	(*1)
Servo ON/OFF signal control	Perform the required process on the PLC.

(*1) The unidirectional positioning function can be used in the machining center system only.

- If parameter "#8209 G60 SHIFT" is not in the indexing unit, a program error (P20) will occur.
- When an axis command that cannot be divided in the command unit is issued, a program error (P20) will occur.
- In a single block operation, the block stop is carried out for the indexing axis at the position specified by parameter #8209, and the clamp or unclamp operation is carried out.

Single block

The clamp and unclamp operations are not executed when the movement commands of the index table indexing axis are successively issued.

Note that the clamp and unclamp operations are executed even when the movement commands are continued during single block operation.

A combination of G codes that executes clamp or unclamp operation with continuous blocks is listed below.

(The unclamp operation is executed before the axis movement of previous block is started, and the clamp operation is executed after the axis movement of the next block.)

(1) Clamp and unclamp operations between continuous blocks

Command	Continuous block	Condition and result
Reference position check (G27)	G00 -> G27	(*1)
	G27 -> G00	(*2)
Start position return (G29)	G00 -> G29	(*1)
	G29 -> G00	(*1)
Tool change position return 1 to 6 Lathe system: G30.1 to G30.5 Machining center system: G30.1 to G30.6	G00 -> G30.1	(*1)
Normal line control cancel (G40.1) (Machining center system only)	G40.1 -> G00	(*1)
Basic machine coordinate system selection (G53)	G00 -> G53	(*3)
	G53 -> G00	
Unidirectional positioning (G60) (Machining center system only)	G00 -> G60	(*1)
	G60 -> G00	(*2)
	G60 -> G60	
Program stop (M00)	M00	(*1)
Optional stop (M01)	M01	(*1)

(*1) The clamp and unclamp operations are executed between blocks.

(*2) The clamp and unclamp operations are NOT executed between blocks.

(*3) The clamp and unclamp operations are executed during workpiece installation error compensation (G54.4) or during inclined surface machining command (G68.2).

9 Miscellaneous Functions

(2) Clamp and unclamp operations between continuous blocks (Reference position return)

The operation during reference position return depends on the ignoring of intermediate points during return, and it depends on the MTB specifications (Parameter "#1091 Mpoint").

Command	Continuous block	Condition and result	
		#1091 = 1	#1091 = 0
1st reference position return (G28)	G00 -> G28	(*1)	(*2)
	G28 -> G00	(*3)	(*4)
2nd to 4th reference position return (G30)	G00 -> G30	(*1)	(*2)
	G30 -> G00	(*3)	(*4)

(*1) Performs the clamp operation at the end of G00 movement, and performs the unclamp operation before reference position return.

(*2) The clamp/unclamp operation will not be performed until the reference position return is completed.

(*3) Performs the clamp operation after the reference position return, and performs the unclamp operation before G00 movement.

(*4) The clamp/unclamp operation will not be performed when movement to the intermediate point is completed. The clamp operation will be performed after the reference position return, and the unclamp operation will be performed before G00 movement.

Macro interruption

Clamp/unclamp operations during macro interrupt are as follows.

Parameter		Settings
#1112 S_TRG	0	Edge trigger mode
#1113 INT_2	0	Immediately start the interrupt program without waiting for the completion of currently executing block.
#8101 MACRO SINGLE	1	

(1) When the macro interrupt program, executed during indexing axis movement, contains a movement command.

The commands in the interrupted block are lost, and the interrupt program is executed. After completion of interrupt program, when executing from the block next to the interrupt block, the clamp/unclamp operation is executed even if the interrupt program and main program specify the continuous movement.

(Example)

[Main program]

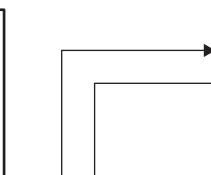
```

O620(MINT MAIN)
N100 G90G94;
N110 G28B0.X0.;
N120 M96P621;
N130 G01X10.B10.F150.;
N132 G01B15.;
N140 G04X3.;
N150 M97;
M02;
    
```

[Interrupt program]

```

O621(MINT SUB)
N100 G01B1.F500.;
M99;
    
```



(a) Performs unclamp operation at the beginning of main program N130 block.

(b) Executes macro interrupt during main program N130 execution.

(c) Performs clamp operation after end of interrupt program O621 N100 block B1.

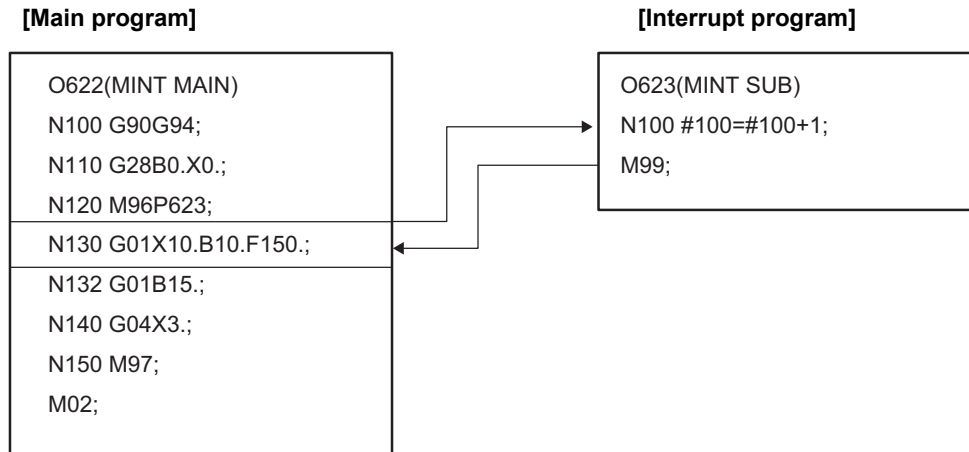
(d) Performs unclamp operation at the beginning of main program N132 block, and performs clamp operation after axis movement.

- (2) When the macro interrupt program, executed during indexing axis movement, does not contain a movement command

When executing the remaining blocks after completion of interrupt program, perform the unclamp operation at the restart of main program.

Also, perform the clamp/unclamp operation even when the next block continues.

(Example)



- (a) Performs unclamp operation at the beginning of main program N130 block.
- (b) Executes macro interrupt during main program N130 execution.
- (c) Interrupt program O623 execution is completed.
- (d) Performs unclamp operation at the restart of main program N130 block, and performs clamp operation after completion of axis movement.
- (e) Performs unclamp operation at the beginning of main program N132 block, and performs clamp operation after completion of axis movement.



Precautions

- (1) Several axes can be set as index table indexing axes.
- (2) The movement speed of index table indexing axes follows the feedrate of the modal (G00/G01) at that time.
- (3) The unclamp command for the indexing axes is also issued when the index table indexing axes are commanded in the same block as other axes. Thus, the movement of other axes commanded in the same block is not carried out until the unclamp operation completes.
Note that the movement of other axes commanded in the same block is carried out during a non-interpolation commands.
- (4) Index table indexing axes are used as normal rotation axes, but this function performs an unclamp operation even for linear axes.
- (5) If some error that makes unclamp command OFF occurs during indexing axis movement in automatic operation, the unclamp state will remain, and the indexing axis will execute a deceleration stop.
Other axes commanded in the same block will also execute a deceleration stop, except during non-interpolation commands.
- (6) If the axis movement is interrupted by an interlock, etc., during indexing axis movement, the unclamp state will remain.
- (7) The clamp and unclamp operations are not executed when the movement commands of the index table indexing axis are successively issued.
Note that the clamp and unclamp operations are executed even when the movement commands are continued during single block operation. Refer to "Single block" of the "Relationship with other functions".
- (8) Make sure that the command position is at a position where clamping is possible.
- (9) Set the unidirectional positioning (G60) parameter "#8209 G60 SHIFT" in indexing increment. A program error (P20) will occur if it is not set in indexing increment.
In a single block operation, the block stop is carried out at the "G60 SHIFT" position, and the clamp or unclamp operation is carried out.

Spindle Functions

10.1 Spindle Functions



Function and purpose

(1) Spindle function (S 8-digit)

This function allows you to designate an S command with an 8-digit number (0 to 99999999) following address S and include one pair of S commands in a single block.

The output signal is a 32-bit binary data with sign and start signal.

Processing and completion sequences are required for all S commands.

(2) Spindle function (S 6-digit analog)

When the S 6-digit function is added, S commands can be designated in the range from S0 to S999999.

This function outputs the appropriate gear signal or the voltage and start signals matching the spindle rotation speed (r/min) to be commanded using the 6-digit numerical command following the S code.

Processing and completion sequences are required for all S commands.

If the gear level is switched manually while an S command is not running, this function obtains the appropriate voltage from the rotation speed designated for the gear level and the previously commanded rotation speed, and outputs the result.

10.2 Constant Surface Speed Control; G96, G97



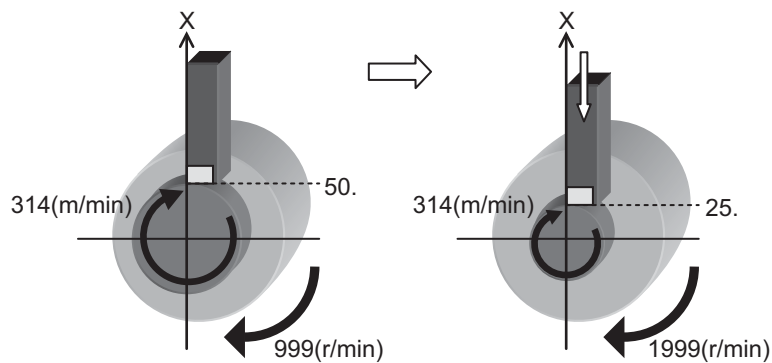
Function and purpose

This function adjusts the spindle rotation speed (constant surface speed control) in accordance with the movement of the tool nose point so that the cutting point always remains at the constant speed (constant surface speed). Using this function for processes such as a cutting-off process is effective in terms of machining time, tool life, etc. Note that when the tool nose point is moving to the workpiece zero point, the rotation may be at the maximum rotation speed defined in the machine specifications; this is dangerous. Be sure to specify the maximum clamp rotation speed with the spindle clamp speed setting command (G92/G50).

Constant surface speed control at constant surface speed command G96 S314 m/min

Workpiece diameter: 50 mm
(Radius value)

Workpiece diameter: 25mm
(Radius value)



To keep the surface speed constant, this function obtains and automatically adjusts the spindle rotation speed in accordance with the movement of the tool nose point.

In the example above, to keep the surface speed (314 (m/min)) constant, the rotation speed is changed from 999 (r/min) to 1999 (r/min) with changes of the workpiece radius (50mm → 25mm).

$$\text{Spindle rotation speed (r/min)} = \frac{\text{Surface speed (m/min)}}{\text{G96 command value}} \div \frac{\text{Workpiece surface (m/r)}}{\text{Automatically calculated from the workpiece zero point and tool nose position}}$$

Note

- (1) When the surface speed constant control is commanded under Inch system, the error of the spindle rotation speed specification depends on the MTB specifications (parameter "#1255 set27/bit0").



Command format

Constant surface speed ON

G96 S__ P__ ;	
S	Surface speed (-99999999 to 99999999 (m/min), -99999999 to 99999999 (feet/min))
P	Constant surface speed control axis 0 to n (n: Number of axes that can be controlled in the part system with G96 commanded)

Note

- (1) The S command is handled as the absolute value (the sign is ignored).
- (2) If the value of the S command exceeds the allowable range, a program error will occur (P35).
- (3) If the value of the P command exceeds the allowable range, a program error will occur (P133).

Constant surface speed cancel

G97 S__ ;	
S	Spindle rotation speed (-99999999 to 99999999 (r/min))

Note

- (1) The S command is handled as the absolute value (the sign is ignored).



Detailed description

- (1) When the P0 or P command is not specified, the operation depends on the MTB specifications (parameter "#1181 G96_ax").
However, if this parameter is set to "0", the first axis is used as the surface speed axis regardless of whether address P is specified or not.
0: Fixed at 1st axis (P command invalid)
1: 1st axis
2: 2nd axis
:
- (2) To change the constant surface speed control axis in the constant surface speed control mode, specify the command in the G96 P_ format. (However, when the parameter above is set to "0", no change can be made.)
If the S command is issued simultaneously, the surface speed can also be changed.

<Example 1>

Machining program	Control axis sequence in constant surface speed command part system			Details of Operation
	1st axis	2nd axis	3rd axis	
: G96 S200 P1; : :	X1 ↓ ↓ ↓	Z1 ↓ ↓ ↓	C1 ↓ ↓ ↓	The X1 axis is used as the constant surface speed control axis. (Controls the spindle rotation so that the surface speed is set to 200 (m/min) for the X1 axis.)
G96 P2; :	↓ ↓	↓ ↓	↓ ↓	The Z1 axis is used as the constant surface speed control axis.

<Example 2>

Machining program	Control axis sequence in constant surface speed command part system			Details of Operation
	1st axis	2nd axis	3rd axis	
: G96 S200 P1; : : :	Z1 ↓ ↓ ↓ ↓	C1 ↓ ↓ ↓ ↓	- - - -	The Z1 axis is used as the constant surface speed control axis. (Controls the spindle rotation so that the surface speed is set to 200 (m/min) for the Z1 axis.)

- (3) The spindle to be controlled is determined in the MTB specifications (parameter "#1300 ext36/bit0").
For multiple-spindle control II (*1), the spindle is determined by the spindle selection signal from the PLC.
(*1) Multiple-spindle control with the PLC signal used. Whether the specification is provided and the details depend on models and MTB specifications.
- (4) Specify the spindle surface speed with the S command when constant surface speed control ON is commanded.
In constant surface speed control mode, the surface speed can only be changed with the S command.
- (5) The spindle clamp speed setting (G92 S__ Q__) is to be commanded when the spindle speed needs to be limited depending on the workpiece to be machined, the chuck to be mounted on the spindle and the tool specifications, etc.
Whether the spindle clamp speed setting is made valid only in the constant surface speed control mode or also made valid for normal spindle rotation commands depends on the MTB specifications (parameter "#1227 aux11/bit5").
Once the maximum clamp rotation speed and the minimum clamp rotation speed are set using the spindle clamp speed setting (G92 S__ Q__), the maximum speed clamp will not be canceled even if the command "G92 S0" is issued.
Whether the commanded spindle clamp speed setting is kept when NC is reset during constant surface speed control depends on the MTB specifications (parameter "#1210 RstGmd/bit19").

- (6) Whether the surface speed is always calculated or at the end of a block when the rapid traverse command is issued depends on the MTB specifications (parameter "#1087 G96_G0").
- (7) The constant surface speed cancel command (G97) cancels constant surface speed control in the part system that has executed the constant surface speed ON command (G96).
The constant surface speed control cannot be canceled from another part system.
The spindle rotation speed is maintained at the speed specified when the constant surface speed cancel command (G97) has been executed.
- (8) If NC is reset during constant surface speed control, the spindle rotation speed is changed to "0" (r/min) after reset.



Relationship with Other Functions

Checking the maximum clamp rotation speed

When the constant surface speed control is commanded, check whether the spindle speed clamp is valid.

If the constant surface speed control axis is near the zero point, it causes the spindle to rotate at the maximum rotation speed.

Check the spindle speed clamp command to prevent the spindle from rotating at high speed.

- (1) In multiple-spindle control II, if the speed clamp command is not valid for the selected spindle, it causes an operation error (M01 1043). When such an error occurs, reset to finish the program, and issue the spindle speed clamp command after selecting a spindle.

When the operation error above occurs, execute the commands in the same block.

- (a) G96 S100 M03 com- When the spindle forward rotation signal is input from the user PLC by the M03
mand: command, the spindle runs forward.
(The spindle speed is set to the previously commanded rotation speed.)

- (b) G96 S100 X30. com- If an error occurs, axis movement is performed until the program is reset.
mand:

- (2) When spindle speed clamp command check is valid, the spindle speed clamp command value is set to "0" if the G92/G50 S0 command is issued.
- (3) In multiple-spindle control II, spindle speed clamp check is conducted for the spindle selected in the G96S command. Specify the spindle speed clamp command for all the currently selected spindles.
- (4) Whether to conduct spindle speed clamp command check depends on the MTB specifications (parameters "#1146 Sclamp" and "#1284 ext20/bit0".)
If parameter "#1146 Sclamp" is set to "0", the spindle speed clamp command cannot be executed when constant surface speed control is turned off; therefore, the spindle speed clamp command cannot be issued before constant surface speed control.

Parameter "#1284 ext20/bit0" has the following setting:

- 0: Checks the spindle speed clamp.
- 1: Does not check the spindle speed clamp.

- (5) The spindle speed clamp may be performed only in the constant surface speed mode depending on the MTB specifications (parameter "#1227 aux11/bit5"). If the program is then reset, the clamp may be rendered ineffective.

For information on whether the setting is configured to keep the clamp status, refer to the MTB specifications. (Parameter "#1210 RstGmd/bit10, bit19")

- BIT10: Group 17, constant surface speed control command modal
- BIT19: Spindle rotation clamp speed

- (6) When operating the system in the initial constant surface speed mode or with the constant surface speed modal by holding the constant surface speed control command modal, the constant surface speed control mode is set by the S command (surface speed). When spindle speed clamp command check is valid, issue the spindle speed clamp command before the S command.

Arbitrary axis exchange control

- (1) If constant surface speed control axes are rearranged by the arbitrary axis exchange command, the spindle rotation speed is maintained at the value specified before rearrangement.
- (2) If a new surface speed is specified by the S command while the spindle rotation speed is maintained, it becomes valid when the rearranged constant surface speed axes are returned to the original status.
- (3) If the constant surface speed command is re-executed when constant surface speed axes are rearranged and the spindle rotation speed is maintained at the constant rotation speed, the kept spindle rotation speed is canceled, and the reissued constant surface speed control command is executed.
- (4) If constant surface speed axes are returned to the original status by rearrangement while constant surface speed control is temporarily canceled, the spindle rotation speed will be maintained. After this, the surface speed becomes constant when it is specified with the S command.
- (5) If the surface speed is specified by the S command with the rearrangement of the constant surface speed axes while the constant surface speed control is in the temporary cancel state, the spindle rotation speed kept at temporary cancellation is applied, and the surface speed becomes constant when the constant surface speed axes are returned to the original arrangement.

Other functions

Function name	Operation
Spindle Clamp Speed Setting (G92/G50)	The spindle clamp speed setting is valid in the constant surface speed control mode. Whether the commanded spindle clamp speed setting is kept when NC is reset during constant surface speed control depends on the MTB specifications. (parameter "#1210 RstGmd BIT19")
Cylindrical Interpolation (G07.1)	The constant surface speed control cannot be commanded during the cylindrical interpolation mode. Program error (P481) will occur. The cylindrical interpolation cannot be commanded during the constant surface speed control mode. Program error (P485) will occur.
Thread Cutting (Designation of lead or number of ridges) (G32)	When the constant surface speed command is issued in the same part system during execution of the thread cutting or thread cutting cycle command or when the thread cutting or thread cutting cycle command is issued in the same part system in the constant surface speed control mode, the spindle rotation speed for constant surface speed control remains unchanged. (The constant surface speed control is not performed.) This function keeps the spindle rotation speed specified at execution of the thread cutting or thread cutting cycle command. When the thread cutting or thread cutting cycle command is terminated, the spindle rotation speed is changed to the value obtained from the position of the constant surface speed control axis and the surface speed. The constant surface speed command cannot be issued from other part systems to the spindle for which the thread cutting command is currently executed. Also, the thread cutting command cannot be issued from other part systems to the spindle in the constant surface speed control mode. An operation error (M01 1113) will occur.

Function name	Operation
Tapping cycle (G84/G88) Synchronous tapping cycle (G84/G88)	<p>If the constant surface speed command is issued in the same part system during execution of the tapping cycle or synchronous tapping cycle command or the tapping cycle command is issued in the same part system in the constant surface speed control mode, the spindle rotation speed for constant surface speed control remains unchanged. (The constant surface speed control is not performed.) This function keeps the spindle rotation speed specified at execution of the tapping cycle command.</p> <p>When the tapping cycle or synchronous tapping cycle command is terminated, the spindle rotation speed is changed to the value obtained from the position of the constant surface speed control axis and the surface speed.</p> <p>The constant surface speed command cannot be issued from other part systems to the spindle for which the tapping cycle or synchronous tapping cycle command is currently executed. Also, the tapping cycle or synchronous tapping cycle command cannot be issued from other part systems to the spindle in the constant surface speed control mode. An operation error (M01 1113) will occur.</p> <p>The synchronous tapping cycle command cannot be executed in the constant surface speed control mode. Program error (P182) will occur.</p> <p>Also, the constant surface speed command cannot be executed during execution of the synchronous tapping cycle command. Program error (P186) will occur.</p>
Spindle-Mode Servo Motor Control	The system also runs when a spindle-mode servo is specified for constant surface speed control
External Spindle Deceleration	If the external spindle deceleration signal is set to OFF for the spindle under constant surface speed control, the spindle is clamped at the external spindle deceleration speed. The S analog maximum/minimum over signal (SOVE) is set on.
High-Speed Simple Program Check	The surface speed is calculated, but the actual rotation speed of the spindle remains set to the value specified before the part system synchronization machine lock high-speed operation is selected.
NC reset (Reset 1/2, reset & rewind)	When NC is reset during constant surface speed control, the spindle rotation speed is set to "0" (r/min).



Precautions

- (1) Under the constant surface speed control (during G96 modal), if the axis targeted for the constant surface speed control (normally X axis for a lathe) moves toward the spindle center, the spindle rotation speed will increase and may exceed the allowable speed of the workpiece or chuck, etc. In this case, the workpiece, etc. may jump out during machining, which may break tools or the machine or injure the operators. Therefore, make sure to use this control while the "spindle speed clamp" is enabled. When the constant surface speed control is commanded, keep enough distance from the program zero point.

Program example

(Example 1) When the parameter "#1146 Sclamp" is set to "0"

G96 S200 ; The spindle rotation speed is controlled so that the surface speed is 200 m/min.
 G92 S4000 Q200 ; The spindle rotation speed is clamped up to 4000 r/min and down to 200 r/min.
 M3 ; The rotation command to the spindle


(Example 2) When the parameter "#1146 Sclamp" is set to "1"

G92 S4000 Q200 ; The spindle rotation speed is clamped up to 4000 r/min and down to 200 r/min.
 G96 S200 ; The spindle rotation speed is controlled so that the surface speed is 200 m/min.
 M3 ; The rotation command to the spindle

<Note>

- ♦For safety, issue the rotation command to the spindle after G92.

WARNING

 **Under the constant surface speed control (during G96 modal), if the axis targeted for the constant surface speed control (normally X axis for a lathe) moves toward the spindle center, the spindle rotation speed will increase and may exceed the allowable speed of the workpiece or chuck, etc. In this case, the workpiece, etc. may jump out during machining, which may break tools or the machine or injure the operators.**

- (2) When the G96 command is issued, do not omit the "S_" surface speed command. If omitted, the system will follow the previous "S_" command.
 The S command ("S_" command) for the spindle in the constant surface speed control mode specifies the surface speed.
- (3) If the spindle speed clamp is not commanded when the constant surface speed control axis is near the zero point, it causes the spindle to rotate at the maximum rotation speed. We recommend that you command the spindle speed clamp before the constant surface speed command.
 In this case, the parameter "#1146 Sclamp" must be made valid, but this function depends on the MTB specifications.
- (4) If an axis number not registered in the command part system is commanded when the constant surface speed command is specified, it causes a program error (P133).

10.3 Spindle Clamp Speed Setting; G92



Function and purpose

The maximum clamp rotation speed of the spindle can be assigned by address S following G92 and the minimum clamp rotation speed by address Q.
 Use this command when the spindle speed needs to be limited depending on the workpiece to be machined, the chuck to be mounted on the spindle and the tool specifications, etc.



Command format

Spindle clamp speed setting

G92 S__ Q__ ;

S	Maximum clamp rotation speed
Q	Minimum clamp rotation speed



Detailed description

- Besides this command, parameters can be used to set the rotation speed range up to 4 stages in 1 r/min units to accommodate gear selection between the spindle and spindle motor. The lowest upper limit and highest lower limit are valid among the rotation speed ranges based on the parameters and based on "G92 S_ Q_".
- Whether to carry out rotation speed clamp only in the constant surface speed mode or even when the constant surface speed is canceled depends on the MTB specifications (parameters "#1146 Sclamp" and "#1227 aux11/bit5").

<Note>

♦G92S command and rotation speed clamp operation

		Sclamp=0		Sclamp=1	
		aux11/bit5=0	aux11/bit5=1	aux11/bit5=0	aux11/bit5=1
Command	In G96	Rotation speed clamp command		Rotation speed clamp command	
	In G97	Spindle rotation speed command		Rotation speed clamp command	
Operation	In G96	Rotation speed clamp execution		Rotation speed clamp execution	
	In G97	No rotation speed clamp		Rotation speed clamp command	No rotation speed clamp

♦The address Q following the G92 command is handled as the spindle speed clamp command regardless of the constant surface mode.


- The command value of the spindle clamp rotation speed will be cleared by modal reset (reset 2 or reset & rewind). Note that the modal is retained if the parameter "#1210 RstGmd / bit19" is ON. It is set to "0" during power ON.



Precautions

- (1) Once the maximum clamp speed and the minimum clamp speed are set using the spindle clamp speed setting (G92 S__ Q __), the maximum speed clamp will not be cancelled even if the command "G92 S0" is issued. During this time, the Q__ value is still valid and $S0 < Q__$ is established. The Q__ value is treated as the maximum speed clamp, and S0 is treated as the minimum speed clamp.
- (2) Note that if the spindle clamp speed setting (G92 S__ Q __) is not commanded, the speed may increase to the machine's maximum specified speed that is set by the parameter. Especially when the constant surface speed control (G96 S__) is commanded, command the spindle clamp speed setting as well as the spindle maximum rotation speed. As the tool moves closer to the spindle center, the spindle rotation speed will increase and may exceed the allowable speed of the workpiece or chuck, etc.

WARNING

-  The spindle clamp speed setting command is a modal command, but make sure to confirm that the G and F modal and coordinate values are appropriate if the operation is started from a block in the middle of the program. If there are coordinate system shift commands or M, S, T and B commands before the block set as the start position, carry out the required commands using the MDI, etc. If the program is run from the set block without carrying out these operations, the machine interference may occur or the machine may operate at an unexpected speed.

10.4 Spindle Position Control (Spindle/C Axis Control)



Function and purpose

This function controls a spindle as the rotary axis. After switching the spindle to the rotary axis, the positioning and the interpolation between the spindle and other NC axes can be operated in the same way as the NC axis by executing the position command (the movement command). Using this function, the NC axis for controlling the spindle stock as the rotary axis or the machinery for switching the spindle and NC axis (such as a gear switching machinery) had been necessary for controlling a spindle stock readily as the rotary axis, but they are not necessary with this function.

For information on how to validate or invalidate this function, each setting to use this function, and the mechanism of your machine, refer to the specifications or the instruction manual issued by the MTB.

There are two methods to switch the spindle and rotary axis: PLC signal method and program command method. The available method depends on the MTB specifications (parameter "#3129 cax_spec/bit0"). For details, refer to the specifications issued by the MTB.

This section describes the program command method.

In this manual, the state of controlling an axis as a spindle is referred to as "spindle mode", and the state of controlling an axis as a rotary axis is referred to as "C axis mode".

The PLC signal processing and operation depends on the MTB specifications. Refer to the instruction manual issued by the MTB for details.

Coordinate zero point and zero point adjustment in C axis mode

For the encoder-based spindle position control (PLG and external encoder), set the Z phase position of the encoder as the first reference point of the C axis. This first reference point is used as the coordinate zero point; however, the spindle zero point position can be adjusted with the spindle/C axis reference position return shift amount parameter, which is determined in the MTB specifications. This parameter is determined in the MTB specifications (parameter "#3113 cax_sft").

Program command method

(1) In the machining program, the program switches to the C axis mode with G00 command, and to the spindle mode with S command. The C axis servo OFF signal (*SVFn) must be always kept ON while the program command method is selected. This depends on the MTB specifications.

When the servo OFF signal is set to OFF, operations are performed as follows.

- The mode cannot be switched from the spindle mode to the C axis mode or from the C axis mode to the spindle mode.
- In the spindle mode, the axis does not run as a spindle even if the forward run command (SRN) or reverse run command (SRI) is executed.
- In the C axis mode, an operation error (M01 0005) occurs if the movement command is executed. In the servo OFF mode, operations follow the setting of the parameter "#1064 svof" (error correction) MTB specifications).

- (2) It depends on the MTB specifications (the parameter "#3129 cax_spec/bit2") either the spindle mode or the C axis mode is set when the power is turned ON.
 If the power is turned ON in the C axis mode setting, the mode shifts to the C axis mode after the Z phase detection and reference position return operations have been performed.
 For Z phase detection, the spindle rotates in the C axis zero point return direction (*2) at the C axis zero point return speed (*1).
- (*1) Depends on the MTB specifications (parameter "#3112 cax_spd").
 (*2) Depends on the MTB specifications (parameter "#3106 zrn_typ/bitA-9").
- (3) It depends on the MTB specifications (the parameter "#3129 cax_spec/bit3") either the spindle mode or the C axis mode is set when NC is reset.



Command format

Switching from spindle mode to C axis mode (C axis)

G00 C__ ;	
C	Target C axis in C axis mode

- ♦ Command "G00 C_" in the NC program during the spindle mode. The axis is positioned directly to the specified position.
 The non-interpolation positioning for each axis is performed by specifying "G00 X__ Z__ C__" regardless of the G00-interpolation OFF parameter ("#1086 G0Intp" in the MTB specifications) setting, and C axis is switched to the C axis mode.
- ♦ Only the G00 command is valid to switch the mode. If the mode is commanded with another G code, it causes a program error (P430).
- ♦ Designate the axis for spindle position control with the absolute value address or absolute value command (G90). If the axis is designated with the incremental value address or incremental value command (G91), it causes a program error (P32).
- ♦ The reference position return type (*1) is set at switching, and the direction to return from the rotation mode to the zero point follows the rotation direction (*2). The direction for returning from the stop mode to the reference position and the interpolation mode depend on the MTB specifications (parameters "#3106 zrn_typ/bit9,bitA", "#3106 zrn_typ/bitD,bitE" and "#1256 set28/bit1").
 (*1) Type to necessarily return to the reference position when switching from the spindle mode to the C axis mode.
 This depends on the MTB specifications (parameter "#3106 zrn_typ/bit8").
 (*2) Depends on the MTB specifications (parameter "#3106 zrn_typ/bitB").
- ♦ If the Z phase is not detected and if switching is commanded, the spindle is rotated in the zero point return direction (*4) at the spindle zero return speed (*3). Then, the zero point return operation is executed after the Z phase detection.
 (*3) Depends on the MTB specifications (parameter "#3112 cax_spd").
 (*4) Depends on the MTB specifications (parameter "#3106 zrn_typ/bitA-9").

[C axis mode switching conditions]

When switching is commanded, all the following conditions must be satisfied.

- ♦ The C axis servo OFF signal (*SVFn) is ON.

Switching C axis mode to spindle mode

- ♦ The switching is performed with the spindle forward run signal (SRN) ON or the spindle reverse run signal (SRI) ON and the S command.
- ♦ The switching is performed with the startup of the spindle forward run signal (SRN) or the spindle reverse run signal (SRI).

[Spindle mode switching condition]

When switching is commanded, all the following conditions must be satisfied.

- ♦ The C axis servo OFF signal (*SVFn) is ON.
- ♦ The C axis selection signal (CMD) is OFF.
- ♦ The C axis is stopped.



Detailed description

Mode switching

- (1) Example in which the mode is switched to the spindle mode with the forward run command and the rotation command (S command)

M03 command -> Forward run command (SRN) ON and reverse run command (SRI) OFF

M04 command -> Reverse run command (SRI) ON and forward run command (SRN) OFF

Program example	Mode	Description
M03 S1000; : :	Spindle mode	The spindle rotates at forward run speed 1000 (r/min).
G00 C90.;	C axis mode	The axis is positioned at 90 degrees directly based on the rotation mode. After positioning, the mode is switched from the spindle mode to the C axis mode.
G01 X10. C20. F100; :		In the C axis mode, the spindle can be commanded as the rotary axis. In the C axis mode, interpolation with another NC axis is possible.
M03 S1500; : :	Spindle mode	The mode is switched from the C axis mode to the spindle mode with the forward run command and rotation command (S command). After being switched to the spindle mode, the spindle rotates at forward run speed 1500 (r/min).
G00 X20.C270.;	C axis mode	The axis is positioned at 270 degrees directly based on the rotation mode, and stops at the position. Simultaneously, the X axis is positioned at 20mm with interpolation. After positioning, the mode is switched from the spindle mode to the C axis mode.

- (2) Example in which the mode is switched to the spindle mode by a change from the forward run command to the reverse run command

M03 command -> Forward run command (SRN) ON and reverse run command (SRI) OFF

M04 command -> Reverse run command (SRI) ON and forward run command (SRN) OFF

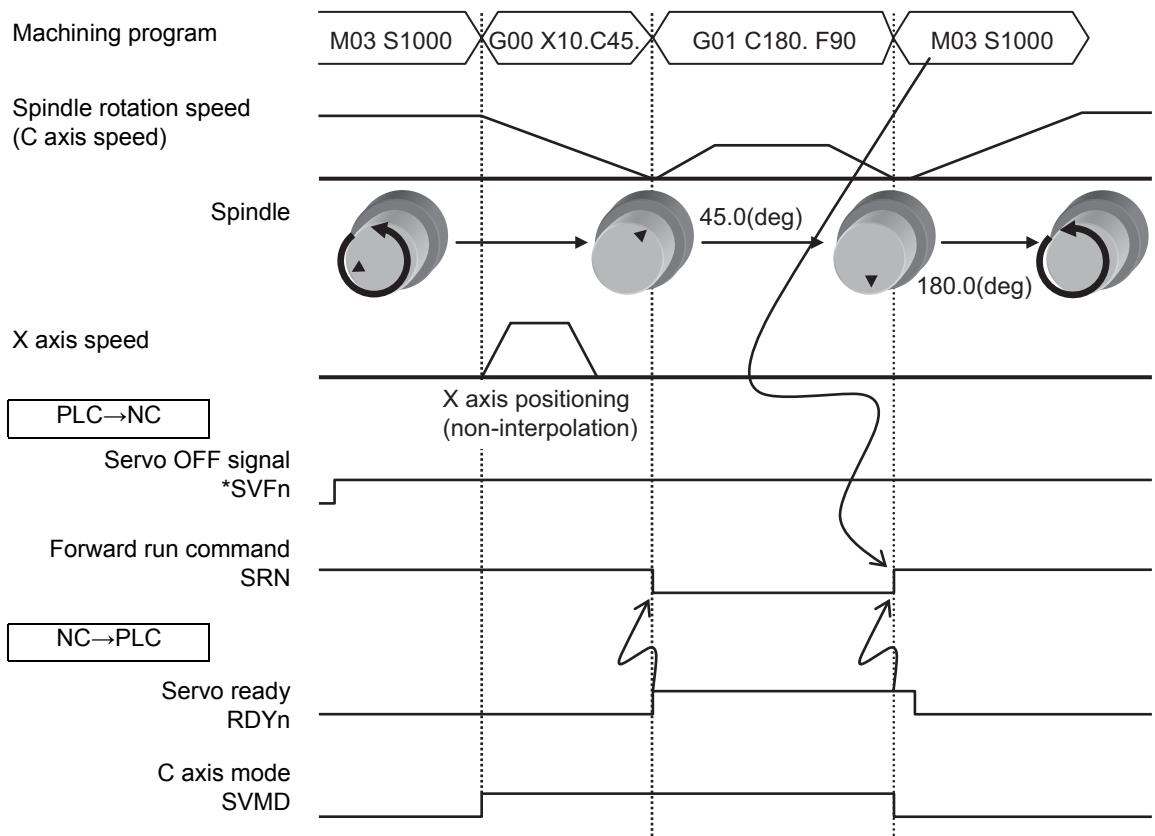
Program example	Mode	Description
M03 S1000; : :	Spindle mode	The spindle rotates at forward run speed 1000 (r/min).
G00 C90.;	C axis mode	The axis is positioned at 90 degrees directly based on the rotation mode. After positioning, the mode is switched from the spindle mode to the C axis mode.
G01 X10. C20. F100 ; :		In the C axis mode, the spindle can be commanded as the rotary axis. In the C axis mode, interpolation with another NC axis is possible.
M4; : :	Spindle mode	The mode is switched from the C axis mode to the spindle mode with the reverse run command. After being switched to the spindle mode, the spindle rotates at reverse run speed 1000 (r/min).
G00 X20.C270.;	C axis mode	The axis is positioned at 270 degrees directly based on the rotation mode, and stops at the position. Simultaneously, the X axis is positioned at 20 mm with interpolation. After positioning, the mode is switched from the spindle mode to the C axis mode.

(3) Example in which the mode is not switched from the C axis mode to the spindle mode
M03 command -> Forward run command (SRN) ON and reverse run command (SRI) OFF

Program example	Mode	Description
M03 S1000; : :	Spindle mode	The spindle rotates at forward run speed 1000 (r/min).
G00 C90.;		
G01 X10. C20. F100; : :	C axis mode	The axis is positioned at 90 degrees directly based on the rotation mode. After positioning, the mode is switched from the spindle mode to the C axis mode.
M3; : :		In the C axis mode, the spindle can be commanded as the rotary axis. In the C axis mode, interpolation with another NC axis is possible.
M3; : :	C axis mode	The rotation command (S command) is omitted between the forward run commands, and the rising edge (change) of the forward run command is not detected; therefore, the mode is not switched to the spindle mode. The forward run command must be changed from OFF to ON with the rotation command (S command) or M3 command.

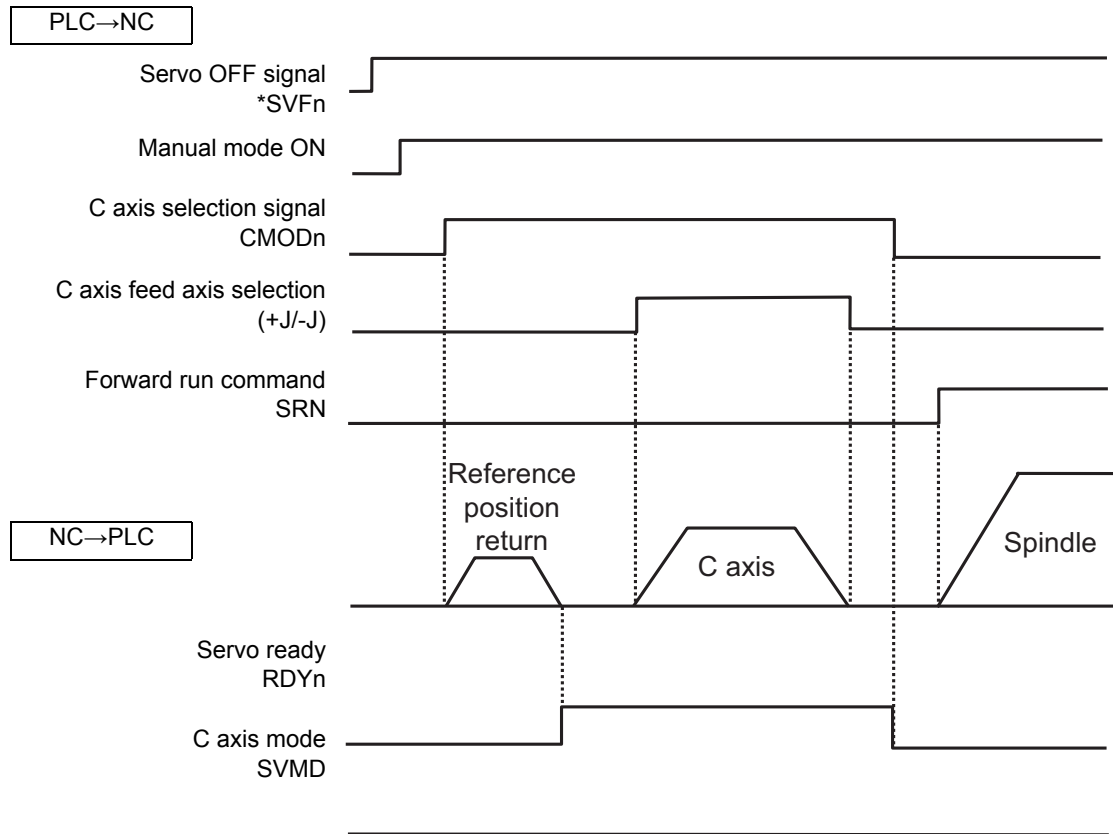
Switching operation

When the program command method is selected, switching operation is performed with the reference position return type.



Manual operation with the program command method selected

When moving the spindle/C axis as the C axis in the manual operation mode, change the C axis selection signal (CMOD) from OFF to ON to switch to the C axis mode. When switching to the spindle mode, change the C axis selection signal from ON to OFF. The switching operation is performed with the reference position return type. In the C axis mode, the axis can be moved by selecting the manual mode (jog mode, handle mode, incremental feed mode, manual arbitrary-feed mode, or reference position return mode).



If the C axis selection signal (CMOD) is changed while either the C axis mode or spindle mode is selected in the program command method, the mode is set as follows. The mode is not switched to the C axis mode or spindle mode in the program command method during automatic running when the C axis selection signal is turned ON. Switching follows the state of the C axis selection signal (CMOD).

C axis selection signal (CMOD)	During automatic operation		During reset	
	C axis mode by "G00 C_ command"	Spindle mode by "S command"	C axis mode	Spindle mode
OFF to ON	C axis mode	C axis mode	C axis mode	C axis mode
ON to OFF	C axis mode	Spindle mode	Spindle mode	Spindle mode
Remarks			Whether the mode is switched to the C axis mode during reset depends on the MTB specifications. (#3129 cax_spec/bit2) (#3129 cax_spec/bit3)	



Relationship with other functions

Spindle forward-run start (SRN) and spindle reverse-run start (SRI)

The mode is switched to the C axis mode regardless of the state of the spindle forward-run start (SRN) or spindle reverse-run start (SRI) signal. In the C axis mode, spindle forward-run start and spindle reverse-run start are invalid. When [PLC signal method] is selected, the spindle rotates by carrying out the spindle forward-run start or spindle reverse-run start again (OFF to ON operation) after the C axis mode has been canceled. When [Program command method] is selected, the spindle rotates by carrying out the spindle forward-run start or spindle reverse-run start again (OFF to ON operation) in the C axis mode or by issuing the S command with the spindle forward-run start or spindle reverse-run start set ON.

Spindle orientation signal (ORC)

In the C axis mode, the spindle orientation command (ORC) is invalid. Also, the spindle position control command (spindle/C axis control) is invalid if issued during spindle orientation.

Spindle gear switching

Gear switching cannot be performed in the C axis mode. After the mode has been changed from the C axis mode to the spindle mode, gear switching is performed. Also, the mode cannot be switched to the C axis mode during gear switching. After gear switching has been completed, the mode is switched to the C axis mode.

Coil switching

Coil switching is invalid in the C axis mode. Conduct coil switching before switching to the C axis mode. If switching to the C axis mode is commanded during coil switching, switching to the C axis mode is executed after coil switching has been completed.

Spindle synchronization I

(1) Spindle synchronization I

If the reference spindle or the synchronized spindle under the synchronization control is switched to the C axis mode, it causes an operation error (M01 1026). Also, if the reference spindle command or synchronized spindle command is issued to the spindle in the C axis mode, it causes an operation error (M01 1026). The alarm can be deactivated by canceling the synchronization command or the C axis mode.

<Note>

- ◆When the specifications of the C axis control mode under spindle synchronization are valid, this function enables the spindle position control by the reference spindle under spindle synchronization control.

Spindle override

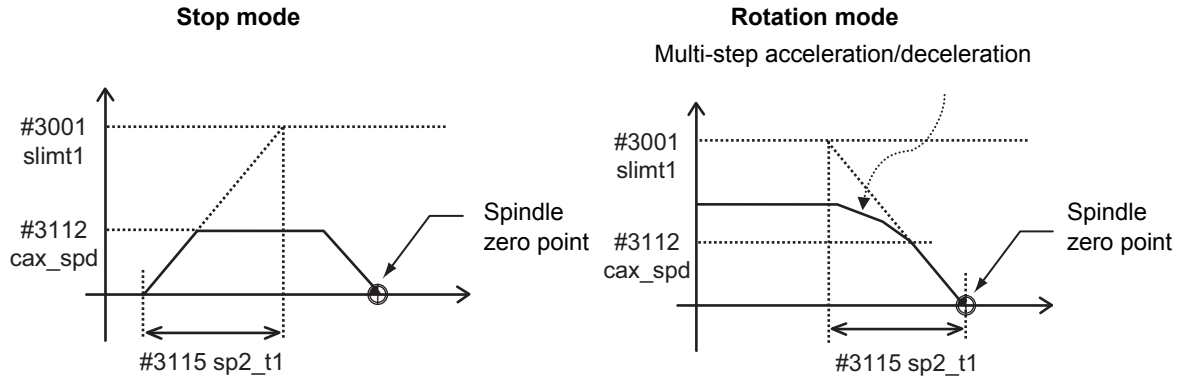
The spindle override is invalid for the reference position return operation at switching to the C axis mode. In the C axis mode, the spindle override is invalid. The cutting feed override or rapid traverse override of the NC axes is valid in the C axis mode.

Spindle-mode servo motor control

The spindle position control is valid, excluding the following differences.

(1) Speed pattern of the reference position return operation

The reference position return from the stop mode is performed by the same operation as for the normal spindle. However, when the spindle returns from the rotation mode to the zero point, the rotation speed decelerates up to the C axis zero point return speed with the multi-step acceleration/deceleration pattern. After the C axis zero point return speed has been reached, the spindle inclines to the zero point, and stops while decelerating at the constant speed. The multi-step acceleration/deceleration pattern and C axis reference position return speed depend on the MTB specifications (parameters "#3054 spct1" to "#3061 spdiv1" and "#3112 cax_spd").



Analog spindle

The spindle position control is invalid.

Absolute position detection

The absolute position detection is invalid in the C axis mode.

Constant surface speed control

When Program command method is selected, the mode is switched from the C axis mode to the spindle mode if the surface speed command S (m/min) and the spindle forward-run start (SRN) or spindle reverse-run start (SRI) signal is set to ON.

Manual arbitrary reverse run

When the program command method is selected, "block switched from spindle mode to C axis mode (example: G00 C_) and "block switched from C axis mode to spindle mode (example: M03 S1000)" is handled as a reverse run prohibited block. The reverse run cannot be carried out back through blocks with the mode switched.

Coordinate system setting (G92), Local coordinate system setting (G52)

The offset values of the coordinate system setting and local coordinate system setting configured in the C axis mode are retained even in the spindle mode.

After this, whether these offset values designated in the previous C axis mode are to be retained when the spindle mode is switched to the C axis mode depends on the MTB specifications (parameter "#3129 cax_spec/bit5").



Precautions and restrictions

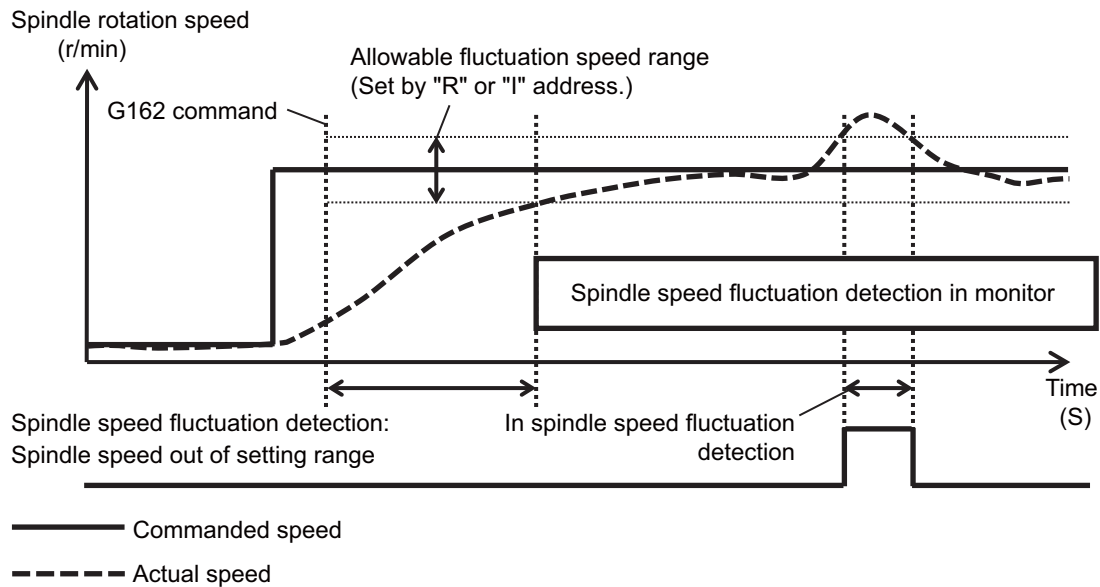
- (1) If the movement is commanded while the servo OFF signal (*SVFn) is set to OFF, it causes an operation error (M01 0005). Reset NC to cancel the error, and set the servo OFF signal on to restart machining. If the spindle command is issued, the spindle does not rotate.
- (2) If the servo OFF signal (*SVFn) is set to OFF during C axis movement, it causes an operation error (M01 0005). Reset NC to cancel the error.
- (3) To switch from the spindle mode to the C axis mode, issue the G00 command. If a command other than the G00 command is issued, it causes a program error (P430).
- (4) The spindle position control axis must be commanded with the absolute value address or absolute value command (G90). If the incremental value address or incremental value command (G91) is used, it causes a program error (P32).
- (5) When the spindle mode is switched to the C axis mode, in-position check is applied regardless of the deceleration check designation type (*1).
(*1) This designation depends on the MTB specifications (parameter "#1193 inpos").

10.5 Spindle Speed Fluctuation Detection; G162/G163



Function and purpose

When this function is valid and the spindle actual speed fluctuates relative to the programmed speed due to external factors such as load fluctuation, the NC outputs the signal (Spindle speed out of setting range) to PLC and causes the operation error (M01 1105) at the same time. PLC can take the necessary measure for the fluctuation of the spindle speed using the output signal (spindle speed out of setting range) from the NC. The operation error (M01 1105) output from the NC does not stop the cycle operation or the spindle. Whether or not to output the operation error during spindle speed fluctuation detection (G162) depends on the MTB specifications (parameter "#1242 set14/bit2").



Term

The following descriptions are the meanings of the terms used in this manual.

Term	Meaning
Spindle command speed	Spindle command speed is the command speed to which the spindle override and the spindle clamp speed have been added. This is the spindle last command speed sent to the spindle drive unit.
Spindle actual speed	This is the speed fed back from the spindle, at which the spindle actually runs.
Allowable fluctuation range	This indicates the allowable deviation range from the command speed in spindle speed fluctuation detection. The calculation result for the command speed of "spindle speed fluctuation allowance rate" (R address or parameter) or "allowable spindle speed fluctuation range" (I address or parameter), whichever is greater, is used as the allowable fluctuation range. When the calculation result for the command speed of "detection range to achieve spindle speed", which is used to determine whether or not the spindle rotation speed achieves the command speed, is greater than the calculation result for the command speed of "allowable fluctuation rate of spindle speed" or "allowable fluctuation range of spindle speed", the range of "detection range to achieve spindle speed" is used as allowable fluctuation range.



Command format

Starting the spindle speed fluctuation detection

```
G162 S__ P__ Q__ R__ I__ ;
```

S	Spindle name of detection target
P	Spindle speed fluctuation detection start delay time
Q	Spindle up-to-speed detection width
R	Allowable fluctuation rate of spindle speed
I	Allowable fluctuation range of spindle speed

Spindle speed fluctuation detection cancel

```
G163 S__ ;
```

S	Spindle name of detection target
---	----------------------------------



Detailed description

Description of each address

Address	Command range (unit)	Remarks
S	1 to 9	<ul style="list-style-type: none"> ◆ This sets the name of the spindle which performs the spindle speed fluctuation detection. The spindle number is used for this command, but the value set by the parameter (#3077) is used when the spindle name method is valid. ◆ When this address is omitted from the G162 command, the spindle selected in the commanded part system is treated as the commanded spindle. ◆ When this address is omitted from the G163 command, the fluctuation detection for all spindles is canceled. ◆ When the set spindle is not mounted, the program error (P35) occurs.
P	0 to 99.999(s)	<ul style="list-style-type: none"> ◆ This sets the delay time from the time when the spindle speed fluctuation detection (G162) is commanded to the time to start the fluctuation detection. Also when the spindle command speed changes, the delay time is set. The change of the spindle command speed means the change of the spindle last command sent to the spindle drive unit. ◆ When this address is omitted, the value set in the parameter (#1242 set14/bit2) is used. ◆ When the command value exceeds the command range, the program error (P35) occurs.
Q	1 to 100 (%)	<ul style="list-style-type: none"> ◆ This sets the range for the spindle speed command, with which the control determines whether the spindle reaches the command speed in order to start fluctuation detection. ◆ When this address is omitted, the value set in the parameter (#3105) is used. ◆ When the command value exceeds the command range, the program error (P35) occurs.

Address	Command range (unit)	Remarks
R	1 to 100 (%)	<ul style="list-style-type: none"> • This sets the allowable fluctuation speed range calculated for the spindle command speed. When the actual spindle speed exceeds the range, the signal is output to PLC and the operation error (M01 1105) occurs. This sets the ratio of the speed deviation to the command speed. • When this address is omitted, the value set in the parameter (#1242 set14/bit2) is used. • When the command value exceeds the command range, the program error (P35) occurs.
I	0 to 999999 (r/min)	<ul style="list-style-type: none"> • This sets the allowable fluctuation speed range calculated for the spindle command speed. When the actual spindle speed exceeds the range, the signal is output to PLC and the operation error (M01 1105) occurs. This sets the speed deviation from the command speed. • When this address is omitted, the value set in the parameter (#1242 set14/bit2) is used. • When the command value exceeds the command range, the program error (P35) occurs.

Enabling conditions of the function

- (1) This function is valid when the spindle speed fluctuation detection (G162) is commanded during the cycle operation.
- (2) When G162 is commanded during the cycle operation, it is valid until G163 command (cancel command), cycle operation end, reset or emergency stop.
- (3) If any function that is unusable with this function is enabled while this function is valid, this function is temporarily canceled. After that, when the said unusable function is disabled, this function is enabled.
- (4) When G162 is commanded while any unusable function is valid, this function is temporarily canceled. After that, when the said unusable function is disabled, this function is enabled.



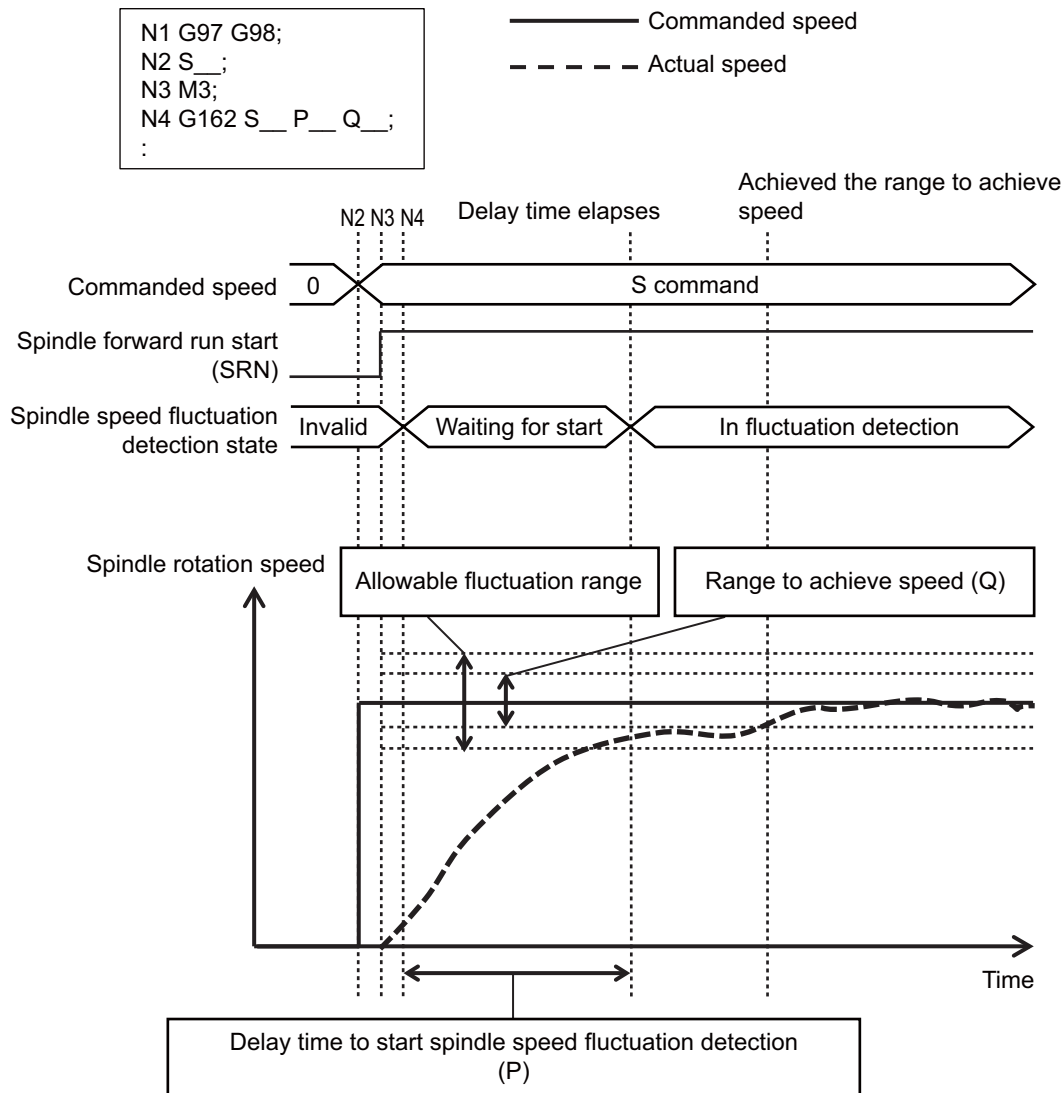
Operation example

Start timing of spindle speed fluctuation detection

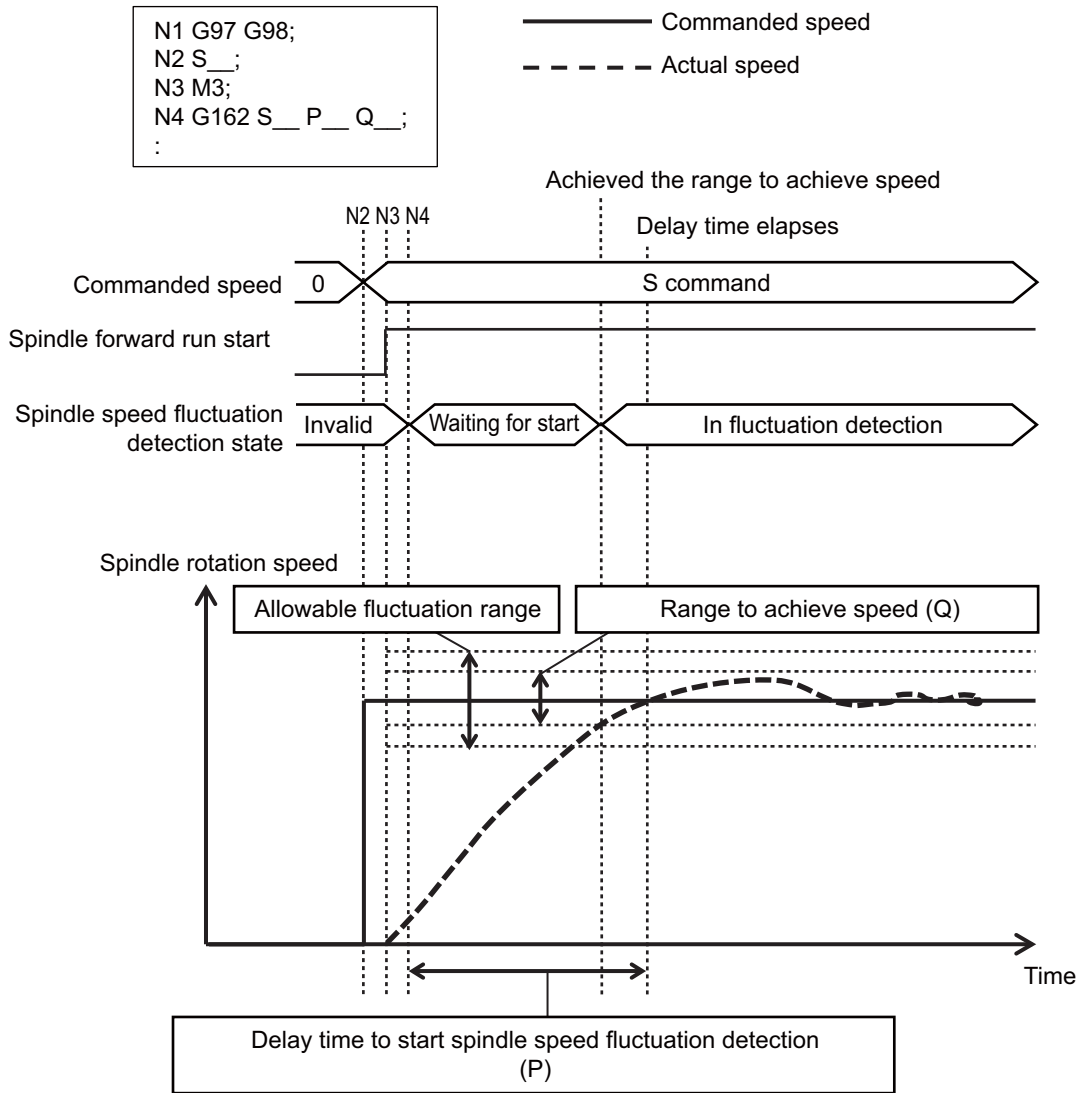
When one of the following conditions is satisfied after G162 command, the spindle speed fluctuation detection starts:

- Case in which the start delay time of the spindle speed fluctuation detection (set by "P") elapses (Refer to (1) figure.)
- Case in which the spindle actual speed is within the detection range to achieve spindle speed (set by "Q") (Refer to (2) figure.)

(1) Case in which the start delay time of the spindle speed fluctuation detection (set by "P") elapses

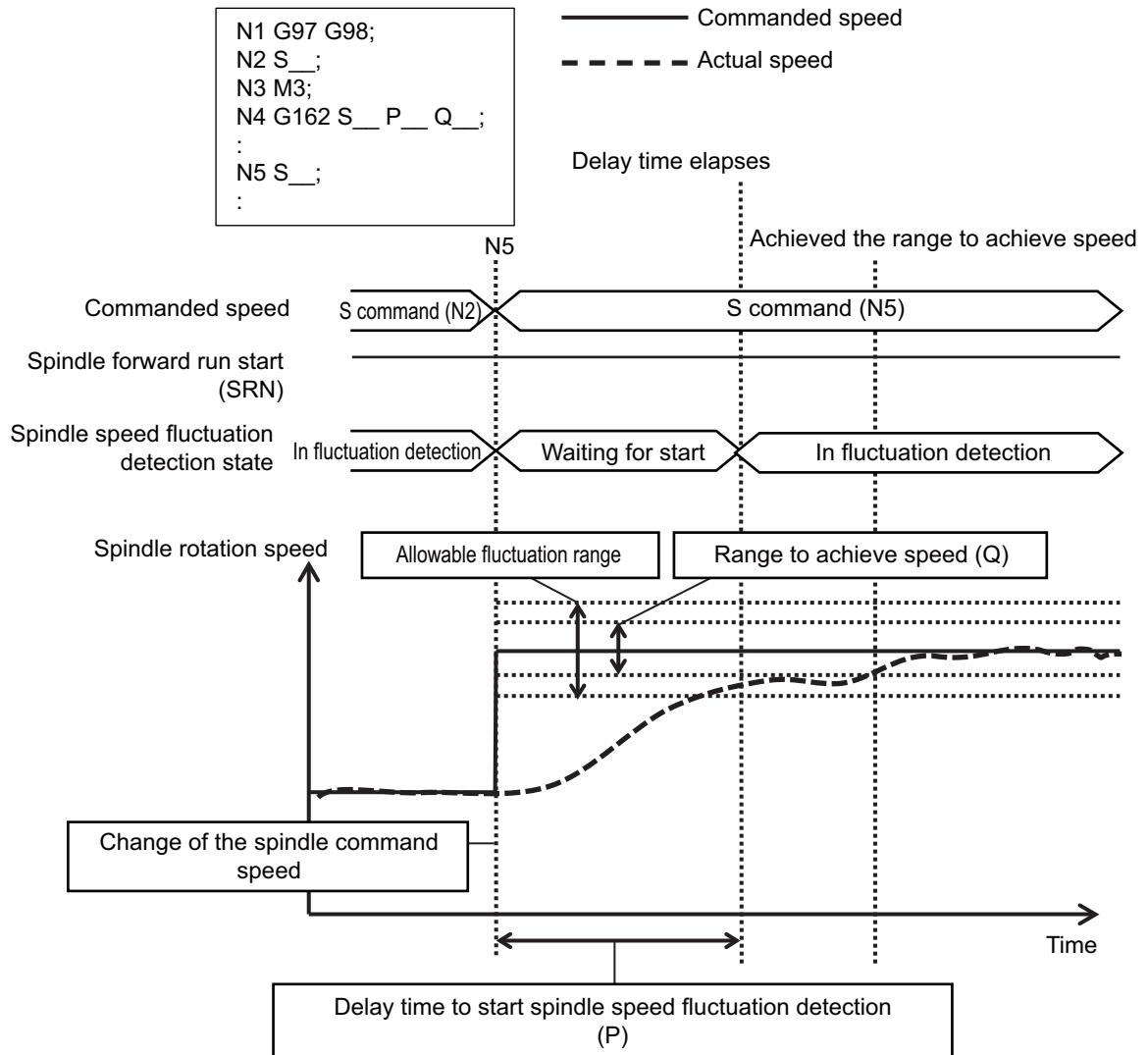


(2) Case in which the spindle actual speed is within the detection range to achieve spindle speed (set by "Q")



Fluctuation detection start timing when the spindle command speed is changed

When the spindle command speed is changed by S command or spindle override, the state is the same as the one immediately after G162 command and the fluctuation detection is not performed until the condition of "Start timing of spindle speed fluctuation detection" is satisfied. When this condition is satisfied, the fluctuation detection is started. Also, if the speed of synchronized spindle, which is driven based on the reference spindle speed under synchronization etc., changes, the control interprets that the spindle command speed is changed. (Spindle synchronization control/Polygon machining/Hobbing/Spindle superimposition control)



Temporary cancellation operation of spindle speed fluctuation detection

The spindle speed fluctuation detection function cannot be combined with the following functions. This is because these functions cause the spindle speed to be changed frequently without ensuring the spindle rotation at a constant speed.

- ♦Synchronous tapping (Synchronous tapping cycle/Pecking tapping cycle/Deep-hole tapping cycle/High-speed synchronous tapping)
- ♦Spindle orientation
- ♦C axis mode of spindle position control (spindle/C axis control)

When any of the functions above is enabled while the spindle speed fluctuation detection is enabled, this function is temporarily canceled. After that, when the said unusable function becomes invalid, this function is enabled. When this function is commanded while an unusable function is running, this function is temporarily canceled. After that, when the said unusable function becomes invalid, this function is enabled. When this function is enabled, the state is the same as the one immediately after G162 command. Fluctuation detection is started if the condition of "Start timing of spindle speed fluctuation detection" is satisfied.

When the spindle speed fluctuation detection command (G162) is performed during the spindle speed fluctuation detection

When the command with the exact same settings is given to the axis where this function is enabled, the command is ignored. On the other hand, when at least one of the settings other than S address (P, Q, I, or R) is different, the setting is changed to the new one. After that, the state is the same as the one immediately after G162 command. Fluctuation detection is started if the condition of "Start timing of spindle speed fluctuation detection" is satisfied. Changing the S address means that the spindle speed fluctuation detection is commanded to a different spindle.

**Relationship with other functions****Synchronous tapping**

This function is temporarily canceled during synchronous tapping (synchronous tapping cycle/pecking tapping cycle/deep-hole tapping cycle/high-speed synchronous tapping).

Multiple-spindle control I/II

When G162 command is commanded omitting S address, the operation is performed to the spindle which is selected by multiple-spindle control.

Spindle orientation

This function is temporarily canceled during spindle orientation.

Spindle position control (Spindle/C axis)

This function is temporarily canceled in the C axis mode. The fluctuation detection can be performed in the spindle mode.

Constant surface speed control

Constant surface speed control can be combined with this function. However, the fluctuation detection is not performed unless the fluctuation detection start condition is satisfied each time the spindle speed fluctuates; therefore, the fluctuation detection may not be performed much.

Manual arbitrary reverse run

Manual arbitrary reverse run cannot be performed to the command of this function.

**Precautions**

- (1) While the spindle is stopped, the spindle speed fluctuation detection is not performed.
- (2) The spindle speed fluctuation detection is not performed to the speed which is the minimum rotation speed (parameter #3032) or less.
- (3) The spindle speed fluctuation detection is not performed during synchronous tapping, spindle orientation, C axis control mode of spindle or C axis.
- (4) When any other command is issued at the same time, the program error (P45) occurs.

Tool Functions (T command)

11.1 Tool Functions (T8-digit BCD)



Function and purpose

The tool functions are also known as T functions and they assign the tool numbers. This control unit specifies a tool number with an 8-digit (0 to 99999999) number following the address T, and up to four sets can be commanded into one block. However, the number of sets that can be commanded within the same block depends on the MTB specifications (parameter "#12009 Tfig").

One of the following output signals is issued depending on the parameter setting (depends on the machine specifications).

- 8-digit BCD code and start signal
- Signed 32-bit binary data and start signal
- Unsigned 32-bit binary data and start signal

If the T function is designated in the same block as a movement command, the commands may be executed in either of the following two orders. The machine specifications determine which sequence applies.

- (1) The T function is executed after completion of the movement.
- (2) The T function is executed simultaneously with the movement command.

Processing and completion sequences are required for all T commands.

Tool Compensation Functions

12.1 Tool Compensation

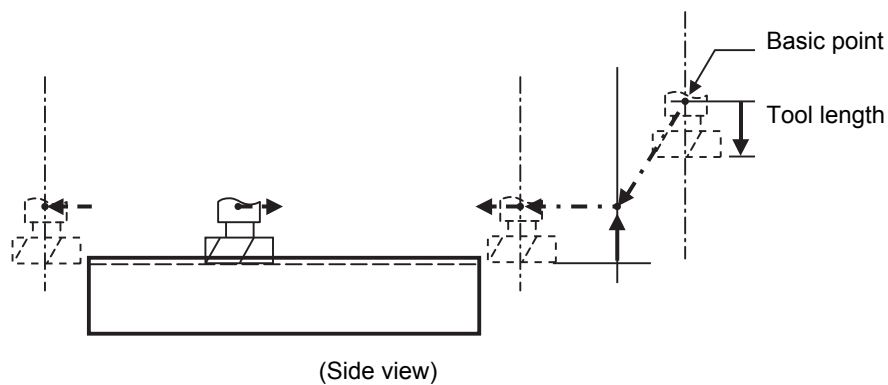
12.1.1 Tool Compensation



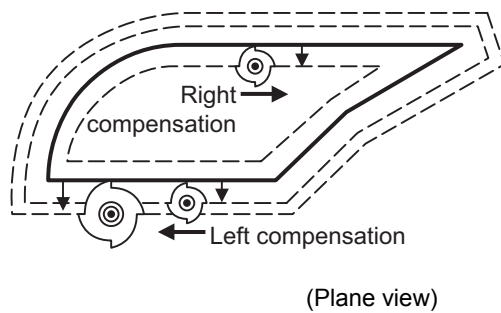
Function and purpose

The basic tool compensation function includes the tool length compensation and tool radius compensation. Each compensation amount is designated with the tool compensation No. Each compensation amount is input from the setting and display unit or the program.

Tool length compensation



Tool radius compensation



Tool compensation memory

There are two types of tool compensation memories, types I and II, used to set and select the tool compensation amount. (The type used is determined by the MTB specifications.)

Each of types I and II can be changed to type III depending on the MTB settings (parameter "#1046 T-ofs disp type"). If the tool compensation memory is changed to type III, you can register the tool compensation amount of the base axes I, J, and K and the tool tip point, enabling tool compensation for a turning tool.

The details of type III are also displayed on the screen.

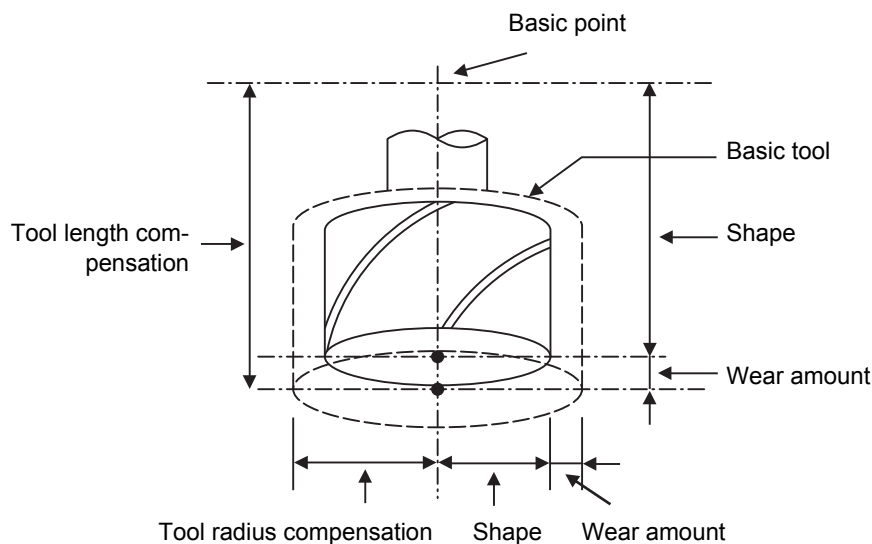
If the tool compensation memory is reset to the original type, the turning tool compensation items are not displayed. However, when it is changed to type III, the previously registered data is displayed.

The compensation amount settings are preset with the setting and display unit.

Type I is selected when parameter "#1037 cmdtyp" is set to "1", and type II is selected when set to "2".

Type of tool compensation memory	Classification of length compensation, radius compensation	Classification of shape compensation, wear compensation
Type I	No	No
Type II	Yes	Yes
Type III	(*1)	Yes

(*1) Distinguished between tool length compensation and tool nose radius compensation.



12 Tool Compensation Functions

Type I

One compensation amount corresponds to one compensation No. as shown below. Thus, these can be used commonly regardless of the tool length compensation amount, tool radius compensation amount, shape compensation amount and wear compensation amount.

$(D1) = a1, (H1) = a1$

$(D2) = a2, (H2) = a2$

:
:

$(Dn) = an, (Hn) = an$

Compensation No.	Compensation amount
1	a1
2	a2
3	a3
:	:
:	:
n	an

If the tool compensation type is changed from type I to type III, the tool compensation amount of type I is handled as tool length Z of type III.

Type II

The shape compensation amount related to the tool length, wear compensation amount, shape compensation related to the tool radius and the wear compensation amount can be set independently for one compensation No. as shown below.

The tool length compensation amount is set with H, and the tool radius compensation amount with D.

$(H1) = b1 + c1, (D1) = d1 + e1$

$(H2) = b2 + c2, (D2) = d2 + e2$

:
:

$(Hn) = bn + cn, (Dn) = dn + en$

Compensation No.	Tool length (H)		Tool radius (D)/(Position compensation)	
	Shape compensation amount	Wear compensation amount	Shape compensation amount	Wear compensation amount
1	b1	c1	d1	e1
2	b2	c2	d2	e2
3	b3	c3	d3	e3
:	:	:	:	:
:	:	:	:	:
n	bn	cn	dn	en

If the tool compensation type is changed from type II to type III, data registered for type II is handled as the following data.

Type II	Type III
Length dimension	Tool length Z
Length wear	Z wear
Radius dimension	Tool nose R
Radius wear	R wear

 CAUTION

 If the tool compensation amount is changed during automatic operation (including during single block stop), it will be validated from the next block or multiple blocks onwards.

Tool compensation No. (H/D)

This address designates the tool compensation No.

- (1) H is used for the tool length compensation, and D is used for the tool position compensation and tool radius compensation.
- (2) The tool compensation No. that is designated once does not change until a new H or D is designated.
- (3) The compensation No. can be commanded once in each block. (If two or more Nos. are commanded, the latter one will be valid.)
- (4) For 40 sets:
Designate with the H01 to H40 (D01 to D40) numbers.
- (5) If a value larger than this is set, the program error (P170) will occur.
- (6) The setting value ranges are as follows for each No.
The compensation amount for each compensation No. is preset with the setting and display unit.

Setting	Shape compensation amount		Wear compensation amount	
	Metric system	Inch system	Metric system	Inch system
#1003=B	± 999.999 (mm)	± 99.9999 (inch)	± 999.999 (mm)	± 99.9999 (inch)
#1003=C	± 999.9999 (mm)	± 99.99999 (inch)	± 999.9999 (mm)	± 99.99999 (inch)
#1003=D	± 999.99999 (mm)	± 99.999999 (inch)	± 999.99999 (mm)	± 99.999999 (inch)
#1003=E	± 999.999999 (mm)	± 99.9999999 (inch)	± 999.999999 (mm)	± 99.9999999 (inch)

12.1.2 Number of Tool Offset Sets Allocation to Part Systems



Function and purpose

The number of tool offset sets can be set per part system.

This function is divided into the following methods and which one is used depends on the MTB specifications (parameters "#1438 Ofs-SysAssign", "#12054 Tol-Ofsnum").

Arbitrary allocation: Arbitrarily allocates to each part system.

Fixed allocation: Automatically and evenly allocates to each part system.

The arbitrary allocation enables the efficient allocation because when a certain part system needs only a small number of offset sets, the rest can be allocated to another part system. If an auxiliary-axis part system does not need the tool offset set at all, the number of tool offset sets can be set to "0" for the auxiliary-axis part system.

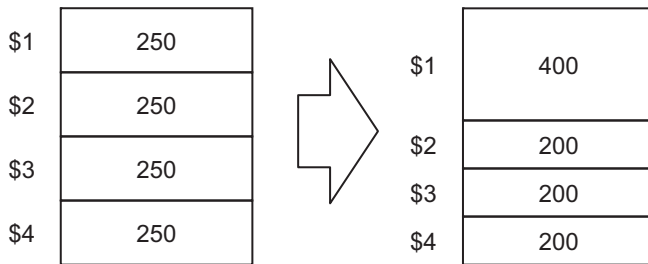
While this function is available if the specification allows allocation by tool compensation memory part system, this parameter depends on the MTB specification parameter "#1051 MemTol"). Subsequent description is an example in the case where the number of tool offset sets in the system is 999. Number of tool offset sets in system is the total number of tool offset sets of all part systems.

(1) Arbitrary allocation (with #1438=1)

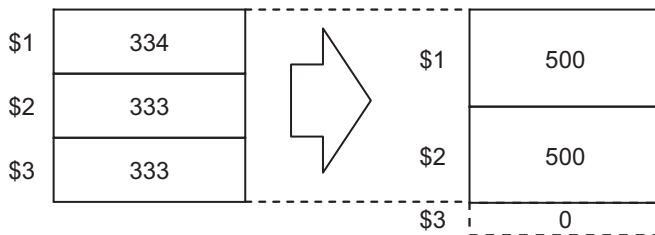
The number of tool offset sets allocated to each part system depends on the MTB specifications (parameter "#12054 Tol-ofsnum").

The following example shows the number of tool offset sets allocated when the lathe system is a 4-part system.

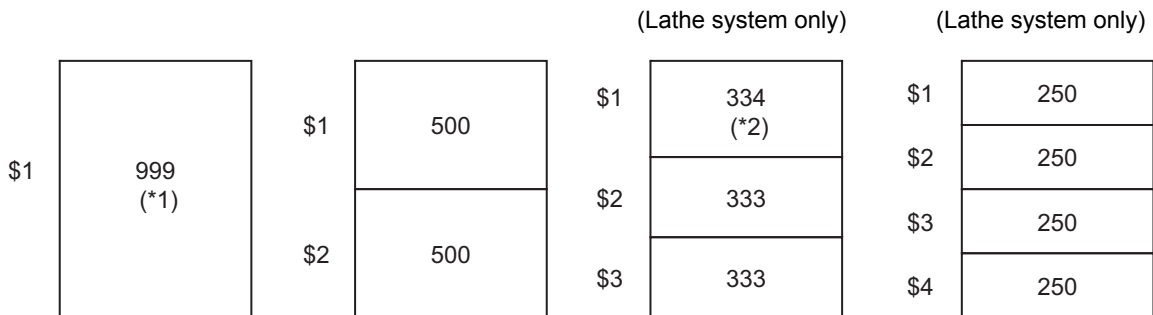
(a) When the number of tool offset sets is increased for the 1st part system of 4-part system



(b) When the number of offset sets is set to "0 sets" for the 3rd part system to use the 3rd part system as auxiliary-axis part system



(2) Fixed allocation (with #1438=0)



(*1) The maximum number of tool offset sets per part system is 999.

(*2) If there is any remainder, the remainder sets are allocated to the 1st part system.



Precautions

- (1) The maximum number of tool offset sets for 1-part system is 999.
- (2) For 1-part system, up to the number of tool offset sets in the system is available regardless of the parameter setting.
- (3) When the value of the parameter "#12054 Tol-Ofsnum" (the number of tool offset sets by arbitrary allocation) is equal to or below the number of tool offset sets in the system, the remainder is not allocated to any part system even if the specification allows arbitrary allocation.
- (4) When the tool compensation memory is provided commonly for the part systems ("#1051 MemTol"=1), the number of tool offset sets in the system are commonly used by all part systems regardless of the parameter setting. The setting of parameter #1051 depends on the MTB specifications, so check it in your machine specifications.
- (5) Even if the specification allows arbitrary allocation, fixed allocation is applied if the parameter is "#12054 Tol-Ofsnum"= 0.
- (6) When entering offset data, if the number of offset data exceeds that of current tool offset sets, the excess offset data cannot be entered.

12.2 Tool Length Compensation/Cancel; G43, G44 / G49



Function and purpose

The end position of the movement command for each axis can be compensated for by the preset amount when this command is issued. A continuity can be applied to the program by setting the actual deviation from the tool length value decided during programming as the compensation amount using this function.



Command format

Tool length compensation start

G43 Zz Hh ; (+ direction)

G44 Zz Hh ; (- direction)

Tool length compensation cancel

G49 Zz ;



Detailed description

Tool length compensation movement amount

The movement amount is calculated with the following expressions when the G43 or G44 tool length compensation command or G49 tool length compensation cancel command is issued.

	Z axis movement amount	Operation
G43 Zz Hh1;	$z + (lh1)$	Compensation in + direction by tool compensation amount
G44 Zz Hh1;	$z - (lh1)$	Compensation in - direction by tool compensation amount
G49 Zz;	$z - (+)(lh1)$	Compensation amount cancel

lh1; Compensation amount for compensation No. h1

Regardless of the absolute value command or incremental value command, the actual end point will be the point compensated for by the compensation amount designated for the programmed movement command end point coordinate value.

The G49 (tool length compensation cancel) mode is entered when the power is turned ON or when M02 has been executed.

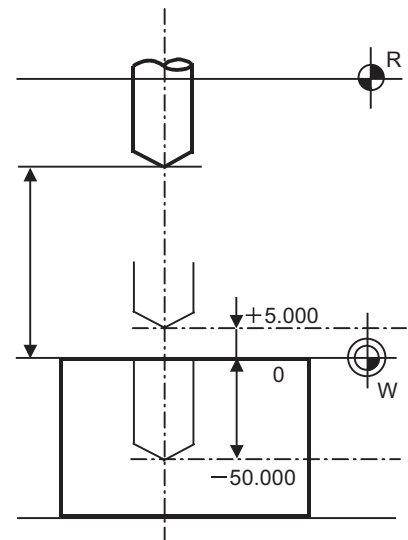
(Example 1) For absolute value command H01
= -100000

```
N1 G28 Z0 T01 M06 ;
N2 G90 G92 Z0 ;
N3 G43 Z5000 H01 ;
N4 G01 Z-50000 F500 ;
```

(Example 2) For incremental value command H01
=-100000

```
N1 G28 Z0 T01 M06 ;
N2 G91 G92 Z0 ;
N3 G43 Z5000 H01 ;
N4 G01 Z-55000 F500 ;
```

Tool length compensation
H01=-100.

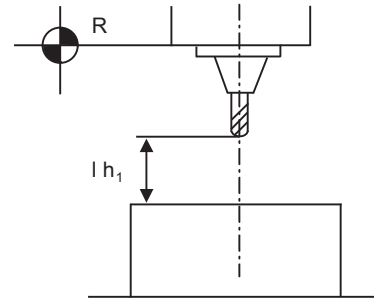


Compensation No.

(1) The compensation amount differs according to the compensation type. The following example shows a case in which "G43 Hh1;" is commanded.

Type I

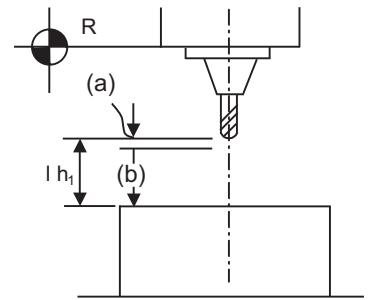
The compensation amount lh1 commanded with compensation No. h1 will be applied commonly regardless of the tool length compensation amount, tool radius compensation amount, shape compensation amount or wear compensation amount.



Type II

The compensation amount lh1 commanded with compensation No. h1 is as follows.

lh1: Shape compensation (b) + wear compensation amount (a)



Type III

The compensation amount lh1 commanded with compensation No. h1 is as follows. (Refer to the figure of type II.)

lh1: Tool length compensation amount in Z axis direction (b) + Wear compensation amount in Z axis report (a)

- (2) The valid range of the compensation No. will differ according to the specifications (No. of compensation sets).
- (3) If the commanded compensation No. exceeds the specification range, the program error (P170) will occur.
- (4) Tool length cancel will be applied when H0 is designated.
- (5) The compensation No. commanded in the same block as G43 or G44 will be valid for the following modals.

(Example 3)

```
G43 Zz1 Hh1 ;           Tool length compensation is executed with h1.
:
G45 Xx1 Yy1 Hh6 ;
:
G49 Zz2 ;             Tool length compensation is canceled.
:
G43 Zz2 ;           Tool length compensation is re-executed with h1.
:
```

(6) If G43 is commanded in the G43 modal, a compensation of the difference between the compensation No. data will be executed.

(Example 4)

```
G43 Zz1 Hh1 ;           The axis moves by "z1 + (lh1)".
:
G43 Zz2 Hh2 ;         The axis moves by "z2 + (lh2-lh1)".
:
```

The same applies for the G44 command in the G44 modal.

Axis valid for tool length compensation

- (1) When parameter "#1080 Dril_Z" is set to "1", the tool length compensation is always applied to the Z axis.
- (2) When parameter "#1080 Dril_Z" is set to "0", the axis will depend on the axis address commanded in the same block as G43. The order of priority is shown below.

Zp > Yp > Xp

(Example 5)

```
G43 Xx1 Hh1;          + compensation to X axis
:
G49 Xx2 ;
:
G44 Yy1Hh2;          - compensation to Y axis
:
G49 Yy2 ;
:
G43 αα1 Hh3;          + compensation to additional axis
:
G49 αα1 ;
:
G43 Xx3Yy3Zz3 ;      Compensation is applied on Z axis.
:
G49 ;
```

The handling of the additional axis will follow the parameters "#1029 aux_I" to "1031 aux_K" settings.

If the tool length compensation is commanded for the rotary axis, set the rotary axis name for one of the parallel axes.

- (3) If H (compensation No.) is not designated in the same block as G43, the Z axis will be valid.

(Example 6)

```
G43 Hh1 ;            Compensation and cancel to Z axis
:
G49 ;
:
```

Movement during other commands in tool length compensation modal

- (1) If reference position return is executed with G28 and manual operation, the tool length compensation will be canceled when the reference position return is completed.

(Example 7)

```
G43 Zz1 Hh1 ;
:
G28 Zz2 ;            Canceled when reference position is reached. (Same as G49)
:
G43 Zz2Hh2 ;
:
G49 G28Zz2 ;        The tool length compensation will be included when positioning the intermediate point.
                    Canceled when reference position is reached.
```

- (2) The movement is commanded to the G53 machine coordinate system, the axis will move to the machine position without tool compensation amount.

When the G54 to G59 workpiece coordinate system is returned to, the position returned to will be the coordinates shifted by the tool compensation amount.

Movement by tool length compensation command

When the independent tool length compensation command (G43/G44) or independent tool length compensation cancel command (G49) is issued, the movement by the compensation amount may not be carried out depending on the MTB specifications (parameter "#1247/bit0" (movement by tool length compensation command)).

	Without movement (#1247/bit0 = 1)		With movement (#1247/bit0 = 0)	
Independent tool length compensation command	: G54 A0. C0. G68.2 G53.1 G00XxYyZz G43H1 : G49			
	The axis does not move when Tool length compensation is commanded independently, but the tool compensation amount is applied to the program position counter.		The axis moves by the compensation amount when Tool length compensation is commanded independently.	
Tool length compensation command with movement	: G54 A0. C0. G68.2 G53.1 G00XxYyZz G43H1Z0 : G49Z10.			
	The axis moves by the compensation amount when the tool length compensation with travel is commanded.		The axis moves by the compensation amount when the tool length compensation with travel is commanded.	

Tool length compensation of multiple axis synchronization control function

Tool length compensation method for multiple axis synchronization control depends on the MTB specifications (parameter "#1255 set27/bit5").

- When "#1255 set27/bit5" is set to "1"

Tool length compensation for multiple axis synchronization control is enabled and different amount of tool length compensation for each axis can be applied to the target axis for multiple axis synchronization control. Even when the tool with different length for each head is mounted with multi head configuration machine, the different amount of tool length compensation can be applied to each head and the tool length error between heads can be compensated.

- When "#1255 set27/bit5" is set to "0"

Tool length compensation for multiple axis synchronization control is disabled and the same tool length compensation amount as the master axis is applied to the slave axis.

[Operation of tool length compensation]

With tool length compensation for multiple axis synchronization control, the tool compensation amount of difference tool compensation No. can be applied to each axis. Tool compensation amount of the tool compensation No. which the value of the "2675 tcmp_top" added to commanded No. is applied.

Tool length compensation for multiple axis synchronization control is applicable only to axes which are actually in the synchronized state. Therefore, the axis where the bit of PLC signal Synchronization control operation method signal (R2589) is turned OFF is not the target for tool length compensation.

Tool length compensation for multiple axis synchronization control is started with G43, G44 command in the same way as the normal tool length compensation. To cancel the tool length compensation, command G49 in the same way as the normal tool length compensation.

With tool length compensation for multiple axis synchronization control, the movement is performed for the amount of tool length compensation in G43, G44, G49 commanded blocks. With tool length compensation for multiple axis synchronization control, the parameter setting "#1247 set19/bit0" is disabled. However, if the target axis for tool length compensation is not the target axis for multiple axis synchronization control, the parameter setting "#1247 set19/bit0" is reflected.

If H0 is commanded in G43, G44 commanded blocks, "0" is applied to the compensation amount value of the axis where "#2675 tcmp_top" is set to "0", and the tool compensation amount value of the tool compensation No. corresponding to the parameter setting value is applied to the axis where "#2675 tcmp_top" is set to other than "0".

<Example of operation>

The following shows the example conditions for the system with XYZUVW axis configuration.

- Set the parameter so that Z axis is the master, U and V axes are the slave axes.
- The machine position when the multiple axis synchronization control is started is the same on master and slave axes.
- Mount the tools with different length.

Other settings are as below.

- Setting values of the parameter "#2675 tcmp_top"
 - Z axis: 0
 - U axis: 20
 - V axis: 33
- Setting values of tool compensation amount
 - Compensation No. 5: 20.0 mm
 - Compensation No. 25: 27.0 mm
 - Compensation No. 38: 35.0 mm
- Tool length compensation command G44 H5

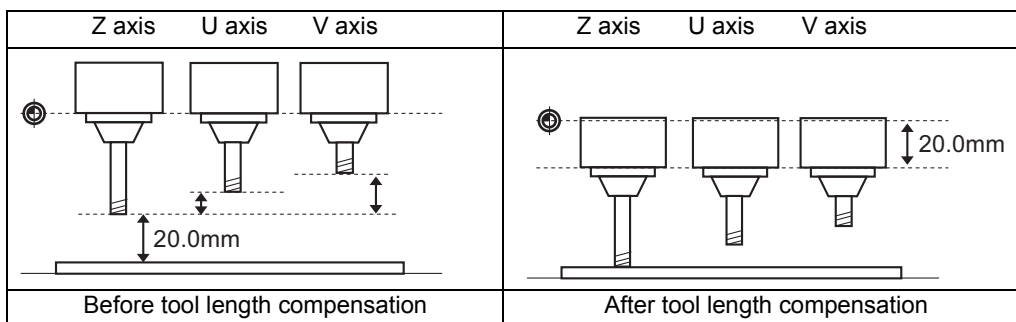
12 Tool Compensation Functions

(1) When "#1255 set27/bit5" is set to "0"

As the same tool length compensation amount as the master axis is applied to the slave axis, 20.0 mm compensation of the compensation No. 5 is applied to Z, U and V axes.

Tool compensation amount setting screen

1	0.000	15	0.000	29	0.000
2	0.000	16	0.000	30	0.000
3	0.000	17	0.000	31	0.000
4	0.000	18	0.000	32	0.000
5	20.000	19	0.000	33	0.000
6	0.000	20	0.000	34	0.000
7	0.000	21	0.000	35	0.000
8	0.000	22	0.000	36	0.000
9	0.000	23	0.000	37	0.000
10	0.000	24	0.000	38	35.000
11	0.000	25	27.000	39	0.000
12	0.000	26	0.000	40	0.000
13	0.000	27	0.000	41	0.000
14	0.000	28	0.000	42	0.000



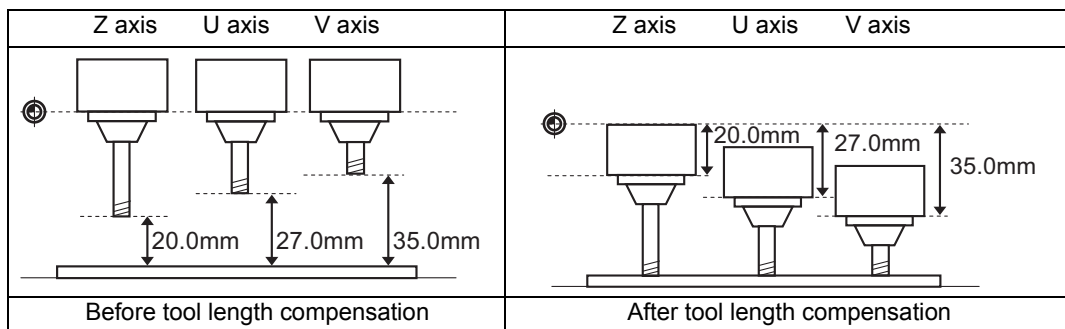
12 Tool Compensation Functions

(2) When "#1255 set27/bit5" is set to "1"

Independent tool length compensation amount is applied to master and slave axes. For the master and slave axes, the tool compensation amount of the tool compensation No. where "5" is added to the No. specified with the parameter "#2675 tcmp_top" is applied; therefore, the tool compensation amount No. 5 is applied to Z axis, No. 25 is applied to U axis and No. 38 is applied to V axis.

Tool compensation amount setting screen

Tool offset					
1	0.000	15	0.000	29	0.000
2	0.000	16	0.000	30	0.000
3	0.000	17	0.000	31	0.000
4	0.000	18	0.000	32	0.000
5	20.000	19	0.000	33	0.000
6	0.000	20	0.000	34	0.000
7	0.000	21	0.000	35	0.000
8	0.000	22	0.000	36	0.000
9	0.000	23	0.000	37	0.000
10	0.000	24	0.000	38	35.000
11	0.000	25	27.000	39	0.000
12	0.000	26	0.000	40	0.000
13	0.000	27	0.000	41	0.000
14	0.000	28	0.000	42	0.000



12.3 Tool Radius Compensation; G38,G39/G40/G41,G42



Function and purpose

This function compensates the radius of the tool. The compensation can be done in the random vector direction by the radius amount of the tool selected with the G command (G38 to G42) and the D command. When using tool nose radius compensation, refer to "12.4 Tool Nose Radius Compensation (for Machining Center System)".



Command format

G40 X__Y__;	Tool radius compensation cancel
-------------	---------------------------------

G41 X__Y__ D__;	Tool radius compensation (Left)
-----------------	---------------------------------

G42 X__Y__ D__;	Tool radius compensation (Right)
-----------------	----------------------------------

G38 I__J__;	Change or hold of compensation vector (Can be commanded only during the radius compensation mode.)
-------------	--

G39 X__Y__;	Corner changeover (Can be commanded only during the radius compensation mode.)
-------------	--



Detailed description

The number of sets for the compensation differ according to machine specification. (The No. of sets is the total of the tool length offset, tool position offset and tool radius compensation sets.)

The H command is ignored during the tool radius compensation, and only the D command is valid.

The compensation will be executed within the plane designated with the plane selection G code or axis address 2 axis, and axes other than those included in the designated plane and the axes parallel to the designated plane will not be affected. Refer to the section on plane selection for details on selecting the plane with the G code.

12.3.1 Tool Radius Compensation Operation



Detailed description

Tool radius compensation cancel mode

The tool radius compensation cancel mode is established by any of the following conditions.

- (1) After the power has been switched on
- (2) After the reset button on the setting and display unit has been pressed
- (3) After the M02 or M30 command with reset function has been executed
- (4) After the compensation cancel command (G40) is issued

The compensation vectors are zero in the compensation cancel mode, and the tool nose point path coincides with the programmed path.

Programs including tool radius compensation must be terminated in the compensation cancel mode.

Tool radius compensation start (startup)

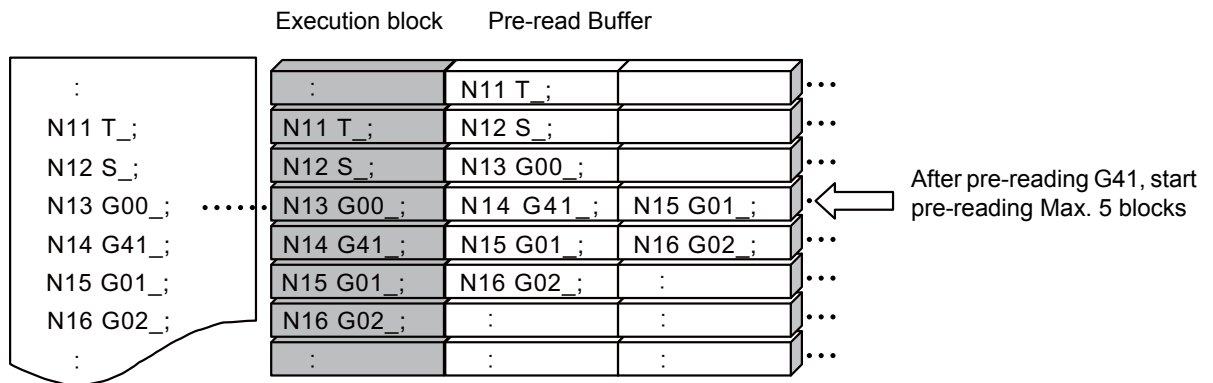
Tool radius compensation starts when all the following conditions are met in the compensation cancel mode.

- (1) The movement command is issued after G41 or G42.
- (2) The tool radius compensation offset No. is $0 < D \leq \text{max. offset No.}$
- (3) The movement command of positioning (G00) or linear interpolation (G01) is issued.

Whether in continuous or single block operation, compensation always starts after reading three blocks, or if the three blocks do not contain any movement command, up to five continuous blocks will be pre-read.

In compensation mode, too, up to 5 blocks are pre-read and the compensation is arithmetically processed.

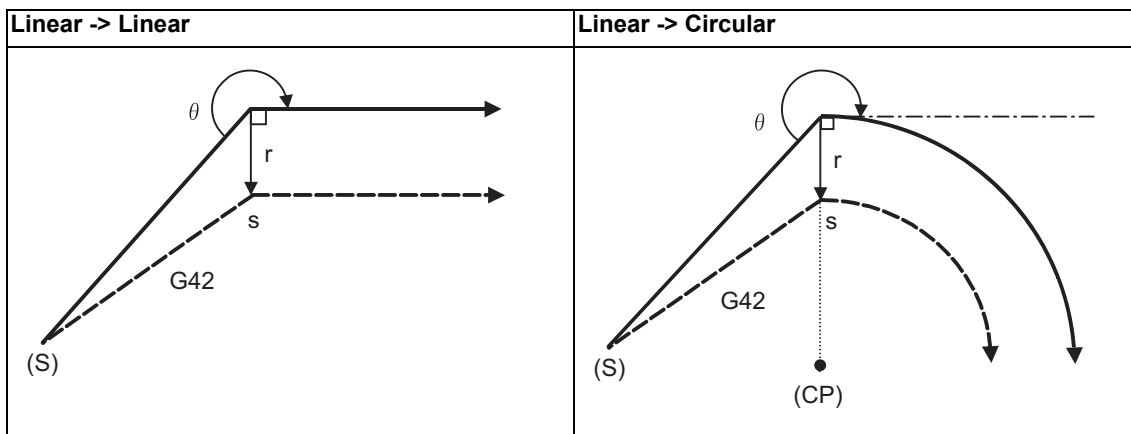
[Control state diagram]



There are two ways of starting the compensation operation: type A and type B. The type depends on the setting of the parameter "#8157 Radius comp type B". This type is used in common with the compensation cancel type.

Start operation for tool radius compensation

(1) Machining an inside corner



(S) Start point

r : Compensation amount

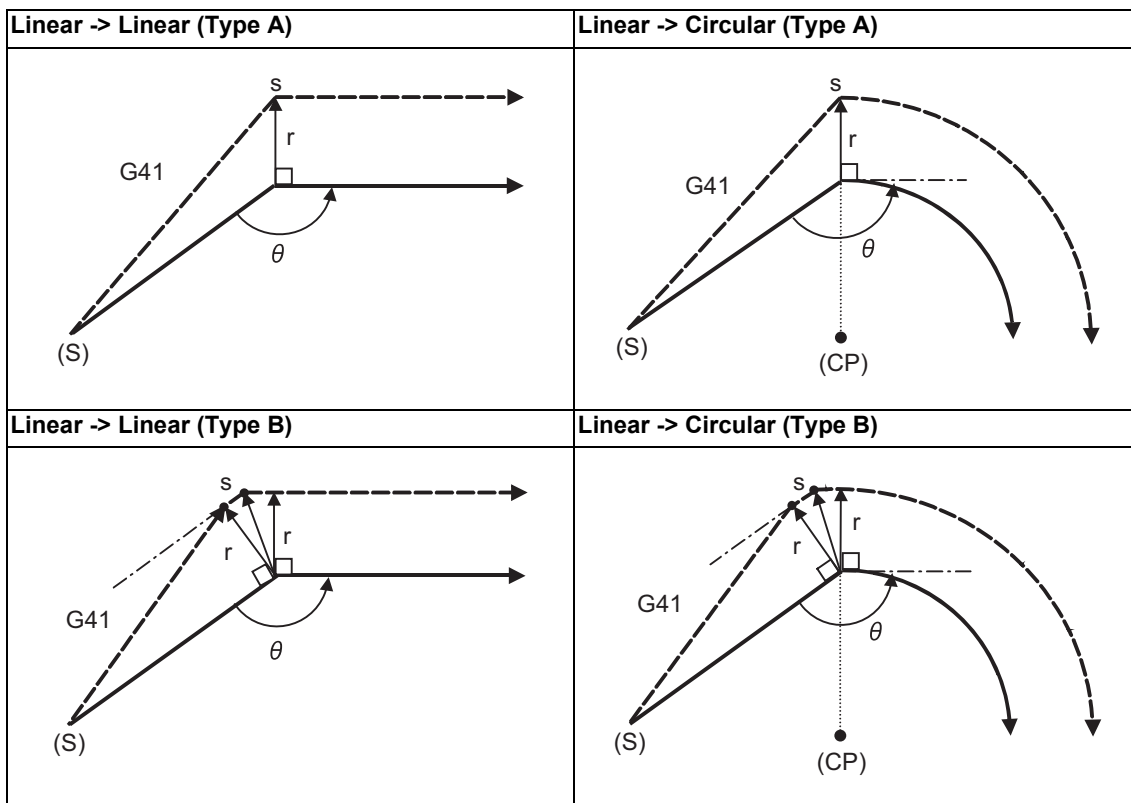
— Program path

(CP) Center of circular

s: Stop point with single block

---- Tool center path

(2) Machining an outside corner (obtuse angle) [$90^\circ \leq \theta < 180^\circ$]



(S) Start point

r : Compensation amount

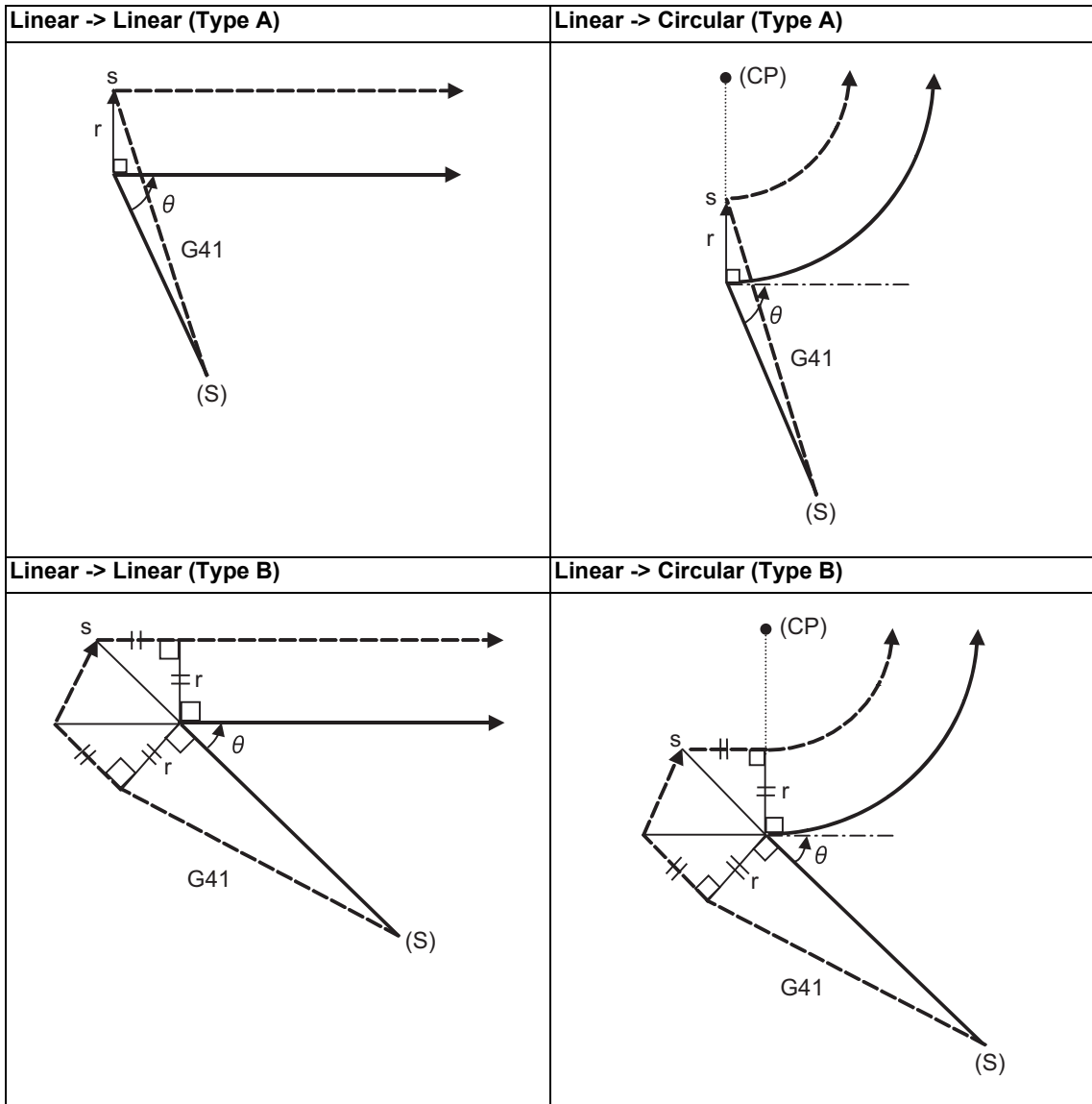
— Program path

(CP) Center of circular

s: Stop point with single block

---- Tool center path

(3) Machining an outside corner (acute angle) [$\theta < 90^\circ$]



(S) Start point

r : Compensation amount

— Program path

(CP) Center of circular

s: Stop point with single block

---- Tool center path

Note

- If there is no axis movement command in the same block as G41 or G42, compensation is performed perpendicularly to the next block's direction.

Operation in compensation mode

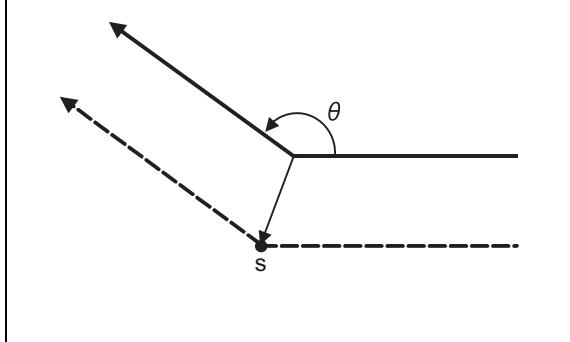
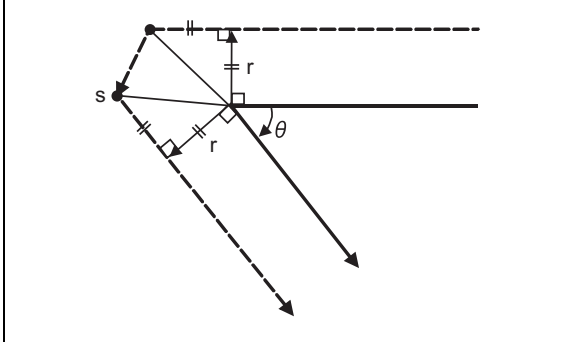
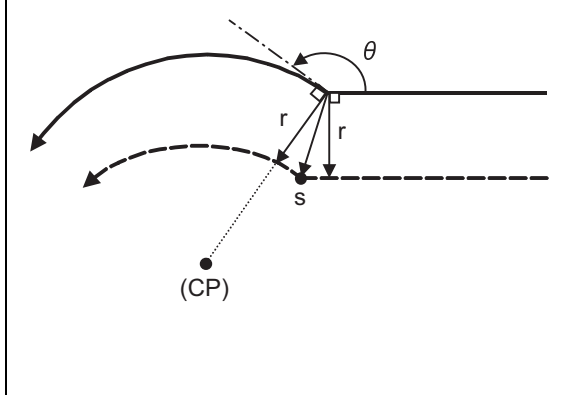
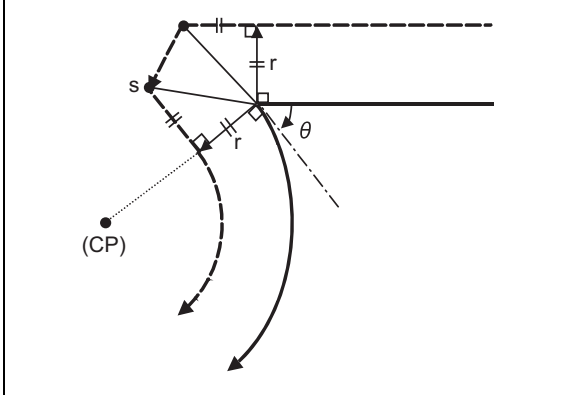
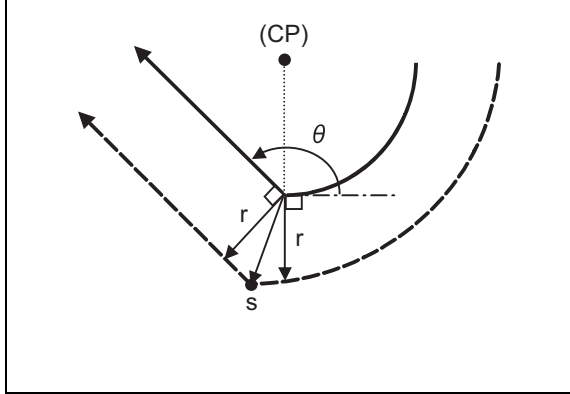
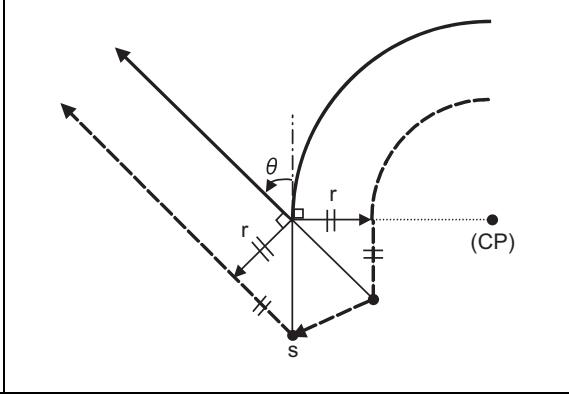
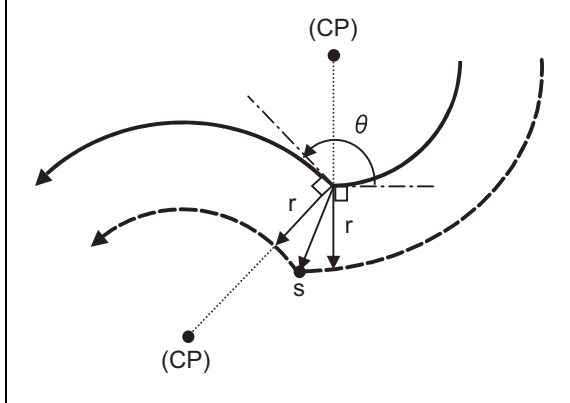
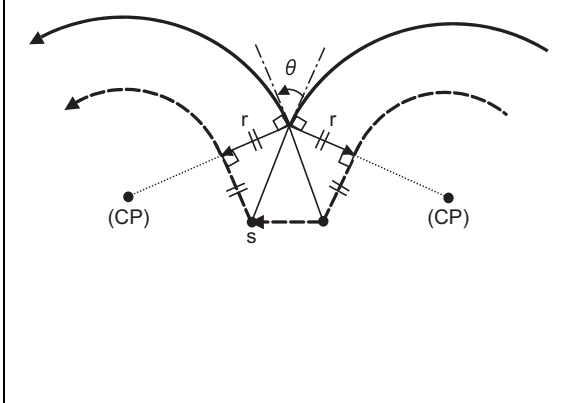
Calculate the tool center path from the linear line/circular arc to perform compensation to the program path (G00, G01, G02, G03).

Even if the same compensation command (G41, G42) is issued in the compensation mode, the command will be ignored.

When 4 or more blocks without movement command are continuously specified in the compensation mode, overcutting or undercutting will occur.

When the M00 command has been issued during tool radius compensation, pre-reading is prohibited.

(1) Machining an outside corner

<p>Linear -> Linear ($90^\circ \leq \theta < 180^\circ$)</p> 	<p>Linear -> Linear ($0^\circ < \theta < 90^\circ$)</p> 
<p>Linear -> Circular ($90^\circ \leq \theta < 180^\circ$)</p> 	<p>Linear -> Circular ($0^\circ < \theta < 90^\circ$)</p> 
<p>Circular -> Linear ($90^\circ \leq \theta < 180^\circ$)</p> 	<p>Circular -> Linear ($0^\circ < \theta < 90^\circ$)</p> 
<p>Circular -> Circular ($90^\circ \leq \theta < 180^\circ$)</p> 	<p>Circular -> Circular ($0^\circ < \theta < 90^\circ$)</p> 

(CP) Center of circular
 r : Compensation amount
 — Program path

s : Single block stop point
 - - - - Tool center path

(2) Machining an inside corner

<p>Linear -> Linear (Obtuse angle)</p>	<p>Linear -> Linear (Acute angle)</p>
<p>Linear -> Circular (Obtuse angle)</p>	<p>Linear -> Circular (Acute angle)</p>
<p>Circular -> Linear (Obtuse angle)</p>	<p>Circular -> Linear (Acute angle)</p>
<p>Circular -> Circular (Obtuse angle)</p>	<p>Circular -> Circular (Acute angle)</p>

(CP) Center of circular

r : Compensation amount

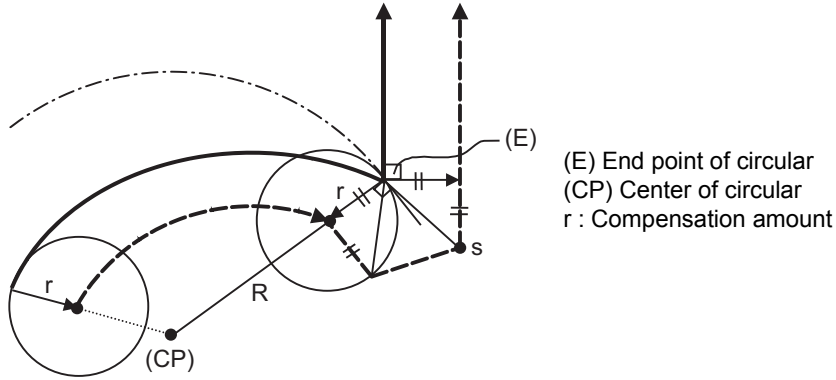
—— Program path

s : Single block stop point

----- Tool center path

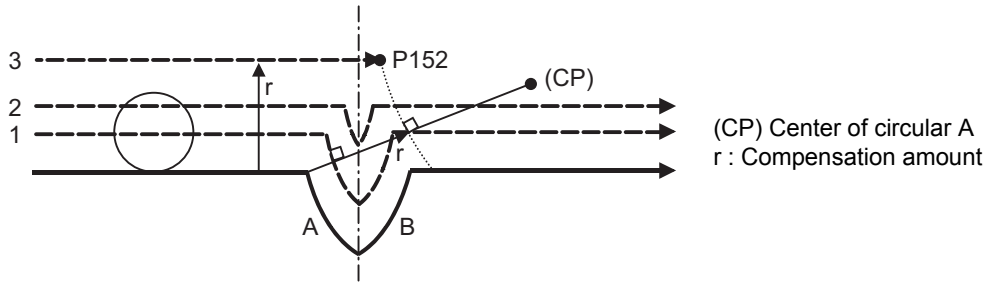
(3) When the circular end point is not on the circular

Spiral circular command ... The area from the arc start point to the end point is interpolated as a spiral arc.
 Normal circular command If the error after compensation is within the parameter value ("#1084 RadErr"), it is interpolated as a spiral arc.



(4) When the inner intersection point does not exist

In cases like the figure below, the intersection point of circulars A and B may not exist depending on the compensation amount.
 In such cases, program error (P152) appears and the tool stops at the end point of the previous block.
 In the pattern 1 and 2 in this figure, machining is possible because compensation amount r is small.
 In pattern 3, compensation r is so large that an intersection does not exist and program error (P152) will occur.



—— Program path

---- Tool center path

Tool radius compensation cancel

Tool radius compensation cancel If either of the following conditions is met in the tool radius compensation mode, the compensation will be canceled.

However, there must be any movement command except a circular command.

If the compensation is canceled by a circular command, program error (P151) will occur.

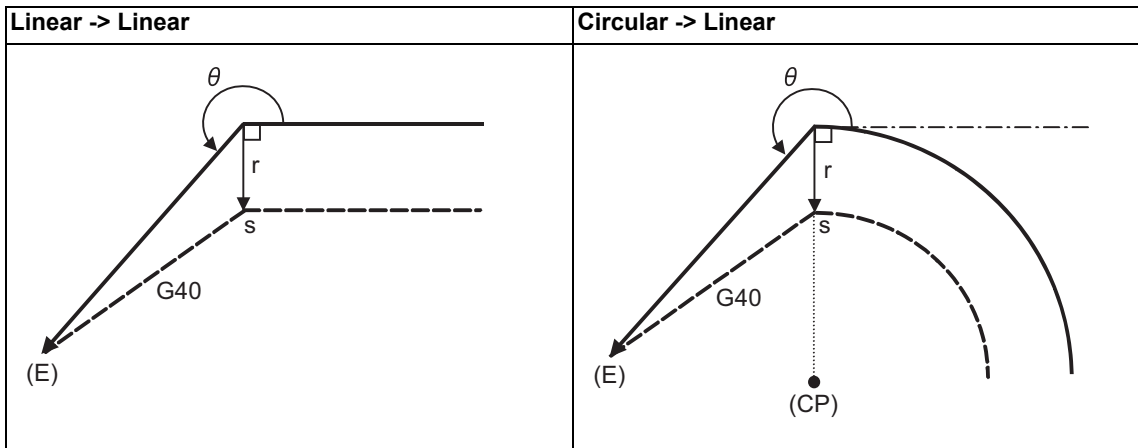
(1)The G40 command has been executed.

(2)Executed the compensation No.D00.

The cancel mode is established once the compensation cancel command has been read, 5-block pre-reading is suspended and 1-block pre-reading will be operated.

Tool radius compensation cancel operation

(1) Machining an inside corner



(E) End point

r : Compensation amount

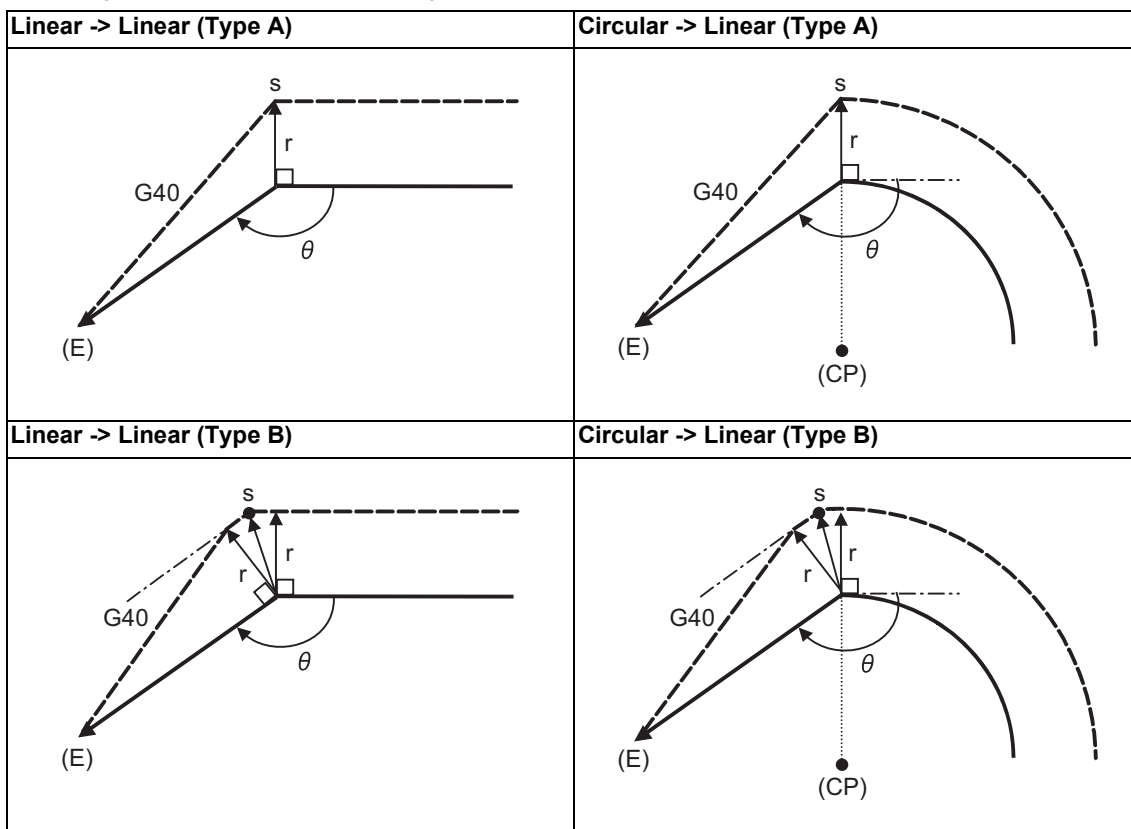
— Program path

(CP) Center of circular

s : Single block stop point

---- Tool center path

(2) Machining an outside corner (obtuse angle) [$90^\circ \leq \theta < 180^\circ$]



(E) End point

r : Compensation amount

—— Program path

(CP) Center of circular

s : Single block stop point

----- Tool center path

(3) Machining an outside corner (acute angle) [$\theta < 90^\circ$]

Linear -> Linear (Type A)	Circular -> Linear (Type A)
Linear -> Linear (Type B)	Circular -> Linear (Type B)

(E) End point
 r : Compensation amount
 — Program path

(CP) Center of circular
 s : Single block stop point
 - - - Tool center path

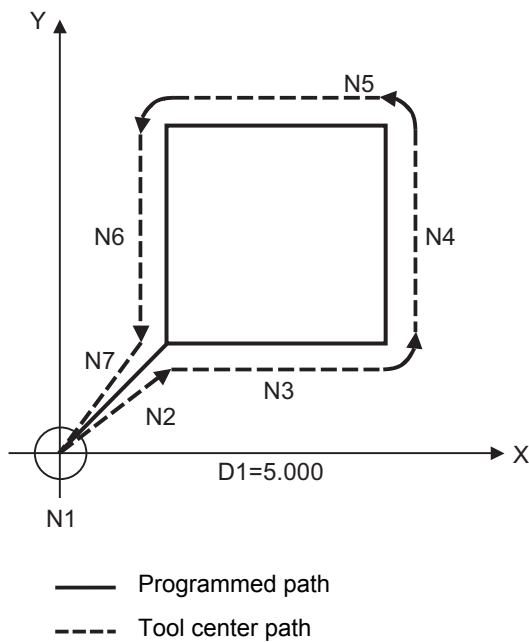
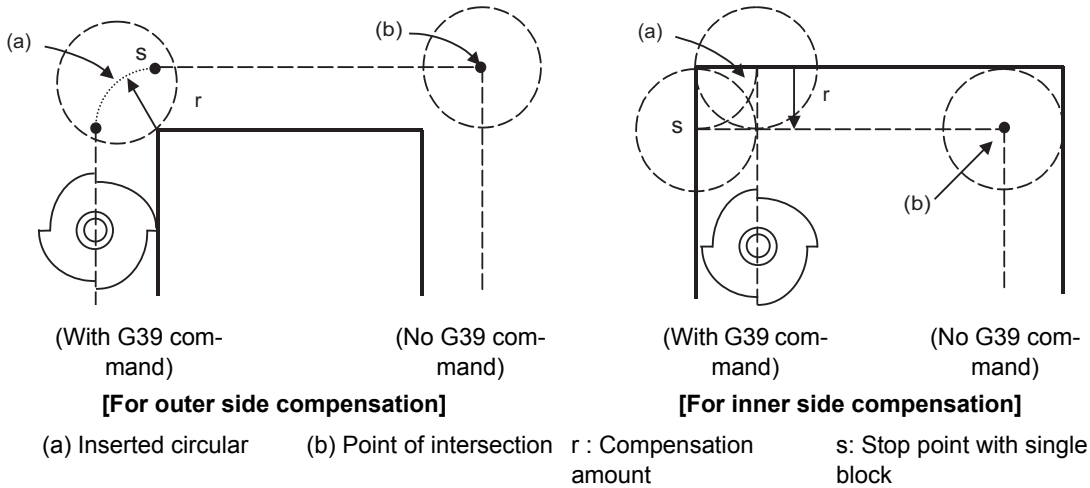
12.3.2 Other Commands and Operations during Tool Radius Compensation



Detailed description

Insertion of corner arc

An arc that uses the compensation amount as the radius is inserted without calculating the point of intersection at the workpiece corner when G39 (corner arc) is commanded.



```

N1 G28 X0 Y0 ;
N2 G91 G01 G42 X20. Y20. D1 F100 ;
N3 G39 X40. ;
N4 G39 Y40. ;
N5 G39 X-40. ;
N6 Y-40. ;
N7 G40 X-20. Y-20. ;
N8 M02 ;
    
```

Changing and holding of compensation vector

The compensation vector can be changed or held during tool radius compensation by using the G38 command.

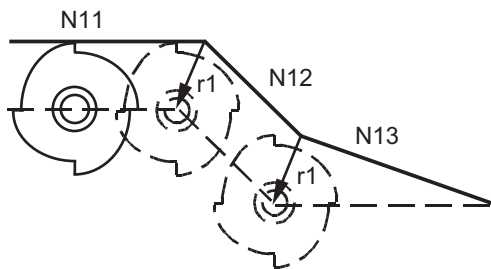
(1) Holding of vector

When G38 is commanded in a block having a movement command, the point of intersection will not be calculated at the program end point, and instead the vector of the previous block will be held.

G38 Xx Yy;

This can be used for pick feed, etc.

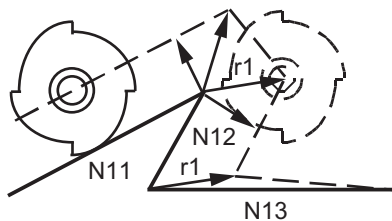
[Holding the inside compensation vector]



```
N11 G01 Xx11 ;
N12 G38 Xx12 Yy12 ;
N13 G40 Xx13 ;
```

r1: Vector at N11-N12 block intersection calculation

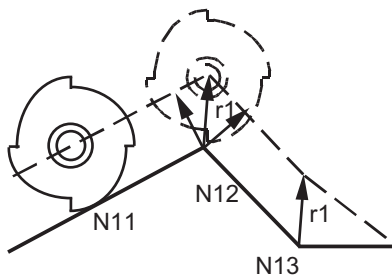
[Holding the outside compensation acute angle]



```
N11 G01 Xx11 Yy11 ;
N12 G38 Xx12 Yy12 ;
N13 G40 Xx13 ;
```

r1: Vector at N11-N12 block intersection calculation

[Holding the outside compensation obtuse angle]



```
N11 G01 Xx11 Yy11 ;
N12 G38 Xx12 Yy12 ;
N13 G40 Xx13 ;
```

r1: Vector at N11-N12 block intersection calculation

— Programmed path

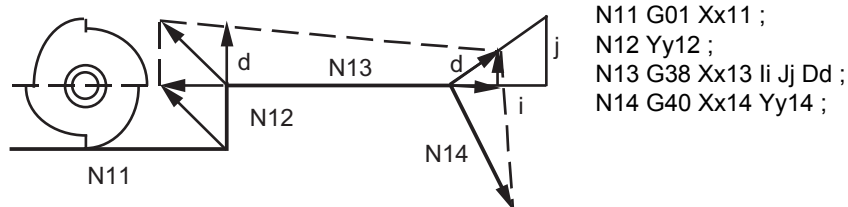
---- Tool center path

(2) Changing of vector

A new compensation vector direction can be commanded with I, J and K, and a new compensation amount with D.

(These can be commanded in the same block as the movement command.)

G38 Ii Jj Dd ; (I, J and K will differ according to the selected plane.)



N11 G01 Xx11 ;
 N12 Yy12 ;
 N13 G38 Xx13 Ii Jj Dd ;
 N14 G40 Xx14 Yy14 ;

— Programmed path

- - - - Tool center path

The compensation amount “d” vector is created in the commanded i and j vector direction.

Note

- If G38 is commanded in the same block as the circular block (G02/G03) I and J commands, I and J will be handled as the G38 vector, and an error will occur.

Changing the compensation direction during tool radius compensation

The compensation direction is determined by the tool radius compensation commands (G41, G42) and compensation amount sign.

G Code	Compensation amount sign +	Compensation amount sign -
G41	Left-side compensation	Right-side compensation
G42	Right-side compensation	Left-side compensation

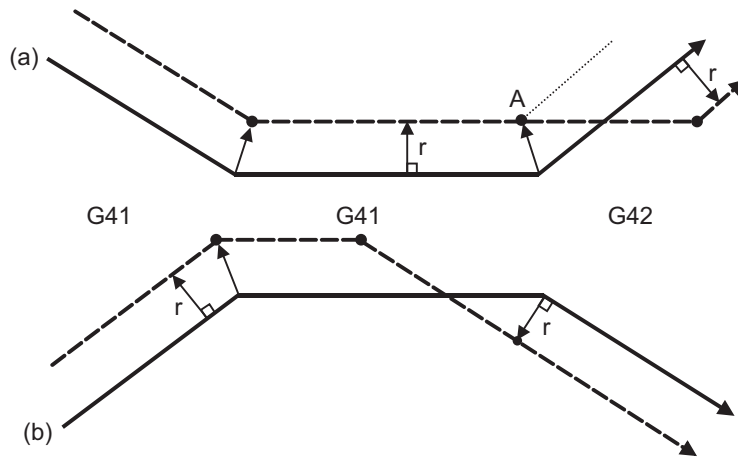
The compensation direction can be changed by changing the compensation command during the compensation mode without canceling the mode.

However, it is impossible to change the direction in the compensation start block and the next block.

(1) Linear -> Linear

(a) When there is an intersection (A) at the change of compensation direction

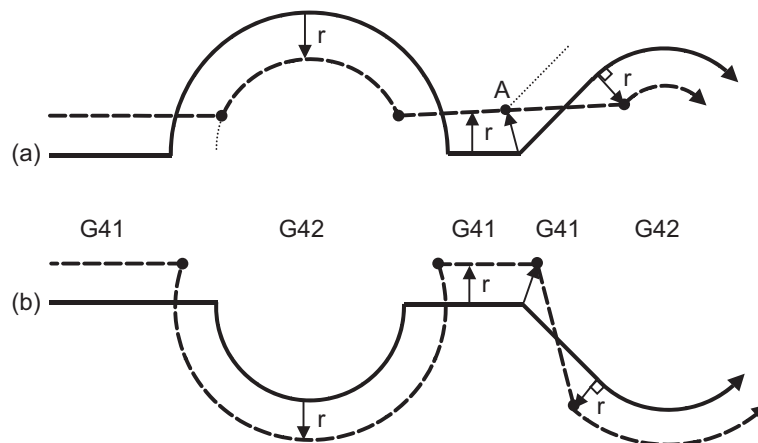
(b) When there is no intersection at the change of compensation direction



(2) Linear <-> Circular

(a) When there is a point of intersection (A) when the compensation direction is changed.

(b) When there is no point of intersection when the compensation direction is changed.



— Program path

- - - Tool center path

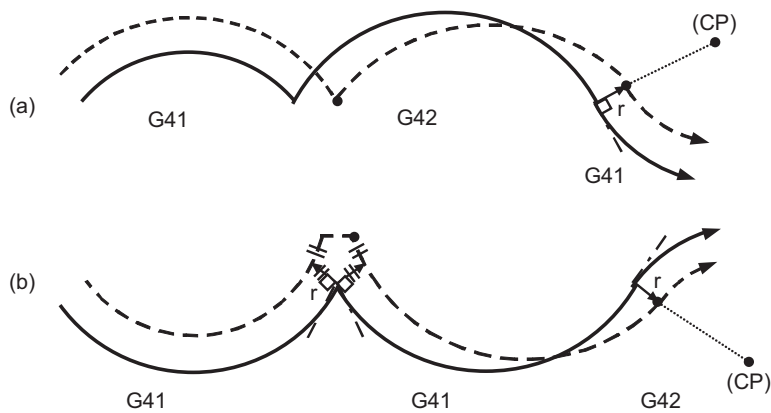
12 Tool Compensation Functions

(3) Circular -> Circular

(a) When there is an intersection at the change of compensation direction

(b) When there is no intersection at the change of compensation direction

(CP) Center of circular

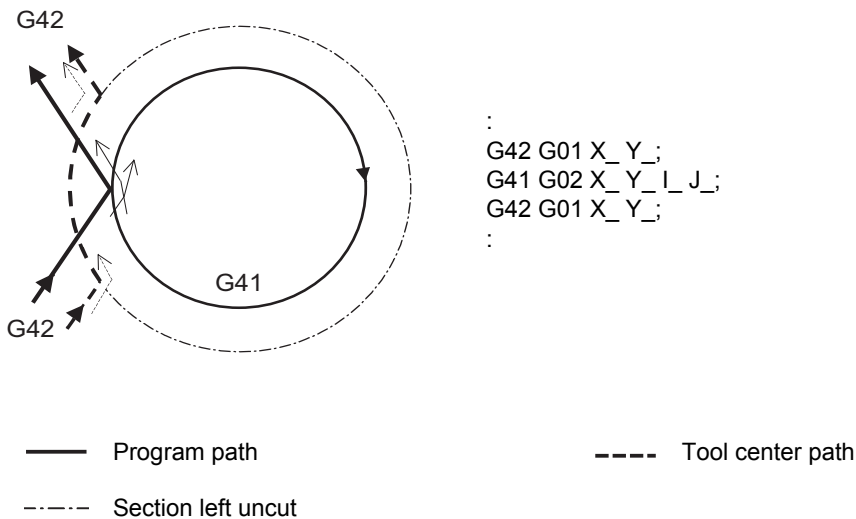


(4) Linear return



(5) When the compensation direction is switched using G41/G42, it is possible that the arc may exceed 360°.

If the arc exceeds 360°, compensation will be performed as shown in the figure and uncut section will be left.



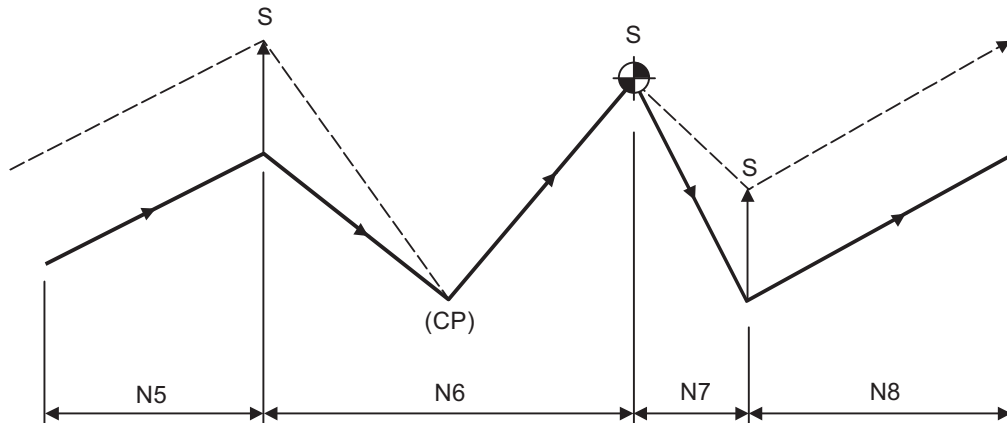
Command for eliminating compensation vectors temporarily

When the following command is issued in the compensation mode, the compensation vectors are temporarily eliminated and then, compensation mode will automatically return.

In this case, the compensation is not canceled, and the tool goes directly from the intersection point vector to the point without vectors, in other words, to the programmed command point. When returning to the compensation mode, it goes directly to the intersection point.

(1) Reference position return command

Temporarily no compensation vectors at intermediate point. (Reference position when there is no intermediate point).



(G41) : (CP) Intermediate point
 N5 G91 G01 X60. Y30. ;
 N6 G28 X50. Y-40. ; ← Temporarily no compensation vectors at intermediate point.
 N7 X30. Y-60. ; (Reference position when there is no intermediate point)
 N8 X70. Y40. ;
 :

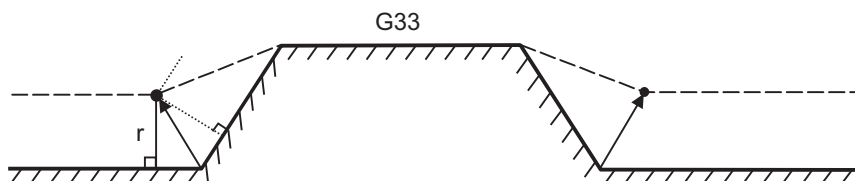
(2) The compensation vector will be eliminated temporarily with the G53 command (Basic machine coordinate system selection).

<Note>

- ♦The compensation vectors do not change with the coordinate system setting (G92) command.

(3) G33 thread cutting command

Tool radius compensation does not apply to the G33 block.



Blocks without movement

The following blocks are known as blocks without movement.

M03 ;	M command
S12 ;	S command
T45 ;	T command
G04X500 ;	Dwell
G22 X200. Y150. Z100 ;	Machining prohibited region setting
G10 L10; P01 R50;	Compensation amount setting
G92 X600. Y400. Z500.	Coordinate system setting
(G17) Z40.	Movement outside the compensation plane
G90 ;	G code only
G91 X0;	Movement amount 0

M00, M01, M02 and M30 are handled as pre-read inhibit M codes.

(1) When command is assigned at start of the compensation

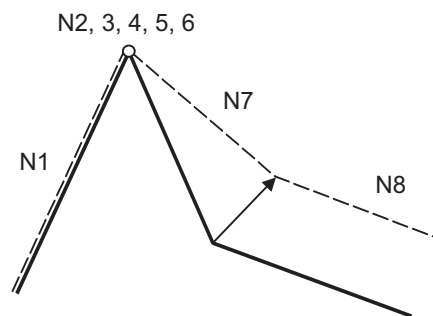
Compensation vector cannot be created when there are four or more successive blocks without movement, or when pre-reading prohibiting M command is issued.

```

N1 X30.Y60 ;
N2 G41 D10 ;
N3 G04 X1000 ;
N4 F100 ;
N5 S500 ;
N6 M3 ;
N7 X20.Y-50. ;
N8 X50.Y-20. ;

```

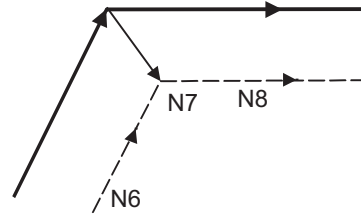
} Block without movement



(2) When command is assigned in the compensation mode

Compensation vector will be created as normal when there are not four or more successive blocks without movement, or when pre-read prohibiting M command is not issued.

```
N6 G91 X100. Y200. ;
N7 G04 X1000;      ... Block without movement
N8 X200. ;
```

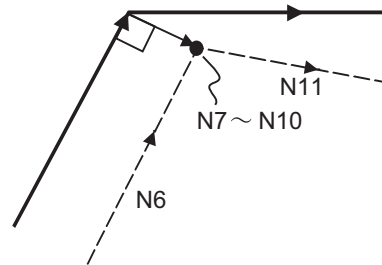


Block N7 is executed at N7 in the figure.

Compensation vector will be created perpendicularly to the end point of the previous block when there are four or more successive blocks without movement, or when pre-read prohibiting M command is issued. In this case, a cut may occur.

```
N6 X100. Y200. ;
N7 G04 X1000 ;
N8 F100 ;
N9 S500 ;
N10 M4 ;
N11 X100. ;
```

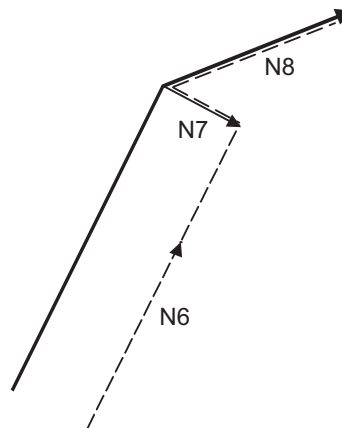
} Block without movement



(3) When commanded together with compensation cancel

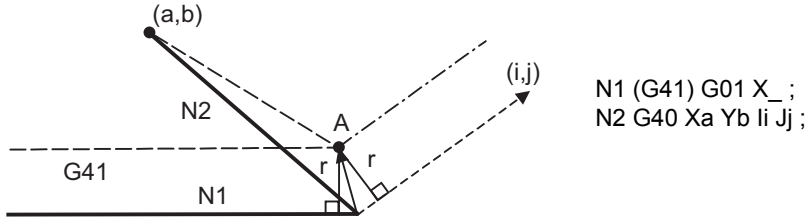
Only the compensation vectors are canceled when a block without movement is commanded together with the G40 command.

```
N6 X100. Y200. ;
N7 G40 M5 ;
N8 X100. Y50. ;
```



When I, J, K are commanded in G40

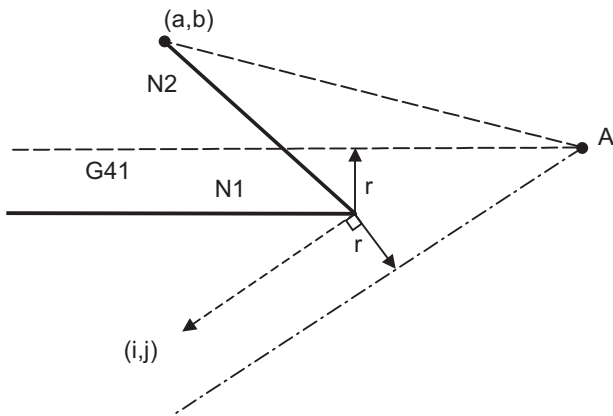
(1) If the final movement command block in the four blocks before the G40 block is the G41 or G42 mode, it will be assumed that the movement is commanded in the vector I, J or K direction from the end point of the final movement command. After interpolating between the hypothetical tool center path and point of intersection, it will be canceled. The compensation direction will not change.



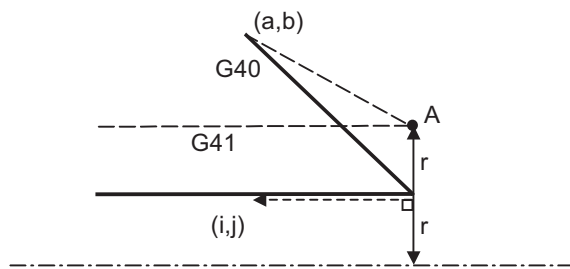
```
N1 (G41) G01 X_ ;
N2 G40 Xa Yb Ii Jj ;
```

In this case, the point of intersection will always be obtained, regardless of the compensation direction, even when the commanded vector is incorrect as shown below.

[When the I and J symbols in the above program example are incorrect]

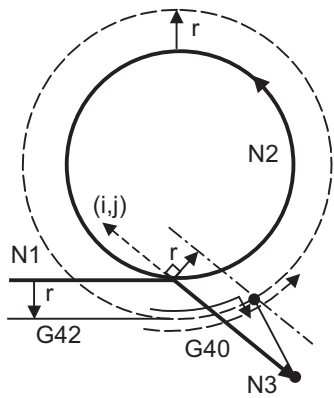


If the compensation vector obtained via a point of intersection calculation is extremely large, a perpendicular vector will be created in the block before G40.



- Program path
- - - Tool center path
- · - · Hypothetical tool center path

(2) If the arc is 360° or more due to the details of I, J and K at G40 after the arc command, an uncut section will occur.



```
N1 (G42,G91) G01 X200. ;
N2 G03 J150.;
N3 G40 G01 X150. Y-150. I-100. J100. ;
```

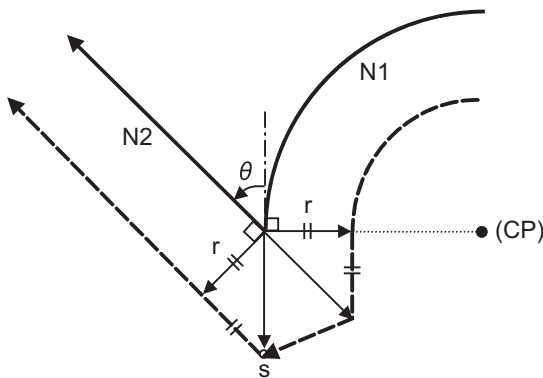
- Programmed path
- - - Tool center path
- · · Section left uncut

Corner movement

When a multiple number of compensation vectors are created at the joints between movement command blocks, the tool will move in a straight line between these vectors. This action is called corner movement.

When the vectors do not coincide, the tool moves in order to machine the corner although this movement is part and parcel of the joint block.

Consequently, operation in the single block mode will execute the previous block + corner movement as a single block and the remaining joining movement + following block will be executed as a single block in the following operation.



(CP) Center of circular
 r: Compensation amount
 s: Single block stop point

- Programmed path
- - - Tool center path

12.3.3 G41/G42 Commands and I, J, K Designation



Function and purpose

The compensation direction can be intentionally changed by issuing the G41/G42 command and I, J, K in the same block.



Command format

G17 (X-Y plane) G41/G42 X__ Y__ I__ J__ ;

G18 (Z-X plane) G41/G42 X__ Z__ I__ K__ ;

G19 (Y-Z plane) G41/G42 Y__ Z__ J__ K__ ;

Assign a linear command (G00, G01) in a movement mode.



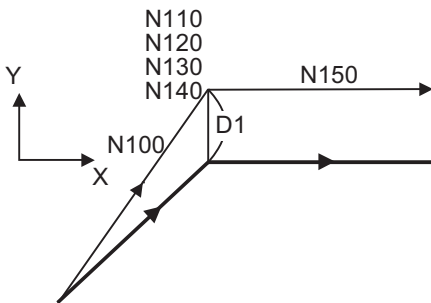
Detailed description

I, J type vectors (G17 X-Y plane selection)

This section describes the new I, J type vectors (G17 plane) created by this command. (Similar descriptions apply to vector K, I for the G18 plane and to J, K for the G19 plane.)

As shown in the following figures, I, J type vectors create compensation vectors which are perpendicular to the direction designated by I, J and equivalent to the compensation amount, without the intersection point calculation of the programmed path. The I, J vectors can be commanded even in the mode (G41/G42 mode in the block before) and even at the compensation start (G40 mode in the block before).

(1) When I, J is commanded at compensation start

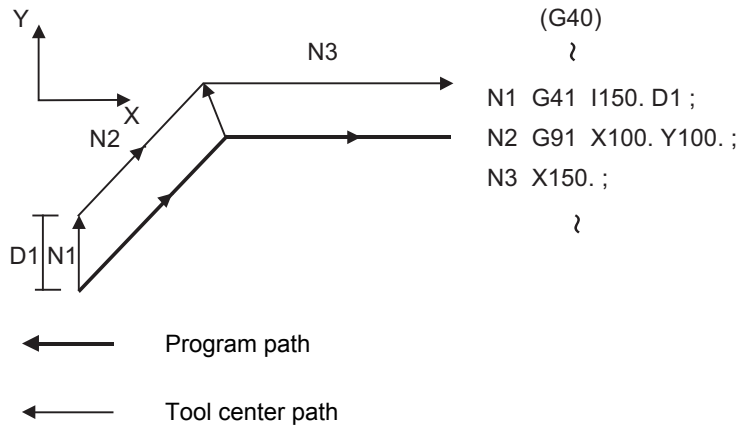


```

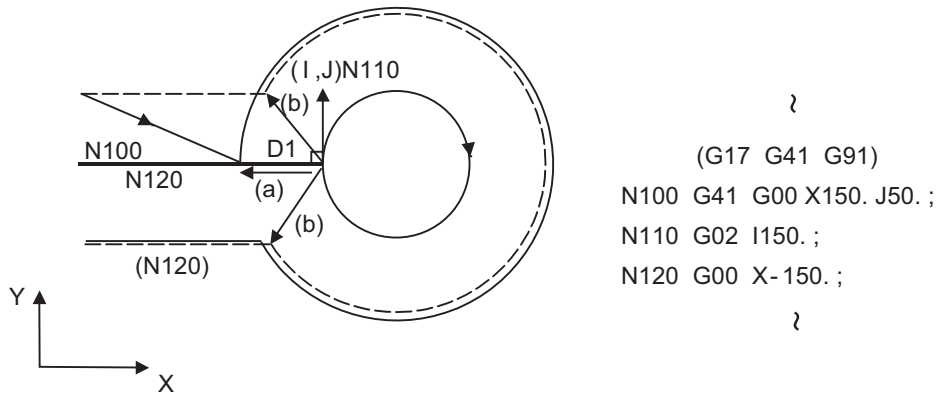
(G40)
{
N100 G91 G41 X100. Y100. I150. D1 ;
N110 G04 X1000 ;
N120 G01 F1000 ;
N130 S500 ;
N140 M03 ;
N150 X150. ;
}
    
```

- ← Program path
- ← Tool center path

(2) When there are no movement commands at the compensation start.



(3) When I, J has been commanded in the G41/42 mode (G17 plane)



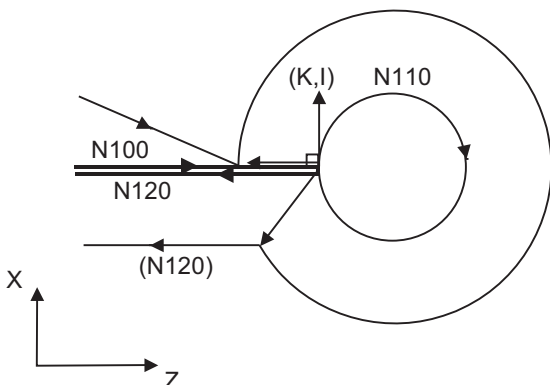
(a) I, J type vector (b) Intersection point calculation type vector

- Program path
- Tool center path
- Path after intersection point calculation

12 Tool Compensation Functions

(Reference)

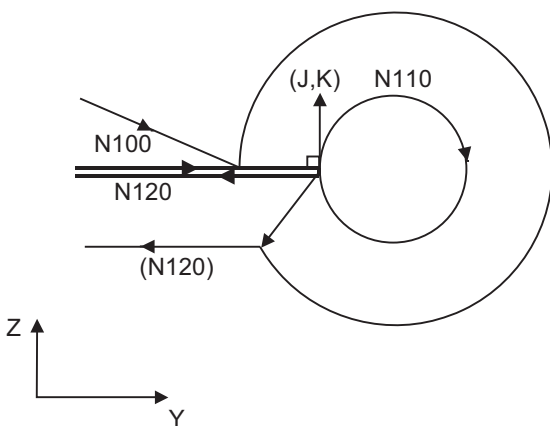
(a) G18 plane



```

}
(G18 G41 G91)
N100 G41 G00 Z150. I50. ;
N110 G02 K50. ;
N120 G00 Z -150. ;
}
    
```

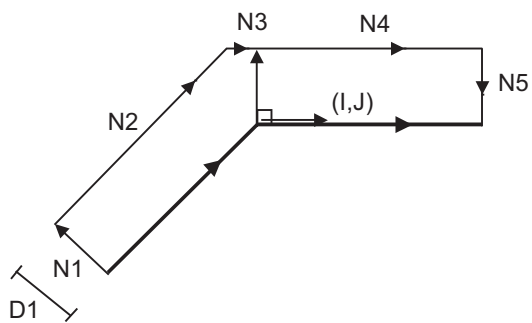
(b) G19 plane



```

}
(G19 G41 G91)
N100 G41 G00 Y150. K50. ;
N110 G02 J50. ;
N120 G00 Y -150. ;
}
    
```

(4) When I, J has been commanded in a block without movement



```

N1 G41 D1 G01 F1000 ;
N2 G91 X100. Y100. ;
N3 G41 I50. ;
N4 X150. ;
N5 G40 ;
    
```

Compensation vector direction

(1) In G41 mode

Direction produced by rotating the direction commanded by I,J by 90° to the left when looking at the zero point from the forward direction of the Z axis (3rd axis).

(Example 1) With I100.		(Example 2) With I-100.	
	(100, 0) IJ direction		(-100, 0) IJ direction
	Compensation vector direction		Compensation vector direction

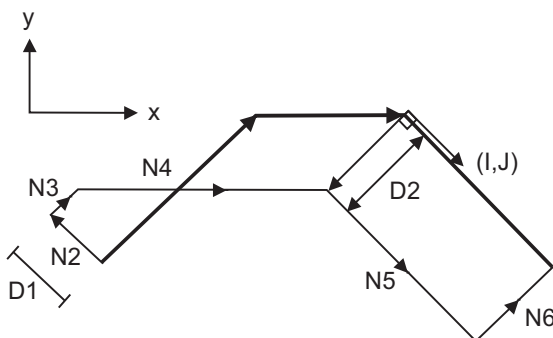
(2) In G42 mode

Direction produced by rotating the direction commanded by I, J by 90° to the right when looking at the zero point from the forward direction of the Z axis (3rd axis).

(Example 1) With I100.		(Example 2) With I-100.	
	(100, 0) IJ direction		(-100, 0) IJ direction
	Compensation vector direction		Compensation vector direction

Selection of compensation modal

G41 and G42 modals can be switched over at any time.



```

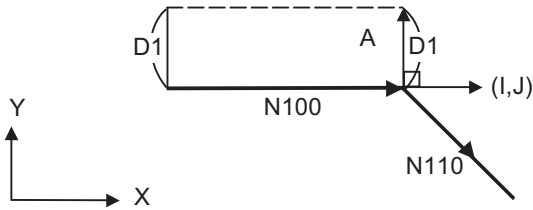
N1 G28 X0 Y0 ;
N2 G41 D1 F1000 ;
N3 G01 G91 X100. Y100. ;
N4 G42 X100. I100. J -100.
    D2 ;
N5 X100. Y-100. ;
N6 G40 ;
N7 M02 ;
%
```

Compensation amount for compensation vectors

The compensation amount is determined by the offset No. (modal) in a block with the IJ designation.

<Example 1>

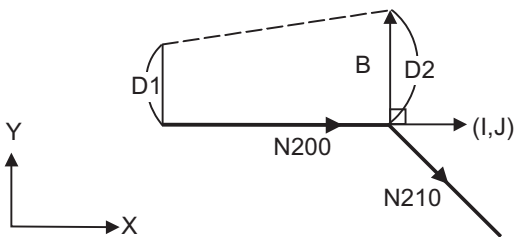
Vector A is the compensation amount registered in the compensation No. modal D1 of the N100 block.



```
(G41 D1 G91)
      }
N100 G41 X150. I50. ;
N110 X100. Y-100. ;
      }
```

<Example 2>

Vector B is the compensation amount registered in the compensation No. modal D2 of the N200 block.

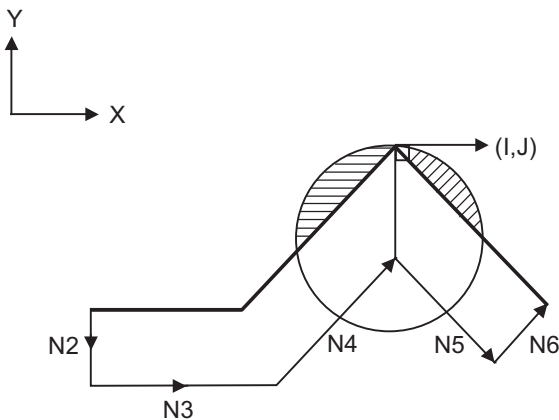


```
(G41 D1 G91)
      }
N200 G41 X150. I50. D2 ;
N210 X100. Y-100. ;
      }
```



Precautions

- (1) Issue the I, J type vector in a linear mode (G00, G01). If it is in an arc mode at the start of compensation, program error (P151) will occur.
When it is in the compensation mode as well as in the arc mode, I, J will be designated at the center of the circular.
- (2) When the I, J type vector is designated, it will not be deleted (Interference avoidance) even if there is interference. Consequently, overcutting may occur.
In the figure below, cutting will occur in the shaded section.



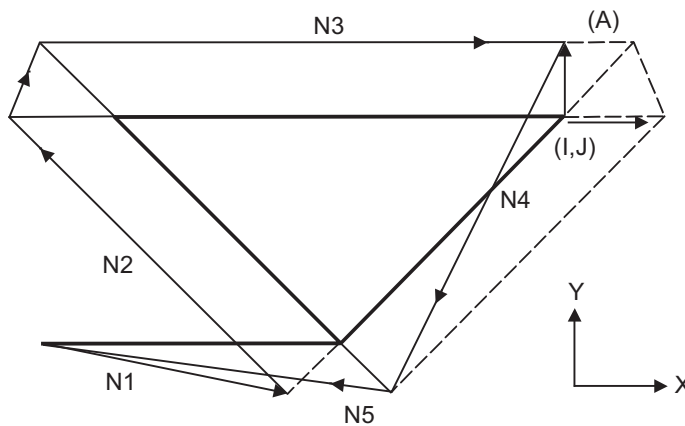
```
N1 G28 X0 Y0 ;
N2 G42 D1 F1000 ;
N3 G91 X100. ;
N4 G42 X100. Y100. I10. ;
N5 X100. Y-100. ;
N6 G40 ;
N7 M02 ;
```

(3) The vectors differ for the G38 I _J_ (K_) command and the G41/G42 I _J_(K_) command.

	G38	G41/G42
Example	⋮ (G41) ⋮ G38 G91 X100. I50. J50. ; ⋮	⋮ (G41) ⋮ G41 G91 X100. I50. J50. ; ⋮
	Vector in IJ direction having a compensation amount (a) size	Vector perpendicular in IJ direction and having a compensation amount (b) size

(4) Refer to the following table for the compensation methods depend on the presence or absence of G41/G42 command and I, K, (J) command.

G41/42	I, J (K)	Compensation methods
No	No	Intersection point calculation type vector
No	Yes	Intersection point calculation type vector
Yes	No	Intersection point calculation type vector
Yes	Yes	I, J, type vector No insertion block



```

N1 G91 G01 G41 X200. D1 F1000 ;
N2 X-150. Y150. ;
N3 G41 X300. I50. ;
N4 X-150. Y-150. ;
N5 G40 X-200. ;
    
```

During the I, J type vector compensation, the A insertion block will not exist.

12.3.4 Interrupts during Tool Radius Compensation



Detailed description

MDI interruption

Tool radius compensation is valid in any automatic operation mode - whether tape, memory or MDI mode. The figure below shows what happens by MDI interruption after stopping the block during tape or memory mode. S in the figure indicates the stop position with single block.

(1) Interrupt without movement (tool path does not change)

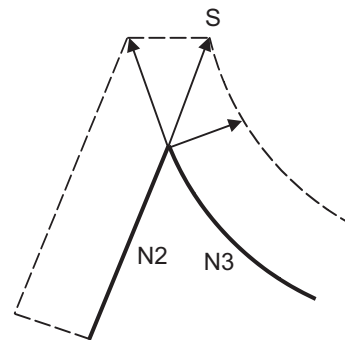
Automatic operation

N1 G41 D1;
N2 X20. Y50. ;

N3 G03 X40. Y-40. R70. ;

MDI interruption

<--- S1000 M3;



(2) Interrupt with movement

The compensation vectors are automatically re-calculated in the movement block after interrupt.

With linear interrupt

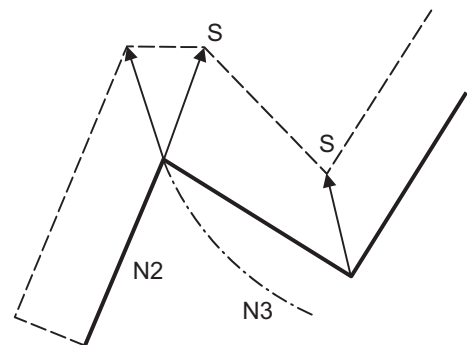
Automatic operation

N1 G41 D1;
N2 X20. Y50. ;

N3 G03 X40.Y-40. R70. ;

MDI interruption

<--- X50. Y-30. ;
X30. Y50. ;



With circular interruption

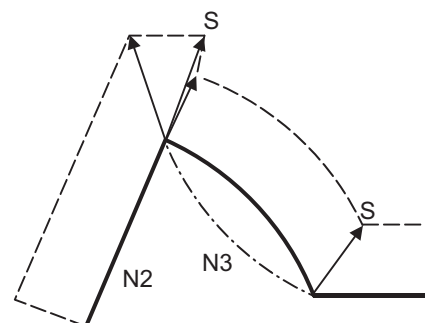
Automatic operation

N1 G41 D1;
N2 X20. Y50. ;

N3 G03 X40. Y-40. R70. ;

MDI interruption

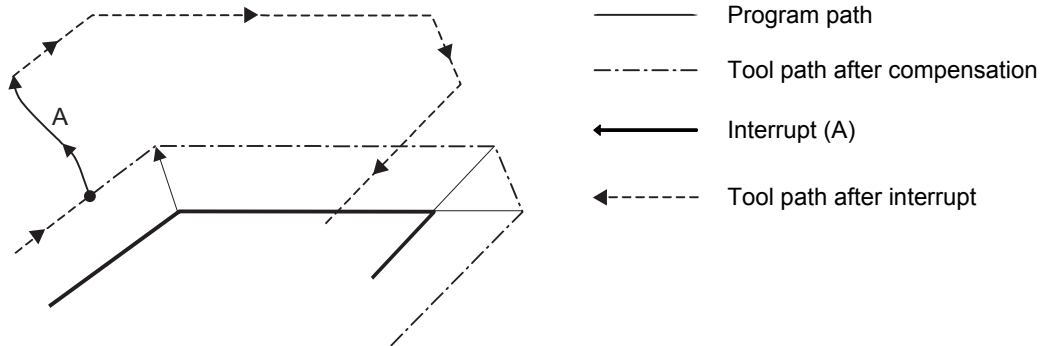
<--- G02 X40. Y-40. R70. ;
G01 X40. ;



Manual interruption

(1) Interrupt with manual absolute OFF.

The tool path will deviate from the compensated path by the interrupt amount.

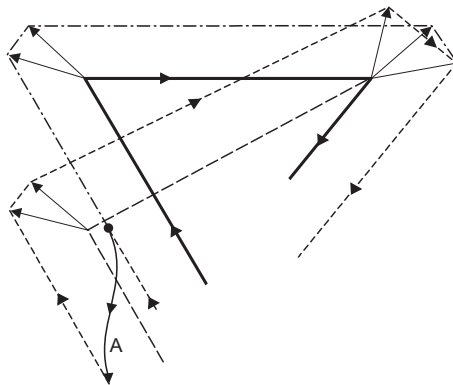


(2) Interrupt with manual absolute ON

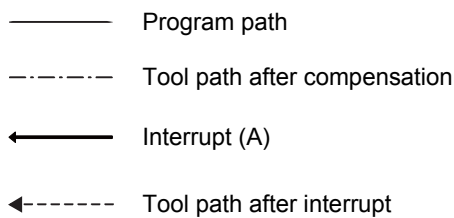
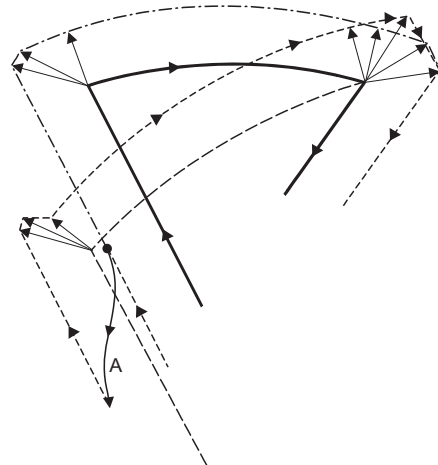
In the incremental value mode, the same operation will be performed as the manual absolute OFF.

In the absolute value mode, however, the tool returns to its original path at the end point of the block following the interrupted block, as shown in the figure.

[Line-Line-Line]



[Line-arc-Line]



12.3.5 General Precautions for Tool Radius Compensation



Precautions

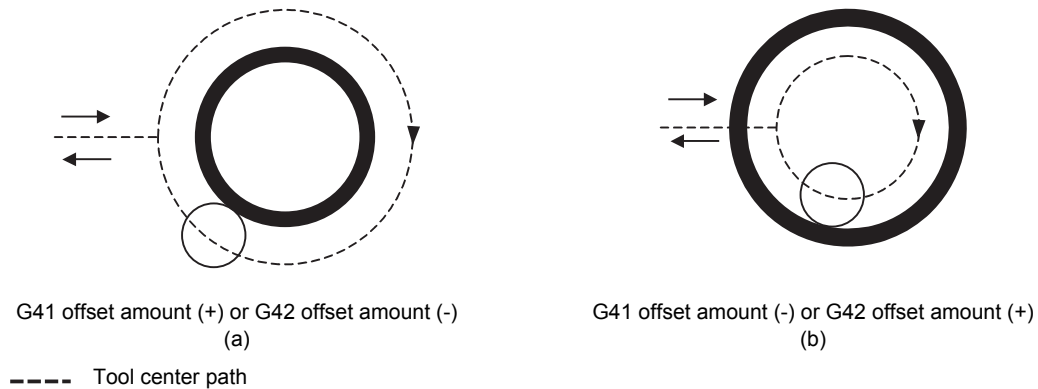
Assigning the compensation amounts

- (1) The offset amounts can be designated with the D code by designating an offset amount No. Once designated, the D code is valid until another D code is commanded. If an H code is designated, the program error (P170) No COMP No will occur.
 Besides being used to designate the compensation amounts for tool radius compensation, the D codes are also used to designate the compensation amounts for tool position compensation.
- (2) Compensation amounts are normally changed when a different tool has been selected in the compensation cancel mode. However, when an amount is changed during the compensation mode, the vectors at the end point of the block are calculated using the compensation amount designated in that block.

Compensation amount symbols and tool center path

If the compensation amount is negative (-), the figure will be the same as if G41 and G42 are interchanged. Thus, the axis that was rotating around the outer side of the workpiece will rotate around the inner side, and vice versa. An example is shown below. Normally, the compensation amount is programmed as positive (+). However, if the tool path center is programmed as shown in (a) and the compensation amount is set to be negative (-), the movement will be as shown in (b). On the other hand, if the program is created as shown in (b) and the offset amount is set to be negative (-), the movement will be as shown in (a). Thus, only one program is required to execute machining of both male and female shapes. The tolerance for each shape can be randomly determined by adequately selecting the offset amount.

(Note that a circle will be divided with type A when compensation is started or canceled.)



12.3.6 Changing of Compensation No. during Compensation Mode



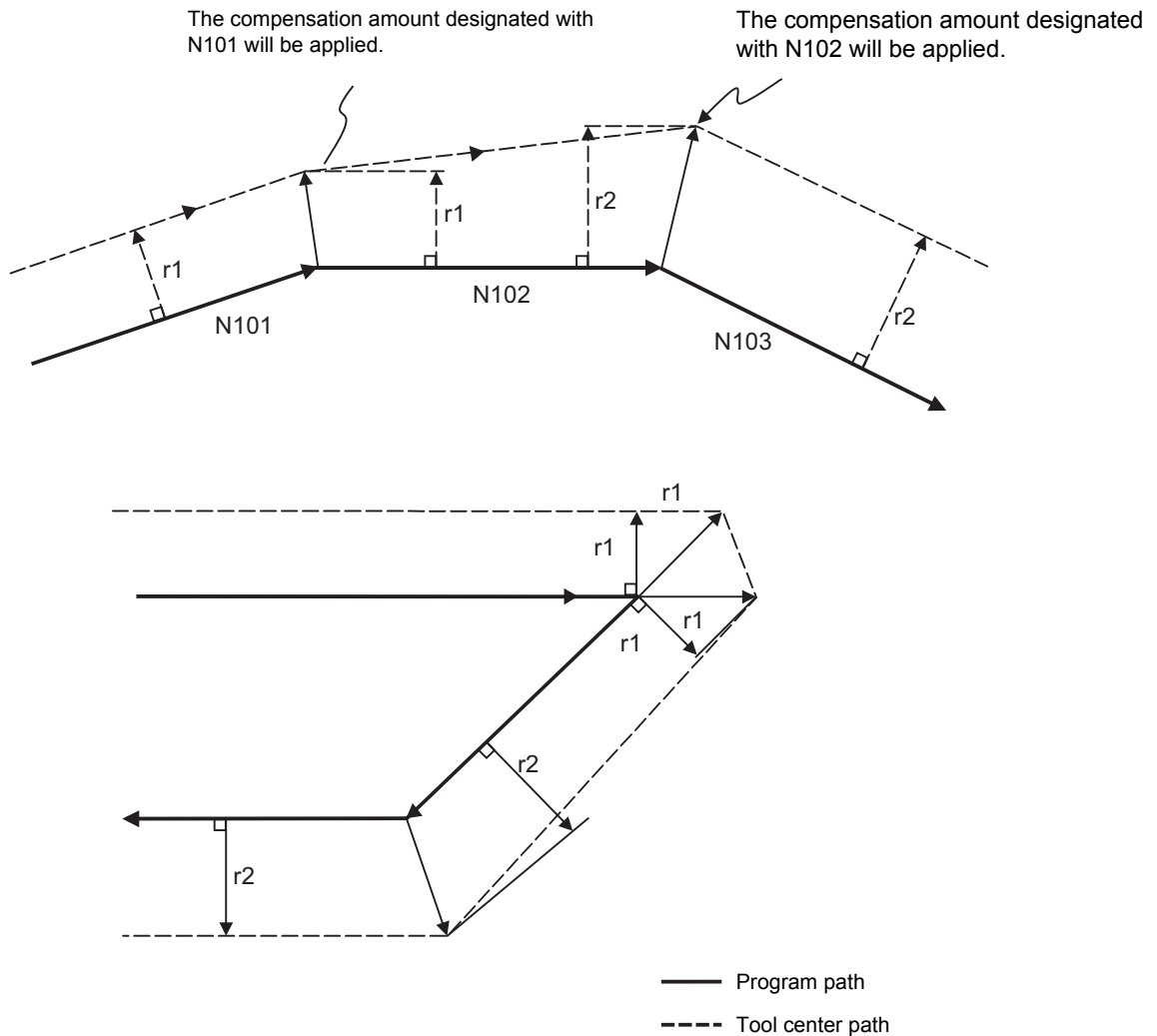
Function and purpose

As a principle, the compensation No. must not be changed during the compensation mode. If changed, the movement will be as shown below.

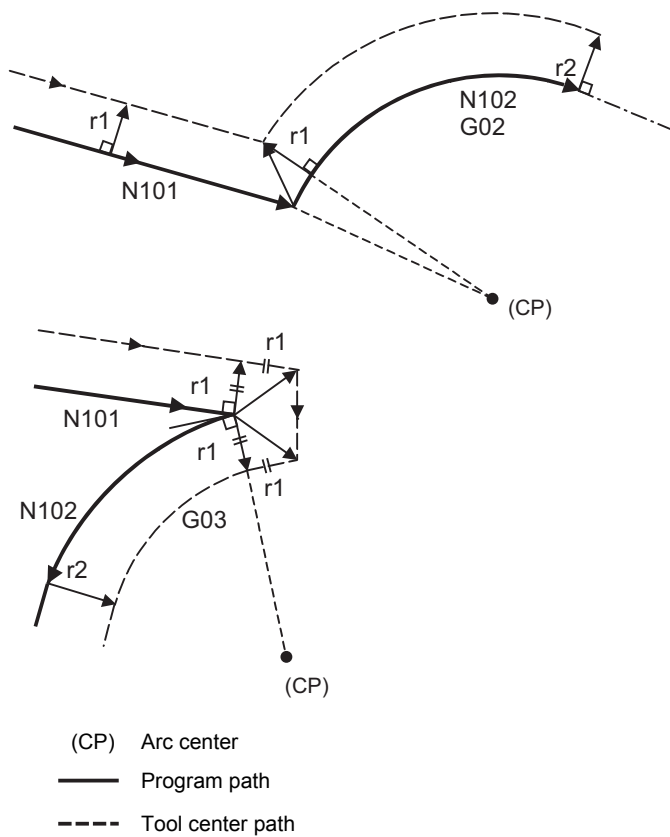
When compensation No. (compensation amount) is changed:

```
G41 G01 ..... Dr1 ;
(α = 0,1,2,3)
N101 G0α Xx1 Yy1 ;
N102 G0α Xx2 Yy2 Dr2 ; ..... Compensation No. changed
N103      Xx3 Yy3 ;
```

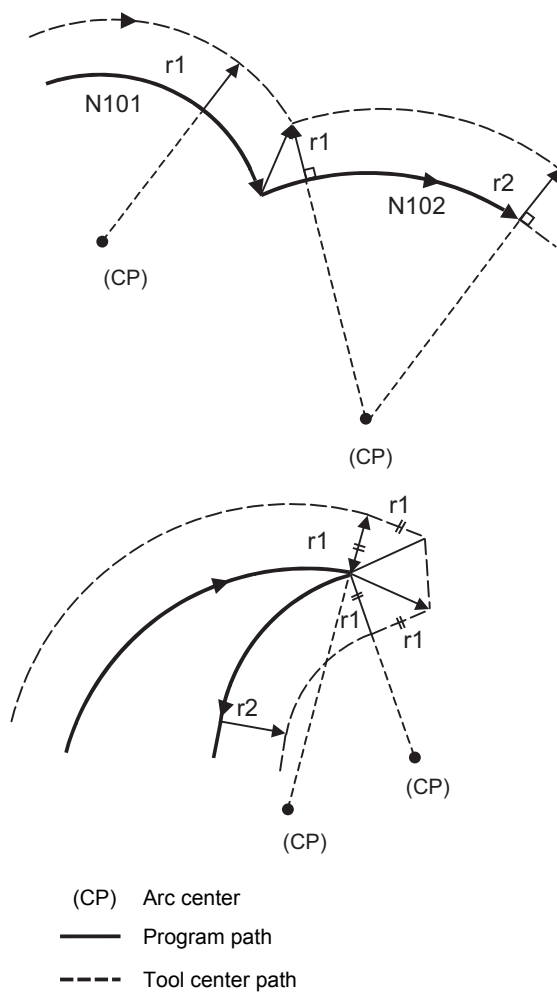
During linear -> linear



Linear ->circular



Circular -> circular



12.3.7 Start of Tool Radius Compensation and Z Axis Cut in Operation



Function and purpose

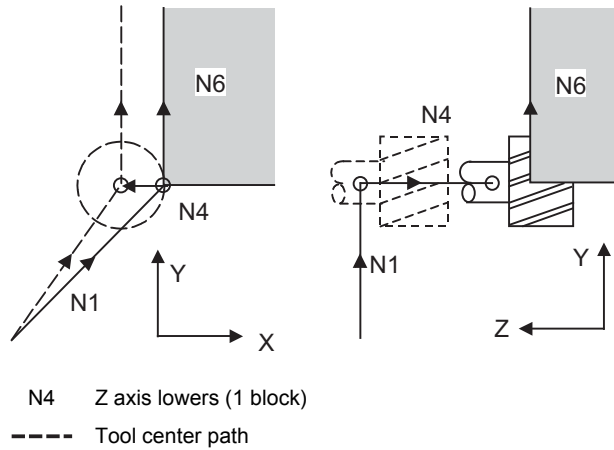
Often when starting cutting, a method of applying a radius compensation (normally the XY plane) beforehand at a position separated for the workpiece, and then cutting in with the Z axis is often used. When using this method, create the program so that the Z axis movement is divided into the two steps of rapid traverse and cutting feed after nearing the workpiece.



Program example

When the following type of program is created:

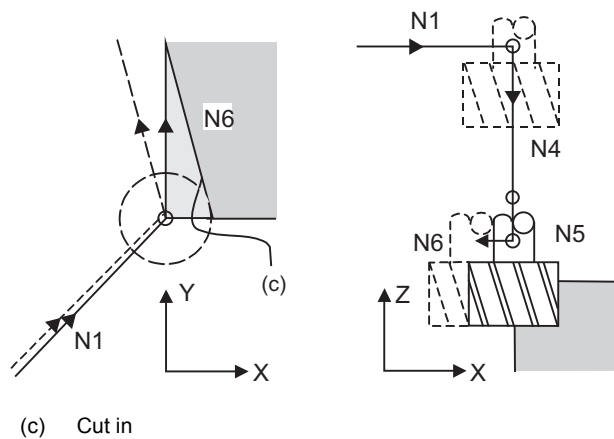
```
N1 G91 G00 G41 X500. Y500. D1 ;
N2 S1000 ;
N3 M3 ;
N4 G01 Z-300. F1 ;
N6 Y100. F2 ;
:
:
```



With this program, at the start of the N1 compensation the program will be read to the N6 block. The relation of N1 and N6 can be judged, and correct compensation can be executed as shown above.

If the above program's N4 block is divided into two

```
N1 G91 G00 G41 X500. Y500. D1 ;
N2 S1000 ;
N3 M3 ;
N4 Z-250. ;
N5 G01 Z-50. F1 ;
N6 Y100. F2 ;
```

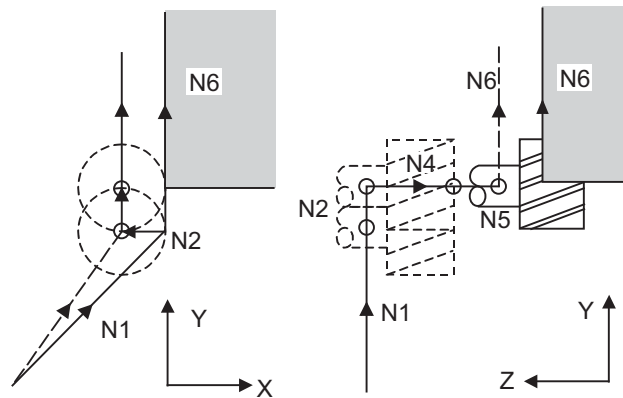


In this case, the four blocks N2 to N5 do not have a command in the XY plane, so when the N1 compensation is started, the program cannot be read to the N6 block.

As a result, the compensation is done based only on the information in the N1 block, and the compensation vector is not created at the start of compensation. Thus, an excessive cut in occurs as shown above.

In this case, consider the calculation of the inner side, and before the Z axis cutting, issue a command in the same direction as the direction that the Z axis advances in after lowering, to prevent excessive cutting.

```
N1 G91 G00 G41 X500. Y400. D1 ;
N2 Y100. S1000 ;
N3 M3 ;
N4 Z-250. ;
N5 G01 Z-50. F1 ;
N6 Y100. F2 ;
```



The movement is correctly compensated as the same direction as the N6 advance direction is commanded in N2.

12.3.8 Interference Check



Function and purpose

When tool radius is larger than the program path, a tool, compensated for by the tool radius compensation function, may sometimes cut into the workpiece. This is known as interference, and interference check is the function which prevents this from occurring.

The table below shows the three functions of interference check and each can be selected for use by parameter.

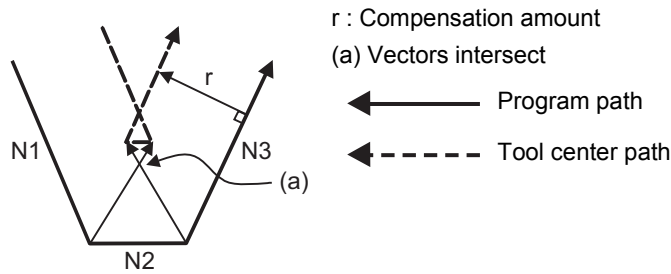
	Function	Parameters		Operation
		#8102 COLL. ALM OFF	#8103 COLL. CHK OFF	
(1)	Interference check alarm function	0	0	Operation stops with a program error (P153) before executing a block which will cause cutting.
(2)	Interference check avoidance function	1	0	The tool path is changed to prevent cutting from occurring. If the path cannot be changed, a program error (P153) occurs and the program will be stopped.
(3)	Interference check invalid function	0/1	1	Cutting continues as is, even if the workpiece is cut into. Use in the fine segment program.



Detailed description

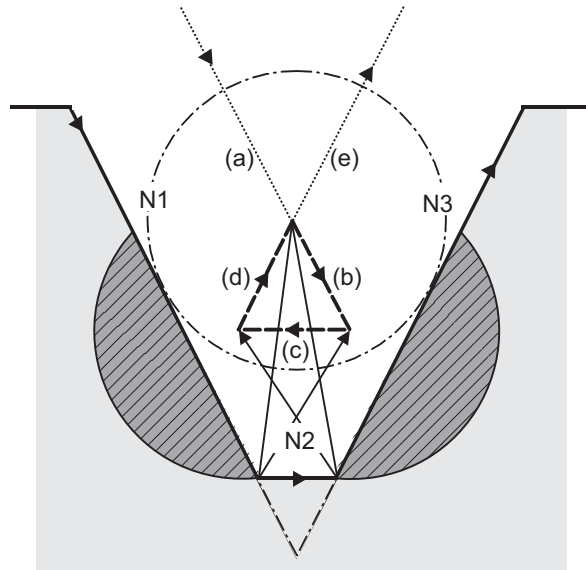
Conditions viewed as interference

When there is a movement command in three of the five pre-read blocks, and if the compensation calculation vectors which are created at the contacts of movement commands intersect each other, it will be viewed as interference.



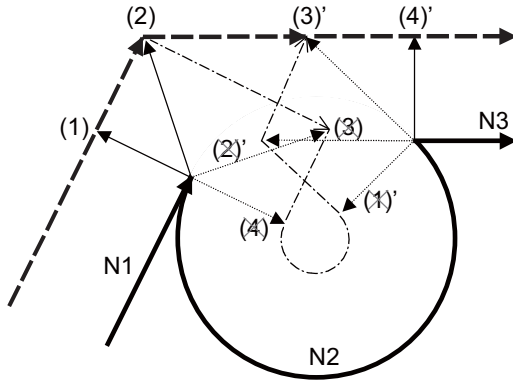
(Example 1) When operating a program including a short segment with a tool with a large radius, cutting will occur in the shaded section.

```
(G41)
N1 G91 G01 X50. Y-100. ;
N2 X70. Y-100. ;
N3 X120. Y0 ;
```



- (1) With alarm function. An alarm is output before N1 is executed. The buffer correction function can thus be used to change N1 to the following, enabling machining to continue: N1 G01 X20. Y-40.;
- (2) With avoidance function
The intersection of N1 and N3 is calculated to create interference avoidance vectors.
Tool center path is (a) -> (e).
- (3) With interference check invalid function.
The tool passes while cutting the N1 and N3 line.
Tool center path is (a)->(b)->(c)->(d)->(e) .

(Example 2) When operating a program including a small circular with a tool with a large radius, cutting occurs near the start point/end point of the circular in the following figure.



Interference check processing

Vectors (1) (4)' check -> No interference



Vectors (2) (3)' check -> No interference



Vectors (3) (2)' check -> Interference -> Erase vectors (3) (2)'



Erase vectors (4) (1)'

(1) With alarm function

The alarm occurs before N1 is executed.

(2) With avoidance function

With the above process, the vectors (1), (2), (3)' and (4)' will remain as the valid vectors. The tool center path will follow the path that connects vectors (1), (2), (3)' and (4)', as the interference avoidance path.

← - - - - (Thick broken line path)

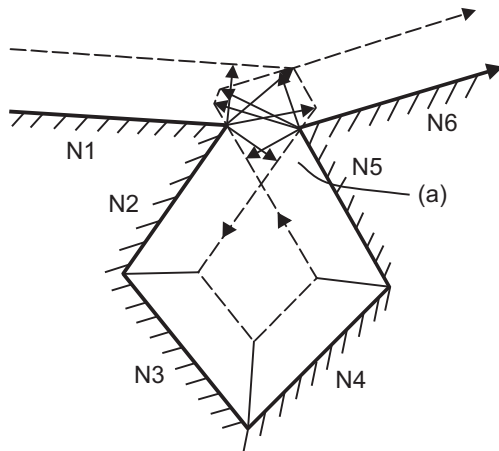
(3) With interference check invalid function

The tool center path will follow the path that connects (1), (2), (3), (4), (1)', (2)', (3)', (4)', as the interference avoidance path while cutting.

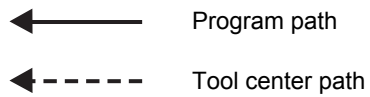
← - . - - - (Thin broken line path)

When interference check cannot be executed

- (1) When three of the movement command blocks cannot be pre-read
(when there are three or more blocks in the five pre-read blocks that are not moving)
- (2) When there is an interference following the fourth movement block

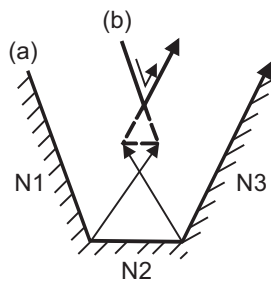


(a) Interference check is not possible

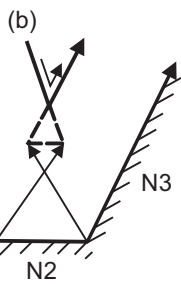


Operation when interference avoidance function is valid

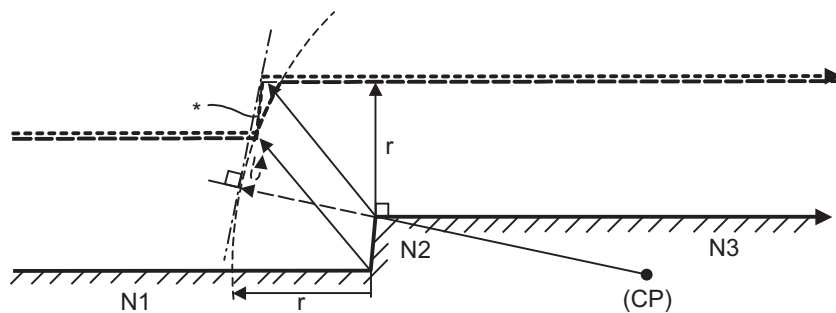
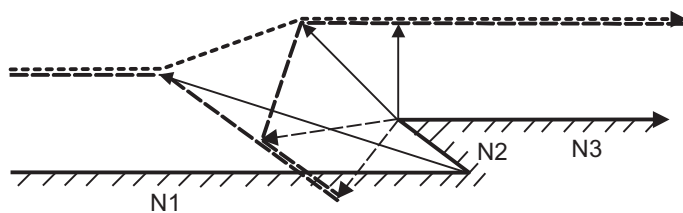
The movement will be as shown below when the interference avoidance check is valid.



(a) Program path

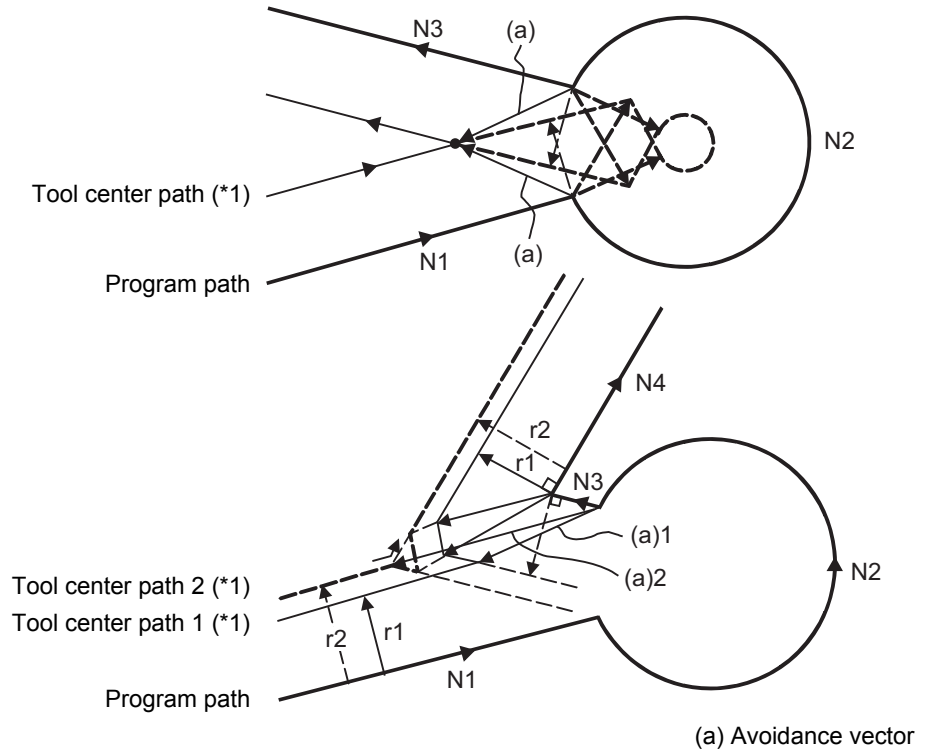


(b) Tool center path



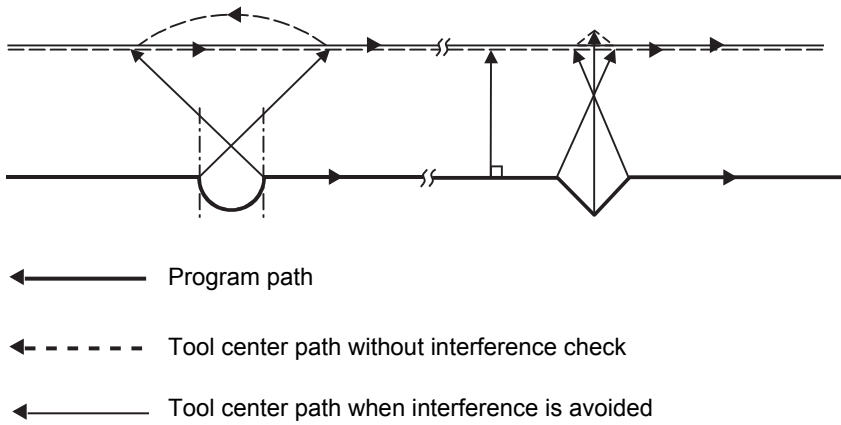
- ← Program path
- ← - - - - - Tool center path without interference check
- ← ······ Tool center path when interference is avoided (*: Linear movement)
- ← Valid vector
- ← - - - - - Invalid vector

If all of the line vectors for the interference avoidance are deleted, create a new avoidance vector as shown in below to avoid the interference.



(*1) Tool center path when interference is avoided

In the case of the figure below, the groove will be left uncut.

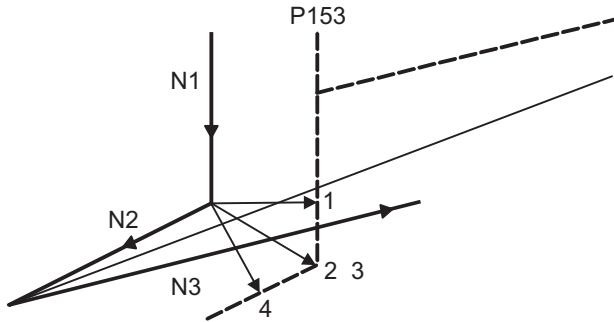


Interference check alarm operation

The interference check alarm occurs under the following conditions.

- (1) When the interference check alarm function has been selected
When all vectors at the end of its own block have been deleted

As shown in the figure below, when vectors 1 through 4 at the end point of the N1 block have all been deleted, program error (P153) will occur prior to N1 execution.

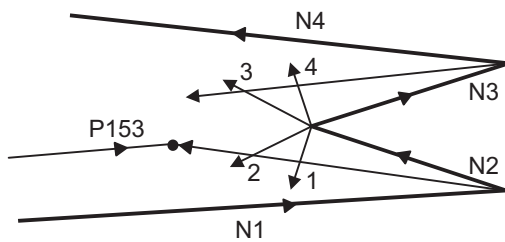


- (2) When the interference check avoidance function has been selected

(Example 1) When there are valid vectors at the end point of the following blocks even when all the vectors at the end point of its own block have been deleted

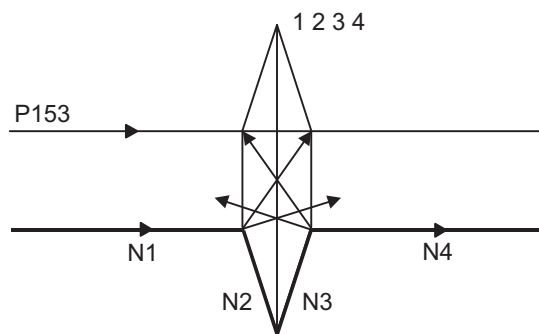
When, in the figure below, the N2 interference check is conducted, the N2 end point vectors are all deleted but the N3 end point vectors are regarded as valid.

Program error (P153) now occurs at the N1 end point and the operation stops.



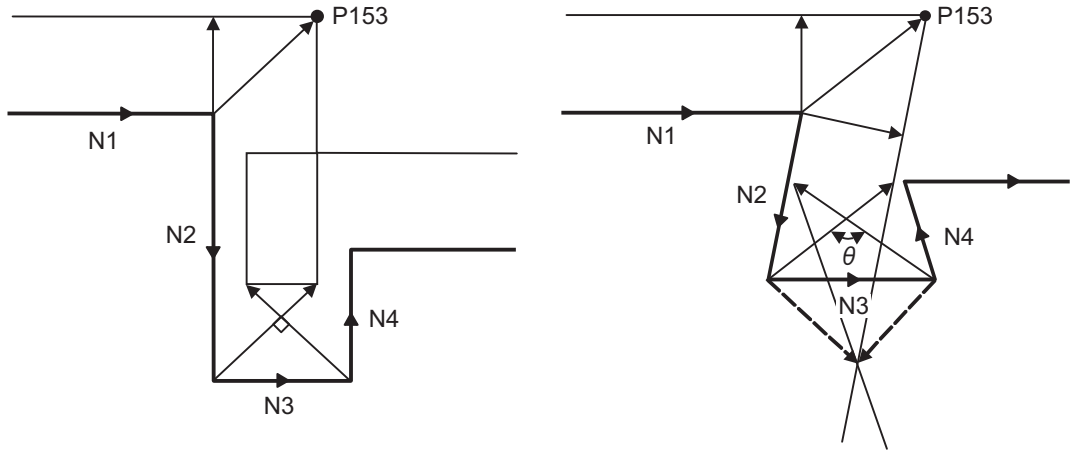
In the case shown in the figure below, the tool will move in the reverse direction at N2.

Program error (P153) now occurs before executing N1 and the operation stops.



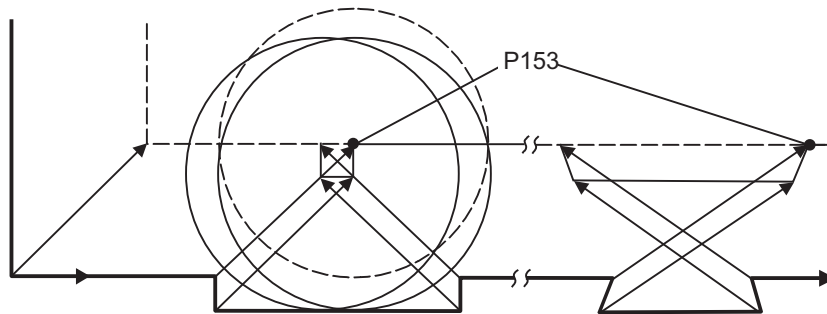
(Example 2) When avoidance vectors cannot be created

Even when, as in the figure below, the conditions for creating the avoidance vectors are satisfied, it may still be impossible to create avoidance vectors, or the interference vectors may interfere with N3. Program error (P153) will occur at the N1 end point when the vector intersecting angle is more than 90° and the operation will stop.



(Example 3) When the program advance direction and the advance direction after compensation are reversed

When grooves that are narrower than the tool diameter with parallel or widening bottom are programmed, it will still be regarded as interference even if there is actually no interference.



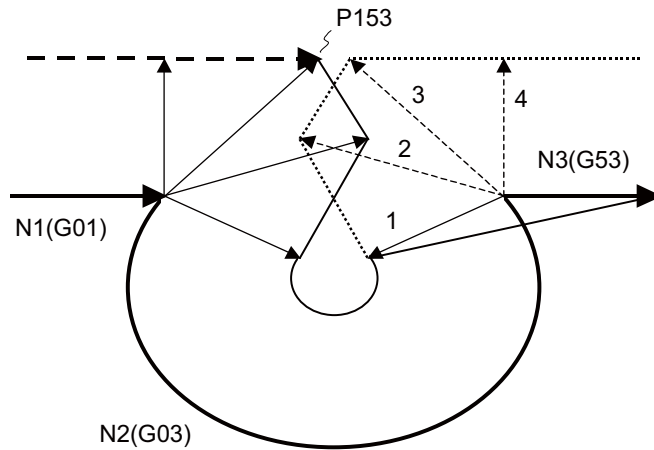
12 Tool Compensation Functions

(Example 4) When vectors at the end point of the block immediately before the command to eliminate compensation vectors temporarily cause an interference

Interference check will be executed also at the end point of the block immediately before the command to eliminate compensation vectors temporarily, similarly with the case compensation vectors are not eliminated. It may be regarded as an interference even if there is actually no interference. If regarded as an interference, program error (P153) will occur.

In the figure below, only vector 1 is left as an end point vector in N2 because of the N3 G53 command to temporarily eliminate compensation vectors. However, the interference check will still be conducted to vector 1 to 4 and an interference will be detected.

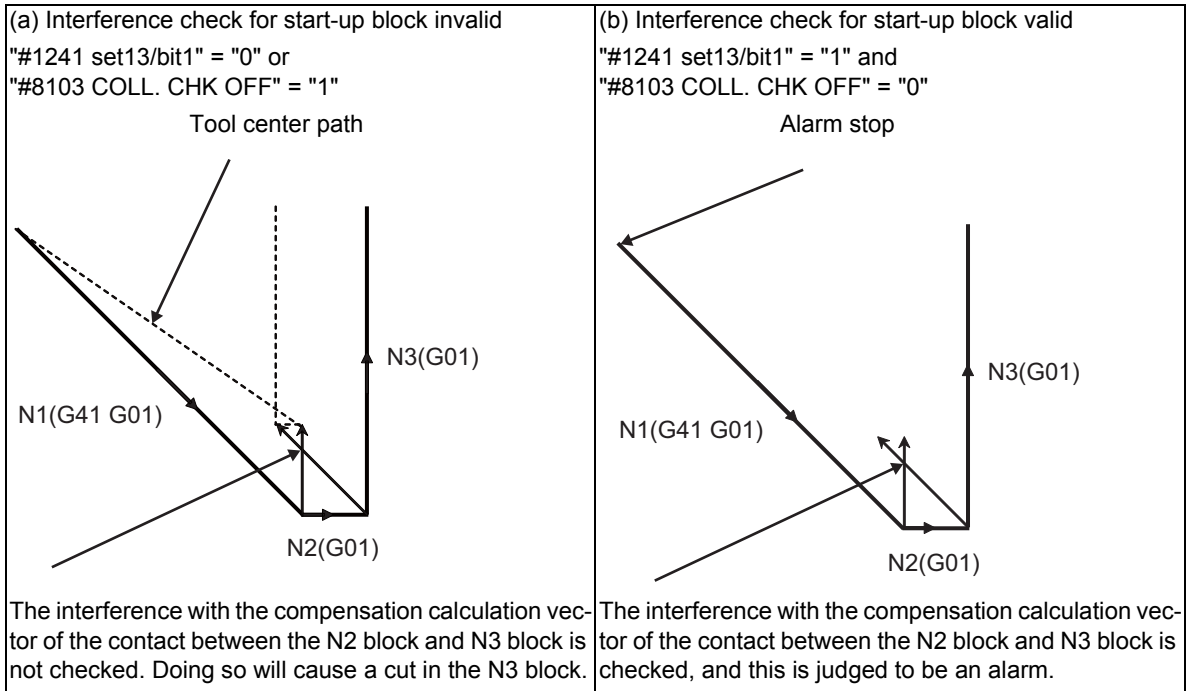
Program error (P153) now occurs at the end point of the previous block and the operation stops.



- ← Program path
- ← - - - - Tool center path
- Tool center path when the interference check is invalid
- Tool center path when the interference check is invalid in N3 due to a command (G01 etc.) not to eliminate compensation vectors.
- ← Valid vector
- ← - - - - Invalid vector (Invalid, however, subject to interference check)

Interference check for start-up block

When starting compensation operation, the tool center path is determined with the movement command of the same block as G41/G42 and the next movement command. The interference check is not executed at that time. To check interference, set the parameter "#1241 set13/bit1" (MTB specifications). Note that an alarm is output and the operation is stopped even when the collision avoidance setting "#8102 COLL. ALM OFF" is set to "1" and that the interference avoidance is not applied.



12.3.9 Diameter Designation of Compensation Amount



Function and purpose

With this function, the tool radius compensation amount can be designated by tool diameter. When the control parameter "#8117 OFS Diam DESIGN" is ON, the compensation amount specified to the commanded tool No. will be recognized as the diameter compensation amount, and the amount will be converted to the radius compensation amount when executing the compensation.

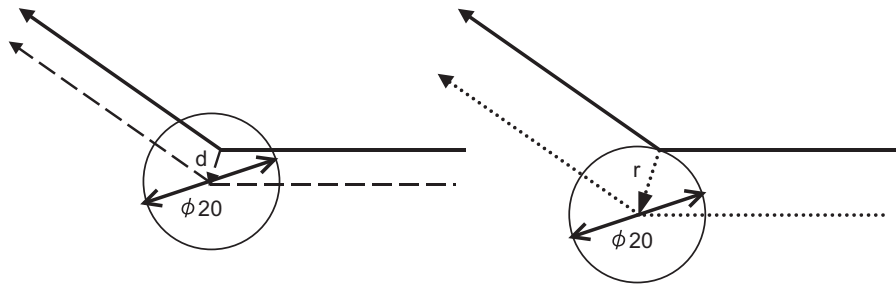


Operation example

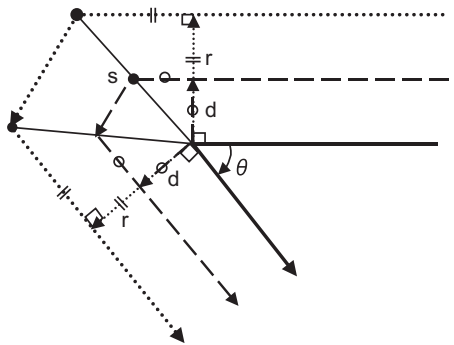
Operations when designating the compensation amount with diameter

When the tool radius compensation amount $D=10.0$ is commanded, tool radius compensation amount "d" is 5.0 if the parameter "#8117" is ON (set to "1").

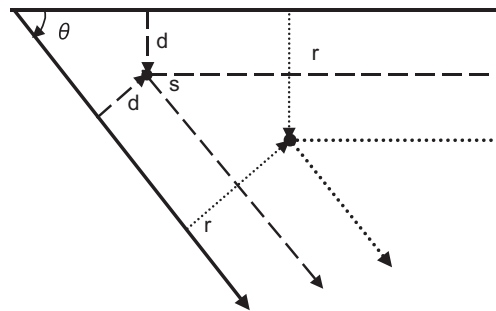
(Tool radius compensation amount "r" is 10.0 if the parameter "#8117" is OFF (set to "0").)



(1) Linear -> linear corner (acute angle)



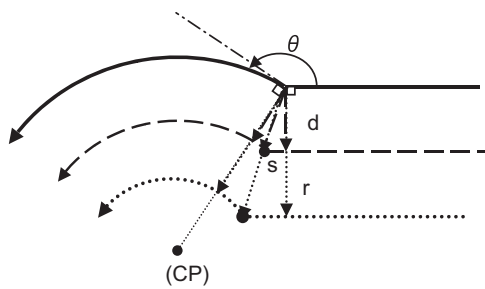
Outside of the corner



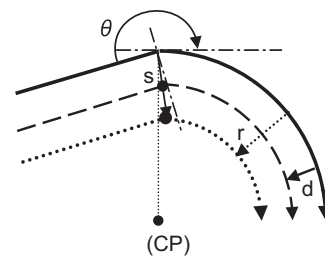
Inside of the corner

- Program path
- - - Tool center path (When #8117 is ON)
- Tool center path (When #8117 is OFF)

(2) Linear -> arc (obtuse angle)

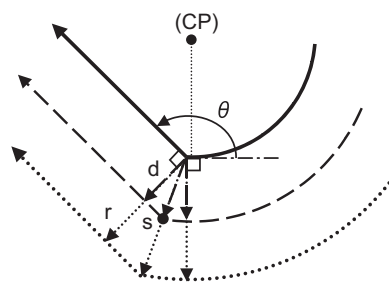


Outside of the corner

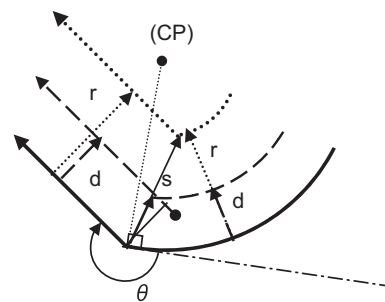


Inside of the corner

(3) Arc -> linear (obtuse angle)



Outside of the corner



Inside of the corner

- Program path
- - - Tool center path (When #8117 is ON)
- Tool center path (When #8117 is OFF)
- (CP) Arc center



Restrictions

- (1) If tool radius compensation amount has already been set, the compensation amount is not be changed even if the parameter "8117" is changed.
- (2) Make sure not to change the parameter #8117 during the compensation. When the parameter is changed using parameter input by program function, the program error (P421) will occur.
- (3) If the parameter #8117 is set to ON with the parameter "#1037 cmdtyp" set to "2", the tool radius wear data is also regarded as the diameter compensation amount, thus, it will be converted to the radius value and compensation will be performed.
- (4) Diameter designation of tool radius compensation amount can be used for the tool life management data.
- (5) There is no effect by #8117 on the tool radius measurement function.

12.3.10 Workpiece Coordinate Changing during Radius Compensation



Function and purpose

When the tool radius compensation is executed, the tool center path is calculated based on the position on the coordinate system. The based coordinate system is different depending on setting of the parameter "#1246/bit2 Switch coordinate systems for radius compensation". (This depends on the MTB specifications.)

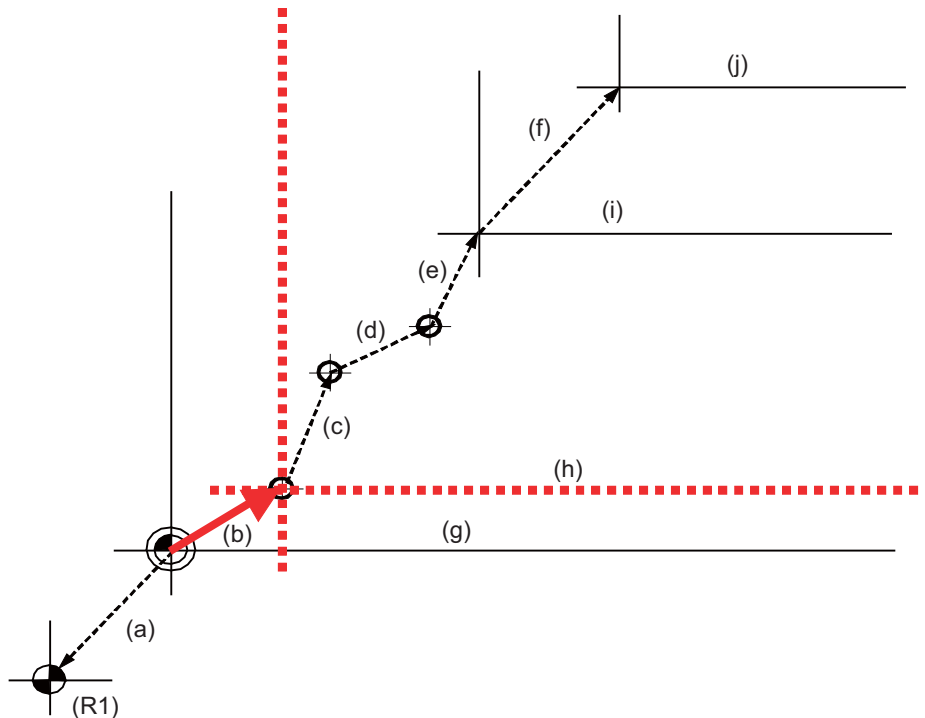


Detailed description

When the parameter is "0", the tool radius compensation is calculated based on the position on the workpiece coordinate system.

When the parameter is "1", the tool radius compensation is calculated based on the position on the program coordinate system.

The program coordinate systems are defined as shown in the figure below.



- (R1) 1st reference position
- (a) 1st reference position offset
- (b) Interrupt amount offset
- (c) Extended workpiece coordinate system offset
- (d) G92 offset
- (e) Workpiece coordinate system offset
- (f) Local coordinate system offset
- (g) G53 Basic machine coordinate system
- (h) Program coordinate system
- (i) G54 to G59/G54.1Pn Workpiece coordinate system/Extended workpiece coordinate system
- (j) G52 Local coordinate system

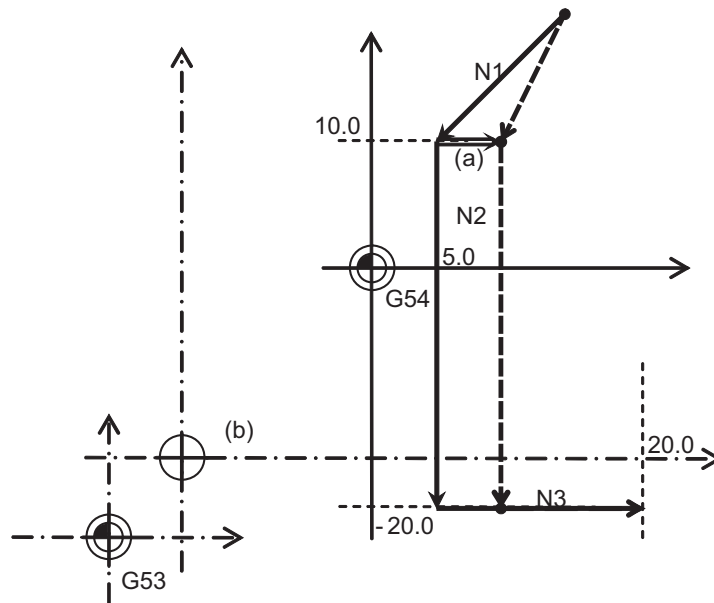
12 Tool Compensation Functions

The coordinate system changed by parameter is as follows.

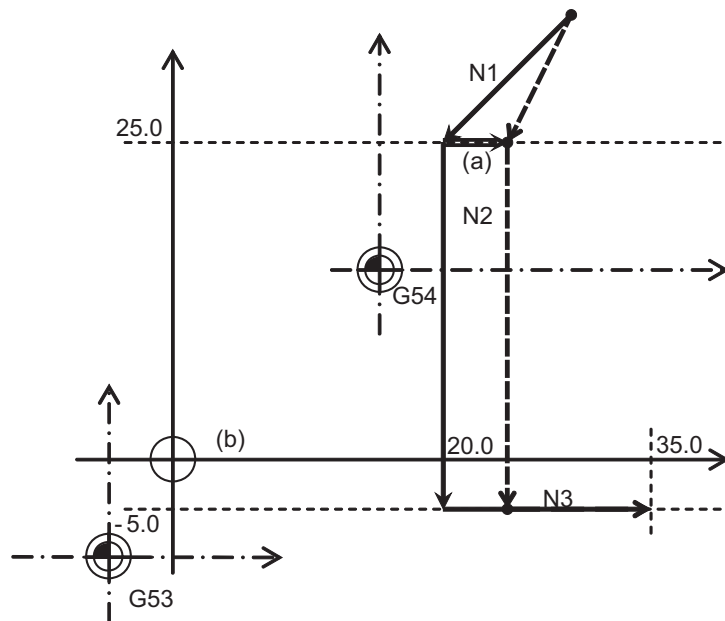
G90 G54 G00 X15. Y20.;
N1 G41 D3 X5. Y10. ;
N2 G01; Y-20 F1000;
N3 G40 X20. ;
M30 ;

D3 =5.000
 G54 offset
 X15.000
 Y15.000

(1) Parameter = 0



(2) Parameter = 1



- Program path
- - - Tool center path
- (a) Compensation vector
- (b) Program coordinate system

12.4 Tool Nose Radius Compensation (for Machining Center System)



Function and purpose

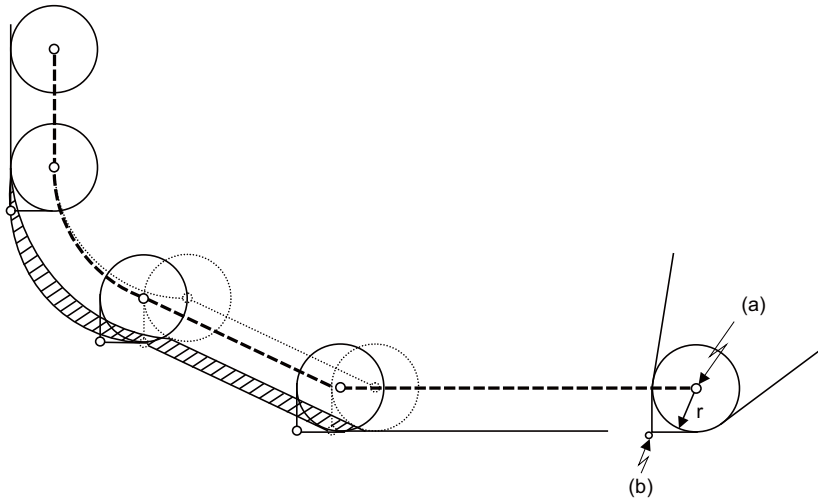
Because a tool nose is generally rounded, a hypothetical tool nose point is used for programming. Due to this roundness of the tool nose, there will be a gap between the programmed shape and the actual cutting shape during taper cutting or circular cutting. Tool nose radius compensation (nose R compensation) compensation is a function for automatically calculating and offsetting this error by setting the tool nose radius (cutter radius) value.

The validity of this function depends on the MTB specifications. (The tool nose must be set to "1" to "8" in the parameter "#1037 cmdtyp".)

If the tool nose is set to "0" or "9", tool radius compensation is carried out.

When G46 is commanded, the tool position offset reduction function is enabled. (The automatic direction identification mode is not available.)

Refer to "Programming Manual Lathe System" for details of the tool nose radius compensation.



(a) Tool nose center

(b) Hypothetical tool nose point

(r) Tool nose radius

..... Tool nose center path with no nose R compensation (Shaded part indicates the cutting shape gap)

----- Tool nose center path with nose R compensation



Command format

G40; Tool nose radius compensation cancel

G41 (X_ Z_ D_); Tool nose radius compensation left

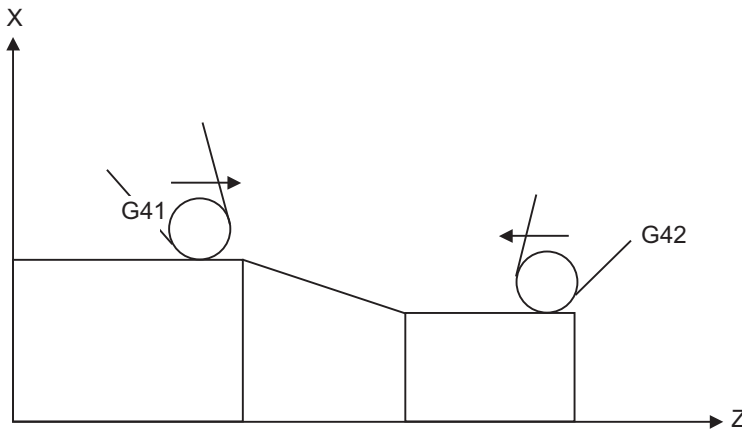
G42 (X_ Z_ D_); Tool nose radius compensation right

X	X axis end point coordinate (Absolute value of workpiece coordinate system)
Z	Z axis end point coordinate (Absolute value of workpiece coordinate system)
D	Compensation No. (The compensation No. setting range will differ according to the specifications (No. of compensation sets).)

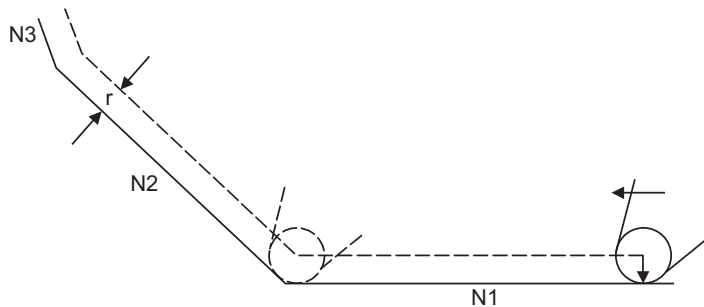


Detailed description

- (1) G41 works on condition that the tool is located on the left of the workpiece to the direction of motion.
G42 works on condition that the tool is located on the right of the workpiece to the direction of motion.
G40 cancels the tool nose radius compensation mode.



- (2) Nose R compensation pre-reads the data in the following two movement command blocks (up to 5 blocks when there is no movement command) and controls the tool nose radius center path by the intersection point calculation method so that it is offset from the programmed path by an amount equivalent to the nose R.
In the figure below, "r" is the tool nose radius compensation amount (nose R).
The nose R compensation amount corresponds to the tool length No. and should be preset along with the tool nose point.



- (3) If there are 4 or more blocks without movement amounts among 5 continuous blocks, overcutting or undercutting will occur.
Blocks in which optional block skip is valid are ignored.
- (4) Tool nose radius compensation is also valid for fixed cycle.
- (5) Compensation mode will be temporarily canceled in 1 block before the thread cutting command block.
- (6) The compensation plane, movement axes and next advance direction vector follow the plane selection command designated by G17, G18 or G19.

G17	XY plane X,Y,I,J
G18	ZX plane Z,X,K,I
G19	YZ plane Y,Z,J,K



Precautions

Assigning the compensation amounts

- (1) For nose radius compensation for machining center system, the compensation amount should always be specified with radius value.
- (2) Compensation amounts are normally changed when a different tool has been selected in the compensation cancel mode. However, when an amount is changed during the compensation mode, the vectors at the end point of the block are calculated using the compensation amount designated in that block.

Corner judgment method

- (1) The criterion to execute the outer rounding at the small corner in tool radius compensation depends on the MTB specifications (parameter "#1289 ext25/bit0").

Designating the tool nose point

- (1) There two methods to set the tool nose point for machining center system. (This depends on the MTB specifications.)
 - Set a value in the system variable #23000+n ("n" corresponds to the compensation number) using the machining program.
 - Set the parameter "#1046 T-ofs disp type" to change the compensation type to III, then, set a value in the tool compensation amount screen.

Errors during tool nose radius compensation

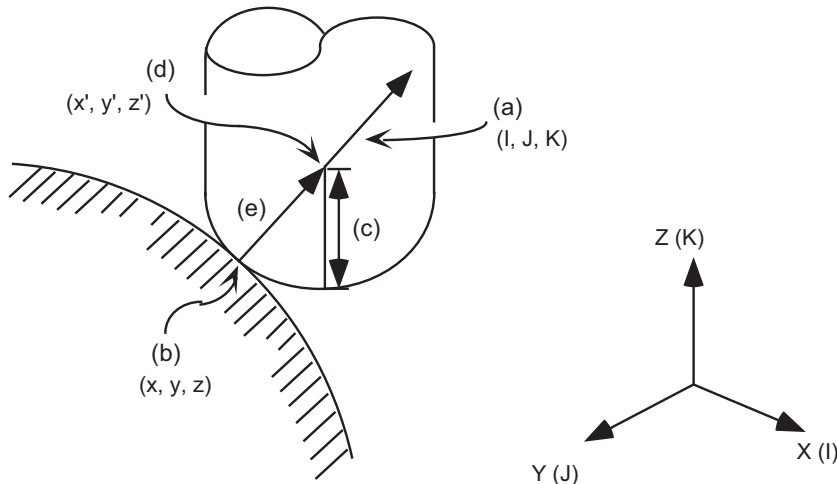
- (1) An error will occur when any of the following commands is programmed during tool nose radius compensation.
 - G17, G18, G19 (when a plane different from the one used during the compensation is commanded (P112))
 - G31 (P608)
 - G74,G75,G76 (P155)
 - G81 to G89(P155)
- (2) A program error will occur when a circular command is issued in the first or last block of the tool nose radius compensation. (P151)
- (3) A program error will occur during tool nose radius compensation when the intersection point of single block skip in the interference block processing cannot be calculated. (P152)
- (4) A program error will occur when there is an error in one of the pre-read blocks during tool nose radius compensation.
- (5) A program error will occur when an interference occurs under no interference avoidance conditions during tool nose radius compensation. (P153)
- (6) A program error will occur when a tool nose radius compensation command is issued even though the tool nose radius compensation specification is not provided. (P150)
- (7) If a tool nose radius compensation command is issued in mirror image, a program error will occur. (P803)

12.5 3-dimensional Tool Radius Compensation; G40/G41, G42



Function and purpose

The 3-dimensional tool radius compensation compensates for the tool in a 3-dimensional space following the commanded three-dimensional vectors.



As shown above, the tool is moved in the tool center coordinate position (x', y', z') (d) which is compensated for by the tool radius "r" (c) in respect to the program coordinate position (x, y, z) (b) following the plane normal line vector (l, j, k) (a).

Though two-dimensional tool radius compensation creates the vectors at a right angle to the (l, j, k) direction, three-dimensional tool radius compensation creates the vector in the (l, j, k) direction. (The vector is created at the end point of the block.)

The three-dimensional compensation vector (compensation) (e) axis elements are as below.

$$H_x = \frac{l}{\sqrt{(l^2 + j^2 + k^2)}} \times r$$

$$H_y = \frac{j}{\sqrt{(l^2 + j^2 + k^2)}} \times r$$

$$H_z = \frac{k}{\sqrt{(l^2 + j^2 + k^2)}} \times r$$

Thus, the tool center coordinate position (x', y', z') (d) is each expressed as below. Note that (x, y, z) are the program coordinate position.

$$x' = x + H_x$$

$$y' = y + H_y$$

$$z' = z + H_z$$

Note

- (1) Three-dimensional compensation vector (H_x, H_y, H_z) refers to the plane normal line vector whose direction is same as the plane normal line vector (l, j, k) and the size equals to the tool radius "r".
- (2) When the machining parameter "#8071 3-D CMP" is set to a value other than "0", $\sqrt{(l^2 + j^2 + k^2)}$ the value of "#8071 3-D CMP" will be used as the value. (Refer to the s Setup Manual for details.)



Command format

3-dimensional tool radius compensation start

G41(G42) X__Y__Z__I__J__K__D__;

New plane normal line vector is commanded in the compensation mode.

X__Y__Z__I__J__K__;

3-dimensional tool radius compensation cancel

G40; (or D00;)

G40 X__Y__Z__; (or X__Y__Z__ D00;)

G41	Three-dimensional tool radius compensation command (+ direction)
G42	Three-dimensional tool radius compensation command (- direction)
G40	Three-dimensional tool radius compensation cancel command
X, Y, Z	Movement axis command compensation space
I, J, K	Plane normal line vector
D	Compensation No. (Note that when "D00" is issued, three dimensional tool radius compensation will be canceled even if G40 is not commanded.)

Command, for all three axes, the compensation No. D and plane normal line vector (I, J, K) in the same block as the three-dimensional tool radius compensation command G41 (G42).

If only one or two axes are commanded, the normal tool radius compensation mode will be applied. (When the command value for I, J, K is set to "0", this command is valid.)

G Code	Compensation amount:		D00
	+	-	
G40	Cancel	Cancel	Cancel
G41	I, J, K direction	Reverse direction of I, J, K	Cancel
G42	Reverse direction of I, J, K	I, J, K direction	Cancel



Detailed description

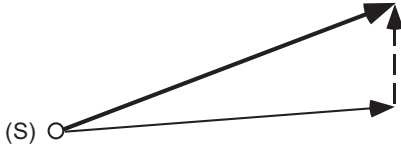
The compensation space is determined by the axis address commands (X, Y, Z, U, V, W) of the block where the three-dimensional tool radius compensation starts.	(Example) G17 ; G41 Xx Yy Zz li Jj Kk ;	X Y Z space
	G17 ; G41 Yy li Jj Kk ;	X Y Z space
Here, U, V and W are each the additional axes for the X, Y and Z axis. If the X axis and U axis (Y and V, Z and W) are commanded simultaneously in the three-dimensional tool radius compensation start block, the currently commanded plane selection axis will have the priority. If the axis address is not commanded, it will be interpreted that the X, Y and Z axes are commanded for the coordinate axes.	G17 V ; G41 Xx Vv Zz li Jj Kk ;	X V Z space
	G17 W ; G41 Ww li Jj Kk ;	X Y W space
	G17 ; G41 Xx Yy Zz Ww li Jj Kk ;	X Y Z space
	G17 W ; G41 Xx Yy Zz Ww li Jj Kk ;	X Y W space
	G17 ; G41 li Jj Kk ;	X Y Z space
	G17 U ; G41 li Jj Kk ;	U Y Z space



Operation example

Compensation start (1) When there is a movement command

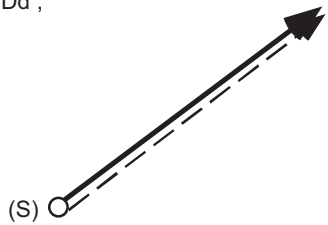
G41 Xx Yy Zz Ii Jj Kk Dd ;



- (S) Start point
- Tool center path
- Programmed path
- Three-dimensional compensation vector

Compensation start : When there is no movement command

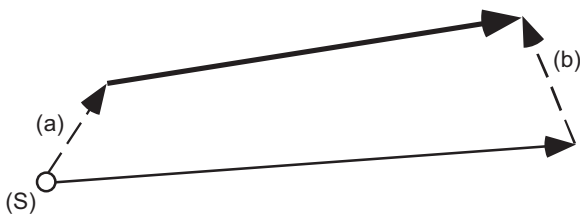
G41 Ii Jj Kk Dd ;



- (S) Start point
- Tool center path
- Three-dimensional compensation vector

Movement during the compensation (3) When there is a movement command and a plane normal line vector command

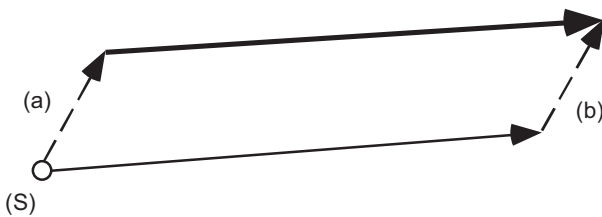
Xx Yy Zz Ii Jj Kk ;



- (S) Start point
- (a) Old vector
- (b) New vector
- Tool center path
- Programmed path

Movement during the compensation (4) When there is no plane normal line vector command

Xx Yy Zz Ii Jj Kk ;



- (S) Start point
- (a) Old vector
- (b) New vector
- Tool center path
- Programmed path

Movement during the compensation (5) For arc or helical cutting

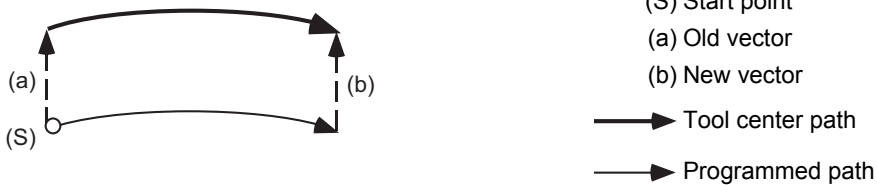
The I, J, K commands for a circular or helical cutting are regarded as the circular center commands, thus, the new vector is equivalent to the old vector.

Even for the R-designation method, commanded I, J, K addresses will be ignored, then the new vector will be equivalent to the old vector.

G02 Xx Yy (Zz) Ii Jj ; I, J(K) means the circular center

Or

G02 Xx Yy (Zz) Rr ; R-designated arc



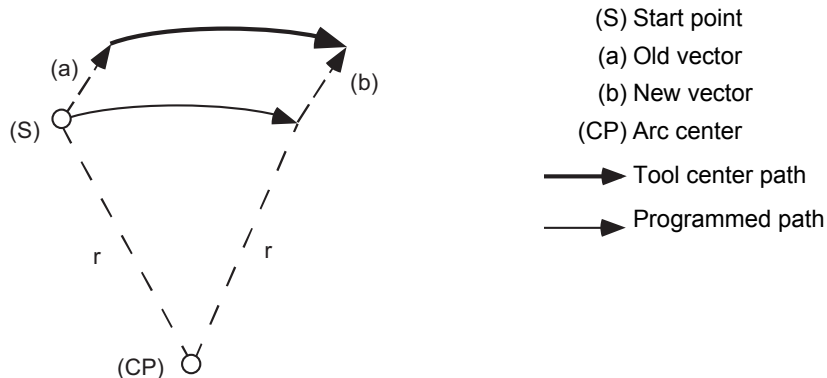
Note

(1) The center coordinate will not shift during the circular or helical cutting. Thus, when I, J, K are commanded with the vector as below, the program error (P70) will occur.

G02 Xx Yy (Zz) Ii Jj ; I, J(K) means the circular center

Or

G02 Xx Yy (Zz) Rr ; R-designated arc

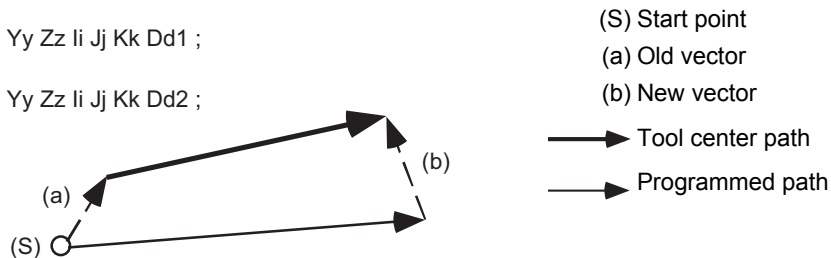


Movement during the tool radius compensation (6) When compensation amount is to be changed

G41 Xx Yy Zz Ii Jj Kk Dd1 ;

:

G41 Xx Yy Zz Ii Jj Kk Dd2 ;

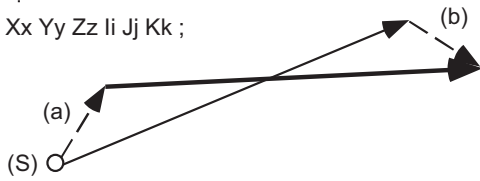


Note

(1) If I, J, K are not commanded in a block where the compensation amount is to be changed, the vector will be equivalent to the old vector. In this case, the modal will change, however, the compensation amount will change when I, J, K are commanded.

Movement during the tool radius compensation (7) When compensation direction is to be changed

```
G41 Xx Yy Zz Ii Jj Kk Dd1 ;
:
G42 Xx Yy Zz Ii Jj Kk ;
```



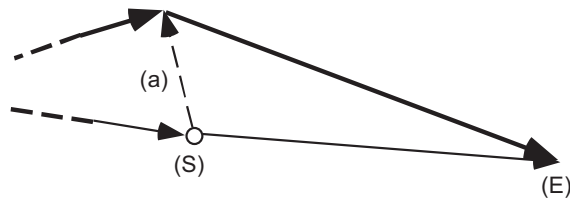
- (S) Start point
- (a) Old vector
- (b) New vector
- Tool center path
- Programmed path

Note

- (1) If I, J and K are not commanded in a block where the compensation direction is to be changed, the vector will be equivalent to the old vector and the compensation direction will not be changed. In this case, the modal will change, however, the compensation direction will change when I, J and K are commanded.
- (2) If the compensation direction is changed in an arc (G02/G03) block, I, J will be the center of the arc, thus, the compensation direction will not change. Even for the R-designation method, commanded I, J and K will be ignored, and the compensation direction cannot be changed.

Movement during the tool radius compensation : When there is a movement command

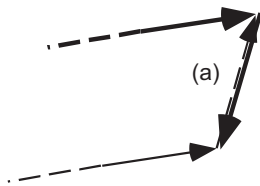
```
G40 Xx Yy Zz ; (or Xx Yy Zz D00 ;)
```



- (S) Start point
- (E) End point
- (a) Old vector
- Tool center path
- Programmed path

Tool radius compensation cancel: When there is no movement command

```
G40; (or D00;)
```



- (a) Old vector
- Tool center path
- Programmed path



Relationship with other functions

Normal tool radius compensation

If the plane normal line vector (I, J, K) is not commanded for all three axes in the three-dimensional tool radius compensation start block, the normal tool radius compensation mode will take place.

If G41 (G42) is commanded without commanding the plane normal line vector during three-dimensional tool radius compensation, the modal will change, however, the old vector will be used.

If G41 (G42) with the plane normal line vector is commanded during tool radius compensation, this command will be ignored and the normal tool radius compensation will take place.

Tool length compensation

Tool length compensation is applied to the coordinate after three-dimensional tool radius compensation.

Tool position offset

Tool position offset is applied to the coordinate after three-dimensional tool radius compensation.

Fixed cycle

The program error (P155) will occur.

Scaling

Scaling is applied to the coordinate before three-dimensional tool radius compensation.

Scaling is not applied to the plane normal line vector (I, J, K).

```
D1=10.
G90 ;
G51 X0 Y0 P0.5 ;
N1 G41 D1 X-10. Y-20. Z-10. I-5. J-5. K-5. ;
N2 X-30. Y-30. Z-20. ;
N3 X-50. Y-20. Z-10. ;
N4 Y0. ;

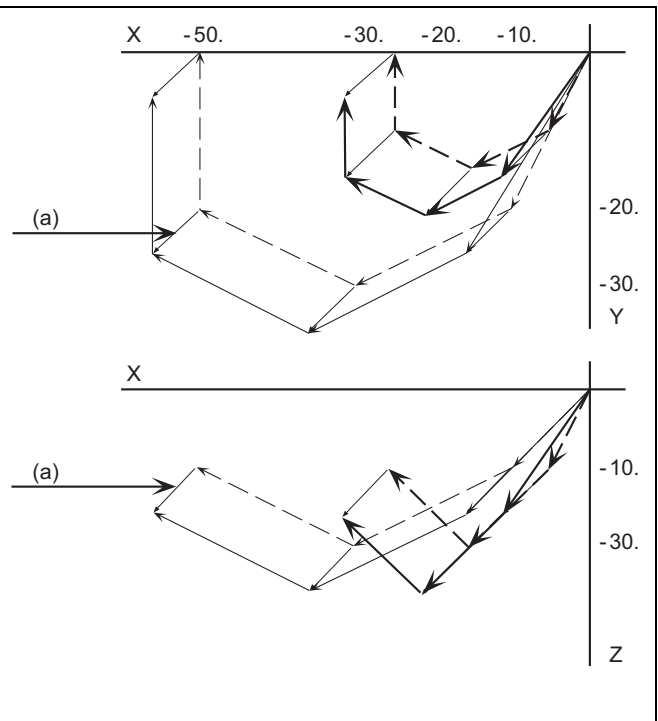
N1( -5.000, -10.000, -10.000 )
N1( -10.773, -15.773, -15.773 )

N2( -15.000, -15.000, -20.000 )
N2( -20.773, -20.773, -25.773 )

N3( -25.000, -10.000, -10.000 )
N3( -30.773, -15.773, -15.773 )

N4( -25.000, 0.000, -10.000 )
N4( -30.773, -5.773, -15.773 )

*Upper: Program position after scaling
Lower: Position after scaling and compensation
```



(a) Plane normal line vector

— — — Programmed path

———— Program path after compensation

· · · · Program path after scaling

———— Program path after scaling and compensation

Coordinate rotation by program

Program coordinate rotation is applied to the coordinate before three-dimensional tool radius compensation. The plane normal line vector (I, J, K) will not rotate.

```

D1=10.
G90 ;
G68 X0 Y0 R45. ;N1 G41 D1 X-10. Y-20. Z-10.
I-5. J-5. K-5. ;
N2 X-30. Y-30. Z-20. ;
N3 X-50. Y-20. Z-10. ;
N4 Y0. ;

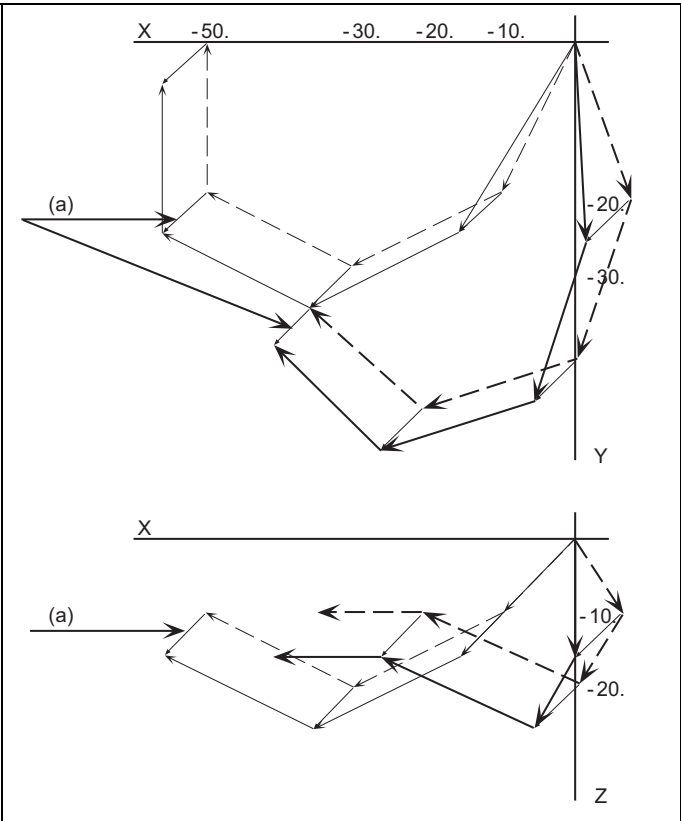
N1( 7.071, -21.213, -10.000 )
N1( 7.071, -29.378, -15.773 )

N2( 0.000, -42.426, -20.000 )
N2( 0.000, -50.591, -25.773 )

N3( -21.213, -49.497, -10.000 )
N3( -21.213, -57.662, -15.773 )

N4( -35.355, -35.355, -10.000 )
N4( -35.355, -43.520, -15.773 )
    
```

* Upper: Program position after coordinate rotation
 Lower: Position after coordinate rotation and compensation



(a) Plane normal line vector

- — — Programmed path
- Program path after compensation
- · - · - Program path after coordinate rotation
- Program path after coordinate rotation and compensation

Coordinate rotation by parameter

Parameter coordinate rotation is applied to the coordinates after three-dimensional tool radius compensation. The plane normal line vector (I, J, K) rotates.

Mirror image

Mirror image is applied to the coordinates after three-dimensional tool radius compensation. Mirror image is applied to the plane normal line vector (I, J, K).

Skip

The program error (P608) will occur.

Reference position check

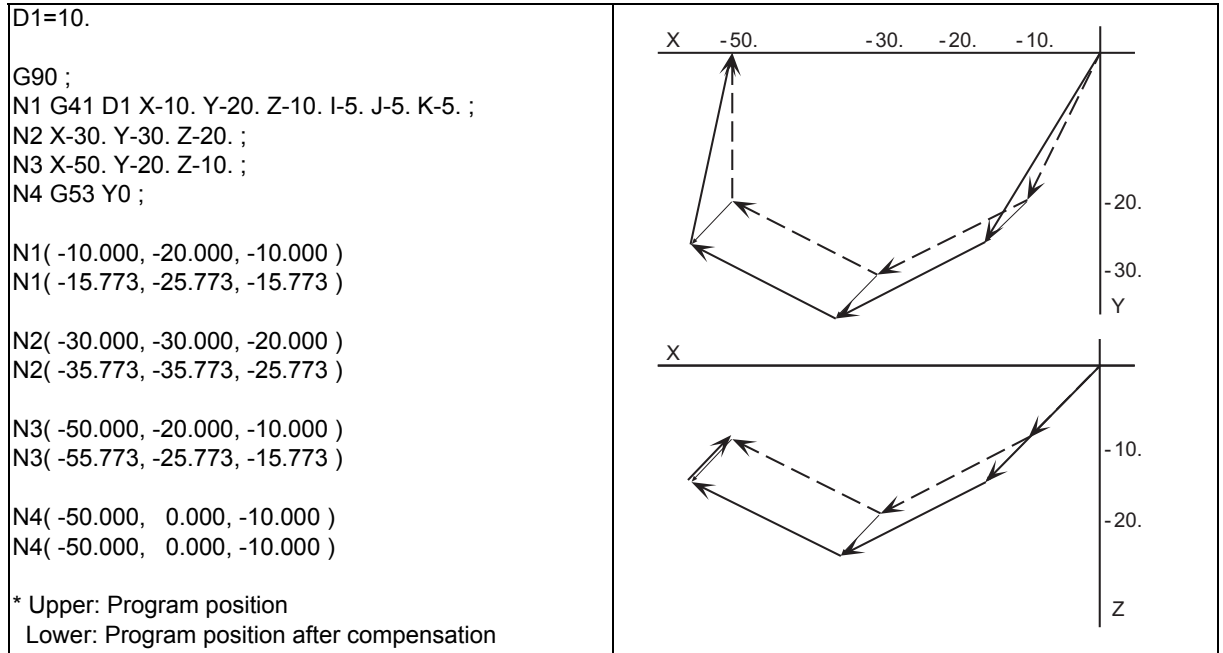
The compensation amount will not be canceled. Thus, if this is commanded during three-dimensional tool radius compensation, the path will be deviated by the compensation amount, thus the program error (P434) will occur.

Automatic corner override

Automatic corner override is invalid during three-dimensional tool radius compensation.

Machine coordinate system selection

(1) For the absolute command, all axes will be temporarily canceled at the commanded coordinate position.

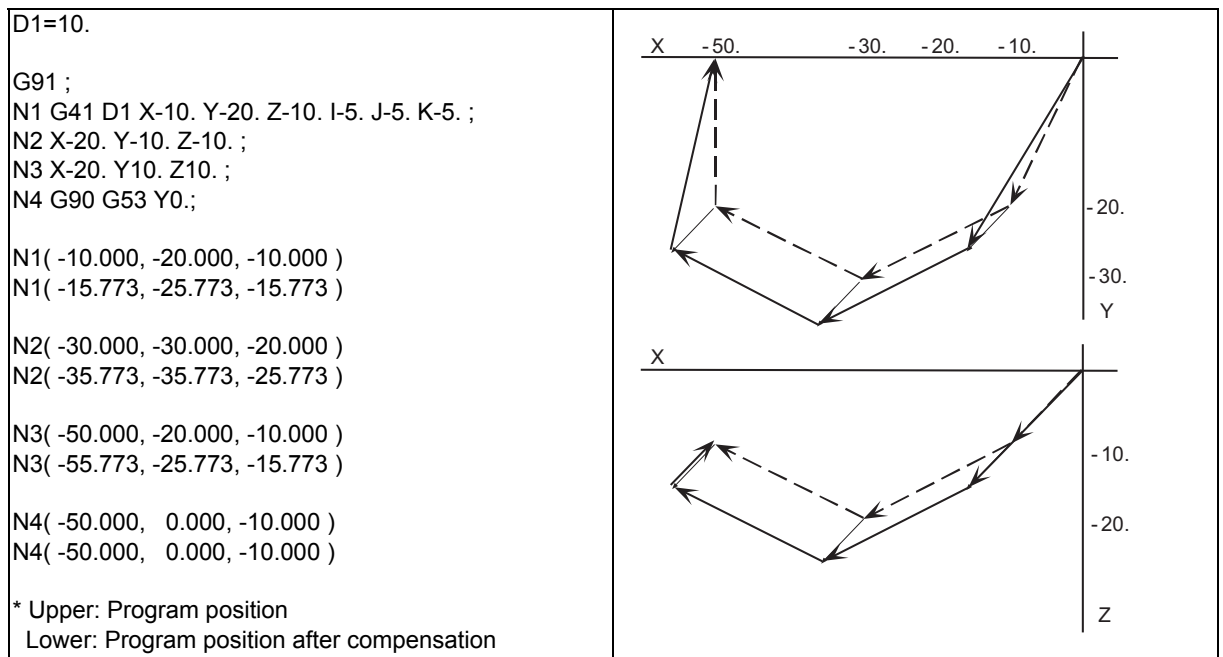


----Programmed path

——Program path after compensation

(2) For the incremental command, the axis will move by the amount obtained by subtracting each axis vector from the incremental movement amount.

(The compensation amount is temporarily canceled.)

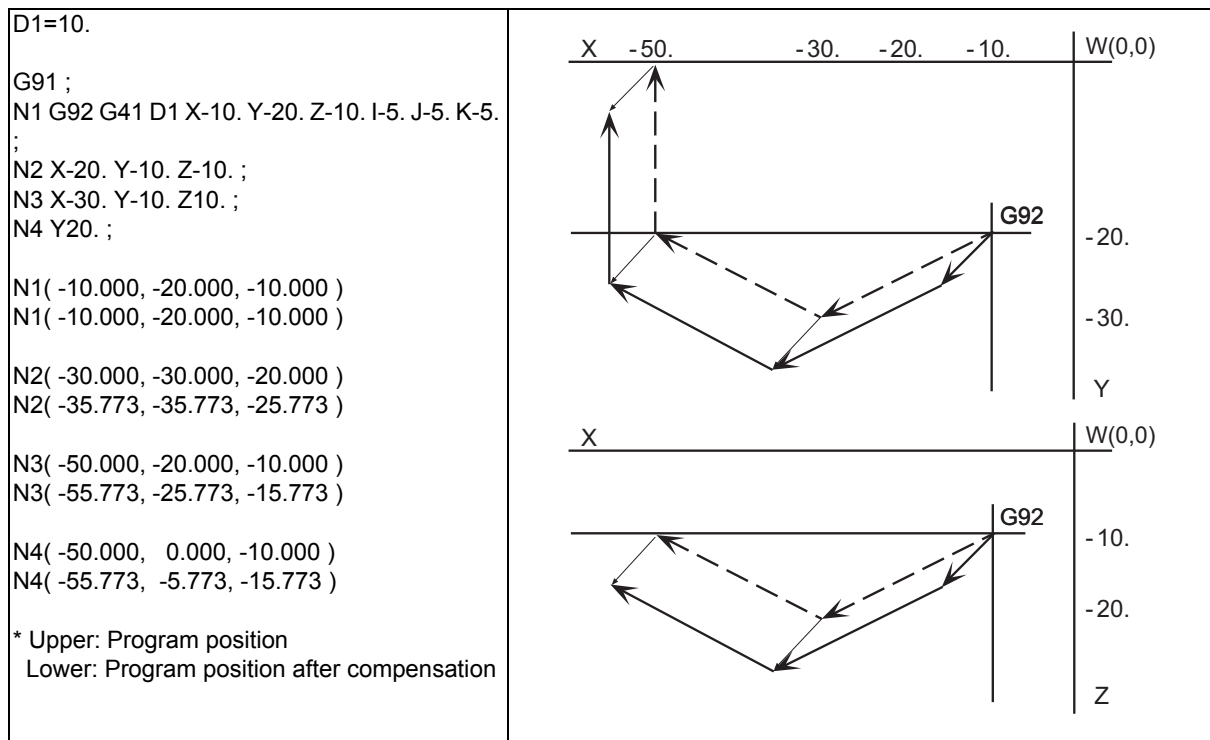


----Programmed path

——Program path after compensation

Coordinate system setting

When commanded in the same block as the coordinate system setting, the coordinate system will be set, and operation will start up independently with the plane normal line vector (I, J, K).



--- Programmed path

— Program path after compensation

Reference position return completed

All the axes will be temporarily canceled at the intermediate point.

```

D1=10.
G91 ;
N1 G41 D1 X-10. Y-20. Z-10. I-5. J-5. K-5. ;
N2 X-20. Y-10. Z-10. ;
N3 X-20. Y10. Z10. ;
N4 G28 X0 Y0 Z0 ;

N1( -10.000, -20.000, -10.000 )
N1( -15.773, -25.773, -15.773 )

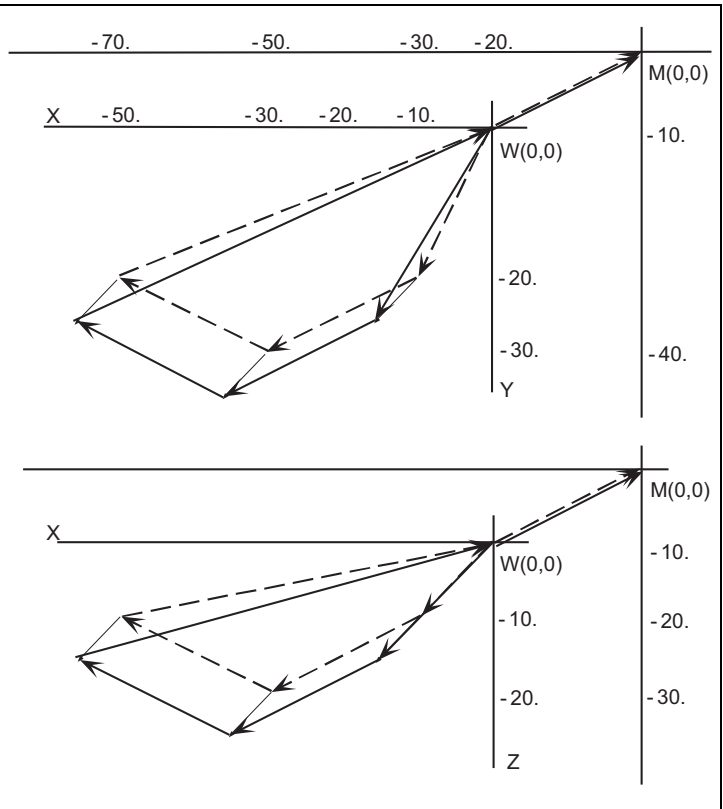
N2( -30.000, -30.000, -20.000 )
N2( -35.773, -35.773, -25.773 )

N3( -50.000, -20.000, -10.000 )
N3( -55.773, -25.773, -15.773 )

N4( 0.000, 0.000, 0.000 )
N4( 0.000, 0.000, 0.000 )

N4( 20.000, 10.000, 10.000 )
N4( 20.000, 10.000, 10.000 )

* Upper: Workpiece coordinate position
  Lower: Program position after compensa-
  tion
    
```



- Programmed path
- Program path after compensation

NC reset

Three-dimensional tool radius compensation will be canceled if NC reset is executed during three-dimensional tool radius compensation.

Emergency stop

Three-dimensional tool radius compensation will be canceled by the emergency stop or emergency stop cancel during three-dimensional tool radius compensation.



Restrictions

- (1) The compensation No. is selected with the D address, however, the D address is valid only when G41 or G42 is commanded. If D is not commanded, the number of the previous D address will be valid.
- (2) Switch the mode to the compensation mode in the G00 or G01 mode. When changed during the arc mode, the program error (P150) will occur.
The compensation direction and compensation amount after the mode change will become valid from the block where I, J and K are commanded in the G00 or G01 mode. If three-dimensional tool radius compensation is commanded in a block not containing the plane normal line vector (I, J, K) during the arc mode, only the modal information will be changed.
The plane normal line vector will be validated from the block where I, J and K are commanded next.
- (3) During the 3-dimensional tool radius compensation mode in a certain space, it is not possible to switch the space to another one and to execute three-dimensional tool radius compensation. To switch the compensation space, always cancel the compensation mode with G40 or D00 first.

(Example)

G41 Xx Yy Zz Ii Jj Kk ;	Compensation starts in X, Y, Z space.
:	:
:	:
G41 Uu Yy Zz Ii Jj Kk ;	Compensation is carried out in X, Y, Z space, and U axis moves by commanded value.

- (4) If the compensation No. D is other than the range of 1 to 40 with the standard specifications or 1 to 800 (max.) with the additional specifications, the program error (P170) will occur.
- (5) Only the G40 and D00 commands can be used to cancel 3-dimensional tool radius compensation.
- (6) If the size ($I^2+J^2+K^2$) of the vector commanded with I, J and K overflows, the program error (P35) will occur.

12.6 Tool Position Offset; G45 to G48



Function and purpose

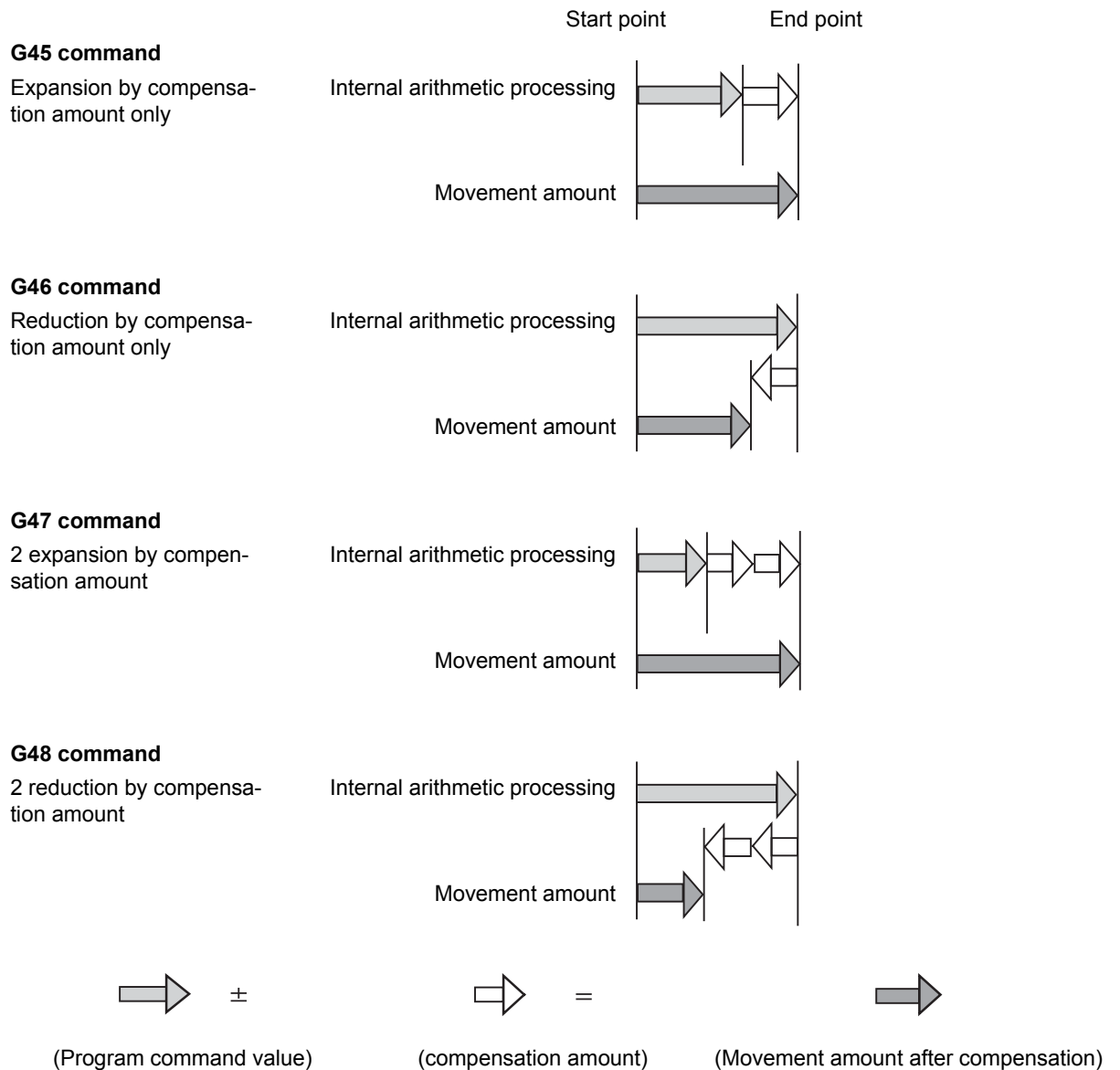
Using the G45 to G46 commands, the movement distance of the axes specified in the same block can be extended or reduced by a preset compensation length.

Furthermore, the compensation amount can be similarly doubled (x 2 expansion) or halved (x 2 reduction) with commands G47 and G48.

The number of sets for the compensation differ according to machine specification. Refer to Specifications Manual.

D01 to Dn

(The numbers given are the total number of sets for tool length compensation, tool position compensation and tool radius compensation.)





Command format

G45 X__ Y__ Z__ D__ ;	Expansion of movement amount by compensation amount set in compensation memory
-----------------------	--

G46 X__ Y__ Z__ D__ ;	Reduction of movement amount by compensation amount set in compensation memory
-----------------------	--

G47 X__ Y__ Z__ D__ ;	Expansion of movement amount by double the compensation amount set in compensation memory
-----------------------	---

G48 X__ Y__ Z__ D__ ;	Reduction of movement amount by double the compensation amount set in compensation memory
-----------------------	---

X, Y, Z	Movement amount of each axis
D	Tool compensation No.



Detailed description

Details for incremental values are given below.

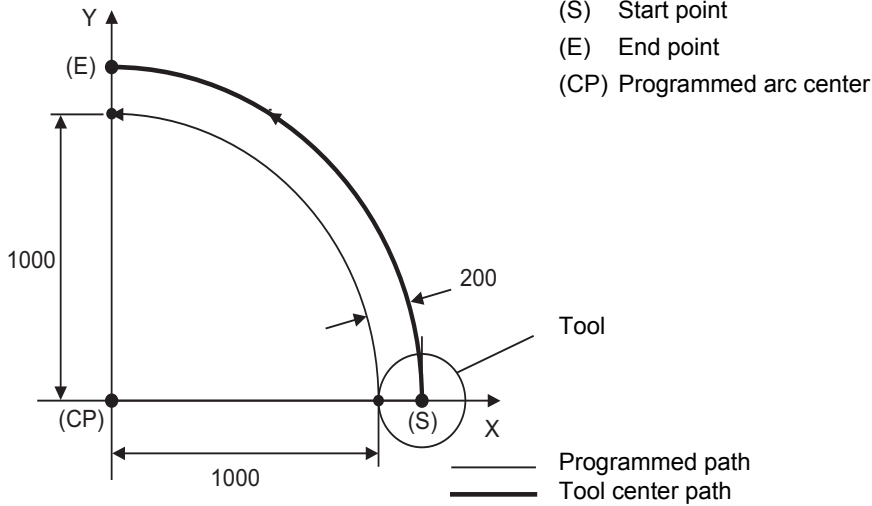
Command	Movement amount of equivalent command (assigned compensation amount = l)	Example (when X = 1000)
G45 Xx Dd	$X(x+l)$	l= 10 X= 1010 l= -10 X= 990
G45 X-x Dd	$X-(x+l)$	l= 10 X= -1010 l= -10 X= -990
G46 Xx Dd	$X(x-l)$	l= 10 X= 990 l= -10 X= 1010
G46 X-x Dd	$X-(x-l)$	l= 10 X= -990 l= -10 X= -1010
G47 Xx Dd	$X(x+2*l)$	l= 10 X= 1020 l= -10 X= 980
G47 X-x Dd	$X-(x+2*l)$	l= 10 X= -1020 l= -10 X= -980
G48 Xx Dd	$X(x-2*l)$	l= 10 X= 980 l= -10 X= 1020
G48 X-x Dd	$X-(x-2*l)$	l= 10 X= -980 l= -10 X= -1020



Program example

(Example 1)

Tool position compensation with 1/4 arc command



It is assumed that compensation has already been provided in the + X direction by D01 = 200.

```
G91 G45 G03 X -1000 Y1000 I -1000 F1000 D01 ;
```

Even if the compensation numbers are not assigned in the same block as the G45 to G48 commands, compensation is provided with the tool position compensation number previously stored in the memory.

If the commanded compensation No. exceeds the specification range, the program error (P170) will occur.

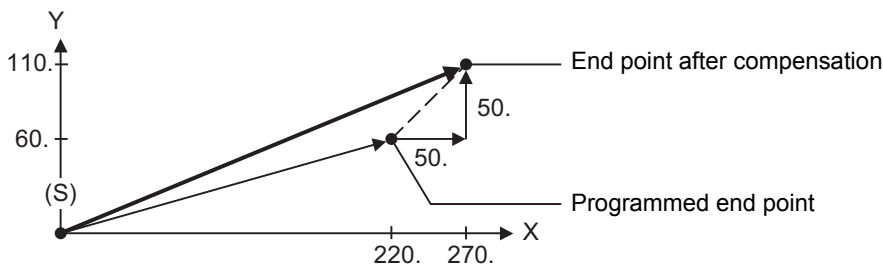
These G codes are unmodal and are effective only in the command block.

Even with an absolute value command, the amount of the movement is extended or reduced for each axis with respect to the direction of movement from the end point of the preceding block to the position assigned by the G45 to G48 block.

In other words, even for an absolute value command, compensation can be applied to movement amounts (incremental values) in the same block.

When a command for "n" number of simultaneous axes is given, the same compensation will be applied to all axes. It is valid even for the additional axes. (but it must be within the range of the number of axes that can be controlled simultaneously.)

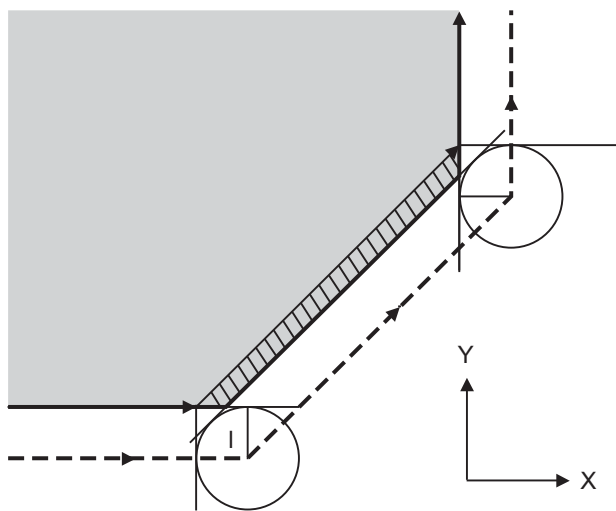
```
G01 G45 X220. Y60. D20 ;  
(D20) = +50. 000
```



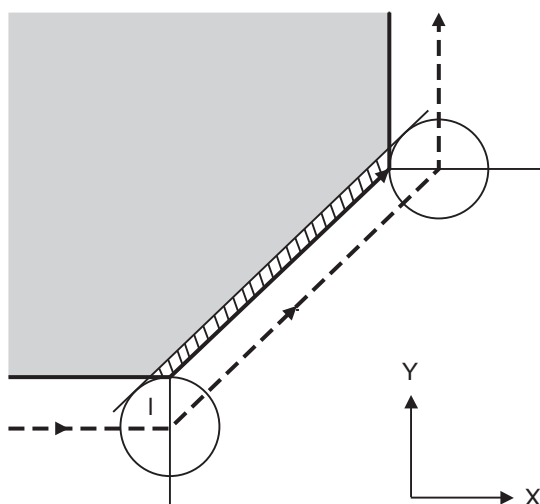
(S) Start point

Note

(1) If compensation is applied to two axes, over-cutting or under-cutting will result, as shown in the figures below. In cases like this, use the cutter compensation commands (G40 to G42).



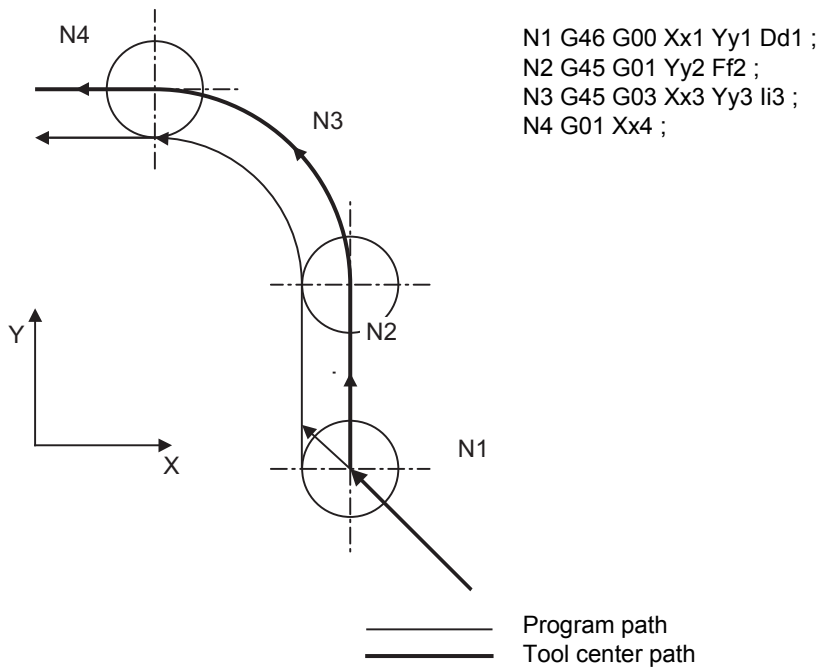
```
G01 G45 Xx 1 Dd1 ;
      Xx2 Yy 2 ;
G45 Yy3 ;
```



```
G01 Xx1 ;
G45 Xx2 Yy2 Dd 2 ;
Yy3 ;
```

- Programmed path
- Tool center path
- Workpiece
- Undercutting (figure above) or overcutting (figure below)
- I = Compensation amount setting

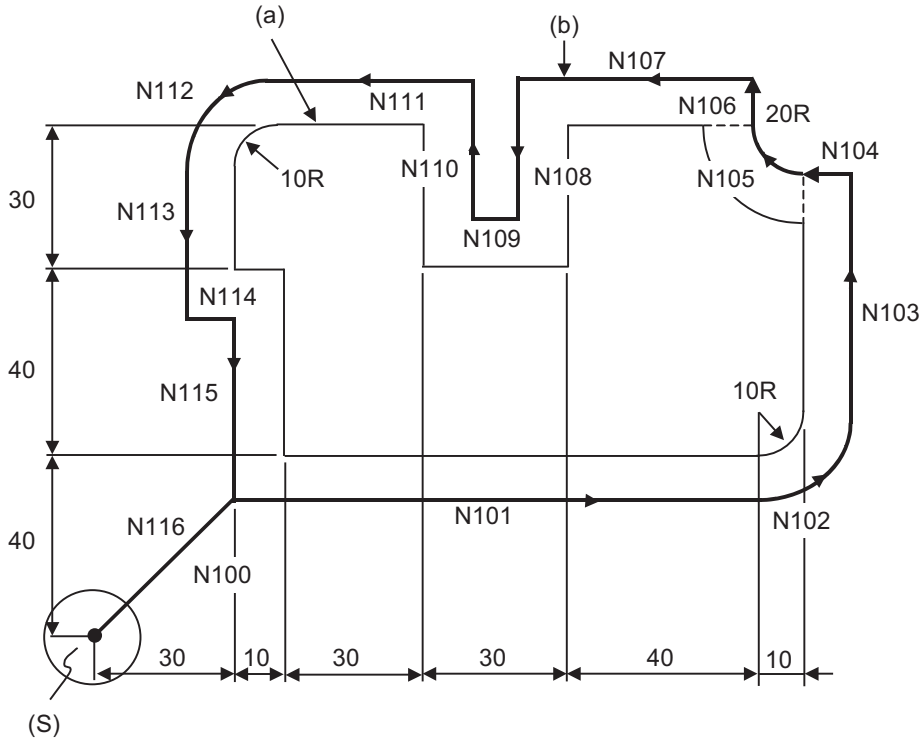
(Example 2)



12 Tool Compensation Functions

(Example 3)

When the G45 to G48 command is assigned, the compensation amount for each pass is the amount assigned by the compensation number, and the tool does not move for the difference from the previous compensation as it would do with the tool length compensation command (G43).



(S) Start point

(a) Programmed path

(b) Tool center path

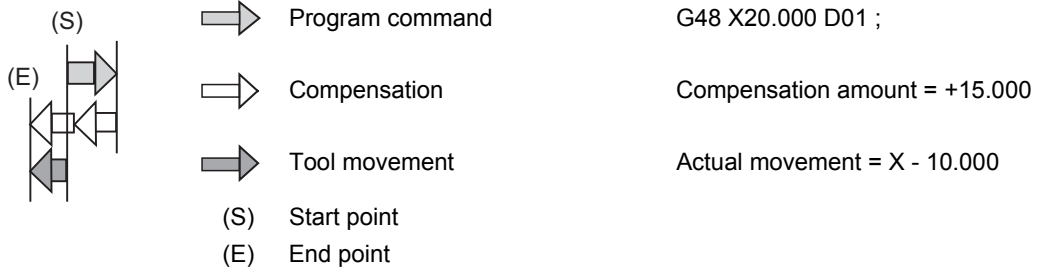
Compensation amount D01 = 10.000mm (Compensation amount of tool radius)

N100	G91	G46	G00	X40.0	Y40.0	D01 ;
N101	G45	G01	X100.0	F200 ;		
N102	G45	G03	X10.0	Y10.0	J10.0 ;	
N103	G45	G01	Y40.0 ;			
N104	G46	X0 ;				
N105	G46	G02	X-20.0	Y20.0	J20.0 ;	
N106	G45	G01	Y0 ;			
N107	G47	X-30.0 ;				
N108		Y-30.0 ;				
N109	G48	X-30.0 ;				
N110		Y30.0 ;				
N111	G45	X-30.0 ;				
N112	G45	G03	X-10.0	Y-10.0	J-10.0 ;	
N113	G45	G01	Y-20.0 ;			
N114		X10.0 ;				
N115		Y-40.0 ;				
N116	G46	X-40.0	Y-40.0 ;			
N117	M02 ;					
%						



Precautions

- (1) These commands should be used when operation is not in a fixed cycle mode. (They are ignored even if they are assigned during a fixed cycle.)
- (2) As a result of the internal arithmetic processing based on the expansion or reduction, the tool will proceed to move in the opposite direction when the command direction is reversed.



- (3) When a zero movement amount has been specified in the incremental value command (G91) mode, the result is as follows.

Compensation No.				: D01
Compensation amount corresponding to D01				: 1234
NC command	G45 X0 D01 ;	G45 X-0 D01 ;	G46 X0 D01 ;	G46 X-0 D01 ;
Equivalent command	X1234;	X -1234 ;	X -1234 ;	X1234;

When a zero movement amount has been specified with an absolute value command, the operation is completed immediately and the tool does not move for the compensation amount.

12 Tool Compensation Functions

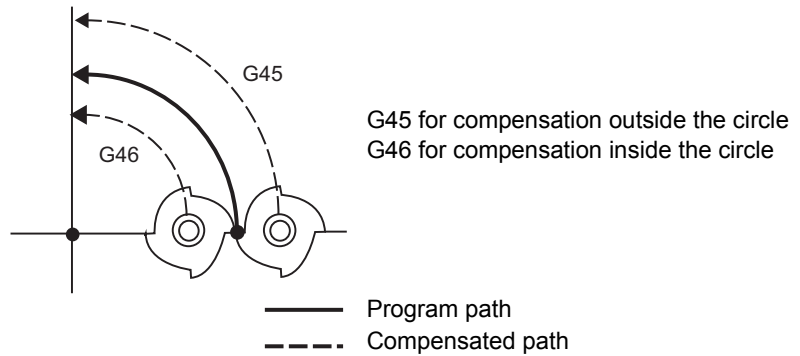
(4) In the case of circular interpolation, tool radius compensation is possible by the G45 to G48 commands only for one quadrant, two quadrants (semi sphere) or three quadrants when the start and end points are on the axis.

The commands are assigned as follows depending on whether the compensation is applied for outside or inside the arc programmed path.

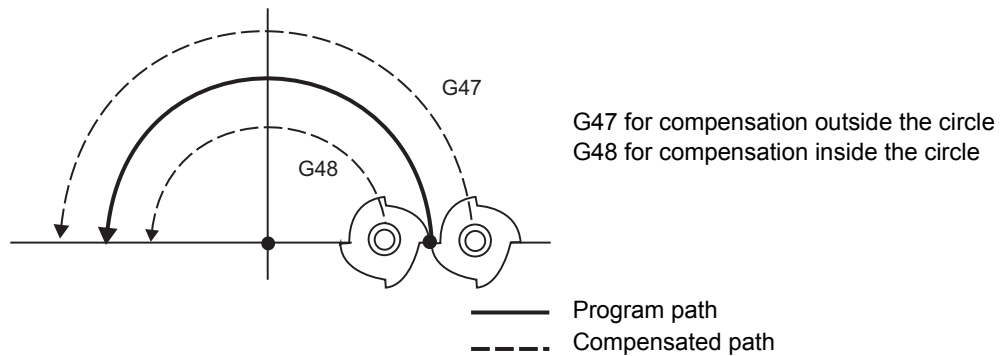
However, in this case, compensation must already be provided in the desired direction at the arc start point. (If a compensation command is assigned for the arc independently, the arc start point and end point radius will shift by an amount equivalent to the compensation amount.)

The program path is indicated by the heavy line in the figure.

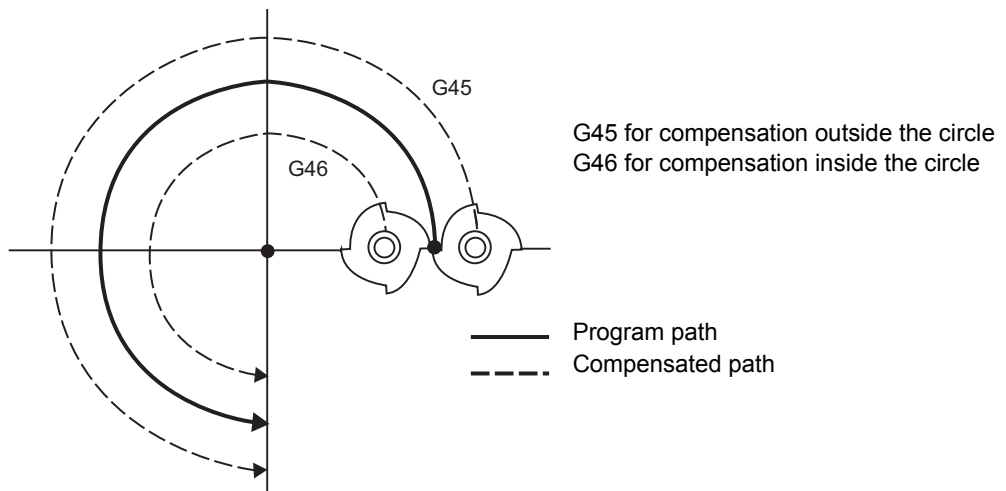
1/4 circle



1/2 circle



3/4 circle



Fixed Cycle

13.1 Fixed Cycles



Function and purpose

These fixed cycles are used to perform prepared sequences of machining programs, such as positioning, hole drilling, boring and tapping in one block. The available machining sequences are listed in the table below.

By editing the standard fixed cycle subprograms, the fixed cycle sequences can be changed by the user. The user can also register and edit an original fixed cycle program. For the standard fixed cycle subprograms, refer to the list of the fixed cycle subprograms in the appendix of the operation manual. The list of fixed cycle functions for this control unit is shown below.

G code	Hole drilling start (-Z direction)	Operation at hole bottom		Return operation (+Z direction)	Retract at high speed	Application
		Dwell	Spindle			
G80	-	-	-	-	-	Cancel
G81	Cutting feed	-	-	Rapid traverse	Possible	Drill, spot drilling cycle
G82	Cutting feed	Yes	-	Rapid traverse	-	Drill, counter boring cycle
G83	Intermittent feed	-	-	Rapid traverse	Possible	Deep hole drilling cycle
G84	Cutting feed	Yes	Reverse rotation	Cutting feed	-	Tapping cycle
G85	Cutting feed	-	-	Cutting feed	-	Boring cycle
G86	Cutting feed	Yes	Stop	Rapid traverse	-	Boring cycle
G87	Rapid traverse	-	Forward rotation	Cutting feed	-	Back boring cycle
G88	Cutting feed	Yes	Stop	Rapid traverse	-	Boring cycle
G89	Cutting feed	Yes	-	Cutting feed	-	Boring cycle
G73	Intermittent feed	Yes	-	Rapid traverse	Possible	Stepping cycle
G74	Cutting feed	Yes	Forward rotation	Cutting feed	-	Reverse tapping cycle
G75	Cutting feed	-	-	Rapid traverse	-	Circular cutting cycle
G76	Cutting feed	-	Oriented spindle stop	Rapid traverse	-	Fine boring cycle
G187	Cutting feed	-	-	Rapid traverse	-	Thread milling cycle

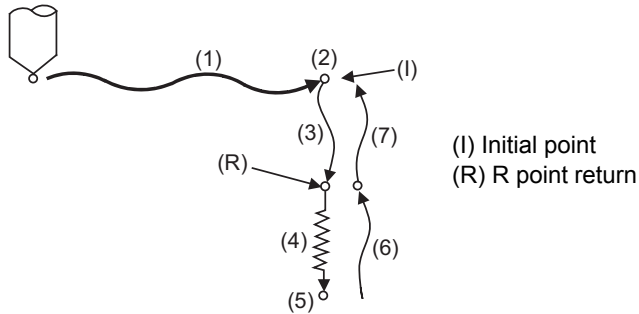
A fixed cycle mode can be canceled by G80 command and other hole machining modes or G command in the 01 group. At the same time, various other data will also be cleared to zero.



Detailed description

Basic operations of fixed cycle for drilling

There are 7 actual operations which are each described below.



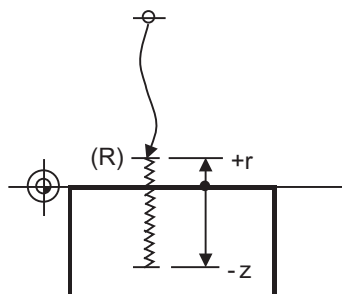
- (1) This indicates the X and Y axes positioning, and executes positioning with G00.
- (2) This is an operation done after positioning is completed (at the initial point), and when G87 is commanded, the M19 command is output from the control unit to the machine. When this M command is executed and the finish signal (FIN) is received by the control unit, the next operation will start. If the single block stop switch is ON, the block will stop after positioning.
- (3) The tool is positioned to the R point by rapid traverse.
- (4) Hole machining is conducted by cutting feed.
- (5) This operation takes place at the hole bottom position, and depending on the fixed cycle mode, the operation can be the spindle stop (M05), the rotary tool reverse rotation (M04), rotary tool forward rotation (M03), dwell or tool shift.
- (6) The tool is retracted to the R point at the cutting feed or the rapid traverse rate, depending on the fixed cycle mode.
- (7) The tool is returned to the initial point at rapid traverse rate.

Note

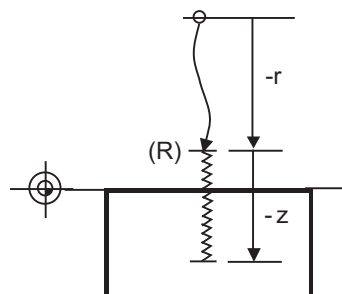
- (1) Whether the fixed cycle is to be completed at operation 6 or 7 can be selected by G98/G99 commands. (Refer to "Initial point and R point level return; G98, G99")

Difference between absolute value command and incremental value command

For absolute value



For incremental value



(R) R point

Positioning plane and hole drilling axis

The fixed cycle has basic control elements for the positioning plane and hole drilling axis. The positioning plane is determined by the G17, G18 and G19 plane selection commands, and the hole drilling axis is the axis perpendicular (X, Y, Z or their parallel axis) to the above plane.

Plane selection	Positioning plane	Hole drilling axis
G17 (X-Y)	Xp-Yp	Zp
G18 (Z-X)	Zp-Xp	Yp
G19 (Y-Z)	Yp-Zp	Xp

Xp, Yp and Zp indicate the basic axes X, Y and Z or an axis parallel to the basic axis.

An arbitrary axis other than the hole drilling axis can be commanded for positioning.

The hole drilling axis is determined by the axis address of the hole drilling axis commanded in the same block as G81 to G89, G73, G74 or G76. The basic axis will be the hole drilling axis if there is no designation.

(Example 1) When G17 (X-Y plane) is selected, and the axis parallel to the Z axis is set as the W axis.

G81 Z_ ; The Z axis is used as the hole drilling axis.
 G81 W_ ; The W axis is used as the hole drilling axis.
 G81 ; (No Z or W) The Z axis is used as the hole drilling axis.

Note

- (1) The hole drilling axis can be fixed to the Z axis with parameter #1080 Dril_Z.
- (2) Changeover of the hole drilling axis must be done with the fixed cycle canceled.

In the following explanations on the movement in each fixed cycle mode, the XY plane is used for the positioning plane and the Z axis for the hole drilling axis. Note that all command values will be incremental values, the positioning plane will be the XY plane and the hole drilling axis will be the Z axis.

Programmable in-position width command in fixed cycle

This commands the in-position width for commanding the fixed cycle from the machining program. The commanded in-position width is valid only in the eight fixed cycles; G81 (drill, spot drill), G82 (drill, counter boring), G83 (deep drill cycle), G84 (tap cycle), G85 (boring), G89 (boring), G73 (step cycle) and G74 (reverse tap cycle). The ",I" address is commanded in respect to the positioning axis, and the ",J" address is commanded in respect to the drilling axis.

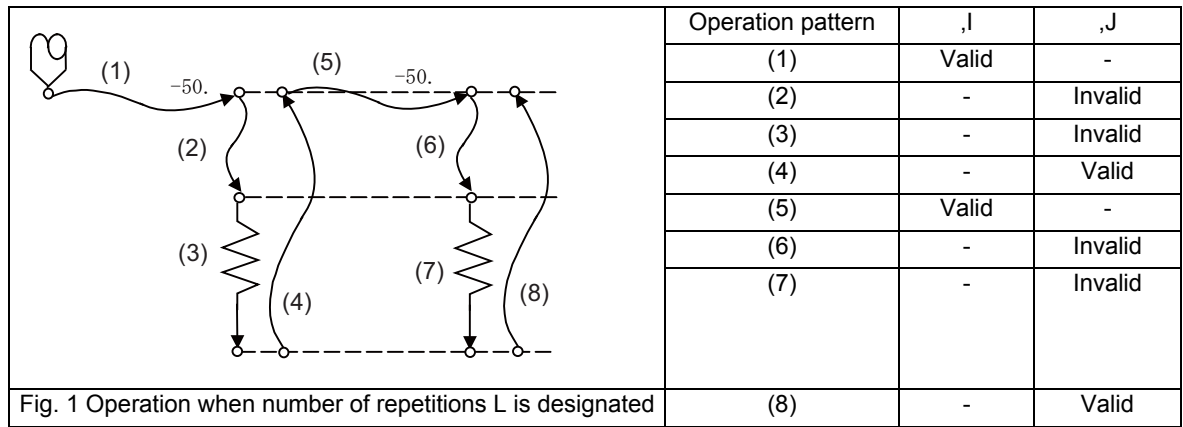
Address	Meaning of address	Command range (unit)	Remarks
,I	Positioning axis in-position width(position error amount)	0.001 to 999.999 (mm)	If a value exceeding the command range is commanded, a program error (P35) will occur.
,J	Drilling axis in-position width(position error amount)		

In-position check in fixed cycle

When L (number of repetitions) is designated twice or more times in the fixed cycle, the commanded in-position width will be valid in the repetition block (5) to (8) below.

```

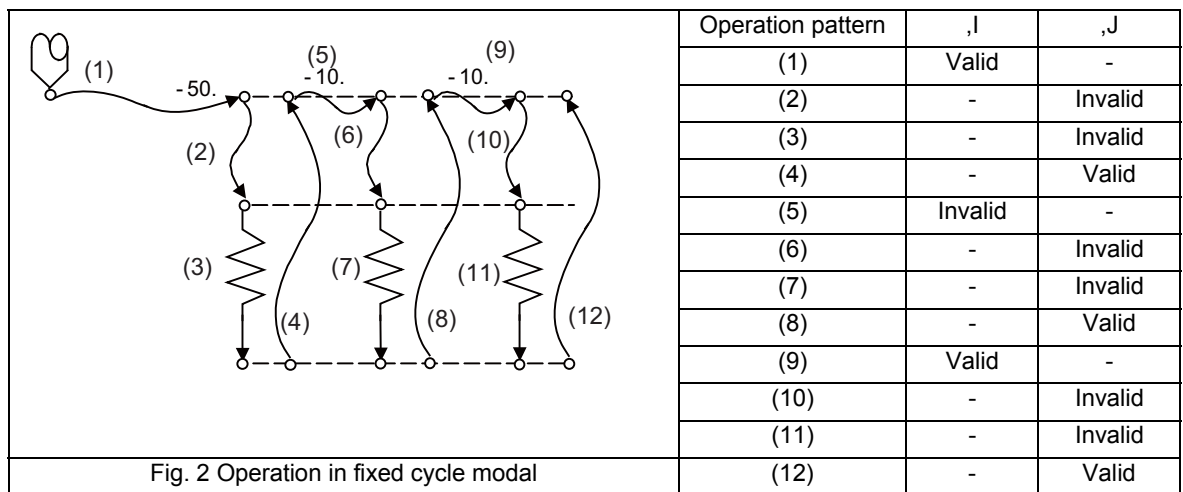
:
G91 G81 X-50. Z-50. R-50. L2 F2000 ,I0.2 ,J0.3;
:
    
```



In the following machining program, the commanded in-position width is valid for the Fig. 2 block. In the (B) block, the in-position width (, I) commanded regarding to positioning in the previous block (A) is invalid (5). However, when returning from the hole bottom, the in-position width (, J) commanded in the previous block (A) is valid (8). To validate the in-position width for positioning, command again as shown in block (C) (9).

```

:
G91 G81 X-50. Z-50. R-50. F2000 ,I0.2 ,J0.3 ;           (A)
X-10. ; .                                               (B)
X-10.,I0.2 ;                                           (C)
    
```



13.1.1 Drilling, Spot Drilling; G81



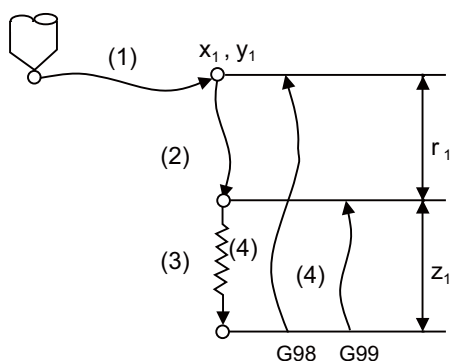
Command format

G81 Xx1 Yy1 Zz1 Rr1 Ff1 LI1,li1,Jj1;

Xx1	Designation of hole drilling position (absolute value or incremental value)
Yy1	Designation of hole drilling position (absolute value or incremental value)
Zz1	Designation of hole bottom position (absolute value or incremental value) (modal)
Rr1	Designation of R point position (absolute value or incremental value) (modal)
Ff1	Designation of feedrate for cutting feed (modal)
LI1	Designation of number of repetitions. (0 to 9999) When "0" is set, no execution
,li1	Positioning axis in-position width
,Jj1	Drilling axis in-position width



Detailed description



Operation pattern	i1	j1	Program	
(1)	Valid	-	G00 Xx1 Yy1;	
(2)	-	Invalid	G00 Zr1;	
(3)	-	Invalid	G01 Zz1 Ff1;	
(4)	-	Valid	G00 Z- z1+r1;	G98 mode
			G00 Z-z1;	G99 mode

The operation stops at after the (1), (2) and (4) commands during single block operation.

13.1.2 Drilling, Counter Boring; G82



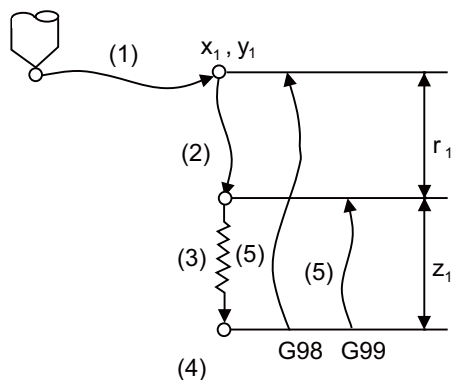
Command format

G82 Xx1 Yy1 Zz1 Rr1 Ff1 Pp1 L1 ,li1 ,Jj1;

Xx1	Designation of hole drilling position (absolute value or incremental value)
Yy1	Designation of hole drilling position (absolute value or incremental value)
Zz1	Designation of hole bottom position (absolute value or incremental value) (modal)
Rr1	Designation of R point position (absolute value or incremental value) (modal)
Ff1	Designation of feedrate for cutting feed (modal)
Pp1	Designation of dwell time at hole bottom position (the values after the decimal points will be ignored) (modal)
L1	Designation of number of repetitions (0 to 9999) When "0" is set, processing is not executed.
,li1	Positioning axis in-position width
,Jj1	Drilling axis in-position width



Detailed description



Operation pattern	i1	j1	Program	
(1)	Valid	-	G00 Xx1 Yy1;	
(2)	-	Invalid	G00 Zr1;	
(3)	-	Invalid	G01 Zz1 Ff1;	
(4)	-	-	G04 Pp1 ;	Dwell
(5)	-	Valid	G00 Z-(z1+r1);	G98 mode
			G00 Z-z1;	G99 mode

The operation stops at after the (1), (2) and (5) commands during single block operation.

13.1.3 Deep Hole Drilling Cycle; G83

13.1.3.1 Deep Hole Drilling Cycle

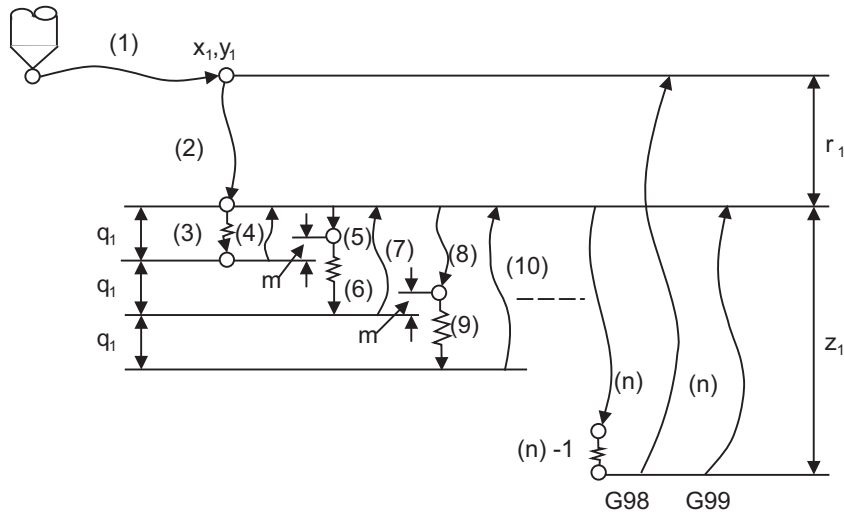
**Command format**

G83 Xx1 Yy1 Zz1 Rr1 Qq1 Ff1 LI1 ,li1 ,Jj1;

Xx1	Designation of hole drilling position (absolute value or incremental value)
Yy1	Designation of hole drilling position (absolute value or incremental value)
Zz1	Designation of hole bottom position (absolute value or incremental value) (modal)
Rr1	Designation of R point position (absolute value or incremental value) (modal)
Qq1	Cut amount for each cutting pass (incremental value) (modal)
Ff1	Designation of feedrate for cutting feed (modal)
LI1	Designation of number of repetitions. (0 to 9999) When "0" is set, no execution
,li1	Positioning axis in-position width
,Jj1	Drilling axis in-position width



Detailed description



Operation pattern	i1	j1	Program	
(1)	Valid	-	G00 Xx1 Yy1;	
(2)	-	Invalid	G00 Zr1;	
(3)	-	Invalid	G01 Zq1 Ff1;	
(4)	-	Invalid	G00 Z-q1;	
(5)	-	Invalid	G00 Z(q1-m);	
(6)	-	Invalid	G01 Z(q1+m) Ff1;	
(7)	-	Invalid	G00 Z-2*q1;	
(8)	-	Invalid	G00 Z(2*q1-m);	
(9)	-	Invalid	G01 Z(q1+m) Ff1;	
(10)	-	Invalid	G00 Z-3*q1;	
:				
(n)-1	-	Invalid		
(n)	-	Valid	G00 Z-(z1+r1);	G98 mode
			G00 Z-z1;	G99 mode

When executing a second and following cuttings in the G83 as shown above, the movement will change from rapid traverse to cutting feed "m" mm before the position machined last. After reaching the hole bottom, the axis will return according to the G98 or G99 mode.

"m" will differ according to the parameter "#8013 G83 return". Program so that $q1 > m$.

The operation stops at after the (1), (2) and (n) commands during single block operation.

13.1.3.2 Small Diameter Deep Hole Drilling Cycle



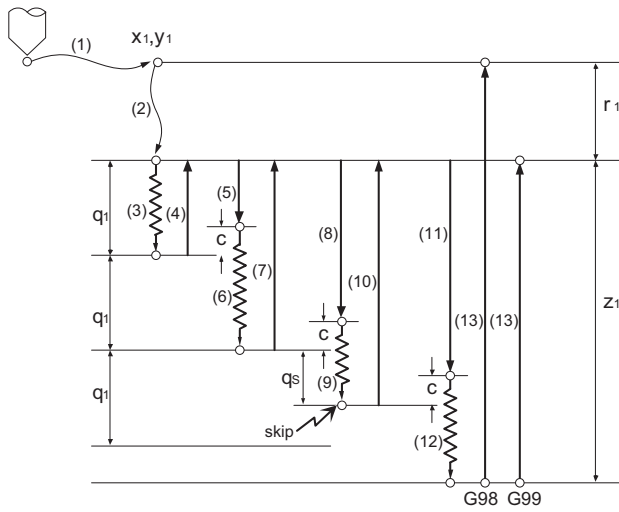
Command format

G83 Xx1 Yy1 Zz1 Rr1 Qq1 Ff1 li1 Pp1;

Xx1	Designation of hole drilling position (absolute value or incremental value)
Yy1	Designation of hole drilling position (absolute value or incremental value)
Zz1	Designation of hole bottom position (absolute value or incremental value) (modal)
Rr1	Designation of R point position (absolute value or incremental value) (modal)
Qq1	Cut amount for each cutting pass (incremental value) (modal)
Ff1	Designation of feedrate for cutting feed (modal)
li1	The feedrate from R point to the cutting start position, the speed (mm/min) for returning from the hole bottom are stored only in the same block as G83, and it is valid until the small diameter deep hole drilling cycle is canceled. (It follows the setting of "#8085 G83S Forward F", "#8086 G83S Back F" when omitted.)
Pp1	Dwell time at hole bottom position



Detailed description



- c: Parameter "#8084 G83S Clearance"
- qs: Cut amount when small diameter deep hole drilling cycle signal (YCCA) is input
- skip: Small diameter deep hole drilling cycle signal (YCCA) input

Operation pattern	Program	
(1)	G00 Xx1 Yy1,li1;	
(2)	G00 Zr1;	
(3)	G01 Zq1 Ff1;	
(4)	G01 Z-q1 Fi2;	
(5)	G01 Z(q1-c) Fi1;	
(6)	G01 Z(q1+c) Ff1;	
(7)	G01 Z-2•q1 Fi2;	
(8)	G01 Z(2•q1-c) Fi1;	
(9)	G01 Z(q1+c) Ff1;	
(10)	G01 Z-(2•q1+qs) Fi2;	
(11)	G01 Z(2•q1+qs-c) Fi1;	
(12)	G01 Z(z1-q1•n-qs) Ff1;	
(13)	G01 Z-(z1+r1) Fi2;	G98 mode
	G01 Z-z1 Fi2;	G99 mode

"i1" follows the parameter "#8085 G83S Forward F" when there is no I command.

"i2" follows the parameter "#8086 G83S Back F" when there is no I command.

In deep hole drilling, cutting and retract are repeated and the workpiece is machined multiple times. In addition, when PLC signals are input during cutting, the cutting for the time concerned is skipped. In this way, this cycle reduces the load applied to the tool.

The small-diameter deep-hole drilling cycle mode is established by designating the M code command that was set in the parameter "#8083 G83S mode M".

If the G83 command is designated in this mode, the small-diameter deep-hole drilling cycle is executed.

The mode is canceled by the following conditions.

- ◆Designation of a fixed cycle cancel command (G80, G commands in Group 1)
- ◆Resetting

It is not immediately switched to the small diameter deep hole drilling cycle mode even the small diameter deep hole drilling cycle switch M command is issued during G83 deep hole drilling cycle modal. Then, when G83 is commanded, the small diameter deep hole drilling cycle mode is applied.

When the small diameter deep hole drilling cycle signal (YCCA) is input during the cutting operation (9), the remaining cutting command is skipped and the axis returns to the R point at the cutting speed "i2".

"In small diameter deep hole cycle signal (XCC1)" is output between the positioning to the R point of drilling axis (2) and the R point/initial point return after finishing the drilling (13).

"c" depends on the parameter "#8084 G83S Clearance".

Program the small diameter deep hole drilling cycle to make it "q1 > c".

The operation stops at after the (1), (2) and (13) commands during single block operation.

If there is no "I" command, or either the parameter "#8085 G83S Forward F" or "#8086 G83S Back F" is set to "0", a program error (P62) will occur.

Confirm the following related parameters before using the small hole diameter drilling cycle.

- ◆#8083 G83S modeM
- ◆#8084 G83S Clearance
- ◆#8085 G83S Forward F
- ◆#8086 G83S Back F

13.1.4 Tapping Cycle; G84



Command format

G84 Xx1 Yy1 Zz1 Rr1 Qq1 Ff1(Ee1) Pp1, Rr2 Ss1 ,Ss2 ,li1 ,Jj1 LI1 (Kk1);
--

Xx1	Designation of hole drilling position (absolute value or incremental value)
Yy1	Designation of hole drilling position (absolute value or incremental value)
Zz1	Designation of hole bottom position (absolute value or incremental value) (modal)
Rr1	Designation of R point position (absolute value or incremental value) (modal)
Qq1	Cut amount for each cutting pass (incremental value) (modal)
Ff1	During synchronous tapping: Designation of drilling axis feed amount (tapping pitch) per spindle revolution (modal) During asynchronous tapping: Designation of the feedrate for cutting feed (modal)
Ee	Cutting feedrate at synchronous tapping (Number of screw threads per inch) If this command is issued simultaneously with the F command, the F command is valid.
Pp1	Designation of dwell time at hole bottom position (the values after the decimal points will be ignored) (modal)
,Rr2	Synchronization method selection (r2=1 synchronous, r2=0 asynchronous) (When omitted, the mode will follow the setting of parameter "#8159 Synchronous tap")
Ss1	Spindle rotation speed command <Note> •At a synchronous tapping mode, "Sn = *****" type S command will be ignored. (n:spindle number, *****: rotation speed) •If an S command is issued during synchronous tapping modal, a program error (P186) will occur.
,Ss2	Spindle rotation speed during return
,li1	Positioning axis in-position width
,Jj1	Drilling axis in-position width
LI1	Designation of number of repetitions (0 to 9999) Not executed when "0" is set.
Kk1	Number of repetitions (It can be commanded when the parameter "#1271 ext07/bit1" is "1")

Note

(1) ",S" command is held as a modal information.

When the value of the ",S" command is lower than the spindle rotation speed (S command), operations depend on the MTB specifications.

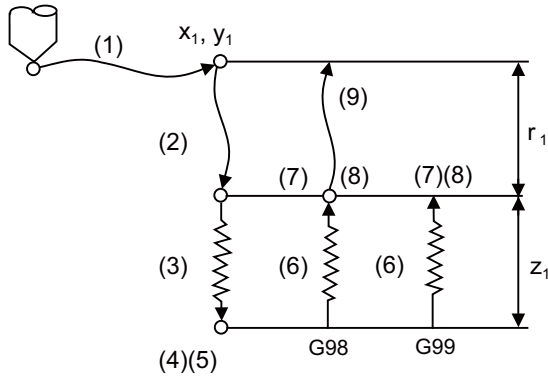
(Whether it operates on the spindle rotation speed at the return or on the spindle rotation speed of S command is determined according to the parameter "#1241 set13/bit7 setting.)

When the spindle speed at the return is not "0", the value of tap return override (#1172 tapovr) is invalid.



Detailed description

Normal tapping cycle (When Q is not designated)



Operation pattern	i1	j1	Program	
(1)	Valid	-	G00 Xx1 Yy1;	
(2)	-	Invalid	G00 Zr1;	
(3)	-	Invalid	G01 Zz1 Ff1;	
(4)	-	-	G04 Pp1;	
(5)	-	-	M4;	Spindle forward rotation
(6)	-	Invalid	G01 Z-z1 Ff1;	
(7)	-	-	G04 Pp1;	
(8)	-	-	M3;	Spindle forward rotation
(9)	-	Valid	G00 Z-r1;	G98 mode
			No movement	G99 mode

When r2 = 1, the synchronous tapping mode will be applied, and when r2 = 0, the asynchronous tapping mode will be applied. If there is no r2 command, mode will follow the parameter setting.

When G84 is executed, the override will be canceled and automatically be set to 100%.

Dry run is valid for the positioning command when the control parameter "G00 DRY RUN" is ON. If the feed hold button is pressed during G84 execution, and the sequence is at (3) to (6), the movement will not stop immediately, and instead will stop after (6). During the rapid traverse in sequence (1), (2) and (9), the movement will stop immediately.

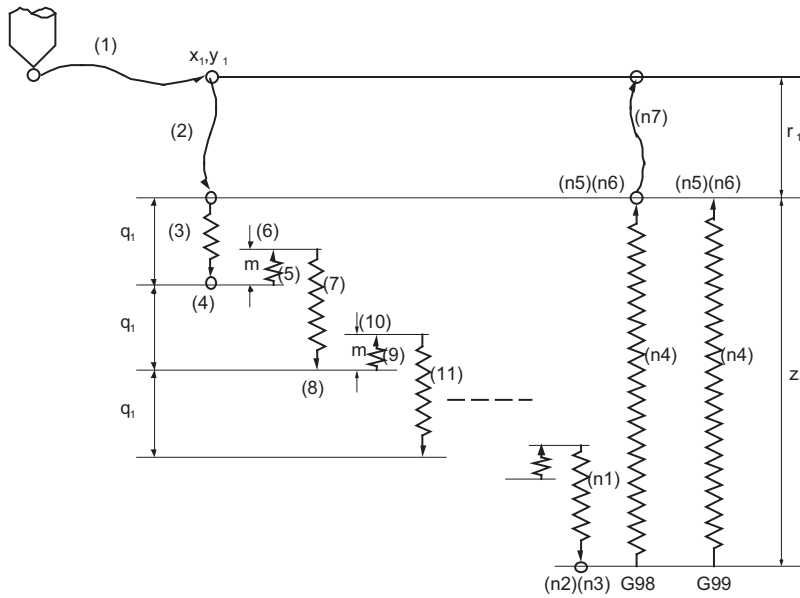
The operation stops at after the (1), (2) and (9) commands during single block operation.

During the G84 mode, the NC signal "Tapping" will be output.

During the G84 synchronous tapping modal, the M3, M4, M5 and S code will not be output.

When it is interrupted by such as the emergency stop during the tapping cycle, enable the "Tap retract" signal (TRV); a tool can be taken out from the workpiece by tap retract operation.

Pecking Tapping Cycle (When the Q command is designated #1272 ext08/bit4=0)



m : parameter (#8018 G84/G74 n)

Note

•This program is for the G84 command.

The spindle forward rotation (M3) and reverse rotation (M4) are reversed with the G74 command.

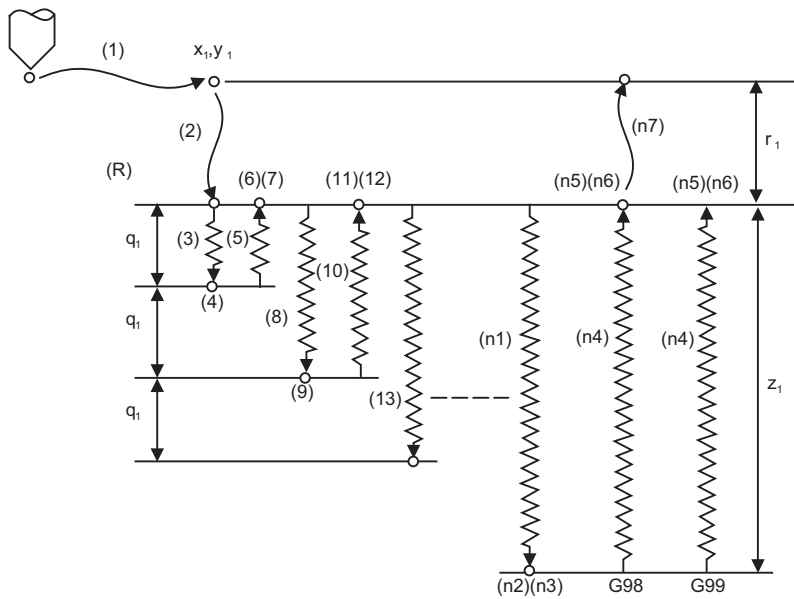
Operation pattern	Program	
(1)	G00 Xx1 Yy1 ,li1;	
(2)	G00 Zr1;	
(3)	G01 Zq1 Ff1;	
(4)	M4;	Spindle reverse rotation
(5)	G01 Z-m Ff1;	
(6)	M3 ;	Spindle forward rotation
(7)	G01 Z(q1+m) Ff1;	
(8)	M4;	Spindle reverse rotation
(9)	G01 Z-m Ff1;	
(10)	M3 ;	Spindle forward rotation
(11)	G01 Z(q1+m) Ff1;	
:	:	
(n1)	G01 Z(z1-q1*n) Ff1;	
(n2)	G04 Pp1;	
(n3)	M4;	Spindle reverse rotation
(n4)	G01 Z-z1 Ff1 Ss2;	
(n5)	G04 Pp1;	
(n6)	M3;	Spindle forward rotation
(n7)	G00 Z-r1 ,Jj1;	G98 mode
	No movement	G99 mode

The load applied to the tool can be reduced by designating the depth of cut per pass (Q) and cutting the workpiece to the hole bottom for a multiple number of passes. The amount retracted from the hole bottom is set to the parameter "#8018 G84/G74 n". Whether the pecking tapping cycle or the deep-hole tapping cycle is valid depends on the MTB specifications (parameter "#1272 ext08/bit4"). When "depth of cut per pass Q" is designated in the block containing the G84 or G74 command with the pecking tapping cycle selected, the pecking tapping cycle will be executed.

In the following cases, the normal tapping cycle will be carried out.

- When Q is not designated.
- When the command value of Q is zero.

Deep-hole Tapping Cycle (When the Q command is designated #1272 ext08/bit4=1)



(R) R point

Note

(1) This program is for the G84 command.

The spindle forward rotation (M3) and reverse rotation (M4) are reversed with the G74 command.

Operation pattern	Program	
(1)	G00 Xx1 Yy1;	
(2)	G00 Zr1;	
(3)	G09 G01 Zq1 Ff1;	
(4)	M4 ;	Spindle reverse rotation
(5)	G09 G01 Z-q1 Ff1;	
(6)	G04 Pp1;	
(7)	M3;	Spindle forward rotation
(8)	G09 G01 Z(2*q1) Ff1;	
(9)	M4 ;	Spindle reverse rotation
(10)	G09 G01 Z-(2*q1) Ff1;	
(11)	G04 Pp1;	
(12)	M3;	Spindle forward rotation
(13)	G09 G01 Z(3*q1) Ff1;	
:	:	
(n1)	G09 G01 Zz1 Ff1;	
(n2)	G04 Pp1;	

Operation pattern	Program	
(n3)	M4;	Spindle reverse rotation
(n4)	G09 G01 Z-z1 Ff1;	
(n5)	G04 Pp1;	
(n6)	M3;	Spindle forward rotation
(n7)	G00 Z-r1;	G98 mode
	No movement	G99 mode

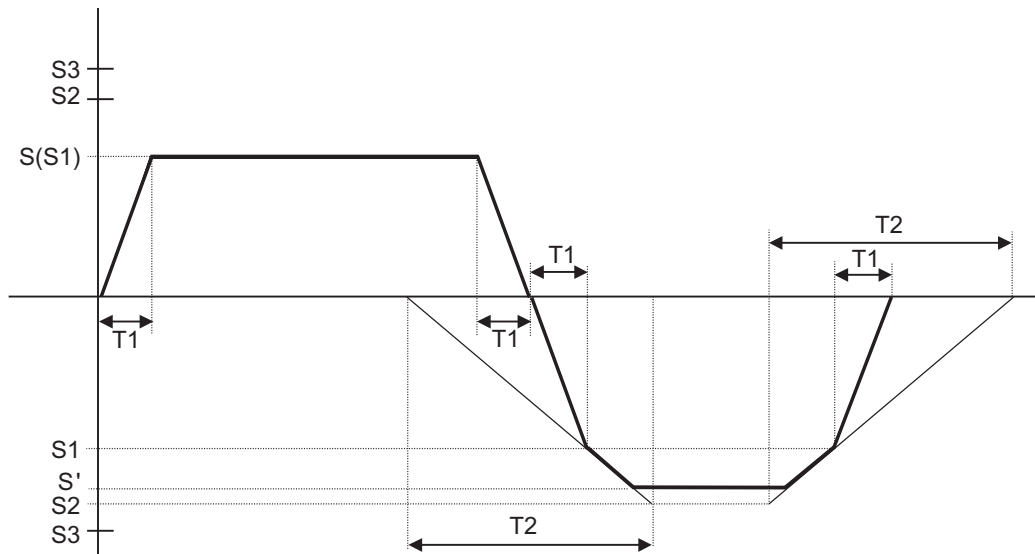
- (a) In the deep-hole tapping, the load applied to the tool can be reduced by designating the depth of cut per pass and cutting the workpiece to the hole bottom for a multiple number of passes.
Under the deep-hole tapping cycle, the tool is retracted to the R-point every time.
- (b) Whether the pecking tapping cycle or the deep-hole tapping cycle is valid depends on the MTB specifications (parameter "#1272 ext08/bit4").
When "depth of cut per pass Q" is designated in the block containing the G84 or G74 command in the state where the deep-hole tapping cycle is selected by parameter, the deep-hole tapping cycle is executed.
In the following cases, the normal tapping cycle will be carried out.
- When Q is not designated.
 - When the command value of Q is zero.
- (c) When G84 is executed, the override will be canceled and the override will automatically be set to 100% in the cutting operation. And the override set in the parameter "#1172 tapovr" will also be disabled.
(When "#1272 ext08/bit5" = 1, the setting of "#1172 tapovr" will be enabled only during a pulling operation)
- (d) Dry run is valid for a positioning command when the parameter "#1085 G00 DRY RUN" is "1" and is valid for the positioning command. If the feed hold button is pressed during G84 execution, the tool does not stop immediately during cutting or returning, and it stops after completing an R point return.
- (e) During single block operation, the tool does not stop during cutting or returning, but stops after completing an R point/initial point return.
- (f) During the G84 mode, the NC signal "Tapping" will be output.
- (g) During the G84 synchronous tapping modal, the M3, M4, M5 and S code will not be output.
- (h) If the command value of F becomes extremely small such as around "F < 0.01mm/rev" during synchronous tapping, the spindle does not rotate smoothly. So make sure to command a value larger than "0.01mm/rev". The unit of F can be selected between mm/rev and mm/min.
- (i) If the external deceleration signal is turned ON during synchronous or asynchronous tapping, the feed rate does not change even when deceleration conditions are satisfied.
- (j) If the operation is interrupted by a cause such as an emergency stop or reset during the deep-hole tapping cycle, a tap retract is executed when the tap retract signal is input.
- (k) When the reference position return signal is input during the deep-hole tapping cycle, a tap retract is carried out, and a reference position return will be executed from the end point of the tap retract.

Spindle acceleration/deceleration pattern during synchronous tapping

This function enables to make spindle acceleration/deceleration pattern closer to that of the speed loop by dividing the spindle and drilling axis acceleration/deceleration pattern into up to three stages during synchronous tapping. The acceleration/deceleration pattern can be set up to three stages for each gear. (This depends on the MTB specifications.)

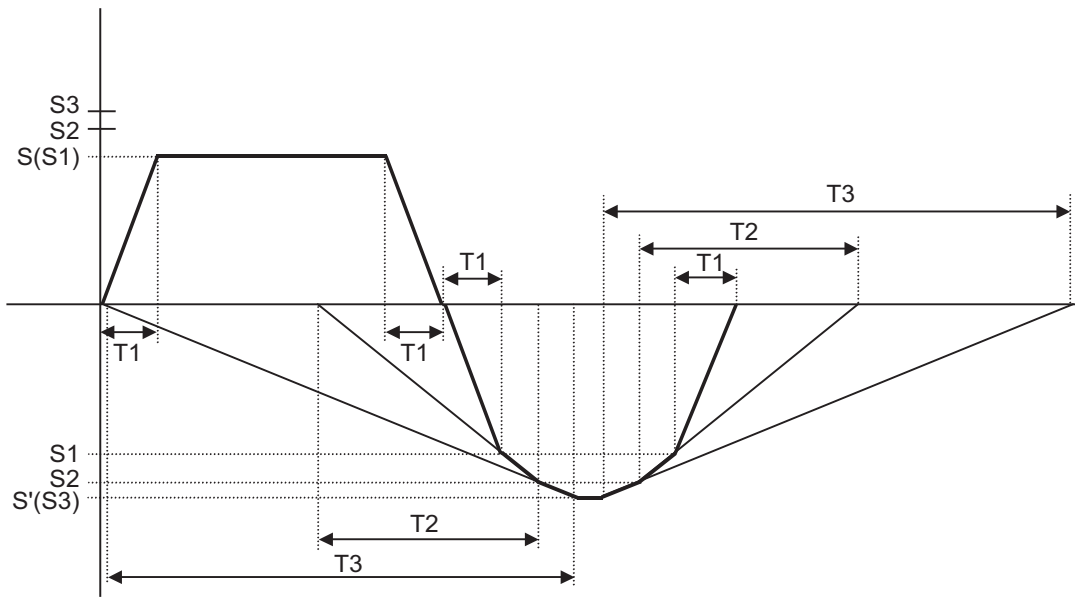
When returning from the hole bottom, rapid return is possible at the spindle rotation speed during return. The spindle rotation speed during return is held as modal information.

- (1) When tapping rotation speed < spindle rotation speed during return ≤ synchronous tapping changeover spindle rotation speed 2



- S Command spindle rotation speed
- S' Spindle rotation speed during return
- S1 Tapping rotation speed (spindle specification parameters #3013 to #3016)
- S2 Synchronous tapping changeover spindle rotation speed 2 (spindle specification parameters #3037 to #3040)
- S3 Maximum spindle rotation speed for synchronous tapping (spindle specification parameters #43046 to #43049)
However, when those parameters are set to "0", processing is performed based on "#3005" to "#3008". #3005 to #3008 can be designated using up to six digits (999999); however, they are limited to five digits (99999) for this function.
- T1 Tapping time constant (spindle specification parameters #3017 to #3020)
- T2 Synchronous tapping changeover time constant 2 (spindle specification parameters #3041 to #3044)

(2) When synchronous tapping changeover spindle rotation speed 2 < spindle rotation speed during return



- S Command spindle rotation speed
- S' Spindle rotation speed during return
- S1 Tapping rotation speed (spindle specification parameters #3013 to #3016)
- S2 Synchronous tapping changeover spindle rotation speed 2 (spindle specification parameters #3037 to #3040)
- S3 Maximum spindle rotation speed for synchronous tapping (spindle specification parameters #43046 to #43049)
However, when those parameters are set to "0", processing is performed based on "#3005" to "#3008". #3005 to #3008 can be designated using up to six digits (999999); however, they are limited to five digits (99999) for this function.
- T1 Tapping time constant (spindle specification parameters #3017 to #3020)
- T2 Synchronous tapping changeover time constant 2 (spindle specification parameters #3041 to #3044)
- T3 Synchronous tapping changeover time constant 3 (spindle specification parameters #3045 to #3048)

Feedrate for tapping cycle and tapping return

The feedrates for the tapping cycle and tapping return are as shown below.

(1) Selection of synchronous tapping cycle/asynchronous tapping cycle

Program G84..., Rxx	Control parameter Synchronous tapping	Synchronous/ asynchronous
,R00	-	Asynchronous
,Rxx	OFF	
No designation	ON	Synchronous
,R01	-	

- is irrelevant to the setting

(2) Selection of asynchronous tapping cycle feedrate

G94/G95	Control parameter F1-digit valid	F command value	Feed designation
G94	OFF	F designation	Feed per minute
	ON	Other than F0 to F8	F1-digit Feed
		F0 to F8 (no decimal point)	
G95	-	F designation	Feed per revolution

- is irrelevant to the setting

(3) Spindle rotation speed during return of synchronous tapping cycle

Address	Meaning of address	Command range (unit)	Remarks
,S	Spindle rotation speed during return	0 to 99999 (r/min)	The data is held as modal information. If the value is smaller than the spindle rotation speed, the spindle rotation speed value will be valid even during return. If the spindle rotation speed is not 0 during return, the tapping retract override value will be invalid.

M code for forward/reverse rotation command in asynchronous tapping cycle

The M code set with the parameter "#3028 sprcmm" is output as the M code for spindle forward/reverse rotation that is output at "hole bottom" or at "R point" during asynchronous tapping cycle.

Note that the M code for forward rotation is output as "M3" and that for reverse rotation is as "M4" if the parameter "#3028 sprcmm" is set to "0".

Feed per minute command of the synchronous tapping

Enable the feed per minute command of the synchronous tapping by the setting of parameter "#1268 ext04/bit2". When this parameter is valid, G94 and G95 modal will be applied.

	During G94 modal (feed per minute)	During G95 modal (feed per revolution)
#1268/bit2 = 1	Feed per minute (*1)	Feed per revolution (*2)
#1268/bit2 = 0	Feed per revolution (*2)	Feed per revolution (*2)

(*1) The F command is set to feed per minute (mm/min, inch/min).

Pitch = F command value / S command value

(*2) The F command is set to feed per revolution (mm/rev, inch/rev).

Note

- (1) The G94 command, which is a modal command, is valid until the G95 (feed per revolution) command is issued next.
- (2) If the E address (number of screw threads per inch) is issued while feed per minute is valid, the program error (P32) will occur.
- (3) The F address of the synchronous tapping command does not affect the F modal for cutting feed.

Range restriction of maximum cutting feedrate command for synchronous tapping

You can restrict the maximum value (minimum value of the E address for the number of screw threads) of the pitch F address for synchronous tapping (parameter "#19004 tap feedrate limit"). The program error (P184) will occur if the machining program is executed when the value of "F" address (pitch) exceeds the maximum value or when the value of "E" address (number of the screw threads per inch) is below the minimum value.

When the parameter "#19004" is set to "0", the pitch command by the F address is set as follows.

Command unit	Pitch F	E setting (number of screw threads) (*1)
B (0.001mm)	0.001 to 999.999 mm/rev	0.0255 to 999.99 screw threads/inch
C (0.0001mm)	0.0001 to 999.9999mm/rev	0.026 to 999.999 screw threads/inch
D (0.00001mm)	0.00001 to 999.99999 mm/rev	0.0255 to 999.9999 screw threads/inch
E (0.000001mm)	0.000001 to 999.999999 mm/rev	0.02541 to 999.99999 screw threads/inch
B (0.0001inch)	0.000001 to 39.370078inch/rev	0.03 to 9999.9999 screw threads/inch
C (0.00001inch)	0.0000001 to 39.3700787inch/rev	0.026 to 9999.99999 screw threads/inch
D (0.000001inch)	0.00000001 to 39.37007874inch/rev	0.0255 to 9999.999999 screw threads/inch
E (0.0000001inch)	0.000000001 to 39.370078740inch/rev	0.02541 to 9999.9999999 screw threads/inch

(*1) When feed per minute is commanded, the pitch calculation result for the spindle rotation speed is range-restricted in this parameter setting.

Synchronous tapping in-position check (Parameter setting values and tapping axis movement)

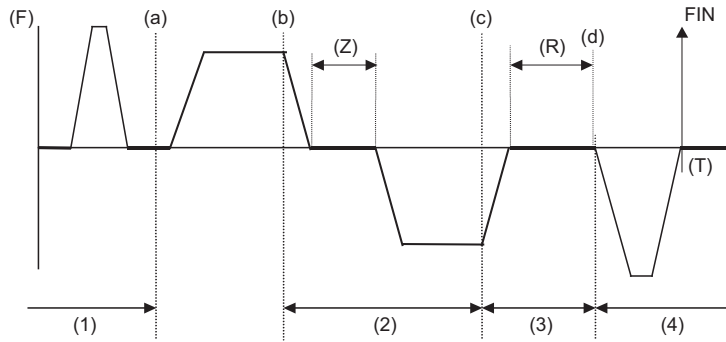
#1223 aux07				"P" designation of G84/ G74 command	In-position check during synchronous tapping		
bit3	bit4	bit5	bit2		Hole bottom	R point	I point -> R point
In-position check during synchronous tapping	Hole bottom	R point	I point-> R point				
0	-	-	-	-	yes	yes	yes
1	-	-	-	No "P" designation Example: G84 F1. Z-5. S1000 R-5.	no	no	no
1	1	1	1	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	(*1)	yes	yes
1	1	0	1	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	(*1)	no	yes
1	0	1	1	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	yes	yes	yes
1	0	0	1	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	no	no	yes
1	1	1	0	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	(*1)	yes	no
1	1	0	0	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	(*1)	no	no
1	0	1	0	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	no	yes	no
1	0	0	0	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	no	no	no

(*1) Carry out in-position check by tapping in-position width.

Note

(1) The I point refers to the initial point.

In-position width and tapping axis movement for a synchronous tapping in-position check



(Z) Hole bottom

(R) R point

(a) In-position completion of the G00 feed from the R point

(b) G01 deceleration start at tapping cut-in

(c) G01 deceleration start at tapping return

(d) Start of G00 feed to the R point

(1) Section in which the in-position check is carried out by G0inps.

(2) Section in which the in-position check is carried out by TapInp.

(3) Section in which the in-position check is carried out by G1inps.

(4) Section in which the in-position check is carried out by sv024.

R point: In-position check by the G1inps

I point: In-position check by the G0inps

Hole bottom: In-position check by the TapInp

Relation between the parameter setting values and tapping axis movement for a synchronous tapping in-position check

#1223 aux07				Hole bottom wait time	Operation at hole bottom	Operation at R point	Operation at I point -> R point
bit3	bit4	bit5	bit2				
In-position check during synchronous tapping	Hole bottom	R point	I point -> R point				
0	-	-	-	Time designated by "P" Several 10 ms as processing time when no "P".	Operation determined by setting of inpos (#1193) and aux07 (#1223/bit1) parameters.	Operation determined by setting of inpos (#1193) and aux07 (#1223/bit1) parameters.	Operation determined by setting of inpos (#1193) and aux07 (#1223/bit1) parameters.
1	0	0	1	The larger value of "P" and TapDwl (#1313) is valid. No dwell is executed if both values are "0".	Wait until time in the left column elapses.		Wait until completion of in-position check by G0inps.
1	0	1	1	The larger value of "P" and TapDwl (#1313) is valid. No dwell is executed if both values are "0".	Wait until time in the left column elapses.	Wait until completion of in-position check by G1inps.	Wait until completion of in-position check by G0inps.
1	1	0	1	The larger value of "P" and TapDwl (#1313) is valid. No dwell is executed if both values are "0".	Wait until dwell time in the left column elapses after completion of in-position check.		Wait until completion of in-position check by G0inps.
1	1	1	1	The larger value of "P" and TapDwl (#1313) is valid. Several 10 ms as processing time when both of them are "0".	Wait until dwell time in the left column elapses after completion of in-position check.	Wait until completion of in-position check by G1inps.	Wait until completion of in-position check by G0inps.
1	0	0	0	The larger value of "P" and TapDwl (#1313) is valid. No dwell is executed if both values are "0".	Wait until time in the left column elapses.		
1	0	1	0	The larger value of "P" and TapDwl (#1313) is valid. No dwell is executed if both values are "0".	Wait until time in the left column elapses.	Wait until completion of in-position check by G1inps.	
1	1	0	0	The larger value of "P" and TapDwl (#1313) is valid. No dwell is executed if both values are "0".	Wait until dwell time in the left column elapses after completion of in-position check.		
1	1	1	0	The larger value of "P" and TapDwl (#1313) is valid.	Wait until dwell time in the left column elapses after completion of in-position check.	Wait until completion of in-position check by G1inps.	

Note

- (1) The I point refers to the initial point.
- (2) Note that vibration or deterioration in accuracy may occur when invalidating the in-position check at R point. Confirm the accuracy when invalidating it.
Processing may take some time if no in-position check is performed at each point.

Manual synchronous tapping

This function allows you to carry out tapping by selecting and moving the drilling axis with manual handle operation after performing block stop or feed hold stop processing during synchronous tapping cycle.

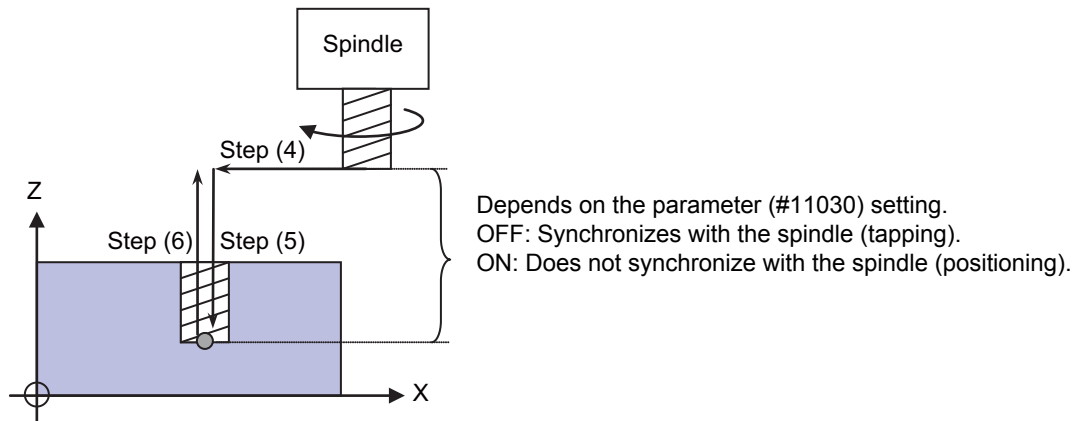
Whether to synchronize the drilling axis for manual synchronous tapping with the spindle when moving it depends on the MTB specifications. (Parameter "#11030 Man tap sync cancl")

Designate this command in the same format as for synchronous tapping.

[Operation procedure]

The following example shows the procedure when the parameter "#11030 Man tap sync cancl" is set to "0".

- (1) Execute the synchronous tapping cycle program in the MDI mode.
G91 G84 X0 Y0 Z0 R0 F2. S1000;
- (2) Set to the handle mode.
- (3) Determine the drilling position using the X/Y axis handle.
- (4) Perform drilling using the drilling axis handle.
- (5) Perform pulling out from the home bottom using the drilling axis handle.
- (6) When continuing machining, return to step (4).
- (7) Reset the G84 modal.



[Precautions and restrictions]

- (1) The manual synchronous tapping is only required in the handle mode.
- (2) If necessary, you can perform the manual synchronous tapping using the handle after switching to another operation mode until it is reset or canceled with the G80 command.
- (3) The spindle is synchronized in the pitch commanded with "F" of machining program G84 (G74).
(Example)
N1 G28 X0 Y0 Z0 ;
N2 G91 G01 F1000 ;
N3 G84 X-50. Y-50. Z-100. R-50. F2. S1000. ,R1 ;
N4 G80 M02 ;
When the handle is turned while the handle magnification is set to "100", the Z axis moves synchronously with the spindle. "2mm/rev" is set to the F2 command, so the spindle rotates by one revolution when the handle is turned by 20 pulses.
- (4) For manual synchronous tapping, the acceleration/deceleration time constant is the same as for the normal handle operation. Also, the tap return override is invalid.
- (5) The spindle rotation speed for manual synchronous tapping is not clamped with parameters "#3013 stap1" to "#3016 stap4" (maximum tapping rotation speed) in the S command of the program.
- (6) Manual synchronous tapping becomes valid from the timing when block stop or feed hold stop mode is set during automatic operation. However, the feed hold stop mode during cutting is changed to the block stop mode after "cutting -> movement to R point".
- (7) The dwell time (P command) at the hole bottom is invalid during manual synchronous tapping.
- (8) When the automatic operation is started in the miscellaneous function lock ("MST lock" ON), the spindle is not synchronized even after block stop or feed hold stop processing has been ended.
- (9) The manual synchronous tapping operation in 3-dimensional coordinate conversion is different from the above. (Not supported.)

Analog spindle synchronous tapping

This function performs the tapping using the analog-connected spindle. By using this function, each tapping cycle function can be used using the analog-connected spindle by inverter, etc.

When this function is used, the analog spindle with the function to control the spindle position needs to be connected. Also, the parameter "#1295 ext31/bit6" (Analog spindle synchronous tapping ON) needs to be set to "1".

The voltage value to be output to the analog spindle is between -10V and 10V and it is determined by the ratio of the setting value of the parameters "#3001 slimit1" to "#3004 slimit4" which correspond to the command to the analog spindle.

[Relationship with other functions]

- (1) The following function cannot be used with the analog spindle synchronous tapping. If the analog spindle synchronous tapping is commanded while the following function is used, the program error (P182) occurs.
 - Coordinate rotation by program
 - 3-dimensional coordinate conversion
 - Parameter coordinate conversion
 - Inclined surface machining
 - Workpiece installation error compensation
 - R-Navi
 - Mixed control
 - Arbitrary axis exchange control
- (2) When the reset or emergency stop is performed during analog spindle synchronous tapping, the tap retract function cannot be used.

[Precautions]

- (1) The pecking tapping cycle or deep-hole tapping cycle cannot be commanded while the analog spindle synchronous tapping is used. If commanded, a program error (P182) occurs.
- (2) The synchronous tap with multi-step acceleration deceleration cannot be used while the analog spindle synchronous tapping is used. Command by setting the parameter "#1223 aux07/bit7" (Synchronous tap method) to "1".
- (3) The high-speed synchronous tapping cannot be used while the analog spindle synchronous tapping is used. Regardless of the value of the parameter "#1281 ext17/bit5" (High-speed synchronous tapping valid), the normal synchronous tapping operation is performed.
- (4) If the synchronous tapping is commanded to the analog spindle in multiple spindle control mode when the parameter "#1295 ext31/bit" (Analog spindle synchronous tapping ON) is turned OFF, the following error occurs.
In multiple spindle control I mode: Program error (P182)
In multiple spindle control II mode: Operation error (M01 0054)
- (5) The analog spindle synchronous tapping can be used only in one part system. In multiple spindle control mode, if the analog spindle synchronous tapping is performed in the part system other than the one where the parameter "#11717 astap_sysno" (Analog spindle synch tap: Part system selection) is set, the following errors occur.
In multiple spindle control I mode: Program error (P182)
In multiple spindle control II mode: Operation error (M01 0054)
- (6) When multiple spindles are selected using the multiple spindle control, do not command the synchronous tapping with the analog-connected spindle and the serial-connected spindle being mixed.
- (7) With the analog spindle synchronous tapping, the synchronous tapping error display function is disabled and the synchronous tapping error display always shows "0".
- (8) When the reset or emergency stop is performed during analog spindle synchronous tapping, the voltage output value to the spindle becomes "0" and the spindle operation stops.

13.1.5 Boring; G85



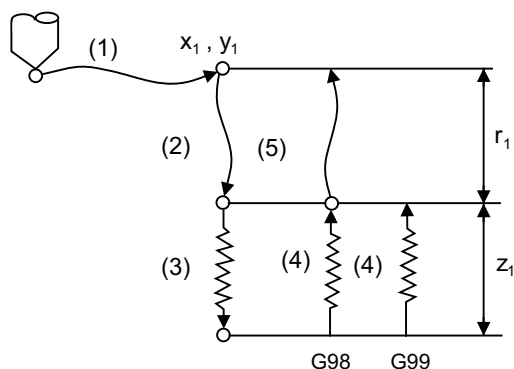
Command format

G85 Xx1 Yy1 Zz1 Rr1 Ff1 LI1 ,li1 ,Jj1;

Xx1	Designation of hole drilling position (absolute value or incremental value)
Yy1	Designation of hole drilling position (absolute value or incremental value)
Zz1	Designation of hole bottom position (absolute value or incremental value) (modal)
Rr1	Designation of R point position (absolute value or incremental value) (modal)
Ff1	Designation of feedrate for cutting feed (modal)
LI1	Designation of number of repetitions (0 to 9999) When "0" is set, processing is not executed.
,li1	Positioning axis in-position width
,Jj1	Drilling axis in-position width



Detailed description



Operation pattern	i1	j1	Program	
(1)	Valid	-	G00 Xx1 Yy1;	
(2)	-	Invalid	G00 Zr1;	
(3)	-	Invalid	G01 Zz1 Ff1;	
(4)	-	Invalid	G01 Z-z1 Ff1;	
(5)	-	Invalid	G00 Z-r1;	G98 mode
			No movement	G99 mode

The operation stops at after the (1), (2), (4) or (5) commands during single block operation.

13.1.6 Boring; G86



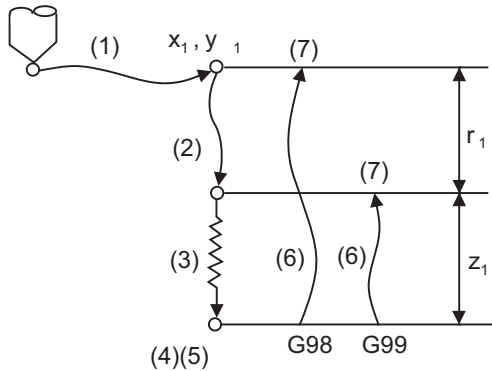
Command format

G86 Xx1 Yy1 Zz1 Rr1 Ff1 Pp1 Ll1 ;

Xx1	Designation of hole drilling position (absolute value or incremental value)
Yy1	Designation of hole drilling position (absolute value or incremental value)
Zz1	Designation of hole bottom position (absolute value or incremental value) (modal)
Rr1	Designation of R point position (absolute value or incremental value) (modal)
Ff1	Designation of feedrate for cutting feed (modal)
Pp1	Designation of dwell time at hole bottom position (the values after the decimal points will be ignored) (modal)
Ll1	Designation of number of repetitions (0 to 9999) When "0" is set, processing is not executed.



Detailed description



Operation pattern	Program	
(1)	G00 Xx1 Yy1;	
(2)	G00 Zr1;	
(3)	G01 Zz1 Ff1;	
(4)	G04 Pp1;	
(5)	M5;	Spindle stop
(6)	G00 Z-(z1+r1);	G98 mode
	G00 Z-z1;	G99 mode
(7)	M3;	Spindle forward rotation

The operation stops at after the (1), (2) and (7) commands during single block operation.

13.1.7 Back Boring; G87



Command format

G87 Xx1 Yy1 Zz1 Rr1 Iq1 Jq2 Kq3 Ff1 LI1;
--

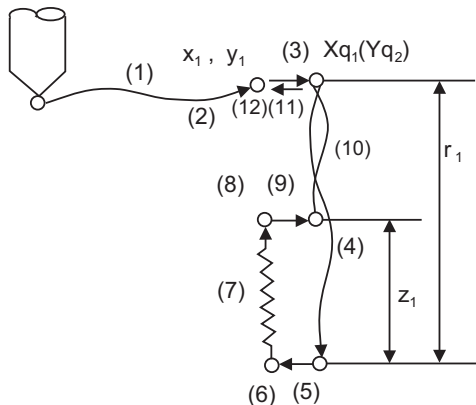
Xx1	Designation of hole drilling position (absolute value or incremental value)
Yy1	Designation of hole drilling position (absolute value or incremental value)
Zz1	Designation of hole bottom position (absolute value or incremental value) (modal)
Rr1	Designation of R point position (absolute value or incremental value) (modal)
Iq1	Designation of shift amount (incremental value) (modal)
Jq2	The command address for each plane selection is as follows. When G17 plane is selected: IJ When G18 plane is selected: KI When G19 plane is selected: JK Depending on the parameter setting, the shift amount can be designated by Q address. Refer to "Designation of shift amount (I,J,K)".
Kq3	
Ff1	
LI1	Designation of number of repetitions. (0 to 9999) When "0" is set, no execution

Note

- (1) Be careful of the designation of "z1" and "r1" (the sign of "z1" is opposite to that of "r1"). Also, the R point return is not performed.



Detailed description



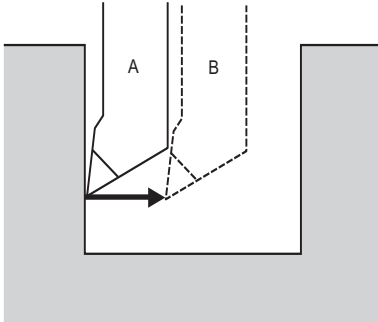
Operation pattern	Program	
(1)	G00 Xx1 Yy1;	
(2)	M19 ;	Spindle orientation
(3)	G00 Xq1 (Yq2) ;	Shift
(4)	G00 Zr1;	
(5)	G00 X-q1(Y-q2)/G01 X-q1(Y-q2)Ff1 ;	Shift
(6)	M3;	Spindle forward rotation
(7)	G01 Zz1 Ff1;	
(8)	M19 ;	Spindle orientation
(9)	G00 Xq1 (Yq2);	Shift
(10)	G00 Z-(r1+z1);	G98 mode
	G00 Z-(r1+z1);	G99 mode
(11)	G00 X-q1(Y-q2);	Shift
(12)	M3;	Spindle forward rotation

The operation stops at after the (1), (4), (6) and (11) commands during single block operation.

Designation of shift amount (I,J,K)

When this command is issued, high precision drilling machining that does not scratch the machining surface can be done. Positioning to the hole bottom and escaping (return) after cutting are carried out in the state shifted to the direction opposite of the cutter.

(The bold arrow shown in the figure indicates the tool shift amount.)



A: Tool position during cutting

B: Tool position when positioning to the hole bottom and, also, when escaping after cutting

Whether the tool moves (*1) by the designated shift amount with the rapid traverse (G00) or linear interpolation (G01) depends on the MTB specifications (parameter "#1255 set27/bit4").

(*1) Refers to the movement using operation pattern (5) shown in the figure in "Detailed description".

When the rapid traverse is selected, the route at positioning is set to the interpolation type regardless of the setting of the parameter "#1086 G0Intp".

When the linear interpolation is selected, the feedrate follows the F command.

Note

- (1) To move the tool by the designated shift amount with the rapid traverse, it is necessary to upgrade the M800/M80 series software to version C4 or later and replace the fixed cycle program. For details, consult the MTB.
- (2) Command I, J, and K with incremental values in the same block as the hole position data. I, J and K will be handled as modal during the fixed cycle.
- (3) If the parameter "#1080 Dril_Z" which fixes the hole drilling axis to the Z axis is set, the shift amount can be designated with address Q instead of I, J. In this case, whether to shift or not and the shift direction are set with parameter "#8207 G76/87 No shift" and "#8208 G76/87 Shift (-)". The sign for the Q value is ignored and the value is handled as a positive value. The Q value is a modal during the fixed cycle. Then, be sure to note that the Q value is commonly used for the cutting amount with G73/G83 or the shift amount with G83/G87.

13.1.8 Boring; G88



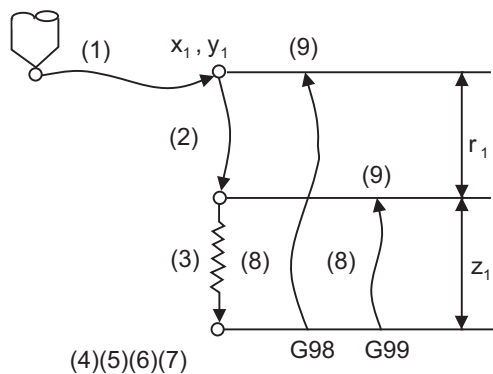
Command format

G88 Xx1 Yy1 Zz1 Rr1 Ff1 Pp1 LI1;

Xx1	Designation of hole drilling position (absolute value or incremental value)
Yy1	Designation of hole drilling position (absolute value or incremental value)
Zz1	Designation of hole bottom position (absolute value or incremental value) (modal)
Rr1	Designation of R point position (absolute value or incremental value) (modal)
Ff1	Designation of feedrate for cutting feed (modal)
Pp1	Designation of dwell time at hole bottom position (the values after the decimal points will be ignored) (modal)
LI1	Designation of number of repetitions (0 to 9999) When "0" is set, processing is not executed.



Detailed description



Operation pattern	Program	
(1)	G00 Xx1 Yy1;	
(2)	G00 Zr1;	
(3)	G01 Zz1 Ff1;	
(4)	G04 Pp1;	
(5)	M5;	Spindle stop
(6)	Stop when single block stop switch is ON	
(7)	Automatic start switch ON	
(8)	G00 Z-(z1+r1);	G98 mode
	G00 Z-z1;	G99 mode
(9)	M3 ;	Spindle forward rotation

The operation stops at after the (1), (2), (6) and (9) commands during single block operation.

13.1.9 Boring; G89



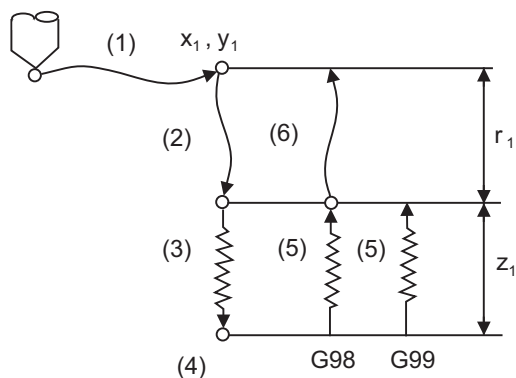
Command format

G89 Xx1 Yy1 Zz1 Rr1 Ff1 Pp1 Ll1 ,li1,Jj1;

Xx1	Designation of hole drilling position (absolute value or incremental value)
Yy1	Designation of hole drilling position (absolute value or incremental value)
Zz1	Designation of hole bottom position (absolute value or incremental value) (modal)
Rr1	Designation of R point position (absolute value or incremental value) (modal)
Ff1	Designation of feedrate for cutting feed (modal)
Pp1	Designation of dwell time at hole bottom position (the values after the decimal points will be ignored) (modal)
Ll1	Designation of number of repetitions (0 to 9999) When "0" is set, processing is not executed.
,li1	Positioning axis in-position width
,Jj1	Drilling axis in-position width



Detailed description



Operation pattern	i1	j1	Program	
(1)	Valid	-	G00 Xx1 Yy1;	
(2)	-	Invalid	G00 Zr1;	
(3)	-	Invalid	G01 Zz1 Ff1;	
(4)	-	-	G04 Pp1;	
(5)	-	Invalid	G01 Z-z1 Ff1;	
(6)	-	Valid	G00 Z-r1;	G98 mode
			No movement	G99 mode

The operation stops at after the (1), (2), (5) or (6) commands during single block operation.

13.1.10 Stepping Cycle; G73



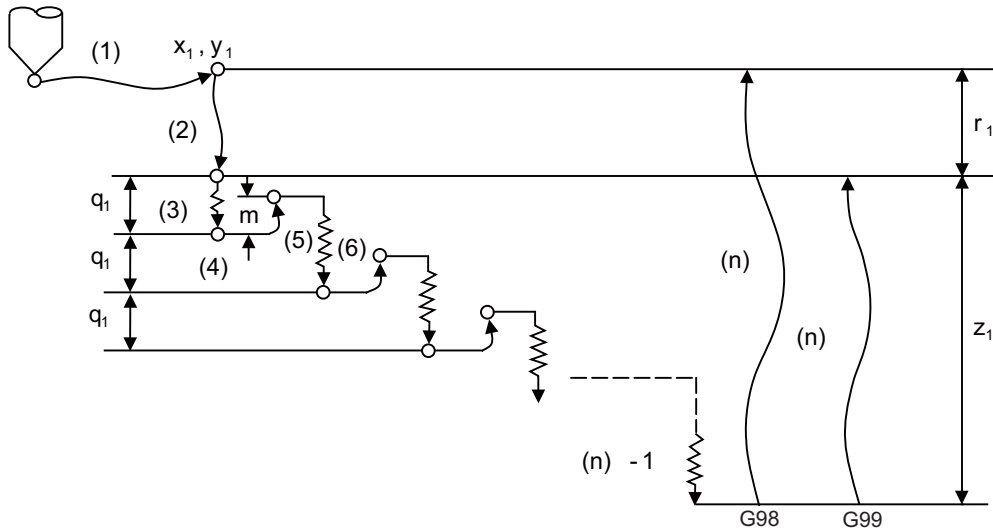
Command format

G73 Xx1 Yy1 Zz1 Qq1 Rr1 Ff1 Pp1 Ll1 ,li1 ,Jj1;
--

Xx1	Designation of hole drilling position (absolute value or incremental value)
Yy1	Designation of hole drilling position (absolute value or incremental value)
Zz1	Designation of hole bottom position (absolute value or incremental value) (modal)
Qq1	Cut amount for each cutting pass (incremental value) (modal)
Rr1	Designation of R point position (absolute value or incremental value) (modal)
Ff1	Designation of feedrate for cutting feed (modal)
Pp1	Designation of dwell time at hole bottom position (the values after the decimal points will be ignored) (modal)
Ll1	Designation of number of repetitions (0 to 9999) When "0" is set, processing is not executed.
,li1	Positioning axis in-position width
,Jj1	Drilling axis in-position width



Detailed description



Operation pattern	i1	j1	Program	
(1)	Valid	-	G00 Xx1 Yy1;	
(2)	-	Invalid	G00 Zr1;	
(3)	-	Invalid	G01 Zq1 Ff1;	
(4)	-	-	G04 Pp1;	
(5)	-	Invalid	G00 Z-m;	
(6)	-	Invalid	G01 Z(q1+m) Ff1;	
:				
(n)-1	-	Invalid		
(n)	-	Valid	G00 Z-(z1+r1);	G98 mode
			G00 Z-z1;	G99 mode

When executing a second and following cutting in the G73 as shown above, the movement will return several "m" mm with rapid traverse and then will change to cutting feed. The return amount "m" will differ according to the parameter "#8012 G73 return".

The operation stops at after the (1), (2) and (n) commands during single block operation.

13.1.11 Reverse Tapping Cycle; G74



Command format

G74 Xx1 Yy1 Zz1 Rr1 Ff1 Pp1 ,Rr2 Ss1 ,Ss2 LI1 ,li1,Jj1;

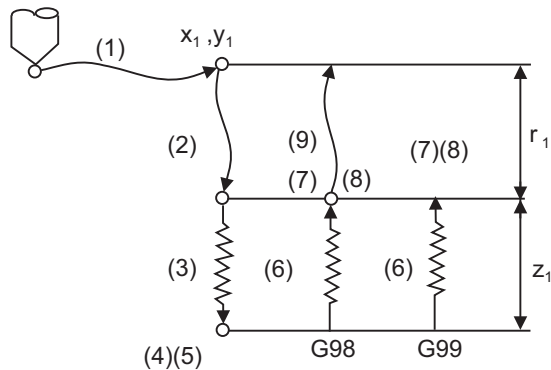
Xx1	Designation of hole drilling position (absolute value or incremental value)
Yy1	Designation of hole drilling position (absolute value or incremental value)
Zz1	Designation of hole bottom position (absolute value or incremental value) (modal)
Rr1	Designation of R point position (absolute value or incremental value) (modal)
Ff1	Z-axis feed amount (tapping pitch) per spindle rotation (modal)
Pp1	Designation of dwell time at hole bottom position (the values after the decimal points will be ignored) (modal)
,Rr2	Synchronization method selection (r2=1 synchronous, r2=0 asynchronous) (modal) (When omitted, the mode will follow the setting of parameter "#8159 Synchronous tap")
Ss1	Spindle rotation speed command <Note> ♦At a synchronous tapping mode, "Sn = *****" type S command will be ignored. (n:spindle number, *****: rotation speed) ♦If an S command is issued during synchronous tapping modal, a program error (P186) will occur.
,Ss2	Spindle rotation speed during return
LI1	Designation of number of repetitions (0 to 9999) When "0" is set, processing is not executed.
,li1	Positioning axis in-position width
,Jj1	Drilling axis in-position width

Note _____

(1) When asynchronous tapping mode is applied, F address becomes the cutting feed speed.



Detailed description



Operation pattern	i1	j1	Program	
(1)	Valid	-	G00 Xx1 Yy1;	
(2)	-	Invalid	G00 Zr1;	
(3)	-	Invalid	G01 Zz1 Ff1;	
(4)	-	-	G04 Pp1;	
(5)	-	-	M3;	Spindle forward rotation
(6)	-	Invalid	G01 Z-z1 Ff1;	
(7)	-	-	G04 Pp1;	
(8)	-	-	M4;	Spindle reverse rotation
(9)	-	Valid	G00 Z-r1;	G98 mode
			No movement	G99 mode

When $r2 = 1$, the synchronous tapping mode will be applied, and when $r2 = 0$, the asynchronous tapping mode will be applied. If there is no $r2$ command, mode will follow the parameter setting.

When G74 is executed, the override will be canceled and the override will automatically be set to 100%.

Dry run is valid for the positioning command when the parameter "#1085 G00 Drn" is set to "1".

If the feed hold button is pressed during G74 execution, and the sequence is at (3) to (6), the movement will not stop immediately, and instead will stop after (6). During the rapid traverse in sequence (1), (2) and (9), the movement will stop immediately.

The operation stops at after the (1), (2) and (9) commands during single block operation.

During the G74 and G84 modal, the "Tapping" NC output signal will be output.

During the G74 synchronous tapping modal, the M3, M4, M5 and S code will not be output.

Spindle acceleration/deceleration pattern during synchronous tapping

Refer to "13.1.4 Tapping Cycle; G84".

Feedrate for tapping cycle and tapping return

Refer to "13.1.4 Tapping Cycle; G84".

M code for forward/reverse rotation command in asynchronous tapping cycle

Refer to "13.1.4 Tapping Cycle; G84".

Parameter setting values and tapping axis

Refer to "13.1.4 Tapping Cycle; G84".

13.1.12 Circular Cutting; G75

**Function and purpose**

Circular cutting starts with the X and Y axes positioned at the center of the circle, and the Z axis cuts into the commanded position. Then, the tool cuts the inner circumference of the circle drawing a true circle and returns to the center position.

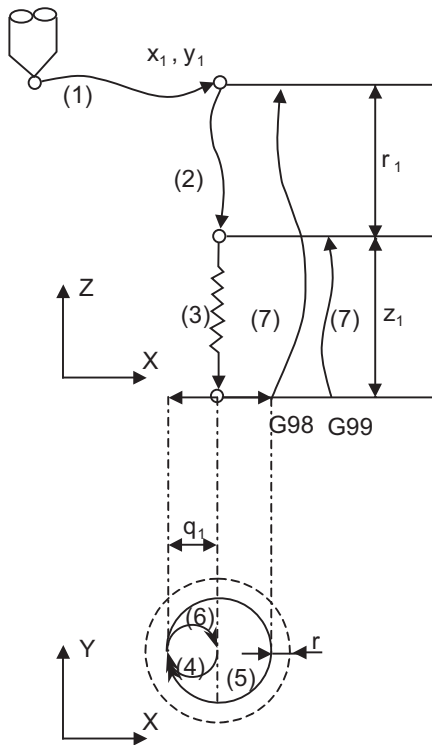
**Command format**

G75 Xx1 Yy1 Zz1 Rr1 Qq1 Pp1 Ff1 Ll1 ;
--

Xx1	Designation of hole drilling position (absolute value or incremental value)
Yy1	Designation of hole drilling position (absolute value or incremental value)
Zz1	Designation of hole bottom position (absolute value or incremental value) (modal)
Rr1	Designation of R point position (absolute value or incremental value) (modal)
Qq1	Radius of outer circumference (modal)
Pp1	Tool radius compensation No. (modal)
Ff1	Designation of feedrate for cutting feed (modal)
Ll1	Designation of number of repetitions (0 to 9999) When "0" is set, processing is not executed.



Detailed description



Operation pattern	Program		n: $q1 \geq 0 \rightarrow G02$ $q1 < 0 \rightarrow G03$ r: Tool radius compensation amount of the No. commanded with p1.
(1)	G00 Xx1 Yy1;		
(2)	G00 Zr1;		
(3)	G01 Zz1 Ff1;		
(4)	Gn X-(q1-r) I-(q1/2);	Inner circumference half circle	
(5)	Iq1;	Outer circumference	
(6)	X(q1-r) I(q1/2);	Inner circumference half circle	
(7)	G00 Z-(z1+r1); G00 Z-z1;	G98 mode G99 mode	

The operation stops at after the (1), (2) and (6) commands during single block operation.

13.1.13 Fine Boring; G76

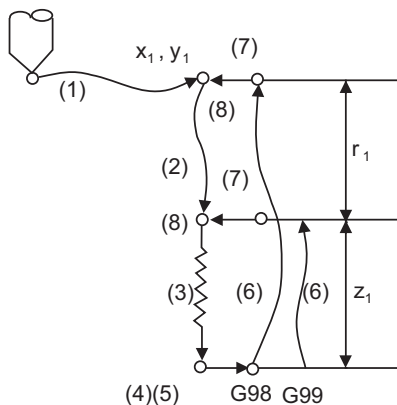


Command format

G76 Xx1 Yy1 Zz1 Rr1 Iq1 Jq2 Kq3 Ff1 LI1;	
Xx1	Designation of hole drilling position (absolute value or incremental value)
Yy1	Designation of hole drilling position (absolute value or incremental value)
Zz1	Designation of hole bottom position (absolute value or incremental value) (modal)
Rr1	Designation of R point position (absolute value or incremental value) (modal)
Iq1 Jq2 Kq3	Designation of shift amount (incremental value) (modal) The command address for each plane selection is as follows: G17 plane: IJ G18 plane: KI G19 plane: JK Depending on the parameter setting, the shift amount can be designated by Q address. Refer to "Designation of shift amount (I,J,K)".
Ff1	Designation of feedrate for cutting feed (modal)
LI1	Designation of number of repetitions (0 to 9999) When "0" is set, processing is not executed.



Detailed description



Operation pattern	Program	
(1)	G00 Xx1 Yy1;	
(2)	G00 Zr1;	
(3)	G01 Zz1 Ff1;	
(4)	M19;	Spindle orientation
(5)	G00 Xq1(Yq2)/G01 Xq1(Yq2)Ff1 ;	shift
(6)	G00 Z-(z1+r1);	G98 mode
	G00 Z-z1;	G99 mode
(7)	G00 X-q1 (Y-q2);	shift
(8)	M3;	Spindle forward rotation

The operation stops at after the (1), (2) and (7) commands during single block operation.

Designation of shift amount (I,J,K)

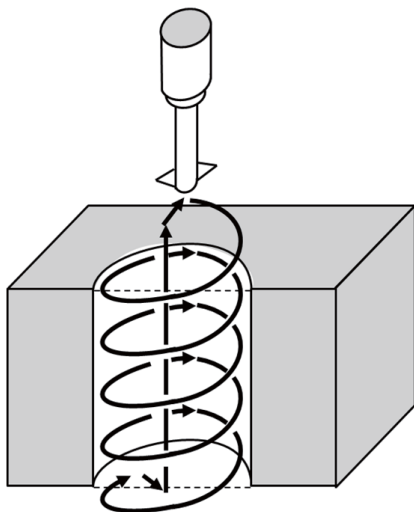
Refer to "Designation of shift amount (I,J,K) in "13.1.7 Back Boring; G87".

13.1.14 Thread Milling Cycle; G187

**Function and purpose**

This function is a fixed cycle that performs thread machining by helicoidally operating the tool referred to as a thread milling tool. This function allows a thick female thread cutting or female thread cutting in any pitch that is impossible using a tapping tool.

This function is a fixed cycle using circular interpolation (helical interpolation); therefore, plane selection is required in advance.

**Command format**

```
G187 Z_ I/J_ P_ F_ D_ Q_ ;
```

Z	Designation of pole bottom position
I/J	Arc radius, approach direction
P	Pitch
F	Feedrate
D	Rotation direction
Q	Dwell time



Detailed description

Detailed address setting

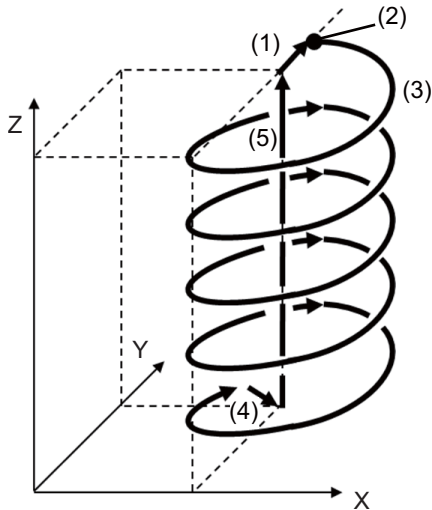
Address	Command range (unit)	Remarks
Z	-99999.999 to 99999.999 (mm)	Designate the hole bottom position. <ul style="list-style-type: none"> ♦If an axis other than the drilling axis is commanded or the address is omitted, a program error (P33) will occur.
I/J	-99999.999 to 99999.999 (mm)	Designate the arc radius and approach direction. The approach direction can be designated with the address I/J and the sign of the commanded value. (Example) If "J-5" is designated, the radius and direction are obtained as shown below. Arc radius: 5 (mm) Approach direction: -Y direction <ul style="list-style-type: none"> ♦ Designate the radius for helical operation, not the hole radius. ♦For the arc radius, designate a radius value. A program error (P33) will occur in the following cases. <ul style="list-style-type: none"> ♦The drilling axis direction (K) has been designated. ♦The command has been omitted, or two axes have been commanded simultaneously. ♦The command value is set to "0".
P	0.001 to 99999.999 (mm)	Designate the pitch (drilling axis feed amount per revolution). <ul style="list-style-type: none"> ♦For the pitch, designate a radius value. ♦If the command is omitted, a program error (P33) occurs. ♦If the command value is set to "0", a program error (P35) occurs.
F	0.001 to 10000000 (mm/min)	Designate the feedrate. Designate the helical operation speed, not the cutting speed. <ul style="list-style-type: none"> ♦The normal F modal value will not change. ♦If the command is omitted or the command value is set to "0", a program error (P62) occurs. ♦In the same way as for the normal helical interpolation, the speed designation is selected by the parameter "#1235 set07/bit0". (Based on the MTB specifications.)
D	0, 1	Designate the rotation direction. 0: CW 1: CCW <ul style="list-style-type: none"> ♦If the command is omitted, the rotation direction is set to CW.
Q	0 to 99999.999(s)	Designate the dwell period from the time when the axis moves from the hole center to the radius direction to the time when the helical interpolation starts. <ul style="list-style-type: none"> ♦When the command is omitted, dwelling is not performed. ♦The relationship between the time and designated value is the same as for the values designated in "G04P".



Operation example

The thread milling cycle runs as shown below.

- (1) The axis moves (approaches) with G01 from the center of the hole to the radius direction.
- (2) If the dwell time is designated, dwelling is performed.
- (3) Helical interpolation is performed in the commanded pitches.
- (4) After the hole bottom position has been reached, the axis moves to the center of the hole bottom with G01.
- (5) Pull out the tool vertically from the center of the hole bottom with G00.



Differences from another fixed cycle for drilling

- (1) Modal/Unmodal
The thread milling cycle (G187) is unmodal, which must be commanded for each cycle.
All the data commanded with the address is also unmodal.
- (2) Hole drilling axis
The drilling axis is determined by plane selection (G17, G18, or G19).
For details, refer to "Relationship between plane selection and drilling axis".
- (3) Command format
The hole position and the number of repetitions cannot be designated.
- (4) Operation
Positioning to the initial point or R point is not performed.
Drilling is started using the position, at which the thread milling cycle (G187) is commanded, as the center.

Relationship between plane selection and drilling axis

The drilling axis is determined by plane selection (G17, G18, or G19).
 The axis (X, Y, Z, or its parallel axis) vertical to the plane designated in G17, G18, or G19 is used as the drilling axis.
 The setting of the parameter "#1080 Dril_Z" is invalid for the thread milling cycle.

Plane selection	Hole drilling axis
G17 (X-Y)	Zp
G18 (Z-X)	Yp
G19 (Y-Z)	Xp

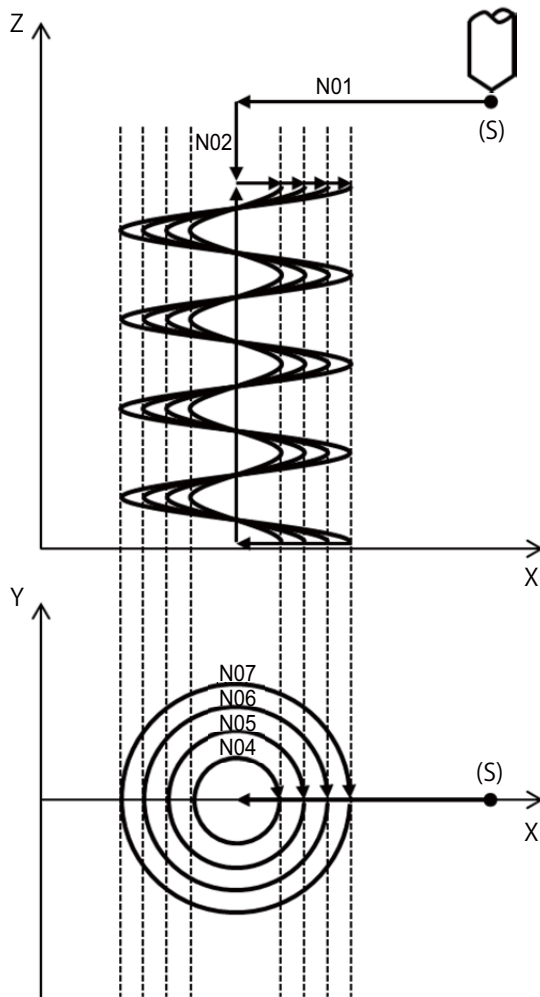
Xp, Yp and Zp indicate the base axes X, Y and Z or an axis parallel to the base axis.

The command format, etc. of this function are explained, assuming that G17 (drilling axis = Z axis) is designated for plane selection.

In the following usage example, finishing is carried out up to the thread top by repeating the command that increases the arc radius by degrees. Always designate the same value for the pitch (P).

```

:
N01 G90 G00 X30.;
N02 Z45.;
N03 G17;
N04 G187 Z25. I2. P5. F100 D0;
N05 G187 Z25. I3. P5. F100 D0;
N06 G187 Z25. I4. P5. F100 D0;
N07 G187 Z25. I5. P5. F100 D0;
:
M30;
    
```



(S) Start point



Precautions

Refer to "13.1.15 Precautions for Using a Fixed Cycle" in addition to the following description.

- (1) When a manual interruption is performed in automatic operation "pause", the end point of the interrupted block and the end point of the block in thread milling cycle move parallel by the manual movement amount.
 (The operation with the manual absolute (ABS) signal set OFF is performed during the thread milling cycle.)

13.1.15 Precautions for Using a Fixed Cycle



Precautions

- (1) Before the fixed cycle is commanded, the spindle must be rotating in a specific direction with a miscellaneous function command (M3; or M4;).
Note that for the G87 (back boring) command, the spindle rotation command is included in the fixed cycle so only the rotation speed command needs to be commanded beforehand.
- (2) If there is data for the basic axis, additional axis or R in the block during the fixed cycle mode, the hole drilling operation will be executed. If there is no data, the hole drilling operation will not be executed.
Note that even when the X axis data exists, the hole will not be drilled if the data is a dwell (G04) time command.
- (3) Command the hole machining data (Q, P, I, J, K) in a block where hole drilling is executed (Block containing a basic axis, additional axis or R data).
- (4) The fixed cycle can be canceled by the G00 to G03 or G33 command besides the G80 command. If these are designated in the same block as the fixed cycle, the following will occur.

m = 00 to 03, 33 n = Fixed cycles

Gm Gn X_Y_Z_R_Q_P_L_F_;

Gm	:	Execution	Gn	:	Ignore	
X_Y_Z	:	Execution	R_Q_P_L	:	Ignore	F : Record

Note that for the G02 and G03 commands, R will be handled as the arc radius.

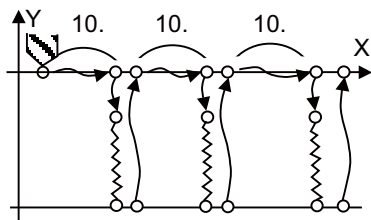
- (5) If M00 or M01 is commanded in a same block with a fixed cycle or during a fixed cycle mode, the fixed cycle will be ignored. Instead, M00 and M01 will be output after positioning. The fixed cycle is executed if X, Y, Z or R is commanded.
- (6) If an M function is commanded in the same block as the fixed cycle command, the M code and MF will be output during the initial positioning. The axis will move to the next operation with FIN (finish signal).
If there is a designation of No. of times, the above control will be executed only for the first drilling.
- (7) If another control axis (ex. rotary axis, additional axis) is commanded in the same block as the fixed cycle control axis, the fixed cycle will be executed after the other control axes start to move.
- (8) If the No. of repetitions L is not designated, L1 will be set. If L0 is designated in the same block as the fixed cycle G code command, the hole machining data will be memorized, but the hole machining will not be executed.

(Example) G73 X_Y_Z_R_Q_P_F_L0_;

Memorize only the codes with an execution address

- (9) When the fixed cycle is executed, only the modal command issued in the fixed cycle program will be valid in the fixed cycle subprogram. The modal of the program which called the fixed cycle will not be affected.
- (10) Other subprograms cannot be called from the fixed cycle subprogram.
- (11) Decimal points in the movement command of the fixed cycle subprogram will be ignored.
- (12) If the No. of repetitions L is 2 or more during the incremental value mode, the positioning will also be incremented each time.

(Example) G91 G81 X10. Z-50. R-20. F100. L3;



13.1.16 Initial Point and R Point Level Return; G98, G99



Function and purpose

Whether to use R point or initial level as the return level in the final sequence of the fixed cycle can be selected.



Command format

G98; ... Initial level return

G99; ... R point level return



Detailed description

The relation of the G98/G99 mode and the number of repetition designation is as shown below.

No. of hole drilling times	Program example	G98 (At power ON, at cancel with M02, M30, and reset button)	G99
Only one execution	G81 X100. Y100. Z-50. R25. F1000 ;	<p>Initial level return is executed.</p>	<p>R point level return is executed.</p>
Two or more executions	G81 X100. Y100. Z-50. R25. L5 F1000 ;	<p>Initial level return is executed for all times.</p>	

(a) First time

(b) Second time

(c) Last time

13.1.17 Setting of Workpiece Coordinates in Fixed Cycle Mode

**Function and purpose**

The designated axis moves in the workpiece coordinate system set for the axis.

The Z axis becomes valid from the R point positioning after positioning is completed or from Z axis movement.

Note

(1) When the workpiece coordinates change, re-program the addresses Z and R, even if the values are the same.

<Example>

G54 Xx1 Yy1 Zz1;

G81 Xx1 Yy2 Zz2 Rr2;

G55 Xx3 Yy3 Zz2 Rr2 Re-command even if Z and R are the same as the previous value.

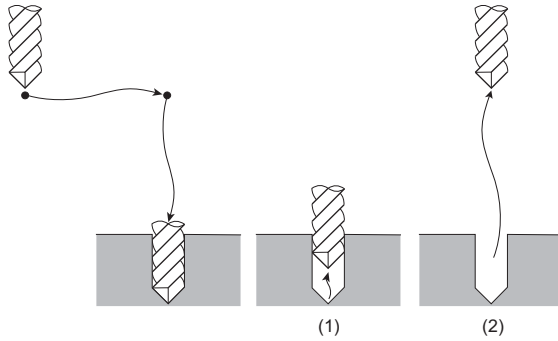
Xx4 Yy4 ;

Xx5 Yy5 ;

13.1.18 Drilling Cycle High-Speed Retract

**Function and purpose**

This function retracts the drill from the hole bottom at high speed in drilling machining. This helps extending the drill life by reducing the time of drilling in vain at hole bottom.



The drill moves up at high-speed (1) and returns to the initial point or R point in rapid traverse (2).

**Command format**

The command format is the same as fixed cycle.



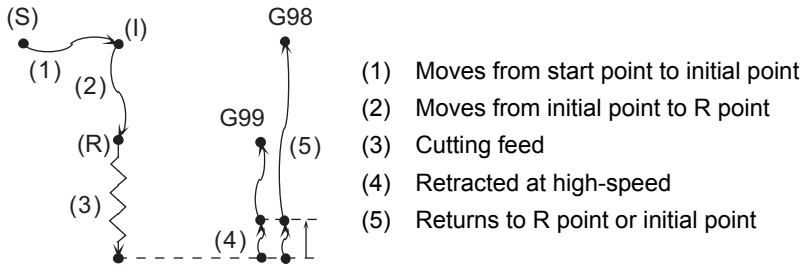
Detailed description

- (1) This function is available only when "#8123 H-spd retract ON" is enabled in the following fixed cycles.
 - G81 (Drill spot drilling cycle)
 - G83 (Deep whole drilling cycle)
 - G73 (Step cycle)
- (2) When "#8123 H-spd retract ON" is ON, the tool is retracted from the hole bottom at high speed using the lost motion compensation function.
 - (a) Set the lost motion compensation type 2 or 3 to the servo parameter. Then set the following parameters to adjust the retract amount.
 - #2170 Lmc1QR (Lost motion compensation gain 1 for high-speed retract)
(correspond to "#2216 SV016(LMC1)" (Lost motion compensation 1))
 - #2171 Lmc2QR (Lost motion compensation gain 2 for high-speed retract)
(correspond to "#2241 SV041(LMC2)" (Lost motion compensation 2))
 - (b) When the lost motion compensation timing, lost motion compensation 3 spring constant, or lost motion compensation 3 viscous coefficient is set in addition to the ordinary lost motion compensations, its setting value depends on the MTB specifications (parameter shown below).
 - #2172 LmcdQR (Lost motion compensation timing for high-speed retract)
(correspond to "#2239 SV039(LMCD)" (Lost motion compensation timing))
 - #2173 LmckQR (Lost motion compensation 3 spring constant for high-speed retract)
(correspond to "#2285 SV085(LMCK)" (Lost motion compensation 3 spring constant))
 - #2174 LmccQR (Lost motion compensation 3 viscous coefficient for high-speed retract)
(correspond to "#2286 SV086(LMCC)" (Lost motion compensation 3 viscous coefficient))
 - (c) If the drilling axis is synchronously controlled, set the same value in both parameters for master axis and slave axis.
- (3) While G80 (Fixed cycle cancel) command is issued, this function will be canceled by issuing any other fixed cycle of the same group (Group 9) or any Group 1 command.
- (4) This function is invalid during the following command modal:
In this case, the drill moves in the ordinary rapid traverse even if "#8123" is enabled.
 - G43.1 (Tool length compensation in the tool axis direction)
 - G43.4, G43.5 (Tool center point control)
 - G68 (3-dimensional coordinate conversion)



Details of Operation

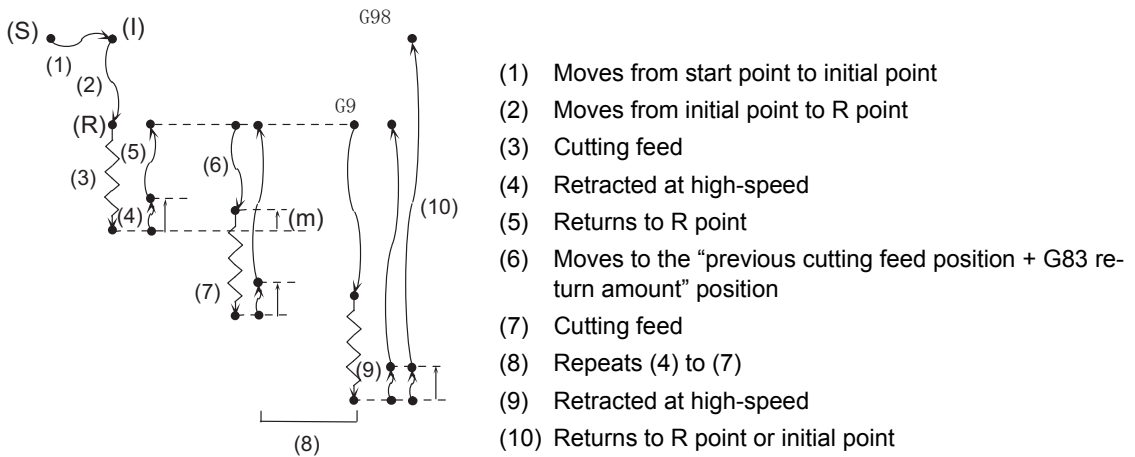
Operation at G81 command



- (I) Initial point
- (R) R point
- (S) Start point

During single block operation, the axis stops after (1), (2) and (5) only.

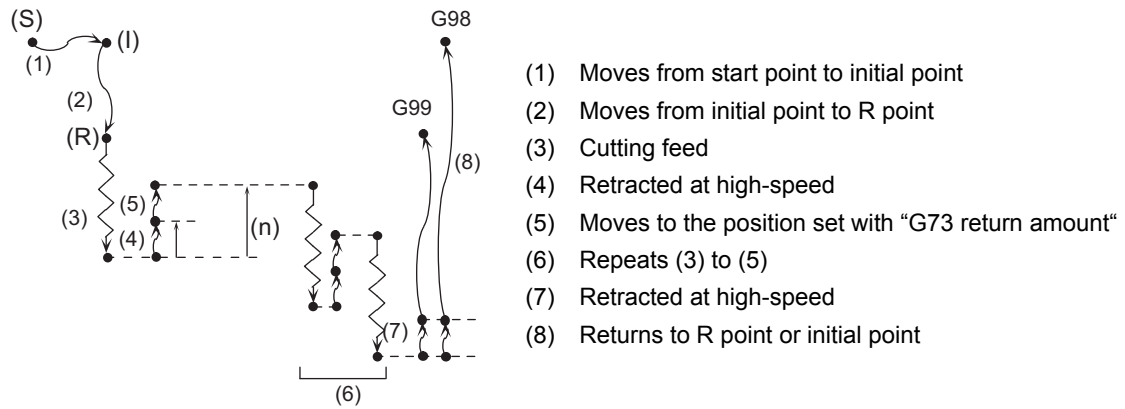
Operation at G83 command



- (I) Initial point
- (R) R point
- (S) Start point
- (m) G83 Return amount

During single block operation, the axis stops after (1), (2) and (10) only.

Operation at G73 command



- (1) Moves from start point to initial point
- (2) Moves from initial point to R point
- (3) Cutting feed
- (4) Retracted at high-speed
- (5) Moves to the position set with "G73 return amount"
- (6) Repeats (3) to (5)
- (7) Retracted at high-speed
- (8) Returns to R point or initial point

(I) Initial point

(S) Start point

(R) R point

(n) G73 Return amount

During single block operation, the axis stops after (1), (2) and (8) only.

If a dwell command is issued, the high-speed retract will be executed after the command.

13.1.19 Acceleration/Deceleration Mode Change in Hole Drilling Cycle

**Function and purpose**

This function switches the acceleration/deceleration mode for fixed cycle for drilling between the inclination constant method and the acceleration/deceleration after interpolation.

**Command format**

The command formats are the same as those of the fixed cycles G83, G87, and G83.2.

**Detailed description**

With parameter "#1253 set25/bit2" (Acceleration/deceleration mode change in hole drilling cycle) enabled, operation will be as follows.

- (1) Acceleration/deceleration mode will be either linear or soft method. (Unless soft acceleration/deceleration is applied, the linear method will always be applied.)
- (2) The operation follows the parameter settings that determine whether the inclination-constant or post-interpolation acceleration/deceleration is applied.

"#2001 rapid" (rapid traverse rate) and "#2004 G0tL" (G0 time constant (linear)) provide G0 (rapid traverse) acceleration/deceleration inclination, and "#2002 clamp" (cutting feedrate for clamp) and "#2007 G1tL" (G1 time constant (linear)) provide G1 (cutting feed) acceleration/deceleration inclination.

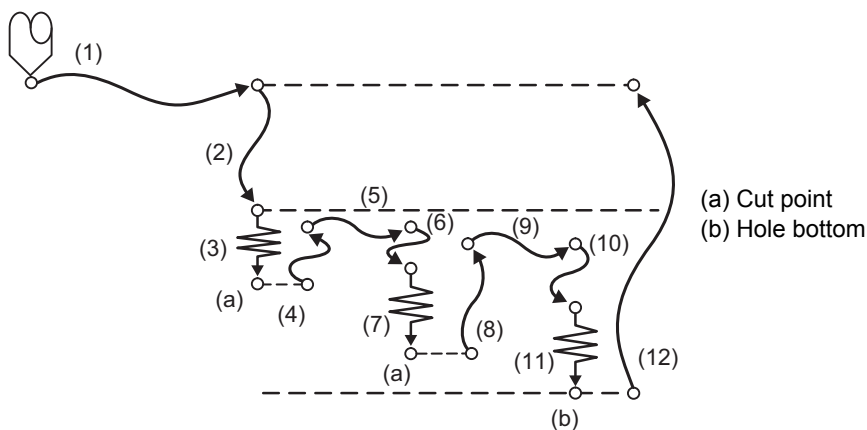
Refer to "Rapid Traverse Constant Inclination Acceleration/Deceleration" for details of constant inclination acceleration/deceleration.



Operation example

Operation example of "acceleration/deceleration mode change in hole drilling cycle" being enabled

The below illustrates the processes of hole-bottom deceleration check of a drilling axis following the parameter "#19417 Hole dec check 2" settings.



#19417		G81	G82	G83	G73
0	(a) Cut point	Deceleration Check		Perform no deceleration check.	
	(b) Hole bottom	Perform no deceleration check.			
1	(a) Cut point	Perform no command deceleration check.		Command deceleration check	
	(b) Hole bottom	Command deceleration check			
2	(a) Cut point	Command deceleration check			Perform in-position check (sv024).
	(b) Hole bottom	Perform in-position check (sv024).			

13.2 Special Fixed Cycle



Function and purpose

The special fixed cycle is used with the standard fixed cycle.

Before using the special fixed cycle, record the hole machining data except for the positioning data (except for X, Y plane) by the standard fixed cycle.

The tool is positioned to the hole drilling position when the special fixed cycle is executed. The drilling operation is executed with the fixed cycle for drilling.

Even after the special fixed cycle is executed, the recorded standard fixed cycle will be kept until canceled.

If the special fixed cycle is designated when not in the fixed cycle mode, only positioning will be executed, and the hole drilling operation will not be carried out.

If the special fixed cycle is commanded without commanding the fixed cycle for drilling, positioning will be executed following the current 01 group modal G code.

13.2.1 Bolt Hole Cycle; G34



Function and purpose

This function is used to drill "n" holes, dividing the circumference by "n", on the circumference with radius R centering the coordinates designated with X and Y. The drilling starts at the point which makes the angle θ with X axis. The hole drilling operation at each hole will follow the standard fixed cycle.

The movement between hole positions will all be done in the G00 mode. G34 will not hold the data after the command is completed.



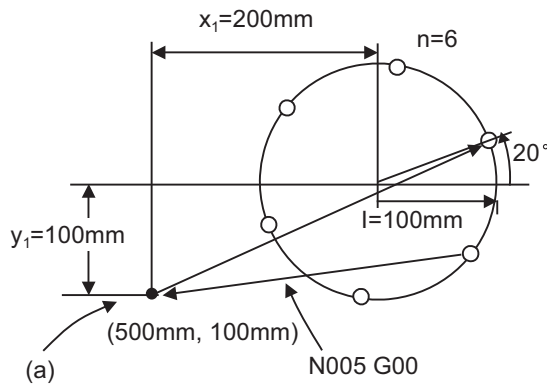
Command format

G34 Xx1 Yy1 Ir J θ Kn ;

Xx1,Yy1	Positioning of bolt hole cycle center. This will be affected by G90/G91.
Ir	Radius r of the circle. The unit follows the input setting unit, and is given with a positive No.
J θ	Angle θ of the point to be drilled first. The CCW direction is positive. (The decimal point position will be the degree class. If there is no decimal point, the unit will be 0.001°.)
Kn	No. of holes to be drilled: n 1 to 9999 can be designated, but 0 cannot be designated. When the value is positive, positioning will take place in the CCW direction, and when negative, will take place in the CW direction. If "0" is designated, a program error (P221) occurs.



Program example



```

N001 G91;
N002 G81 Z-10.000 R5.000 L0 F200 ;
N003 G90 G34 X200.000 Y100.000 I100.000 J20.000
      K6;
N004 G80 ; ----- (G81 cancel)
N005 G90 G00 X500.000 Y100.000. ;
    
```

(a) Position before G34 execution

As shown in the example, the tool position after the G34 command is completed is above the final hole. When moving to the next position, the coordinate value must be calculated to issue the command with an incremental value. Thus, use of the absolute value mode is handy.

Note

- (1) If an address other than the selected plane's vertical axis, horizontal axis, G, N, I, J, K, H, O, P, F, M, S or 2nd miscellaneous function is issued in the same block as the G34 command, a program error (P32) will occur.

13.2.2 Line at Angle; G35

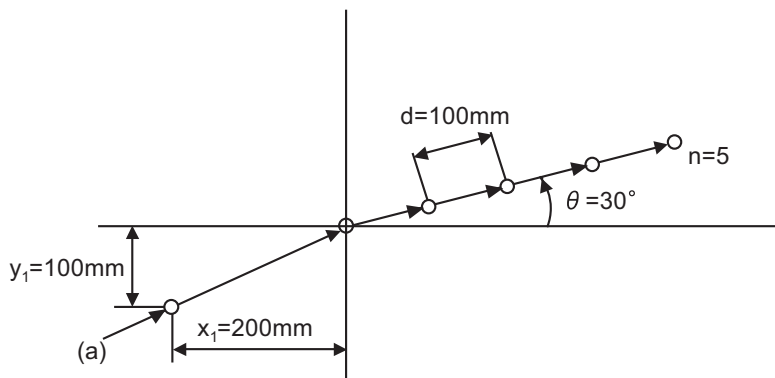
**Function and purpose**

Using the position designated by X and Y as the start point, the n holes will be drilled with interval d in the direction which makes an angle θ with X axis. The hole drilling operation at each hole will follow the standard fixed cycle. The movement between hole positions will all be done in the G00 mode. G35 will not hold the data after the command is completed.

**Command format**

G35 Xx1 Yy1 Id J θ Kn ;

Xx1,Yy1	Designation of start point coordinates. This will be affected by G90/G91.
Id	Interval d. The unit follows the input setting unit. If d is negative, the drilling will take place in the direction symmetrical to the center of the start point.
J θ	Angle θ . The CCW direction is positive. (The decimal point position will be the degree class. If there is no decimal point, the unit will be 0.001°.)
Kn	Number of holes: n 1 to 9999 can be designated, and the start point is included.

**Program example**

```
G91 ;
G81   Z-10.000 R5.000 L0 F100 ;
G35   X200.000 Y100.000
      I100.000 J30.000 K5 ;
```

(a) Position before G35 execution

Note

- (1) If the K command is K0 or if there is no K command, the program error (P221) will occur.
- (2) If the K value is more than four digits, the last four digits will be valid.
- (3) If an address other than the selected plane's vertical axis, horizontal axis, G, N, I, J, K, H, O, P, F, M, S or 2nd miscellaneous function is issued in the same block as the G35 command, a program error (P32) will occur.
- (4) If G command of group 0 is issued in the same block as the G35 command, the command issued later has the priority.
(Example) G35 G28 Xx1 Yy1 Ii1 Jj1 Kk1 ; G35 is ignored G 28 is executed as Xx1 Yy1
- (5) If there is G72 to G89 commands in the same block as the G35 command, the fixed cycle will be ignored, and the G35 command will be executed.

13.2.3 Arc; G36



Function and purpose

The "n" holes aligned with the angle interval $\Delta\theta$ will be drilled starting at the point which makes the angle θ with the X axis on the circumference with a radius R centering the coordinates designated with X and Y. The hole drilling operation at each hole will follow the standard fixed cycle.

The movement between hole positions will all be done in the G00 mode. G36 will not hold the data after the command is completed.



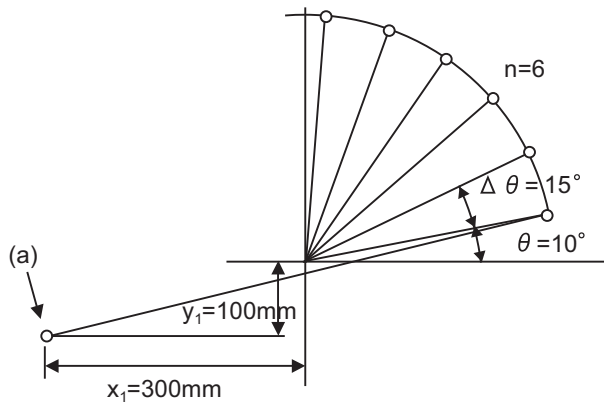
Command format

G36 Xx1 Yy1 Ir J θ P $\Delta\theta$ Kn ;

Xx1,Yy1	Center coordinates of arc. This will be affected by G90/G91.
Ir	Radius r of arc. The unit follows the input setting unit, and is given with a positive No.
J θ	Angle θ of the point to be drilled first. The CCW direction is positive. (The decimal point position will be the degree class. If there is no decimal point, the unit will be 0.001°.)
P $\Delta\theta$	Angle interval θ . When the value is positive, the drilling will take place in the CCW direction, and in the CW direction when negative. (The decimal point position will be the degree class. If there is no decimal point, the unit will be 0.001°.)
Kn	No. of holes n to be drilled. The setting range is 1 to 9999.



Program example



```

N001 G91 ;
N002 G81 Z-10.000 R5.000 F100 ;
N003 G36 X300.000 Y100.000 I300.000 J10.000
      P15000 K6 ;
    
```

(a) Position before G36 execution

Note

- (1) If an address other than the selected plane's vertical axis, horizontal axis, G, N, I, J, K, H, O, P, F, M, S or 2nd miscellaneous function is issued in the same block as the G36 command, a program error (P32) will occur.

13.2.4 Grid; G37.1



Function and purpose

The n_x points on a grid are drilled with an interval Δx parallel to the X axis, starting at the position designated with X, Y. The hole drilling operation at each hole will follow the standard fixed cycle. The movement between hole positions will all be done in the G00 mode. G37.1 will not hold the data after the command is completed.



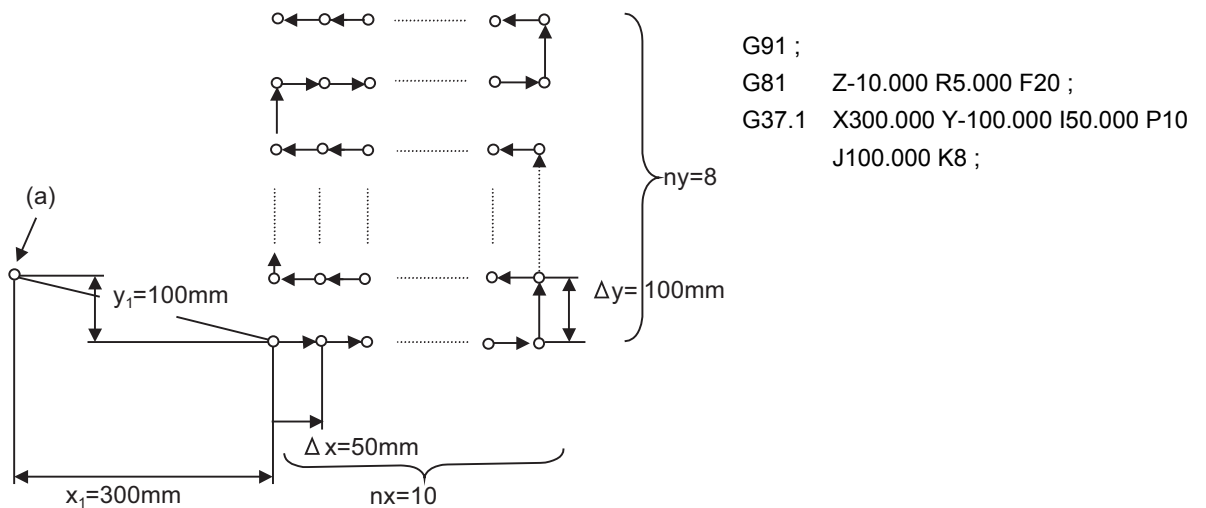
Command format

G37.1 Xx1 Yy1 IΔx Pnx JΔy Knny ;

Xx1,Yy1	Designate the coordinates at the start point. This will be affected by G90/G91.
I Δx	Interval Δx of the X axis. The unit will follow the input setting unit. If Δx is positive, the interval will be in the forward direction looking from the start point, and when negative, will be in the reverse direction looking from the start point.
Pnx	No. of holes n_x in the X axis direction. The setting range is 1 to 9999.
J Δy	Interval Δy of the Y axis. The unit will follow the input setting unit. If Δy is positive, the interval will be in the forward direction looking from the start point, and when negative, will be in the reverse direction looking from the start point.
Knny	No. of holes n_y in the Y axis direction. The setting range is 1 to 9999.



Program example



(a) Position before G37.1 is executed

Note

- (1) If the P and K commands are P0 or K0, or if there is no P or K command, the program error (P221) will occur. If the P or K value is more than four digits, the last four digits will be valid.
- (2) If an address other than the selected plane's vertical axis, horizontal axis, G, N, I, J, K, H, O, P, F, M, S or 2nd miscellaneous function is issued in the same block as the G37.1 command, a program error (P32) will occur.
- (3) If G command of group 0 is issued in the same block as the G37.1 command, the command issued later has the priority.
- (4) If there is G72 to G89 command in the same block as the G37.1 command, the fixed cycle will be ignored, and the G37.1 command will be executed.
- (5) If the G22/G23 command is programmed in the same block as the G37.1 command, the G22/G23 command will be ignored, and the G37.1 command will be executed.

13.3 Fixed Cycle for Turning Machining



Function and purpose

When performing rough cutting and other cuttings by turning machining, fixed cycles are effective in simplifying machining programs. The whole commands can be performed in a single block, which normally requires several blocks. The types of fixed cycles for turning machining are listed below.

G code	Function
G174	Longitudinal cutting cycle
G175	Thread cutting cycle
G176	Face cutting cycle



Detailed description

- (1) Fixed cycle commands are modal G codes. They are valid until another command in the same modal group or a cancel command is issued.
- (2) The fixed cycle call becomes the movement command block call.
By the movement command block call, the fixed cycle macro subprogram is called only when there is an axis movement command during the fixed cycle mode. It is executed until the fixed cycle is canceled.
- (3) A manual interruption can be applied while a fixed cycle for turning machining is being executed. Upon completion of the interrupt, however, the tool must be returned to the position where the manual interruption was applied and then the fixed cycle for turning machining should be resumed.
If it is resumed without returning the tool, all subsequent operations will deviate from the original path by the manual interruption amount.

13.3.1 Longitudinal Cutting Cycle; G174

**Function and purpose**

The longitudinal cutting cycle performs continuous straight and taper cutting in the longitudinal direction.

**Command format****Straight cutting**

G174 X__ Z__ F__ ;

X	X axis end point coordinate (*1)
Z	Z axis end point coordinate (*1)
F	Feedrate

Taper cutting

G174 X__ Z__ R__ F__ ;

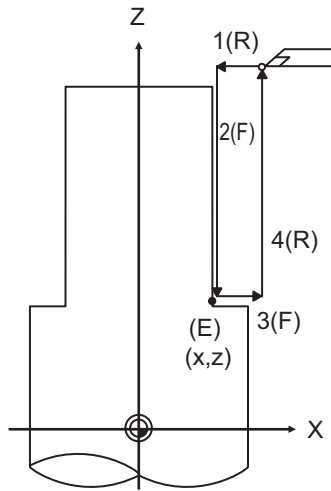
X	X axis end point coordinate (*1)
Z	Z axis end point coordinate (*1)
R	Taper depth (radius designation, incremental value, sign required)
F	Feedrate

(*1) If the commanded axis and selected plane are different when the fixed cycle for turning machining is commanded, or if the movement amount is not specified in either or both of the selected plane axis commands, a program error (P114) will occur. Whether this is judged to be a program error depends on the MTB specifications (parameter "#1241 set13/bit4" ("fixed cycle for turning machining" selected-plane axis check disabled)).



Detailed description

Straight cutting

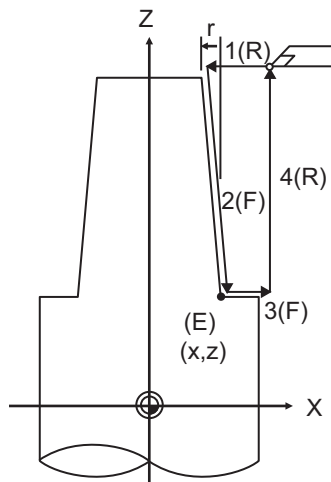


(R) Rapid traverse

(F) Cutting feed

(E) End point coordinates

Taper cutting



(R) Rapid traverse

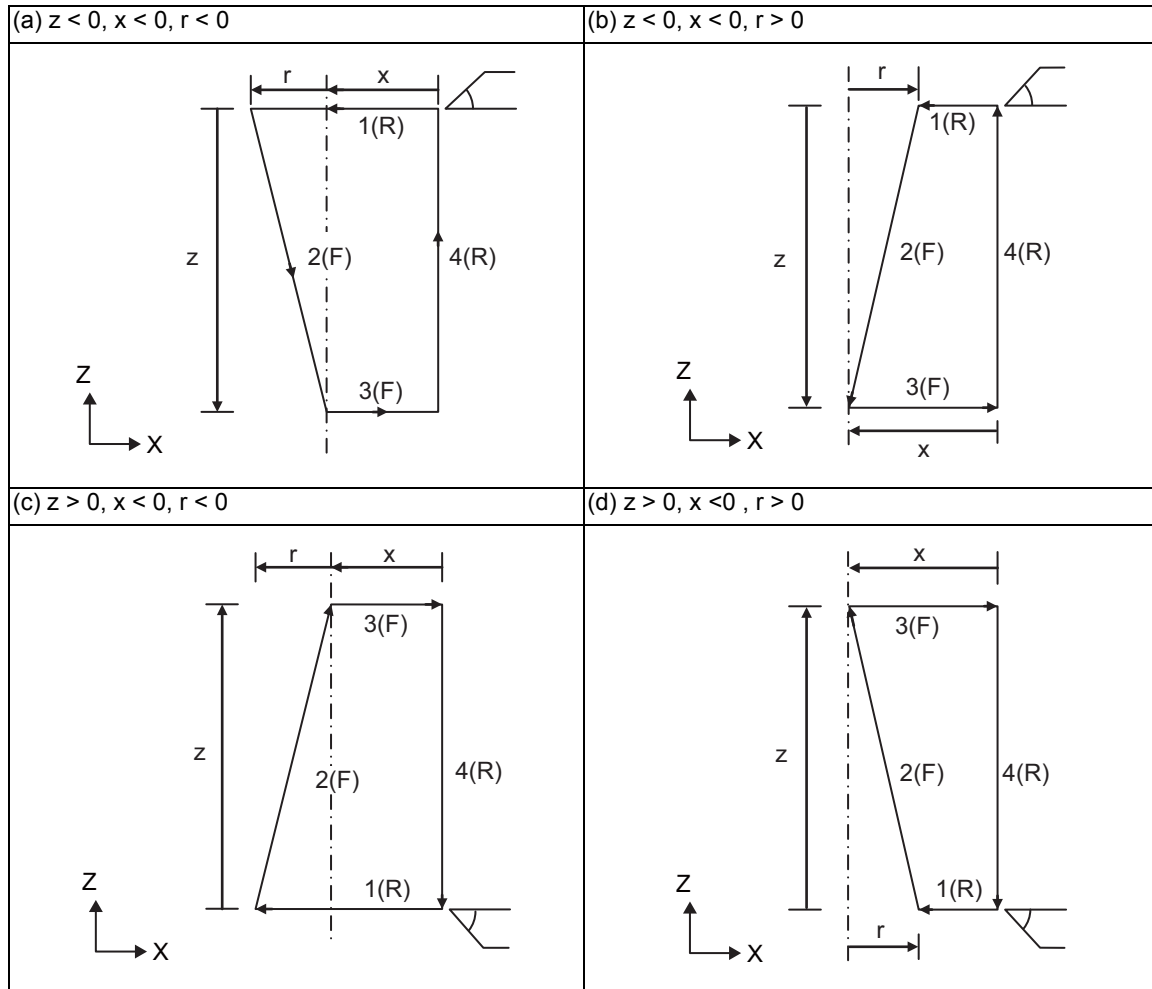
(F) Cutting feed

(E) End point coordinates

Detailed description

With a single block, the tool stops at the end points of operations 1, 2, 3 and 4 shown above.

Depending on the signs of x, z and r, the following shapes are created.



Program error (P191) will occur in (b) and (c) unless the following condition is satisfied.

$$|x| \geq |r|$$

13.3.2 Thread Cutting Cycle; G175

**Function and purpose**

Thread cutting cycle is a fixed cycle which performs straight and taper thread cutting.

**Command format****Straight thread cutting**

```
G175 X__ Z__ F/E__ Q__;
```

X	X axis end point coordinate (*1)
Z	Z axis end point coordinate (*1)
F/E	Lead of long axis (axis which moves most) direction
Q	Thread cutting start shift angle

Taper thread cutting

```
G175 X__ Z__ R__ F/E__ Q__;
```

X	X axis end point coordinate (*1)
Z	Z axis end point coordinate (*1)
R	Taper depth (radius designation, incremental value, sign required)
F/E	Lead of long axis (axis which moves most) direction
Q	Thread cutting start shift angle

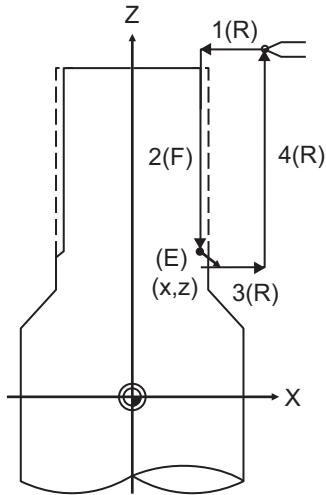
(*1) If the commanded axis and selected plane are different when the fixed cycle for turning machining is commanded, or if the movement amount is not specified in either or both of the selected plane axis commands, a program error (P114) will occur. Whether this is judged to be a program error depends on the MTB specifications (parameter "#1241 set13/bit4" ("fixed cycle for turning machining" selected-plane axis check disabled)).



Detailed description

Straight thread cutting

With a single block, the tool stops at the end points of operations 1, 3 and 4.



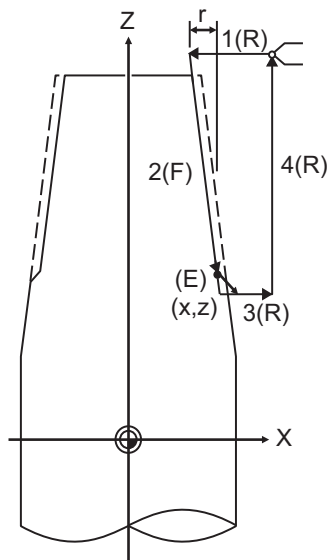
(R) Rapid traverse

(F) Thread cutting cycle

(E) End point coordinates

Taper thread cutting

With a single block, the tool stops at the end points of operations 1, 3 and 4.



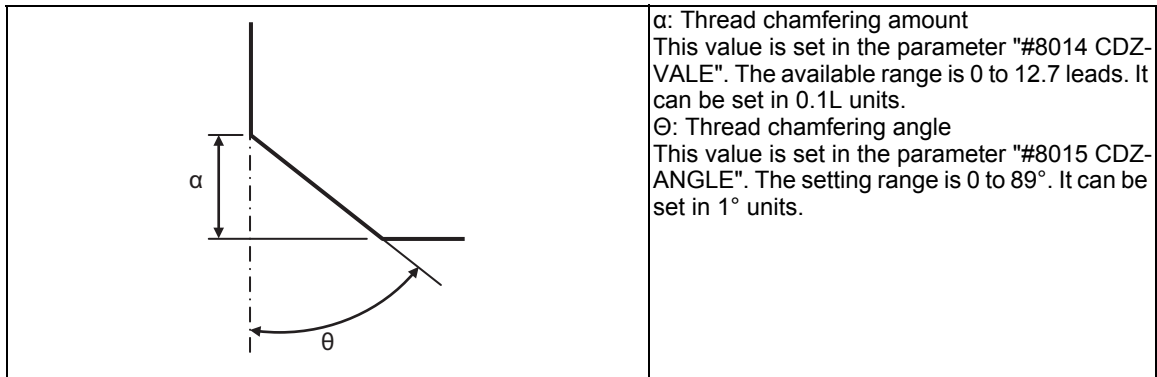
(R) Rapid traverse

(F) Thread cutting cycle

(E) End point coordinates

Detailed description

(1) Details for chamfering



If thread chamfering amount is bigger than a thread lead length, a program error(P192) will occur before starting thread cutting.

The operation in the thread cutting cycle when the result of the thread cutting tool retract with chamfering exceeds the cycle start point depends on the MTB specifications.

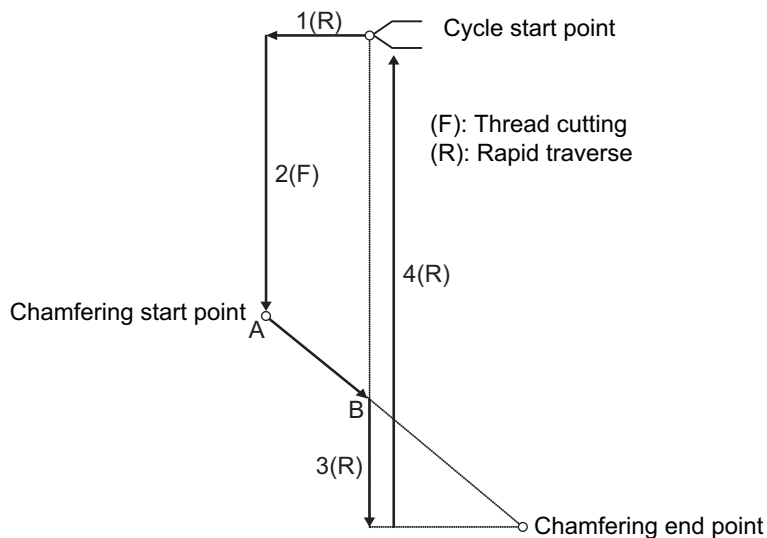
<#1270 ext06/bit4 = 0>

A program error (P192) will occur before thread cutting is started.

<#1270 ext06/bit4 = 1>

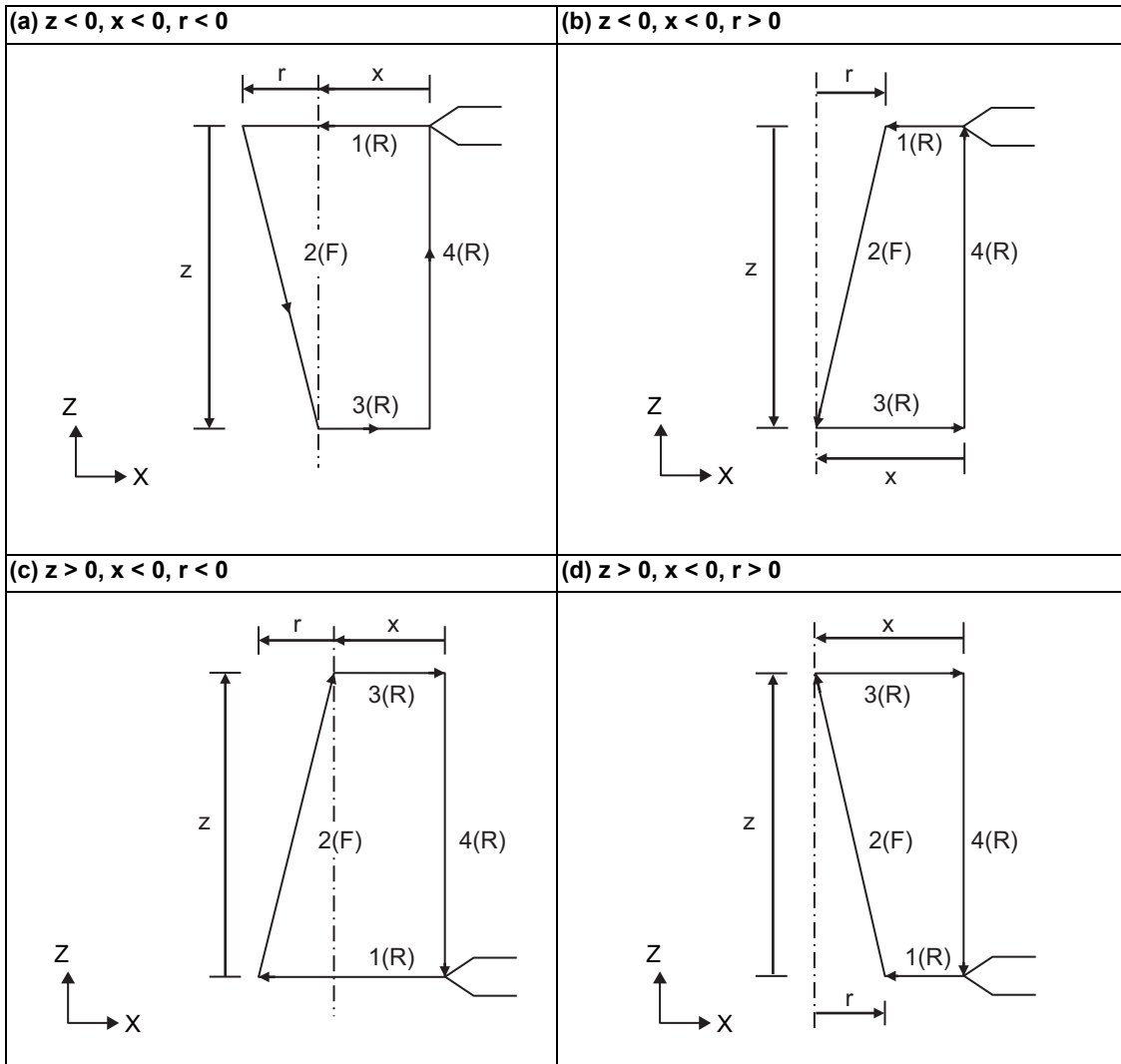
A program error (P192) will not occur. After the thread cutting tool retraction is stopped and the thread cutting block ends at the cycle start point ("A" in the figure), the axis moves to the end coordinate ("B" in the figure) at rapid traverse feed.

Thread cutting tool retraction when "#1270 ext06/bit4" is set to "1"



- (2) When the feed hold is applied during the thread cutting cycle, automatic operation will stop if it is applied when thread cutting is not being executed or when a cutting command is issued but the axis is yet to move. When the feed hold is applied during thread cutting, the thread cutting cycle retract is performed.

(3) Depending on the signs of x, z and r, the following shapes are created.



Program error (P191) will occur in (b) and (c) unless the following condition is satisfied.

$$|x| \geq |r|$$

<Note>

- ♦The thread cutting start shift angle is not modal. If there is no Q command with G175, this will be handled as "Q0".
- ♦If a value exceeding "360.000" is commanded in G175 Q address, this will be handled as "Q360.000".
- ♦G175 cuts one row with one cycle. To cut two rows, change the Q value, and issue the same command.
- ♦Also, follow the precautions for the thread cutting command (G33).

13.3.3 Face Cutting Cycle; G176

**Function and purpose**

The face cutting cycle performs continuous straight and taper cutting in the face direction.

**Command format****Straight cutting**

```
G176 X__ Z__ F__ ;
```

X	X axis end point coordinate (*1)
Z	Z axis end point coordinate (*1)
F	Feedrate

Taper cutting

```
G176 X__ Z__ R__ F__ ;
```

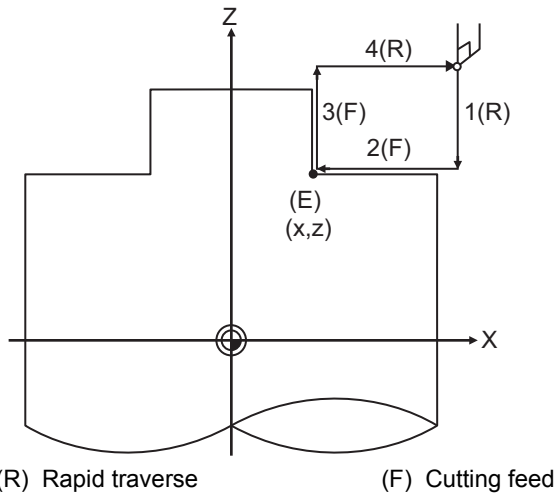
X	X axis end point coordinate (*1)
Z	Z axis end point coordinate (*1)
R	Taper depth (radius designation, incremental value, sign required)
F	Feedrate

(*1) If the commanded axis and selected plane are different when the fixed cycle for turning machining is commanded, or if the movement amount is not specified in either or both of the selected plane axis commands, a program error (P114) will occur. Whether this is judged to be a program error depends on the MTB specifications (parameter "#1241 set13/bit4" ("fixed cycle for turning machining" selected-plane axis check disabled)).



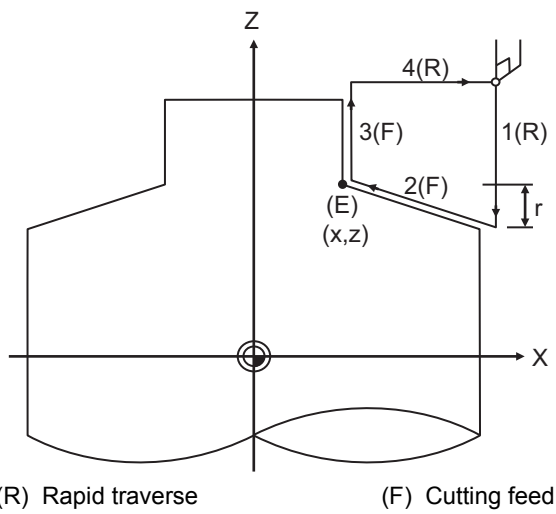
Detailed description

Straight cutting



(E) End point coordinates

Taper cutting

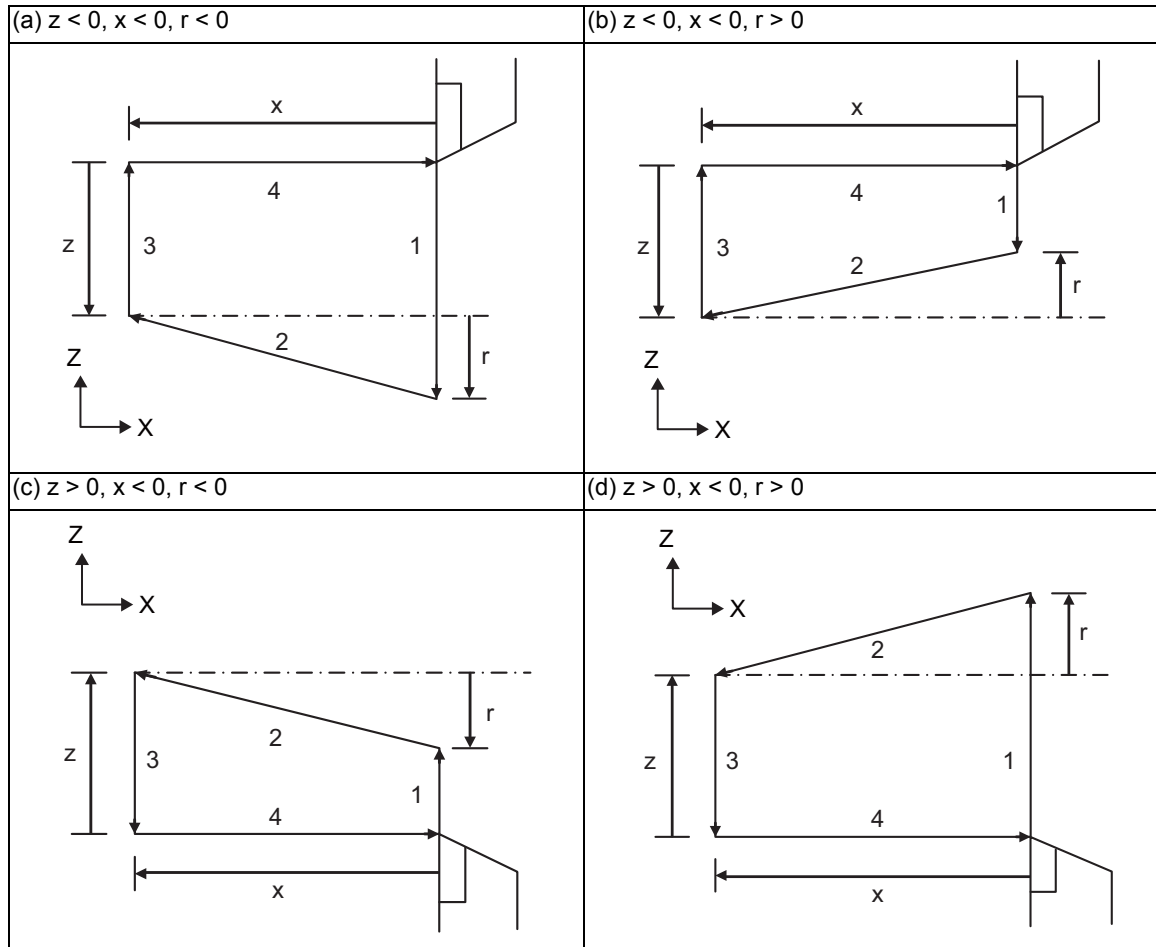


(E) End point coordinates

Detailed description

With a single block, the tool stops at the end points of operations 1, 2, 3 and 4 shown above.

Depending on the signs of x, z and r, the following shapes are created.



Program error (P191) will occur in (b) and (c) unless the following condition is satisfied.

$$|z| \geq |r|$$

Macro Functions

14.1 Subprogram Control; M98, M99, M198

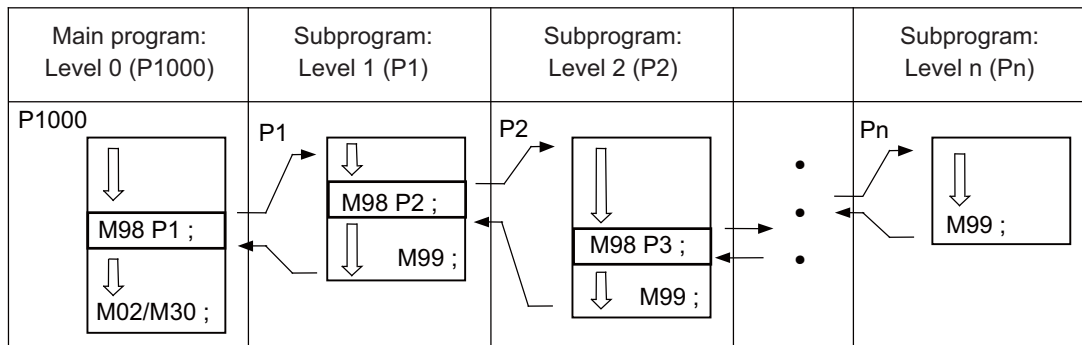
14.1.1 Subprogram Call; M98, M99



Function and purpose

Fixed sequences or repeatedly used parameters can be stored in the memory as subprograms that can then be called from the main program when required. M98 serves to call subprograms and M99 serves to return operation from the subprogram to the main program. Furthermore, it is possible to call other subprograms from particular subprograms.

The nesting depth depends on the model.



The table below shows the functions that can be executed by adding and combining the tape memory/editing functions, subprogram control functions and fixed cycle functions.

	Case 1	Case 2	Case 3	Case 4
1. Tape memory and editing	Yes	Yes	Yes	Yes
2. Subprogram control	No	Yes	Yes	No
3. Fixed cycles	No	No	Yes	Yes
Function				
1. Memory mode	○	○	○	○
2. Tape editing (main memory)	○	○	○	○
3. Subprogram call	×	○	○	×
4. Subprogram variable designation (*2)	×	○	○	×
5. Subprogram nesting call (*3)	×	○	○	×
6. Fixed cycles	×	×	○	○
7. Editing subprogram for fixed cycle	×	×	○	○

(*1) Symbol "○" denotes available functions and symbol "×" denotes unavailable functions.

(*2) Variables cannot be transferred with the M98 command, but variable commands in subprograms are available if the variable command specifications are provided.

(*3) The depth of nesting call depends on the model.



Command format

Subprogram call

```
M98 P__ H__ L__ ,D__ ;
```

```
M98 <file name> H__ L__ ,D__ ;
```

P	Program number in subprogram to be called (own program if omitted) Note that P can be omitted only for memory mode, MDI operation, high-speed program server operation, SD card operation, hard disk operation, or USB operation. (Numeric value of up to 8 digits) Use a parameter to specify a 4- or 8-digit subprogram No. starting with O. However, if the commanded value is bigger than the digit number set with parameter, a subprogram call is carried out as commanded.
<File name>	File name A file name can be specified instead of a program No. In this case, enclose the file name inside brackets < >. (The file name can have up to 32 characters including the extension.) (Example) M98 <BUHIN-12. RAF> ;
H	Program sequence number in subprogram to be called (head block if omitted)
L	Number of subprogram repetitions (If omitted, it is assumed to be "L1", and processing is not carried out when "L0" is set.) (1 to 9999 times depending on the 4-digit value) For instance, "M98 P1 L3;" is equivalent to the following: M98 P1 ; M98 P1 ; M98 P1 ;
,D [M800/M80]	Subprogram device No. (0 to 4) The subprogram is searched according to the setting of parameter "#8890 Subpro srch odr D0" to "#8894 Subpro srch odr D4" when ",D" is omitted. The device No. is set to the parameter, such as "#8880 Subpro stor D0: dev".
,D [C80]	Subprogram device No. (0 to 1) When ",D" command is omitted, the subprogram is searched in the area where main program is being executed. ,D0: Basic area (memory) ,D1: Extended area (memory2) (*1)

(*1) To use memory2 (extended area), additional specification of part program storage capacity of 2560 [m] or 5120 [m] is required. If the specification is invalid, program error occurs when ",D1" is designated.

(*2) Program in extended area can be called as a subprogram from the program in basic area and program within the other extended area. Note that program in extended area can call a program in basic area as well.

Return to main program from subprogram

```
M99 P__ ;
```

P	Sequence No. of return destination (returned to block that follows the calling block)
---	---



Detailed description

Creating and registering subprograms

Subprograms have the same format as machining programs for normal memory mode, except that the subprogram completion instruction M99(P_); must be registered as an independent block in the last block.

O***** ;	Program No. as subprogram No.
..... ; : ;	Main body of subprogram
M99 ;	Subprogram return command
%(EOR)	Registration completion code

- (1) The above program is registered by editing operations at the setting and display unit. For further details, refer to the section on "program editing" in the Instruction Manual.
- (2) Only those subprogram Nos. ranging from 1 to 99999999 designated by the optional specifications can be used. When there are no program Nos. on the tape, they are registered as the setting No. for "program input."
- (3) If a program is called from a subprogram over the nesting depth determined in the specifications, the program error (P230) will occur.
- (4) Main programs and subprograms are registered in the order they were read without distinction. Therefore, main programs and subprograms should not be given the same Nos. (If they are, error "E11" will be displayed at registration.)
- (5) Main programs can be executed during memory, tape, MDI, or BTR mode, but subprograms must be in the memory mode.
- (6) Besides the M98 command, subprogram nesting is subject to the following commands:
 - G65: Macro call
 - G66: Modal call
 - G66.1: Modal call
 - G Code call
 - Miscellaneous function call
 - MDI interruption
 - Automatic tool length measurement
 - Macro interruption
 - Multiple-step skip function
- (7) The following commands can be called even if the nesting depth exceeds the determined depth in the specifications because they are not subject to subprogram nesting.
 - Fixed cycles
 - Pattern cycles
- (8) To repeatedly use the subprogram, it can be repeated I1 times by programming M98 Pp1 L11;
- (9) When using the multi-part system, if the subprogram attributed to the part system with the call command is empty, the subprogram call operation will change according to the parameters. (These parameters depend on the MTB specifications.)

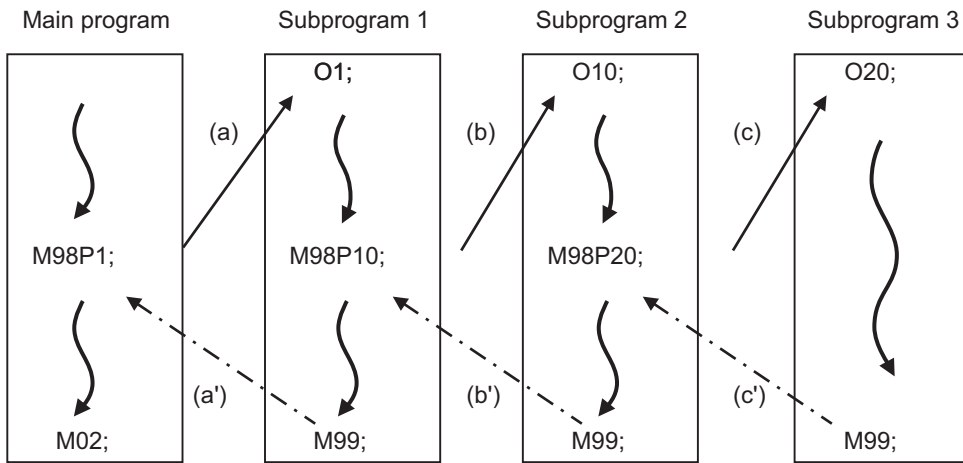
#1285 ext21/ bit1	Description
OFF	The subprogram registered in the memory for the selected part system is called out.
ON	The subprogram registered in the memory for the selected part system is called out. If the subprogram in the selected part system is empty, the subprogram with the same No. in the 1st part system is called out.



Program example

Program example 1

When there are 3 subprogram calls (known as 3 nesting levels)

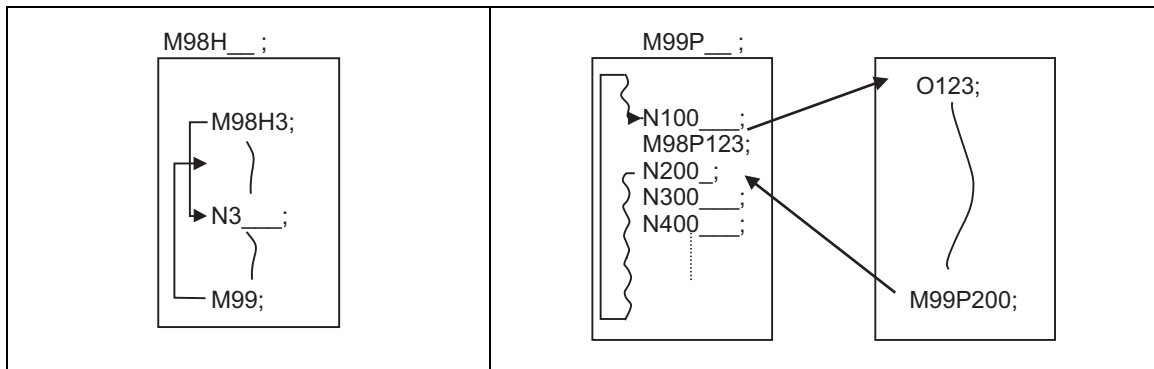


Sequence of execution: (a)-(b)-(c)-(c')-(b')-(a')

- (1) For nesting, the M98 and M99 commands should always be paired off on a 1:1 basis; (a)' for (a), (b)' for (b), etc.
- (2) Modal information is rewritten in the order of execution sequence without distinction between main programs and subprograms. Therefore, after calling a subprogram, attention must be paid to the modal data status when programming.

Program example 2

The M98 H_ ; M99 P_ ; commands designate the sequence Nos. in a program with a call instruction.





Precautions

- (1) The program error (P232) will occur when the designated P (program No.) cannot be found.
- (2) The M98 P_ ; M99 ; block does not perform a single block stop. If any address except O, N, P, L or H is used, single block stop can be executed. (With "X100. M98 P100 ;", the operation branches to O100 after X100. is executed.)
- (3) When M99 is commanded by the main program, operation returns to the head. (This is the same as for MDI.)
- (4) Branching from tape and BTR mode to the subprogram with M98 P_ ; is possible, but the return destination sequence No. cannot be designated with M99 P_ ; . (P_ is ignored.)
- (5) Note that it takes time to search when the sequence No. is designated by M99 P_ ;.
- (6) When using a file name for the subprogram, specify the file name with 32 characters or less, including the extension. If a file name exceeding 32 characters is specified, a program error (P232) will occur.
- (7) All the programs are registered as files. For example, when calling a file "0100" as a subprogram, "0100" cannot be searched with M98P100 or M98P0100. When numerical values are specified after P, 0 is ignored. In this case, it is regarded that the program No. (file) "100" is specified. To call a program like "0100" , specify the file name using the M98<0100> format.
- (8) A subprogram added O No. is searched with the parameter setting (#8129="1" or "2") which calls a subprogram with O No. as priority.
If a subprogram with O No. is not found, a subprogram with a name specified with the P command is searched.

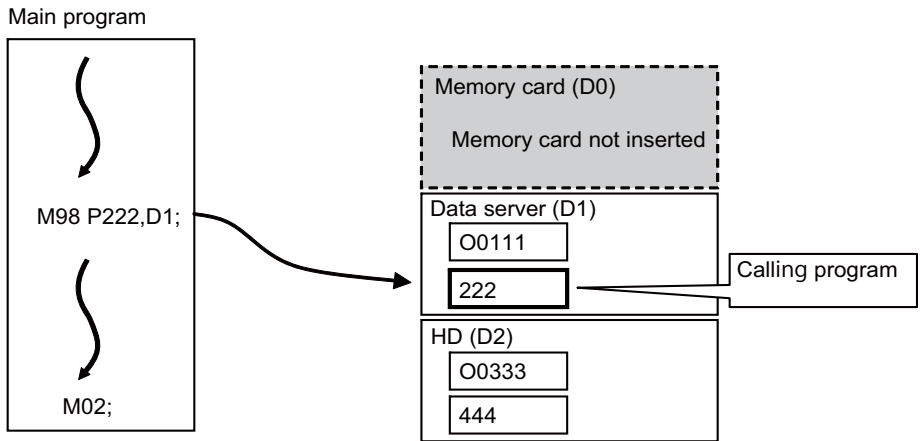
<Note>

- To prevent any unintended program call, avoid using program names that may be confused. (For example, 123, O0123 and O0000123 can be considered identical.)

Refer to the next page for operation examples of subprogram search with the setting which calls subprograms with O No. as priority.

(a) With designation of device No.

Only the designated devices are subject to search. (The following is an example of M800/M80 series.)



[Parameter setting]

- #8129 Subpro No. select = 1 (Four-digit program No. beginning with O No.)
- #8880 Subpro stor D0 dev = R (Memory card)
- #8882 Subpro stor D1 dev = D (Data server)
- #8884 Subpro stor D2 dev = G (Hard disk)

(b) Without designation of device No.

[M800/M80]

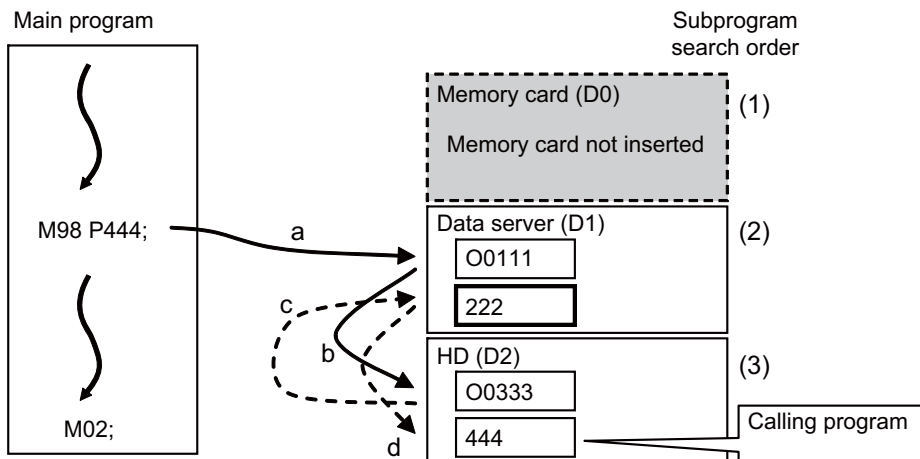
A subprogram with O No. is searched according to the settings of #8890 (D0 in order of subprogram search) to #8894 (D4 in order of subprogram search). (Refer to the solid line arrows "a" and "b" in the figure.)

If a subprogram with O No. is not found, subprograms with a name designated with the P command are searched in order of the parameter setting. (Refer to the broken lines "c" and "d" in the figure.)

If none of the designated subprogram storage locations are subject to search, memories are searched.

Note

- If any device or directory designated as the subprogram storage location is not found due to a reason such as absence, poor contact and contact failure of a memory card, the said device or directory will be excluded from the search target.



[Parameter setting]

#8129 Subpro No. select = 1 (Four-digit program No. beginning with O No.)

#8880 Subpro stor D0 dev = R (Memory card)

#8882 Subpro stor D1 dev = D (Data server)

#8884 Subpro stor D2 dev = G (Hard disk)

#8890 Subpro srch odr D0 = 1

#8891 Subpro srch odr D1 = 2

#8892 Subpro srch odr D2 = 3

[C80]

Subprogram is searched inside of the currently selected device.

- (9) When a program in an external device such as a USB memory device is executed, a period of processing time is required in the subprogram call or in the instruction to change the flow of the program such as GOTO or DO-END; therefore, interpolation may be decelerated or stopped.

14.1.2 Subprogram Call; M198

**Function and purpose**

Programs registered in the SD card can be called as a subprograms. To call a program in the SD card as a subprogram, command the following with the main program.

**Command format****Subprogram call**

```
M198 P__ L__ ;
```

```
M198 <File name> L__ ;
```

P	Program No. in SD card to be called as a subprogram. (Max. 8 digits) Use a parameter to specify a 4- or 8-digit subprogram No. starting with O. However, if the commanded value is bigger than the digit number set with parameter, a subprogram call is carried out as commanded.
<File name>	File name A file name can be specified instead of a program No. In this case, enclose the file name inside brackets < >. (The file name can have up to 32 characters including the extension.)
L	Number of subprogram repetitions. (Max. 4 digits) This can be omitted. (In this case, the subprogram will be called once.) When "L0" is designated, the subprogram call will not be executed.

Note

(1) Sequence No. call (M198 H****) cannot be commanded.

Return to main program from subprogram

```
M99 ;
```

**Detailed description**

- (1) The device that can be used for M198 subprogram call differs depending on the NC models.
The SD card in the front side is available for M800S/M80, and the SD card in the control unit is available for M800W. (The M198 command is not available for the C80 series.)
- (2) The subprogram can be called with the M198 command once in the subprogram nest. The subprogram can be called only from the memory or MDI program.
- (3) The section from the head of the program to the first LF (line feed code, 0x0A hexadecimal) is invalid, and is not run or displayed. Note that if the head starts with a O No., the program will be valid from the head.
- (4) A program registered in an SD card can be executed from only one part system. A program error will occur if an attempt is made to execute the programs in the SD card simultaneously by two or more part systems. If all the part system is reset when the error occurred, programs will be displayed as only "%" except for the first part system.
- (5) Refer to "14.1.1 Subprogram Call; M98, M99" for <File name> and calling the subprogram with O No.

14.1.3 Figure Rotation; M98 I_J_K_

**Function and purpose**

If the same pattern is used repeatedly on a concentric circle, one of the rotating machining patterns can be registered as a subprogram. When the subprogram is called from the main program, if the rotation center is designated, a path similar to the rotary phase can be easily created on the concentric circle. This simplifies creation of the program.

**Command format****Subprogram call command**

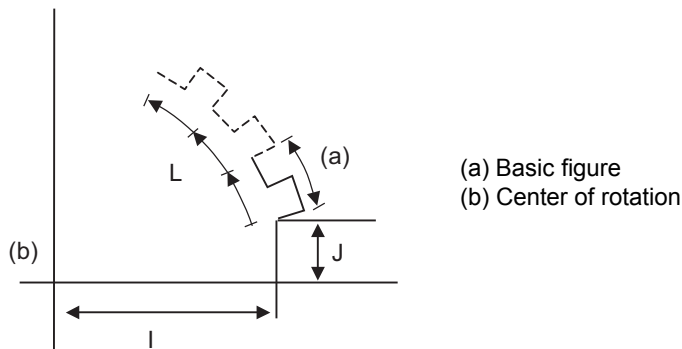
```
M98 I__ J__ K__ P__ H__ L__ ,D__ ;
```

```
M98 I__ J__ K__ <file name> H__ L__ ,D__ ;
```

I, J, K	Rotation center coordinates
P	Program number in subprogram to be called (own program if omitted) Note that P can be omitted only for memory mode, MDI operation, high-speed program server operation, SD card operation, hard disk operation, or USB operation. (Numeric value containing up to eight digits) Use a parameter to specify a 4- or 8-digit subprogram No. starting with O.
<File name>	A file name can be specified instead of a program No. In this case, enclose the file name with brackets < >. (The file name can have up to 32 characters including the extension.) (Example) M98 <BUHIN-12.RAF>;
H	Program sequence number in subprogram to be called (head block if omitted)
L	Number of subprogram repetitions (If omitted, it is assumed to be "L1", and processing is not carried out when "L0" is set.) (1 to 9999 times depending on the 4-digit value)
,D	Subprogram device No. (0 to 4). The subprogram in the memory can be used when ,D is omitted. The device No. is set with the machining parameters.



Detailed description



- (1) The first subprogram called out with subprogram call is executed at 0° rotation angle. The path is created as commanded.
- (2) If the number of repetitions is set to twice or more, the rotation angle is obtained from the called subprogram's start point, end point and rotation center coordinate. The path of the first subprogram is used as the basic figure and is rotated and arranged for the designated number of call repetitions, using the rotation center coordinates as a reference.
- (3) All blocks in the subprogram are rotated.
- (4) If the subprogram start point and end point are not on the same circle having the commanded figure rotation center coordinates as the center, the axis will interpolate using the subprogram's end point as the start point, and the end point in the first movement command block in the rotated subprogram as the end point.
- (5) Both absolute values and incremental values can be used in the figure rotation subprogram. Even if commanded with an absolute value command, the rotation will be the same as when commanded with an incremental value.
- (6) I, J and K are commanded with the incremental amount from the start point.
- (7) A subprogram of which figure is rotating cannot be branched to the other subprogram.
- (8) The figure is rotated on the workpiece coordinate system, and can be shifted with the G92, G52, G54 to G59 (workpiece coordinate system shift) command.
- (9) Functions (reference position return, uni-direction positioning, etc.) on the machine coordinate system for the rotary plane axis cannot be used while the figure is rotated. However, the machine coordinate system functions can be used for axes other than the rotation plane.
- (10) Refer to "14.1.1 Subprogram Call; M98, M99" for <File name> and calling the subprogram with O No.



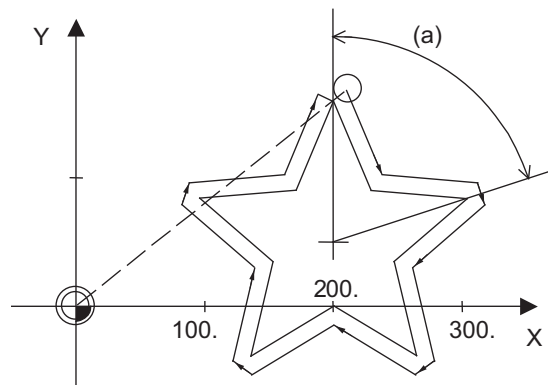
Program example

Main program (O1000)

```
N01 G90 G54 G00 X0 Y0 ;
N02 G01 G41 X200. Y150. D01 F500 ;
N03 G01 Z-50. F300 ;
N04 M98 P2200 L5 J-100. ;
N05 G90 G01 Z50. F500 ;
N06 G40 ;
N07 G00 X0 Y0 ;
```

Subprogram (O2200)

```
N01 G91 G01 X29.389 Y-59.549 ;
N02 X65.717 Y-9.549 ;
N03 M99 ;
```



(a) Basic figure



Precautions

- (1) A program error will occur if figure rotation is commanded during figure rotation.
- (2) Figure rotation and program coordinate rotation cannot be commanded simultaneously. The program error will occur.

14.2 Variable Commands



Function and purpose

Programming can be endowed with flexibility and general-purpose capabilities by designating variables, instead of giving direct numerical values to particular addresses in a program, and by assigning the variable values depending on the conditions that exist when executing the program.

All common variables are retained even when the power is turned OFF.

When the power is turned OFF or reset, the common variables can be set to <null> by setting the parameter ("#1128 RstVCI", "#1129 PwrVCI").



Command format

$\Delta\Delta\Delta$ = $\circ\circ\circ\circ\circ\circ$;

*** = [formula] ;



Detailed description

Variable expressions

		Example
#m	m = value consisting of 0 to 9	#100
# [f]	f = one of the followings in the formula	#[-#120]
	Numerical value m	123
	Variable	#543
	Formula Operator Formula	#110+#119
	- (minus) formula	-#120
	[Formula]	[#119]
	Function [formula]	SIN[#110]

Note

- (1) The 4 standard operators are +, -, * and /.
- (2) Functions cannot be used unless the user macro specifications are available.
- (3) Error (P241) will occur when a variable No. is negative.
- (4) Examples of incorrect variable expressions are given below.

Incorrect	Correct
#6/2	# [6/2] (#6/2 is regarded as [#6] /2.)
#--5	#[-[-5]]
#-[#1]	#[-#1]

Types of Variables

The following table gives the types of variables.

The common variables are divided into the following two types.

Common variables 1: Used in common through all part systems

Common variables 2: Used in common in the programs of the part system

Type		No.		Function
Common variable		Common variables 1	Common variables 2	♦Can be used in common throughout main, sub and macro programs. ♦When using common variables in the multi-part system, the number of common variables shared between the part systems can be specified depending on the MTB specifications (parameter "#1052 MemVal"). (*2) ♦(Only for C80 series) Can be read from/written to each common variable by using ZR80000 or later. Refer to "PLC Interface Manual" for the correspondence between the ZR device No. and the common variable No.
1 part system	200 sets	500 to 599	100 to 199	
	600 sets	500 to 999 100100 to 800199 (*4)	100 to 199	
	700 sets	400 to 999 (*1) 100100 to 800199 (*4)	100 to 199	
	8000 sets	400 to 999 (*1) 100100 to 800199 (*4) 900000 to 907399 (*3)	100 to 199	
Multi-part systems (n = number of part systems)	600 + 100 * n sets	400 to 999 (*1) 100100 to 800199 (*4)	100 to 199 *n	
	7900 + 100 * n sets	400 to 999 (*1) 100100 to 800199 (*4) 900000 to 907399 (*3)	100 to 199 *n	
Local variables		1 to 33		Can be used as local variables in macro programs.
System variable		1000 to		Application is fixed by system.
Fixed cycle variables		1 to 32		Local variables in fixed cycle programs.

(*1) Common variable address #400s can only be used when there are 700 or more sets of common variables and the MTB specifications are valid (parameter "#1336 #400_Valtyp").

When common variable address #400s can be used, these can be displayed and set on the common variable screen.

It also becomes possible to input/output data of common variable address #400s.

(*2) When the parameter "#1052 MemVal" is set to "1" in multi-part system (MTB specifications), some or all of common variables "#100 to #199" and "#500 to #999" can be shared and used between part systems. The number of variables sharable in part systems depends on the MTB specifications (parameters "#1303 V1comN" and "#1304 V0comN").

(Example) When "#1304 V0comN" is set to "5":

#500 to #504: Common for the part systems

#505 to #999: Each part system

Depending on the MTB specifications, the common variables #100 to #199 are used for each part system, and variables #500 to #999 are common for the part systems (parameter "#1052 MemVal"). Address #400s, that can be used as common variable with 700 or more sets of variable, is common for the part systems regardless of the setting of parameter "#1052 MemVal".

(*3) When "#1052 MemVal" is set to "1", #900000 to #907399 available for 8,000 sets of variable are not available.

(*4) When the parameter "#1316 CrossCom" is set to "1", the common variables #100100 to #800199 can be shared between the part systems. (This depends on the MTB specifications.) The part system common variable which can be used is shown in the table below.

Variable sets		Common variables 1 (When "#1316 CrossCom" = "1")
Variable sets specification	600 sets (500 + 100 sets)	#100100 to #100199 (Equivalent to # 100 to #199 in 1st part system) #200100 to #200199 (Equivalent to # 100 to #199 in 2nd part system)
	700 sets (600 + 100 sets)	#300100 to #300199 (Equivalent to # 100 to #199 in 3rd part system) #400100 to #400199 (Equivalent to # 100 to #199 in 4th part system)
	8000 sets (7900 + 100 sets)	#500100 to #500199 (Equivalent to # 100 to #199 in 5th part system) #600100 to #600199 (Equivalent to # 100 to #199 in 6th part system) #700100 to #700199 (Equivalent to # 100 to #199 in 7th part system) #800100 to #800199 (Equivalent to # 100 to #199 in 8th part system)

(Example)

<1-part system>

- #100100=200 ; Equivalent to #100 = 200 ;
- #200105=#100 ; "200" is set to #200105.
- #300110=#100100 ; "200" is set to #300110.
- #800199=#500120 ; The variable value of "#500120" is set to #800199.

<Multi-part system>

Common variables for each part system #100 to #199" in other part system can be used.

- \$1
- #200100=-100 ; "-100" is set to #100 of 2nd part system.
- #101=#200102 ; "#101" is set to #102 of 2nd part system.
- #300105=#200103 ; "#103" of 2nd part system is set to #105 of 3rd part system.
- #110=#500107 ; The variable value of "#500107" is set to #110.

- ♦The PLC data reading function cannot be used, which uses system variables #100100 to #100110, and variables #100100 to #100110 are used as common variables.
- ♦The setting of number of common variables shared between the part systems (The parameter #1052 MemVal" is set to "1") becomes invalid, thus the movement is the same as "0" is set.
- ♦When the parameters "#1128 RstVCI", "#1129 PwrVCI" are set to "1", the operation is as follows.
 - "#1128 RstVCI"
 - The common variables shared between the part systems equivalent to #100 to #199 of the reset part system are cleared.
 - (Example) If the 1st part system is reset, #100100 to #100199 are cleared.
 - If the 2nd part system is reset, #200100 to #200199 are cleared.
 - "#1129 PwrVCI"
 - The common variables shared between the part systems equivalent to #100 to #199 in the valid part system are cleared.
 - (Example) In 1st part system, #100100 to #100199 are cleared.
 - In 2nd part system, #100100 to #100199 and #200100 to #200199 are cleared.
- ♦Common variables shared between the part systems #100100 to #800199 can be displayed and set on the common variable screen.
- ♦If common variables #100100 to #800199 are used when the number of sets of common variables is less than 600 sets or the parameter "#1316 CrossCom" is "0", a program error (P241) will occur.

Note

- (1) When inputting the common variable data, if the following illegal variable No. data exist in the input file, the illegal variable No. data is ignored and only the correct common variable data will be input.
- ♦ Variable data that is not common variables such as local variables (#1 to #33) or system variables (#1000 and after)
 - ♦ Variable data of which the number of common variable sets does not match the pre-specified value
- (Example)
- If variables of # numbers undefined in the specifications exist in the input file when there are 700 sets of common variables (#100 to #199, #500 to #999, and #100100 to #800199), they are ignored, and only the variables defined in the specifications are input.

Variable quotations

Variables can be used for all addresses except O, N and / (slash).

- (1) When the variable value is used directly:

X#1 Value of #1 is used as the X value.

- (2) When the complement of the variable value is used:

X-#2 Value with the #2 sign changed is used as the X value.

- (3) When defining variables:

#3 = #5 Variable #3 uses the equivalent value of variable #5.

#1 = 1000 Variable #1 uses the equivalent value 1000. ("1000" is assumed to be "1000".)

- (4) When defining the variable arithmetic formula:

#1 = #3 + #2 - 100 Value of the operation result of "#3 + #2 - 100." is used as the #1 value.

X[#1 + #3 + 1000] Value of the operation result of "#1 + #3 + 1000" is used as the X value.

Note

- (1) A variable cannot be defined in the same block as an address. It must be defined in a separate block.

Incorrect		Correct
X#1 = #3 + 100 ;	→	#1 = #3 + 100 ; X#1 ;

- (2) Up to five sets of square parentheses [] may be used.

#543 = -[[[[[#120]/2+15.]*3-#100]/#520+#125+#128]*#130+#132]

- (3) There are no restrictions on the number of characters and number of variables for variable definition.

- (4) The variable values should be within the range of 0 to ±99999999.

If this range is exceeded, the arithmetic operations may not be conducted properly.

- (5) The variable definitions become valid when definitions are made.

#1 = 100 ; #1 = 100

#1 = 200 #2 = #1 + 200 ; #1 = 200, #2 = 400

#3 = #1 + 300 ; #3 = 500

- (6) Variable quotations are always regarded as having a decimal point at the end.

When #100 is set to "10"

"X#100;" is set to "X10."

Protection of common variable

- (1) If the common variable protection function is valid, the common variables in the range specified in the parameters (#12111 to #12114) cannot be changed from machining program or screen operation, or user operation such as file input. This function depends on the MTB specifications (parameter "#1391 User level protect").
- (2) If an attempt is made to change the value or name of the protected variable on the machining program, the program error (P243) will occur, causing the operation to be stop. Such a variable value or name can be changed using the machine tool builder macro program, but cannot be done by the user.
Multiple variable names can be changed in one block with the SETVNn command. However, if at least one of them is protected, the program error (P243) will occur.
- (3) If "#1128 RstVCl" is set to "1", the variables (#100 to #199) are cleared after reset even if common variables (#100 to #199) are protected.
- (4) If "#1129 PwrVCl" is set to "1", the variables (#100 to #199) are cleared at the power-ON even if common variables (#100 to #199) are protected.
- (5) For common variables used common to the part systems, the variable values and variable names can be changed by the displayed part system.

14.3 User Macro

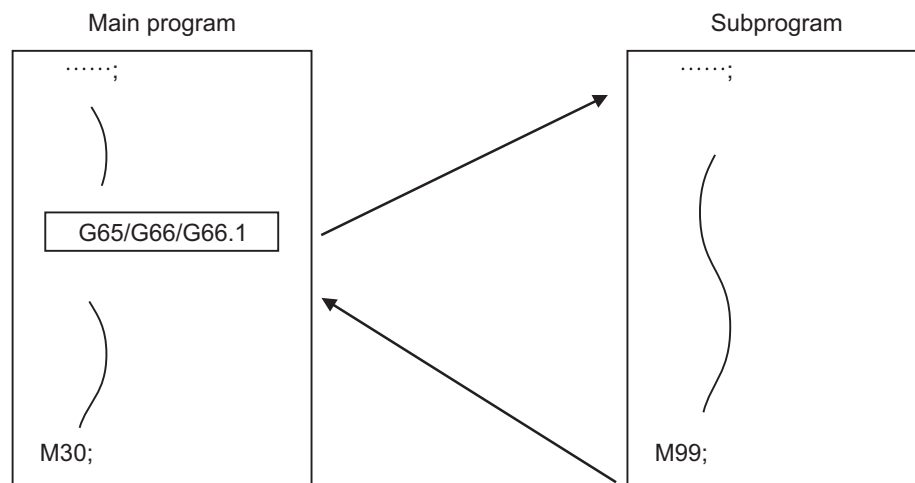


Function and purpose

A group of control and arithmetic instructions can be registered and used as a macro program to make it one integrated function.

Macro programs use variables, control and arithmetic instructions to create subprograms which function to provide special-purpose controls.

By combining the user macros with variable commands, it is possible to use the macro program call, arithmetic operations, data input/output with PLC, control, decision, branch and many other instructions for measurement and other such applications.



These special-purpose control functions (macro programs) are called by the macro call instructions from the main program when needed.

G code	Function
G65	User macro Simple call
G66	User macro Modal call A (Movement command call)
G66.1	User macro Modal call B (Per-block call)
G67	User macro Modal call (G66, G66.1) cancel



Detailed description

- (1) When the G66 or G66.1 command is entered, the specified user macro program will be called every time a block is executed or after a movement command in blocks with a movement command is executed, until the G67 (cancel) command is entered.
- (2) The G66 (G66.1) and G67 commands must be paired in a same program.

14.4 Macro Call Instructions



Function and purpose

Macro call commands include the simple calls which call only the instructed block and the modal calls (types A and B) which call a block in the call modal.

When the macro argument L/P valid function is enabled, the addresses L (number of subprogram repetitions) and P (calling program No.) used as commands in user macro can be used as arguments.

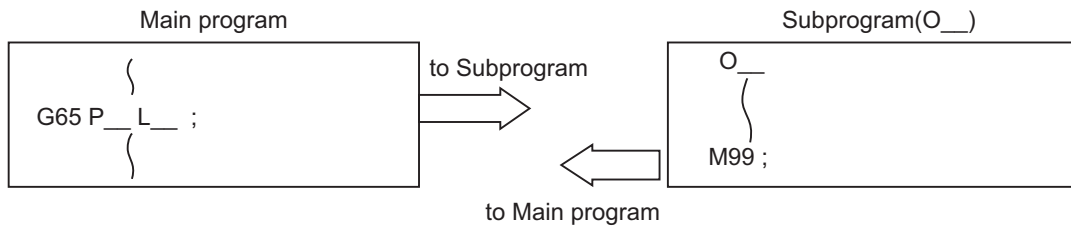
The validity of this parameter depends on the MTB specifications. (Parameter "#1241 set13"/bit5 (Macro argument L/P valid))

When a program in an external device such as a USB memory device is executed, a machining program stored in USB memory cannot be called with a macro call such as G65, G66, or G66.1. Using such a macro calls a macro program in memory.

14.4.1 Simple Macro Calls; G65



Function and purpose



M99 is used to terminate the user macro subprogram.



Command format

Simple macro calls

G65 P__ L__ argument ;

Simple macro calls

G65 <File name> L__ argument ;

P	Program No. (*1) Use a parameter to specify a 4- or 8-digit subprogram No. starting with O.
<File name>	File name A file name can be specified instead of a program No. In this case, enclose the file name inside brackets < >. (The file name can have up to 32 characters including the extension.)
L	Number of repetitions (*1) If omitted, this value is set to "1". (0 to 9999)
Argument	Specify variable data

(*1) Can also be used as an argument at the same time as the macro argument L/P valid function is enabled.



Detailed description

- (1) When the argument must be transferred as a local variable to a user macro subprogram, the actual value should be designated after the address.
In this case, regardless of the address, a sign and decimal point can be used in the argument. There are 2 ways in which arguments are designated.

Argument designation I

Format : A_ B_ C_X_ Y_ Z_

- (a) Arguments can be designated using any address except G, L, N, O and P.
 (b) I, J and K must be designated in alphabetical order.
 I_ J_ K...Correct
 J_ I_ K...Incorrect
 (c) Except for I, J and K, there is no need for designation in alphabetical order.
 (d) Addresses which do not need to be designated can be omitted.
 (e) The following table shows the correspondence between the addresses which can be designated by argument designation I and the variable numbers in the user macro main body.

Address and variable No. correspondence		Addresses available for call instructions	
Argument designation I address	Variable in macro	G65, G66	G66.1
A	#1	○	○
B	#2	○	○
C	#3	○	○
D	#7	○	○
E	#8	○	○
F	#9	○	○
G	#10	×	× *
H	#11	○	○
I	#4	○	○
J	#5	○	○
K	#6	○	○
L	#12	×	× *
M	#13	○	○
N	#14	×	× *
O	#15	×	×
P	#16	×	× *
Q	#17	○	○
R	#18	○	○
S	#19	○	○
T	#20	○	○
U	#21	○	○
V	#22	○	○
W	#23	○	○
X	#24	○	○
Y	#25	○	○
Z	#26	○	○

○: Available

×: Unavailable

*: Can be used while G66.1 command is modal

Argument designation II

Format: A_B_C_I_J_K_I_J_K_...

- (a) In addition to address A, B and C, up to 10 groups of arguments with I, J, K serving as 1 group can be designated.
- (b) When the same address is duplicated, designate the addresses in the specified order.
- (c) Addresses which do not need to be designated can be omitted.
- (d) The following table shows the correspondence between the addresses which can be designated by argument designation II and the variable numbers in the user macro main body.

Argument designation II address	Variable in macro
A	#1
B	#2
C	#3
I1	#4
J1	#5
K1	#6
I2	#7
J2	#8
K2	#9
I3	#10
J3	#11
K3	#12
I4	#13
J4	#14
K4	#15
I5	#16

Argument designation II address	Variable in macro
J5	#17
K5	#18
I6	#19
J6	#20
K6	#21
I7	#22
J7	#23
K7	#24
I8	#25
J8	#26
K8	#27
I9	#28
J9	#29
K9	#30
I10	#31
J10	#32
K10	#33

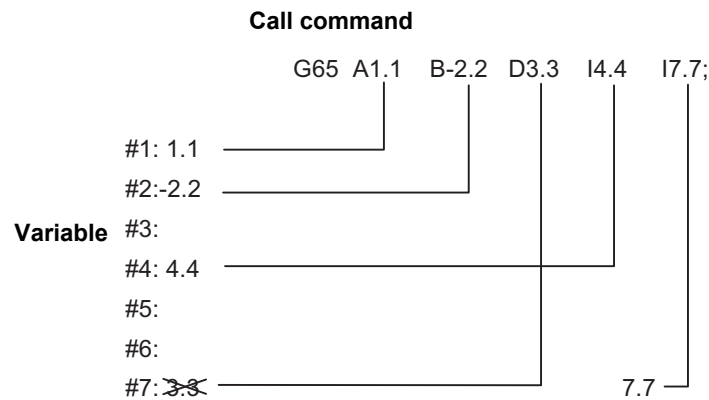
Note

- (1) The numbers 1 to 10 accompanying I, J and K indicate the sequence of the commanded sets, and are not required in the actual command.

Using arguments designations I and II together

- (1) If addresses corresponding to the same variable are commanded when both types I and II are used to designate arguments, the latter address will become valid.

(Example 1)



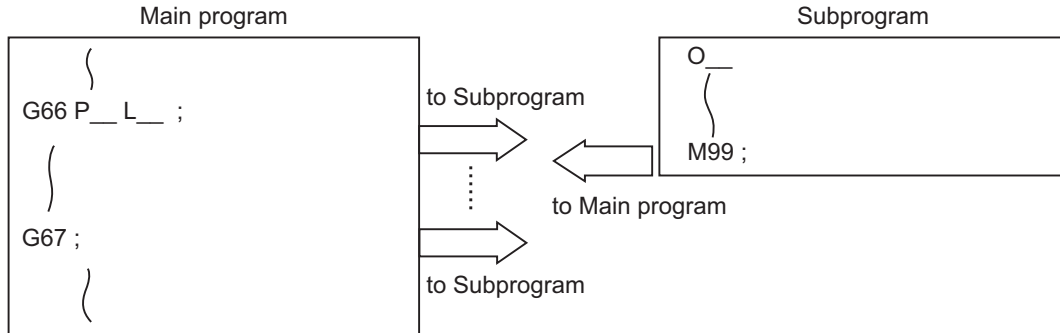
In the above example, I7.7 argument is valid when both arguments D3.3 and I7.7 are commanded for the #7 variable.

- (2) If calling a subprogram numbered with O is enabled, a sub program number starting with O and specified by P command value is called with a priority.
However, when P command value is less than the digit number set with parameter "#8129 Subpro No. select", increase the digit number of command value by adding leading zeros.
(Example) When parameter "#8129 Subpro No. select"="1", call the subprogram "O0012" with "G65 P12" command.
- (3) In the following cases, a subprogram of P command value without O No. is called even with a setting to call a subprogram with O No.
- ♦The digit number of P command value is over the digit number of the program number set with parameter "#8129 Subpro No. select".
 - ♦A subprogram starting with commanded O No. does not exist.

14.4.2 Modal Call A (Movement Command Call) ; G66



Function and purpose



When the block with a movement command is commanded between G66 and G67, the movement command is first executed and then the designated user macro subprogram is executed. A number of user macro subprograms are designated with "L".

The argument is the same as for a simple call.



Command format

Modal call A

G66 P__ L__ argument ;

G66 <File name> L__ argument ;

P	Program No. (*1) Use a parameter to specify a 4- or 8-digit subprogram No. starting with O.
<File name>	File name A file name can be specified instead of a program No. In this case, enclose the file name inside brackets < >. (The file name can have up to 32 characters including the extension.)
L	Number of repetitions (*1)
Argument	Specify variable data

(*1) Can also be used as an argument at the same time as the macro argument L/P valid function is enabled.

Modal call end

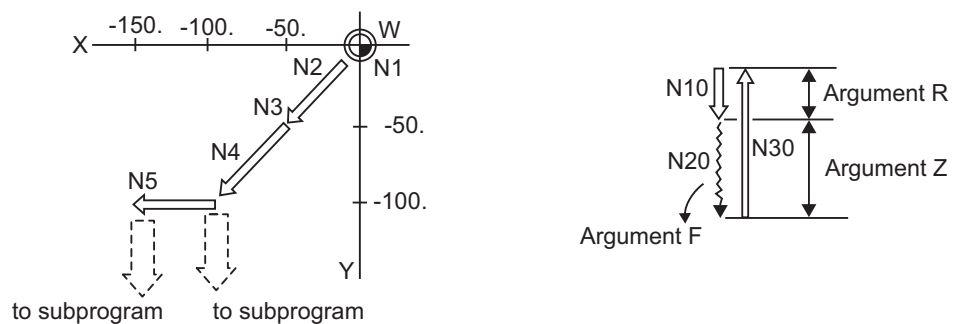
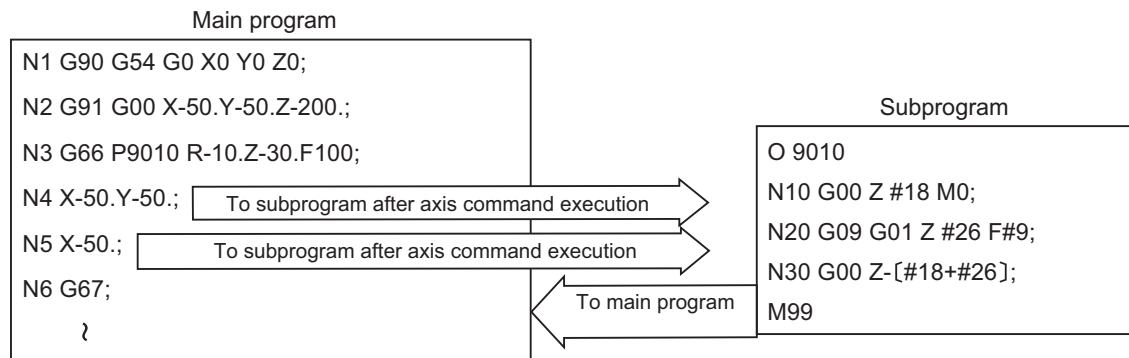
G67;



Detailed description

- (1) When the G66 command is entered, the specified user macro program will be called after the movement command in a block with the movement commands has been executed, until the G67 (cancel) command is entered.
- (2) The G66 and G67 commands must be paired in a same program.
A program error will occur when G67 is issued without G66.

(Example) Drill cycle



<Note>

- ♦After the axis command is executed in the main program, the subprogram is executed.
 - ♦The subprogram is not executed in the blocks following G67.
- (3) If calling a subprogram numbered with O is enabled, a sub program number starting with O and specified by P command value is called with a priority.
However, when P command value is less than the digit number set with parameter "#8129 Subpro No. select", increase the digit number of command value by adding leading zeros.
(Example)
When parameter "#8129 Subpro No. select"="1", call the subprogram "O0012" with "G66 P12" command.
 - (4) In the following cases, a subprogram of P command value without O No. is called even with a setting to call a subprogram with O No.
 - ♦The digit number of P command value is over the digit number of the program number set with parameter "#8129 Subpro No. select".
 - ♦A subprogram starting with commanded O No. does not exist.

14.4.3 Modal Call B (for Each Block); G66.1

**Function and purpose**

The specified user macro subprogram is called unconditionally for each command block that is assigned between G66.1 and G67 and the subprogram will be repeated for the number of times specified in L. The argument is the same as for a simple call.

**Command format****Modal call B**

G66.1 P__ L__ argument ;

G66.1 <File name> L__ argument ;

P	Program No. (*1) Use a parameter to specify a 4- or 8-digit subprogram No. starting with O.
<File name>	File name A file name can be specified instead of a program No. In this case, enclose the file name inside brackets < >. (The file name can have up to 32 characters including the extension.)
L	Number of repetitions (*1)
Argument	Specify variable data

(*1) Can also be used as an argument at the same time as the macro argument L/P valid function is enabled.

Modal call end

G67;



Detailed description

- (1) In the G66.1 mode, everything except the O, N and G codes in the various command blocks which are read are handled as the argument without being executed. Any G code designated last or any N code commanded after anything except O and N will function as the argument.
- (2) All significant blocks in the G66.1 mode are handled as when G65 P_ is assigned at the head of a block.
(Example 1)
In "G66.1 P1000;" mode, "N100 G01 G90 X100. Y200. F400 R1000 ;" is the same as "N100 G65 P1000 G01 G90 X100. Y200. F400 R1000 ;".
<Note>
 - ♦The call is performed even in the G66.1 command block in the G66.1 mode and the correspondence between the argument address and the variable number is the same as for G65 (simple call).
- (3) The range of the G and N command values that can be used anew as variables in the G66.1 mode is subject to the restrictions as normal NC command values.
- (4) Program number O, sequence numbers N and modal G codes are updated as modal information.
- (5) If calling a subprogram numbered with O is enabled, a sub program number starting with O and specified by P command value is called with a priority.
However, when P command value is less than the digit number set with parameter "#8129 Subpro No. select", increase the digit number of command value by adding leading zeros.
(Example)
When parameter "#8129 Subpro No. select"="1", call the subprogram "O0012" with "G66.1 P12" command.
- (6) In the following cases, a subprogram of P command value without O No. is called even with a setting to call a subprogram with O No.
 - ♦The digit number of P command value is over the digit number of the program number set with parameter "#8129 Subpro No. select".
 - ♦A subprogram starting with commanded O No. does not exist.

14.4.4 G Code Macro Call



Function and purpose

User macro subprogram with prescribed program numbers can be called merely by issuing the G code command.



Command format

G code macro call

G** P__ L__ argument ;

G**	G code for macro call
P	(*1)
L	(*1)

(*1) Functions as an argument when the macro argument L/P valid function is enabled.
It cannot be used when the macro argument L/P valid function is disabled.



Detailed description

- (1) The above instruction functions in the same way as the instructions below, however, the correspondence between G codes and instructions can be set by parameters.
 - a: M98 P****;
 - b: G65 P**** <Argument>;
 - c: G66 P**** <Argument>;
 - d: G66.1 P**** <Argument>;

When the parameters corresponding to "c" and "d" above are set, issue the cancel command (G67) either in the user macro or after the call code has been commanded so as to cancel the modal call.

- (2) The correspondence between the "****" which conducts the macro call and the macro program number P**** to be called is set by parameters.
- (3) Up to 10 G codes from G100 to G999 can be used with this instruction. (G codes from G01 to G99 used in the system can also be used with parameter "#1081 Gmac_P". This parameter setting depends on the MTB specifications.)

<Note>

♦G101 to G110 and G200 to G202 are user macro I codes, but if the parameters are set as the G code call codes, the G code call will be the priority, and these codes cannot be used for user macro I.

- (4) These commands cannot be issued in a program which has been called by a G code macro. If issued in such a program, they will be handled as ordinary G commands.
- (5) When ",D" or "<(Character string)>" is commanded in a block that is calling a G code macro, a miscellaneous command macro, or an ASCII macro while the macro argument L/P valid function is enabled, a program error (P33) will occur.
This parameter setting depends on the MTB specifications. (Parameter "#1241 set13/bit5")

14.4.5 Miscellaneous Command Macro Call (for M, S, T, B Code Macro Call)

**Function and purpose**

The user macro subprogram of the specified program number can be called merely by issuing an M (or S, T, B) code. (Registered M code and all S, T and B codes.)

**Command format****Miscellaneous command macro call**

M** P__ L__ ; (or S** ;, T** ;, B** ;)

M**	M code for macro call (or S, T, B code)
P	(*1)
L	(*1)

(*1) Functions as an argument when the macro argument L/P valid function is enabled.
It cannot be used when the macro argument L/P valid function is disabled.

**Detailed description**

(1) The above instruction functions in the same way as the instructions below, however, the correspondence between M codes and instructions can be set by parameters. (Same for S, T and B codes)

a: M98 P**** ;	M98, M** are not output.
b: G65 P**** M** ;	
c: G66 P**** M** ;	
d: G66.1 P**** M** ;	

When the parameters corresponding to "c" and "d" above are set, issue the cancel command (G67) either in the user macro or after the call code has been commanded so as to cancel the modal call.

(2) The correspondence between the "M**" which conducts the macro call and the macro program number P**** to be called is set by parameters. Up to 10 M codes from M00 to M9999 can be registered.
Note that the codes to be registered should exclude those basically required for the machine and the following M codes.

M0, M1, M2, M30, M96, M97, M98, M99, M198, and M codes for G83 specified in the parameter "#8083"

(3) As with M98, it is displayed on the screen display of the setting and display unit but the M codes and MF are not output.

(4) Even if the registered miscellaneous commands above are issued in a user macro subprogram which are called by an M code, it will not be regarded as a macro call and will be handled as a normal miscellaneous command. (Same for S, T and B codes)

(5) All S, T and B codes call the subprograms in the prescribed program numbers of the corresponding S, T and B functions.

(6) Up to 10 M codes can be set.

<Note>

- When "1 to 3" is set to "#7002 M[01] type", the macro call will be equivalent call to G65/G66/G66.1. In this case, the alphabet before the M,S,T,B code macro is not handled as an argument. For example, commanding the M code and T code in the same block changes the operation depending on the order of the address.

(Example) To register M06 in M code macro

M06 T02 The value of T is treated as variable #20 in macro. The value is entered in the T code at the same time.

T02 M06 The value is not entered in the variable #20 in macro. The value is entered in the T code.

(7) The address L and address P can be used as an argument when the macro argument L/P valid function is enabled.

This parameter setting depends on the MTB specifications (parameter "#1241 set13"/bit5).

The argument address L is designated with variable #12 in the macro, and the argument address P with variable #16 in the macro.

(8) When the macro argument L/P function is enabled and the value is designated with macro type "M98", if address L and address P are commanded, a program error (P33) will occur.

(9) Even when the macro argument L/P valid function is enabled, argument codes G, L, N, O, and P are not displayed on the local variable screen.

(10) When ",D" or "<(Character string)>" is commanded in a block that is calling a G code macro, a miscellaneous command macro, or an ASCII code macro while the macro argument L/P valid function is enabled, a program error (P33) will occur.

This parameter setting depends on the MTB specifications. (Parameter "#1241 set13"/bit5)

(11) Even when the miscellaneous function lock signal (AFL) is enabled, the macro call instruction is executed.

14.4.6 Detailed Description for Macro Call Instruction



Detailed description

Differences between M98 and G65 commands

- (1) The argument can be designated for G65 but not for M98.
- (2) The sequence number can be designated for M98, but not for G65, G66 and G66.1.
- (3) M98 executes subprograms after all the commands except M, P, H and L in the M98 block are executed, but G65 branches directly to the subprogram without any further operation.
- (4) When any address except O, N, P, H or L is included in the M98 block, the single block stop will be conducted, but not for the G65.
- (5) The level of the M98 local variables is fixed but it varies in accordance with the nesting depth for G65. ("#1" before and after M98, for instance, has the same significance, but they have different significance in G65.)
- (6) The M98 nesting depth extends up to 10 levels in combination with G65, G66 and G66.1. The G65 nesting depth extends up to only 4 levels in combination with G66 and G66.1.

Macro call command nesting depth

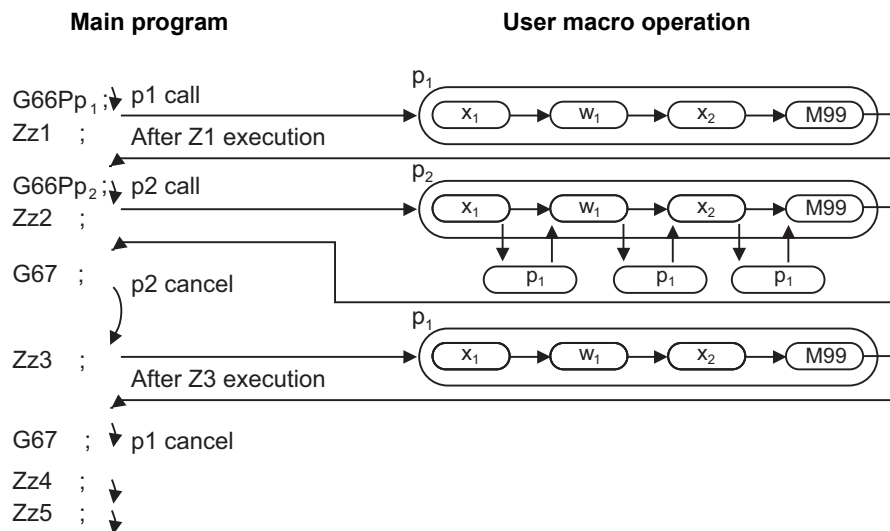
Up to 4 nesting levels are available for macro subprogram calls by simple call or modal call.

The argument for a macro call instruction is valid only within the called macro level. Since the nesting depth for macro calls extends up to 4 levels, the argument can be used as a local variable for the programs of each macro call of each level.

Note

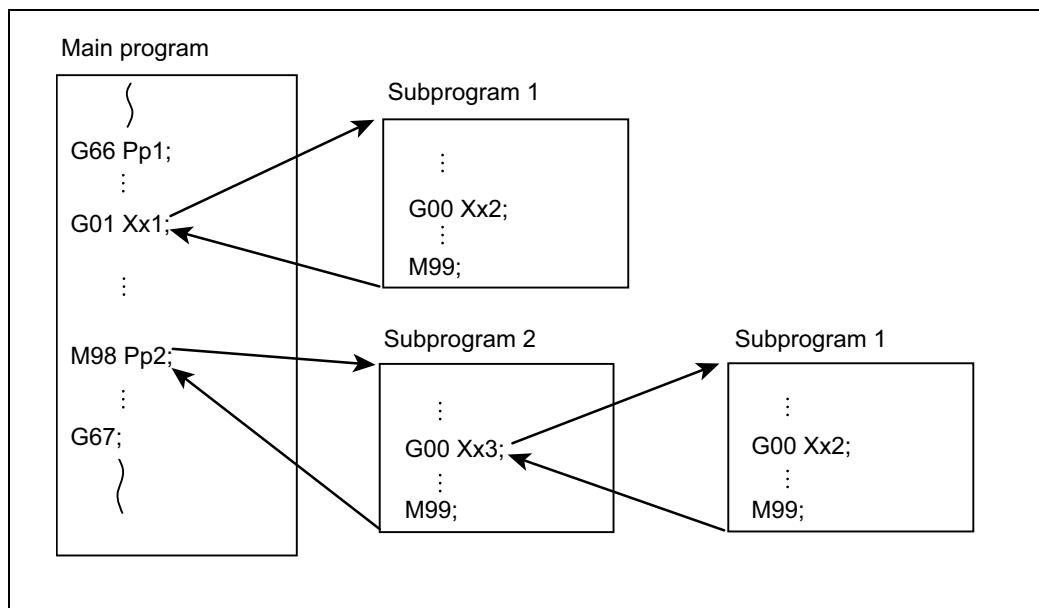
- (1) When a G65, G66, G66.1 G code macro call or miscellaneous command macro call is conducted, this is regarded as a nesting level and the level of the local variables is also incremented by one.
- (2) With modal call A, the designated user macro subprogram is called every time a movement command is executed. However, when the G66 command is duplicated, the next user macro subprogram is called to movement commands in the macro every time an axis is moved. User macro subprograms are called from the one commanded last.

(Example 1)



- (3) When M98 command is executed in G66 (G66.1) modal, the program designated by G66 (G66.1) will be executed after completing the movement command in the subprogram called by M98 (in case of G66.1, after completing each block).

(Example 2)

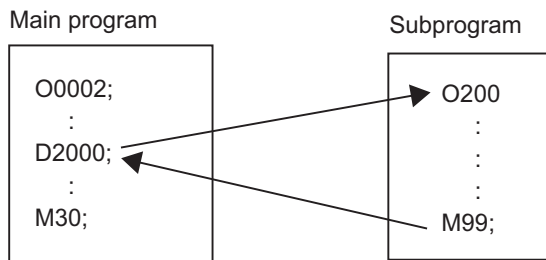


When the program numbers of p1 and p2 are same, the program numbers of subprograms 1 and 2 will be same.

14.4.7 ASCII Code Macro

**Function and purpose**

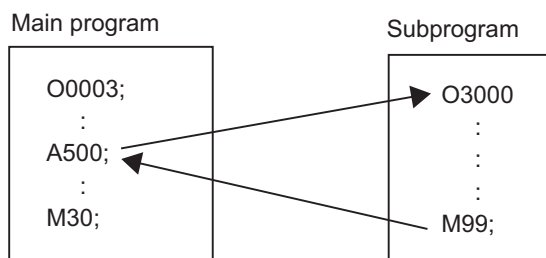
A macro program can be called out by setting the correspondence of a subprogram (macro program) preregistered with the parameters to codes, and then commanding the ASCII code in the machining program. This function can be used in addition to the G, M, S, T and B miscellaneous command macro call function. These parameters depend on the MTB specifications.

(Execution example 1) M98 type

After outputting 2000 to common variable #146, the program No. 200 subprogram is called with the M98 subprogram call type.

Parameters

#7401 (ASCII call Valid/Invalid)	1 (Valid)
#7402 (ASCII code)	D
#7403 (Call type)	0 (M98 type)
#7404 (ASCII [01] Program No.)	200
#7405 (ASCII [01] Variable)	146

(Execution example 2) G65 type

After outputting 500 to local variable #1, the program No. 3000 subprogram is called out with the G65 macro call type.

Parameters

#7411 (ASCII call Valid/Invalid)	1 (Valid)
#7412 (ASCII code)	A
#7413 (Call type)	1 (G65 type)
#7414 (ASCII [01] Program No.)	3000
#7415 (ASCII [01] Variable)	100 (Not used)



Command format

□**** P__ L__ ; ... Designates the address and code

□	ASCII code for calling out a macro (one character)
****	Value or expression output to variable (Setting range: ±999999.9999)
P	(*1)
L	(*1)

(*1) Functions as an argument when the macro argument L/P valid function is enabled.
It cannot be used when the macro argument L/P valid function is disabled.



Detailed description

(1) The above command performs the same operations as the commands listed below. The correspondence of commands is set for each ASCII code with the parameters.

- 0: M98 P**** ;
- 1: G65 P**** <Argument>;
- 2: G66 P**** <Argument>;
- 3: G66.1 P**** <Argument>;

When parameters corresponding to items "2" and "3" above are set, the modal call will be canceled. Thus, command the cancel command (G67) after commanding the call code or during the user macro.

- (2) The ASCII code for calling the macro and the program No. P**** to be called are set with the parameters. Up to two ASCII codes can be registered.
- (3) The code section is output to the variables, but the output destination differs according to the call type and address.

(a) For M98 type

The code section is output to a common variable and the variable No. is set with a parameter. When corresponding to the first address (parameter #7401), the section is output to the common variable which is indicated by the first variable No. (parameter #7404). (These parameters depend on the MTB specifications.)

(b) For G65/G66/G66.1 type

The code section is output to a local variable. The variable No. differs according to the address, and corresponds to the following table.

Address	#	Address	#	Address	#
A	1	K	6	U	21
B	2	L	12	V	22
C	3	M	13	W	23
D	7	N	14	X	24
E	8	O	15	Y	25
F	9	P	16	Z	26
G	10	Q	17		
H	11	R	18		
I	4	S	19		
J	5	T	20		

Note

- (1) The following addresses can be used.
A, B, D, F, H, I, J, K, M, Q, R, S, T



Precautions

Calling a macro with an ASCII code from a macro-called program

A macro cannot be called with an ASCII code from a macro-called program with an ASCII code.

The other patterns are shown below.

If it is judged that a macro cannot be called, the command will be handled as a normal command.

		Called side			
		ASCII	GMSTB macro	G65/66/66.1	M98
Calling side	ASCII	×	×	○	○
	GMSTB macro	×	×	○	○
	G65/66/66.1	○	○	○	○
	M98	○	○	○	○

Nest level of macro call commands

Up to 4 nesting levels are available for macro subprogram calls using simple call (G65) and modal call (G66/G66.1).

The macro call command's argument is valid only in the called macro level.

Since the macro call nest level is four, the argument can be used in the program as a local variable for each macro call.

Nest level of subprogram call command

Counting the main program as 0, up to ten levels of subprograms can be called (M98) from a subprogram.

The following commands are used for subprogram nesting.

- (1) M98
- (2) G65 G66 G66.1
- (3) G code call Miscellaneous function call (M/S/T/B)
- (4) MDI interruption
- (5) Automatic tool length measurement
- (6) Multiple-step skip function

The following commands can be issued regardless of nesting.

- (7) Fixed cycles
- (8) Macro interruption

Order of command priority

If "M" is designated for the ASCII code address, it may overlap with the codes basically necessary for that machine. In this case, commands will be identified with the following priority using code values.

- (1) M98, M99 (subprogram call command)
 - M00 (program stop command)
 - M01 (optional stop command)
 - M02, M30, M198, M199 (end command)
 - M96, M97 (macro interruption command)
- (2) When corresponding to ASCII code macro command
- (3) Used as normal command

If "S", "T" and "B" are designated for the ASCII code address, commands will be identified with the following priority using code values.

- (a) When corresponding to miscellaneous code (S, T, B) call macro command
- (b) When corresponding to ASCII code macro command
- (c) When used as normal command.

If the other addresses do not correspond to the ASCII code macro command, they will be identified as normal commands. If the command to be used, overlaps with an ASCII code macro command, it must be commanded in the macro-called program with the ASCII code.

Note that there are cases where the command will be unconditionally handled as a normal command, as explained in below.

Conditions where the address set is handled as a normal command

- (1) When there is a data setting command (G10) in the same block.
- (2) When ASCII code macro call is executed after the G code macro call command in the same block (also applies for M, S, T, B and ASCII)
 - (Example) When address "D" (G65 type) is set in the ASCII code macro, and M50 is set in the macro call (G65 type).

M50 D200 ; Execute M code macro with argument (200 set in #7)

- (3) When inputting parameters
- (4) When there is a comma (,) before the address. (For example, ",D", ",R", etc.)
- (5) When commanded in fixed cycle
- (6) When commanded in macro subprogram called with G code macro call
 - (Also applies when macro is called with M, S, T, B or ASCII)
- (7) When ",D" or "<(Character string)>" is commanded in a block that is calling a G code macro, a miscellaneous command macro, or an ASCII macro while the macro argument L/P valid function is enabled, a program error (P33) will occur.
 - This parameter setting depends on the MTB specifications. (Parameter "#1241 set13/bit5")

14.5 Variables Used in User Macros



Function and purpose

Both the variable specifications and user macro specifications are required for the variables that are used with the user macros.

The compensation amounts of the local, common and system variables among the variables for this NC system except #33 are retained even when the unit's power is switched off. (Common variables can also be cleared by parameter "#1129 PwrVCI".)



Detailed description

Use of multiple variable

When the user macro specifications are applied, variable Nos. can be turned into variables (multiple uses of variables) or replaced by <formula>.

Only one of the four basic arithmetic rule (+, -, *, /) operations can be conducted with <formula>.

(Example 1) Multiple uses of variables

#1=10 #10=20 #20=30 ;	
#5=# [#1] ;	# [#1] = # [#10] from #1 = 10.
	# [#10] = #20 from #10 = 20.
	Therefore, #5 = #20 or #5 = 30.

#1=10 #10 =20 #20=30 #5=1000;	
#[#1]=#5;	# [#1] = # [#10] from #1 = 10.
	# [#10] = #20 from #10 = 20.
	Therefore, #20 = #5 or #20 = 1000.

(Example 2) Example of multiple designations of variables

#10=5;	<Formula>##10 = 100; is handled in the same manner as # [#10] = 100.
##10=100 ;	In which case, #5 = 100.

(Example 3) Replacing variable Nos. with <formula>

#10=5 ;	
#[#10 + 1] = 1000 ;	In which case, #6 = 1000.
#[#10 - 1] = -1000 ;	In which case, #4 = -1000.
#[#10 * 3] = 100 ;	In which case, #15 = 100.
#[#10/2] = -100 ;	In which case, #2 = -100.

Undefined variables

When applying the user macro specifications, variables which have not been used even once after the power was switched on or local variables which were not specified by the G65, G66 or G66.1 commands, can be used as <Blank>. Also, variables can forcibly be set to <Blank>.

Variable #0 is always used as the <Blank> and cannot be defined in the left-side member.

(1) Arithmetic expressions

#1 = #0;	#1 = <Blank>
#2 = #0 + 1;	#2 = 1
#3 = 1 + #0;	#3 = 1
#4 = #0 * 10;	#4 = 0
#5 = #0 + #0;	#5 = 0

Note that <Blank> in an arithmetic expression is handled in the same way as 0.

<Blank> + <Blank> = 0

<Blank> + <Constant> = Constant

<Constant> + <Blank> = Constant

(2) Variable quotations

When only the undefined variables are quoted, they are ignored including the address itself.

When #1 = <Blank>

G00 X#1 Z1000 ; Equivalent to G00 Z1000 ;

G00 X#1+10 Z1000 ; Equivalent to G00 X10 Z1000 ;

(3) Conditional expressions

<Blank> differs from "0", only for EQ and NE. (#0 is <Blank>.)

When #101 = <Blank>	When #101 = 0
#101EQ#0 <Blank> = <Blank> Established	#101EQ#0 0 = <Blank> Not established
#101NE0 <Blank> ≠ 0 Established	#101NE0 0 ≠ 0 Not established
#101GE#0 <Blank> ≥ <Blank> Established	#101GE#0 0 ≥ <Blank> Established
#101GT0 <Blank> > 0 Not established	#101GT0 0 > 0 Not established
#101LE#0 <Blank> ≤ <Blank> Established	#101LE#0 0 ≤ <Blank> Established
#101LT0 <Blank> < 0 Not established	#101LT0 0 < 0 Not established

Note

- (1) EQ and NE should be compared only for integers. For comparison of numeric values with decimals, GE, GT, LE, and LT should be used.

14.5.1 Common Variables

**Detailed description**

Common variables can be used commonly from any position. Number of the common variables sets depends on the specifications.

Refer to the explanation about Variable Commands for details.

Variable name setting and quotation

Any name (variable name) can be given to common variables #100 to #199 and #500 to #599. It must be composed of not more than 7 alphanumeric characters and it must begin with a letter. Do not use "#" in variable names. It causes an alarm when the program is executed.

SETVn [NAME1,NAME2,] ;	
n	Head No. of variable to be named
NAME1	#n name (variable name)
NAME2	#n + 1 name (variable name)

Variable names are separated by a comma (,).

- (1) Once variable names have been set, they will not be cleared even when the power is turned off.
- (2) Variables in programs can be quoted by their variable names. In this case, the variables should be enclosed in square parentheses [].
(Example 1) G01X [#POINT1] ;
- (3) The variable Nos., data and variable names are displayed on the screen of the setting and display unit.
(Example 2)
Program... SETVN500 [A234567, DIST, TOOL25] ;

#	Value	Name
500	-12345.6780	A234567
501	5670.0200	DIST
502	-156.5000	TOOL25
503		
504		
505		
506		
507		
508		
509		
510		
511		
512		
513		

Note

- (1) Do not use characters (SIN, COS, etc.) predetermined by the NC and used for operation commands at the head of a variable name.

14.5.2 Local Variables (#1 to #33)

**Detailed description**

Local variables can be defined as an <argument> when a macro subprogram is called, and also used locally within main programs and subprograms. They can be duplicated because there is no relationship between macros. (up to 4 levels)

G65 P__ L__ <argument> ;

P	Program No.
L	Number of repetitions

The <argument> is assumed to be Aa1 Bb1 Cc1..... Zz1.

The following table shows correspondences points between the addresses designated by <argument> and the local variable numbers used in the user macro main bodies.

[Argument designation I]

Call command		Argument address	Local variable No.
G65 G66	G66.1		
○	○	A	#1
○	○	B	#2
○	○	C	#3
○	○	D	#7
○	○	E	#8
○	○	F	#9
×	× *	G	#10
○	○	H	#11
○	○	I	#4
○	○	J	#5
○	○	K	#6
Δ	Δ *	L	#12
○	○	M	#13
×	× *	N	#14
×	×	O	#15
Δ	Δ *	P	#16

Call command		Argument address	Local variable No.
G65 G66	G66.1		
○	○	Q	#17
○	○	R	#18
○	○	S	#19
○	○	T	#20
○	○	U	#21
○	○	V	#22
○	○	W	#23
○	○	X	#24
○	○	Y	#25
○	○	Z	#26
		-	#27
		-	#28
		-	#29
		-	#30
		-	#31
		-	#32
		-	#33

"×" in the above table denotes argument addresses which cannot be used. However, provided that the G66.1 mode has been established, an argument address denoted by the asterisk can be added for use.

The hyphen (-) mark indicates that there is no corresponding address.

A "Δ" mark denotes an argument address which may be used depending on the MTB specifications. (Parameter "#1241 set13"/bit5)

[Argument designation II]

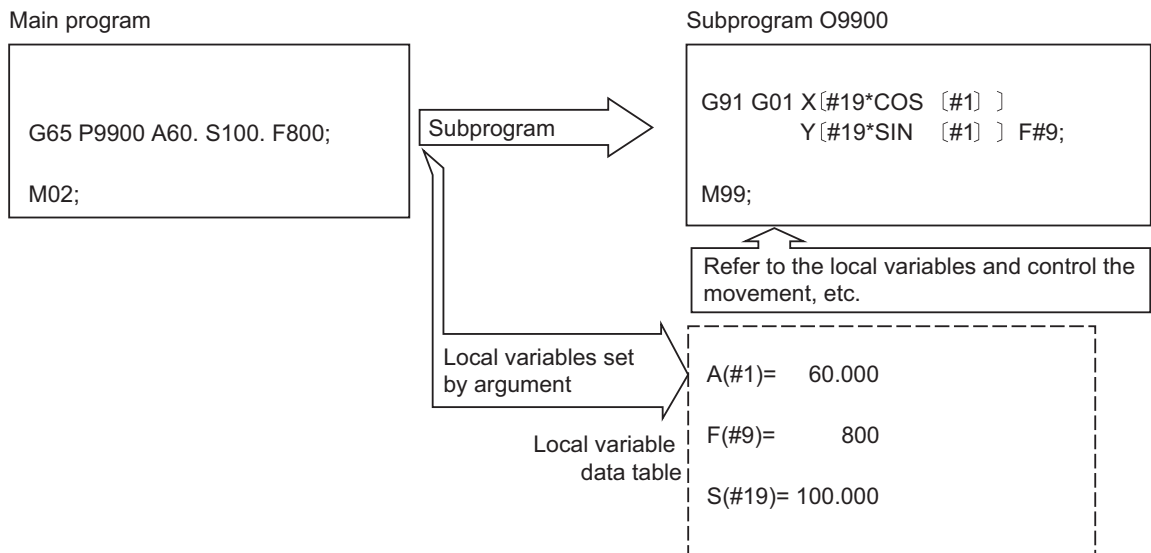
Argument designation II address	Variable in macro
A	#1
B	#2
C	#3
I1	#4
J1	#5
K1	#6
I2	#7
J2	#8
K2	#9
I3	#10
J3	#11
K3	#12
I4	#13
J4	#14
K4	#15
I5	#16

Argument designation II address	Variable in macro
J5	#17
K5	#18
I6	#19
J6	#20
K6	#21
I7	#22
J7	#23
K7	#24
I8	#25
J8	#26
K8	#27
I9	#28
J9	#29
K9	#30
I10	#31
J10	#32
K10	#33

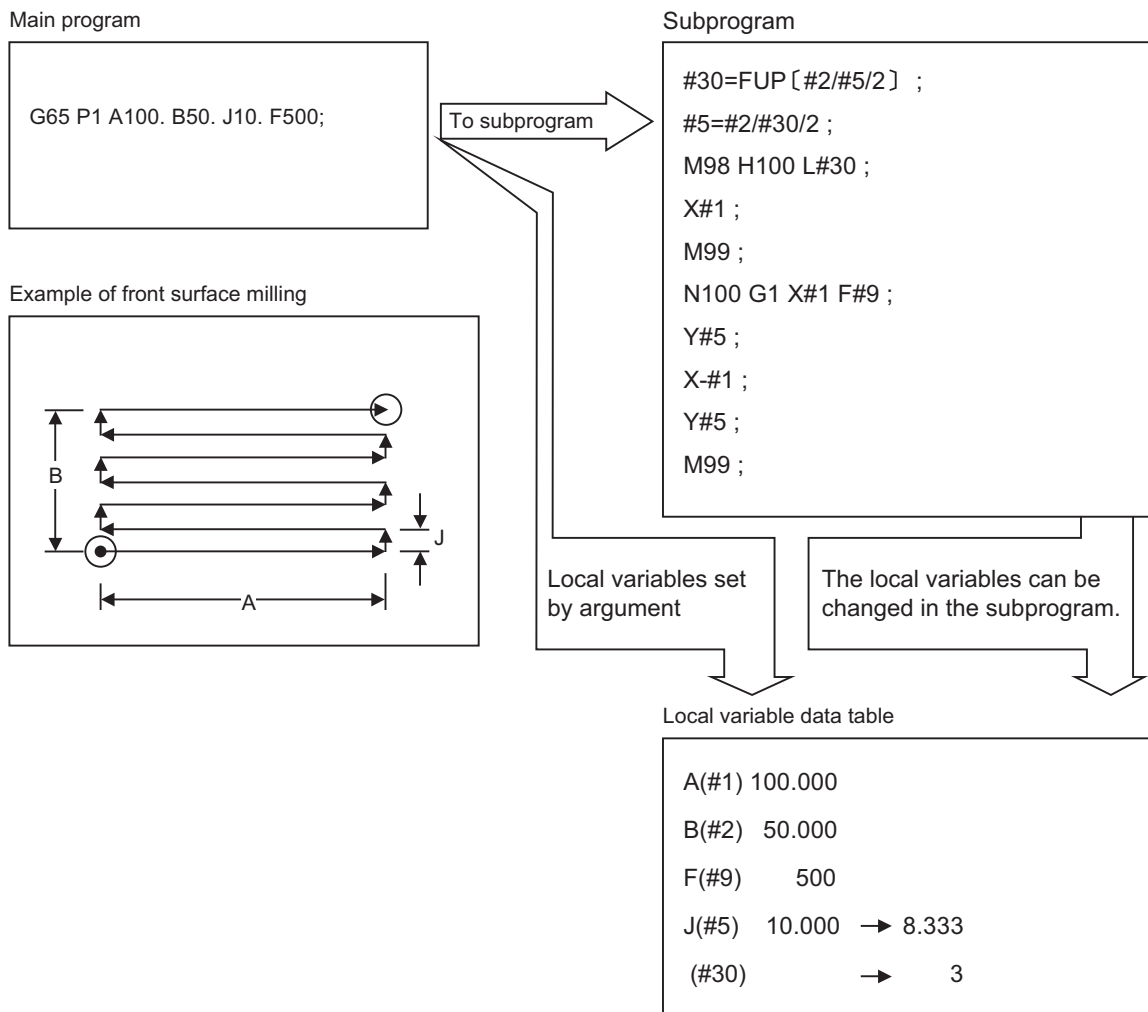
Note

•The numbers 1 to 10 accompanying I, J and K indicate the sequence of the commanded sets, and are not required in the actual command.

(1) Local variables in subprograms can be defined by means of the <argument> designation during macro call. (Local variables can be used freely in those subprograms.)



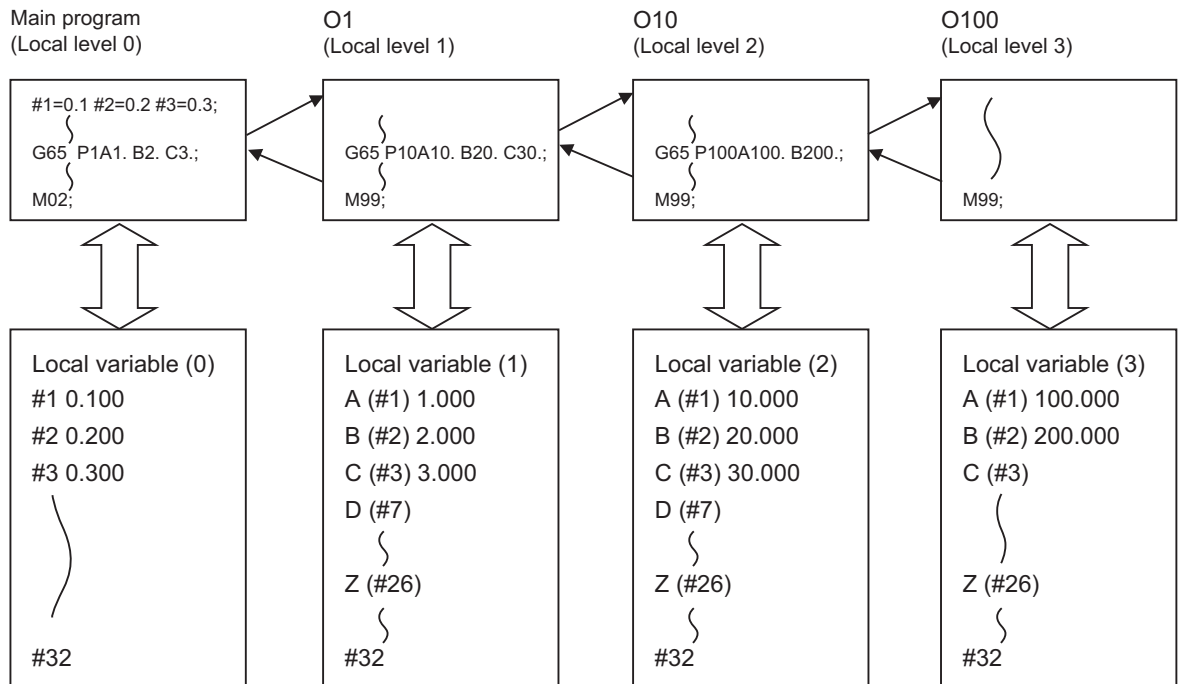
(2) The local variables can be used freely in that subprogram.



In the front surface milling example, argument J is programmed as the milling pitch 10. mm. However, this is changed to 8.333 mm to create an equal interval pitch.

The results of the No. of reciprocation data calculation is set in local variable #30.

- (3) Local variables can be used independently on each of the macro call levels (4 levels). Local variables are also provided independently for the main program (macro level 0). Arguments cannot be used for the level 0 local variables.



The status of the local variables is displayed on the setting and display unit. Refer to the Instruction Manual for details.

14.5.3 System Variables

Data such as the workpiece offset amount can be read using system variables other than common variables or local variables.

Refer to "23 System Variables" for details.

14.6 User Macro Commands

14.6.1 Operation Commands



Function and purpose

A variety of operations can be performed between variables.



Command format

#i = <formula> ;

<Formula> is a combination of constants, variables, functions and operators.

Constants can be used instead of #j and #k below.

(1) Definition and substitution of variables	#i = #j	Definition, substitution
(2) Addition operation	#i = #j + #k	Addition
	#i = #j - #k	Subtraction
	#i = #j OR #k	Logical sum (at every bit of 32 bits)
	#i = #j XOR #k	Exclusive OR (at every bit of 32 bits)
(3) Multiplication operation	#i = #j * #k	Multiplication
	#i = #j / #k	Division
	#i = #j MOD #k	Remainder
	#i = #j AND #k	Logical product (at every bit of 32 bits)
(4) Functions	#i = SIN [#k]	Sine
	#i = COS [#k]	Cosine
	#i = TAN [#k]	Tangent tan θ uses sinθ/cosθ.
	#i = ASIN [#k]	Arcsine
	#i = ATAN [#k]	Arctangent (ATAN or ATN may be used)
	#i = ACOS [#k]	Arccosine
	#i = SQRT [#k]	Square root (SQRT or SQR may be used)
	#i = ABS [#k]	Absolute value
	#i = BIN [#k]	Conversion from BCD to BIN
	#i = BCD [#k]	Conversion from BIN to BCD
	#i = ROUND[#k]	Rounding off (ROUND or RND may be used)
	#i = FIX [#k]	Discarding fractions after decimal point
	#i = FUP [#k]	Add for fractions less than 1
	#i = LN [#k]	Natural logarithm
#i = EXP [#k]	Exponent with e (=2.718) as bottom	
#i = POW [#j, #k]	Power	

Note

- (1) A value without a decimal point is basically treated as a value with a decimal point at the end (1 = 1.000).
- (2) Compensation amounts from #10001 and workpiece coordinate system compensation values from #5201 are handled as data with a decimal point. Consequently, data with a decimal point will be produced even when data without a decimal point have been defined in the variable numbers.
(Example)

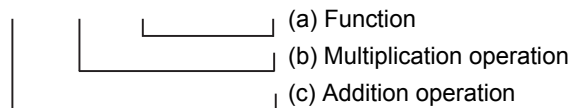
Operation Commands	Common variables after execution
#101 = 1000 ; #10001 = #101 ; #102 = #10001 ;	#101 1000.000 #102 1000.000

- (3) The <formula> after a function must be enclosed in the square parentheses [].

**Detailed description****Sequence of operations**

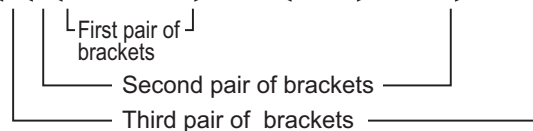
- (1) The sequence of the operations (a) to (c) is performed in the following order; the function, the multiplication operation and the addition operation.

```
#101=#111+#112*SIN [#113]
```



- (2) The part to be given priority in the operation sequence should be enclosed in square parentheses []. Up to 5 pairs of such parentheses, including those for the functions, may be used.

```
#101=SQRT [ [ [ [#111-#112 ] *SIN [#113 ] +#114 ] *#115 ] ;
```



Examples of operation commands

(1) Main program and argument designation	G65 P100 A10 B20.; #101 = 100.000 #102 = 200.000;	#1 10.000 #2 20.000 #101 100.000 #102 200.000		
(2) Definition and substitution =	#1 = 1000 #2 = 1000.	#1 1000.000 #2 1000.000		
	#3 = #101 #4 = #102	#3 100.000 #4 200.000	From common variables	
	#5 = #10001 (#10001 = -10.)	#5 -10.000	From tool compensation	
(3) Addition and subtraction + -	#11 = #1 + 1000 #12 = #2 - 50. #13 = #101 + #1 #14 = #10001 - 3. (#10001 = -10.) #15 = #10001 + #102	#11 2000.000 #12 950.000 #13 1100.000 #14 -13.000 #15 190.000		
(4) Multiplication and division * /	#21 = 100 * 100 #22 = 100. * 100 #23 = 100 * 100. #24 = 100. * 100. #25 = 100 / 100 #26 = 100. / 100 #27 = 100 / 100. #28 = 100. / 100. #29 = #10001 * #101 (#10001 = -10.) #30 = #10001 / #102	#21 10000.000 #22 10000.000 #23 10000.000 #24 10000.000 #25 1.000 #26 1.000 #27 1.000 #28 1.000 #29 -1000.000 #30 -0.050		
(5) Remainder MOD	#19 = 48 #20 = 9 #31 = #19 MOD #20	#19/#20 = 48/9 = 5 Remainder 3 #31 = 3		
(6) Logical sum OR	#3 = 100 #4 = #3 OR 14	#3 = 01100100 (binary) 14 = 00001110 (binary)		
		#4 = 01101110 = 110		
(7) Exclusive OR XOR	#3 = 100 #4 = #3 XOR 14	#3 = 01100100 (binary) 14 = 00001110 (binary)		
		#4 = 01101010 = 106		
(8) Logical product AND	#9 = 100 #10 = #9 AND 15	#9 = 01100100 (binary) 15 = 00001111 (binary)		
		#10 = 00000100 = 4		
(9) Sine SIN	#501 = SIN [60] #502 = SIN [60.] #503 = 1000 * SIN [60] #504 = 1000 * SIN [60.] #505 = 1000. * SIN [60] #506 = 1000. * SIN [60.] <Note> ♦SIN [60] is equivalent to SIN [60.]	#501 0.866 #502 0.866 #503 866.025 #504 866.025 #505 866.025 #506 866.025		
(10) Cosine COS	#541 = COS [45] #542 = COS [45.] #543 = 1000 * COS [45] #544 = 1000 * COS [45.] #545 = 1000. * COS [45] #546 = 1000. * COS [45.] <Note> ♦COS [45] is equivalent to COS [45.]	#541 0.707 #542 0.707 #543 707.107 #544 707.107 #545 707.107 #546 707.107		

14 Macro Functions

(11) Tangent TAN	#551 = TAN [60] #552 = TAN [60.] #553 = 1000 * TAN [60] #554 = 1000 * TAN [60.] #555 = 1000. * TAN [60] #556 = 1000. * TAN [60.] <Note> ♦TAN [60] is equivalent to TAN [60.]	#551 #552 #553 #554 #555 #556	1.732 1.732 1732.051 1732.051 1732.051 1732.051	
(12) Arcsine ASIN	#531 = ASIN[100.500 / 201.] #532 = ASIN[100.500 / 201] #533 = ASIN[0.500] #534 = ASIN[-0.500]	#531 #532 #533 #534	30.000 30.000 30.000 -30.000	
		<Note> ♦When #1273/bit 0 is set to 1, #534 will be 330°.		
(13) Arc tangent ATN or ATAN	#561 = ATAN [173205 / 100000] #562 = ATAN [173205 / 100000.] #563 = ATAN [173.205 / 100] #564 = ATAN [173.205 / 100.] #565 = ATAN [1.73205]	#561 #562 #563 #564 #565	60.000 60.000 60.000 60.000 60.000	
(14) Arccosine ACOS	#521 = ACOS [100 / 141.421] #522 = ACOS [100. / 141.421]	#521 #522	45.000 45.000	
(15) Square root SQR or SQRT	#571 = SQRT [1000] #572 = SQRT [1000.] #573 = SQRT [10. * 10. + 20. * 20] <Note> ♦In order to increase the accuracy, proceed with the operation inside parentheses as much as possible.	#571 #572 #573	31.623 31.623 22.360	
(16) Absolute value ABS	#576 = -1000 #577 = ABS [#576] #3 = 70. #4 = -50. #580 = ABS [#4 - #3]	#576 #577 #580	-1000.000 1000.000 120.000	
(17) BIN, BCD	#1 = 100 #11 = BIN [#1] #12 = BCD [#1]	#11 #12	64 256	
(18) Rounding off RND or ROUND	#21 = ROUND [14 / 3] #22 = ROUND [14. / 3] #23 = ROUND [14 / 3.] #24 = ROUND [14. / 3.] #25 = ROUND [-14 / 3] #26 = ROUND [-14. / 3] #27 = ROUND [-14 / 3.] #28 = ROUND [-14. / 3.]	#21 #22 #23 #24 #25 #26 #27 #28	5 5 5 5 -5 -5 -5 -5	
(19) Discarding frac- tions below decimal point FIX	#21 = FIX [14 / 3] #22 = FIX [14. / 3] #23 = FIX [14 / 3.] #24 = FIX [14. / 3.] #25 = FIX [-14 / 3] #26 = FIX [-14. / 3] #27 = FIX [-14 / 3.] #28 = FIX [-14. / 3.]	#21 #22 #23 #24 #25 #26 #27 #28	4.000 4.000 4.000 4.000 -4.000 -4.000 -4.000 -4.000	
(20) Adding fractions less than 1 FUP	#21 = FUP [14 / 3] #22 = FUP [14. / 3] #23 = FUP [14 / 3.] #24 = FUP [14. / 3.] #25 = FUP [-14 / 3] #26 = FUP [-14. / 3] #27 = FUP [-14 / 3.] #28 = FUP [-14. / 3.]	#21 #22 #23 #24 #25 #26 #27 #28	5.000 5.000 5.000 5.000 -5.000 -5.000 -5.000 -5.000	

(21) Natural logarithms LN	#10 = LN [5] #102 = LN [0.5] #103 = LN [-5]	#101 #102 Error	1.609 -0.693 "P282"	
(22) Exponents EXP	#104 = EXP [2] #105 = EXP [1] #106 = EXP [-2]	#104 #105 #106	7.389 2.718 0.135	
(23) Power POW	#107 = POW [2, 3] #108 = POW [2, -3] #109 = POW [2.5, 3.5] #110 = POW [0, -1] #111 = POW [-2, 2.5]	#107 #108 #109 Error Error	8.000 0.125 24.705 "P282" "P282"	



Precautions

(1) Notes on logical relation

EQ, NE, GT, LT, GE and LE conduct the same calculation as addition and subtraction. Pay careful attention to errors. For example, to determine whether or not #10 and #20 are equal in the following example, it is not always possible to judge correctly because of the error.

```
IF [#10 EQ #20]
```

Therefore when the difference between #10 and #20 falls within the designated error range, both values should be considered equal.

```
IF [ABS [#10 - #20] LT 200000]
```

(2) If an operation command using a function is executed, a program error (P282) will occur when:

- A number that sets $\cos\theta$ to "0" has been designated in the argument of the tangent command (TAN).
- A negative number has been designated in the argument of the square root command (SQR).
- A negative number has been designated in the argument of the logarithm natural command (LN).
- "0" has been set to argument 1 and "0" or less to argument 2 in the power command (POW).
- A negative number has been set to argument 1 and a non-integer to argument 2 in the power command (POW).

14.6.2 Control Commands

**Function and purpose**

The flow of programs can be controlled by "IF-GOTO-", "IF-THEN-ELSE-ENDIF", and "WHILE-DO-".

When a program in an external device such as a USB memory device is executed, a period of processing time is required in the subprogram call or in the instruction to change the flow of the program such as GOTO or DO-END; therefore, interpolation may be decelerated or stopped.

**Detailed description****Branch (IF-GOTO-)**

IF [conditional expression] GOTO n; (n = sequence number in the program)

When the condition is satisfied, control branches to "n" and when it is not satisfied, the next block is executed.

IF [conditional expression] can be omitted and, when it is, control branches to "n" unconditionally.

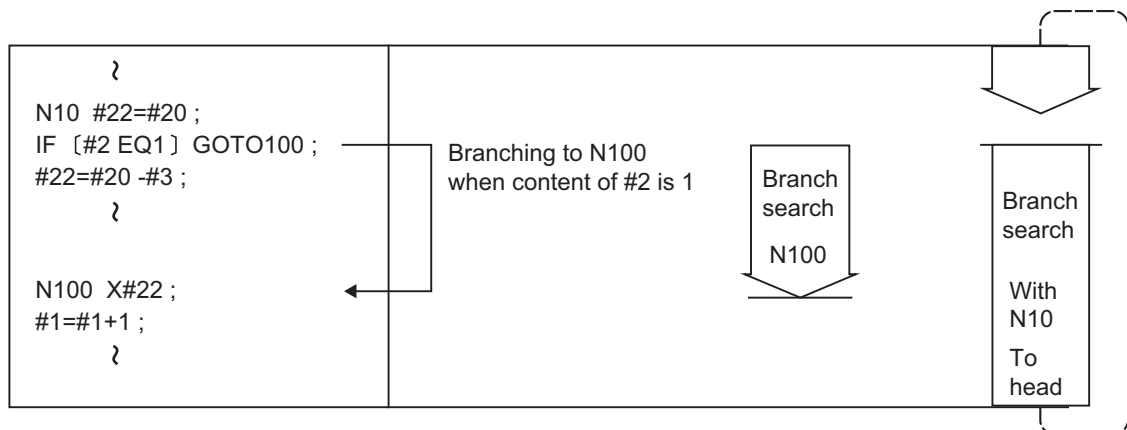
The following types of [conditional expressions] are available.

#i EQ #j	= When #i and #j are equal
#i NE #j	≠ When #i and #j are not equal
#i GT #j	> When #i is greater than #j
#i LT #j	< When #i is less than #j
#i GE #j	>= When #i is #j or more
#i LE #j	<= When #i is #j or less

"n" of "GOTO n" must always be in the same program. If not, program error (P231) will occur. A formula or variable can be used instead of i, #j and n.

In the block with sequence number "n" which will be executed after a "GOTO n" command, the sequence number "Nn" must always be at the head of the block. Otherwise, program error (P231) will occur.

If "/" is at the head of the block and "Nn" follows, control can be branched to the sequence number.

**Note**

- (1) When searching the sequence number of the branch destination, the search is conducted up to the end of the program (% code) from the block following IF.....; and if it is not found, it is then conducted from the top of the program to the block before IF.....; Therefore, branch searches in the opposite direction to the program flow will take longer time compared with branch searches in the forward direction.
- (2) EQ and NE should be compared only for integers. For comparison of numeric values with decimals, GE, GT, LE, and LT should be used.

Branch (IF-THEN-ELSE-ENDIF)

(a)	IF [conditional expression] THEN ; Macro statement or executable statement : ELSE ; Macro statement or executable statement : ENDIF ;
(b)	IF [conditional expression] THEN operation command ; ELSE operation command ;
(c)	IF [conditional expression] THEN operation command ELSE operation command ;

- (1) When the condition is satisfied, THEN-side processing is executed. Otherwise, ELSE-side processing is executed.
- (2) Designate the conditional expression in the same way as for the "IF-GOTO-" command.
- (3) If neither the THEN nor ELSE command is designated in the same block as for the IF command (the IF statement is commanded individually), a program error (P33) will occur.
- (4) When the run command has an executable statement or multiple commands are designated, enclose them in the IF, THEN, ELSE, and ENDIF rows as shown in format (a).
- (5) If the run command is an operation instruction, it can be described following THEN or ELSE as shown in formats (b) and (c).
- (6) If the ENDIF command is omitted in format (a), a program error (P289) will occur.
In format (b) or (c), the ENDIF command can be described like format (a).
When the IF statement is used as a nesting in format (b) or (c), designate the ENDIF command.
<Example> When the IF statement is used as a nesting, the operation is performed as shown below.

```

IF[ #100 EQ 0 ] THEN ;           A
IF[ #110 EQ 1 ] THEN #120 = 10 ; B
ENDIF ;                          C
ELSE ;
#120 = 20;
ENDIF ;
    
```

[Operation]

The following operation is performed depending on whether the ENDIF command is designated in "C".

- ENDIF command designated: ELSE processing is executed when the IF condition of A is false.
- ENDIF command undesignated: ELSE processing is executed when the IF condition of B is false.

- (7) The THEN-side processing or ELSE-side processing can be omitted in any of formats (a) to (c).

When ELSE-side processing is omitted:	IF[#100 EQ 0] THEN ; #100 = 2 ; G00 X#101 ; ENDIF ;
When THEN-side processing is omitted:	IF[#100 EQ 0] ELSE #110 =10 ;

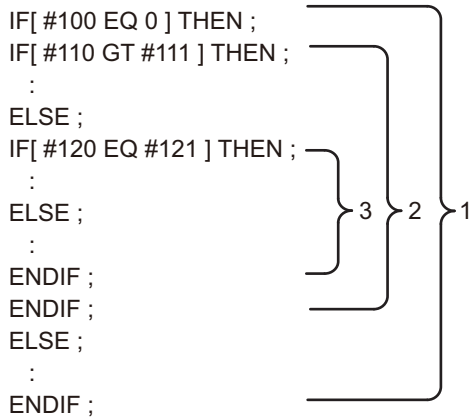
- (8) Formats (a) and (b) can be combined to issue commands.

```

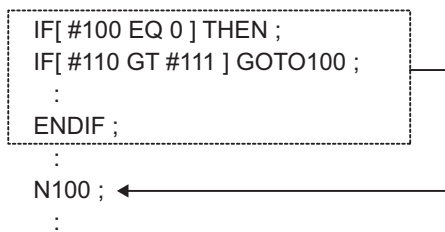
IF[ #100 EQ 0 ] THEN ;
#100 = 2 ;
G00 X#101 ;
ELSE #110 =10 ;
ENDIF ;
    
```

- (9) If any of THEN, ELSE, and ENDIF is commanded with no IF command issued, a program error (P289) will occur.

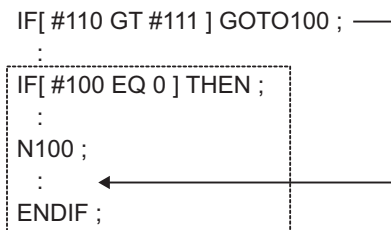
- (10) The IF statement can be commanded up to 10 nesting levels. When the nesting level exceeds 10, a program error (P288) will occur. The following shows an example in which the nesting level is set to 3.



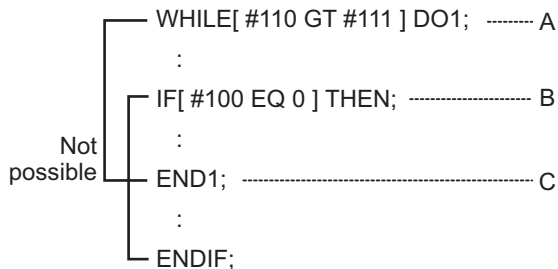
- (11) You can set a branch from the inside of the IF to ENDIF range to the outside.



- (12) Do not set a branch to the inside of the IF to ENDIF range, including the ENDIF block. Branching disables the skipped IF command and executes all the commands designated up to the ENDIF command that is paired with the IF command.



- (13) If IF to ENDIF is intersected with WHILE-DO to END, a program error (P288, P289, or P294) will occur.



If "A" to "C" are repeated 11 times or more while the IF condition in "B" is true, a program error (P288) will occur in "B".

When the IF condition in "B" is false, "C" is not executed, so "A" to "C" are not processed repeatedly.

- (14) You can call a subprogram (M98, G65, G66, etc.) from the inside of the IF to ENDIF range. Also, you can execute the IF, THEN, ELSE, and ENDIF commands in a subprogram. The IF statement can be commanded up to 10 nesting levels even in a subprogram. (The IF statement can be commanded up to 10 nesting levels for each program.)
- (15) The IF statement processing (IF to ENDIF) must be ended in the same program. If processing is not ended in the same program, a program error (P289) will occur.
- (16) The block skip ("/") for the IF/THEN/ELSE/ENDIF command block is valid only when it is assigned to the head of the block. The block skip ("/") assigned to the middle of the IF/THEN/ELSE/ENDIF command block is handled as shown below regardless of the setting of the optional block skip type (parameter "#1226 aux10/bit1").
 - When the block skip ("/") is assigned just after the THEN or ELSE command, it is ignored.
 - Otherwise, the block skip ("/") is handled as a division command.

<Example> When the block skip ("/") is assigned to the head of the IF statement block, the operation is performed as shown below.

/ IF[#100 EQ 0] THEN #100 =10 ;	When "Optional block skip" signal is ON, IF statement will not be executed.
-----------------------------------	---

Repetitions

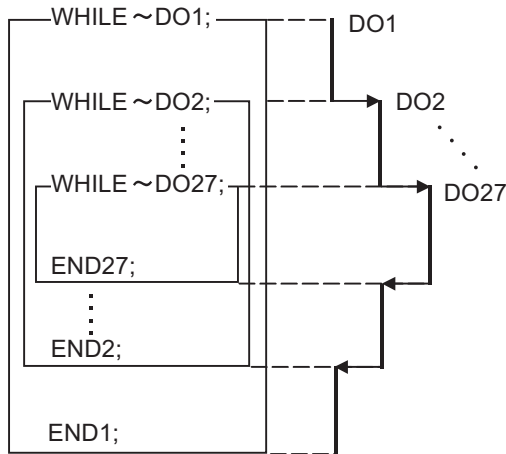
```

WHILE [conditional expression] DOm ; (m =1, 2, 3 ..... 127) :
:
END m ;
    
```

While the conditional expression is established, the blocks from the following block to ENDm are repeatedly executed; when it is not established, execution moves to the block following ENDm. DOm may come before WHILE. "WHILE [conditional expression] DOm" and "ENDm" must be used as a pair. If "WHILE [conditional expression]" is omitted, these blocks will be repeatedly ad infinitum. The repeating identification Nos. range from 1 to 127. (DO1, DO2, DO3,DO127) Up to 27 nesting levels can be used.

<p>(1) Same identification No. can be used any number of times.</p> <div style="margin: 10px 0;"> <pre> WHILE ~DO1; [] END1; </pre> </div> <div style="margin: 10px 0;"> <pre> WHILE ~DO1; [] END1; </pre> </div>	<p>(2) Any number may be used as the WHILE-DOm identification No.</p> <div style="margin: 10px 0;"> <pre> WHILE ~DO1; } END1; } - WHILE ~DO3; } END3; } - WHILE ~DO2; } END2; } - WHILE ~DO1; } END1; </pre> </div>
---	---

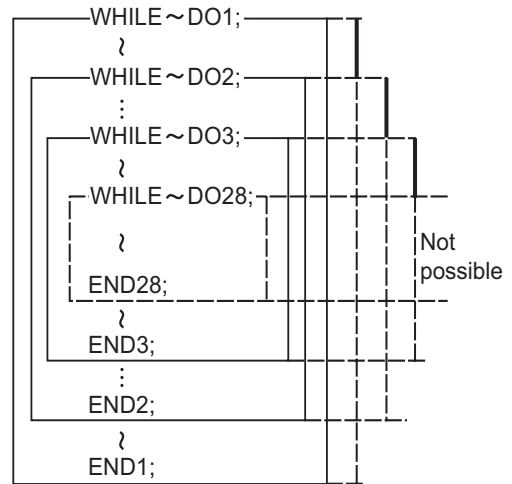
(3) Up to 27 nesting levels can be used for WHILE-DOm. "m" is any number from 1 to 127 for the nesting depth.



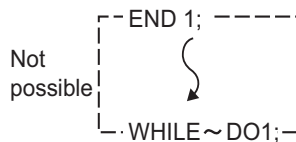
<Note>

- For nesting, "m" which has been used once cannot be used.

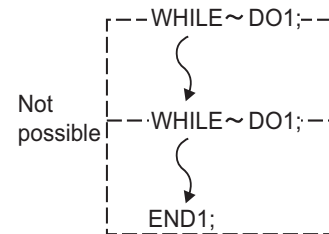
(4) The number of WHILE-DOm nesting levels cannot exceed 27.



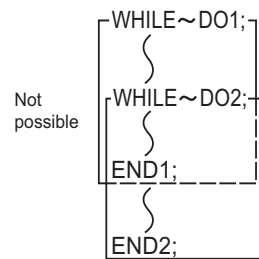
(5) WHILE - DOm must be designated first and ENDm last.



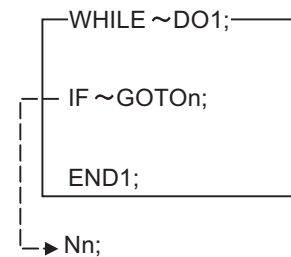
(6) WHILE - DOm and ENDm must correspond on a 1:1 (pairing) basis in a same program.



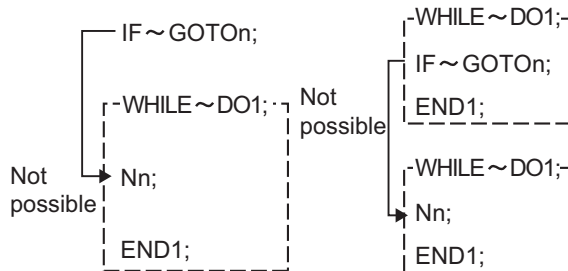
(7) Two WHILE - DOm's must not overlap.



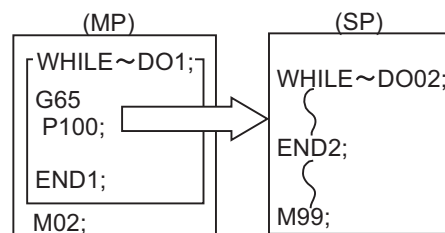
(8) Branching externally out of the WHILE - DOm range, is possible.

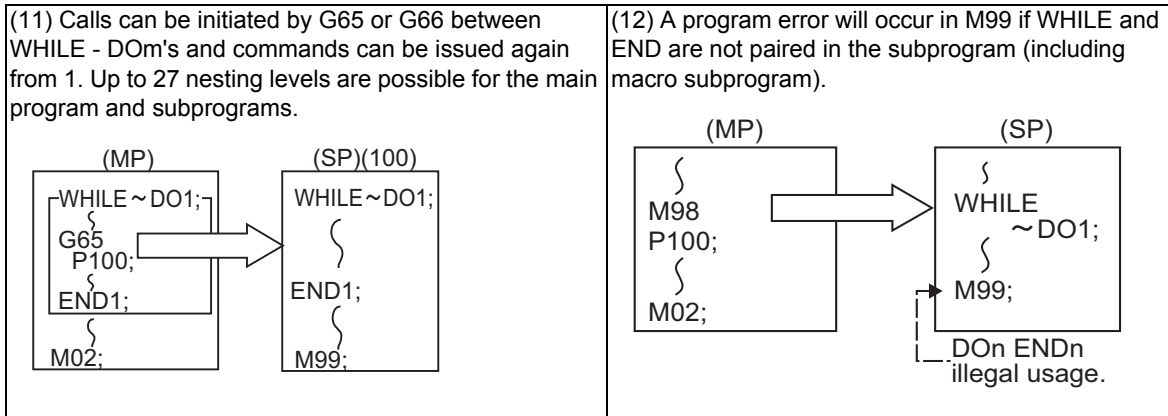


(9) No branching into WHILE - DOm, is possible.



(10) Subprograms can be called by M98, G65 or G66 between WHILE - DOm's.





(MP) Main program

(SP) Subprogram

Note

- Even if a fixed cycle containing WHILE is called, the nesting level will be counted up.

14.6.3 External Output Commands; POPEN, PCLOS, DPRNT

**Function and purpose**

Besides the standard user macro commands, the following macro instructions are also available as external output commands. They are designed to output the variable values or characters to external devices. The data output port can be chosen from RS-232C or memory card.

**Command format****Open command**

POPEN

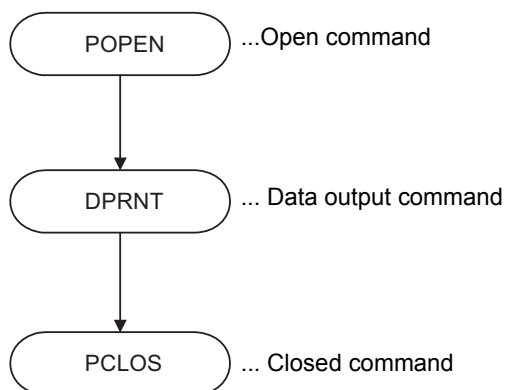
Closed command

PCLOS

Data output command

DPRNT

POPEN	For preparing the data outputs
PCLOS	For terminating the data outputs
DPRNT	For character output and digit-by-digit variable numerical output

Command sequence



Detailed description

Open command : POPEN

- (1) The command is issued before the series of data output commands.
- (2) The DC2 control code and % code are output from the NC system to the external output device.
- (3) Once POPEN; is issued, it will remain valid until PCLOS; is issued.

Close command : PCLOS

- (1) This command is issued when all the data outputs are completed.
- (2) The DC4 control code and % code are output from the NC system to the external output device.
- (3) This command is used together with the open command and it should not be issued unless the open mode has been established.
- (4) Issue the close command at the end of the program even when the operation is suspended by resetting or some other operation during data output.

Data output command : DPRNT

DPRNT [I1#v1 [d1 c1] I2#v2 [d2 c2]] ;		
I1	Character string	
v1	Variable No.	
d1	Significant digits above decimal point	c + d <= 8
c1	Significant digits after decimal point	

- (1) The character output and decimal output of the variable values are done with ISO codes.
- (2) The commanded character string is output as it is by the ISO code.
Alphanumerics (A to Z, 0 to 9) and special characters (+, -, *, /) can be used.
Note that asterisk (*) is output as a space code.
- (3) The required significant digits above and below the decimal point of the variable values are each commanded within square parentheses. As a result, the commanded number of digits of variable values are output in ISO code in decimal notation from the high-order digits including the decimal point. In this case, trailing zeroes are not omitted.
- (4) Leading zeroes are omitted.
The omitted leading zero can be replaced by a space by the setting of a parameter.
This can justify the last digit of the data output to the printer.
- (5) Linefeed (LF) code will be output to the end of the output data.
And by setting the parameter "#9112 to #9512 DEV0 - 4 CR OUTPUT" to "1", (CR) code will be written in just before EOB (LF) code.

Note

♦A data output command can be issued even in two-part system mode. In this case, however, note that the output channel is shared by both part systems. So, be careful not to execute data output by both part systems simultaneously.

Data output port

- (1) The output port can be selected by the parameter "#9007 MACRO PRINT PORT".
- (2) When the port is a memory card, the file name of the port can be designated by the parameter "#9054 MACRO PRINT FILE".
- (3) When the port is a memory card, the port directory is fixed to root directory.



Use example:

<Parameter setting>

#1127 DPRINT (DPRINT alignment)	= 1 (Align the minimum digit and output)
#9007 MACRO PRINT PORT	= 9 (Output to a memory card by an external output command)
#9008 MACRO PRINT DEV.	= 0 (Device 0 is selected for an external output command)
#9054 MACRO PRINT FILE	= DPRNT_OUT (File name to store output data of an external output command)
#9112 DEV0 CR OUTPUT	= 1 (Insert the CR code just before the LF code)

<Machining program>

```
#1=12.34;
#2=#0
#100=-123456789.;
#500=-0.123456789;
POPEN;
DPRNT[]; (*1)
DPRNT[VAL-CHECK];
DPRNT[1234567890];
DPRNT[#1[44]];
DPRNT[#2[44]];
DPRNT[#100[80]];
DPRNT[#500[80]];
DPRNT[#100[08]];
DPRNT[#500[08]];
PCLOS;
M30;
%
```

(*1) Designate a blank line when opening an output file on an edit screen. If not, it will be regarded that no information is provided in the head line of the file on the NC edit screen.

<Output file (file name = DPRNT_OUT)>

Blank										Linefeed code	
(CR)	(LF)									(CR)	(LF)
V	A	L	-	C	H	E	C	K			
1	2	3	4	5	6	7	8	9	0	(CR)	(LF)
			1	2	.	3	4	0	0	(CR)	(LF)
				0	.	0	0	0	0	(CR)	(LF)
-	2	3	4	5	6	7	8	9		(CR)	(LF)
							-	0		(CR)	(LF)
-	.	0	0	0	0	0	0	0	0	(CR)	(LF)
-	.	1	2	3	4	5	6	7	9	(CR)	(LF)

Values above the number of significant figures are rounded down

Values below the number of significant figures are rounded off



Precautions

- (1) An external output command during restart-searching is ignored.
After using the restart-search type 2 to restart-search between a POPEN command and a PCLOS command, execute a POPEN command by such as an MDI interruption before restarting the program.
- (2) An external output command during graphic check is ignored.
- (3) A program error (P460) will occur if an external output command is issued when the output device is unable to output due to a lack of connection, a low free space etc.
- (4) The NC automatically conducts a closing processing when it is reset between a POPEN command and a PCLOS command. So, execute a POPEN command by such as an MDI interruption before executing the rest of the machining program.
- (5) If a program error occurs between a POPEN command and a PCLOS command, NC will not automatically conduct a closing processing. So, there is no need to execute a POPEN command by such as an MDI interruption before executing the rest of the machining program.
- (6) If a program error occurs to the output port due to the setting of the memory card, execute an NC reset and close the output file before demounting the card.
- (7) When the output port is a memory card, the output file may be destroyed if the card is dismounted or the power is turned off without issuing a PCLOS command or NC reset after a POPEN command is issued.
- (8) As for M800 series, output data of an external output command can be output to a memory card only when the drive name of the card is "E:" or "F:". Drive name "E" is given the priority. A program error (P460) will occur if the output port executes the external output command of the memory card when the drive name is neither "E:" nor "F:".
- (9) When the data is output to a memory card, the maximum number of files that can be created is determined by the FAT16 format.

14.6.4 Precautions



Precautions

When the user macro commands are employed, it is possible to use the M, S, T and other NC control commands together with the arithmetic, decision, branching and other macro commands for preparing the machining programs. When the former commands are made into executable statements and the latter commands into macro statements, the macro statement processing should be accomplished as quickly as possible in order to minimize the machining time, because such processing is not directly related to machine control.

By setting the parameter "#8101 MACRO SINGLE", the macro statements can be processed concurrently with the execution of the executable statement.

(During normal machining, set the parameter OFF to process all the macro statements together, and during a program check, set it ON to execute the macro statements block by block. Setting can be chosen depending on the purpose.)

There is a signal to notify this macro single setting status to the sequence program. This signal is set ON when "1" (valid) is set to the control parameter "#8101 MACRO SINGLE". (Only for C80 series)

The operation of the PLC signal depends on the MTB specifications.

Program example

N1 G91 G28 X0 Y0 ;(1)	
N2 G92 X0 Y0 ;(2)	
N3 G00 X-100. Y-100. ;(3)	
N4 #101 = 100. * COS[210.] ;(4)	(4),(5) Macro statements
N5 #103 = 100. * SIN[210.] ;(5)	
N6 G01 X#101 Y#103 F800 ;(6)	

Macro statements are:

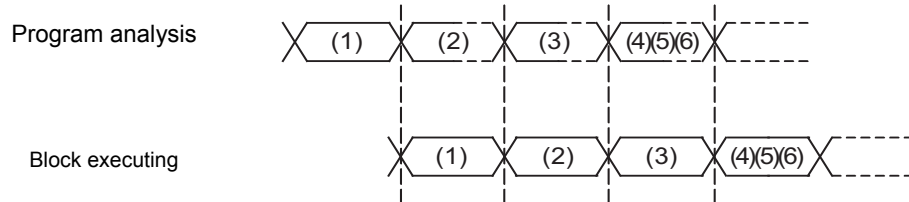
- (a) Arithmetic commands (block including =)
- (b) Control commands (block including GOTO, DO-END, etc.)
- (c) Macro call commands (including macro calls based on G codes and cancel commands (G65, G66, G66.1, G67))

Execution statements refer to statements other than macro statements.

Flow of processing by the Program Example in the previous page

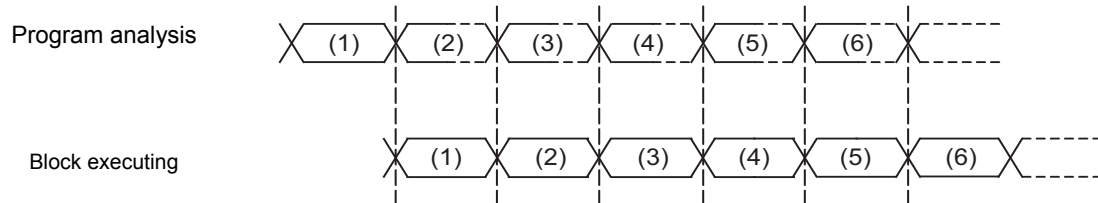
<Macro single OFF>

N4, N5 and N6 are processed in parallel with the control of the executable statement of N3. If the N4, N5 and N6 analysis is in time during N3 control, the machine movement will be continuously controlled.



<Macro single ON>

N4 is processed in parallel with the control of the executable statement of N3. After N3 is finished, N5 and N6 are analyzed, and then N6 is executed. So the machine control is held on standby during the N5 and N6 analysis time.



14.6.5 Actual Examples of Using User Macros



Program example

The following three examples will be described.

(Example 1) SIN curve

(Example 2) Bolt hole circle

(Example 3) Grid

(Example 1) SIN curve

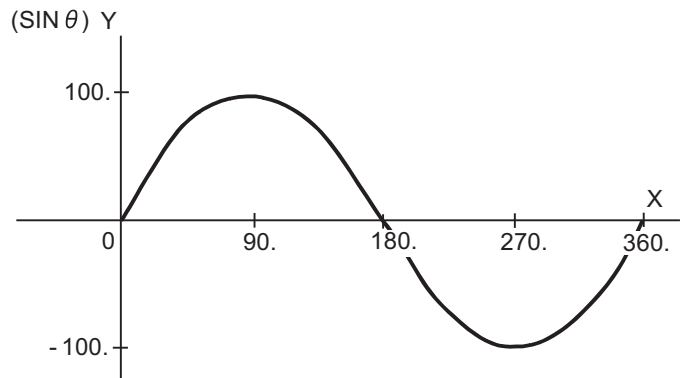
G65 Pp1 Aa1 Bb1 Cc1 Ff1;

a1; Initial value 0°

b1; Final value 360°

c1; R of R*SINθ

f1; Feedrate



Main Program

```

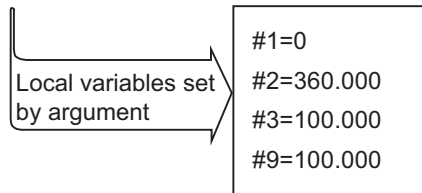
}
G65 P9910 A0 B360.C100.F100;
}
    
```

To subprogram

O9910(Subprogram)

```

WHILE [#1LE#2] DO1;
#101=#3*SIN [#1] ;
G90 G01 X#1 Y#10 F#9; } (Note 1)
#1=#1+10.;
END1;
M99;
    
```

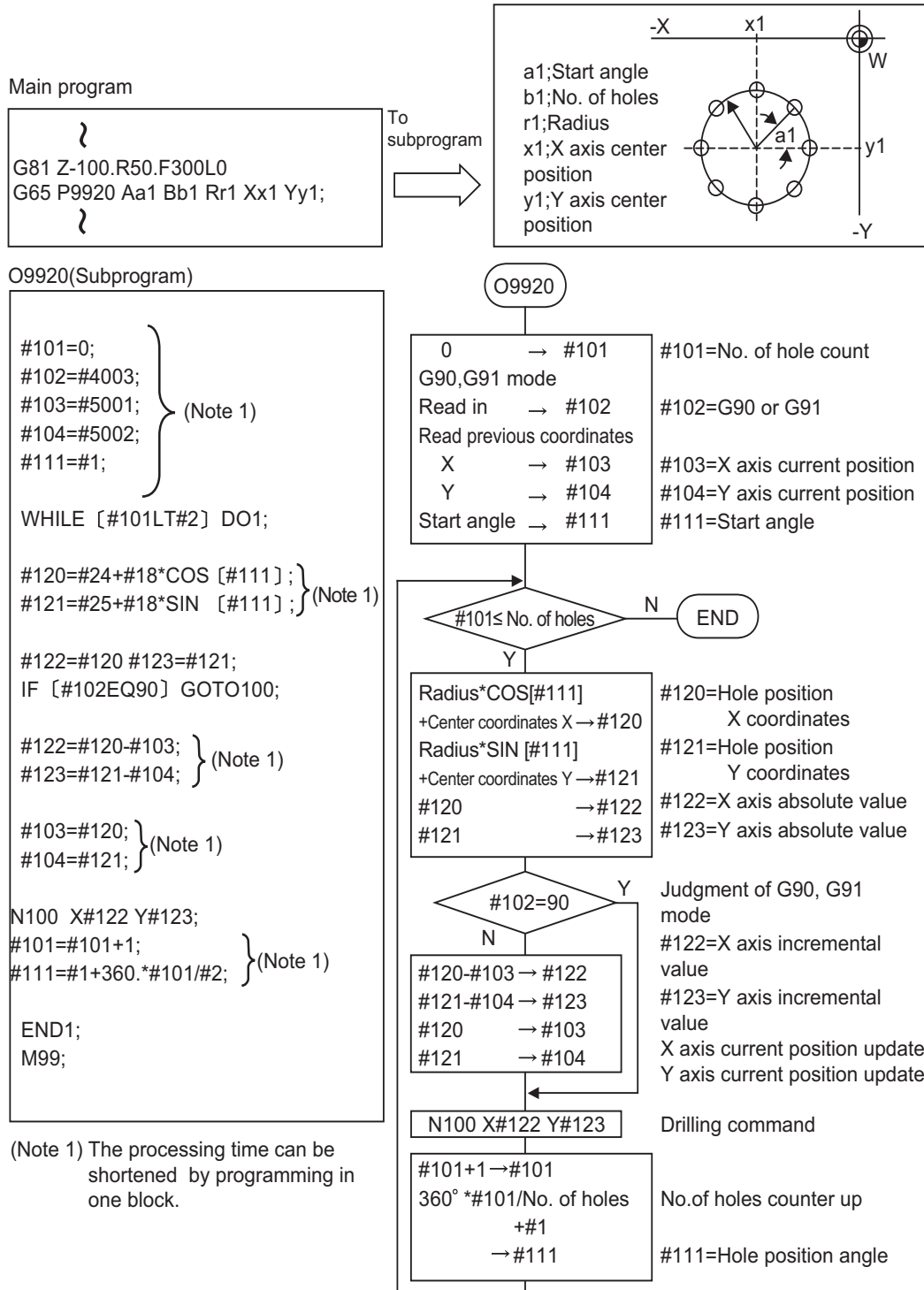


(Note 1) Commanding with one block is possible when G90G01X#1Y[#3*SIN[#1]] F#9; is issued.

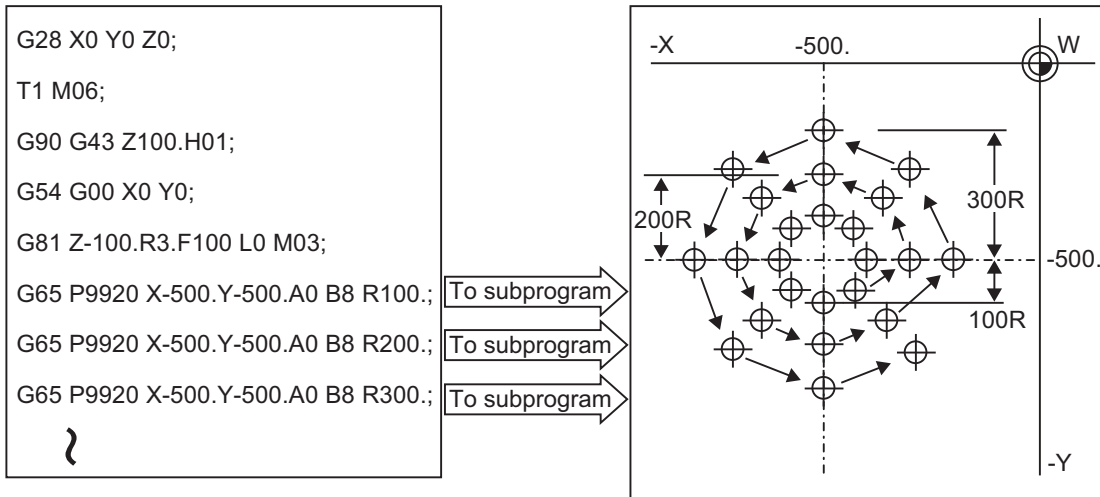
14 Macro Functions

(Example 2) Bolt hole circle

After defining the hole data with fixed cycle (G72 to G89), the macro command is issued as the hole position command.

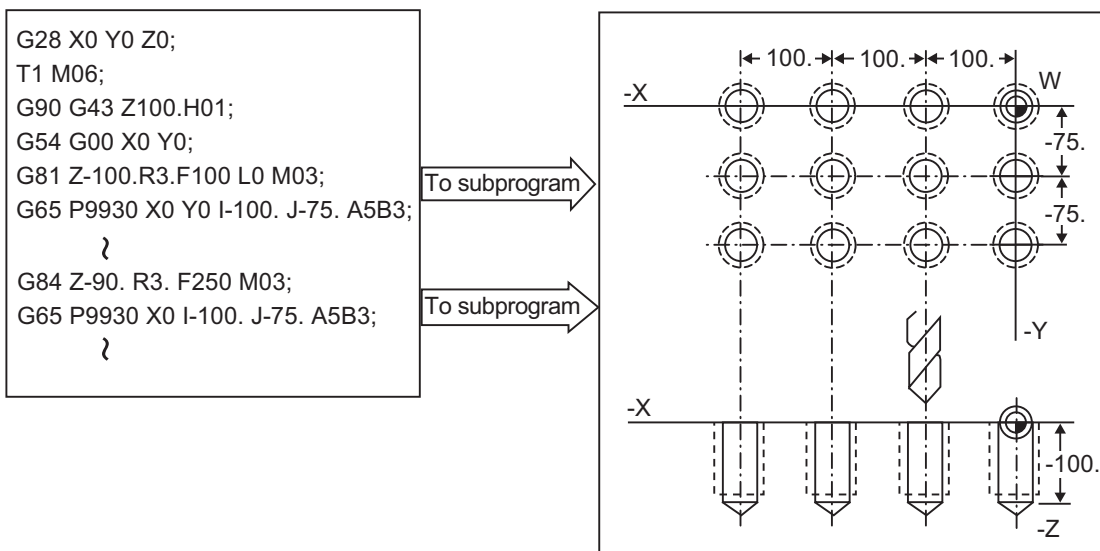
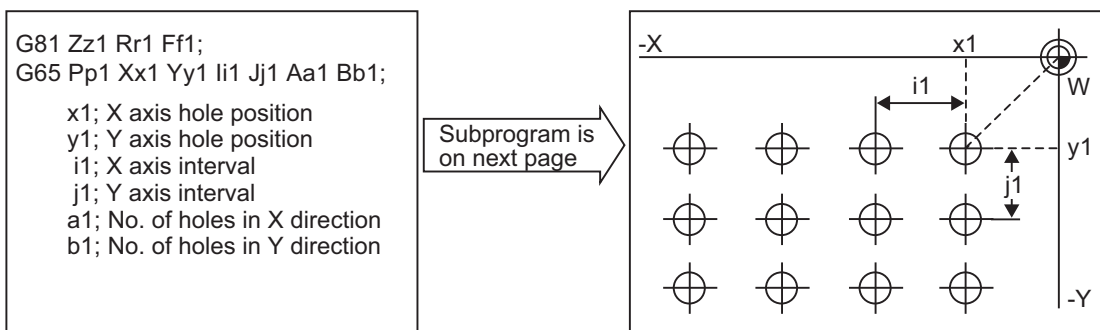


(Note 1) The processing time can be shortened by programming in one block.



(Example 3) Grid

After defining the hole data with the fixed cycle (G72 to G89), macro call is commanded as a hole position command.



14 Macro Functions

O9930(Subprogram)

```

#101=#24;
#102=#25;

#103=#4;
#104=#5;

#106=#2;

WHILE [#106GT0] DO1;

#105=#1;

WHILE [#105GT0] DO2;
G90 X#101 Y#102;

#101=#101+#103;
#105=#105-1;

END2;

#101=#101-#103;
#102=#102+#104;

#103=-#103;
#106=#106-1;

END1;

M99;
    
```

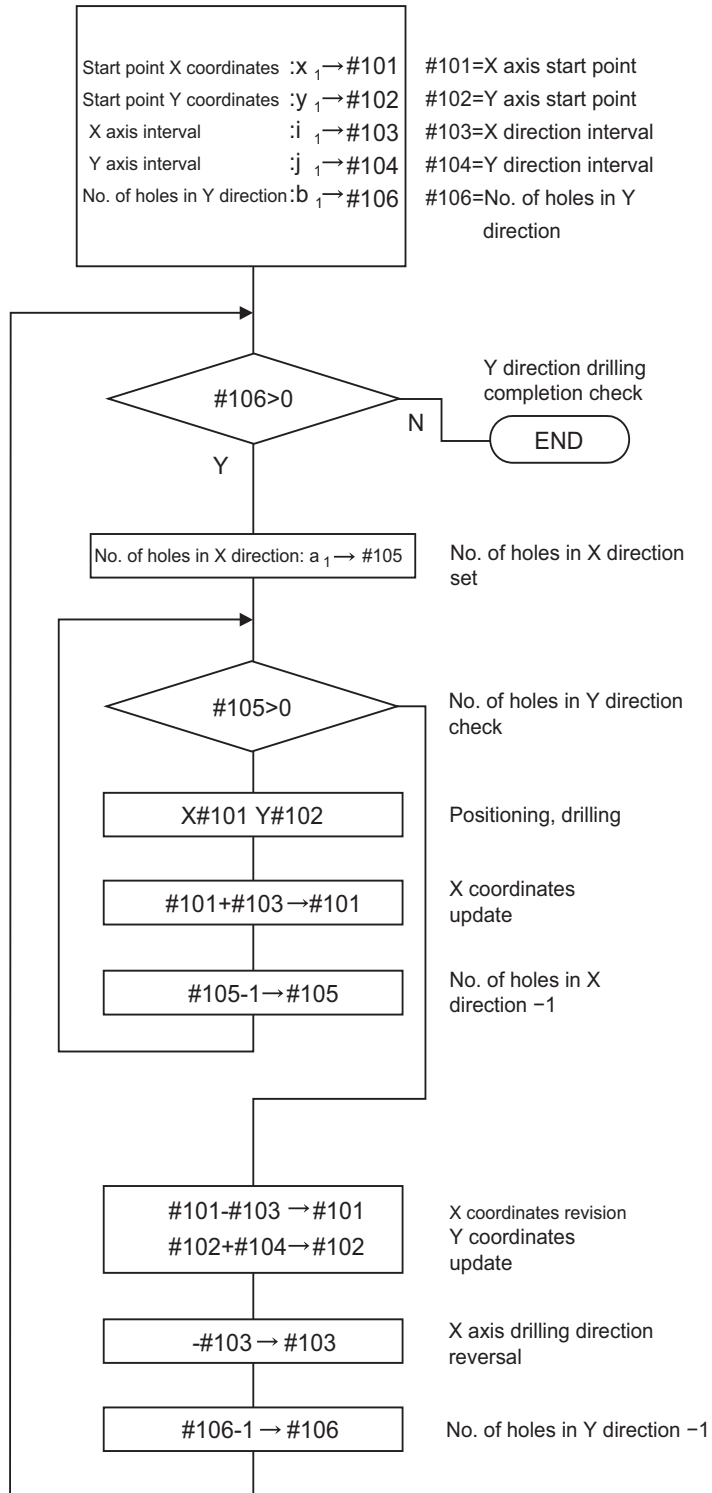
(Note 1)

(Note 1)

(Note 1)

(Note 1) The processing time can be shortened by programming in one block.

O9930



14.7 Macro Interruption; M96, M97



Function and purpose

A user macro interrupt signal (UIT) is input from the machine to interrupt the program currently being executed, and instead calls and executes another program. This is called the user macro interrupt function. Use of this function allows the program to operate flexibly enough to meet varying conditions.



Command format

User macro interruption enable

M96 P__ H__ ;

M96 <File name> H__ ;

P	Interrupt program No. Use a parameter to read out a 4- or 8-digit interrupt program No. starting with O.
<File name>	File name A file name can be specified instead of a program No. In this case, enclose the file name inside brackets < >. (The file name can have up to 32 characters including the extension.)
H	Interrupt sequence No.

User macro interruption disable

M97 ;



Detailed description

- (1) The user macro interrupt function is enabled and disabled by the M96 and M97 commands programmed to make the user macro interrupt signal (UIT) valid or invalid. That is, if an interrupt signal (UIT) is input from the machine side in a user macro interruption enable period from when M96 is issued to when M97 is issued or the NC is reset, a user macro interruption is caused to execute the program specified by P__ instead of the one being executed currently.
- (2) Another interrupt signal (UIT) is ignored until M96 is commanded while one user macro interrupt is in service. It is also ignored in a user macro interrupt disable state such as after an M97 command is issued or the system is reset.
- (3) M96 and M97 are processed internally as user macro interrupt control M codes.
- (4) If calling a subprogram numbered with O is enabled, a program number starting with O and specified by P command value is called with a priority.
However, when P command value is less than the digit number set with parameter "#8129 subprogram number selection", increase the digit number of command value by adding leading zeros.
(Example) When parameter "#8129 subprogram number selection"="1", call the subprogram "O0012" with "M96 P12" command.
- (5) In the following cases, a subprogram of P command value without O No. is called even with a setting to call a subprogram with O No.
 - The digit number of P command value is over the digit number of the program number set with parameter "#8129 subprogram number selection".
 - An interrupt program starting with commanded O No. does not exist.

Enabling conditions

A user macro interruption is enabled only during execution of a program.

The enabling conditions are as follows:

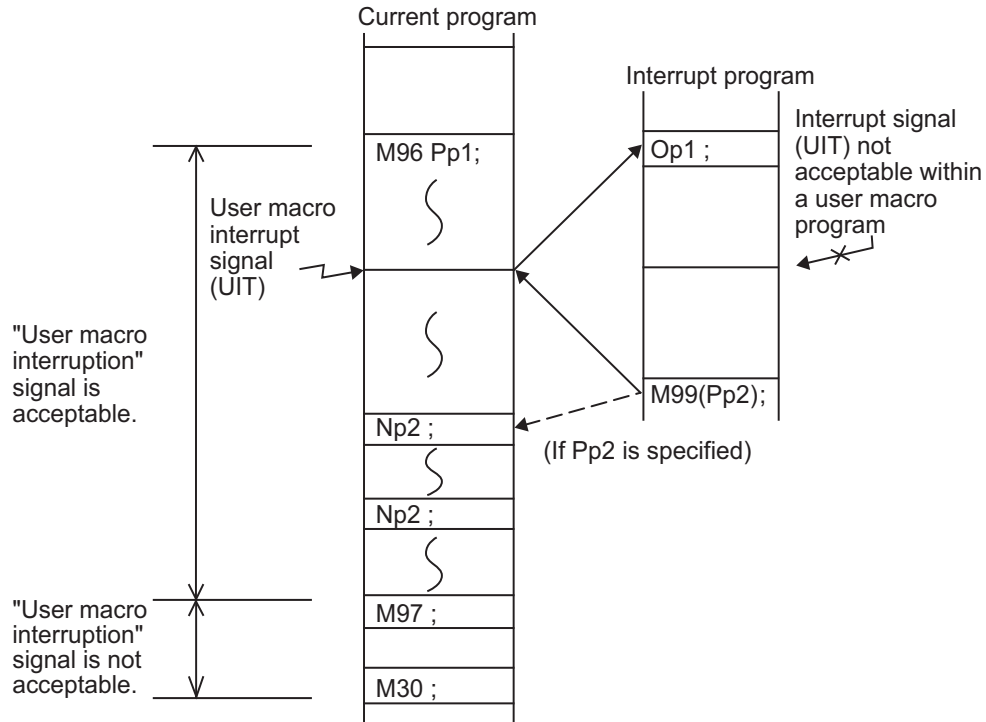
- (1) An automatic operation mode or MDI has been selected.
- (2) The system is running in automatic mode.
- (3) No user macro interruption is being processed.

Note

- (1) A macro interruption is disabled in manual operation mode (JOG, STEP, HANDLE, etc.)

Outline of operation

- (1) When a user macro interrupt signal (UIT) is input after an M96Pp1 ; command is issued by the current program, interrupt program Op1 is executed. When an M99 ; command is issued by the interrupt program, control returns to the main program.
- (2) If M99 Pp2 ; is specified, the blocks from the one next to the interrupted block to the last one are searched. If none is found, blocks between the first block of the program and the one before the interrupted block are searched. Control then returns to the block with sequence number Np2 that is found first in the above search.



Interrupt type

Interrupt types 1 and 2 can be selected by the parameter "#1113 INT_2".

[Type 1]

- (1) When an interrupt signal (UIT) is input, the system immediately stops moving the tool and interrupts dwell, then permits the interrupt program to run.
- (2) If the interrupt program contains a move or miscellaneous function (MSTB) command, the commands in the interrupted block are lost. After the interrupt program completes, the main program resumes operation from the block next to the interrupted one.
- (3) If the interrupted program contains no move and miscellaneous (MSTB) commands, it resumes operation, after completion of the interrupt program, from the point in the block where the interrupt was caused.

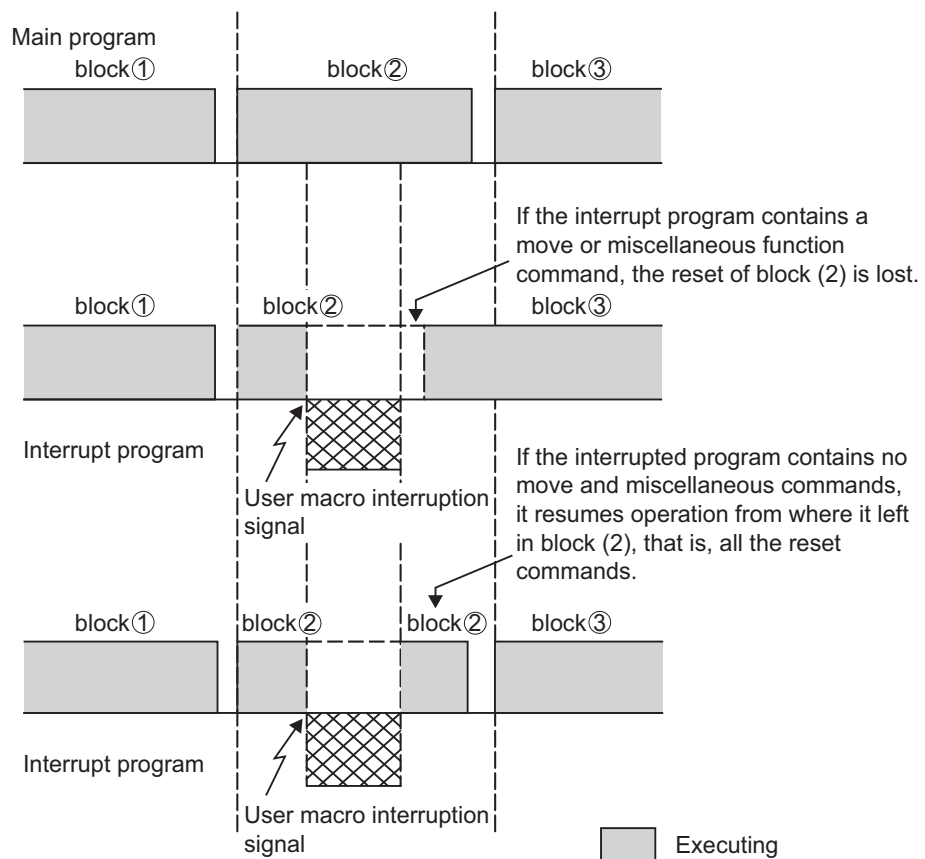
If an interrupt signal (UIT) is input during execution of a miscellaneous function (MSTB) command, the NC system waits for a completion signal (FIN). The system thus executes a move or miscellaneous function command (MSTB) in the interrupt program only after input of FIN.

[Type 2]

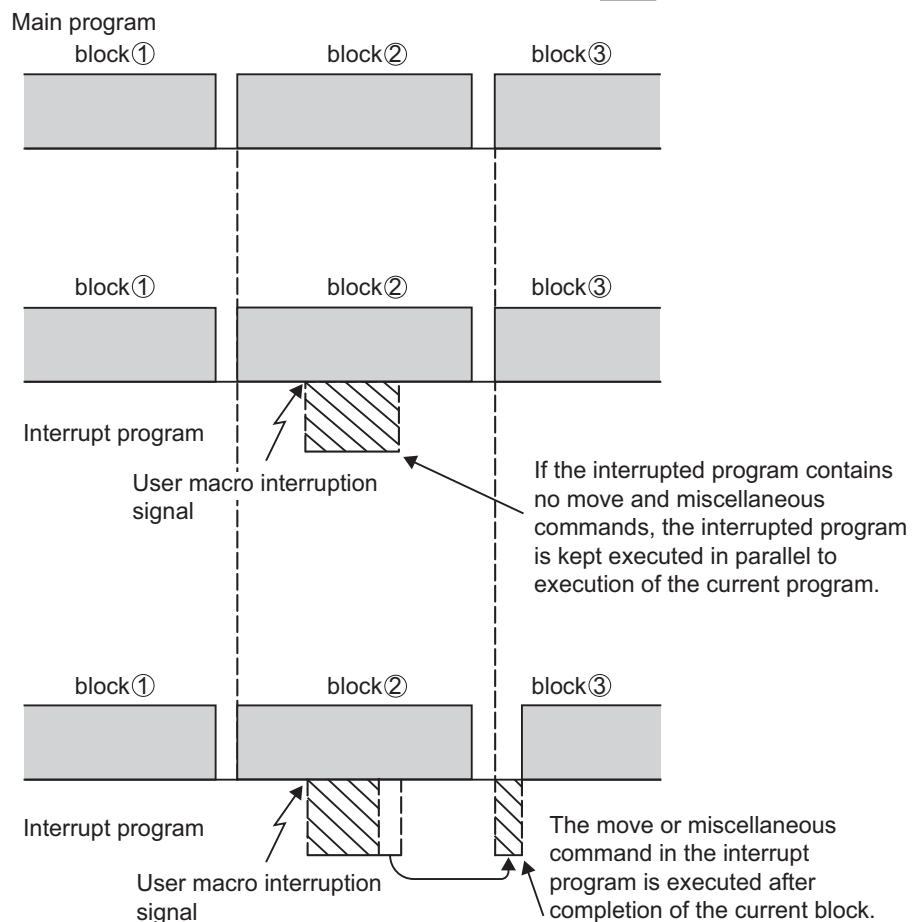
- (1) When an interrupt signal (UIT) is input, the interrupt program is executed after the commands in the block executed at that time have been completed. Even if the interrupt program contains a move or miscellaneous function (MSTB) command, the same processing is performed.
- (2) If the interrupt program contains no move and miscellaneous function (MSTB) commands, the interrupt program is executed without interrupting execution of the current block.

However, if the interrupt program has not ended even after the execution of the original block is completed, the system may stop machining temporarily.

[Type 1]



[Type 2]



Calling method

User macro interruption is classified into the following two types depending on the way an interrupt program is called. These two types of interrupt are selected by parameter "#8155 Sub-pro interrupt". This setting also involves the MTB settings (parameter "#1229 set01/bit0").

Both types of interrupt are added to the calculation of the nest level. The subprograms and user macros called in the interrupt program are also added to the calculation of the nest level.

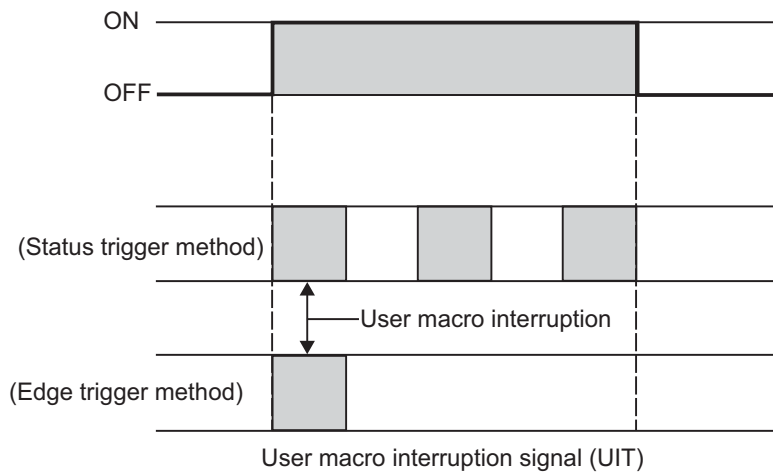
Subprogram type interrupt	The user macro interruption program is called as a subprogram. As with calling by M98, the local variable level remains unchanged before and after an interrupt.
Macro type interruption	The user macro interpretation program is called as a user macro. As with calling by G65, the local variable level changes before and after an interrupt. No arguments in the main program can be passed to the interrupt program.

Acceptance of user macro interruption signal (UIT)

A user macro interruption signal (UIT) is accepted in the following two methods. These two methods are selected by a parameter "#1112 S_TRG".

Status trigger method	The user macro interruption signal (UIT) is accepted as valid when it is ON. If the interrupt signal (UIT) is ON when the user macro interrupt function is enabled by M96, the interrupt program is activated. By keeping the interrupt signal (UIT) ON, the interrupt program can be executed repeatedly.
Edge trigger method	The user macro interrupt signal (UIT) is accepted as valid at its rising edge, that is, at the instance it turns ON. This mode is useful to execute an interrupt program once.

User macro interruption signal (UIT)



Returning from user macro interruption

M99 (P__);

An M99 command is issued in the interrupt program to return to the main program.

Address P is used to specify the sequence number of the return destination in the main program.

The blocks from the one next to the interrupted block to the last one in the main program are first searched for the block with designated sequence No. If it is not found, all the blocks before the interrupted one are then searched.

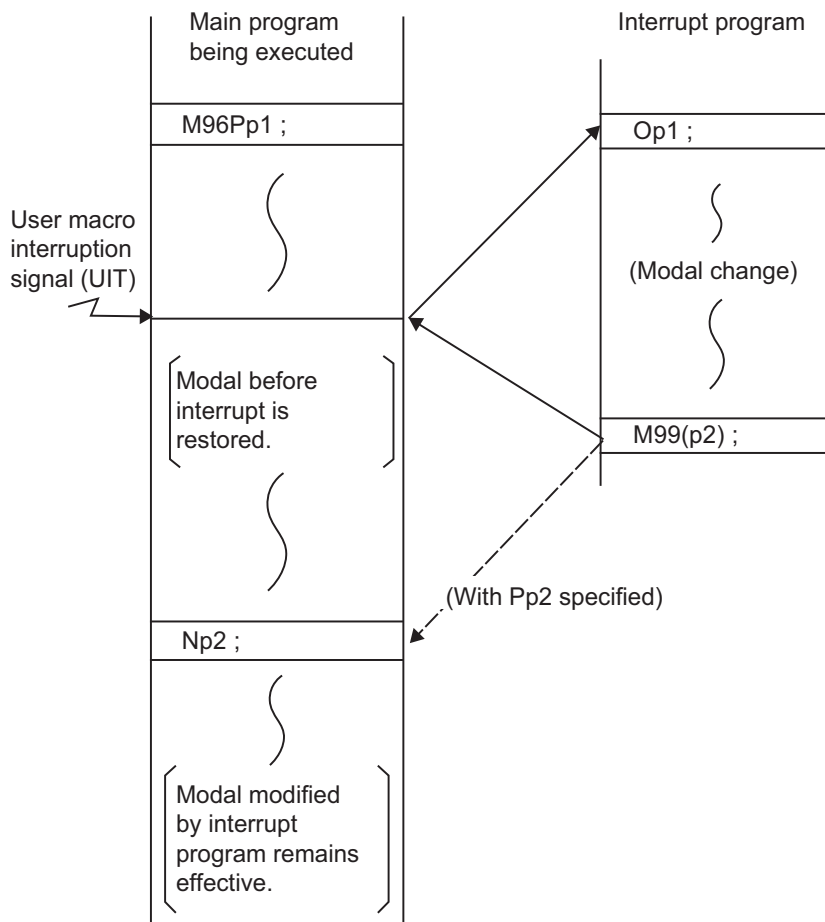
Control thus returns to the block with sequence No. that is found first in the above search.

(This is equivalent to M99P__ used after M98 calling.)

Modal information affected by user macro interruption

If modal information is changed by the interrupt program, it is handled as follows after control returns from the interrupt program to the main program.

Returning with M99;	The change of modal information by the interrupt program is invalidated and the original modal information is restored. With interrupt type 1, however, if the interrupt program contains a move command, miscellaneous function (MSTB) command, or specific command (*1), the original modal information is not restored.
Returning with M99P__ ;	The original modal information is updated by the change in the interrupt program even after returning to the main program. This is the same as in returning with M99P__ ; from a program called by M98, etc.



Modal information affected by user macro interruption

(*1) When interrupt type 1 is applied, modal information is not restored for the commands shown below after control has been returned from the interrupt program.

Command		Function
Machining center system	Lathe system	
G04	G04	Dwell
G11	G11	Data input by program cancel Tool life management data registration cancel
G27	G27	Reference position check
G92	G92	Spindle clamp speed setting
G92.1	G92.1	Workpiece coordinate system preset
-	G110	Mixed control (cross control) I
-	G111	Axis name switch
-	G113	Spindle synchronization I cancel Tool spindle synchronization IA (spindle - spindle synchronization) cancel Tool spindle synchronization II (hobbing) cancel
-	G114.1	Spindle synchronization I
-	G114.2	Tool spindle synchronization IA (Spindle - spindle synchronization)
-	G114.3	Tool spindle synchronization II (Hobbing)
G115	G115	Start point designation timing synchronization Type 1
G116	G116	Start point designation timing synchronization Type 2
G120.1	G120.1	Machining condition selection I
G121	G121	Machining condition selection I cancel
-	G125	Control axis synchronization between part systems
-	G126	Control axis superimposition
G127	G127	All part system reverse run prohibit command
!	!	Timing synchronization (! code)

The lathe-system commands are shown based on the G code system 3.

Hyphen "-" indicates that no specification is provided.

Modal information variables (#4401 to #4520)

Modal information when control passes to the user macro interruption program can be known by reading system variables #4401 to #4520.

The unit specified with a command applies.

System variable	Modal information	
#4401 : #4421	G code (group 01) : G code (group21)	Some groups are not used.
#4507	D code	
#4509	F code	
#4511	H code	
#4513	M code	
#4514	Sequence No.	
#4515	Program No. (*1)	
#4519	S code	
#4520	T code	

The above system variables are available only in the user macro interrupt program.

If they are used in other programs, program error (P241) will occur.

(*1) Programs are registered as files. When the program No. (file name) is read with #4515, the character string will be converted to a value.

(Example 1)

The file name "123" is the character string 0×31, 0×32, 0×33,

so the value will be $(0 \times 31 - 0 \times 30) \times 100 + (0 \times 32 - 0 \times 30) \times 10 + (0 \times 33 - 0 \times 30) = 123.0$.

Note that if the file name contains characters other than numbers, it will be "blank".

(Example 2)

If the file name is "123ABC", it contains characters other than numbers, so the result will be "blank".

M code for control of user macro interruption

The user macro interruption is controlled by M96 and M97. However, these commands may have been used for other operations. To be prepared for such cases, these command functions can be assigned to other M codes. (This invalidates program compatibility.)

User macro interrupt control with substitute M codes is possible by setting the substitute M code in parameters "#1110 M96_M" and "#1111 M97_M" and by validating the setting by selecting parameter "#1109 subs_M".

If the parameter "#1109 subs_M" used to enable the substitute M codes is not selected, the M96 and M97 codes remain effective for user macro interrupt control.

In either case, the M codes for user macro interrupt control are processed internally and not output to the outside.

Parameters

- (1) Subprogram call validity "#8155 Sub-pro interrupt" ("#1229 set01/bit0" (*1))
 - 1: Subprogram type user macro interruption
 - 0: Macro type user macro interruption
 - (2) Status trigger mode validity "#1112 S_TRG" (*1)
 - 1: Status trigger mode
 - 0: Edge trigger mode
 - (3) Interrupt type 2 validity "#1113 INT_2" (*1)
 - 1: The executable statements in the interrupt program are executed after completion of execution of the current block. (Type 2)
 - 0: The executable statements in the interrupt program are executed before completion of execution of the current block. (Type 1)
 - (4) Validity of substitute M code for user macro interruption control "#1109 subs_M" (*1)
 - 1: Valid
 - 0: Invalid
 - (5) Substitute M codes for user macro interruption control (*1)
 - Interrupt enable M code (equivalent to M96) "#1110 M96_M"
 - Interrupt disable M code (equivalent to M97) "#1111 M97_M"
 - Specify "03" to "97", excluding "30".
 - (6) Subprogram number selection "#8129 Subpro No. select"
 - Select a subprogram number to be called preferentially under subprogram control.
 - 0: Commanded program number
 - 1: 4-digits program number beginning with O No.
 - 2: 8-digits program number beginning with O No.
- (*1) These parameter settings depend on the MTB specifications.

Note

- ♦The program called by the user macro, figure rotation, macro interruption, or compound type fixed cycle also conforms to this setting.

**Precautions**

- (1) If the user macro interruption program uses system variables #5001 and after (position information) to read coordinates, the coordinates pre-read in the buffer are used.
- (2) If an interrupt is caused during execution of the tool nose R compensation or tool radius compensation, a sequence No. (M99P__;) must be specified with a command to return from the user macro interrupt program. If no sequence No. is specified, control cannot return to the main program normally.
- (3) With interrupt type 1, when the interrupt program contains a move or MSTB command, do not command the macro interruption to the waiting part system among multiple part systems. Doing so stops machining while the part system that does not perform an interruption remains set in the waiting standby state. If an interruption is carried out, machining can be started by the "ignore the timing synchronization between part systems" signal; however, the operation of this signal depends on the MTB specifications.

Program Support Functions

15.1 Corner Chamfering I/Corner Rounding I



Function and purpose

Chamfering at any angle or corner rounding is performed automatically by adding ",C_" or ",R_" to the end of the block to be commanded first among those command blocks which shape the corner with lines only.

15.1.1 Corner Chamfering I ; G01 X_ Y_ ,C



Function and purpose

This chamfers a corner by connecting the both side of the hypothetical corner which would appear as if chamfering is not performed, by the amount commanded by ",C_".



Command format

```
N100 G01 X__ Y__ ,C__ ;
N200 G01 X__ Y__ ;
```

,C	Length up to chamfering starting point or end point from hypothetical corner
----	--

Corner chamfering is performed at the point where N100 and N200 intersect.



Detailed description

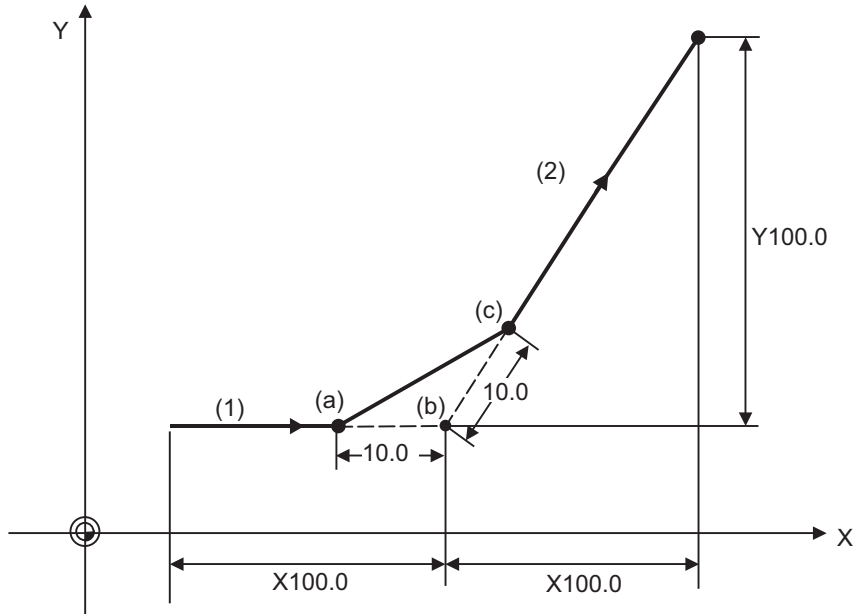
- (1) The start point of the block following the corner chamfering is the hypothetical corner intersection point.
- (2) If there are multiple or duplicate corner chamfering commands in a same block, the last command will be valid.
- (3) When both the corner chamfer and corner rounding commands exist in the same block, the latter command is valid.
- (4) Tool compensation is calculated for the shape which has already been subjected to corner chamfering.
- (5) When the block following a command with corner chamfering does not contain a linear command, a corner chamfering/corner rounding II command will be executed.
- (6) Program error (P383) will occur when the movement amount in the corner chamfering block is less than the chamfering amount.
- (7) Program error (P384) will occur when the movement amount in the block following the corner chamfering block is less than the chamfering amount.
- (8) Program error (P382) will occur when a movement command is not issued in the block following the corner chamfering I command.



Program example

```
(1) G91 G01 X100. ,C10.;
```

```
(2) X100. Y100.;
```



- (a) Chamfering start point
- (b) Hypothetical corner intersection point
- (c) Chamfering end point

15.1.2 Corner Rounding I ; G01 X_ Y_ ,R_

**Function and purpose**

The hypothetical corner, which would exist if the corner were not to be rounded, is rounded with an arc that has a radius commanded by ",R_" only when configured of linear lines.

**Command format**

```
N100 G01 X_ Y_ ,R_ ;
N200 G01 X_ Y_ ;
```

,R	Arc radius of corner rounding
----	-------------------------------

Corner rounding is performed at the point where N100 and N200 intersect.

**Detailed description**

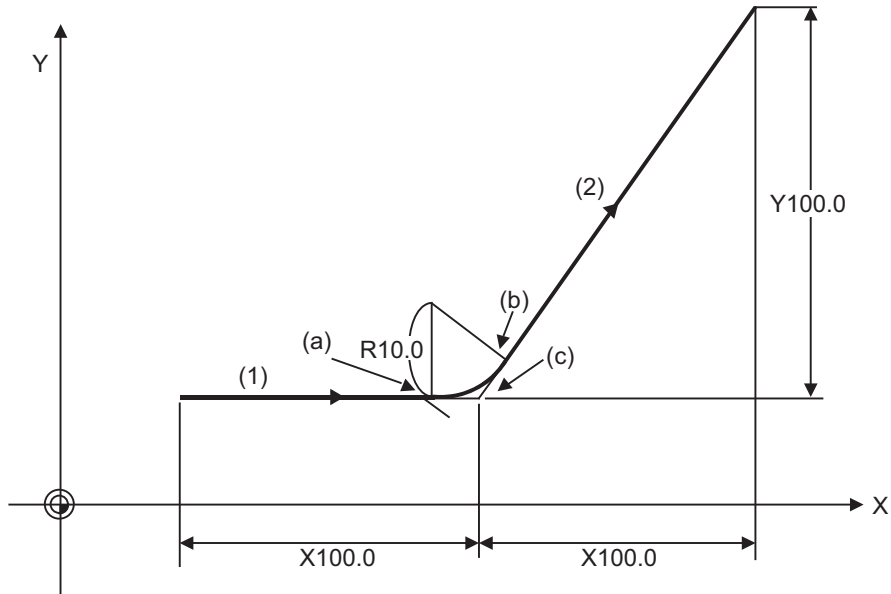
- (1) The start point of the block following the corner rounding is the hypothetical corner intersection point.
- (2) When both corner chamfering and corner rounding are commanded in the same block, the latter command will be valid.
- (3) Tool compensation is calculated for the shape which has already been subjected to corner rounding.
- (4) When the block following a command with corner rounding does not contain a linear command, a corner chamfering/corner rounding II command will be executed.
- (5) Program error (P383) will occur when the movement amount in the corner rounding block is less than the R value.
- (6) Program error (P384) will occur when the movement amount in the block following the corner rounding block is less than the R value.
- (7) Program error (P382) will occur if a movement command is not issued in the block following the corner rounding.



Program example

```
(1) G91 G01 X100. ,R10.;
```

```
(2) X100. Y100.;
```



(a) Corner rounding start point

(b) Corner rounding end point

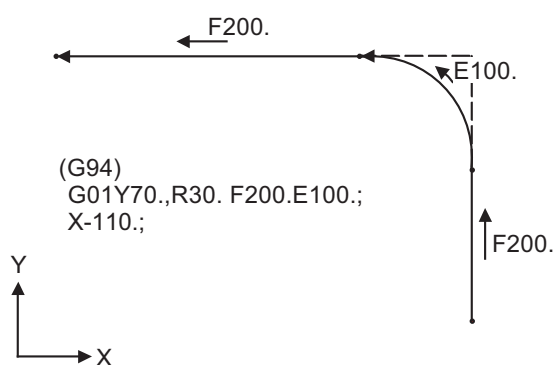
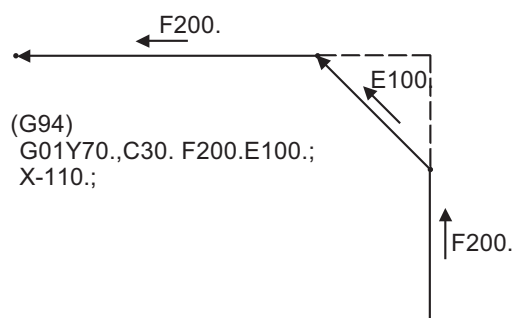
(c) Hypothetical corner intersection point

15.1.3 Corner Chamfering Expansion/Corner Rounding Expansion

**Function and purpose**

Using an E command, the feedrate can be designated for the corner chamfering and corner rounding section. In this way, the corner section can be cut into a correct shape.

Example

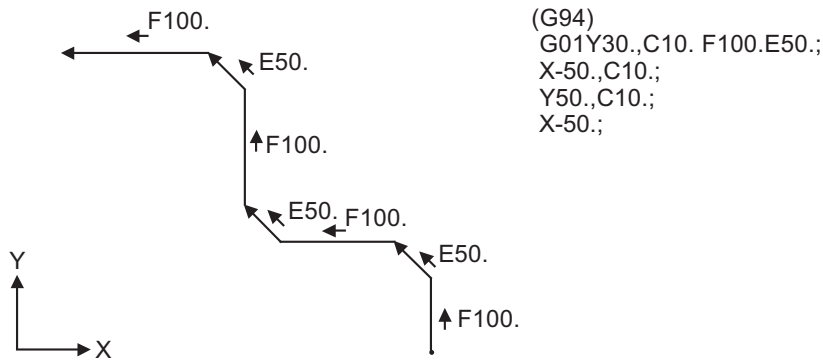




Detailed description

- (1) The E command is modal. It is also valid for the feed in the next corner chamfering/corner rounding section.

Example

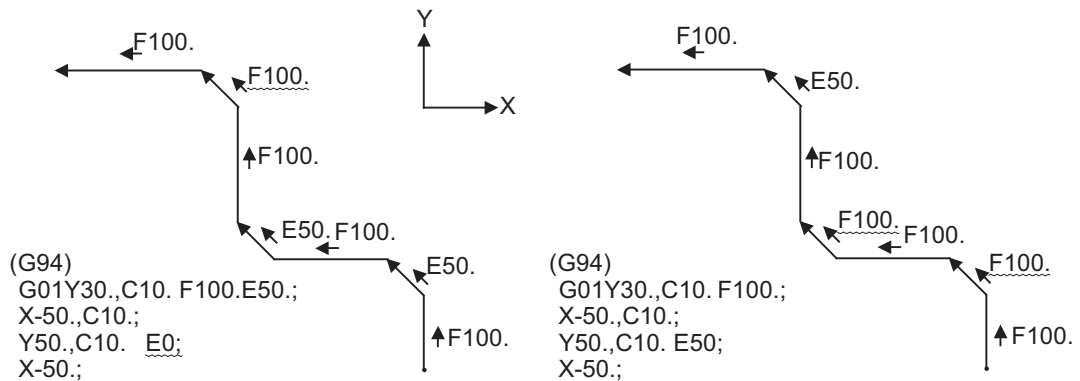


- (2) E command modal has separate asynchronous feedrate modal and synchronous feedrate modal functions.

Which one is validated depends on the asynchronous/synchronous mode (G94/G95).

- (3) When the E command is 0, or when there has not been an E command up to now, the corner chamfering/corner rounding section feedrate will be the same as the F command feedrate.

Example



- (4) E command modal is not cleared even if the reset button is pressed.

It is cleared when the power is turned OFF. (In the same manner as F commands.)

- (5) All E commands except those shown below are at the corner chamfering/corner rounding section feedrate.

- E commands during thread cutting modal
- E commands during thread cutting cycle modal

15.1.4 Interrupt during Corner Chamfering/Interrupt during Corner Rounding

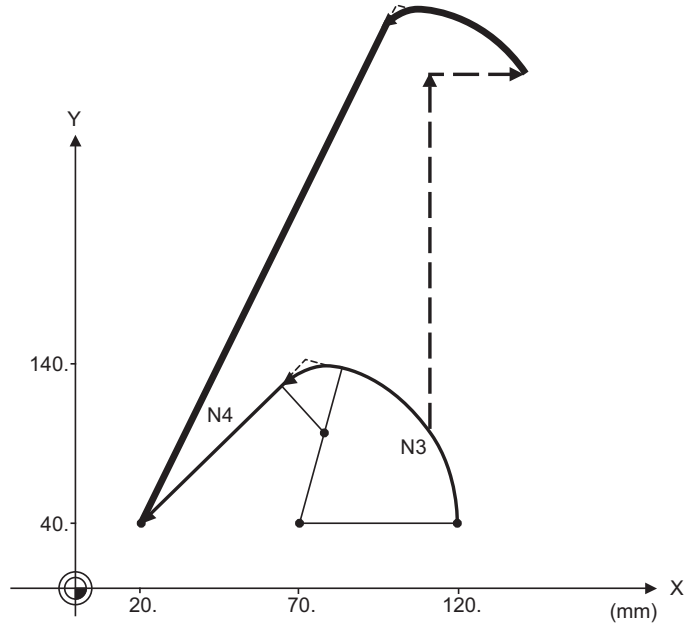


Detailed description

(1) Shown below are the operations of manual interruption during corner chamfering or corner rounding.

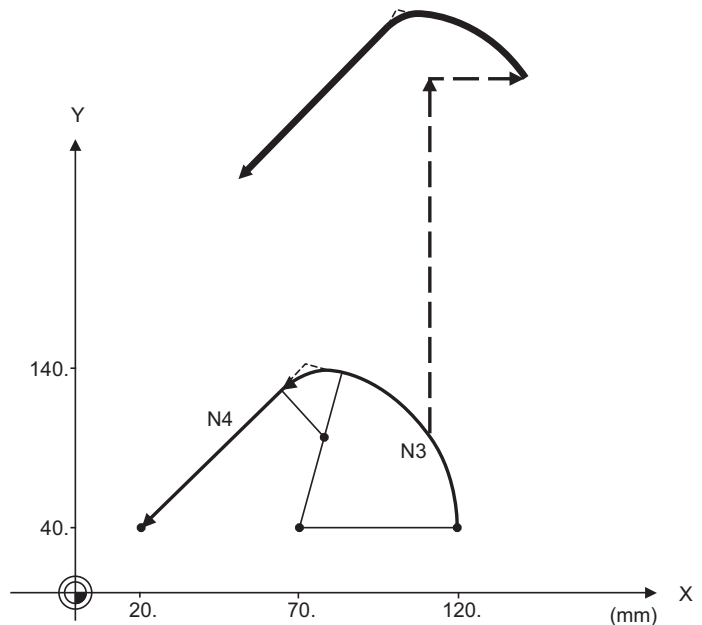
With an absolute value command and manual absolute switch ON.

```
N1 G28 XY;
N2 G00 X120.Y20. ;
N3 G03 X70. Y70.I-50. ,R20. F100 ;
N4 G01 X20. Y20. ;
```



With an incremental value command and manual absolute switch OFF

```
N1 G28 XY;
N2 G00 X120. Y20. ;
N3 G03 X-50. Y50. I-50. ,R20. F100 ;
N4 G01 X-50. Y-50.;
```



- Interrupt amount
- Path in interrupt case
- Path in non-interrupt case

(2)With a single block during corner chamfering or corner rounding, the tool stops after these operations are executed.

15.2 Corner Chamfering II/Corner Rounding II



Function and purpose

Corner chamfering and corner rounding can be performed by adding ",C" or ",R" to the end of the block which is commanded first among the block that forms a corner with continuous arbitrary angle lines or arcs.

15.2.1 Corner Chamfering II ; G01/G02/G03 X_ Y_ ,C_



Function and purpose

The corner is chamfered by commanding ",C" in the 1st block of the two continuous blocks containing an arc. For an arc, this will be the chord length.



Command format

```
N100 G03 X__ Y__ I__ J__ ,C__ ;
N200 G01 X__ Y__ ;
```

,C	Length up to chamfering starting point or end point from hypothetical corner
----	--

Corner chamfering is performed at the point where N100 and N200 intersect.



Detailed description

- (1) If this function is commanded while the corner chamfer or corner rounding command is not defined in the specifications, it causes a program error (P381).
- (2) The start point of the block following the corner chamfering is the hypothetical corner intersection point.
- (3) If there are multiple or duplicate corner chamfering commands in a same block, the last command will be valid.
- (4) When both corner chamfering and corner rounding are commanded in the same block, the latter command will be valid.
- (5) Tool compensation is calculated for the shape which has already been subjected to corner chamfering.
- (6) Program error (P385) will occur when positioning or thread cutting is commanded in the corner chamfering command block or in the next block.
- (7) Program error (P382) will occur when the block following corner chamfering contains a G command other than group 01 or another command.
- (8) Program error (P383) will occur when the movement amount in the block, commanding corner chamfering, is less than the chamfering amount.
- (9) Program error (P384) will occur when the movement amount is less than the chamfering amount in the block following the block commanding corner chamfering.
- (10) Even if a diameter is commanded, it will be handled as a radial command value during corner chamfering.
- (11) Program error (P382) will occur when a movement command is not issued in the block following the corner chamfering II command.

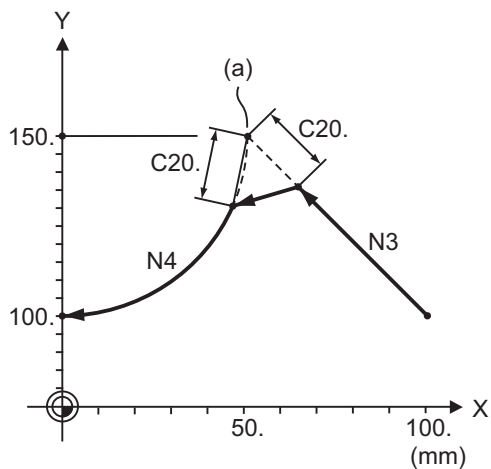


Program example

(1) Linear - arc

Absolute value command
 N1 G28 XY;
 N2 G90 G00 X100. Y100.;
 N3 G01 X50. Y150., C20. F100;
 N4 G02 X0 Y100. I-50. J0;
 :

Relative value command
 N1 G28 XY;
 N2 G91 G00 X100. Y100.;
 N3 G01 X-50. Y50., C20. F100;
 N4 G02 X-50. Y-50. I-50. J0;
 :

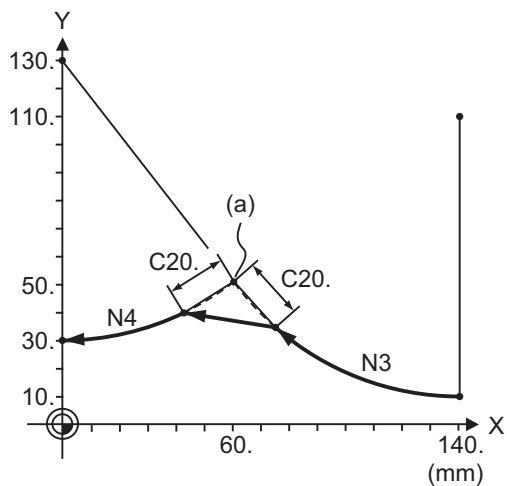


(a) Hypothetical corner intersection point

(2) Arc - arc

Absolute value command
 N1 G28 XY;
 N2 G91 G00 X140. Y10.;
 N3 G02 X60. Y50. I0 J100. , C20. F100;
 N4 X0 Y30. I-60. J80.;
 :

Relative value command
 N1 G28 XY;
 N2 G91 G00 X140. Y10.;
 N3 G02 X-80. Y40. R100. , C20. F100;
 N4 X-60. Y-20. I-60. J80.;
 :



(a) Hypothetical corner intersection point

15.2.2 Corner Rounding II ; G01/G02/G03 X_ Y_ ,R_

**Function and purpose**

The corner is rounded by commanding ",R_" in the 1st block of the two continuous blocks containing an arc.

**Command format**

```
N100 G03 X_ Y_ I_ J_ ,R_ ;
N200 G01 X_ Y_ ;
```

,R	Arc radius of corner rounding
----	-------------------------------

Corner rounding is performed at the point where N100 and N200 intersect.

**Detailed description**

- (1) If this function is commanded while the corner chamfer or corner rounding command is not defined in the specifications, it causes a program error (P381).
- (2) The start point of the block following the corner rounding is the hypothetical corner intersection point.
- (3) When both corner chamfering and corner rounding are commanded in a same block, the latter command will be valid.
- (4) Tool compensation is calculated for the shape which has already been subjected to corner rounding.
- (5) Program error (P385) will occur when positioning or thread cutting is commanded in the corner rounding command block or in the next block.
- (6) Program error (P382) will occur when the block following corner rounding contains a G command other than group 01 or another command.
- (7) Program error (P383) will occur when the movement amount in the corner rounding block is less than the R value.
- (8) Program error (P384) will occur when the movement amount is less than the R value in the block following the corner rounding.
- (9) Even if a diameter is commanded, it will be handled as a radial command value during corner rounding.
- (10) A program error (P382) will occur if a movement command is not issued in the block following corner rounding.

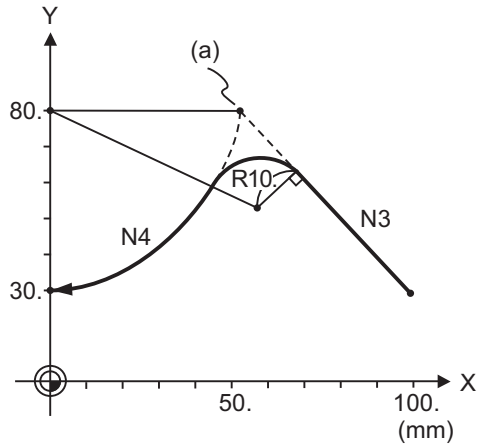


Program example

(1) Linear - arc

Absolute value command
 N1 G28 XY;
 N2 G90 G00 X100. Y30.;
 N3 G01 X50.Y80.,R10. F100;
 N4 G02 X0 Y30. I-50.J0;
 :

Relative value command
 N1 G28 XY;
 N2 G91 G00 X100. Y30.;
 N3 G01 X-50.Y50.,R10. F100;
 N4 G02 X-50. Y-50. I-50.J0;
 :

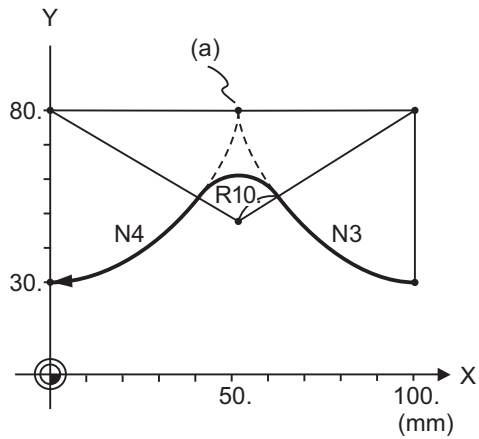


(a) Hypothetical corner intersection point

(2) Arc - arc

Absolute value command
 N1 G28 XY;
 N2 G90 G00 X100. Y30.;
 N3 G02 X50.Y80. R50.,R10.F100;
 N4 X0 Y30. R50.;
 :

Relative value command
 N1 G28 XY;
 N2 G91 G00 X100. Y30.;
 N3 G02 X-50.Y50. I0 J50.,R10.F100;
 N4 X-50. Y-50. I-50. J0;
 :



(a) Hypothetical corner intersection point

15.2.3 Corner Chamfering Expansion/Corner Rounding Expansion

For details, refer to "Corner Chamfering I / Corner Rounding" and "Corner Chamfering Expansion / Corner Rounding Expansion".

15.2.4 Interrupt during Corner Chamfering/Interrupt during Corner Rounding

For details, refer to "Corner Chamfering I / Corner Rounding" and "Interrupt during Corner Chamfering Interrupt during / Corner Rounding".

15.3 Linear Angle Command; G01 X_/Y_ A_/A_



Function and purpose

The end point coordinates are automatically calculated by commanding the linear angle and one of the end point coordinate axes.

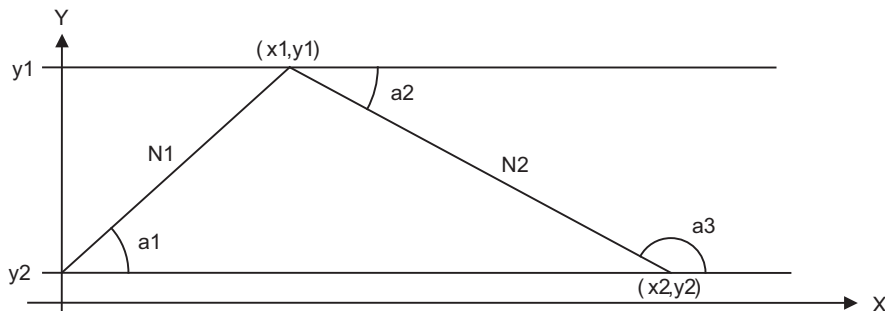


Command format

```
N1 G01 Xx1(Yy1) Aa1;
N2 G01 Xx2(Yy2) A-a2; (A-a2 can also be set as Aa 3.)
```

```
N1 G01 Xx1(Yy1) ,Aa1;
N2 G01 Xx2(Yy2) ,A-a2;
```

This designates the angle and the X or Y axis coordinates.
Select the command plane with G17 to G19.



Detailed description

- (1) The angle is set based on the positive (+) direction of the horizontal axis for the selected plane. The counter-clockwise (CCW) direction is indicated by a positive sign (+), and the clockwise (CW) direction by a negative sign (-).
- (2) Either of the axes on the selected plane is commanded for the end point.
- (3) The angle is ignored when the angle and the coordinates of both axes are commanded.
- (4) When only the angle has been commanded, this is treated as a geometric command.
- (5) The angle of either the start point (a1) or end point (a2) may be used.
- (6) This function is valid only for the G01 command; it is not valid for other interpolation or positioning commands.
- (7) The range of slope "a" is between -360.000 and 360.000.
When a value outside this range is commanded, it will be divided by 360 (degrees) and the remainder will be commanded.
(Example) If 400 is commanded, 40° (remainder of 400/360) will become the command angle.
- (8) If an address A is used for the axis name or the 2nd miscellaneous function, use ",A" as the angle.
- (9) If "A" and ",A" are commanded in a same block, ",A" will be interpreted as the angle.

Note

- A program error (P33) will occur if this function is commanded during the high-speed machining mode or high-speed high-accuracy mode.

15.4 Geometric; G01 A_



Function and purpose

When it is difficult to calculate the intersection point of two straight lines in a continuous linear interpolation command, the end point of the first straight line will be automatically calculated inside the CNC and the movement command will be controlled, provided that the slope of the first straight line as well as the end point coordinates and slope of the second straight line are commanded.

Note

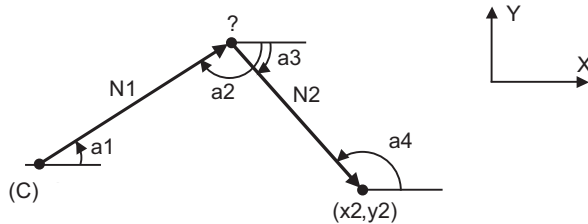
(1) If the parameter (#1082 Geomet) is set to 0, geometric I will not function.



Command format

```
N1 G01 Aa1 (A-a2) Ff1;
N2 Xx2 Yy2 A-a4 (A-a3) Ff2;
```

Aa1, A-a2, A-a3, Aa4	Angle
Ff1, Ff2	Speed
Xx2, Yy2	Next block end point coordinates



(C) Current position



Detailed description

- (1) Program error (P396) will occur when the geometric command is not on the selected plane.
- (2) The slope indicates the angle to the positive (+) direction of the horizontal axis for the selected plane. The counterclockwise (CCW) direction is indicated by a positive sign (+), and the clockwise (CW) direction by a negative sign (-).
- (3) The range of slope "a" is between -360.000 and 360.000.
When a value outside this range is commanded, it will be divided by 360 (degrees) and the remainder will be commanded.
(Example) If 400 is commanded, 40° (remainder of 400/360) will become the command angle.
- (4) The slope of the line can be commanded on either the start or end point side. Whether designated slope is the starting point or the end point will be automatically identified in NC.
- (5) The end point coordinates of the second block should be commanded with absolute values. If incremental values are used, program error (P393) will occur.
- (6) The feedrate can be commanded for each block.
- (7) When the angle where the two straight lines intersect is less than 1°, program error (P392) will occur.
- (8) Program error (P396) will occur when the plane is changed in the 1st block and 2nd block.
- (9) This function is ignored when address A is used for the axis name or as the 2nd miscellaneous function.
- (10) Single block stop is possible at the end point of the 1st block.
- (11) Program error (P394) will occur when the 1st and 2nd blocks do not contain the G01 or G33 command.



Relationship with Other Functions

(1) Corner chamfering and corner rounding can be commanded after the angle command in the 1st block.

<p>(Example 1) N1 Aa1 ,Cc1 ; N2 Xx2 Yy2 Aa2 ;</p>	
<p>(Example 2) N1 Aa1 ,Rr1 ; N2 Xx2 Yy2 Aa2 ;</p>	

(2) The geometric command I can be issued after the corner chamfering or corner rounding command.

<p>(Example 3) N1 Xx2 Yy2 ,Cc1 ; N2 Aa1 ; N3 Xx3 Yy3 Aa2 ;</p>	
---	--

(3) The geometric command I can be issued after the linear angle command.

<p>(Example 4) N1 Xx2 Aa1 ; N2 Aa2 ; N3 Xx3 Yy3 Aa3 ;</p>	
--	--

15.5 Geometric IB



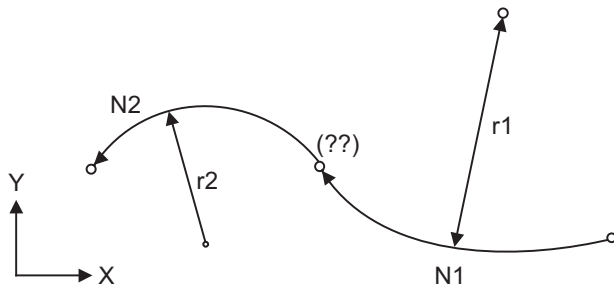
Function and purpose

With the geometric IB function, the contact and intersection are calculated by commanding an arc center point or linear angle in the movement commands of two continuous blocks (only blocks with arc commands), instead of commanding the first block end point.

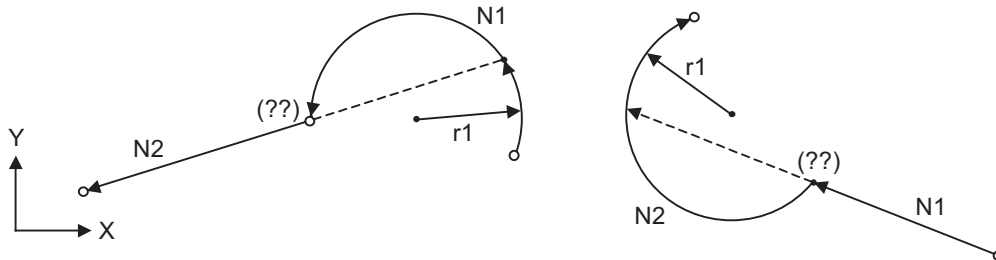
Note

(1) If the parameter (#1082 Geomet) is not set to 2, geometric IB will not function.

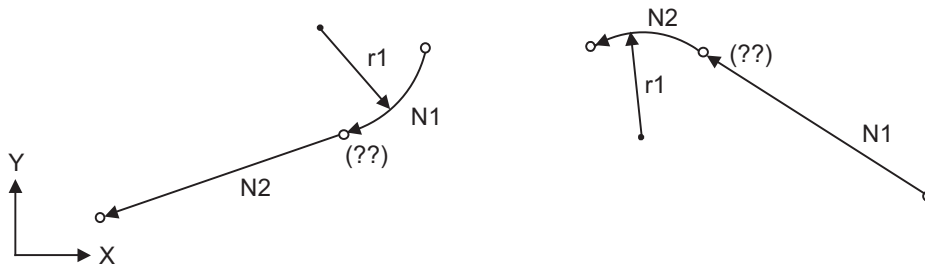
Two-arc contact



Linear - arc (arc - linear) intersection



Linear - arc (arc - linear) contact



15.5.1 Geometric IB (Automatic Calculation of Two-arc Contact) ; G02/G03 P_Q_/R_

**Function and purpose**

When the contact of two continuous contacting arcs is not indicated in the drawing, it can be automatically calculated by commanding the 1st circular center coordinate value or radius, and the 2nd arc end point absolute value and center coordinate value or radius.

**Command format**

```
N1 G02(G03) Pp1 Qq1 Ff1;
N2 G03(G02) Xx2 Yy2 Pp2 Qq2 Ff2;
```

```
N1 G02(G03) Pp1 Qq1 Ff1;
N2 G03(G02) Xx2 Yy2 Rr2 Ff2;
```

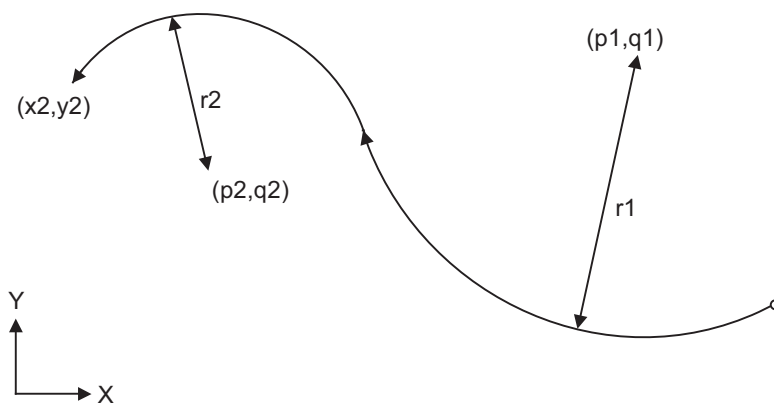
```
N1 G02(G03) Rr1 Ff1;
N2 G03(G02) Xx2 Yy2 Pp2 Qq2 Ff2;
```

P,Q	X and Y axes circular center coordinate absolute value (diameter/radius value command)The center address for the 3rd axis is commanded with A.
R	Arc radius (when a (-) sign is attached, the arc is judged to be 180° or more)

* I and J (X and Y axes arc center coordinate incremental value) commands can be issued instead of P and Q.

1st block arc : Incremental amount from the start point to the center

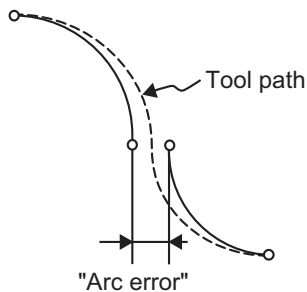
2nd block arc : Incremental amount from the end point to the center



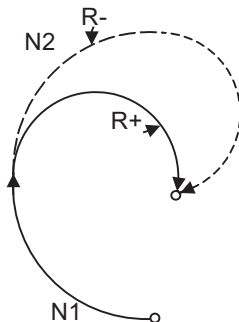


Detailed description

- (1) Program error (P393) will occur before the 1st block if the 2nd block is not a coordinate absolute value command.
- (2) Program error (P398) will occur before the 1st block if there is no geometric IB specification.
- (3) Program error (P395) will occur before the 1st block if there is no R (here, the 1st block is designated with P, Q (I, J)) or P, Q (I, J) designation in the 2nd block.
- (4) Program error (P396) will occur before the 1st block if another plane selection command (G17 to G19) is issued in the 2nd block.
- (5) Program error (P397) will occur before the 1st block if two arcs that do not contact are commanded.
- (6) The contact calculation accuracy is $\pm 1\mu\text{m}$ (fractions rounded up).
- (7) Single block operation stops at the 1st block.
- (8) When I or J is omitted, the values are regarded as I0 and J0. P and Q cannot be omitted.
- (9) The error range in which the contact is obtained is set in parameter "#1084 RadErr".



- (10) For an arc block perfect circle command (arc block start point = arc block end point), the R designation arc command finishes immediately, and there is no operation. Thus, use a PQ (IJ) designation arc command.
- (11) G codes of the G modal group 1 in the 1st/2nd block can be omitted.
- (12) Addresses being used as axis names cannot be used as command addresses for arc center coordinates or arc radius.
- (13) When the 2nd block arc inscribes the 1st block arc and the 2nd block is an R designation arc, the R+ sign becomes the inward turning arc command, and the R- sign becomes the outward turning arc command.



15.5.2 Geometric IB (Automatic Calculation of Linear - Arc Intersection) ; G01 A_ , G02/G03 P_Q_H_



Function and purpose

When the contact point of a shape in which contact between a line and an arc is not indicated in the drawing, it can be automatically calculated by commanding the following program.



Command format (For G18 plane)

```
N1 G01 Aa1 (A-a2) Ff1;
N2 G02(G03) Xx2 Yy2 Pp2 Qq2 Hh2 Ff2 ;
```

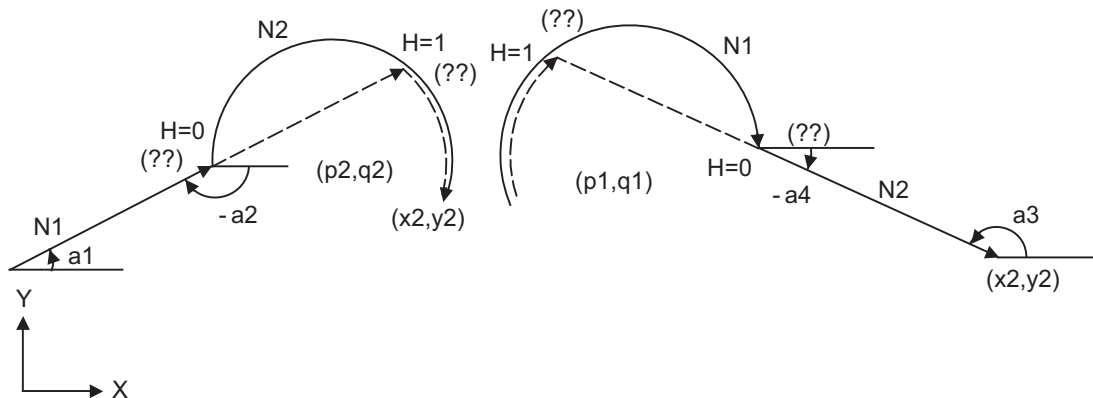
```
N1 G02(G03) Pp1 Qq1 Hh1 (,Hh1) Ff1 ;
N2 G1 Xx2 Yy2 Aa3 (A-a4) Ff2 ;
```

A	Linear angle (-360.000° to 360.000°)
P,Q	X and Y axes circular center coordinate absolute value (diameter/radius value command)The center address for the 3rd axis is commanded with A.
H (,H)	Selection of linear - arc intersection 0: Intersection of the shorter line 1: Intersection of the longer line

* I and J (X and Y axes arc center coordinate incremental value) commands can be issued instead of P and Q.

1st block arc : Incremental amount from the start point to the center

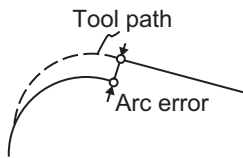
2nd block arc : Incremental amount from the end point to the center



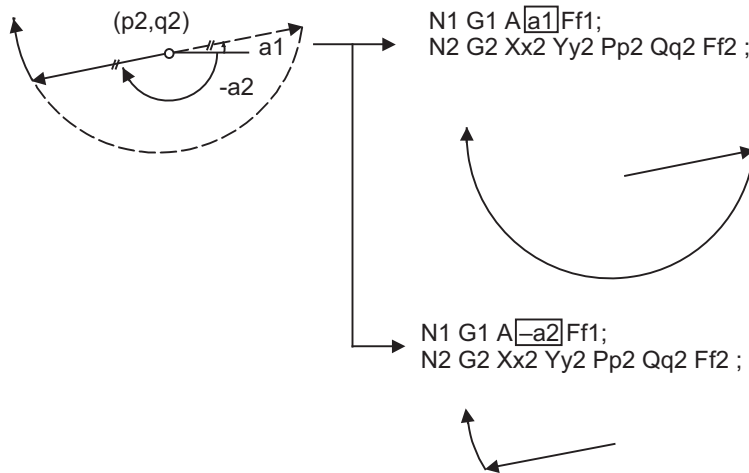


Detailed description

- (1) When the 2nd miscellaneous function address is A, the 2nd miscellaneous function is validated and this function is invalidated.
- (2) Program error (P393) will occur before the 1st block if the 2nd block is not a coordinate absolute value command.
- (3) Program error (P398) will occur before the 1st block if there is no geometric IB specification.
- (4) In case of the 2nd block arc, a program error (P395) will occur before the 1st block if there is no P, Q (I, J) designation. A program error (P395) will also occur if there is no A designation for the line.
- (5) Program error (P396) will occur before the 1st block if another plane selection command (G17 to G19) is issued in the 2nd block.
- (6) Program error (P397) will occur before the 1st block if a straight line and arc that do not contact or intersect are commanded.
- (7) Single block operation stops at the 1st block.
- (8) When I or J is omitted, the values are regarded as I0 and J0. P and Q cannot be omitted.
- (9) When H is omitted, the value is regarded as H0.
- (10) The linear - arc contact is automatically calculated by designating R instead of P, Q (I, J).
- (11) The error range in which the intersect is obtained is set in parameter "#1084 RadErr".



- (12) As seen from the + direction of the horizontal axis of the selected plane, the counterclockwise (CCW) direction is considered to be + and the clockwise direction (CW) -.
- (13) The slope of the line can be commanded on either the start or end point side. Whether designated slope is the starting point or the end point will be automatically identified.
- (14) When the distance to the intersection from the line and arc is same (as in the figure below), the control by address H (short/long distance selection) is invalidated. In this case, the judgment is carried out based on the angle of the line.

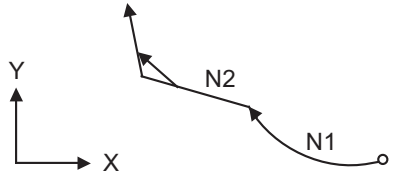


- (15) The intersect calculation accuracy is $\pm 1\mu\text{m}$ (fractions rounded up).
- (16) In linear - arc intersections, the arc command can only be PQ (IJ) command. When the arc block start point and arc block end point are the same point, the arc is a perfect circle.

- (17) G codes of the G modal group in the 1st block can be omitted.
- (18) Addresses being used as axis names cannot be used as command addresses for angles, arc center coordinates or intersection selections.
- (19) When geometric IB is commanded, two blocks are pre-read.



Relationship with other functions

Command	Tool path
Geometric IB + corner chamfering N1 G02 P_ Q_ H_ ; N2 G01 X_ Y_ A_ ,C_ ; G01 X_ Y_ ;	

15.5.3 Geometric IB (Automatic Calculation of Linear - Arc Intersection) ; G01 A_ , G02/G03 R_H_



Function and purpose

When the intersection of a shape in which a line and an arc intersect is not indicated in the drawing, it can be automatically calculated by commanding the following program.

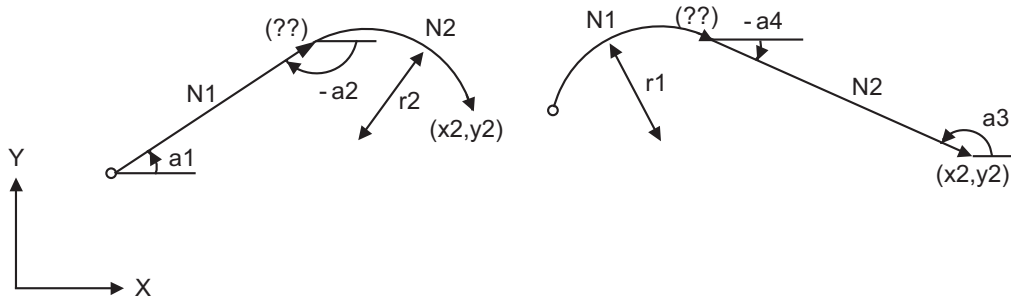


Command format (For G18 plane)

```
N1 G01 Aa1 (A-a2) Ff1;
N2 G03(G02) Xx2 Yy2 Rr2 Ff2;
```

```
N1 G03(G02) Rr1 Ff1;
N2 G01 Xx2 Yy2 Aa3 (A-a4) Ff2 ;
```

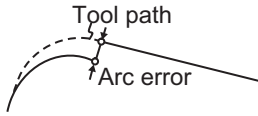
A	Linear angle (-360.000° to 360.000°)
R	Circular radius



Detailed description

- (1) When the 2nd miscellaneous function address is A, the 2nd miscellaneous function is validated and this function is invalidated.
- (2) Program error (P393) will occur before the 1st block if the 2nd block is not a coordinate absolute value command.
- (3) Program error (P398) will occur before the 1st block if there is no geometric IB specification.
- (4) Program error (P396) will occur before the 1st block if another plane selection command (G17 to G19) is issued in the 2nd block.
- (5) A program error (P397) will occur before the 1st block if a straight line and arc that do not contact are commanded.
- (6) In case of the 2nd block arc, a program error (P395) will occur before the 1st block if there is no R designation. A program error (P395) will also occur if there is no A designation for the line.
- (7) Single block operation stops at the 1st block.
- (8) The linear - arc contact is automatically calculated by designating R instead of P, Q (I, J).

(9) The error range in which the contact is obtained is set in parameter "#1084 RadErr".



(10) The line slope is the angle to the positive (+) direction of its horizontal axis. Counterclockwise (CCW) is positive (+). Clockwise (CW) is negative (-).

(11) The slope of the line can be commanded on either the start or end point side. Whether the commanded slope is on the start or end point side is identified automatically inside the NC unit.

(12) The intersect calculation accuracy is $\pm 1\mu\text{m}$ (fractions rounded up).

(13) In linear - arc contact, the arc command can only be an R command. Thus, when the arc block start point = arc block end point, the arc command finishes immediately, and there will be no operation. (Perfect circle command is impossible.)

(14) G codes of the G modal group 1 in the 1st block can be omitted.

(15) Addresses being used as axis names cannot be used as command addresses for angles or arc radius.

(16) When geometric IB is commanded, two blocks are pre-read.



Relationship with other functions

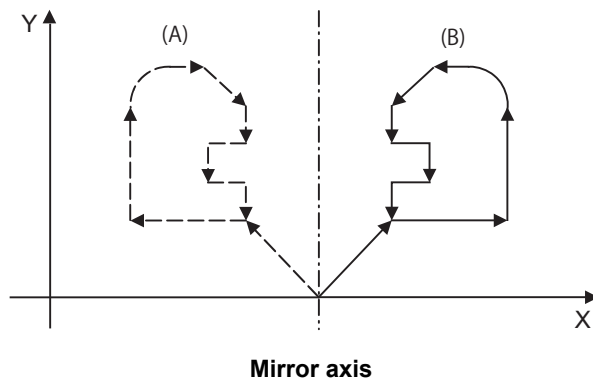
Command	Tool path
Geometric IB + corner chamfering N1 G03 R_ N2 G01 X_ Y_ A_ ,C_ G01 X_ Y_ 	
Geometric IB + corner rounding N1 G03 R_ N2 G01 X_ Y_ A_ ,R_ G01 X_ Y_ 	

15.6 Mirror Image by G code ; G50.1,G51.1



Function and purpose

When cutting a shape that is symmetrical on the left and right, programming time can be shortened by machining one side and then using the same program to machine the other side. The mirror image function is effective for this. For example, when using a program as shown below to machine the shape on the left side (A), a symmetrical shape (B) can be machined on the right side by applying mirror image and executing the program.



Command format

Mirror image ON

G51.1 Xx1 Yy1 Zz1;

x1, y1, z1

Mirror image center coordinates
(Mirror image will be applied regarding this position as a center)

Mirror image OFF

G50.1 Xx2 Yy2 Zz2;

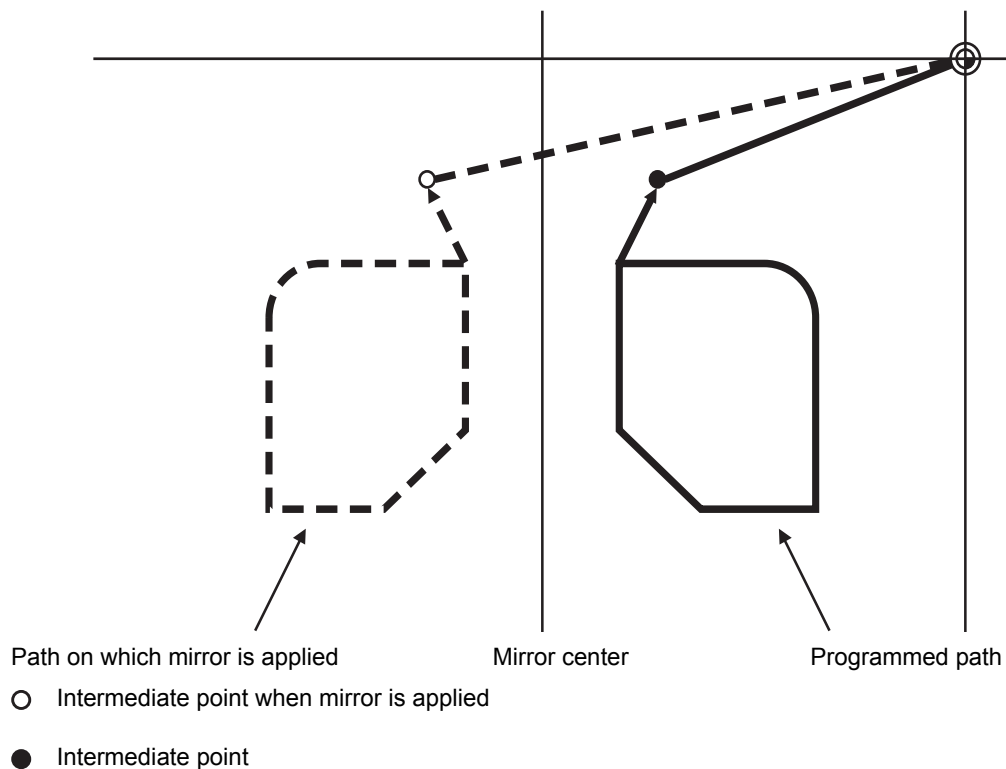
x2, y2, z2

Mirror image cancel axis
(The values of x2, y2, z2 will be ignored.)



Detailed description

- (1) At G51.1, command the mirror image axis and the coordinate to be a center of mirror image with the absolute command or incremental command.
- (2) At G50.1, command the axis for which mirror image is to be turned OFF. The values of x2, y2, and z2 will be ignored.
- (3) If mirror image is applied on only one axis of the designated plane, the rotation direction and compensation direction will be reversed for the arc or tool radius compensation and coordinate rotation, etc.
- (4) This function is processed on the local coordinate system, so the center of the mirror image will change when the counter is preset or when the workpiece coordinates are changed.
- (5) Reference position return during mirror image
If the reference position return command (G28, G30) is executed during the mirror image, the mirror image will be valid during the movement to the intermediate point, but will not be applied to the movement to the reference position after the intermediate point.



- (6) Return from zero point during mirror image
If the return command (G29) from the zero point is commanded during the mirror image, the mirror will be applied to the intermediate point.
- (7) The mirror image will not be applied to the G53 command.



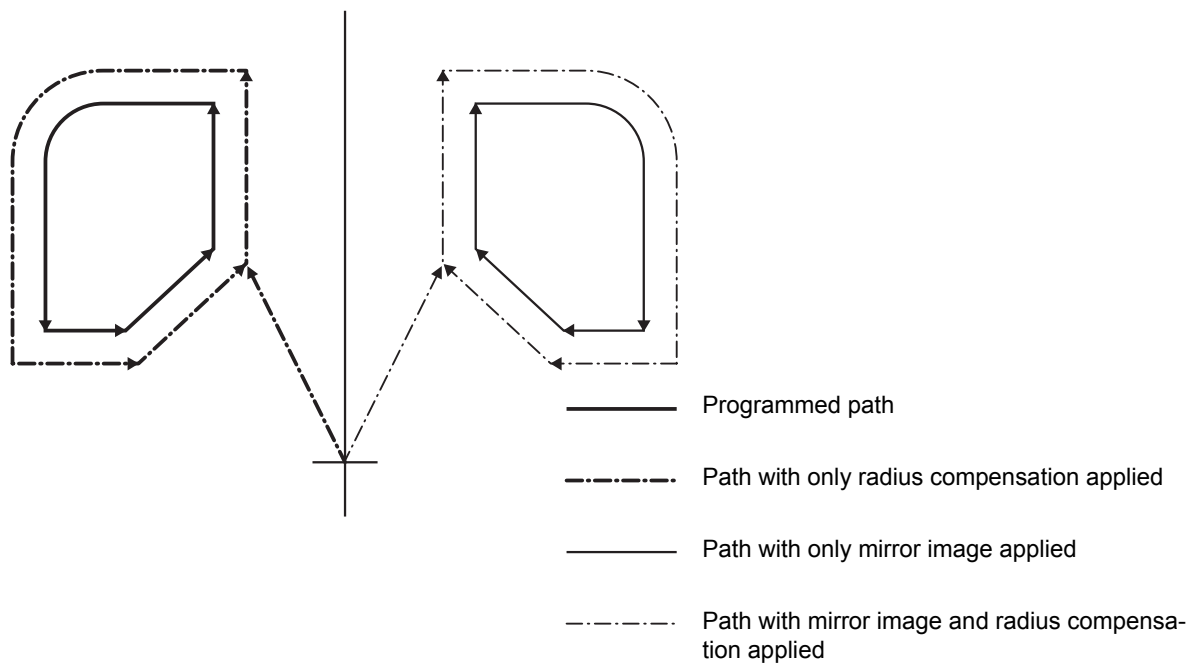
Relationship with other functions

(1) Combination with radius compensation

The mirror image (G51.1) will be processed after the radius compensation (G41, G42) is applied, so the following type of cutting will take place.

Programmed path

Path with mirror image applied





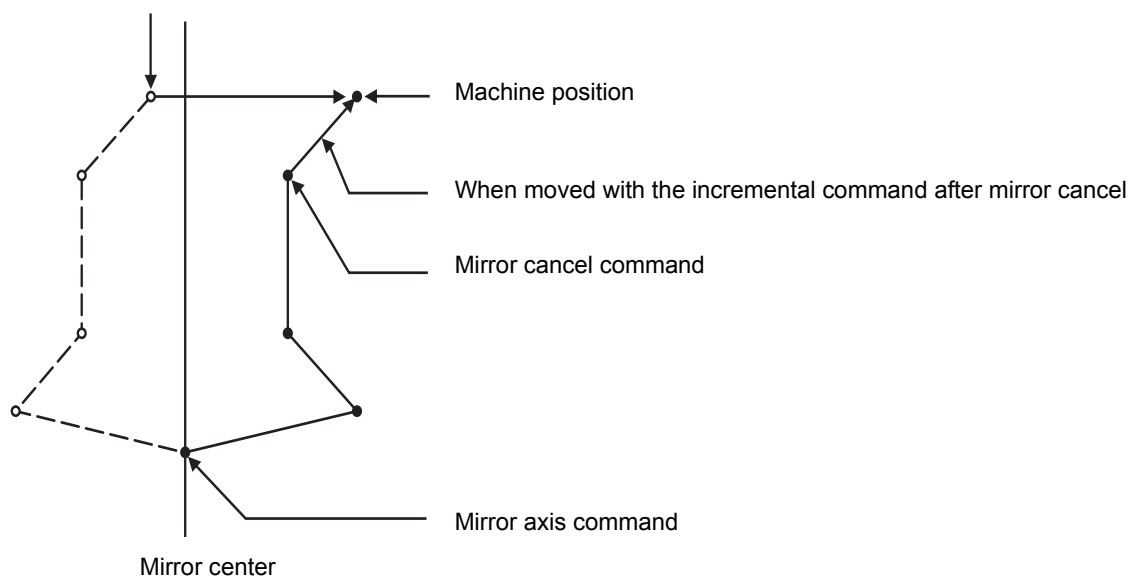
Precautions

CAUTION

Turn the mirror image ON and OFF at the mirror image center.

If mirror image is not canceled at the mirror center, the absolute value and machine position will deviate as shown below. (This state will last until an absolute value command (positioning with G90 mode) is issued, or a reference position return with G28 or G30 is executed.) The mirror center is set with an absolute value, so if the mirror center is commanded again in this state, the center may be set to an unpredictable position. Cancel the mirror at the mirror center or position with the absolute value command after canceling.

Absolute value (position commanded in program)



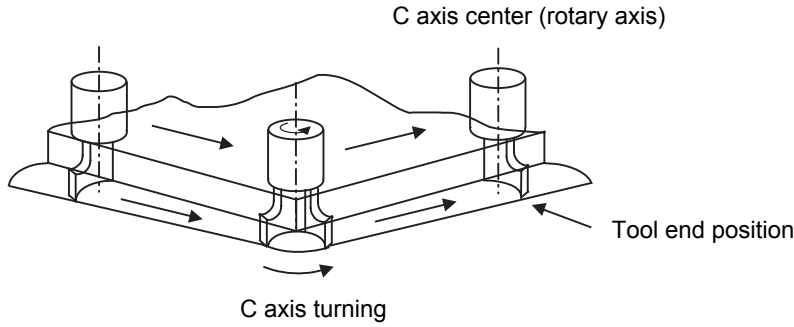
15.7 Normal Line Control; G40.1/G41.1/G42.1 (G150/G151/G152)



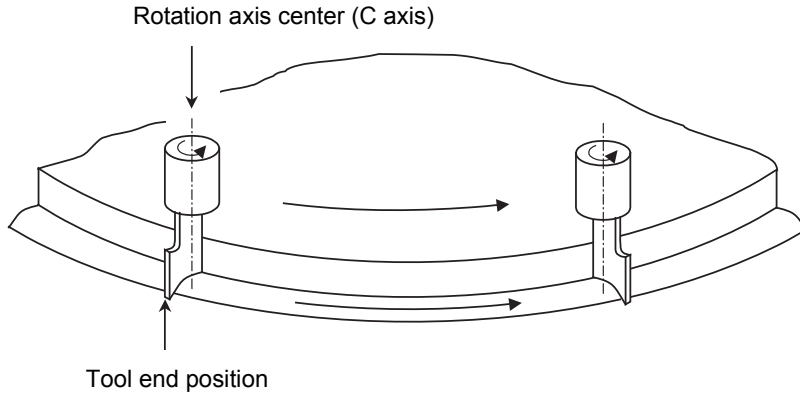
Function and purpose

The C axis (rotary axis) turning will be controlled so that the tool constantly faces the normal line direction in respect to the movement of the axes in the selected plane during program operation.

At the block seams, the C axis turning is controlled so that the tool faces the normal line direction at the next block's start point.



During arc interpolation, the rotary axis turning is controlled in synchronization with the operation of the arc interpolation.



The normal line control I and II can be used according to the C axis turning direction during normal line control. Which method is to be used depends on the MTB specifications (parameter "#1524 C_type").

Normal line control type	Turning direction	Turning speed	Turning speed in arc interpolation
Type I (#1524 C_type = 0)	Direction that is 180° or less (shortcut direction)	Parameter speed (#1523 C_feed)	Speed when the program path follows the F command
Type II (#1524 C_type = 1)	As a principle, the commanded direction	Feedrate	Speed when the tool nose follows the F command



Command format

Normal line control cancel

G40.1 (G150)X__ Y__ F__ ;

Normal line control left ON

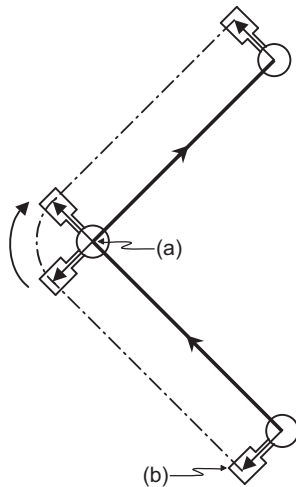
G41.1 (G151)X__ Y__ F__ ;

Normal line control right ON

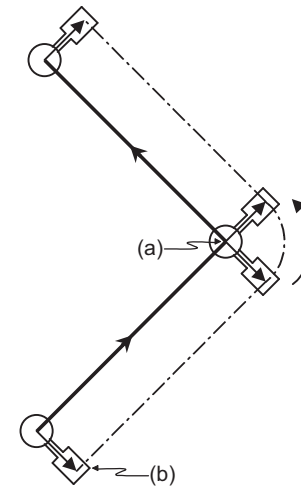
G42.1 (G152)X__ Y__ F__ ;

X	X axis end point coordinate
Y	Y axis end point coordinate
F	Feedrate

G41.1 Normal line control left side



G42.1 Normal line control right side



- (a) Center of rotation
- (b) Tool end
- Programmed path
- - - - Tool end path

The normal line control axis depends on the MTB specifications (parameter #1522 C_axis).

Normal line control is carried out in respect to the movement direction of the axis which is selecting the plane.

G17 plane I-J axes

G18 plane K-I axes

G19 plane J-K axes

Whether the normal line control is canceled at resetting depends on the MTB specifications (parameter "#1210 Rst-Gmd/ bitE").



Detailed description

Definition of the normal line control angle

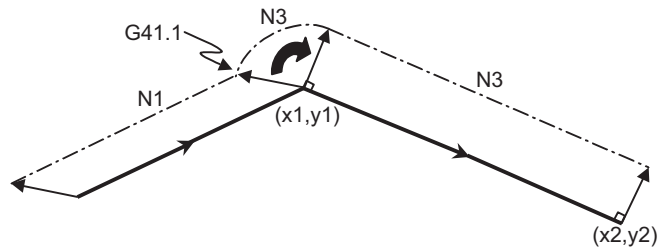
The normal line control angle is 0° (degree) when the tool is facing the horizontal axis (+ direction) direction. The counterclockwise direction turning is + (plus), and the clockwise direction turning is - (minus).

<p>G17 plane (I - J axes) ... The axis angle is 0°(degree) when the tool is facing the +I direction.</p>	
<p>G18 plane (K - I axes) ... The axis angle is 0°(degree) when the tool is facing the +K direction.</p>	
<p>G19 plane (J - K axes) ... The axis angle is 0°(degree) when the tool is facing the +J direction.</p>	

Normal line control turning operation in respect to movement command

(1) Start-up

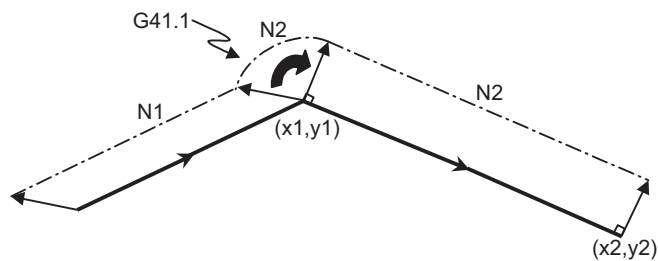
After the normal line control axis turns to the right angle of the advance direction at the start point of the normal line control command block, the axis which is selecting the plane is moves. Note that the normal line control axis at the start up turns in the direction that is 180° or less (shortcut direction) in both the normal line control type I and II.



```

:
N1 G01 Xx1 Yy1 Ff1 ;
N2 G41.1 ;           ... Independent
                    block
N3 Xx2 Yy2 ;
:
    
```

N2 is fixed



```

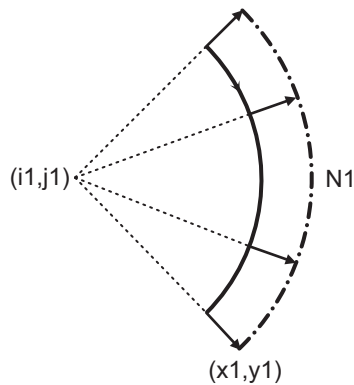
:
N1 G01 Xx1 Yy1 Ff1 ;
N2 G41.1 Xx2 Yy2 ; ... Same block
:
    
```

(2) During normal line control mode

(a) Operation in block

During interpolation of the linear command, the angle of the normal line control axis is fixed, and the normal line control axis does not turn.

During the arc command, the normal line control axis turns in synchronization with the operation of the arc interpolation.



```

:
G41.1 ;
N1 G02 Xx1 Yy1 Ii1 Jj1 ;
:
    
```

— Programmed path

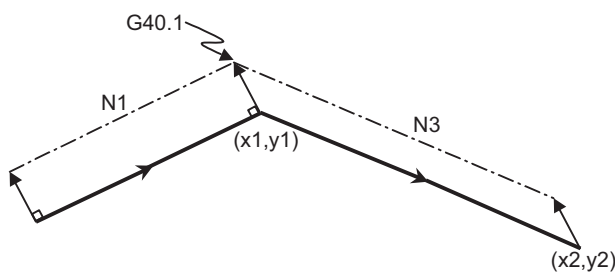
- - - - - Tool end path

(b) Block seam

<p>No tool radius compensation</p>	<p>After the normal line control axis is turned to be at the right angle of the plane selecting movement in the next block, the operation moves to the next block.</p> <div style="display: flex; justify-content: space-around;"> <div style="text-align: center;"> <p>Liner - Liner</p> </div> <div style="text-align: center;"> <p>Liner - Arc</p> </div> <div style="text-align: center;"> <p>Arc - Arc</p> </div> </div> <p>— Programmed path - - - - Tool end path</p>
<p>With tool radius compensation</p>	<p>If tool radius compensation is applied, normal line control is carried out along the path to which the tool radius compensation is applied.</p> <div style="display: flex; justify-content: space-around;"> <div style="text-align: center;"> <p>Liner - Liner</p> </div> <div style="text-align: center;"> <p>Liner - Arc</p> </div> <div style="text-align: center;"> <p>Arc - Arc</p> </div> </div> <p>— Programmed path — Tool radius compensation path - - - - Tool end path</p>

(3) Cancel

The normal line control axis will not turn, and the plane selecting axis will be moved by the program command.

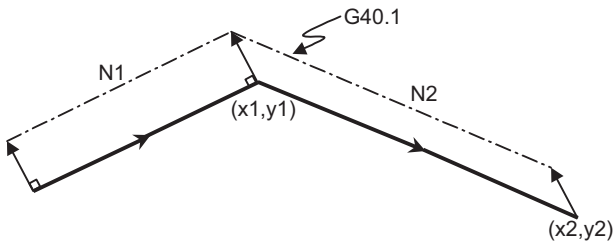


```

:
N1 G01 Xx1 Yy1 Ff1 ;
N2 G40.1 ;           ... Independent block
N3 Xx2 Yy2 ;
:

```

N2 is fixed



```

:
N1 G01 Xx1 Yy1 Ff1 ;
N2 G40.1 Xx2 Yy2 ; ... Same block
:

```

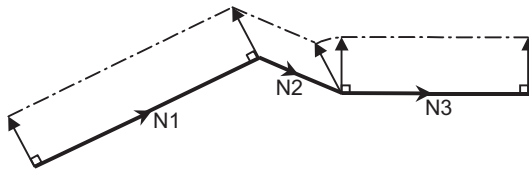
Normal line control temporary cancel

During normal line control, the turning operation for the normal line control axis is not carried out at the seam between a block and the next block, in which the movement amount is smaller than that set with the parameter (#1535 C_leng).

(1) For liner block;

When the movement amount of the N2 block is smaller than the parameter(#1535 C_leng), the normal line control axis is not turned at the seam between the N1 block and N2 block. It stays the same direction as the N1 block.

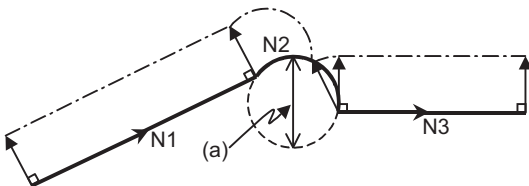
N2 block movement amount < Parameter(#1535 C_leng)



(2) For arc block;

When the diameter value of the N2 block is smaller than the parameter(#1535 C_leng), the normal line control axis is not turned at the seam between the N1 block and N2 block. During arc interpolation of the N2 block, the normal line control axis does not turn in synchronization with the operation of arc interpolation.

N2 block diameter value < Parameter (#1535 C_leng)



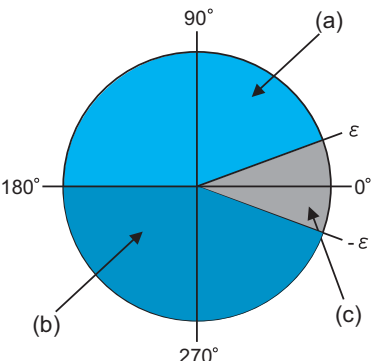
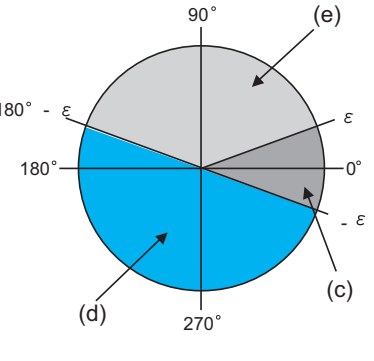
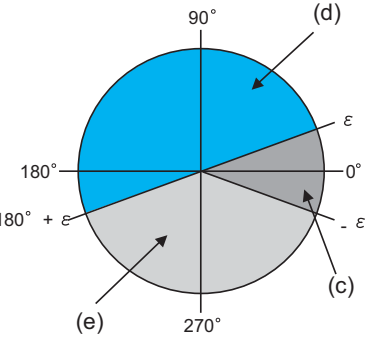
(a) Diameter value

Note

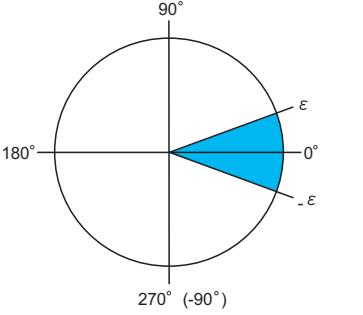
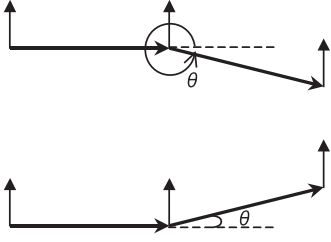
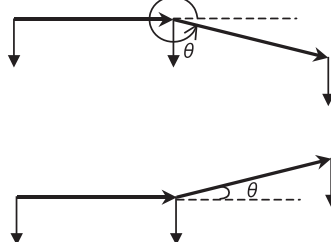
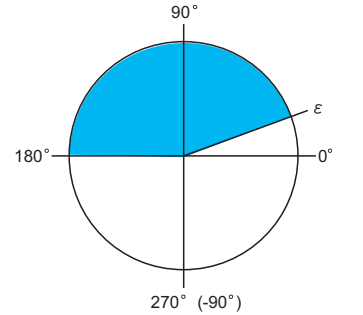
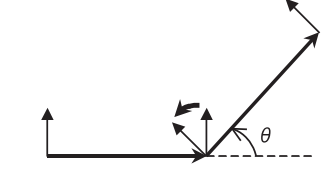
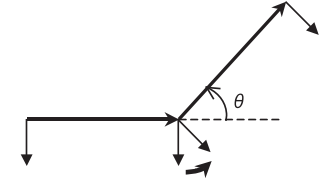
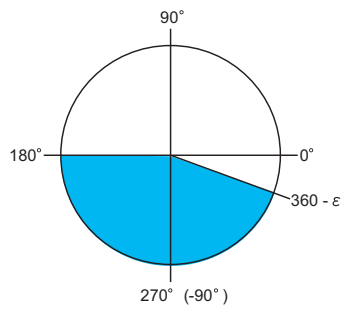
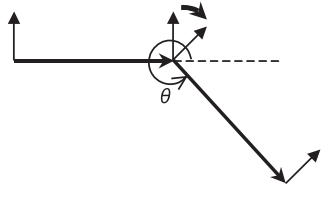
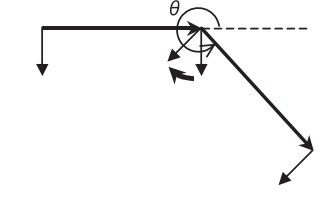
- ◆ Since operation fractions are created by calculating the intersection point of two segments, the turning operation may or may not be carried out when the parameter (#1535 C_leng) and the segment length are equal.

Normal line control axis turning direction at block seam

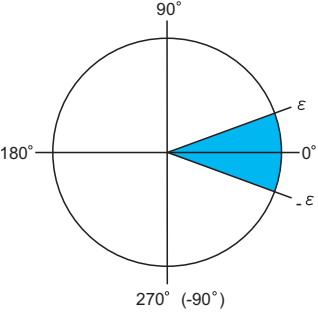
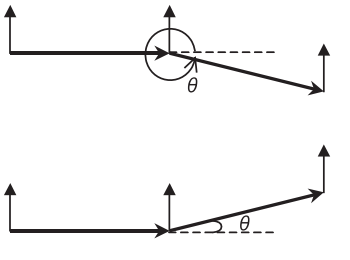
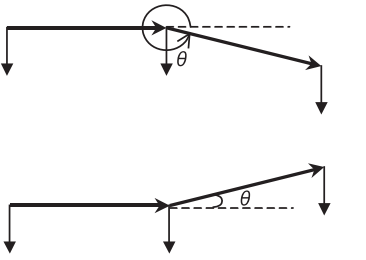
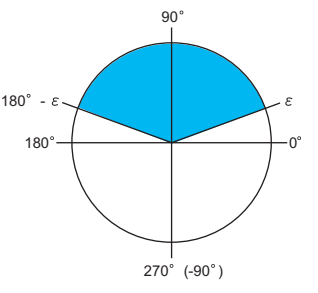
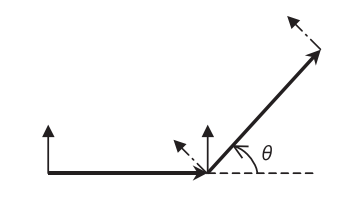
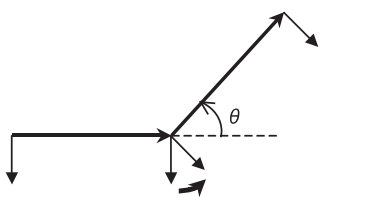
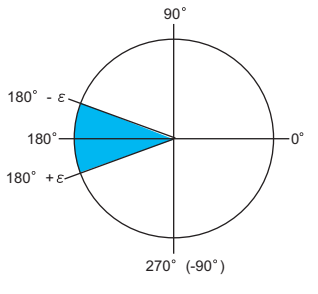
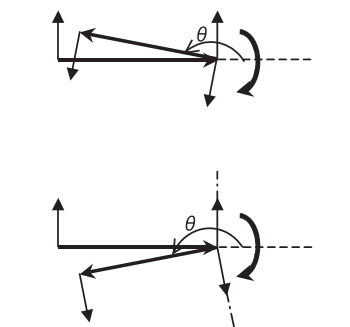
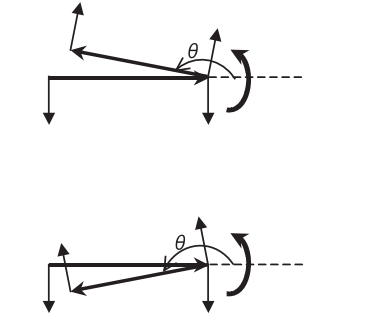
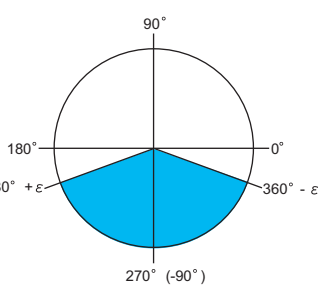
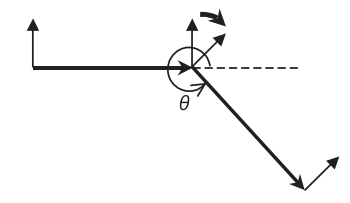
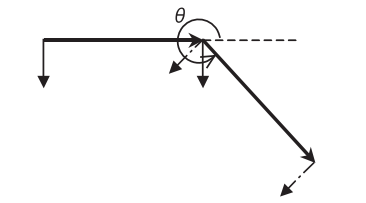
The normal line control axis turning direction at block seam differs according to the normal line control type I or II (parameter "#1524 C_type"). The turning angle is limited by the angle ϵ set with the parameter (#1521 C_min). These parameter settings depend on the MTB specifications.

Item	Normal line control type I	Normal line control type II
Normal line control axis turning direction at block seam	Direction that is 180° or less. (shortcut direction)	G41.1: - direction (CW) G42.1: + direction (CCW)
Normal line control axis turning angle at block seam	<p>♦ When $\theta < \epsilon$, turning is not performed. θ: Turning angle ϵ: Parameter (#1521 C_min)</p> <p>♦ When the turning angle is 180 degrees, the turning direction is undefined regardless of the command mode.</p> <p>[G41.1/G42.1 When the normal line control axis is at 0°]</p>  <p>(a) The normal line control axis turns counterclockwise. (b) The normal line control axis turns clockwise. (c) The axis does not turn.</p>	<p>♦ When $\theta < \epsilon$, turning is not performed. θ: Turning angle ϵ: Parameter (#1521 C_min)</p> <p>The operation error (0118) will occur in the following cases: [For G41.1] $\epsilon \leq \theta < 180^\circ - \epsilon$ [For G42.1] $180^\circ + \epsilon < \theta \leq 360^\circ - \epsilon$</p> <p>[G41.1 When the normal line control axis is at 0°]</p>  <p>[G42.1 When the normal line control axis is at 0°]</p>  <p>(c) The axis does not turn. (d) The normal line control axis turns. (e) Operation error (0118)</p>

(1) Normal line control type I

Normal line control axis turning angle at block seam: θ	G41.1	G42.1
<p>1. $-\epsilon < \theta < \epsilon$</p> 	 <p>No turning</p>	 <p>No turning</p>
<p>2. $\epsilon \leq \theta < 180^\circ$</p> 		
<p>3. $180^\circ \leq \theta \leq 360^\circ - \epsilon$</p> 	 <p>Shortcut direction</p>	 <p>Shortcut direction</p>

(2) Normal line control type II

Normal line control axis turning angle at block seam: θ	G41.1	G42.1
<p>1. $-\epsilon < \theta < \epsilon$</p> 	 <p>No turning</p>	 <p>No turning</p>
<p>2. $\epsilon \leq \theta < 180^\circ - \epsilon$</p> 	 <p>Operation error (0118) (*1)</p>	
<p>3. $180^\circ - \epsilon \leq \theta \leq 180^\circ + \epsilon$</p> 		
<p>4. $180^\circ + \epsilon < \theta \leq 360^\circ - \epsilon$</p> 		 <p>Operation error (0118) (*1)</p>

(*1) If the axis turns into the command direction, it turns inside the workpiece, causing an operation error.

Operation to be performed when the turning angle set before the circular interpolation starts falls below the minimum turning angle

The turning angle falls to or below the minimum turning angle (parameter "#1521 C_min") before the circular interpolation starts; therefore, turning operation may not be inserted.

In this case, it depends on the parameter "#12105 C_minTyp" whether to interpolate the turning angle which was not inserted before the tool reaches the end point of circular interpolation.

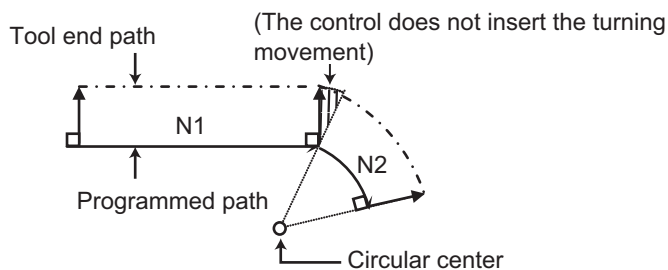
These parameters depends on the MTB specifications.

If the turning angle set before the linear interpolation starts falls to or below the minimum turning angle, turning is not carried out.

[The turning angle is interpolated up to the end point of the arc ("#12105 C_minTyp" = 0).]

The turning angle in the section in which the normal line control axis is not turned is interpolated up to the end point of the circular interpolation.

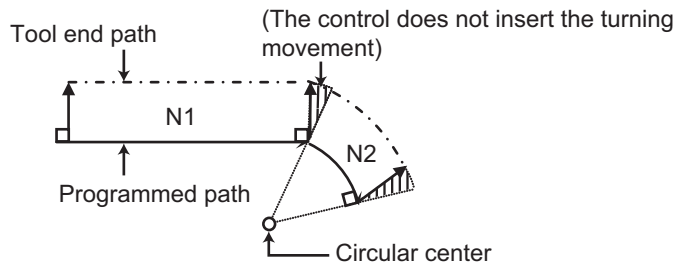
The turning angle falls below the value of parameter #1521.



[The turning angle is not interpolated ("#12105 C_minTyp" = 1).]

The turning angle in the section in which the normal line control axis is not turned is not interpolated during circular interpolation.

The turning angle falls below the value of parameter #1521.



Therefore, an operation error will occur.

Normal line control axis turning speed Turning speed at block seam (select from type I or type II)

(1) Normal line control axis turning speed at block seam

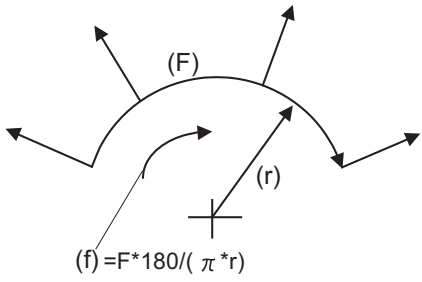
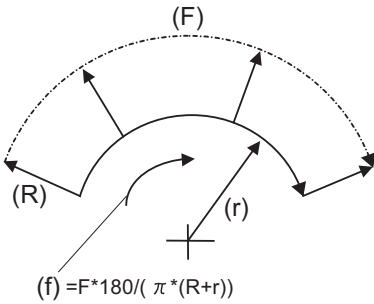
(a) Rapid traverse

Normal line control type I	Normal line control type II
<p>♦Dry run OFF</p> <p>The rapid traverse rate (#2001 rapid) is applied.</p> <p>Normal line control axis turning speed f = Rapid traverse rate * (Rapid traverse override) (° / min)</p>	<p>♦Dry run OFF</p> <p>Normal line control axis turning speed f = $F * 180 / (\pi * R) * (\text{Rapid traverse override})$ (°/min)</p> <p>When R = 0, follow the formula below.</p> <p>Normal line control axis turning speed f = $F * (\text{Rapid traverse override})$ (° /min)</p> <p>F: Rapid traverse rate (#2001 rapid) (mm/min) R: Parameter (#8041 C-rot.R) (mm) (Length from normal line control axis center to tool nose)</p> <p><Note></p> <p>(1) If the normal line control axis turning speed exceeds the rapid traverse rate (#2001 rapid), the rapid traverse rate will be applied.</p>
<p>♦Dry run ON</p> <p>The manual feedrate is applied.</p> <p>Normal line control axis turning speed f = Manual feedrate * (Cutting feed override) (° /min)</p> <p><Note></p> <p>(1) When the manual override valid is ON, the cutting feed override is valid.</p> <p>(2) If the normal line control axis turning speed exceeds the cutting feed clamp speed (#2002 clamp), the cutting feed clamp speed will be applied.</p> <p>(3) When the rapid traverse is ON, the dry run is invalid.</p>	<p>♦Dry run ON</p> <p>Normal line control axis turning speed f = $F * 180 / (\pi * R) * (\text{Rapid traverse override})$ (°/min)</p> <p>When R = 0, follow the formula below.</p> <p>Normal line control axis turning speed f = $F * (\text{Rapid traverse override})$ (° /min)</p> <p>F: Rapid traverse rate (#2001 rapid) (mm/min) R: Parameter (#8041 C-rot.R) (mm) (Length from normal line control axis center to tool nose)</p> <p><Note></p> <p>(1) If the normal line control axis turning speed exceeds the cutting feed clamp speed (#2002 clamp), the cutting feed clamp speed will be applied.</p> <p>(2) If the normal line control axis turning speed exceeds the rapid traverse rate (#2001 rapid), the rapid traverse rate will be applied.</p> <p>(3) When the rapid traverse is ON, the dry run is invalid.</p>

(b) Cutting feed

Normal line control type I	Normal line control type II
<p>•Dry run OFF</p> <p>The normal line control axis turning speed set with the parameter (#1523 C_feed) is applied. Normal line control axis turning speed f = Parameter (#1523 C_feed) * (Cutting feed override) (° /min)</p>	<p>The feedrate at the tool nose is the F command. The normal line control axis turning speed is the normal line control axis speed that follows this F command.</p> <p>Normal line control axis turning speed f = $F * 180 / (\pi * R) * (\text{Cutting feed override})$ (°/min)</p>
<p>•Dry run ON (Rapid traverse ON)</p> <p>The cutting feed clamp speed (#2002 clamp) is applied. Normal line control axis turning speed f = Cutting feed clamp speed (°/min)</p>	<p>When R = 0, follow the formula below. Normal line control axis turning speed f = F (° /min) F: Feedrate command (mm/min) R: Parameter (#8041 C-rot.R) (mm) (Length from normal line control axis center to tool nose)</p>
<p>•Dry run ON (Rapid traverse OFF)</p> <p>The manual feedrate is applied. Normal line control axis turning speed f = Manual feedrate * (Cutting feed override) (° /min)</p> <p><Note></p> <p>(1) When the manual override valid is ON, the cutting feed override is valid.</p> <p>(2) If the normal line control axis turning speed exceeds the cutting feed clamp speed (#2002 clamp), the cutting feed clamp speed will be applied.</p> <div data-bbox="414 1120 686 1433" data-label="Diagram"> <p>The diagram shows a tool path starting from a vertical line, moving horizontally to the right, and then turning upwards at an angle. An arrow labeled 'F' indicates the feedrate along the angled path. A curved arrow labeled 'f' indicates the normal line control axis turning speed at the start of the horizontal segment.</p> </div> <p>F: Feedrate command f: Normal line control axis turning speed</p>	<p><Note></p> <p>(1) If the normal line control axis turning speed exceeds the cutting feed clamp speed (#2002 clamp), the cutting feed clamp speed will be applied.</p> <p>(2) When the dry run is ON, the normal line control axis turning speed is obtained by the same expression as the rapid traverse.</p> <div data-bbox="1021 1142 1212 1366" data-label="Diagram"> <p>The diagram shows a tool path with a horizontal segment and an angled segment. A curved arrow labeled 'f' is shown at the start of the horizontal segment. A dashed line indicates the radius 'R' from the center of the normal line control axis to the tool nose. The formula $f = F * 180 / (\pi * R)$ is written below the diagram.</p> </div> <p>F: Feedrate command f: Normal line control axis turning speed R: Parameter (#8041 C-rot.R)</p>

(2) Normal line control axis turning speed during circular interpolation

Normal line control type I	Normal line control type II
<p>The normal line control axis turning speed is the rotation speed obtained by feedrate F.</p> <p>Normal line control axis turning speed f $= F * 180 / (\pi * r)$ (° /min) F : Feed command speed (mm/min) r : Arc radius (mm)</p>  <p>$(f) = F * 180 / (\pi * r)$</p>	<p>The feedrate at the tool nose is the F command. The normal line control axis turning speed is the rotation speed that follows this F command.</p> <p>Normal line control axis turning speed f $= F * 180 / (\pi * (R + r))$ (° /min) F : Feed command speed (mm/min) R : Parameter (#8041 C-rot. R) (mm) (Length from normal line control axis center to tool nose) r : Arc radius (mm)</p>  <p>$(f) = F * 180 / (\pi * (R + r))$</p>

Note

- (1) If the normal line control axis turning speed exceeds the cutting feed clamp speed (#2002 clamp), the speed will be as follows; - Normal line control axis turning speed = Cutting feed clamp speed.
 - Normal line control axis turning speed = Cutting feed clamp speed
 - Moving speed during arc interpolation = The speed according to the normal line control axis turning speed

Automatic corner arc insertion function

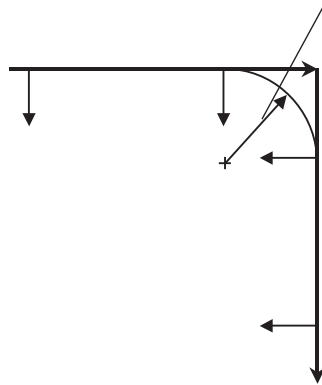
During normal line control, an arc is automatically inserted at the corner in the axis movement of the plane selection. This function is for the normal line control type I.

The radius of the arc to be inserted is set with the parameter (#8042 C-ins.R).

This parameter can be read and written using the macro variable #1901.

Normal line control is performed also during the interpolation for the arc to be inserted.

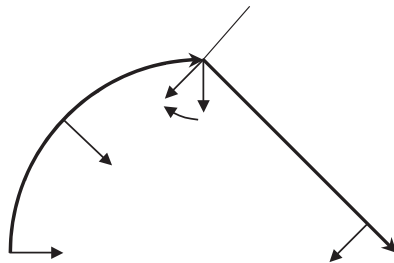
Parameter (#8042 C-rot. R)



<Supplements>

- The corner arc is not inserted in the following cases: linear and arc, arc and arc, linear and moveless or moveless and linear blocks or when a line is shorter than the radius of the arc to insert.

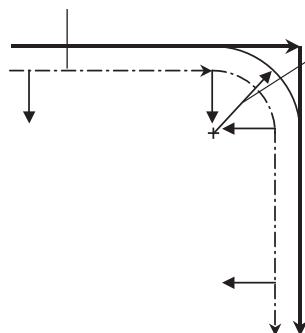
Corner R is not inserted.



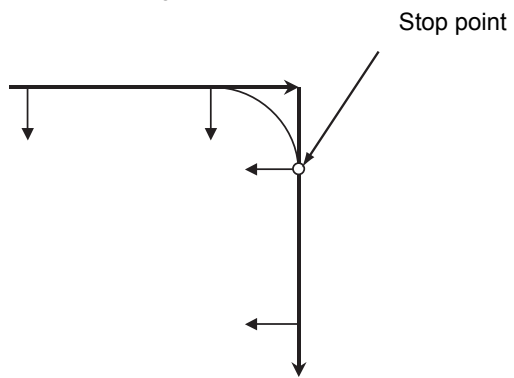
- During the radius compensation, the radius compensation is applied to the path that the corner arc is inserted.

Radius compensation path

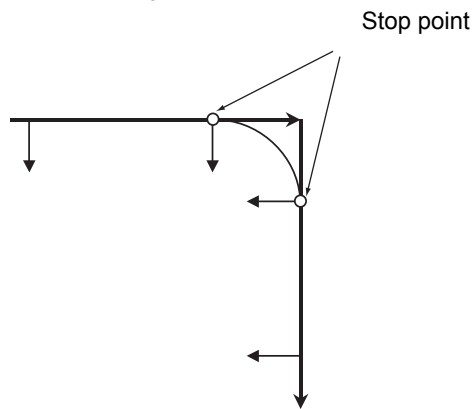
Parameter (#8042 C-rot. R)



◆The stop point of the single block and block start interlock is as follows.



◆The stop point of the cutting start interlock is as follows.





Program example

Normal line control type I

Main program

O500

```
G91X0Y0;G28C0;
G90G92G53X0Y0;
G00G54X25.Y-10.;
G03G41.1X35.Y0.R10.F10.;
#10=10;
WHILE[#10NE0]DO1;

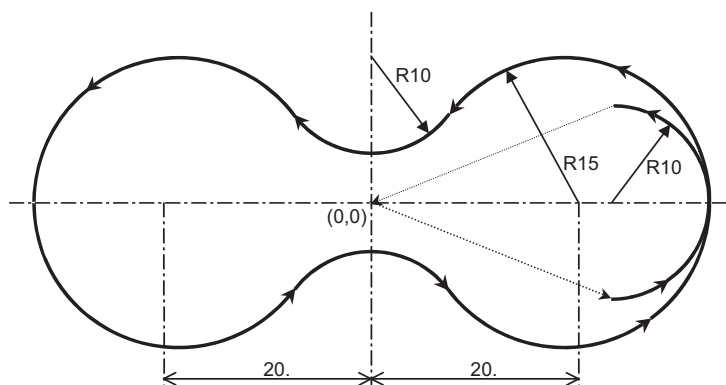
M98P501;

#10=#10-1;
END1;
G03X25.Y10.R10.;
G40.1;
G28X0Y0;
M02;
```

Sub-program

O501

```
G03X8.Y9.R15.;
G02X-8.R10.;
G03Y-9.R-15.;
G02X8.R10.;
G03X35.Y0.R15.;
M99;
```



Normal line control type II

(Example 1)

Main program

Sub-program

O500

O1001

G91X0Y0;
 G28Z0;
 G28C0;
 G90G92G53X0Y0Z0;
 G00G54G43X35.Y0.Z100.H1;
 G00Z3.;
 G01Z0.1F6000;
 G42.1;
 M98P1001L510;
 M98P1002L2;
 G91G01Y10.Z0.05;
 G40.1;
 G90G00Z100.;
 G28X0Y0Z0;
 M02;

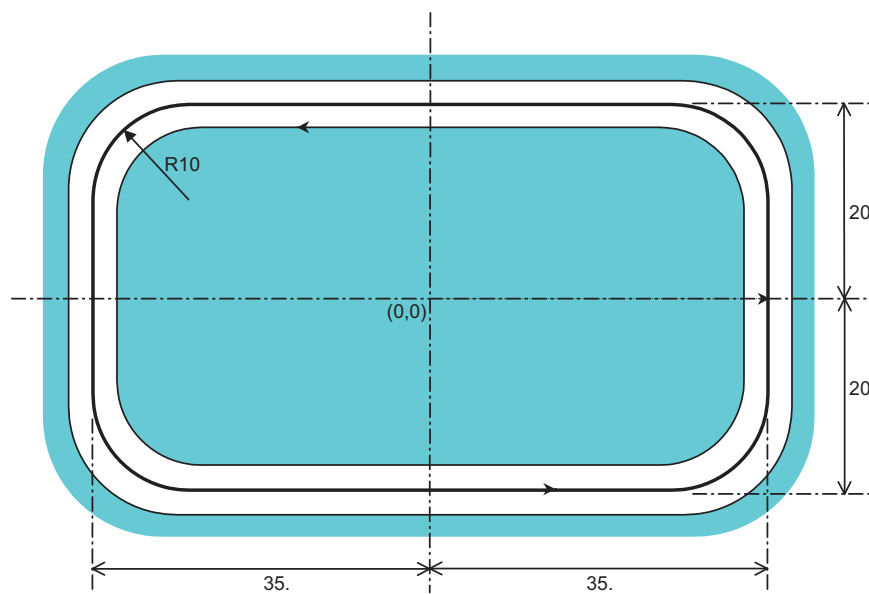
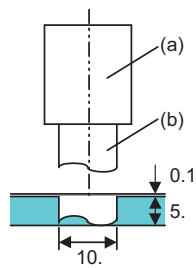
G17G91G01Y20.,R10.Z-0.01;
 X-70.,R10.;
 Y-40.,R10.;
 X70.,R10.;
 Y20.;
 M99;

(Corner chamfering/Corner R specifications are required)

O1002

G17G91G01Y20.,R10.;
 X-70.,R10.;
 Y-40.,R10.;
 X70.,R10.;
 Y20.;
 M99;

(Corner chamfering/Corner R specifications are required)



(a) C-axis
 (b) Tool

15 Program Support Functions

(Example 2)

Main program

O2000

G91G28Z0;
 G28X0Y0;
 G28C0;
 G90G92G53X0Y0Z0;
 G00G54X30.Y0.;
 G00Z3.;
 G41.1G01Z0.1F5000;
 M98P2001L510;
 M98P2002L2;
 G91G01X-30.Z0.05;
 G40.1;
 G90G00Z100.;
 G28X0Y0Z0;
 M02;

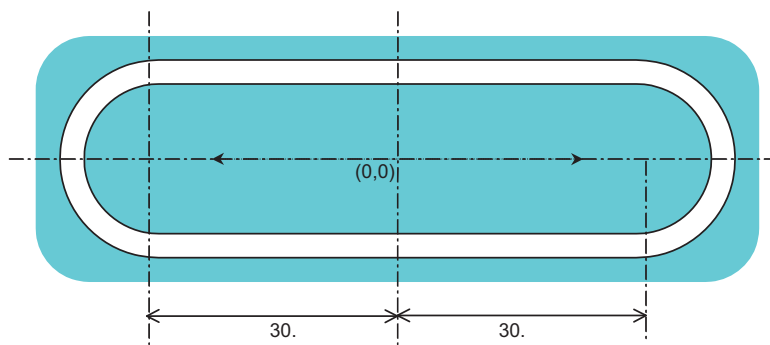
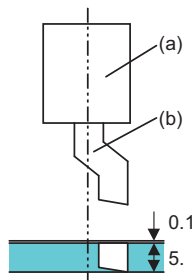
Sub-program

O2001

G17G91G01X-60.Z-0.01;
 X60.;
 M99;

O2002

G17G91G01X-60.;
 X60.;
 M99;



(a) C-axis
 (b) Tool



Relationship with other functions

Function name	Notes
Unidirectional positioning	Normal line control is not applied.
Helical cutting	Normal line control is applied normally.
Spiral interpolation	As the start point and end point are not on the same arc, a normal line control will not be applied correctly.
Exact stop check	The operation will not decelerate and stop for the turning movement of the normal line control axis.
Error detect	Error detect is not applied to the turning movement of the normal line control axis.
Override	Override is applied to the turning movement by normal line control axis.
Coordinate rotation by program	Normal line control is applied to the shape after coordinate rotation.
Scaling	Normal line control is applied to the shape after scaling.
Mirror image	Normal line control is applied to the shape after mirror image.
Thread cutting	Normal line control is not applied.
Geometric command	Normal line control is applied to the shape after geometric command.
Automatic reference position return	Normal line control is not applied.
Start position return	Normal line control is not applied to the movement to the intermediate point position. If the base specification parameter "#1086 G0Intp" is OFF, normal line control is applied to the movement from the intermediate point to a position designated in the program.
High-speed machining mode III	This cannot be commanded during normal line control. A program error (P29) will occur. The normal line control command during high-speed machining mode III cannot be issued, either. A program error (P29) will occur.
High-accuracy control	This cannot be commanded during normal line control. A program error (P29) will occur. The normal line control command during high-accuracy control cannot be issued, either. A program error (P29) will occur.
Spline	This cannot be commanded during normal line control. A program error (P29) will occur. The normal line control command during spline cannot be issued, either. A program error (P29) will occur.
High-speed High-accuracy control I/II	This cannot be commanded during normal line control. A program error (P29) will occur. The normal line control command during high-speed High-accuracy control I/II cannot be issued either. A program error (P29) will occur.
Cylindrical interpolation	This cannot be commanded during normal line control. A program error (P486) will occur. The normal line control command during cylindrical interpolation cannot be issued, either. Program error (P481) will occur.
Workpiece coordinate system offset	The workpiece coordinate system cannot be changed during normal line control. A program error (P29) will occur. The program parameter input (G10L2) cannot be commanded either. A program error (P29) will occur.
Local coordinate system offset	The local coordinate system cannot be changed during normal line control. A program error (P29) will occur.
Program restart	The program including the normal line control command cannot be restarted. "E98 CAN'T RESEARCH" will occur.
Dry run	The feedrate is changed by the dry run signal even in respect to the turning movement of the normal line control axis.

Function name	Notes
Graphic check	The section turned by normal line control is not drawn. The axes subject to graphic check are drawn.
G00 non-interpolation	Normal line control is not applied.
Polar coordinate interpolation	This cannot be commanded during normal line control. A program error (P486) will occur. The normal line control command during polar coordinate interpolation cannot be issued either. Program error (P481) will occur.
Exponential interpolation	If the normal line control axis is the same as the rotary axis of exponential interpolation, a program error (P612) will occur. If they are different, an error will not occur, but normal line control is not applied.
Plane selection	This cannot be commanded during normal line control. A program error (P903) will occur.
Mixed control	This cannot be commanded during normal line control. An operation error (M01 1035) will occur.
System variable	The block end coordinate (#5001 -) for the normal line control axis during normal line control cannot obtain a correct axis position.



Precautions

- (1) During normal line control, the program coordinates are updated following the normal line control axis movement. Thus, program the normal line control on the program coordinate system.
- (2) The normal line control axis will stop at the turning start position for the single block, cutting block start interlock and block start interlock.
- (3) If the movement command is issued to the normal line control axis (C axis) during normal line control, it is ignored.
- (4) The coordinate system preset command (G92 C_;) cannot be issued to the normal line control axis during C axis normal line control (during G41.1 or G42.1 modal). The program error (P901) will occur if commanded.
- (5) When a mirror image is applied to the axis in plane selection mode, normal line control is carried out for the shape processed with the mirror image.
- (6) The rotary axis must be designated as the normal line control axis (parameter "#1522 C_axis"). Designate so that the axis is not duplicated with the axis on the plane where normal line control is to be carried out. If an illegal axis is designated, the program error (P902) will occur when the program (G40.1, G41.1, G42.1) is commanded. The program error (P902) will also occur if the parameter "#1522 C_axis" is "0" when commanding a program. This parameter setting depends on the MTB specifications.
- (7) The movement of the normal line control axis is counted as one axis of number of simultaneous contouring control axes.
If the number of simultaneous contouring control axes exceeds the specification range by movement of the normal line control axis, the program error (P10) will occur.

15.8 Manual Arbitrary Reverse Run Prohibition ; G127



Function and purpose

The manual arbitrary reverse run function controls the feedrate, which is under automatic operation in memory or MDI mode, in proportion to the manual feedrate by the jog or the rotation speed by the manual handle, and manually carries out the reverse run.

After the automatic operation has been stopped in a block, the reverse run can be carried out back through the blocks (up to 20 blocks) that were executed before the block. If necessary, it is possible to correct the program buffer and execute the fixed program after carrying out the reverse run up to the return position.

This function (G127) is available to prevent the program from backing to blocks before the commanded block when carrying out the manual arbitrary reverse run.

The detailed setting and operation vary depending on the machine specifications. Refer to the Instruction Manual issued by the MTB.

"Forward run" means to execute blocks in the same order as for the automatic operation.

"Reverse run" means to process the executed blocks backward.

Whether the reverse run is prohibited for each part system depends on the MTB specifications (system variable #3003). Refer to "List of System Variables" for details.



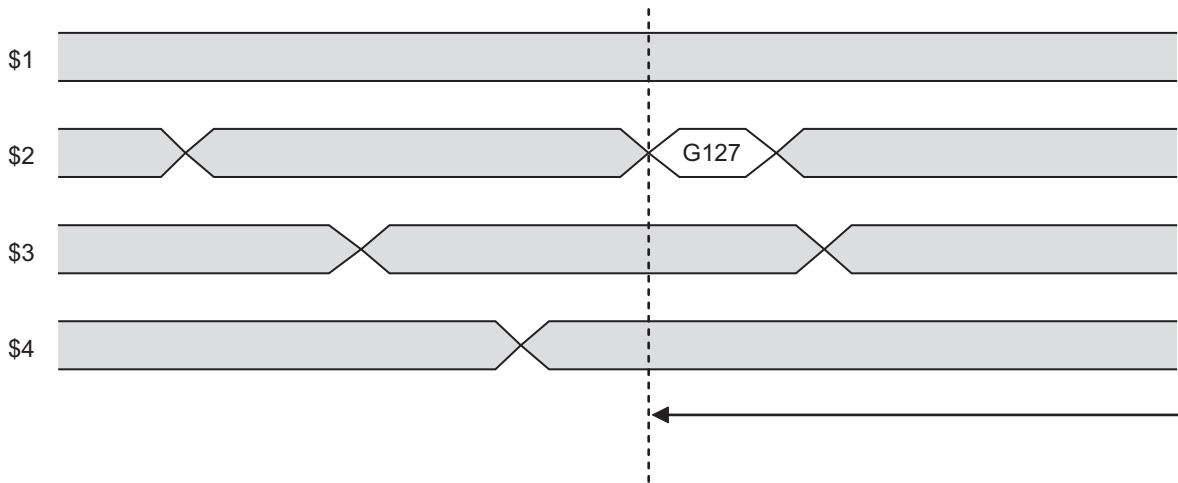
Command format

All part system reverse run prohibit command

```
G127 ;
```

This command disables the program from running reverse to blocks before G127. In part systems that do not have this command executed, the program cannot run reverse before the timing with G127 commanded in any part system even if a block is in process.

No commands in the machining program can be backed in the reverse run mode. For some G codes, the operation differs from the above. Refer to "Relationship with Other Functions".



The reverse run is disabled before the G127 block in the 2nd part system.
The reverse run is canceled in the middle of a block in part systems other than the 2nd part system.



Relationship with other functions

The following shows the relationship between the manual arbitrary reverse run command and G code.

Note

- ♦Fixed cycles or MSTB commands may be prohibited to reverse run or the reverse run operation on tapping cycle may differ depending on the MTB specifications (parameter "#1260 set32" or related PLC signals). Refer to the specifications of your machine tool.

Symbol in "Reverse run" column	Operation
○ *1	Block with reverse run enabled
○ *2	Block with restricted-reverse run enabled Refer to the Remarks for restrictions.
Δ	Block with reverse run ignored. This block is ignored in both the forward and reverse run modes.
× *3	Block with reverse run prohibited. This is intended only for the command blocks.
× *4	Block with reverse run prohibited. The reverse run is also prohibited for all blocks after the mode has been switched by this block.
× *5	Prohibits the reverse run in all part systems.

G code	Function name	Reverse run	Remarks
G00	Positioning	○ *1	-
G01	Linear interpolation	○ *1	-
G02	Circular interpolation CW and spiral/conical interpolation CW (type2)	× *3	-
G03	Circular interpolation CCW and spiral/conical interpolation CCW (type2)	× *3	-
G02.3	Exponential interpolation CW	× *3	-
G03.3	Exponential interpolation CCW	× *3	-
G02.4	3-dimensional circular interpolation	× *3	-
G03.4	3-dimensional circular interpolation	× *3	-
G04	Dwell	○ *1	Dwell skip is invalid.
G05	High-speed high-accuracy control II/III / High-speed machining mode	× *4	-
G05.1	High-speed high-accuracy control I / Spline	× *4	-
G06.2	NURBS interpolation	× *4	-
G07	Hypothetical axis interpolation	× *3	-
G07.1 G107	Cylindrical interpolation	× *4	-
G08	High-accuracy control	× *4	-
G09	Exact stop check	○ *1	-
G10	Program data input (Parameter / Compensation amount / Coordinate rotation by parameter data) / Life management data registration	Δ	The reverse run is enabled, but data is not recovered.
G10.6	Tool retract command	× *3	-
G11	Program parameter input / cancel	Δ	The reverse run is enabled, but data is not recovered.
G12	Circular cutting CW	× *3	-
G13	Circular cutting CCW	× *3	-
G12.1 G112	Polar coordinate interpolation ON	× *4	-
G13.1 G113	Polar coordinate interpolation cancel	× *4	-

15 Program Support Functions

G code	Function name	Reverse run	Remarks
G15	Polar coordinate command OFF	× *4	-
G16	Polar coordinate command ON	× *4	-
G17	X-Y plane selection	○ *2	Data is recovered using the modal information storage block.
G18	Z-X plane selection	○ *2	Data is recovered using the modal information storage block.
G19	Y-Z plane selection	○ *2	Data is recovered using the modal information storage block.
G20	Inch command	○ *1	Switched with the movement command just after commanded.
G21	Metric command	○ *1	Switched with the movement command just after commanded.
G22	Stroke check before travel ON	× *3	-
G23	Stroke check before travel OFF	× *3	-
G27	Reference position check	× *3	-
G28	Automatic reference position return	× *3	-
G29	Start position return	× *3	-
G30	2nd, 3rd and 4th reference position return	× *3	-
G30.1	Tool change position return 1	× *3	-
G30.2	Tool change position return 2	× *3	-
G30.3	Tool change position return 3	× *3	-
G30.4	Tool change position return 4	× *3	-
G30.5	Tool change position return 5	× *3	-
G30.6	Tool change position return 6	× *3	-
G31	Skip/Multi-step skip function 2	× *3	-
G31.1	Multi-step skip function 1-1	× *3	-
G31.2	Multi-step skip function 1-2	× *3	-
G31.3	Multi-step skip function 1-3	× *3	-
G33	Thread cutting	○ *2	The reverse run is enabled, but the synchronous feed is invalid. Actual cutting mode available.
G34	Special fixed cycle (bolt hole circle)	× *4	-
G35	Special fixed cycle (write at angle)	× *4	-
G36	Special fixed cycle (arc)	× *4	-
G37	Automatic tool length measurement	× *3	-
G37.1	Special fixed cycle (grid)	× *4	-
G38	Tool radius compensation vector designation	× *3	-
G39	Tool radius compensation corner arc	× *3	-
G40	Tool radius compensation cancel / 3-dimensional tool radius compensation cancel	○ *2	Data is recovered using the modal information storage block.
G41	Tool radius compensation left / 3-dimensional tool radius compensation left	○ *2	Data is recovered using the modal information storage block.
G42	Tool radius compensation right / 3-dimensional tool radius compensation right	○ *2	Data is recovered using the modal information storage block.
G40.1 G150	Normal line control cancel	× *4	-
G41.1 G151	Normal line control left ON	× *4	-
G42.1 G152	Normal line control right ON	× *4	-
G43	Tool length compensation (+)	○ *2	Data is recovered using the modal information storage block.

G code	Function name	Reverse run	Remarks
G44	Tool length compensation (-)	○ *2	Data is recovered using the modal information storage block.
G43.1	Tool length compensation along the tool axis	× *3	-
G43.4	Tool center point control type1 ON	× *4	-
G43.5	Tool center point control type2 ON	× *4	-
G45	Tool position offset (expansion)	○ *2	Data is recovered using the modal information storage block.
G46	Tool position offset (reduction)	○ *2	Data is recovered using the modal information storage block.
G47	Tool position offset (double)	○ *2	Data is recovered using the modal information storage block.
G48	Tool position offset (decreased by half)	○ *2	Data is recovered using the modal information storage block.
G49	Tool length compensation cancel / Tool center point control cancel	○ *1/ × *3	If tool length compensation cancel is designated, reverse running is enabled.
G50.2	Scaling cancel	× *4	-
G51.2	Scaling ON	× *4	-
G50.1	Mirror image by G code cancel	× *3	-
G51.1	G command mirror image ON	× *3	-
G52	Local coordinate system setting	○ *2	Data is recovered using the modal information storage block.
G53	Machine coordinate system selection	○ *2	Data is recovered using the modal information storage block.
G54	Workpiece coordinate system 1 selection	○ *2	Data is recovered using the modal information storage block.
G55	Workpiece coordinate system 2 selection	○ *2	Data is recovered using the modal information storage block.
G56	Workpiece coordinate system 3 selection	○ *2	Data is recovered using the modal information storage block.
G57	Workpiece coordinate system 4 selection	○ *2	Data is recovered using the modal information storage block.
G58	Workpiece coordinate system 5 selection	○ *2	Data is recovered using the modal information storage block.
G59	Workpiece coordinate system 6 selection	○ *2	Data is recovered using the modal information storage block.
G54.1	Workpiece coordinate system selection 48 / 96 sets extended	○ *2	Data is recovered using the modal information storage block.
G60	Unidirectional positioning	× *3	-
G61	Exact stop check mode	○ *1	-
G61.1	High-accuracy control ON	× *4	-
G61.2	High-accuracy spline	× *4	-
G62	Automatic corner override	○ *1	-
G63	Tapping mode	○ *1	-
G63.1	Synchronous tapping mode (Forward tapping)	× *4	-
G63.2	Synchronous tapping mode (Reverse tapping)	× *4	-
G64	Cutting mode	○ *1	-
G65	Macro call Simple call	○ *1	-
G66	User macro Modal call A	○ *1	-
G66.1	User macro Modal call B	○ *1	-
G67	User macro Modal call cancel	○ *1	-

G code	Function name	Reverse run	Remarks
G68	Coordinate rotation by program mode ON / 3-dimensional coordinate conversion mode ON	× *4	-
G68.2	Inclined surface machining command	× *3	-
G68.3	Inclined surface machining command (Based on tool axis direction)	× *3	-
G69	Coordinate rotation by program mode cancel / 3-dimensional coordinate conversion mode cancel / Inclined surface machining command cancel	× *4	-
G70	User fixed cycle	× *3	-
G71	User fixed cycle	× *3	-
G72	User fixed cycle	× *3	-
G73	Fixed cycle (step)	○ *1	Data is created for each movement block in the fixed cycle.
G74	Fixed cycle (reverse tap)	○ *2	The reverse run is enabled, but the synchronous feed is invalid. Actual cutting mode available.
G75	Fixed cycle (circular cutting cycle)	○ *1	Data is created for each movement block in the fixed cycle.
G76	Fixed cycle (Fine boring)	○ *1	Data is created for each movement block in the fixed cycle.
G77	User fixed cycle	× *3	-
G78	User fixed cycle	× *3	-
G79	User fixed cycle	× *3	-
G80	Fixed cycle for drilling cancel	○ *1	-
G81	Fixed cycle (drill/spot drill)	○ *1	Data is created for each movement block in the fixed cycle.
G82	Fixed cycle (drill/counter boring)	○ *1	Data is created for each movement block in the fixed cycle.
G83	Fixed cycle (deep drilling)	○ *1	Data is created for each movement block in the fixed cycle.
G84	Fixed cycle (tapping)	○ *2	The reverse run is enabled, but the synchronous feed is invalid. Actual cutting mode available.
G85	Fixed cycle (boring)	○ *1	Data is created for each movement block in the fixed cycle.
G86	Fixed cycle (boring)	○ *1	Data is created for each movement block in the fixed cycle.
G87	Fixed cycle (back boring)	○ *1	Data is created for each movement block in the fixed cycle.
G88	Fixed cycle (boring)	○ *1	Data is created for each movement block in the fixed cycle.
G89	Fixed cycle (boring)	○ *1	Data is created for each movement block in the fixed cycle.
G90	Absolute value command	○ *2	Switched with the movement command just after commanded.
G91	Incremental value command	○ *2	Switched with the movement command just after commanded.
G92	Coordinate system setting	○ *1	-
G92.1	Workpiece coordinate system preset	○ *1	-
G93	Inverse time feed	○ *1	-
G94	Asynchronous feed (feed per minute)	○ *1	-
G95	Synchronous feed (feed per revolution)	○ *1	-

G code	Function name	Reverse run	Remarks
G96	Constant surface speed control ON	○ *2	Switched with the movement command just after commanded.
G97	Constant surface speed control OFF	○ *2	Switched with the movement command just after commanded.
(G94)	Asynchronous feed (feed per minute)	○ *1	-
(G95)	Synchronous feed (feed per revolution)	○ *1	-
G98	Fixed cycle (Initial level return)	○ *1	-
G99	Fixed cycle R point level return	○ *1	-
G115	Start point designation timing synchronization Type 1	○ *1	-
G116	Start point designation timing synchronization Type 2	○ *1	-
G118.2	Parameter switching (Spindle)	× *3	-
G119.2	Inertia Estimation (Spindle)	× *3	-
G100 to G225	User macro (G code call) Max. 10	○ *1	-
M98	Subprogram call	○ *1	-

15.9 Data Input by Program

15.9.1 Parameter Input by Program; G10 L70/L100, G11



Function and purpose

The parameters set from the setting and display unit can be changed in the machining programs.

G10 L70	For commanding data with decimal point, and character string data. The data's command range conforms to the parameter setting range described in Setup Manual.
G10 L100	For setting/changing the tool shape for 3D solid program check.



Command format

Data setting start command

G10 L70 ;	
P__ S__ A__ H□__ ;	Bit parameter
P__ S__ A__ D__ ;	Numerical value parameter
P__ S__ A__ <character string> ;	Character string parameter

P	Parameter No.
S	Part system No.
A	Axis No.
H	Bit type data
D	Numeric type data
character string	Character string data

Data setting end command

G11 ;

Note

- (1) The sequence of addresses in a block must be as shown above.
When an address is commanded two or more times, the last command will be valid.
- (2) The part system No. is set in the following manner. "1" for the 1st part system, "2" for 2nd part system, and so forth.
If the address S is omitted, the part system of the executing program will be applied.
As for the parameters common to part systems, the command of part system No. will be ignored.
- (3) The axis No. is set in the following manner. "1" for 1st axis, "2" for 2nd axis, and so forth.
If the address A is omitted, the 1st axis will be applied.
As for the parameters common to axes, the command of axis No. will be ignored.
- (4) Address H is commanded with the combination of setting data (0 or 1) and the bit designation □ (0 to 7).
Hd0: Sets the dth bit OFF. (d: 0 to 7)
Hd1: Sets the dth bit ON. (d: 0 to 7)
- (5) Only the decimal number can be commanded with the address D.
The value that is smaller than the input setting increment (#1003 iunit) will be round off to the nearest increment.
- (6) The character string must be put in angled brackets "<" and ">".
If these brackets are not provided, the program error (P33) will occur.
Up to 63 characters can be set.

- (7) Issue the "G10 L100" and "G11" commands in independent blocks. A program error (P33, P421) will occur if not commanded in independent blocks.
- (8) The parameter "#1078 Decimal pnt type 2" is disabled.
- (9) The following data cannot be changed with the G10 L70 command:
- ♦Tool compensation data
 - ♦workpiece coordinate data
 - ♦PLC switch
 - ♦PLC axis parameter
 - ♦Device open parameters
 - ♦SRAM open parameters
 - ♦DeviceNet parameters
- (10) The settings of the parameters with (PR) in the parameter list will be enabled after the power is turned OFF and ON.
Refer to the parameter list in your manual.

Data setting start command

```
G10 L100 ;
P__ T__ K__ D__ H__ I__ J__ C__ ;
```

Data setting end command

```
G11 ;
```

P	Line No. of the tool set area 1 to 80 (Required to command) (*1)
T	Tool No. 0 to 99999999 (Required to command)
K	Command the tool type using a numerical value. 0: Default tool (3: Sets to drill.) 1: Ball end mill 2: Flat end mill 3: Drill 4: Bull nose end mill 5: Chamfer 6: Tap 7: Face mill
D	Tool diameter/radius (Decimal point input available) (*2)
H	Tool length (Decimal point input available)
I	Tool shape data 1 (Decimal point input available)
J	Tool shape data 2 (Decimal point input available)
C	Command the tool color using a numerical value. 0: Default color (2: Sets to red.) 1: Gray 2: Red 3: Yellow 4: Blue 5: Green 6: Light blue 7: Purple 8: Pink

(*1) Line No. corresponds with a line No. in the tool shape set area (tool shape set screen).

(*2) The setting of "#8117 OFS Diam DESIGN" determines tool diameter or tool radius.

(*3) For details of the data, refer to the explanation of Instruction Manual "Program Check (3D)".

(*4) Omitted addresses cannot be set or changed.

(*5) When address T is set to 0, the designated line is deleted.

(*6) In the following cases, a program error (P421) occurs and the parameter in the block is not changed.

- ♦When a block contains an address whose data are out of range
- ♦When there is an illegal address
- ♦When P or T is omitted

(*7) Issue the "G10 L100" command and "G11" commands in independent blocks. A program error (P421) will occur if not commanded in independent blocks.

- (*8) The parameter "#1078 Decimal pnt type 2" is enabled.
- (*9) The parameter "#8044 Unit*10" is disabled.
- (*10) The display or operation at graphic check varies depending on the model or display unit.



Precautions

Parameter update timing

The timing for updating the spindle parameter and the NC axis parameter settings depends on the MTB specifications (parameter "#1254 set26/bit3").

#1254 set26/bit3	Spindle parameter	NC axis parameter
Invalid	The program updates the parameter settings, waiting for "all axes smoothing zero" in all part systems.	
Valid	The program updates the parameter settings without waiting for "smoothing zero". (*1)	The program updates the parameter settings, waiting for "all axes smoothing zero" in control part systems. (*2)

- (*1) The parameters of the target spindle are not updated while the functions below are active. The parameters are updated after the functions have been completed.
 - ♦Synchronous tapping cycle
 - ♦The spindle for spindle position control is in C axis mode and the C axis is in motion.
- (*2) The program updates the exchange axis under the arbitrary axis exchange control, waiting for "all axes smoothing zero" in the exchange destination part system.



Program example

(1) For G10 L70

G10 L70 ;	
P6401 H71 ;	Sets "1" to "#6401 bit7".
P8204 S1 A2 D1.234 ;	Sets "1.234" to "#8204 of the 1st part system 2nd axis".
P8621 <X> ;	Sets "X" to "#8621".
G11 ;	

(2) When G10 L100 command

G10 L100;	
P1 T1 K3 D5. H20. I0 J0 C2 ;	Set the data of Line 1
P2 T10 D10. ;	Set "10." for the tool diameter/radius of Line 2
P8 T0 ;	Clear the data of Line 8
G11 ;	

15.9.2 Compensation Data Input by Program ; G10 L2/L10/L11/L12/L13/L20, G11

**Function and purpose**

The tool compensation and workpiece offset can be set or changed by the program using the G10 command.

During the absolute value (G90) mode, the commanded offset amount serves as the new offset, whereas during the incremental value (G91) mode, the currently set offset plus the commanded offset serves as the new offset.

**Command format****Workpiece coordinate system offset input (L2)**

G90 (G91) G10 L2 P_X_Y_Z_;	
P	0: External workpiece 1: G54 2: G55 3: G56 4: G57 5: G58 6: G59
X, Y, Z	Offset amount of each axis

Note

(1) The compensation amount in the G91 will be an incremental amount and will be cumulated each time the program is executed. Command G90 or G91 before the G10 as a cautionary means to prevent this type of error.

Extended workpiece coordinate system offset input (L20)

G10 L20 P_X_Y_Z_;	
P	n No. of G54.n (1 to 300)
X, Y, Z	Offset amount of each axis

Offset input to the currently selected workpiece coordinate system (When the L command is omitted)

G10 P_X_Y_Z_;	
P	(1) During G54 to G59 modal 0 : External workpiece offset (EXT) 1 to 6 : Workpiece offset input (G54 to G59) Other than 0 to 6 : Program error (P35)
	(2) During G54.n modal 1 to 300 : Extended workpiece coordinate offset amount setting (G54.n) Other than 1 to 300 : Program error (P35)
X, Y, Z	Offset amount of each axis

Tool compensation input (L10/L11/L12/L13)

Tool compensation memory type I

G10 L10 P_ R_ ;	
P	Compensation No.
R	Compensation amount

Tool compensation memory type II

G10 L10 P_ R_ ;	Tool length compensation shape compensation
G10 L11 P_ R_ ;	Tool length compensation wear compensation
G10 L12 P_ R_ ;	Radius shape compensation
G10 L13 P_ R_ ;	Radius wear compensation

Note

(1) Type I is selected when parameter "#1037 cmdtyp" is set to "1", and type II is selected when set to "2".

Compensation input cancel**G11 ;**

Common to workpiece coordinate system offset, extended workpiece coordinate system offset, and tool compensation.



Detailed description

- (1) Even if this command is displayed on the screen, the offset No. and variable details will not be updated until actually executed.
- (2) G10 is an unmodal command and is valid only in the commanded block.
- (3) The G10 command does not contain movement, but must not be used with G commands other than G54 to G59, G90 or G91.
- (4) Do not command G10 in the same block as the fixed cycle and sub-program call command. This will cause malfunctioning and program errors.
- (5) The workpiece offset input command (L2 or L20) should not be issued in the same block as the tool compensation input command (L10).
- (6) If an illegal L No. or compensation No. is commanded, the program errors (P172 and P170) will occur respectively.
If the offset amount exceeds the maximum command value, the program error (P35) will occur.
- (7) Decimal point inputs can be used for the offset amount.
- (8) The offset amounts for the external workpiece coordinate system and the workpiece coordinate system are commanded as distances from the basic machine coordinate system zero point.
- (9) The workpiece coordinate system updated by inputting the workpiece coordinate system will follow the previous modal (G54 to G59) or the modal (G54 to G59) in the same block.
- (10) L2/L20 can be omitted when the workpiece offset is input.
- (11) When the P command is omitted for workpiece offset input, it will be handled as the currently selected workpiece compensation input.
- (12) If the G command that cannot be combined with G10 is issued in the same block, a program error (P45) will occur.
- (13) The setting range for the compensation amount is given below.
Program error (P35) occurs for any value not listed in the table after command unit conversion.
With an incremental value command, the setting range for the compensation amount is the sum of the present setting value and command value.

Setting	Compensation amount	
	Metric system	Inch system
#1003=B	± 9999.999 (mm)	± 999.9999 (inch)
#1003=C	± 9999.9999 (mm)	± 999.99999 (inch)
#1003=D	± 9999.99999 (mm)	± 999.999999 (inch)
#1003=E	± 9999.999999 (mm)	± 999.9999999 (inch)



Program example

- (1) Input the compensation amount.

```
.....; G10 L10 P10 R-12.345 ; G10 L10 P05 R9.8765 ; G10 L10 P30 R2.468 ; .....
```

```
H10=-12.345 H05=9.8765 H30=2.468
```


(2) Updating of compensation amount

(Example 1) Assume that H10 = -1000 is already set.

N1 G01 G90 G43 Z-100000 H10 F100 ;	(Z=-101000)
N2 G28 Z0 ;	
N3 G91 G10 L10 P10 R-500 ;	(The mode is the G91 mode, so -500 is added.)
N4 G01 G90 G43 Z-100000 H10 ;	(Z=-101500)

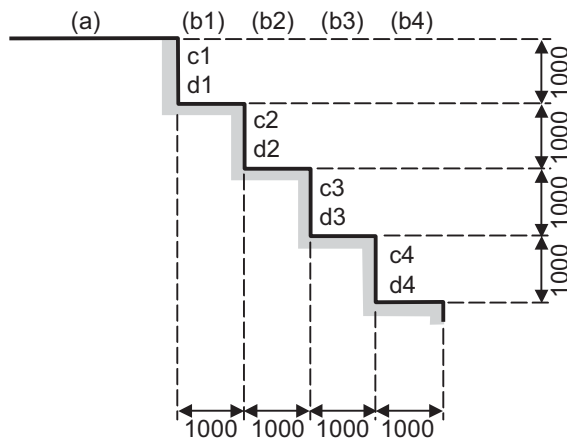
(Example 2) Assume that H10 = -1000 is already set.

Main program

N1 G00 X100000 ;	a
N2 #1=-1000 ;	
N3 M98 P1111 L4 ;	b1, b2, b3, b4

Subprogram O1111

N1 G01 G91 G43 Z0 H10 F100 ;	c1, c2, c3, c4
G01 X1000 ;	d1, d2, d3, d4
#1=#1-1000 ;	
G90 G10 L10 P10 R#1 ;	
M99 ;	



<Note>

•Final offset amount will be H10= -5000.

(Example 3) The program for Example 2 can also be written as follows.

Main program

N1 G00 X100000 ;
N2 M98 P1111 L4 ;

Subprogram O1111

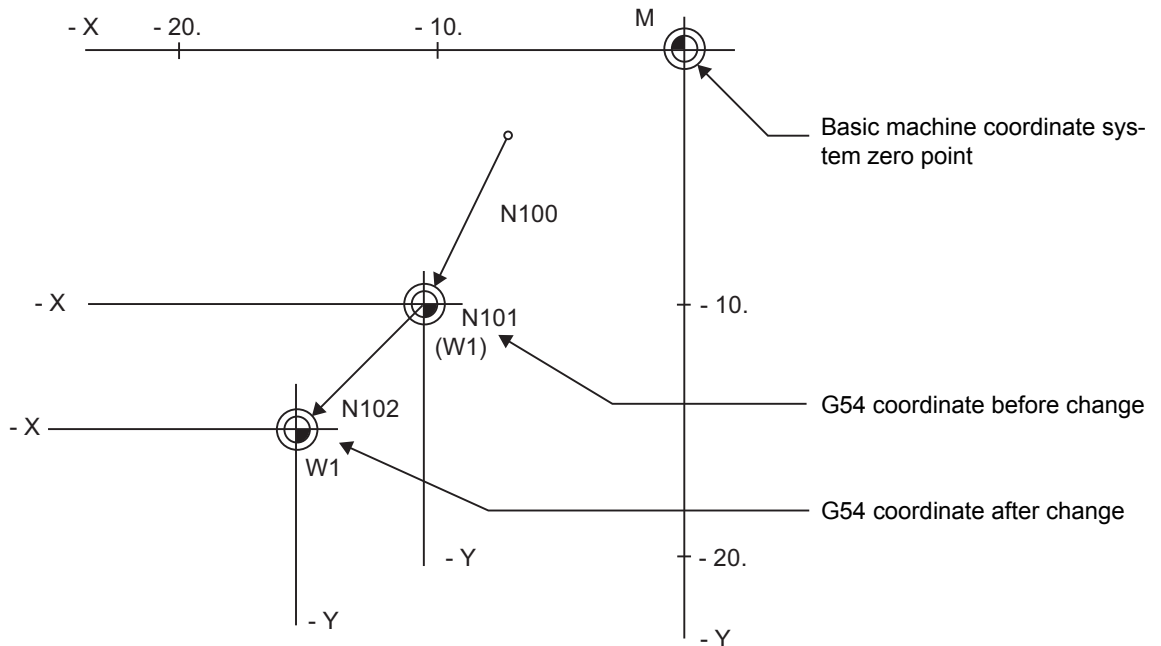
N1 G01 G91 G43 Z0 H10 F100 ;
N2 G01 X1000 ;
N3 G10 L10 P10 R-1000 ;
N4 M99 ;

(3) When updating the workpiece coordinate system offset amount

Assume that the previous workpiece coordinate system offset amount is as follows.

X=-10.000 Y=-10.000

N100 G00 G90 G54 X0 Y0 ;
N101 G90 G10 L2 P1 X-15.000 Y-15.000 ;
N102 X0 Y0 ;
M02 ;



<Note>

◆Changes of workpiece current position display in N101

The G54 workpiece position display data will change before and after the workpiece coordinate system is changed with G10 in N101.

X = 0 → X = +5.000

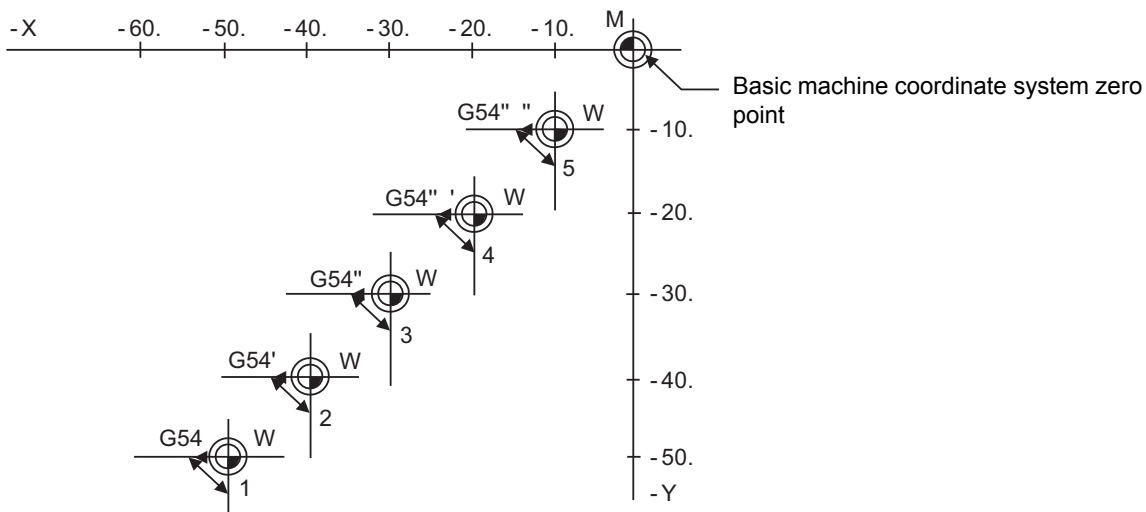
Y = 0 → Y = +5.000

When workpiece coordinate system offset amount is set in G54 to G59

G90 G10 L2 P1 X-10.000 Y-10.000 ;
G90 G10 L2 P2 X-20.000 Y-20.000 ;
G90 G10 L2 P3 X-30.000 Y-30.000 ;
G90 G10 L2 P4 X-40.000 Y-40.000 ;
G90 G10 L2 P5 X-50.000 Y-50.000 ;
G90 G10 L2 P6 X-60.000 Y-60.000 ;

(4) When using one workpiece coordinate system as multiple workpiece coordinate systems

Main program	:
	#1=-50. #2=10. ;
	M98 P200 L5 ;
	M02 ;
	%
Subprogram O200	N1 G90 G54 G10 L2 P1 X#1 Y#1 ;
	N2 G00 X0 Y0 ;
	N3 X-5. F100 ;
	N4 X0 Y-5. ;
	N5 Y0 ;
	N6 #1=#1+#2 ;
	N7 M99 ;
	%



Precautions

(1) Even if this command is displayed on the screen, the offset No. and variable details will not be updated until actually executed.

N1 G90 G10 L10 P10 R-100 ;	
N2 G43 Z-10000 H10 ;	
N3 G00 X-10000 Y-10000 ;	
N4 G90 G10 L10 P10 R-200 ;	The H10 offset amount is updated when the N4 block is executed.

15.9.3 Compensation Data Input by Program (Turning Tool) ; G10 L12/L13, G11

**Function and purpose**

If the tool compensation type is changed to type III by the compensation type selection function, it is possible to write the offset amount for three base axes, nose R compensation amount, and tool nose point (parameter "#1046 T-ofs disp type"). During the absolute value (G90) mode, the commanded tool compensation amount serves as a new one. During the incremental value (G91) mode, the currently set compensation amount plus the commanded compensation amount serves as the new compensation amount.

**Command format****Turning tool compensation input (L12/L13)**

G10 L12 P__X__Y__Z__R__Q__ ; (Shape compensation)

P	Tool shape compensation No. (1 to number of tool compensation sets)
X, Y, Z	Compensation amount for each axis
R	Nose R compensation amount
Q	Hypothetical tool nose point

G10 L13 P__X__Y__Z__R__Q__ ; (Wear compensation)

P	Wear compensation No. (1 to number of tool compensation sets)
X, Y, Z	Compensation amount for each axis
R	Nose R compensation amount
Q	Hypothetical tool nose point

Compensation input cancel

G11 ;



Detailed description

The commanded range and unit of the compensation amount are as follows.

Program error (P35) occurs for any value not listed in the table after command unit conversion. With an incremental value command, the commanded range for the compensation amount is the sum of the present setting value and command value.

Setting	Compensation amount	
	Metric system	Inch system
#1003=B	± 9999.999 (mm)	± 999.9999 (inch)
#1003=C	± 9999.9999 (mm)	± 999.99999 (inch)
#1003=D	± 9999.99999 (mm)	± 999.999999 (inch)
#1003=E	± 9999.999999 (mm)	± 999.9999999 (inch)



Precautions

- (1) The X, Y, and Z addresses are set to the axis names specified in the parameters for three base axes (parameters "#1026 base_J", "#1027 base_J", and "#1028 base_K").
The compensation data input by program of the tool offset is not available for an axis address that is not specified in the parameters for three base axes. Therefore, be sure to carry out compensation data input by program after specifying the parameters for three base axes.
- (2) The compensation data input by program is available using a command (G10 L10, L11, L12, or L13) in a normal machining center system, but only the compensation amount of the Z axis and nose R can be input as data.

G10 L10 P__ R__;	Z axis shape compensation
G10 L11 P__ R__;	Z axis wear compensation
G10 L12 P__ R__;	Nose R shape compensation
G10 L13 P__ R__;	Nose R wear compensation

[Correspondence between tool types and shape data]

[Mill tool]

Shape data	Item by tool type						
	Ball end mill	Flat end mill	Drill	Radius end mill	Chamfer	Tap	Face mill
1	Tool length						
2	Tool radius (*1)						
3	-	-	Tool nose angle	Corner rounding	End angle	Pitch	Cutter length
4	-	-	-	-	End diameter	Thread diameter	Shank diameter
5	-	-	-	-	-	-	-
6	-	-	-	-	-	-	-

[Turning tool]

Shape data	Item by tool type				
	Turning	Slotting	Thread Cutting	Turning drill	Turning tap
1	Tool length A				
2	Tool length B			Tool length B (*1)	
3	Tool nose radius	Tool nose radius	-	Tool nose angle	Pitch
4	Tool nose angle	Tool nose width	-	-	Thread diameter
5	Cutting edge angle	Max. slot depth	-	-	-
6	Tool width	Tool width	Tool width	-	-

(*1) When "#8968 Tool shape radius validity" is set to "0", input the diameter value. When it is set to "1", input the radius value.

Note

- (1) Omitted addresses cannot be set.
- (2) If address "P" or "T" is omitted, a program error (P422) will occur.
- (3) For M80 Series, the tool shape data will be rewritten during the graphic check.
- (4) For M800W and M800S Series, this change is only reflected on the graphic check drawing. The tool shape data is not rewritten.



Detailed description

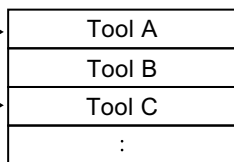
Tool shape settings from the program

The 3D check switches the drawing of tools at the timing of a tool change command. Therefore, the machining program should be prepared to run a tool shape setting command prior to the tool change command being issued.

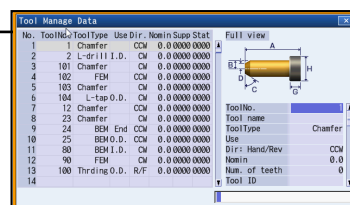
Machining program

```
O200
G10 L100;
P1 T201 ...; (Change the shape of tool A.)
P3 T203 ...; (Newly register the shape of
              tool C.)
G11;
:
T201; (Replaced with tool A.)
:
T202; (Replaced with tool B.)
:
T203; (Replaced with tool C.)
:
```

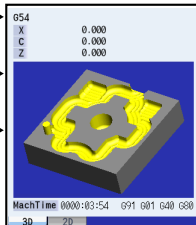
Tool shape data



Tool management screen



3D check screen



- (a) The tool is drawn with the shape that has been changed by the machining program.
- (b) The tool is drawn with a shape that has been registered on the tool management screen.
- (c) The tool is drawn with a new shape that has been registered by the machining program.



Program example

(1) Tool shape settings from the program

G10 L100;	
P1 T1 K3 D5. H20. I0 J0 C2 ;	Sets the data of data No. 1.
P2 T10 D10. ;	Sets the tool diameter of data No. 2 to "10."
P8 T0;	Sets the tool shape data of data No. 8 to "0".
G11;	



Precautions

- (1) If the G10 or G11 command is not issued in an independent block, a program error (P422) will occur.
- (2) If a block contains an address whose data is out of range, a program error (P35) will occur.
- (3) If a block contains an illegal address, a program error (P32) will occur.
- (4) The parameter "#1078 Decimal pnt type 2" is valid for the position command.
Other command addresses comply with the minimum input unit ("#1015 cunit"). (Based on the MTB specifications.)
- (5) The parameter "#8044 UNIT*10" is invalid.
- (6) The command unit of parameters to be input in mm/inch can be switched by G20/G21.

15.9.5 R-Navi Data Input by Program; G10 L110/L111, G11, G68.2, G69

**Function and purpose**

The R-Navi setup parameters can be configured from a machining program.

Command setting values with absolute values.

The input unit conforms to the input setting unit of the 1st part system and the initial inch.

In either case, the input unit depends on the MTB specifications (parameters "#1003 iunit" and "#1041 I_inch").

The parameter "#8044 UNIT*10" is invalid.

**Command format****Workpiece registration and setting**

G69;	Canceling the selected machining surface
-------------	---

G10 L110;	Start setting workpiece data
------------------	-------------------------------------

Q_<>F_C_R_X_Y_Z_I_J_K_;	Data setting
--------------------------------------	---------------------

G11;	End data setting
-------------	-------------------------

♦Command G10 and G11 in independent blocks.

A program error (P423) will occur if not commanded in independent blocks.

♦Address Q cannot be omitted. If omitted, a program error (P423) will occur.

♦For the omitted addresses, data remains unchanged.

♦Cancel the selected machining surface before data setting.

If data is set to a machining workpiece including the selected machining surface, a program error (P423) will occur.

Q	Workpiece registration No. (1 to 10)
< >	Workpiece name Designate the name using up to 20 one-byte alphanumeric characters, including symbols. (If "0" is entered, the setting value is cleared.)
F	Workpiece shape 0: Rectangular parallelepiped 1: Circular cylinder
C	Basic coordinate system of machining workpiece 0 to 5: G54 to G59 6 to 305: G54.1P1 to G54.1P300
R	Marked point No. When the workpiece shape is set to rectangular parallelepiped, designate the marked point to set the basic coordinate system zero point. (0 to 8)
X, Y, Z,	Workpiece size When the shape is set to circular cylinder, designate the diameter with X and the height with Y. (0.000 to 99999.999)
I, J, K	Workpiece shift Set the shift amount from the marked point to the basic coordinate system zero point. (-99999.999 to 99999.999)

- Symbols "\", "/", ";", "*", "?", ":", "<", ">", "|", " " (space), "@", and "~" cannot be used as one-byte symbols. If an available symbol is set, a program error (P35) or (P32) will occur.
- For details on each of input data, refer to the instruction manual.

Machining surface registration and setting

G69;	Canceling the selected machining surface
-------------	---

G10 L111;	Start setting machining surface data
------------------	---

P0 Q_ D_ <_> X_ Y_ Z_ A_;	Machining surface setting (Refer to (1).)
--	--

P1 M_ B_ C_ E_ F_ H_ I_;	Designate the coordinate axis direction (1st axis). (Refer to (2).)
---------------------------------	--

P2 M_ B_ C_ E_ F_ H_ I_;	Designate the coordinate axis direction (2nd axis). (Refer to (2).)
---------------------------------	--

G11;	End data setting
-------------	-------------------------

G68.2 P10 Q_ D_;	Selecting the registered machining surface
-------------------------	---

- Command G10 and G11 in independent blocks.
A program error (P423) will occur if not commanded in independent blocks.
- Addresses P, Q, and D cannot be omitted. If omitted, a program error (P423) will occur.
- For the omitted addresses, data remains unchanged.
- For the machining surface designated with P0, set the coordinate axis direction with P1 and P2. Be sure to first command P0.
If P1 or P2 is commanded before P0, a program error (P423) will occur.
- The machining surface cannot be registered for an undefined workpiece.
If the registration command is issued, a program error (P423) will occur.
- Cancel the selected machining surface before data setting.
If data is set to the selected machining surface, a program error (P423) will occur.

(1) Command address to register the machining surface

P	Machining surface registration (0)
Q	Workpiece registration No. (1 to 10)
D	Machining surface registration No. (2 to 17)
< >	Designate the name of the machining surface using up to 15 one-byte alphanumeric characters, including symbols. (If "0" is entered, the setting value is cleared.)
X, Y, Z	Designate the coordinate system zero point (feature coordinate system zero point) of the machining surface with the offset from the basic coordinate zero point. In this case, designate the coordinate axis direction of the basic coordinate system. (-99999.999 to 99999.999)
A	From three orthogonal axes (X, Y, and Z axes), select two coordinate axes to designate the coordinate axis direction along the machining surface. 0: Z/X axis 1: Y/Z axis 2: X/Y axis

- Symbols "\", "/", ";", "*", "?", ":", "<", ">", "|", " " (space), "@", and "~" cannot be used as one-byte symbols. If an available symbol is set, a program error (P35) or (P32) will occur.
- For details on each of input data, refer to the instruction manual.

(2) Command address to designate the coordinate axis direction

P	Coordinate axis direction designation axis 1: 1st axis 2: 2nd axis
M	Coordinate axis direction designation method Designate the method to set the coordinate axis direction along the machining surface. 0: [Method 1] On-axis point (+) 1: [Method 2] Latitude/Longitude 2: [Method 3] Latitude/Projection angle 3: [Method 4] Start point/End point 4: [Method 5] Indexing angle (Z axis direction only)
B, C, E, F, H, I	Coordinate axis direction setting (*1) (-99999.999 to 99999.999)

(*1) The setting details vary depending on the coordinate axis direction designation method (M address).

[M address: 0 (On-axis point (+))]

B, C, E: Coordinate value on X, Y, or Z axis

F to I: Vacuous

[M address: 1 (Latitude/Longitude)]

B: Latitude (θ_1)

C: Longitude (θ_2)

E to I: Vacuous

[M address: 2 (Latitude/Projection angle)]

B: Latitude (θ_1)

C: Projection angle (θ_2)

E to I: Vacuous

[M address: 3 (Start point/End point)]

B: Start point coordinate value (X)

C: Start point coordinate value (Y)

E: Start point coordinate value (Z)

F: End point coordinate value (X)

H: End point coordinate value (Y)

I: End point coordinate value (Z)

[M address: 4 (Indexing angle)]

B: 1st rotation angle (θ_1)

C: 2nd rotation angle (θ_2)

E to I: Vacuous

•Method 5 (indexing angle) in the coordinate axis direction designation method is only available in the Z axis direction.

If a command is issued to an axis other than the Z axis designated by the coordinate axis selection command (P0Ax), a program error (P423) will occur.

P0 A0 (Z/X axis)	P2 M4 setting causes an error. (Method 5 is not able to be selected on the 2nd axis.)
P0 A1 (Y/Z axis)	
P0 A2 (X/Y axis)	P1M4 or P2M4 setting causes an error. (Method 5 is not able to be selected.)

•For details on each of input data, refer to the instruction manual.



Operation example

This function enables the R-Navi setup parameters to be configured from a machining program. After the parameters have been configured from the program, you can check the values or select the machining surface from the setup screen.

Machining program

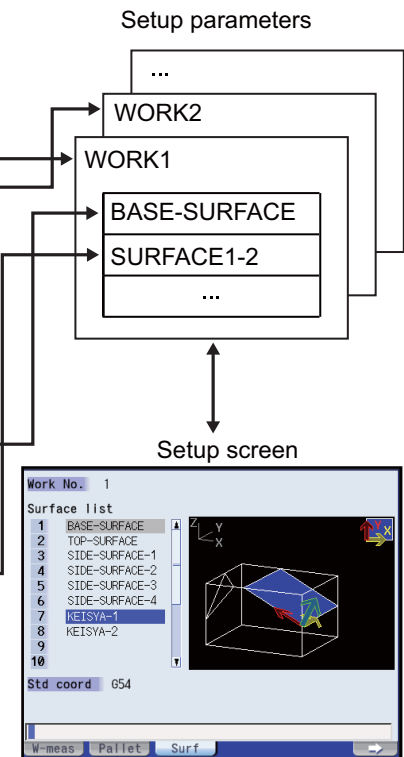
```

O200

Canceling the selected machining surface
G69;
(Setting of workpiece)
G10 L110;
Q1 <WORK1> F0 C0 R1 X50.0 Y30.0 Z20.0 I0.0 J0.0 K0.0 ;
Q2 <WORK2> F0 C0 R5 X50.0 Y30.0 Z20.0 I-10.0 J-5.0 K-40.0 ;
...
G11;

(Setting the machining surface)
G10 L111;
(Workpiece : 1, surface : 2)
P0 Q1 D2 <SURFACE1_2> X50.0 Y-30.0 Z-20.0 A0;
P1 M0 B1.0 C0.0 E1.732 F0.0 H0.0 I0.0;
P2 M1 B90.0 C0.0 E0.0 F0.0 H0.0 I0.0;
(Workpiece : 1, surface : 17)
P0 Q1 D17 <SURFACE1_17> X20.0 Y20.0 Z10.0 A0;
P1 M1 B0.0 C30.0 E0.0 F0.0 H0.0 I0.0;
P2 M0 B0.0 C30.0 E0.0 F0.0 H0.0 I0.0;
...
(Workpiece : 2, surface : 2)
P0 Q2 D2 <SURFACE2_2> X20.0 Y20.0 Z10.0 A0;
P1 M0 B1.0 C0.0 E1.732 F0.0 H0.0 I0.0;
P2 M1 B90.0 C0.0 E0.0 F0.0 H0.0 I0.0;
...
G11;

```



Restrictions

- (1) If the machining surface is selected or canceled while the block start interlock signal (*BSL) is turned OFF, an operation error (M01 0109) will occur. After this, if the block start interlock signal (*BSL) is turned ON, the machining surface is selected or canceled. The operation of the PLC signal depends on the MTB specifications.

15.10 Inputting the Tool Life Management Data; G10,G11

15.10.1 Inputting the Tool Life Management Data by G10 L3 Command; G10 L3,G11



Function and purpose

Using the G10 command (unmodal command), the tool life management data can be registered, changed and added to, and preregistered groups can be deleted.

Only group No. 1 can be used to register, change and add for the tool life management III.



Command format

Start of life management data registration

```
G10 L3;
P_L_Q_ ; (First group)
T_H_D_ ;
T_H_D_ ;
P_L_Q_ ; (Next group )
T_H_D_ ;
```

P	Group No.
L	Life
Q	Control method
T	Tool No. The spare tools are selected in the order of the tool Nos. registered here.
H	Length compensation No.
D	Radius compensation No.

Start of life management data change or addition

```
G10 L3 P1;
P_L_Q_ ; (First group)
T_H_D_ ;
T_H_D_ ;
P_L_Q_ ; (Next group )
T_H_D_ ;
```

P	Group No.
L	Life
Q	Control method
T	Tool No.
H	Length compensation No.
D	Radius compensation No.

Start of life management data deletion

G10 L3 P2;
 P_ ; (First group)
 P_ ; (Second group)

P Group No.

End of life management data registration, change, addition or deletion

G11 ;



Detailed description

Command range

Item		Command range
Group No.	(Pn)	1 to 99999999 (Only group No. 1 can be used for the tool life management III)
Life	(Ln)	0 to 65000 times (No. of times control method) 0 to 4000 minutes (time control method)
Control method	(Qn)	1 to 3 1: Number of mounts control 2: Time control 3: Number of cutting times control
Tool No.	(Tn)	1 to 99999999
Length compensation No.	(Hn)	0 to 999 (*)
Radius compensation No.	(Dn)	0 to 999 (*)

(*) The setting range of the tool compensation No. differs according to the specification of the "number of tool offset sets".

If a value exceeding each command range is issued, a program error (P35) will occur.



Operation example

	Program example	Operation
Data registration	G10 L3 ; P10 L10 Q1 ; T10 H10 D10 ; G11 ; M02 ;	<ol style="list-style-type: none"> 1. After deleting all group data, the registration starts. 2. Group No. 10 is registered. 3. Tool No. 10 is registered in group No. 10. 4. The registration ends. 5. The program ends.
Group change, addition	G10 L3 P1 ; P10 L10 Q1 ; T10 H10 D10 ; G11 ; M02 ;	<ol style="list-style-type: none"> 1. Changing and addition of the group and tool starts. 2. The change and addition operation takes place in the following manner. <ul style="list-style-type: none"> (1)When group No. 10 has not been registered. <ul style="list-style-type: none"> •Group No. 10 is additionally registered. •Tool No. 10 is registered in group No. 10. (2)When group No. 10 has been registered, but tool No. 10 has not been registered. <ul style="list-style-type: none"> •Tool No. 10 is additionally registered in group No. 10 (3)When group No. 10 and tool No. 10 have been both registered. <ul style="list-style-type: none"> •The tool No. 10 data is changed. 3. The group and tool change and addition ends. 4. The program ends.
Group deletion	G10 L3 P2 ; P10 ; G11 ; M02 ;	<ol style="list-style-type: none"> 1. The group deletion starts. 2. The group No. 10 data is deleted. 3. The group deletion ends. 4. The program ends.

15.10.2 Inputting the Tool Life Management Data by G10 L30 Command; G10 L30,G11

**Function and purpose**

Using the G10 command (unmodal command), the tool life management data can be registered, changed and added to, and preregistered groups can be deleted.

Only group No. 1 can be used to register, change and add for the tool life management III.

To specify additional compensation amount or direct compensation amount by control method, the length compensation and diameter compensation can be registered/changed with the tool compensation amount format.

**Command format****Start of life management data registration**

```
G10 L30 ;
P_L_Q_ ; (First group)
T_H_R_ ;
T_H_R_ ;
P_L_Q_ ; (Next group )
T_H_R_ ;
```

P	Group No.
L	Life
Q	Control method
T	Tool No. The spare tools are selected in the order of the tool Nos. registered here.
H	Length compensation No. or length compensation amount
R	Radius compensation No. or radius compensation amount

L_, Q_, T_, H_, and R_ cannot be omitted. If omitted, a program error (P33) occurs.

Start of life management data change or addition

```
G10 L30 P1;
P_L_Q_ ; (First group)
T_H_R_ ;
T_H_R_ ;
P_L_Q_ ; (Next group )
T_H_R_ ;
```

P	Group No.
L	Life
Q	Length compensation data format, radius compensation data format, control method
T	Tool No.
H	Length compensation No. or length compensation amount
D	Radius compensation No. or radius compensation amount

L_, Q_, T_, H_, and R_ cannot be omitted. If omitted, a program error (P33) occurs.

Start of life management data deletion

```
G10 L30 P2;
P_ ; (First group)
P_ ; (Second group)
```

P	Group No.
---	-----------

End of life management data registration, change, addition or deletion

G11 ;



Detailed description

Command range

Item		Command range
Group No.	(Pn)	1 to 99999999 (Only group No. 1 can be used for the tool life management III)
Tool No.	(Tn)	1 to 99999999
Control method	(Qabc)	abc: Three integer digits a. Tool length compensation data format 0: Compensation No. 1: Incremental value compensation amount 2: Absolute value compensation amount b. Tool radius compensation data format 0: Compensation No. 1: Incremental value compensation amount 2: Absolute value compensation amount c. Tool management method 0: Usage time 1: Number of mounts 2: Number of usages
Life	(Ln)	Refer to the table below.
Length compensation (No./amount)	(Hn)	
Radius compensation (No./amount)	(Rn)	

Control method (Qabc)			Length compensation (Hn)	Radius compensation (Rn)	Life (Ln)
a	b	c			
x	x	0	-	-	0 to 4000 (Usage time)
x	x	1	-	-	0 to 65000 (Number of mounts)
x	x	2	-	-	0 to 65000 (Number of usages)
x	0	x	-	0 to 999 (Compensation No.) (*1)	-
x	1	x	-	±99999.999 (Incremental value compensation amount)	-
x	2	x	-	±99999.999 (Absolute value compensation amount)	-
0	x	x	0 to 999 (Compensation No.) (*1)	-	-
1	x	x	±99999.999 (Incremental value compensation amount)	-	-
2	x	x	±99999.999 (Absolute value compensation amount)	-	-

(*1) The setting range of the tool compensation No. differs according to the specification of the "number of tool offset sets".

If a value exceeding each command range is issued, a program error (P35) will occur.



Operation example

	Program example	Operation
Data registration	G10 L30 ; P10 L10 Q001 ; T10 H10 R10 ; G11 ; M02 ;	<ol style="list-style-type: none"> 1. After deleting all group data, the registration starts. 2. Group No. 10 is registered. Tool management method is number of mounts Compensation No. method is applied to tool length compensation and tool radius compensation. 3. Tool No. 10 is registered in group No. 10. 4. The registration ends. 5. The program ends.
Group change, addition	G10 L30 P1 ; P10 L10 Q122 ; T10 H0.5 R0.25 ; G11 ; M02 ;	<ol style="list-style-type: none"> 1. Changing and addition of the group and tool starts. 2. The change and addition operation takes place in the following manner. <ol style="list-style-type: none"> (1) When group No. 10 has not been registered. <ol style="list-style-type: none"> (a) Group No. 10 is additionally registered. About the change and addition tool; <ul style="list-style-type: none"> ♦Tool management method: number of usages ♦Tool length compensation: the incremental value compensation amount method ♦Tool radius compensation: the absolute value compensation amount method. (b) Tool No. 10 is registered in the group No. 10 (c) "0.5" is registered to the length data which the compensation No. of the tool length shows and "0.25" is registered to the radius data which the compensation No. of the tool radius shows. (2) When group No. 10 has been registered, but tool No. 10 has not been registered. <ul style="list-style-type: none"> ♦Tool No. 10 is additionally registered in group No. 10 (3) When group No. 10 and tool No. 10 have been both registered. <ul style="list-style-type: none"> ♦The tool No. 10 data is changed. 3. The group and tool change and addition ends. 4. The program ends.
Group deletion	G10 L30 P2 ; P10 ; G11 ; M02 ;	<ol style="list-style-type: none"> 1. The group deletion starts. 2. The group No. 10 data is deleted. 3. The group deletion ends. 4. The program ends.

15.10.3 Precautions for Inputting the Tool Life Management Data



Relationship with other functions

- (1) During the following operations, the tool usage data will not be counted.
- ♦Machine lock
 - ♦Auxiliary axis function lock
 - ♦Dry run
 - ♦Single block
 - ♦Skip



Precautions

- (1) The tool life data is registered, changed, added to or deleted by executing the program in the memory or MDI mode.
- (2) The group No. and tool No. cannot be commanded in duplicate. The program error (P179) will occur.
- (3) When two or more addresses are commanded in one block, the latter address will be valid.
- (4) If the life data (L_) is omitted in the G10L3 command, the life data for that group will be "0".
- (5) If the control method (Q_) is omitted in the G10L3 command, the control method for that group will follow the base specification parameter "#1106 Tcount".
Note that when carrying out the No. of cutting times control method, command the method from the program.
- (6) If the control method (Q_) is not designated with 3-digit by G10 L30 command, the omitted high-order are equivalent to "0".
Therefore, "Q1" is equivalent to "Q001", and "Q12" is equivalent to "Q012".
- (7) If the length compensation No. (H_) is omitted in the G10L3 command, the length compensation No. for that group will be "0".
- (8) If the radius compensation No. (D_) is omitted in the G10L3 command, the radius compensation No. for that group will be "0".
- (9) Programming with a sequence No. is not possible between G10 L3 or G10 L30 and G11. The program error (P33) will occur.
- (10) If the usage data count valid signal (YC8A) is ON, G10 L3 or G10 L30 cannot be commanded. The program error (P177) will occur.
- (11) The registered data is held even if the power is turned OFF.
- (12) When G10 L3 or G10 L30 is commanded, the commanded group and tool will be registered after all of the registered data is erased.
- (13) The change and addition conditions in the G10 L3 P1 or G10 L30 P1 command are as follows:
- (a) Change conditions
- ♦Both the commanded group No. and tool No. are registered.
Change the commanded tool No. data.
- (b) Additional conditions
- ♦Neither the commanded group No. nor tool No. is registered.
Additionally register the commanded group No. and tool No. data.
 - ♦The commanded group No. is registered, but the commanded tool No. is not registered.
Additionally register the commanded tool No. data to the commanded group No.
- (14) When the tool No. is newly registered with G10L30 command, the same length compensation No. and radius compensation No. as the tool No. are automatically allocated.
When the tool which is outside the compensation No. range is newly registered, the length compensation No. and radius compensation No. of that tool are "0".
By selecting the compensation No. with the management method, the length compensation No. and radius compensation No. can be changed to arbitrary Nos.
- (15) G10L3 command/G10L30 command cannot be used for the tool life management I. If commanded, the program error (P39) occurs.
- (16) The setting range of the tool compensation No. depends on the MTB specifications.
- (17) Only group No. 1 can be used to register, change and add for the tool life management III.

15.10.4 Allocation of the Number of Tool Life Management Sets to Part Systems



Function and purpose

The number of tool life management sets can be set per part system.

This function is divided into following methods and which one is used depends on the MTB specifications (parameters "#1439 Tlife-SysAssign", "#12055 Tol-lifenum").

Arbitrary allocation: Arbitrarily allocates the number of tool life management sets to each part system.

Fixed allocation: Automatically and evenly allocates the number of tool life management sets to each part system. The arbitrary allocation enables the efficient allocation because when a certain part system needs only a small number of tool life management sets, the rest can be allocated to another part system. If an auxiliary-axis part system does not need the tool life management sets at all, the number of tool life management sets can be set to "0" for the auxiliary-axis part system.

Subsequent description is an example in the case where the number of tool life management sets in the system is 999 sets.

(1) Arbitrary allocation (with #1439=1)

The number of sets allocated to each part system depends on the MTB specifications (parameter "#12055 Tol-lifenum").

The following example shows the number of tool offset sets allocated when the lathe system is a 4-part system.

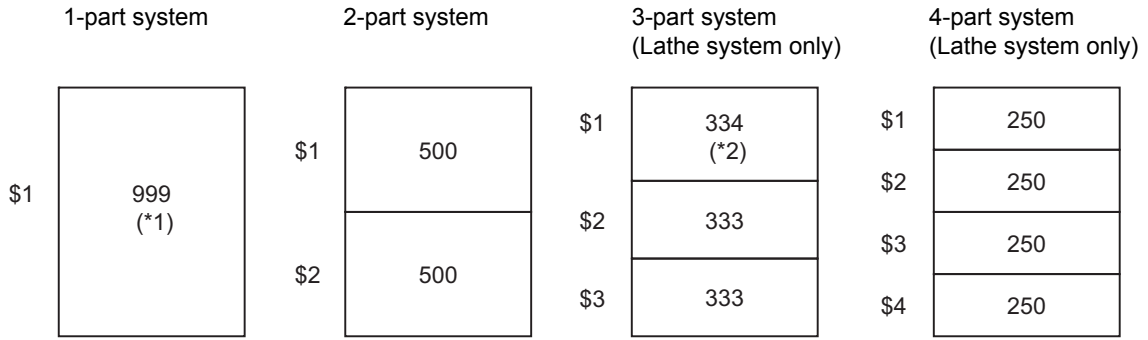
(a) When the number of tool life management sets is increased for the 1st part system (\$1) of 4-part system

\$1	250	➔	\$1	400
\$2	250		\$2	200
\$3	250		\$3	200
\$4	250		\$4	200

(b) When the number of tool life management sets is set to "0 sets" for the 3rd part system (\$3) of 3-part system to use that part system as an auxiliary-axis part system

\$1	334	➔	\$1	500
\$2	333		\$2	500
\$3	333		\$3	0

(2) Automatic and even allocation (with #1439=0)



(*1)The maximum number of tool life management sets per part system is 999.

(*2) If there is any remainder, the remainder is allocated to the 1st part system.



Precautions

- (1) The maximum number of tool life management sets for 1-part system is 999.
- (2) For 1-part system, up to the number of tool life management sets in the system is available regardless of the parameter setting.
- (3) When the value of the parameter "#12055 Tol-lifenum" is equal to or lower than the number of tool life management sets in the system, the remainder is not allocated to any part system even if the specification allows arbitrary allocation.
- (4) When the value of the parameter "#12055 Tol-lifenum" is equal to or lower than the number of tool life management sets in the system, system alarm (Y05) is generated even if the specification allows arbitrary allocation.
- (5) Even if the specification allows arbitrary allocation, fixed allocation is applied if the parameter is "#12055 Tol-lifenum"= "0" for all part systems.
- (6) When entering data into the tool life management file, if the number of tool life management data exceeds that of current tool life management sets, the excess tool life management data cannot be entered.

15.11 Interactive Cycle Insertion; G180

15.11.1 Interactive Cycle Insertion



Function and purpose

The machining and setup support cycles can be interactively inserted to a program which is opened on the edit screen.

Using this function leads the programming time to be shortened.

The cycle can be easily inserted by editing the data on the interactive window.

The block of the cycle once inserted to the program can be directly edited in the edit screen. As long as the cycle format is not changed, the program can be reedited in the cycle edit window.



Command format

G180 P__ A__ ;

P	Cycle information identification No. 1: Cycle header 0: Cycle footer 11: Arbitrary shape header 10: Arbitrary shape footer 31: Hole position header 30: Hole position footer
A	Cycle ID (Only when the cycle information identification No. is set to "1")

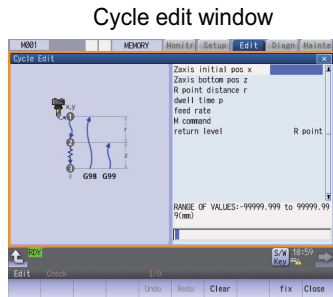
Blocks between cycle header block (G180 P1) and cycle footer block (G180 P0) are handled as blocks of interactive cycle insertion.

G180 is a G code in group 0, and an unmodal command.



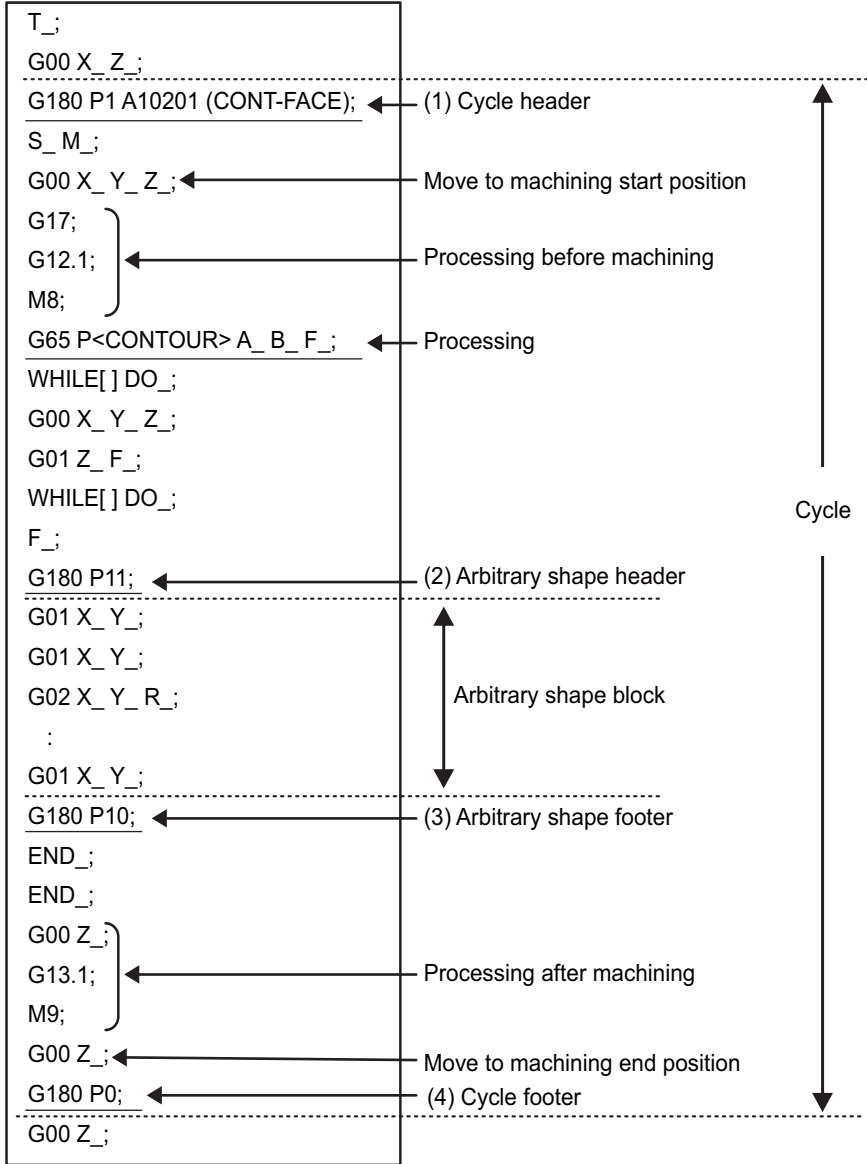
Detailed description

Program format of cycle inserted with this function is indicated as follows:



Edit the cycle data interactively and output to the program.

Program being edited on the edit screen



No.	Process part	Description	Program image
(1)	Cycle header	Block that indicates the cycle starting. Header includes cycle ID (8-digit number) and cycle name. <Cycle ID> Used to identify the cycle type at the time of cycle reediting. <Cycle name> Used to show cycle name in the monitor screen or check screen.	G180 P1 A_ (cycle name); A: Cycle ID
(2)	Arbitrary shape header	The header that indicates the start of the arbitrary shape is output before an arbitrary shape block.	G180 P11;
(3)	Arbitrary shape footer	The footer that indicates the end of the arbitrary shape is output at the end of an arbitrary shape block.	G180 P10;

No.	Process part	Description	Program image
(4)	Cycle footer	The footer that indicates the end of the cycle is output at the end of the cycle.	G180 P0;
-	Hole position header	The header that indicates the start of the hole position is output before the block to specify the hole position.	G180 P31;
-	Hole position footer	The footer that indicates the end of the hole position is output after the block to specify the hole position.	G180 P30;

Note

- (1) The program block output between header and footer differs for each cycle.
- (2) G180 block does not perform since it is the block to identify the cycle information. The operation is same as the block which only has EOB (;). The operation will also be the same when specified other than cycle information identification No. designated by format (G180 P99 etc.).
When the specifications of interactive cycle insertion are invalid, it will result the program error (P39) at the G180 block.
- (3) Since G code of G180 is inserted automatically, manual input is not required.

15.11.2 Interactive Macro



Function and purpose

Interactive macro means a macro program used for interactive cycle insertion. It is stored in the dedicated area. The command format is the same as when an interactive cycle is inserted. Refer to "15.11.1 Interactive Cycle Insertion".



Detailed description

A macro call command during a cycle inserted by this function calls the interactive macro stored in the interactive macro area.

You cannot edit the interactive macros which are stored in this area.

However, if you set "1" to "#8133 Intrctv macro call", a machining program in the machining program area can be called as a macro program. You can newly create and edit the program because it is in the machining program area. When you want programs to be operated differently from the standard interactive macro, you can edit and call them there.

Parameters	Program area	Program display	ONB display	Buffer correction	
#8133	0	Interactive macro area	Hide	Hide	Disabled
	1	Machining program area (*1)	Show	Show	Enabled
		Interactive macro area (*2)	Hide	Hide	Disabled

(*1) When a program with the same name as standard interactive macro exists in the machining program area, the program in the machining program area is called first.

(*2) When a program with the same name as a standard interactive macro does not exist in the machining program area, the program in the interactive macro area is called.



Precautions and restrictions

- (1) Do not change the name of macro program for macro call command. If you change the name to other than standard interactive macro program name, the program error (P232) occurs.
When you change it to the standard interactive macro program name, you can call the interactive macro program which has the name you have changed; however, the cycle cannot be reedited.
- (2) Do not add macro call (G65) or subprogram call (M98) during the cycles (G180P1 to G180P0). If you add the macro call other than the standard interactive macro programs, the program error (P232) occurs.
When adding with the standard interactive macro program name, you can call the interactive macro you have added; however, the cycle cannot be reedited if you insert a block.
- (3) Do not add the "macro interruption command (M96/M97)" during the cycles (G180P1 to G180P0). If you add, the program error (P232) occurs to call the macro in the interactive macro area. Once "1" is set to the parameter "#8133 Intrctv macro call", a macro program or subprogram in the machining program area can be called even during a cycle; however, the cycle cannot be reedited if a block is inserted.
- (4) Subprogram and macro program call nesting levels include interactive macros. The maximum nesting level in which a macro program or subprogram can be called depends on your CNC specifications.
- (5) For the programs inserted cycles using this function, if modifications that do not conform to the cycle format have been made manually, data may not be read properly at the time of reedit. In that case, even if the menu "Reedit" is pressed in the cycle list window, an error occurs, and the cycle cannot be reedited.
- (6) G180 block does not perform anything. Therefore, even if G180 block is added to the program manually, the error does not occur, and it handled as the same operation as the block only EOB (;).
- (7) In this function, saving cycles to the program is possible even if cycle data is unset state. The setting values of unset items are output as "0" or "?". If the program output "?" is operated, the program error (P33) occurs at the block which unset data is output.
- (8) Only the machining program, which contains the cycle of the type designated in the parameter "#8992 Cycle switch", is operable. If the machining program containing a different type of cycle is operated, a program error (P232) occurs in the block.

15.12 Axis Name Extension



Function and purpose

The axis name (command axis name) used for giving the absolute/incremental value command to NC control axis can be expanded to two characters. When this function is invalid, the command axis name (#1013 axname) is set with one character from A, B, C, U, V, W, X, Y or Z, therefore the number of axes is limited if the increment command axis name is used (*1).

When this function is valid, the incremental axis name can be used for all axes by this function.

The name-extended axis cannot be designated in the parameter which sets the command axis name such as plane configuration axis I, J or K (*2). Thus, apply this function to a miscellaneous axis which is not used for machining (cutting).

(*1) When two alphabetical characters are used per axis.

(*2) This indicates the axis names of I, J or K set by the parameters "#1026 base_I" to "#1028 base_K".

[Use example]

<Axis name setting>

The settings of these parameters depend on the MTB specifications.

No.	Item		1st axis	2nd axis	3rd axis	4th axis	5th axis	6th axis	7th axis
1013	axname	Axis name	X	Z	C	X	Z	X	Z
1014	incax	Incremental value command axis name	U	W	H (*3)	U	W	U	W
1601	axnameEx	Axis name extension character	No	No	No	A	A	B	B
Absolute value command axis name			X	Z	C	XA	ZA	XB	ZB
Incremental value command axis name			U	W	H	UA	WA	UB	WB

(*3) "H" can be set to "incax".

Term

The following descriptions are the meanings of the terms used in this manual.

Term	Meaning
Name-extended axis	Axis of which the command axis name is specified with two characters by this function
Name-unextended axis	Axis of which the command axis name is specified with a single character (the axis where this function is not used)
Axis name extension character	2nd character of the name-extended axis

Enabling conditions

In order to use this function, validate this function by the parameter and set the second character of the name-extended axis.

These parameters depend on the MTB specifications (parameters "#1266 ext02/bit0" and "#1601 axnameEx").



Detailed description

Program commands for axis name extension

(1) Relationship between parameter setting and command axis name

Relationship between parameter related to axis name and command axis name is as follows: When the first character is not set, the program command cannot be issued to that axis.

[Parameter configuration example]

#	Item		1st axis	2nd axis	3rd axis	4th axis
1013	axname	Axis name (*1)	X	Z	Z	Y
1014	incax	Incremental value command axis name (*1)	U	W	None	V
1601	axnameEx	Axis name extension character (*2)	A	A	B	None
Absolute value command axis name			XA	ZA	ZB	Y
Incremental value command axis name (*3)			UA	WA	None	V

(*1) This sets the first character of the axis name.

(*2) This sets the second character of the axis name.

(*3) This depends on the MTB specifications (available when the parameter "#1076 AbsInc" is set to "1").

(2) Program commands to name-extended axis

When the axis name of the name-extended axis is "XA", the program command format to the name-extended axis is as follows. When "X" and "XA" exist in the command axis name in the part system, "XA10000" in the command code example below is not interpreted as "X0 A10000" because "XA" is judged preferentially.

Type of command	Command code example
Numerical command	XA10000;
Decimal point command	XA12.345;
Variable command	XA[#100];

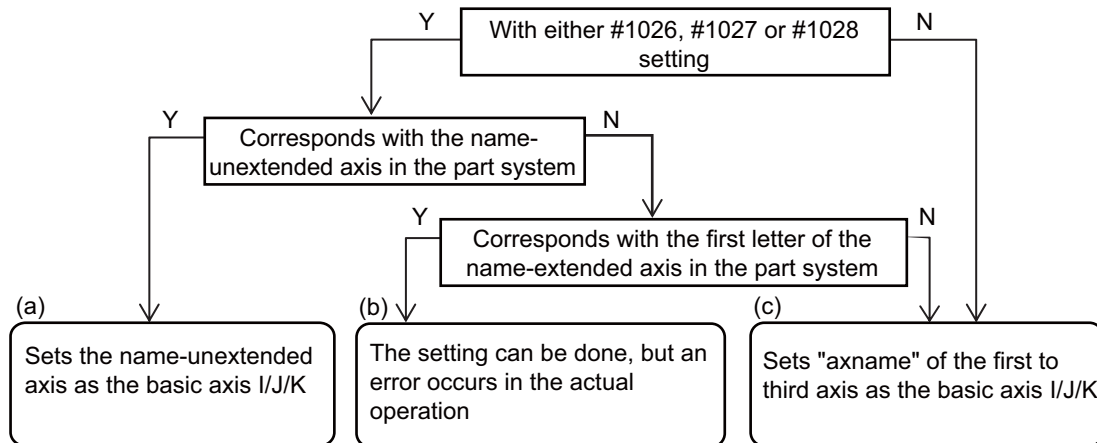
Relationship with axis name setting parameters

Name-extended axis cannot be designated because only one letter can be set to the parameter which sets axis name shown below. Thus, apply axis name extension to miscellaneous axis which is not used for machining (cutting).

#	Item	Description
1026	base_I	Name of base axis configuring plane (*1)
1027	base_J	
1028	base_K	
1029	aux_I	Name of axis parallel to "base_I"
1030	aux_J	Name of axis parallel to "base_J"
1031	aux_K	Name of axis parallel to "base_K"
8317	-	Name of delivery axis when the right chuck and tailstock barrier is movable.
8621	-	Axis name of the plane (horizontal axis) for coordinate rotation control
8622	-	Axis name of the plane (vertical axis) for coordinate rotation control

- (*1) Name-extended axis cannot be designated for parameters "#1026 base_I" to "#1028 base_K" (base axes I, J, and K); however, the NC operates as follows depending on the current setting of base axis I, J, or K:
- (a) When the setting value of the base axis I, J, or K corresponds to any name-unextended axis in the part system, the corresponded axis is identified as base axis I, J, or K.
 - (b) When the NC is operated with the setting value of the base axis I, J, or K as follows, the program error (P11) occurs.
 - It does not correspond with any of the name-unextended axes in the part system.
 - It corresponds with the first character of any of the name-extended axis.
 - (c) When the base axis I, J, or K is not set or in the following state, they are set as shown in the table below:
 - It does not correspond with any of the name-unextended axes in the part system.
 - It does not correspond with the first character of any of the name-extended axis.

Parameter of base axis I, J, or K	Which axis "# 1013 axname" is used for the base axes I, J, and K?	
	L system	M system
#1026 base_I	1st axis	1st axis
#1027 base_J	3rd axis	2nd axis
#1028 base_K	2nd axis	3rd axis



Relationship with arbitrary axis exchange control

Because in the environment where the arbitrary axis exchange control is available, only the name of name-unextended axis can be designated in the parameters "#12071 adr_abs[1]" to "#12078 adr_abs[8]", other axes cannot be assigned to the command axis name of name-extended axis. However, regardless of the setting of these parameters, you can designate the axis name of name-extended axis or use the name-extended axis as target axis for axis exchange. (For the name-unextended axis, the axis name cannot be designated without setting these parameters.)

(1) to (3) show the examples of command code in the case of the following axis configuration.

[Example of axis configuration]

\$1	1st axis	2nd axis	3rd axis	4th axis
#1013 axname	X	Z	X	Z
#1022 axname2	X1	Z1	X9	Z9
#1601 axnameEx	-	-	A	A
Command axis name	X	Z	XA	ZA

(1) Other axes cannot be assigned to the command axis name of name-extended axis.

G140 XA=X1;	Program error (P33)
-------------	---------------------

(2) Name-extended axis can be used as the target axis for axis exchange.

G140 X=X1 Z=Z9;	Assign "Z9 axis" to the command address Z.
-----------------	--

To return the command address to "ZA", carry out the axis exchange return with G141 or G142.

(3) You can command a name of name-extended axis regardless of G140 command.

G140 X=X1 Z=Z1; G00 X10. XA15.;	Both X1 and X9 axes move to the commanded coordinate.
------------------------------------	---

G codes which can use name-extended axis

The following list shows the G codes whose functions are available for the name-extended axis among the G codes using an axis name as argument. Also, the operations when the name-extended axis is commanded in the same block or in the G-code mode are shown.

(1) List of G codes that can be used for the name-extended axis (M system)

G code	Group	G code function	Operation when the name-extended axis is commanded in the same block or in the G-code mode
G00	1	Positioning	Move to the commanded coordinates
G01	1	Linear interpolation	Move to the commanded coordinates
G09	0	Exact stop check	Move to the commanded coordinates
G28	0	Automatic reference position return	Reference position return
G30	0	2nd, 3rd and 4th reference position return	Return to 2nd, 3rd and 4th reference positions
G53	0	Basic machine coordinate system selection	Move to the commanded machine coordinates
G54	12	Workpiece coordinate system selection 1	Move to the commanded coordinates on G54
G55	12	Workpiece coordinate system selection 2	Move to the commanded coordinates on G55
G56	12	Workpiece coordinate system selection 3	Move to the commanded coordinates on G56
G57	12	Workpiece coordinate system selection 4	Move to the commanded coordinates on G57
G58	12	Workpiece coordinate system selection 5	Move to the commanded coordinates on G58
G59	12	Workpiece coordinate system selection 6	Move to the commanded coordinates on G59
G54.1	12	Extended workpiece coordinate system selection	Move to the commanded coordinates on G54.1Pn
G60	00 (01)	Unidirectional positioning	Move to the commanded coordinates
G61	13	Exact stop check mode	Move to the commanded coordinates
G160	0	Torque limitation skip	Move to the commanded coordinates



Relationship with other functions

Spindle position control (Spindle/C axis control)

The name of C axis which performs spindle/C axis control cannot be extended. When the name of C axis is set as name-extended axis, the program error (P11) occurs at the start of the automatic operation.

Relationship with data input by program

(1) G10 command in the machining program

Data input can be performed to the commands which designate the data input target axis by the axis number; however, it cannot be performed to the commands which designate by the axis name. When the name-extended axis is designated, the program error (P33) occurs.

(a) Commands to which the data input of name-extended axis is disabled (when designating by axis number)

(b) Commands to which the data input of name-extended axis is disabled (when designating by axis name)

(c) Command where input data is not per axis (when the axis is not designated)

(a)		(b)		(c)			
G10 L70	Parameters	G10 L2	Workpiece offset	G10 L100	Tool shape for 3D check		
		G10 L20	Extended workpiece offset	G10 L10	Tool length shape compensation (*2)		
		G10 L10	Tool length shape compensation (*1)	G10 L11	Tool length wear compensation (*2)		
		G10 L11	Tool length wear compensation (*1)	G10 L12	Tool radius shape compensation		
		G10 L14	Current limit	G10 L13	Tool radius wear compensation		
						G10 L3	Tool life management
						G10 L30	Tool life management
						G10 I_J_K_	Coordinate Rotation Parameter

(*1) Lathe system

(*2) Machining center system

(2) G10 command in input/output file

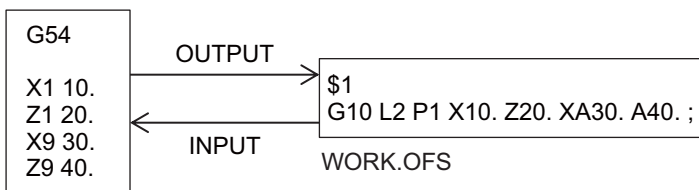
The input/output can be performed for the workpiece offset (G10 L2/L20) and L system tool offset (G10 L10/L11) by G10 command written in the file (WORK.OFS, TOOL.OFS), and the data input/output for the name-extended axis can be performed by G10 command in this case.

The example shows the relationship between the NC internal data and the file contents in the case of the following axis configuration:

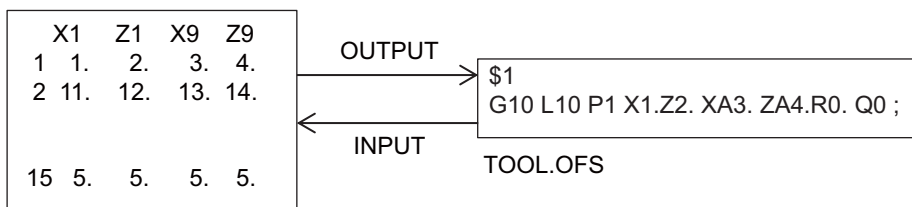
[Example of axis configuration]

\$1	1st axis	2nd axis	3rd axis	4th axis
#1013 axname	X	Z	X	Z
#1022 axname2	X1	Z1	X9	Z9
#1601 axnameEx	-	-	A	A
Command axis name	X	Z	XA	ZA

[Workpiece offset file (WORK.OFS)]



[Tool offset file (TOOL.OFS)]



R address: Tool nose radius compensation amount

Q address: Tool nose point P number



Precautions

- (1) When the second axis name parameter "#1022 axname2" is not set, the command axis name is set automatically at the time of power ON.

Name-extended axis	The axis names after the extension (1st character: "#1013 axname", 2nd character: "#1601 axnameEx") is set.
Name-unextended axis	The axis name set in "#1013 axname" is set.

- (2) When judging a character string described in the machining program, and if the result after analyzing the character string from the top is a reserved word (*1) of the user macro, it is identified as reserved word. When the character string is not the reserved word, it is identified as an axis name, but the name of the name-extended axis is identified preferentially. When the axis name and the reserved word are written in a row, enclose the macro command in "[]" and do not omit the axis command value "0" so that it does not become an unintentional command.

(*1) Reserved word here indicates as follows:

- ♦ Available functions during program mode (such as ABS and SIN)
- ♦ Control statement (such as IF and WHILE)
- ♦ Comparison operator (such as EQ and LT)

Case in which the following axis names exist	Run command	Operation
AB	#100 = ABS[#101];	Set to the ABS command of a macro. (This is not regarded as "#100 = AB0 S[#101];".)
AB, XA	XA[ABS[#100]];	The ABS command result of a macro is used as the command value of the XA axis.
	XAABS[#100];	Same as above (This is not regarded as "XA0 AB0 S[#100];".)
X, XA	XABS[#100];	This is regarded as "XA0 B0 S[#100];". (If the B axis does not exist, a program error (P32) will occur.) When you want to use the ABS command result of a macro as the command value of the X axis, describe "X[ABS[#100]];".
AX	,AX100.;	This is regarded as ",A0 X100.;". (*)

(*) When the name of name-extended axis is written following comma ",", the command address with comma is identified preferentially.

- (3) In the following cases, the program error (P11) occurs because the axis names are duplicated:
- ♦ When the name "#1013 axname" of name-unextended axis in the part system is duplicated
When "#1076 AbsInc" = "1", the duplication check is performed including the increment command axis name "#1014 incax".
 - ♦ When the name of name-extended axis in the part system (1st letter: "#1013 axname", 2nd letter: "#1601 axnameEx") is duplicated
When "#1076 AbsInc" = "1", the duplication check is performed including the increment command axis name (1st character: "#1014 incax", 2nd character: "#1601 axnameEx").
- (4) When the name-unextended axis is configured following the name-extended axis in the part system, the system error (Z100) occurs at the time of power ON.

\$1	Normal axis configuration				Abnormal axis configuration (System error (Z100) occurs.)			
	1st axis	2nd axis	3rd axis	4th axis	1st axis	2nd axis	3rd axis	4th axis
#1013 axname	X	Z	X	Z	X	X	Z	Z
#1601 axnameEx	-	R	A	A	-	A	-	A
Command axis name	X	Z	XA	ZA	X	XA (*)	Z	ZA

(*) This is incorrect axis configuration because XA axis is set before Z axis.

- (5) You cannot input "#1601 axnameEx" by the parameter input by program (G10 L70). When designated, the program error (P421) occurs.

15.13 Machining Interruption [C80]; G26

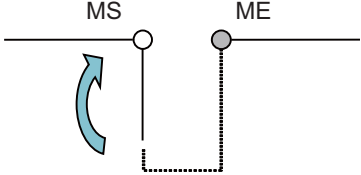
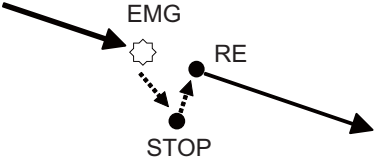


Function and purpose

Machining interruption is a function which enables interrupt operations in the table below while a program is normally executed.

Interruption programs (called "retraction programs") need to be prepared in advance to validate various interruption operations.

This function is available only during memory mode operation, but not during MDI operation.

Interrupt operation	Details of operation	Purpose
<p>Program-based retraction</p>	<p>When the retract button is pressed, the program immediately branches to the predetermined retraction program. A safe tool-path that avoids interference can be programmed in the retraction program. On returning from the retraction program, the tool returns to the programmed position called "machining start point", from which the operation can be restarted.</p>  <p>Returns to the machining start point from the status of machining in progress.</p> <p>MS: Machining start point ME: Machining end point</p>	<p>e.g. when machining is temporarily suspended due to events such as when tools break, and then restarted after the issue is solved.</p>
<p>Emergency stop</p>	<p>When an emergency stop occurs during program execution, machining stops in a feed hold status. When the NC is restarted after the cancellation of an emergency stop, a tool returns from the stop position (caused by the emergency stop) to the interpolation restart position, and then operation restarts along the original program path.</p>  <p>EMG: Emergency stop STOP: Stop position RE: Interpolation restart position</p> <p>Returns to the interpolation interrupted position from the emergency stop position as shown above, then the interpolation restarts.</p>	<p>e.g. when machining is temporarily suspended due to such as the emergency stop.</p>

Term

Term	Meaning
Machining start point	Starting point of a single machining module. The program returns to this block after the retraction program is executed.
Machining end point	Ending point of a single machining module, which is paired with the machining start point. Program-based retraction is possible in a section from the machining start point to the machining end point.
Process start point	Starting point of machining process by a single tool. Indicates the tool change command block.
Selected point	Indicates one of the machining start point, machining end point, and process start point.
Retraction program	Program to retract the tool safely during each process. Executed when retraction is started.
Machining interruption program	Program (retraction program) that is started by the interruption operation.
Slide amount	Distance from the current machine position to the output position obtained by interpolation calculation.
Slide-movement	Movement (equivalent to the slide amount) from the current machine position to the output position obtained by the interpolation calculation. This movement is conducted automatically when the program is restarted after an emergency stop.
Slide speed	Speed to perform slide-movement operation. This setting depends on the MTB specifications (parameter "#12125 slide-F").



Command format

To perform an interruption operation such as a program-based retraction, add the dedicated commands to the program in advance.

Machining start point and machining end point

Program a mark that indicates both the machining start point and machining end point.

,Qqqqqq;	Machining start point
,Q0;	Machining end point

,Q	For "qqqqq", command the sequence No. of a retraction program. When ",Q0" is specified, it is judged to be the machining end point.
----	--

- Command the machining start point and machining end point in an independent block.
- If a command other than "N" (sequence No.) is included in the block that has the machining start point and machining end point specified, a program error (P33) occurs.
- After commanding the machining start point, always specify the machining end point. If the machining start point is commanded while the machining end point remains unspecified, a program error (P727) occurs.

Process start point

The tool change command is handled as the process start point.

```
T** M06;
```

- ♦ "M06" refers to the miscellaneous function code for the tool change command.
- ♦ The M code of the miscellaneous command for tool change, which is assumed to be the process start point, depends on the MTB specifications (parameter "#12126 Mcngit_Tch_M").

Note

- ♦ When the stop code (T03 0320) (stop at the selected point) is issued at the process start point, it indicates that the tool change command has not yet been executed. The tool change command is executed when the automatic operation is activated.

Retraction program

Create a retraction program in the same program following the machining program.

A single retraction program begins with the sequence No. that matches "qqqq" (setting value of ",Q" address) of the machining start point.

Nqqqq;	Mark that indicates the head of a retraction program. ("qqqq" indicates the sequence No.)
G26 Z0.; :	Selected point return command/tapping retract command
M99;	Retraction program end command

- ♦ The retraction program is configured in the standard subprogram format.
- ♦ The G26 (selected point return/tapping retract) command is available in the retraction program.
- ♦ To end the retraction program, command M99.
- ♦ Issue the M99 command in an independent block. If a command other than "N" (sequence No.) is issued in the same block, a program error (P33) occurs.
- ♦ The retraction programs are required as many programs as the number of the sequence Nos. specified in ",Q".

G26 (Selected point return/tapping retract)

The G26 command returns the axis commanded in the same block to the selected point that was executed previously. (*1)

The G26 command is available only in the machining interruption (retraction) program. If the command is issued in a program other than the machining interruption program, a program error (P721) occurs.

G26 X0. Y0. Z0.;	Selected point return
-------------------------	------------------------------

- The specified axis is returned to the selected point that was executed previously.
- The specified coordinate position is ignored. In normal circumstances, specify "0".
- The rotating type rotary axis moves with a shortcut even if the parameter "#8213 Rotation axis type" is set to "Short-cut invalid" (0).
- On returning to the selected point, linear interpolation is applied with the feedrate of the F modal value.

(*1) If retraction start is performed while the tapping cycle is running, only the first G26 command in the retraction program is operated as the tapping retract command, not the selected point return command. (The second and subsequent G26 commands are operated as the selected point return command.)

G26 Zz1.;	Tapping retract
------------------	------------------------

Zz1	Commands the pull-up position (*1) with the absolute value or the incremental value from point R (*2). When the hole drilling axis is a diameter axis, command the diameter value even in the incremental value command.
-----	---

(*1) Do not command a position under point R (*2) (in the hole bottom direction). If commanded, a program error (P730) occurs.

(*2) When point R is not specified in the tapping cycle, the initial point is assumed to be specified.

- Pull up the hole drilling axis to the commanded position.
- The hole drilling axis moves only when retraction start is performed during cutting or dwelling in the tapping cycle. If retraction start is performed in other states, the hole drilling axis does not move.
- Only the hole drilling axis can be commanded in the same block as the tapping retract command. In addition, the S code cannot be commanded. When an axis other than the hole drilling axis or the S code is commanded, a program error (P33) occurs.
- Do not perform axis movement in the process from the head of the retraction program to the tapping retract command. In addition, when the currently executed tapping cycle is in synchronous tapping mode, do not issue the S command. (If commanded, a program error (P729) occurs.)

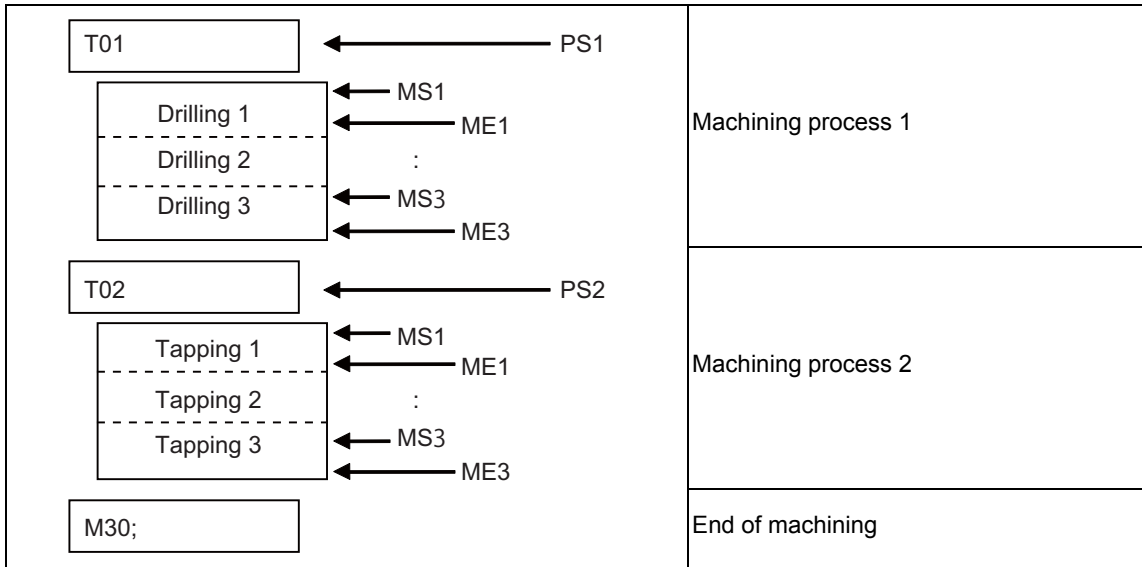


Detailed description

Machining start point and machining end point, Process end point

In a general machining program, a single machining process consists of one or more machining processes (for example, multiple hole drilling) by one tool. A machining program consists of such machining processes that are described in sequence.

A machining process begins with the tool change command. This position on the program is referred to as "process start point".



MS: Machining start point
 ME: Machining end point
 PS: Process start point

Operation with single block at selected point

"Operation with single block at selected point" refers to an operation in which the machining stops at the machining start point, machining end point, or process start point.

When the "operation mode with single block at selected point" signal is set to ON, the operation with single block at selected point is performed.

"Stop at selected point" refers to an operation that the machining stops at the machining start point, machining end point, or process start point. In this case, the stop code is T320.

The following types of selected point stop states exist.

Various interruption operations are possible in the selected point stop state.

Block where "Stop at selected point" is applied	Available interruption operation
,Qqqqqq;	-
,Q0;	Program-based retraction
M06; (M system) T****; (L system)	-

Note

- (1) Macro single ON is set during the operation mode with single block at selected point. (The machining stops at the macro block during single-block operation.)

Stop at selected point

While "Stop at selected point" is active, the machining will stop before executing the block to be stopped at. At this time, a signal indicating each stop state is output. ("Stopping at machining start point", "Stopping at machining end point", or "Stopping at process start point" signal)

When cycle start is restarted from this stop state, machining is restarted from a command in the stopped block.

Program-based retraction

Program-based retraction is available only when the "retraction executable" signal is set to ON (*1). When the "retraction start" signal is set to ON while the "retraction executable" signal is ON, the program branches immediately to the sequence No. indicated by the address Q (the head of the retraction program) that is commanded at the machining start point.

(*1) The operation (the condition to be "ON", etc.) of the "retraction executable" signal or "retraction start" signal depends on the MTB specifications.

For the retraction program, describe a program to retract the tool safely during each process.

In the retraction program, the G26 command can be used to return the commanded axis to the machining start point. In addition, if retraction start is performed while the tapping cycle is running, the G26 command operates as a tapping retract command to pull up the hole drilling axis to the specified position.

The retraction program ends with "M99;" in the same way as for a normal subprogram. In this case, the axis that is not returned to the machining start point automatically returns to the machining start point.

The return to the machining start point depends on the command type:

- For the selected point return command (G26), linear interpolation is applied.
- For the retraction program end command (M99), the travel varies depending on the modal of the currently executed G code (group 1).

(When G00 is commanded, the axis travels in rapid traverse mode; otherwise, it travels in linear interpolation mode.)

In addition, when G26 is operated as the tapping retract command, the hole drilling axis travels in linear interpolation mode. However, when tapping retract is performed while synchronous tapping cycle is running, the spindle rotates synchronously with the movement of the hole drilling axis.

When the retraction program ends, the modal is restored to the state that is set at the machining start point. Each axis stops at the machining start point block after returned to the machining start point. If cycle start is performed in this block stop state, the commanded rotation speed and commanded signal (forward rotation start/reverse rotation start, etc.) of the spindle are restored to the state that was set at the machining start point.

- (1) When reset is performed while the retraction program is running, the currently executed program ends. In addition, if the "reset 2" signal is input, the program is executed from the machining start point at the next cycle start operation.
- (2) If an MDI interruption is conducted, the retraction executable state is released. An MDI interruption can not be conducted while the retraction program is running.
- (3) When retraction start is performed while the miscellaneous function is running (in the state in which the corresponding strobe signal is set to ON and waiting the FIN signal), the strobe signal is set to OFF, and also the FIN signal wait state is canceled.
- (4) Tool radius compensation is canceled when the retraction program is started.
Command the tool radius compensation after the machining start point, and cancel it before the machining end point.
- (5) When the retraction program is executed, the program nesting level is incremented by one in the same way as subprogram calling. Therefore, the retraction program cannot be started if the nesting level has reached the maximum.
- (6) The operation at the start of the retraction program conforms to that at the start of macro interruption. However, the interruption method is fixed to Type 1 (operation to immediately stop the currently executed block and start the retraction program).

- (7) To return each modal to the state that was set at the machining start point when the retraction program is ended (M99 is executed), the machining operates as shown below.

[G96 mode at machining start point]

The spindle stops with "S0" just before the retraction program ends.

[G97 mode at machining start point]

When the following multiple conditions are specified together, the rotation speed may increase.

- ♦The constant surface speed control axis is near the center of the spindle.
- ♦The G96 modal and surface speed were recovered when the retraction program was ended.

Therefore, retract the constant surface speed control axis to the position at which the following conditions are satisfied before the retraction program is ended.

- ♦Position sufficiently far away from the center of the spindle
- ♦Position at which the axis does not pass near the center of the spindle when it moves to the machining start point

- (8) In the retraction executable state, control is moved to the retraction program by retraction start even when the block of timing synchronization between part systems is executed. When the machining of another part system executes a block late, the block is not completed. Do not perform timing synchronization operation between part systems in the retraction executable section (from the machining start point to the machining end point).
- (9) Do not perform the following operations related to the coordinate system in the retraction executable section (from the machining start point to the machining end point) and in the retraction program.
- ♦Local coordinate system setting
 - ♦Change of external workpiece coordinate system offset
 - ♦Coordinate system setting
 - ♦Workpiece coordinate system preset

Emergency stop

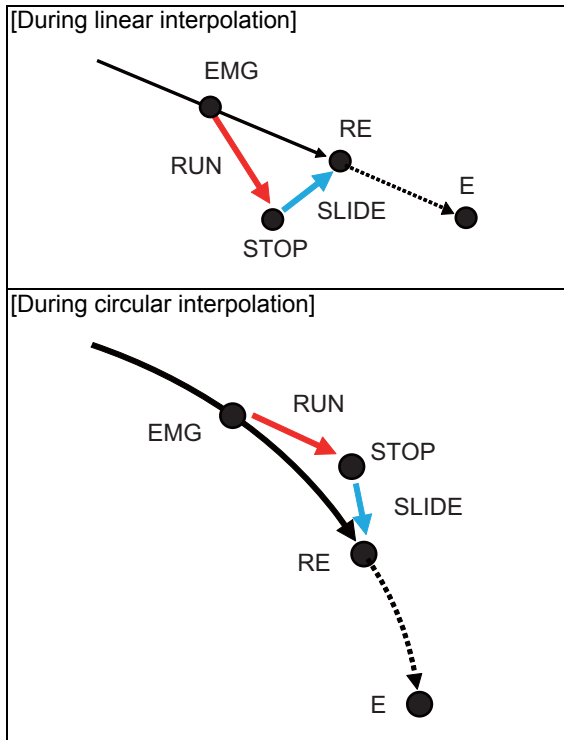
When the parameter "#11020 Mcngit_Spec/bit0" is set to "1" (Enable), operation will not be ended even if emergency stop occurs during automatic operation, which causes the machining to be placed in the automatic operation pause state (stop code (T02 0200)). After an emergency stop is released, operation can be restarted by the automatic operation start. (This parameter setting depends on the MTB specifications.)

When operation is restarted, slide-movement (moved with rapid traverse to the interruption point obtained by the interpolation calculation) and spindle state recovery is performed, and then the interrupted block operation (interpolation) is restarted. The travel speed to the interruption point depends on the MTB specifications (parameter "#12125 slide-F").

For details on the spindle recovery, refer to the "PLC Interface Manual".

Note

- (1) The interruption point obtained by the interpolation calculation precedes the actual machine position; therefore, the restart position will be placed before the emergency stop position.
- (2) After an emergency stop occurred during MDI operation, the operation can be restarted; however, slide-movement and spindle state recovery are not performed at restart.
- (3) Operation can also be restarted while the machining interruption program is running.
To restart the operation, use the start signal to suit the currently executed machining interruption program.
In addition, while the retraction program is running, slide-movement is not performed even when there is a slide amount (difference between the interruption position obtained by the interpolation calculation and the actual machine position). However, the slide amount is compensated for the axis moved by G26 (selected point return/tapping retract command) or M99 (retraction program end command), or the axis with the absolute value specified. (The slide amount is added to the travel amount required to the end point.)
- (4) Slide-movement is performed not only after an emergency stop is released but also when the manual interruption amount is provided when the automatic operation is restarted. The manual interruption amount can be viewed on the operation screen or drive monitor screen.
- (5) If the automatic operation pause occurs by Door open I, slide-movement and spindle state recovery are not performed even when Door open I is released.
- (6) When you want to use the C axis mode of the spindle position control (spindle/C axis) function, the parameter "#11020/bit0" must be set to "0" (Disable). (This parameter setting depends on the MTB specifications.)



EMG: Emergency stop
 RUN: Coasting after an emergency stop
 STOP: Stop position
 SLIDE: Slide position
 RE: Interpolation restart position
 E: Program end point



Program example

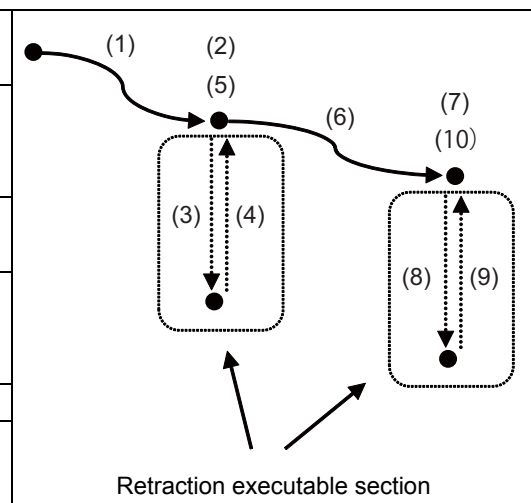
Drilling

This example shows program-based retraction (drilling) at two locations using the drill tool.

- (1), (6): Positioning command to the drilling position.
- (2), (7): Indicates the machining start point just before drilling.
- (5), (10): Indicates the machining end point after drilling.

The "retraction executable" signal is set to ON and program-based retraction can be executed in the sections between (3) and (5) and between (8) and (10).

(1) G00 G91 X100. Y50.;	
(2) ,Q1000;	
(3) G01 Z-165.2 F600;	Retraction executable section
(4) Z165.2 F2000;	
(5) ,Q0;	
(6) G00 X100. Y50.;	
(7) ,Q1000;	
(8) G01 Z-165.2 F600;	Retraction executable section
(9) Z165.2 F2000;	
(10) ,Q0;	
M30;	
N1000 G26 Z0.;	Retraction program
M99;	





Relationship with other functions

All functions

Column A: Case to start retraction while the related function is running

Column B: Case to execute the related function during retraction

Column C: Case that emergency stop occurs while the related function is running, and that operation is restarted after an emergency stop is released.

○: Can be combined with the related function.

×: Cannot be combined with the related function.

Do not use the functions, which cannot be combined (indicated as "x") with the retraction start operation, in the section from the machining start point to the machining end point.

-: Not related.

Δ: It is not impossible to create this combination; however, machining cannot be continued even if specified.

Functions not described in this table cannot be combined (same as "x") as a rule.

Related function	A	B	C
Positioning/Interpolation			
Positioning	○	○	○
Unidirectional positioning	○	○	Δ
Linear interpolation	○	○	○
Circular interpolation	○	○	○
Helical interpolation	○	○	○
Spiral/conical interpolation	○	○	○
Cylindrical interpolation	×	×	Δ
Polar coordinate interpolation	×	×	Δ
Milling interpolation	×	×	Δ
Curve interpolation			
Involute Interpolation	×	×	Δ
Exponential interpolation	×	×	Δ
Spline interpolation	×	×	Δ
NURBS interpolation	×	×	Δ
3-dimensional circular interpolation	×	×	Δ
Speed			
Feed per minute	○	○	-
Feed per revolution	○	○	-
Inverse time feed	○	○	-
F1-digit feed	○	○	-
Manual speed command	○	○	-
Rapid traverse override	○	○	-
Cutting feed override	○	○	-
2nd cutting feed override	○	○	-
Override cancel	○	○	-
Acceleration/Deceleration			
Automatic acceleration/deceleration after Interpolation	○	○	-
Rapid traverse constant inclination acceleration/deceleration	○	○	-
Rapid traverse constant inclination multi-step acceleration/deceleration	○	○	-
Thread cutting			
Thread cutting	○	Δ	Δ

15 Program Support Functions

Related function	A	B	C
Variable lead thread cutting	○	△	△
Synchronous tapping cycle	○	×	△
Chamfering	○	-	△
High-speed synchronous tapping	○	×	△
Spindle, tool, miscellaneous function			
Spindle control	○	○	○
Constant surface speed	○	○	○
Multiple-spindle control I	○	○	○
Multiple-spindle control II	○	○	○
Spindle orientation	○	○	○
Spindle position control (Spindle/C axis control)	○	○	×
Spindle synchronization	×	×	×
T function	○(*1)	○	-
Miscellaneous functions	○(*1)	○	-
2nd miscellaneous functions	○(*1)	○	-
Tool compensation			
Tool length offset	○	○	-
Tool radius compensation	○	×	-
3-dimensional tool radius compensation	×	×	-
Tool nose radius compensation (G40/41/42)	○	×	-
Operation support functions			
Program restart	-	-	-
Program support functions			
Subprogram control	○	○	-
Scaling	×	×	○
Macro call	×	×	-
Macro interruption	×	×	-
Fixed cycle for drilling	○	×	△
Special fixed cycle	○	×	△
Fixed cycle for turning machining	○	×	△
Compound type fixed cycle for turning machining	×	×	△
Mirror image	×	×	○
Coordinate rotation by program	×	×	○
3-dimensional coordinate conversion	×	×	○
Corner chamfering/Corner R	○	○	○
Linear angle command	○	○	○
Geometric command	○	○	○
Polar coordinate command	×	×	○
Chopping	×	×	-
Normal line control	×	×	×
Circular cut	×	×	△
Timing synchronization between part systems	×	×	×
Start point designation synchronization	×	×	×
Mixed control (Cross axis control)	×	×	-
Control axis synchronization between part systems	×	×	×
Balance cut	×	×	△
Common memory for part systems	×	×	-
2-part system simultaneous thread cutting	×	×	△
Multi-part system program management	○	○	-
Machining modal	○	○	-

Related function	A	B	C
Automatic corner override	-	○	-
Deceleration check	-	○	-
High-speed machining mode	×	×	△
High-speed high-accuracy control I (G5.1Q1)	×	×	△
High-speed high-accuracy control II (G5P10000)	×	×	△
High-accuracy control (G61.1/G08)	×	×	△
High-accuracy spline interpolation 1 (G61.2)	×	×	△
High-accuracy spline interpolation 2 (G61.3)	×	×	△
SSS Control	×	×	△
Automation support functions			
Skip	×	×	△
Automatic tool length measurement	×	×	×
Safety and maintenance			
Emergency stop	-	○	-
Stored stroke limit	-	○	-
Stroke check before travel	-	○	-
Chuck barrier/tailstock barrier check	-	○	-
External deceleration	○	○	-
Machine support functions			
Synchronous control	○	○	○
Inclined axis control	×	×	×

(*1) If retraction start is executed while a miscellaneous function is running (when the corresponding strobe signal is set to ON and waiting the FIN signal), the strobe signal is set to OFF, and the FIN signal wait state is canceled.

Tapping cycle

To perform retraction start while tapping cycle, pecking tapping cycle, or deep-hole tapping cycle is running, execute the tapping retract command (G26) to pull up the tool.

In the synchronous tapping cycle, if tapping retract is performed, the spindle rotates in the direction opposite to the cutting work synchronously with the movement of the hole drilling axis. In the asynchronous tapping cycle, the spindle does not rotate even if the tapping retract is executed. Therefore, issue the spindle rotation command before the tapping retract command as necessary.

The below table shows the fixed cycles in which tapping retract is possible.

Tapping cycle currently being executed
G84 (Forward tapping cycle)
G74 (Reverse tapping cycle)

- (1) When retraction is performed while the tapping cycle is running, only the G26 first commanded in the retraction program operates as the tapping retract command. The subsequently issued G26 commands operate as the selected point return command.
- (2) The hole drilling axis is pulled up by the tapping retract command only when retraction start is performed during cutting or dwelling in the tapping cycle. If retraction start is performed during non-cutting in the tapping cycle, the hole drilling axis does not move with the tapping retract command.
- (3) In the retraction program, the axis movement cannot be performed before the tapping retract command. In addition, when retraction start is performed during the synchronous tapping cycle, the S command cannot be executed. (If commanded, a program error (P729) occurs.)
- (4) In the retraction program for tapping cycle, always execute the tapping retract commands (G26 and hole drilling axis commands). If the retraction program is ended (the M99 block is executed) while the tapping retract command remains unexecuted, a program error (P729) occurs.
- (5) The spindle rotation speed for tapping retract is the same as for cutting.
The ",S" command and the parameter "#1172 tapovr" (MTB specifications) are invalid.

[Example to apply program-based retraction to the asynchronous tapping cycle]

G00 G90 X100. Y50. Z0.;	
,Q1000;	Selected point (Machining start point)
S1000 M03;	(*1)
G84 R-20. Z-100. F800 P500 ,R0;	
G80;	
M05;	
,Q0;	
:	
M30;	
N1000 M04;	Reverses the spindle to perform asynchronous tapping retract. (*2)
N1010 G26 G91 Z5.;	Pulls up the hole drilling axis to "5" (" -15" for the absolute value) as the incremental value from point R. (*3)
N1020 M05;	Spindle stop (*2)
N1030 G26 X0. Y0. Z0. F1000;	The second and subsequent G26 commands operate as the selected-point return command.
M99;	

(*1) In the synchronous tapping cycle, specify the spindle rotation speed with the synchronous tapping command block.

(*2) In the synchronous tapping cycle, the spindle reverse or spindle stop block is not required.

(*3) When tapping retract is performed during non-cutting (while the axis is moving from the initial point to point R), the hole drilling axis does not move.

When performing tapping successively at multiple hole drilling positions, specify the machining start point and machining end point at each hole drilling position. If the machining start point and machining end point are collectively specified for one hole drilling position, tapping is performed from the first step each time retraction start is executed. Therefore, tapping is also performed at the hole drilling position at which machining has already ended.

[Example to apply program-based retraction to the cycle with continuous tapping processes]

G00 G90 X0. Y0. Z0.;	
,Q1000;	
G84 Z-100. F1 S1000 ,R1;	(*3)
,Q0;	(*2)
,Q1000;	(*2)
X10.;	
,Q0;	(*2)
,Q1000;	(*2)
X20.;	(*1)
,Q0;	
G80;	
M30;	
N1000;	
:	
M99;	

(*1) If program-based retraction is performed during tapping at this hole drilling position, tapping is restarted at this hole drilling position after the retraction program was ended.

(*2)(*3) When both commands ",Q1000" and ",Q0" are not issued for each command of hole drilling position, the block to be executed will return to the hole drilling position indicated with (*3) after the retraction program ends.

Combinations of G commands while retraction is executed

o: Enabled, x: Disabled

G code	Group	Operation
G00	01	o
G01	01	o
G02	01	o
G03	01	o
G02.1	01	o
G03.1	01	o
G02.3	01	x
G03.3	01	x
G02.4	01	x
G03.4	01	x
G04	00	o
G05	00	x
G05.1	00	x
G06.2	01	x
G07	00	x
G07.1	21	x
G107		
G08	00	x
G09	00	o
G10	00	o
G11	00	o
G12	00	x
G13	00	x
G12.1	21	x
G112		
G13.1	21	x
G113		
G15	18	x
G16	18	x
G17	02	o
G18	02	o
G19	02	o
G20	06	o
G21	06	o
G22	04	o
G23	04	o
G27	00	o
G28	00	o
G29	00	o
G30	00	o

G code	Group	Operation
G30.1	00	o
G30.2	00	o
G30.3	00	o
G30.4	00	o
G30.5	00	o
G30.6	00	o
G31	00	x
G31.1	00	x
G31.2	00	x
G31.3	00	x
G33	01	x
G34	00	x
G35	00	x
G36	00	x
G37	00	x
G37.1	00	x
G38	00	x
G39	00	x
G40	07	x
G41	07	x
G42	07	x
G40.1	15	x
G41.1	15	x
G42.1	15	x
G43	08	o
G44	08	o
G43.1	08	x
G43.4	08	x
G43.5	08	x
G45	00	x
G46	00	x
G47	00	x
G48	00	x
G49	08	o
G50	11	x
G51	11	x
G50.1	19	x
G51.1	19	x
G52	00	x
G53	00	o

G code	Group	Operation
G54	12	○
G55	12	○
G56	12	○
G57	12	○
G58	12	○
G59	12	○
G54.1	12	○
G60	00	○
G61	13	○
G61.1	13	×
G61.2	13	×
G62	13	○
G63	13	○
G63.1	13	×
G63.2	13	×
G64	13	○
G65	00	○
G66	14	×
G66.1	14	×
G67	14	×
G68	16	×
G69	16	×
G70	09	×
G71	09	×
G72	09	×
G73	09	×
G74	09	×

G code	Group	Operation
G75	09	×
G76	09	×
G77	09	×
G78	09	×
G79	09	×
G80	09	×
G81	09	×
G82	09	×
G83	09	×
G84	09	×
G85	09	×
G86	09	×
G87	09	×
G88	09	×
G89	09	×
G90	03	○
G91	03	○
G92	00	×
G92.1	00	×
G93	05	○
G94	05	○
G95	05	○
G96	17	○
G97	17	○
G98	10	×
G99	10	×
G100 - G225	00	○



Precautions

- (1) If emergency stop occurs during synchronous control, a synchronization error between the reference axis and synchronized axis is not corrected in the slide-movement process after an emergency stop is released. To use the synchronous control function, the "automatic correction of synchronization error at servo-ON" must be enabled. (This setting depends on the MTB specifications (parameter "#1281 ext17/bit3").)
- (2) The fixed-cycle modal calling is not performed in the machining interruption program.
In addition, the fixed cycle (G code group 9) cannot be commanded.
(If commanded, a program error (P728) occurs.)
- (3) The macro modal calling is not performed in the machining interruption program.
In addition, the macro modal call (G code group 14) cannot be commanded.
(If commanded, a program error (P728) occurs.)

Multi-part System Control

16.1 Timing Synchronization Operation

CAUTION

When programming a multi-part system, carefully observe the movements caused by other part systems' programs.

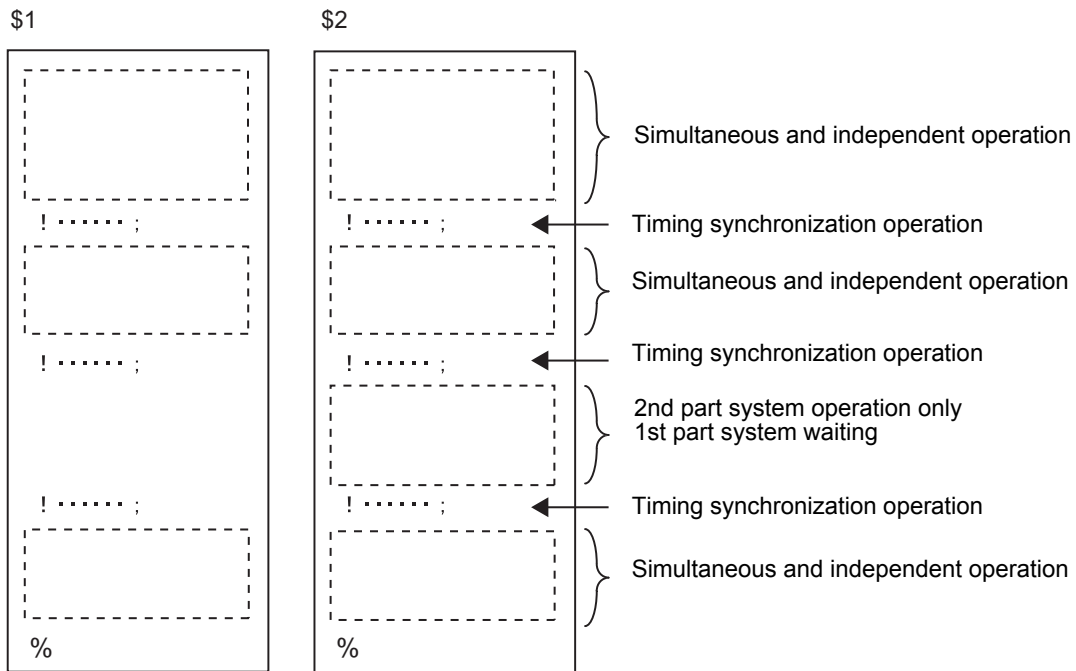
16.1.1 Timing Synchronization Operation (!n (!m ...) L



Function and purpose

The multi-axis, multi-part system complex control CNC system can simultaneously run multiple machining programs independently. The synchronization-between-part systems function is used in cases when, at some particular point during operation, the operations of 1st and 2nd part systems are to be synchronized or in cases when the operation of only one part system is required.

When timing synchronization is executed in the 1st part system (\$1) and the 2nd part system (\$2), operations will be as follows.



Command format

`!n (!m ...) L_ ;`

!n, !m, ...	Timing synchronization operation (!) and part system No. (n:1 - number of part system that can be used) Follows the settings of the parameter "#19419 Timing sync system" if part system number is omitted.
L	Timing Synchronization Operation No. 0 to 9999



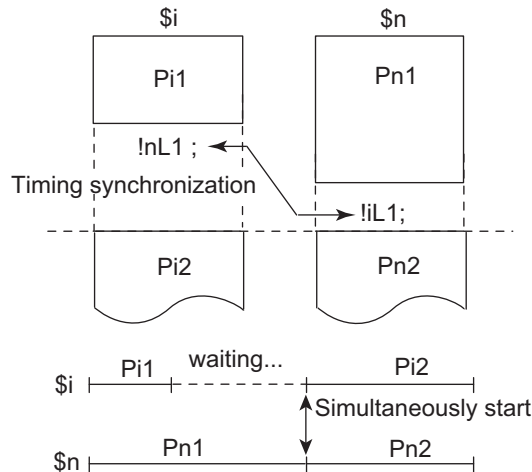
Detailed description

- (1) Timing synchronization between part systems during automatic operation

If !n L_ is commanded from a part system (i), operation of the part system i program will wait until !i L_ is commanded from the part system n program.

When !i L_ is commanded, the programs for the two part systems will start simultaneously.

Timing synchronization between 2 part systems



- (2) The timing synchronization operation is normally issued in a single block. However, if a movement command or M, S or T command is issued in the same block, whether to synchronize after the movement command or M, S or T command or to execute the movement command or M, S or T command after synchronization will depend on the MTB specifications (#1093 Wmvfin).

#1093 Wmvfin

0 : Wait before executing movement command.

1 : Wait after executing movement command.

- (3) If there is no movement command in the same block as the timing synchronization operation, when the next block movement starts, synchronization may not be secured between the part systems. To synchronize the part systems when movement starts after waiting, issue the movement command in the same block as the timing synchronization operation.
- (4) The L command is the timing synchronization identification No. The same Nos. are waited but when they are omitted, the Nos. are handled as L0.
- (5) "SYN" will appear in the operation status section during timing synchronization operation. The timing synchronization operation signal will be output to the PLC I/F.
- (6) In a timing synchronization operation, other part system to be waited for is specified but the own part system can be specified with the other part system.
- (7) The timing synchronization operation of a specific part system can be ignored depending on the MTB specifications.
Operation will be determined by the combination of the timing synchronization operation ignore signal and parameter "#1279 ext15/bit0".
For setting combination, refer to "Time synchronization when timing synchronization ignore is set".
For the specifications of the machine you are using, see the instructions issued by the MTB.



Precautions

- (1) When the M code can be used, both the M code and ! code can be used.
- (2) While the timing synchronization operation M code is valid, if one part system is standing by with an M code, an alarm will occur if there is a ! code timing synchronization operation command in the other part system.
- (3) While the timing synchronization operation M code is valid, if one part system is standing by with a ! code, an alarm will occur if there is an M code timing synchronization operation command in the other part system.
- (4) When macro interruption is carried out in a part system waiting, the part system can stop while waiting even if the conditions for time synchronization are met. In this case, you will be able to continue the program, ignoring the timing synchronization with timing synchronization operation ignore signal.
For details, contact the MTB.

16.1.2 Timing Synchronization Operation with Start Point Designated (Type 1) ; G115

**Function and purpose**

The part system can wait for the other part system to reach the start point before starting itself. The start point can be set in the middle of a block.

**Command format**

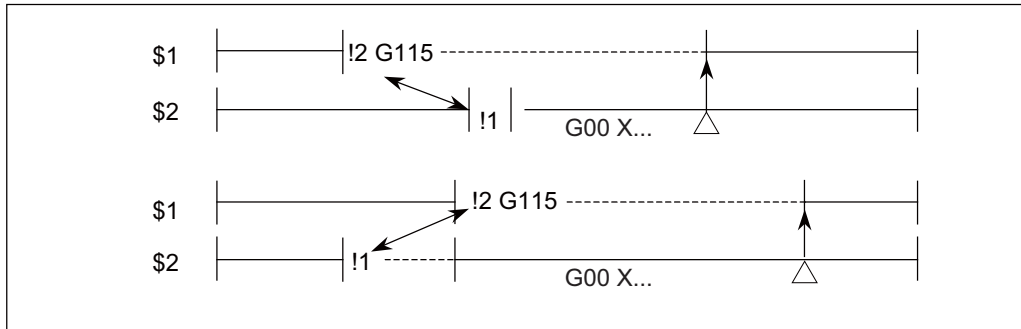
```
In L_ G115 X_ Y_ Z_;
```

!n	Timing synchronization operation (!) and part system No. (n:1 - number of part system that can be used) Part systems follow the settings of the parameter "#19419 Timing sync system" if the number is omitted.
L	Timing Synchronization Operation No. 0 to 9999 (It will be regarded as "L0" when omitted.)
G115	G command
X Y Z	Start point (Command by axis and workpiece coordinate value)



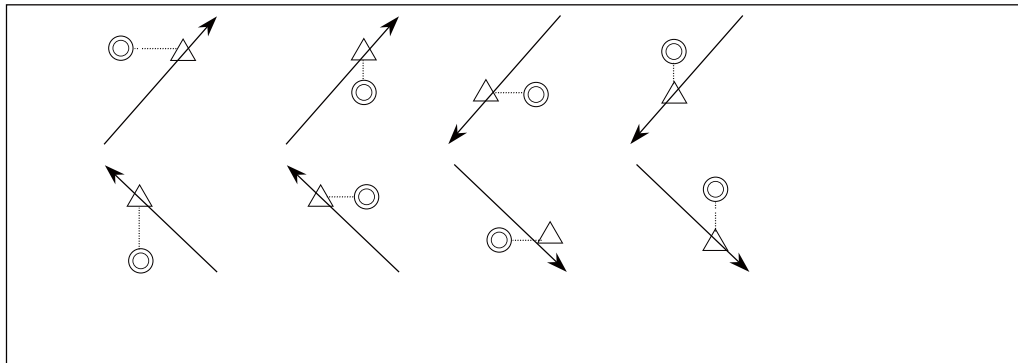
Detailed description

- (1) Designate the start point using the workpiece coordinates of the other part system (ex. \$2).
- (2) The start point check is executed only for the axis designated by G115.
 (Example) !L2 G115 X100. ;
 Once the other part system reaches X100, the own part system (ex. \$1) will start. The other axes are not checked.
- (3) The other part system starts first when timing synchronization operation is executed.
- (4) The own part system waits for the other part system to move and reach the designated start point, and then starts.



↔ Timing synchronization △ Designated start point

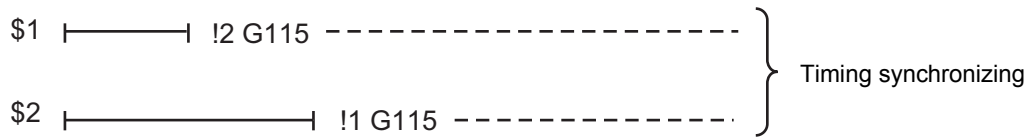
- (5) When the start point designated by G115 is not on the next block movement path of the other part system, the own part system starts once all the designated axis of the other part system has reach the designated start point.



← Movement ⊙ Designated start point △ Actual start point

- (6) After waiting, if the start point cannot be obtained with movement command of the other timing synchronization block, the operations depend on the MTB specifications (parameter "#1229 set01/bit5").
 - (a) When the parameter is ON
 Wait till the own part system reaches the start point by moving after the next block.
 - (b) When the parameter is OFF
 When the next block finishes moving, the own part system will start.

- (7) The timing synchronization status continues when the G115 command has been duplicated between part systems. (Operations will not restart.)



- (8) The single block stop function does not apply for the G115 block.
- (9) A program error (P32) will occur if an address other than an axis is designated in G115 command block.
- (10) In the timing synchronization operation, other part system to be waited for is specified but the own part system can be specified with the other part system.
- (11) The timing synchronization operation of a specific part system can be ignored depending on the MTB specifications. Operation will be determined by the combination of the timing synchronization operation ignore signal (PLC signal) and parameter "#1279 ext15/bit0".
For setting combination, refer to "Time synchronization when timing synchronization ignore is set".
For the specifications of the machine you are using, see the instructions issued by the MTB.



Precautions

- (1) Parameter "#1093 Wmvfin" that selects the timing of the timing synchronization operation and commands on the same block does not work for the start point command block (G115/G116). After synchronization, the start point check will be executed by G115/G116.
- (2) Be careful about the timing when interrupting during the time synchronization of G115/G116. For example, assume interruption with the macro interrupt type 1 while a part system is waiting for time synchronization with G116. In this case, if there is a movement command or MSTB command in the interrupt program, the program will continue after the interrupt program completes without waiting for the start point.
- (3) The L command is the timing synchronization identification No. The same Nos. are waited but when they are omitted, the Nos. are handled as L0.

16.1.3 Timing Synchronization Operation with Start Point Designated (Type 2) ; G116

**Function and purpose**

The own part system can make the other part system to wait until it reaches the start point. The start point can be set in the middle of a block.

**Command format**

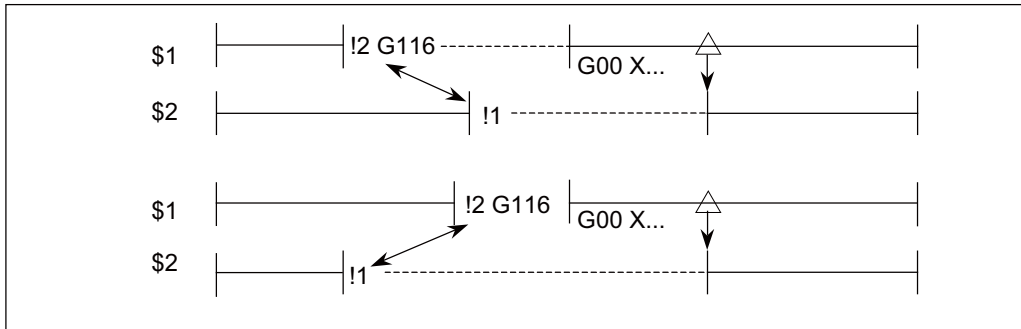
```
!n L_ G116 X_ Y_ Z_ ;
```

!n	Timing synchronization operation (!) and part system No. (n:1 - number of part system that can be used) Part systems follow the settings of the parameter "#19419 Timing sync system" if the number is omitted.
L	Timing Synchronization Operation No. 0 to 9999 (It will be regarded as "L0" when omitted.)
G116	G command
X Y Z	Start point (Command by axis and workpiece coordinate value)



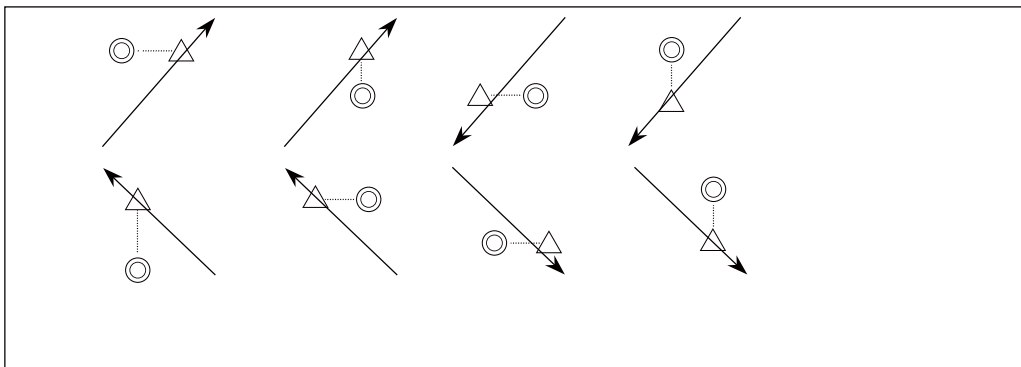
Detailed description

- (1) Designate the start point using the workpiece coordinates of the own part system (ex. \$1).
- (2) The start point check is executed only for the axis designated by G116.
 (Example) !L1 G116 X100. ;
 Once the own part system reaches X100, the other part system (ex. \$2) will start. The other axes are not checked.
- (3) The own part system starts first when timing synchronization operation is executed.
- (4) The other part system waits for the own part system to move and reach the designated start point, and then starts.



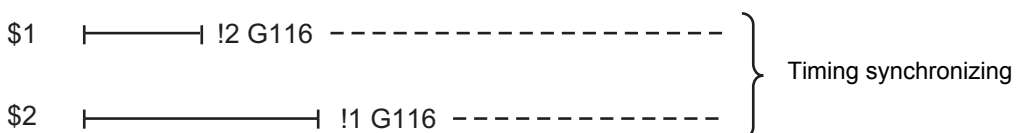
↔ Timing synchronization △ Designated start point

- (5) When the start point designated by G116 is not on the next block movement path of own part system, the other part system starts once all the designated axes of the own part system has reach the designated start point.



← Movement ⊙ Designated start point △ Actual start point

- (6) If the start point cannot be obtained with the movement of the own part system to the next block, the operations depend on the MTB specifications (parameter "#1229 set01/bit5").
 - (a) When the parameter is ON
 The own part system will have a program error (P511) before moving.
 - (b) When the parameter is OFF
 When the next block finishes moving, the other part system will start.
- (7) The timing synchronization status continues when the G116 command has been duplicated between part systems. (Operations will not restart.)



- (8) The single block stop function does not apply for the G116 block.
- (9) A program error (P32) will occur if an address other than an axis is designated in G116 command block.
- (10) In the timing synchronization operation, other part system to be waited for is specified but the own part system can be specified with the other part system.
- (11) The timing synchronization operation of a specific part system can be ignored depending on the MTB specifications. Operation will be determined by the combination of the timing synchronization operation ignore signal (PLC signal) and parameter "#1279 ext15/bit0".
For setting combination, refer to "Time synchronization when timing synchronization ignore is set".
For the specifications of the machine you are using, see the instructions issued by the MTB.



Precautions

Refer to "Start point designation timing synchronization (Type 1) ; G115".

16.1.4 Timing Synchronization Operation Function Using M codes ; M***

**Function and purpose**

The timing synchronization operation function between part systems is conventionally commanded with the "!" code, but by using this function, the part systems can be waited with the M code commanded in the machining program. If the timing synchronization operation M code is commanded in either part system during automatic operation, the system will wait for the same M code to be commanded in the other part system before executing the next block. The timing synchronization operation M code is used to control the timing synchronization operation between the 1st part system and 2nd part system. Whether the timing synchronization operation M code can be used depends on the MTB specifications.

**Command format**

M*** ;

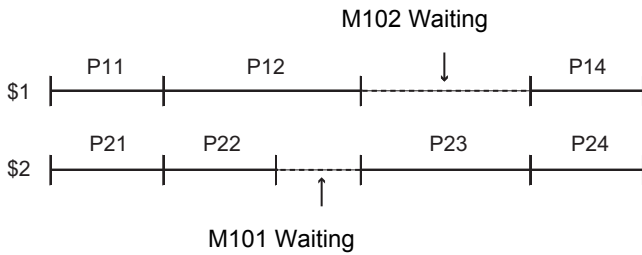
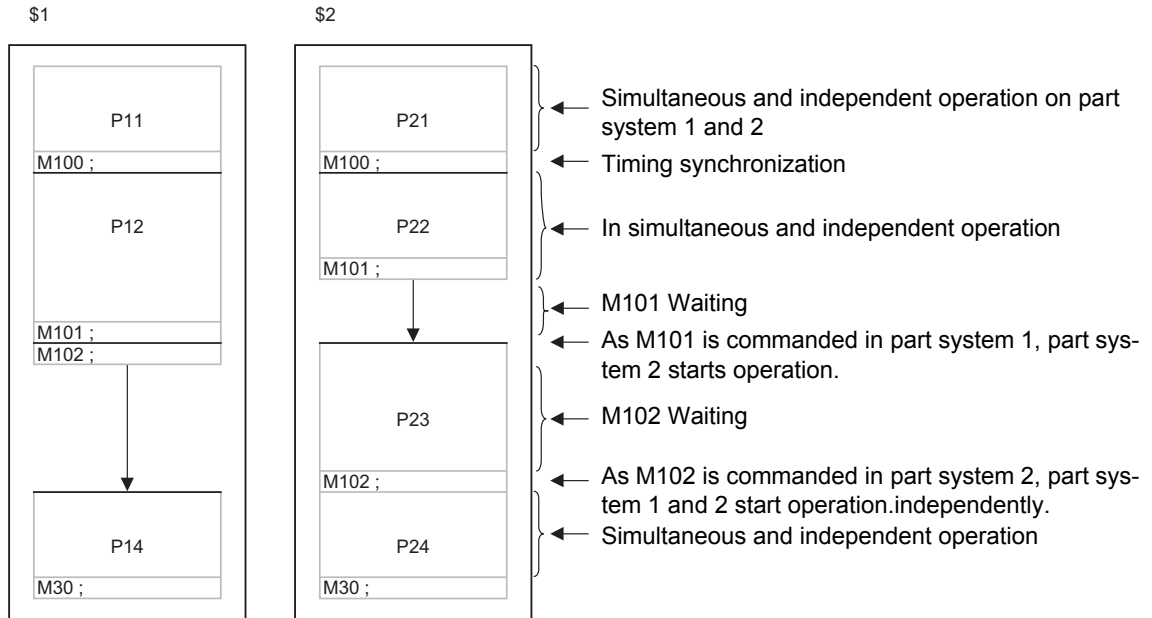
***	Timing synchronization operation M code
-----	---

M code used for timing synchronization depends on the MTB specifications (parameter "#1310 WtMmin)", "#1311 WtMmax").

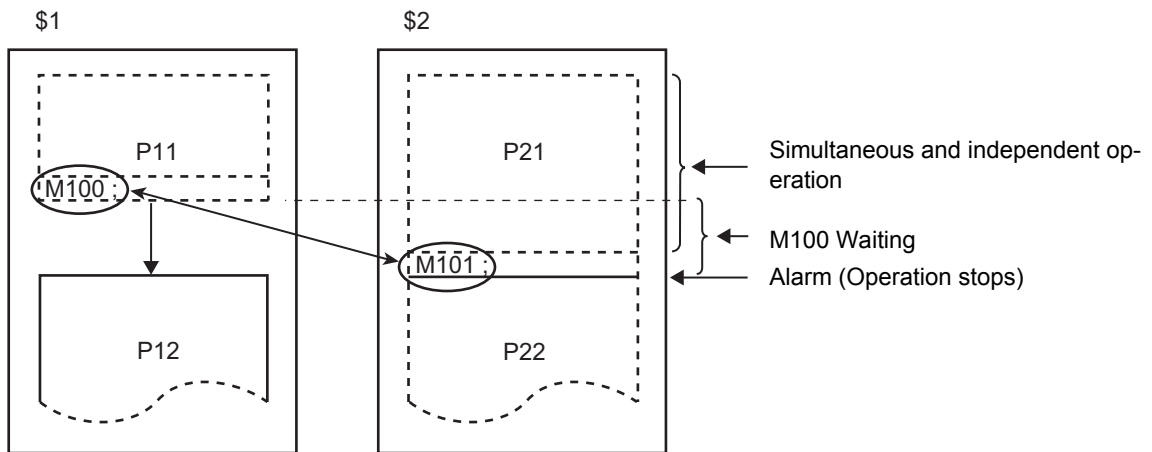


Detailed description

- (1) When the timing synchronization operation M code is commanded in the machining program, the two part systems will be waited and operation will start in the commanded block. If the timing synchronization operation M code is commanded in either part system during automatic operation, the system will wait for the same M code to be commanded in the other part system before executing the next block.



- (2) When the timing synchronization operation M code has been commanded in one part system, and the part system is standing by for waiting, an alarm will occur if a different M code is commanded in the other part system.



(3) The part systems are waited with the M code following the parameters below.

These settings depend on the MTB specifications. Refer to these settings. For details, refer to the specifications of your machine.

(a) M code range designation parameter (M code minimum value \leq M code \leq M code maximum value)

#	Item	Details	Setting range
1310	WtMmin	Timing synchronization M code ABS. MIN.	0, 100 ~ 99999999
1311	WtMmax	Timing synchronization M code ABS. MAX.	0, 100 ~ 99999999

This function is invalid if either parameter is set to "0".

The timing synchronization operation M code cannot be used if the M code maximum value is smaller than the minimum value.

When the timing synchronization operation M code is valid, both the M code and I code can be used for timing synchronization operation.

(b) Timing synchronization operation method parameters

#	Item	Details	Setting range
1279 (PR)	ext15 (bit0)	Method for timing synchronization operation between part systems	0 / 1

Select an operation for timing synchronization operation between part systems.
 0: If one of the part systems is not in automatic operation, ignore the timing synchronization operation and execute the next block.
 1: Operate according to the timing synchronization operation ignore signal.
 If the timing synchronization operation ignore signal is "1", the timing synchronization operation will be ignored. If "0", the part systems will be waited.

Depending on the timing synchronization operation method selection parameter and timing synchronization operation ignore signal combination, the timing synchronization operation will be determined by the parameters, regardless of the command format ("I" code and M code).

This parameter requires the CNC to be turned OFF after the settings. Turn the power OFF and ON to enable the parameter settings.

#	Item	Details	Setting range
1093	Wmvfin	Method for timing synchronization operation between part systems	0 / 1

Parameter to designate the timing synchronization operation between part systems method when using multi-part systems.
 When there is a movement command in the timing synchronization operation (!, M) block:
 0 : Wait before executing movement command.
 1 : Wait after executing movement command.



Relation with other functions

Refer to "Timing Synchronization Operation (! code);In (!m ...) L".



Precautions

For precautions for time synchronization, also refer to "Timing Synchronization (!code);!n (!m ...) L"

- (1) When timing synchronization operation with the M code, always command the M code in an independent block.
- (2) When standing by after commanding the timing synchronization operation M code in one part system, an alarm will occur if a different M code is commanded in the other part system. Operation will stop in both part systems.
- (3) The timing synchronization operation (! code, M code) in the machining program can be ignored with the timing synchronization operation ignore signal. (This depends on the MTB specifications.) Operation with a single part system is possible without deleting the timing synchronization operation (! code, M code) in the machining program.
- (4) Unlike other M codes, the timing synchronization operation M code does not output code signals and strobe signals.
- (5) When the M code can be used, both the M code and ! code can be used.
- (6) While the timing synchronization operation M code is valid, if one part system is standing by with an M code, an alarm will occur if there is a ! code timing synchronization operation command in the other part system.
- (7) While the timing synchronization operation M code is valid, if one part system is standing by with a ! code, an alarm will occur if there is an M code timing synchronization operation command in the other part system.
- (8) If there is a timing synchronization operation with M code after the 3rd part system, an alarm will occur.
- (9) The G115 and G116 commands cannot be used when waiting with the M code.
- (10) If the M code command Nos. are overlapped, the order of priority will be M code macro, M command synchronous tapping, timing synchronization operation M code and normal M code.
- (11) When macro interruption is carried out in a part system waiting, the part system can stop while waiting even if the conditions for time synchronization are met. In this case, you will be able to continue the program, ignoring the timing synchronization with timing synchronization operation ignore signal. For details, contact the MTB.
- (12) "SYN" will appear in the operation status section during timing synchronization operation.

16.1.5 Timing Synchronization When Timing Synchronization Ignore Is Set



Function and purpose

Turning on the timing synchronization operation ignore signal makes it possible to ignore the timing synchronization operation of that part system.

With a 2-part system, if the timing synchronization operation ignore signal of the other part system is ON, timing synchronization is not executed. In the following section, a 3-part system is used as an example to make it easier to understand the functions.

This signal is also used in the following functions.

- Timing synchronization (! code, M code)
- Start point timing synchronization (G115, G116)
- Balance cut (G15) Lathe system only

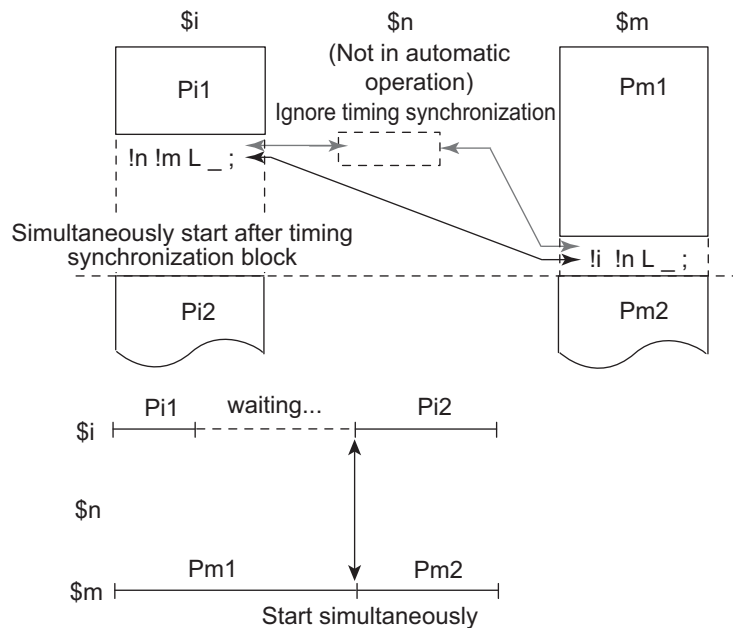
Note

(1) For sub part system control function, refer to "16.3 Sub Part System Control".

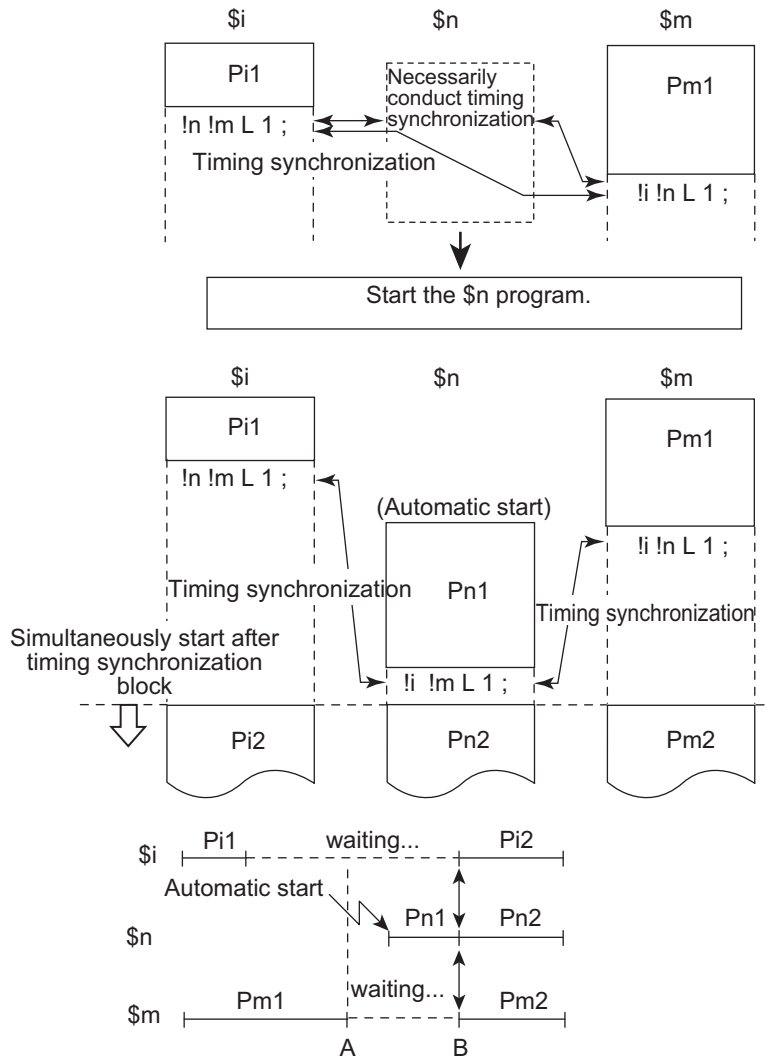
		Timing synchronization operation ignore signal (PLC signal)	
		OFF	ON
Parameter (#1279 ext15/bit0)	0	(1) Ignores the timing synchronization with a part system not in automatic operation	
	1	(2) Does not ignore the timing synchronization regardless of whether or not a part system is in automatic operation (the timing synchronization is executed until the conditions for timing synchronization are established.)	(3) Ignores the timing synchronization regardless of whether or not a part system is in automatic operation (ignores the timing synchronization command for the part system with the timing synchronization ignore signal ON and the timing synchronization operation for that part system)

The following operation diagram gives an example of ! code.

(1) A case that "Ignores the timing synchronization with a part system not in automatic operation"

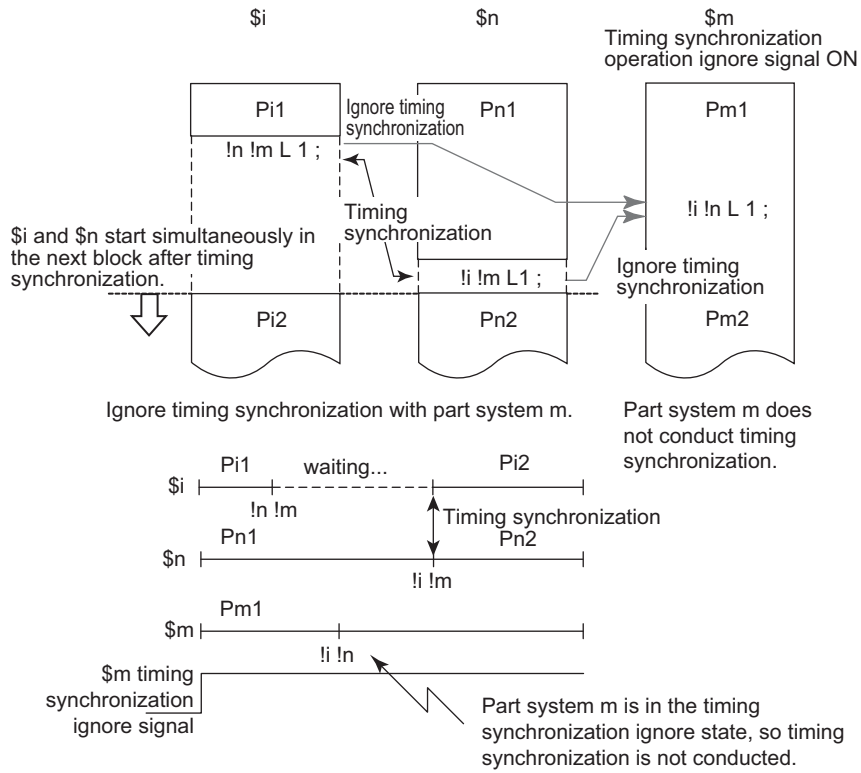


(2) A case that "Does not ignore the timing synchronization regardless of whether or not in automatic operation"



- A: When timing synchronization operation between part systems (parameter "#1279 ext15/bit0" = 1), the timing synchronization status continues until the conditions for timing synchronization are established.
- B: Part system n is automatically started. If the conditions for timing synchronization are established, the next block will start.

(3) A case that "Ignores the timing synchronization regardless of whether or not in automatic operation"



16.2 Mixed Control

16.2.1 Arbitrary Axis Exchange ; G140, G141, G142



Function and purpose

With this function, an arbitrary axis can be exchanged freely across part systems.

The machining can be freer in the multiple part systems by exchanging an axis that can be commanded for machining programs in each part system.

This makes it possible to perform operations which are not possible with regular axis configurations: for instance, tools which are provided only on the 1st part system can be used for machining on the 2nd part system.

This chapter illustrates an example based on the placements of the basis axes below.

	X axis	Y axis	Z axis	C axis
1st part system (\$1)	X1	Y1	Z1	-
2nd part system (\$2)	X2	Y2	-	C2

Refer to "Programming Manual Lathe System" (IB-1501275, IB-1501276) for details of the arbitrary axis exchange.



Command format

When commanding the arbitrary axis exchange

G140 command address = axis address ... ;

Command Address	It is a command address used in a movement or other command after arbitrary axis exchange command (G140). Designate the command address with one alphabetical character set to parameters ("#12071 adr_abs[1]"to "#12078 adr_abs[8]").
Axis address	Set the axis name for arbitrary axis exchange. Designate the command with two alphanumeric characters set to the parameter "#1022 axname2".

When returning the exchanged axis

G141; Arbitrary axis exchange return

Returns the control right of the axis, exchanged by the previous arbitrary axis exchange command (G140) in the commanded part system, to the state before the axis exchange.

G142; Reference axis arrange return

Returns the control right of the axis, exchanged by the arbitrary axis exchange command (G140) in the commanded part system, to the power-on state.



Detailed description

Arbitrary axis exchange command (G140)

There are two methods for axis exchange operations with arbitrary axis exchange command (G140). The methods for your machine depends on the MTB specifications (parameter "#1434 G140Type2").

Method	Operation
Method for exchanging all axes ("#1434 G140Type2" = 0)	Designates axes to be used in the part system with a command address. The command addresses axes that are not designated will be released as uncontrol axes.
Method for exchanging command axes ("#1434 G140Type2" = 1)	Designates axes to be used in the part system with a command address. The command addresses axes that are not designated will maintain the current state.

(1) Operation example of the method for exchanging all axes ("#1434 G140Type2"=0)

Below is the control axis of each part system when running the following machining programs (1st part system, 2nd part system)

\$1 Machining program	\$2 Machining program	Control axes						Uncontrol axes
		\$1			\$2			
		X	Y	Z	X	Y	C	
G140 X=X1 Y=Y1 Z=Z1; (a) G00 X10.; G01 X5. F1; :	G140 X=X2 Y=Y2 C=C2; (d) G00 X20.; G01 X15. F2; :	X1	Y1	Z1	X2	Y2	C2	-
G140 X=X1 Y=Y2; (b) G00 Y25.; G01 X8. F2; :	G140 Y=Z1; (e) G00 Y10.; G01 Y8. F0.05; :	X1	Y2	-	X2	-	C2	Y1,Z1
G140 X=X1 Y=Y1 Z=Z1; (c) G00 X20. Y15.; G01 X15. F5; :	G140 X=X2 Y=Y2 C=C2; (f) G00 X0; :	X1	Y1	Z1	-	-	-	X2,Y2,C2
					X2	Y2	C2	-

1st part system (\$1)	(a),(c)	Declares the use of X1 axis, Y1 axis and Z1 axis.
	(b)	Declares the use of X1 axis and Y2 axis. The control right of Y2 axis shifts to the 2nd part system from the 1st part system. Y1 axis, exchanged for Z1 axis and Y2 axis which were not designated, will be an uncontrol axis.
2nd part system (\$2)	(d),(f)	Declares the use of X2 axis, Y2 axis and C2 axis.
	(e)	Declares the use of Z1 axis. X2 axis and C2 axis which were not designated will be uncontrol axes.

(2) Operation example of the method for exchanging command axes ("#1434 G140Type2"=1)

Below is the control axis of each part system when running the following machining programs (1st part system, 2nd part system)

\$1 Machining program	\$2 Machining program	Control axes						Uncontrol axes
		\$1			\$2			
		X	Y	Z	X	Y	C	
G140 X=X1 Y=Y1 Z=Z1; (a) G00 X10.; G01 X5. F1; :	G140 X=X2 Y=Y2 C=C2; (d) G00 X20.; G01 X15. F2; :	X1	Y1	Z1	X2	Y2	C2	-
G140 Y=Y2; (b) G00 Y25.; G01 X8. F2; :	G140 Y=Y1; (e) G00 Y10.; G01 Y8. F0.05; :	X1	Y2	Z1	X2	-	C2	Y1
G140 Y=Y1 ; (c) G00 X20. Y15.; G01 X15. F5; :	G140 X=X2 Y=Y2 C=C2; (f) G00 X0; :	X1	Y1	Z1	X2	-	C2	Y2
					X2	Y2	C2	-

1st part system (\$1)	(a)	Declares the use of X1 axis, Y1 axis and Z1 axis.
	(b)	Declares the use of Y2 axis. The control right of Y2 axis shifts to the 2nd part system from the 1st part system. Y1 axis which was exchanged for Y2 axis will be an uncontrol axis.
	(c)	Declares the use of Y1 axis. The control right of Y1 axis shifts to the 2nd part system from the 1st part system. Y2 axis which was exchanged for Y1 axis will be an uncontrol axis.
2nd part system (\$2)	(d)	Declares the use of X2 axis, Y2 axis and C2 axis.
	(e)	Declares the use of Y1 axis.
	(f)	Declares the use of X2 axis, Y2 axis and C2 axis.

Unavailable state of axis exchange

"Unavailable state of axis exchange" indicates a "condition in which a target axis for axis exchange is not available for exchange because the designated target axis for axis exchange is being used by other part systems or for other reasons" through the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141), the reference axis arrange return command (G142).

When the conditions for unavailable state of axis exchange fall through, no axis exchange mode will be cancelled. It will be cancelled when a reset signal or emergency stop is entered.

16.3 Sub Part System Control

16.3.1 Sub Part System Control I; G122



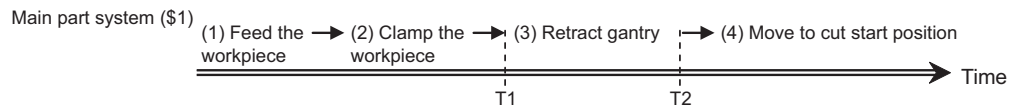
Function and purpose

This function activates and operates any non-operating part system (sub part system) in the multi-part system. Sub part system control I can be used in the same manner as calling subprogram in a non-operating part system. An auxiliary axis machining program can be controlled in the sub part system by commanding Sub part system control I (G122) from the main part system.

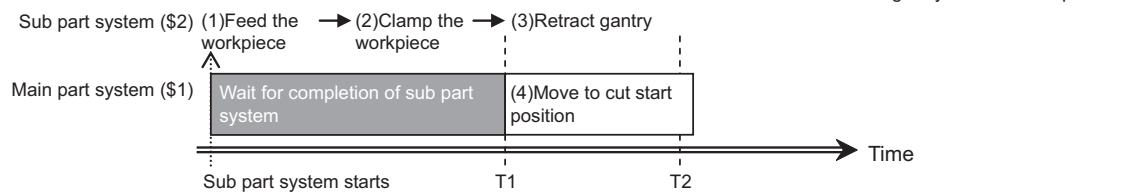
In the usage example below, the tool positioning starts to the machining start point at the same time (time T1) as the start of gantry retract by using Sub part system control I (G145) in the flow from feeding the workpiece to moving to cut start position in order to reduce the cycle time.

Select whether main part system or sub part system for each part system in Sub part system control I. When using a part system as a sub part system, by setting the operation mode to "Sub part system I operation mode" with the PLC signal and commanding sub part system control I (G122) from an operating part system, it is possible to activate the part system in the sub part system I operation mode as a sub part system.

Machining process when Sub part system control is OFF



Machining process when Sub part system control is ON



The following describes the meanings of the terms used in this chapter.

Term	Meaning
Main part system	Indicates a part system located on the uppermost stream side of a sub part system call flow.
Sub part system	Indicates a part system activated by the sub part system activation command.
Calling part system	Indicates a part system that issued the sub part system activation command.

The examples below shows many part systems to provide an easy-to-understand explanation. The actually available number of part systems depends on your machine's specifications.

Enabling conditions

- (1) This function can be used in multi-part systems of two or more part systems.
- (2) In order to activate a sub part system using the sub part system control I command, the following conditions must be satisfied. There are enabling conditions that are only applicable to the M80 series.

[Condition 1]

This condition must be satisfied only for the M80 series/C80 series.

The number of sub part systems has been set in the base common parameter "#1483 SBS1_sys num" (the number of part systems in sub part system I).

- (a) Part systems as many as the number specified in #1483, counted from the end of the valid part system (the part system for which "#1001 SYS_ON" is set to "1"), will be reserved as sub part systems.
- (b) If the number of sub part systems or main part systems exceeds the maximum number defined in the system specifications, an MCP alarm (Y05 1483) will occur.
- (c) (M80 series only) If the values set for "#1483 SBS1_sys num" and "#1474 SBS2_sys num" are both "1" or more, an MCP alarm (Y05 1483) will occur.

[Condition 2]

The identification No. (B command value) used to activate a sub part system has been set in the base common parameter "#12049 SBS_no" (sub part system I identification No.) for sub part systems.

- (a) If an identification No. that is not set in the parameter "#12049 SBS_no" is specified when the sub part system control I command is issued, a program error (P650) (sub part system identification No. illegal) will occur.

[Condition 3]

The PLC signal SBSM (Sub part system I operation mode) of the sub part system is set to "1".

- (a) In a part system operating the sub part system I operation mode, the operation mode appears as "SUB" in the part system display of the operation screen.
- (b) If the sub part system control I command is issued to a part system that is not operating the sub part system I operation mode, an operation error (M01 1111) will occur. However, while the operation error (M01 1111) is occurring, the operation can be started by setting SBSM to "1".



Command format

Call sub part system

G122 A_P_Q_K_D_B_H_ (argument);

G122 <file name> P_Q_K_D_B_H_ (argument);

A	Program No. (1 to 99999999 or 100010000 to 199999998)
<File name>	File name of the program (up to 32 characters)
P	Start sequence number (Head of the program if omitted.)
Q	End sequence number (To end (M99) of the program if omitted.)
K	Number of repetitions (1 to 9999)
D	Synchronization control (0/1)
B	Sub part system identification No. (1 to 7)
H	Sub part system reset type (0/1)
Argument	Argument of a sub part system local variable (Setting rage of local variable (decimal point command is valid))

Complete sub part system

M99; (command in sub part system side)

Cancel the standby status for completion of sub part system

G145; (command in the sub part system side when the D0 command is issued)

Note

(1) G145 is ignored in a sub part system activated in the parallel control method (D1 command).



Detailed description

This function can be used in multi-part systems of two or more part systems.
Main part system and sub part system are switched according to the MTB specifications.

Description of each address

Address	Meaning	Command range (unit)	Remarks
A	Program No.	1 to 99999999 or 100010000 to 199999998 (*1)	Program No. or file name of the machining program operating in the sub part system. •Programs in an external device cannot be designated. •If address A and <file name> are designated at the same time, precedence is given to address A. •If designation of the program is omitted, the machining program defined by the MTB will be used (parameter "#12050 SBS_pro").
<File name>	File name of the program	Up to 32 characters.	
P	Start sequence No.	1 to 99999999	Sequence No. to start the machining program operating in the sub part system. •If there is no command, the operation will start from the head of the machining program.
Q	End sequence No.	1 to 99999999	Sequence No. to end the machining program operating in the sub part system. •If there is no command, the program will run up to "M99".
K	Number of repetitions	1 to 9999	The number of times to repeat the machining program for continuous operation in the sub part system. •If there is no command, the program will only run once. (No repetition)
D	Synchronous control	0 / 1	Validity of synchronous control 0: The next block is processed after the sub part system operation completes. 1: The next block is processed at the same time as the start of a sub part system operation. If there is no command, it is handled in the same manner as 0 is designated.
B	Sub part system identification No.	1 to 7	Identification No. used for timing synchronization with sub part system, etc. •The sub part system to be activated is designated by an identification No. The correspondence between identification No. and part system No. depends on the MTB specifications (parameter "#12049 SBS_no"). •If there is no command, it is handled in the same manner as 1 is designated.
H	Sub part system reset type (*2)	0 / 1	0: The G command modal is maintained by the reset when a sub part system is complete. 1: The G command modal is initialized by the reset when a sub part system is complete. •If there is no command, it is handled in the same manner as 0 is designated.
(Argument)	Argument of a sub part system local variable	Setting range of local variable (Decimal point command is possible.)	•Argument is passed to the sub part system as a local variable (level 0). However, addresses A, B, D, G, H, K, O, P, and Q cannot be used as an argument. •For the correspondence between address and variable number, refer to the following table.

(*1) When the parameter "#1253 set25/bit0" is set to "1", the command range is "100010000 to 199999989".

(*2) If a sub part system ends by M99 or the end sequence No., resetting processing is performed automatically in the sub part system.

Correspondence of argument designation address and variable number in sub part system

Argument designation address	Variable number in sub part system
A	-
B	-
C	#3
D	-
E	#8
F	#9
G	-
H	-
I	#4
J	#5
K	-
L	#12
M	#13

Argument designation address	Variable number in sub part system
N	#14
O	-
P	-
Q	-
R	#18
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

Note

- (1) Addresses can be designated in an arbitrary order.
- (2) Addresses which do not need to be designated can be omitted.
- (3) Local variables in a sub part system are initialized every time the sub part system is activated. Default value is <empty>.
- (4) To use local variables in a sub part system, user macros must be available. For the available functions of each model, refer to the list.

Operation mode of a sub part system

- (1) The operation mode of sub part systems is used as "sub part system I operation mode". If the memory mode/MDI mode and the sub part system I operation mode are entered at the same time, the stop code (T01 0108) will be generated.
- (2) In a part system operating the sub part system I operation mode, the operation mode appears as "SUB" in the part system display of the operation screen.
If an alarm or warning occurs in a sub part system, the part system No. appears as "SUB" in the alarm/warning message of the operation screen.
- (3) If the sub part system control I command is issued to a part system that is not operating the sub part system I operation mode, an operation error (M01 1111) will occur.

Activation part system of a sub part system

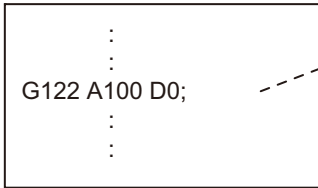
When issuing the sub part system control I command, designate the sub part system identification No. with command address B. (When there is no B command, it will be handled as the B1 command.) The sub part system identification No. and the sub part system No. to be called depend on the MTB specifications. (Parameter "#12049 SBS_no")

(Example 1) and (Example 2) show the operations when parameters are set as shown below. The available number of part systems depends on your machine's specifications.

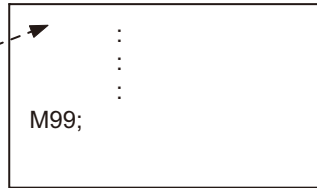
#12049	SBS_no	Sub part system I identification No.	\$1	\$2	\$3	\$4	\$5	\$6	\$7	\$8
			0	0	0	0	1	2	3	4

(Example 1) If the B command is omitted, \$5 corresponding to B1 will be activated.

Calling part system (\$1)

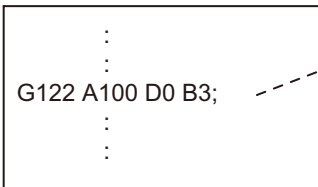


Sub part system (\$5)

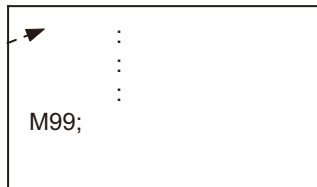


(Example 2) Sub part system identification No. (the part system No. to be activated and correspondence) can be specified with the B command.

Calling part system (\$1)



Sub part system (\$7)



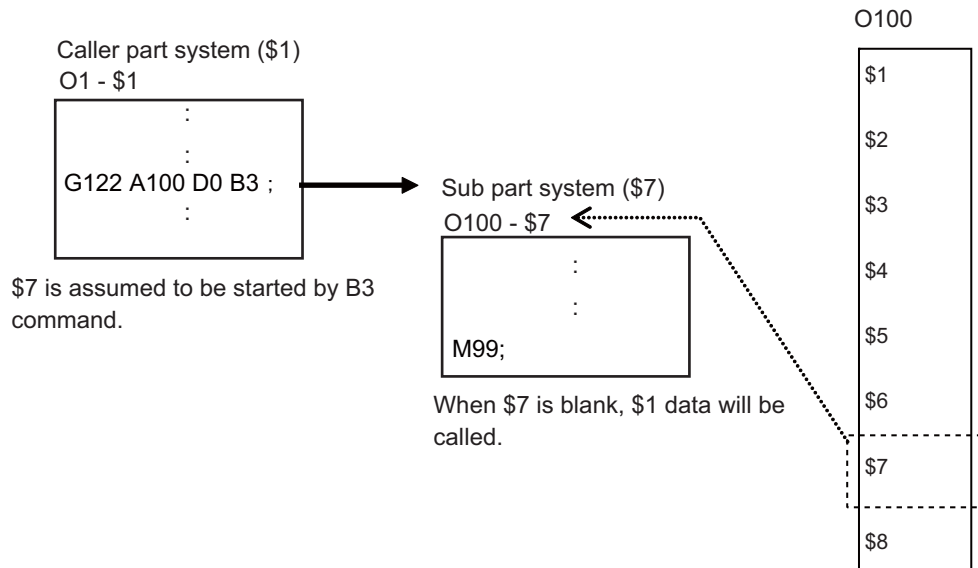
Operation program of a sub part system

When issuing the sub part system control I command, designate the program No. or program name to be operated in the sub part system with command address A or <file name>. If designation of the program is omitted, the machining program set in parameter "#12050 SBS_pro" will be started.

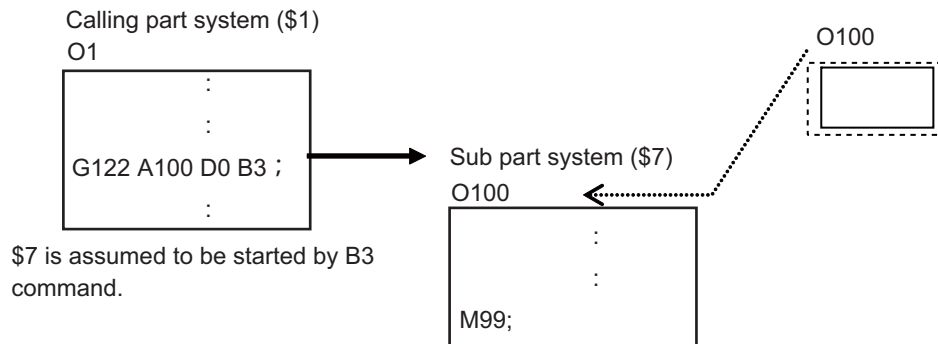
If a machining program is managed for each part system, the program of the part system designated as a sub part system will be operated (*1). If a machining program is commonly managed between part systems, the designated program will be operated.

(*1) If the program of the part system No. for the sub part system is empty, the program of the 1st part system (\$1) will be operated. If the program of the 1st part system is also empty, a program error (P461) will occur.

(1) If program is managed for each part system



(2) If program is commonly managed between part systems

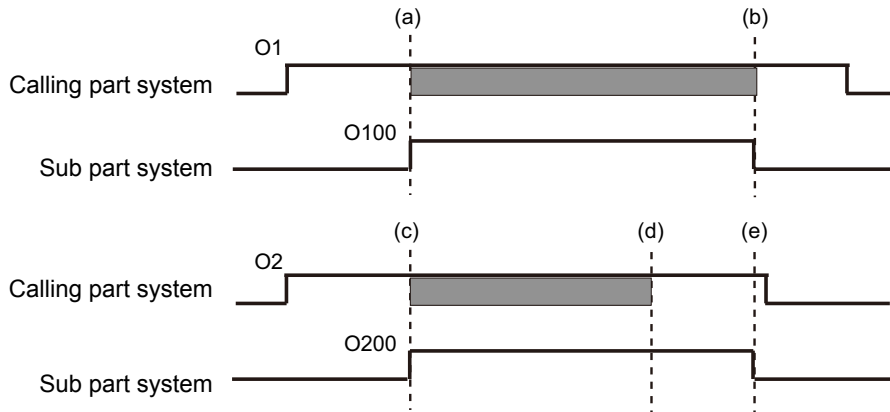
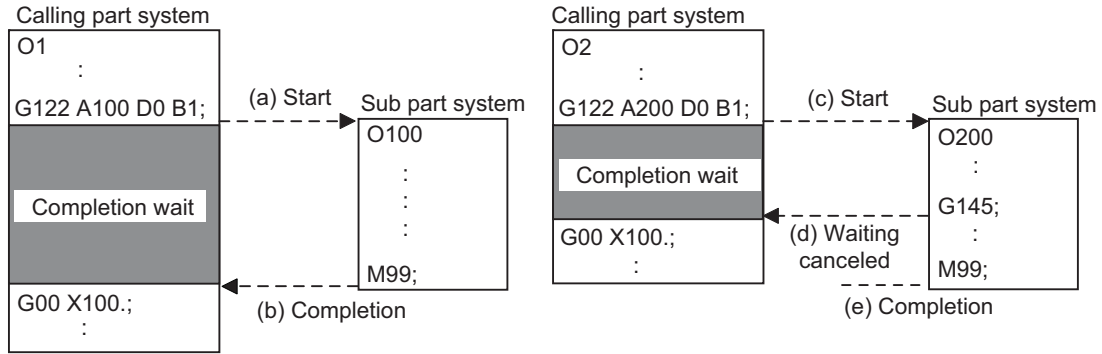


Sub part system activation with the completion wait method (D=0)

If "0" is designated for command address D when the sub part system control I command is issued, or if command address D is omitted, the calling part system will wait for the called sub part system to complete (to M99 or the end sequence No.) before starting the next block.

Meanwhile, if the completion wait cancel command (G145) is issued in a sub part system while the calling part system is in the sub part system completion standby state, the machine will shift to a parallel processing mode.

The following shows the operation and the activation timing of each part system.



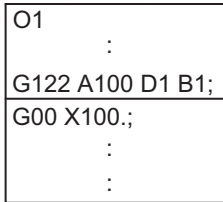
█ : Completion wait

Activation of a sub part system with parallel processing mode (D=1)

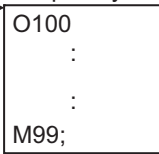
If "1" is designated for command address D when the sub part system control I command is issued, the following blocks of the calling part system and the first and the following blocks of the sub part system will be operated in parallel.

The following shows the operation and the activation timing of each part system.

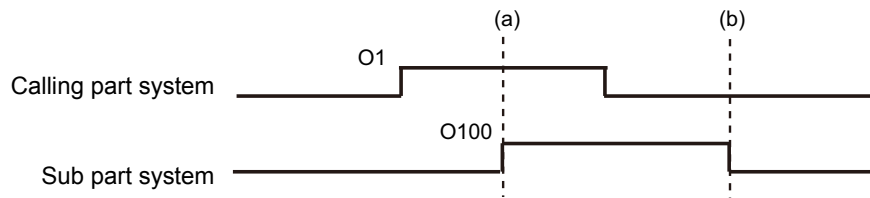
Calling part system



(a) Start → Sub part system



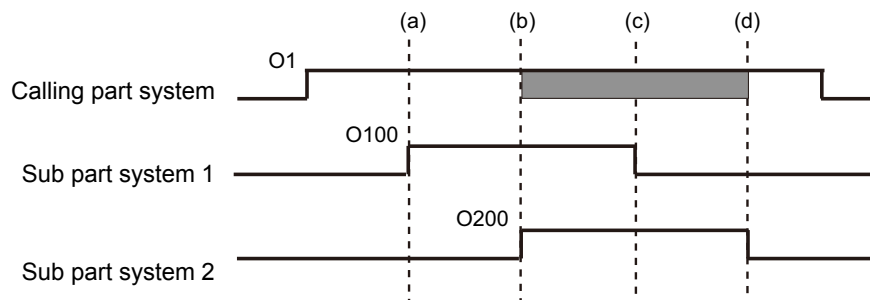
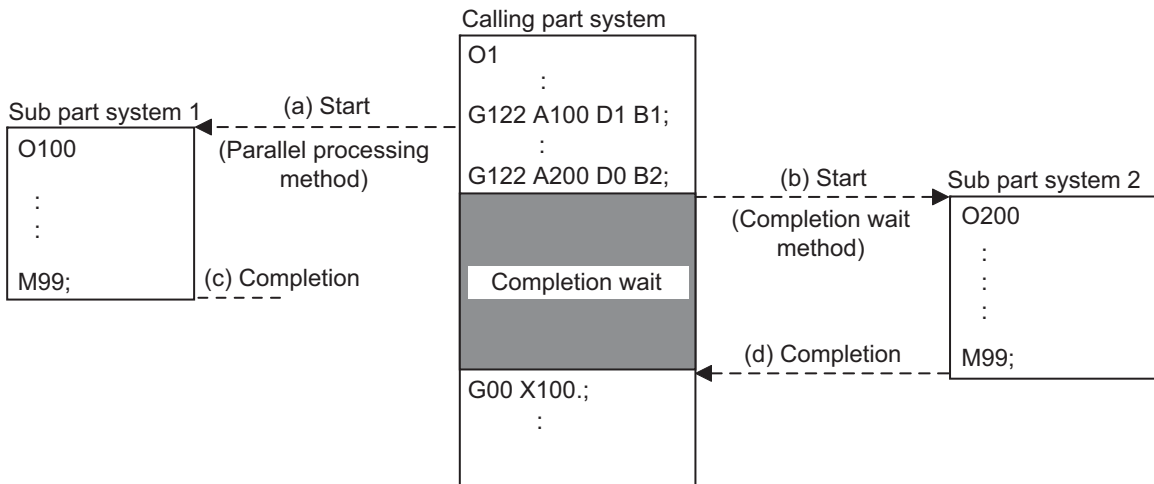
← (b) Completion



Activation of multiple sub part systems

Multiple sub part systems can be activated in parallel during separate processes by calling from a single part system. The number of sub part systems to be processed simultaneously depends on the model.

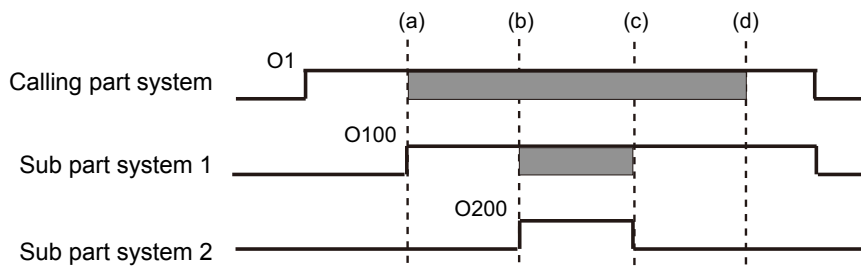
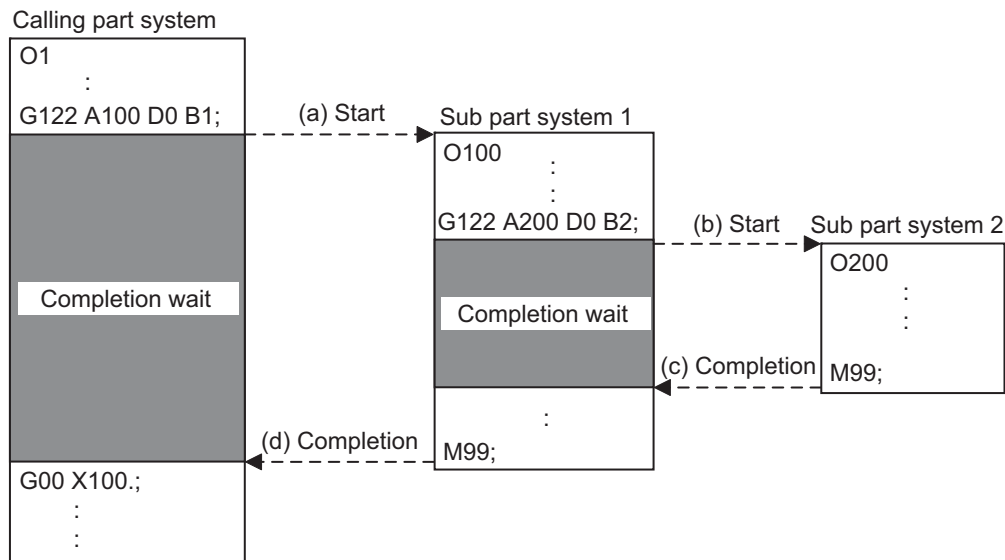
The following shows the operation and the activation timing of each part system.



█ : Completion wait

Activate a sub part system from another sub part system

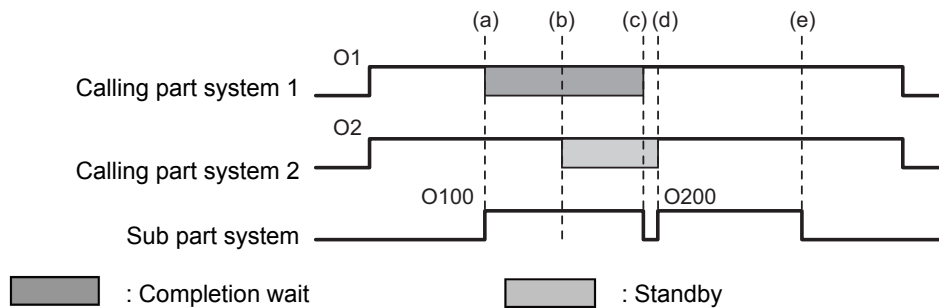
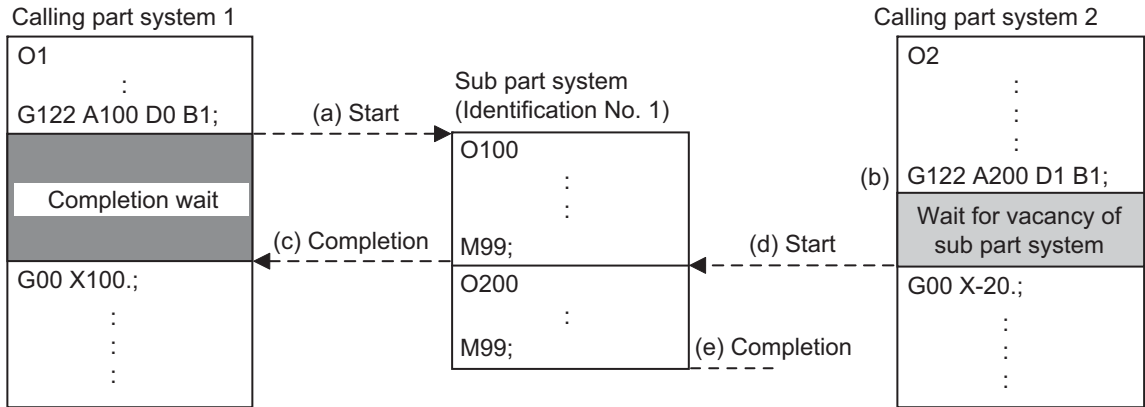
A sub part system can be activated from another sub part system.
 The number of sub part systems to be processed simultaneously depends on the model.
 The following shows the operation and the activation timing of each part system.



: Completion wait

Sub part system activation command to a sub part system being activated

If G122 is commanded while a sub part system is being activated, using the same identification No. (B command), the machine will wait for the earlier sub part system to complete activation, before activating the next sub part system.



Operation example

In the following example, the machining start timing is accelerated by controlling auxiliary axis with a sub part system and operating the main part system and the sub part system in parallel. The tool positioning starts to the machining start point at the same time (time T1) as the start of gantry retract by using sub part system completion wait cancel command (G145) in the flow from mounting the workpiece to moving to cut start position, after feeding and mounting the workpiece with the gantry, in order to reduce the cycle time. (The machine configuration below is a sample only.)

[Axis configuration]

Main part system (\$1) : X1 axis, Z1 axis => Tool

Sub part system (\$2) : X2 axis, Z2 axis => Workpiece feed gantry

[Machining process]

- (a) Feed the workpiece
- (b) Clamp the workpiece
- (c) Retract the gantry
- (d) Move to cut start position

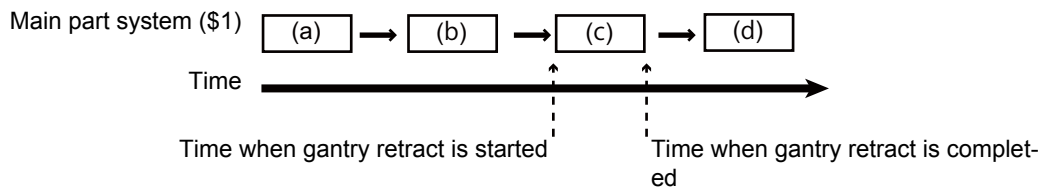
16 Multi-part System Control

(1) Machining process when sub part system control is OFF

Main part system (\$1)

```
O1
:
:
G140 X=X2 Z=Z2; ... (a)
G00 X50.;
G00 Z20.;
M20; ... (b)
G00 X0. Z0.; ... (c)
G141;
G00 X30. Z-15.; ... (d)
G01 Z-20. F10.;
:
```

G140: Arbitrary axis exchange command (Lathe system only)
 G141: Arbitrary axis exchange return command (Lathe system only)
 M20 : M code of workpiece mounting



After the gantry is retracted, cut start position is determined.

(2) Machining process when sub part system control is ON

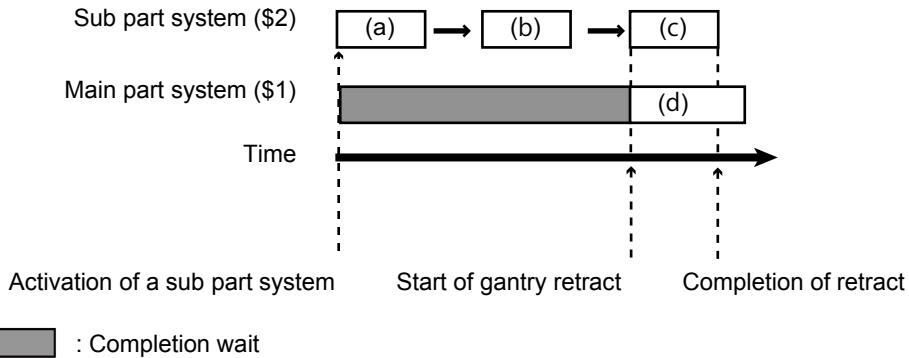
Main part system (\$1)

```
O1
:
:
G122 A100 D0 B1;
G00 X30. Z-15.; ... (d)
G01 Z-20. F10.;
:
```

Sub part system (\$2)

```
O100
G00 X50.; ... (a)
G00 Z20.;
M20; ... (b)
G145;
G00 X0. Z0.; ... (c)
:
M99;
```

M20 : M code of workpiece mounting



Processes after "(c) Retract gantry" and "(d) Move to cut start position" will be operated in parallel.



Relationship with other functions

Timing synchronization with sub part system

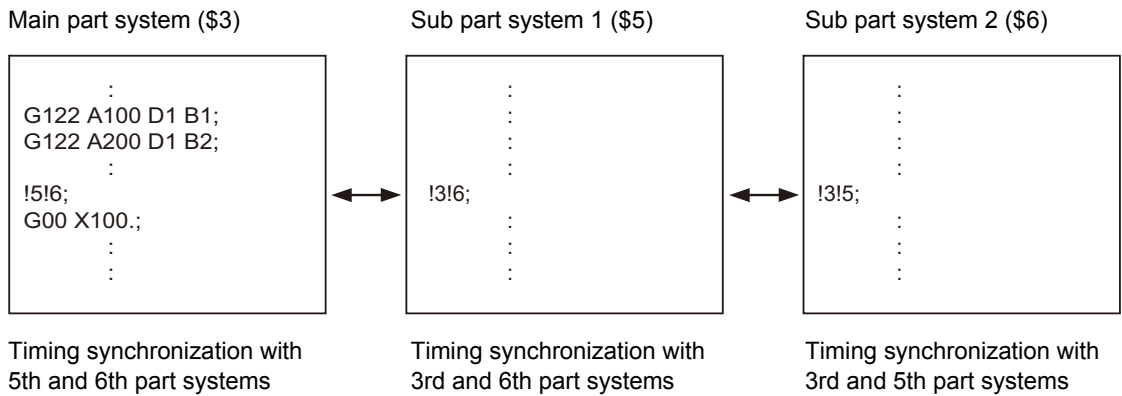
While a sub part system is under control, timing synchronization between part systems can be issued with the "![Part system No.]" command. To synchronize timing between a main part system and a sub part system, or between sub part systems, it is also possible to designate a sub part system identification No. (B command) as shown below. However, the number of part systems that can be used is limited by the specifications.

![Sub part system identification No.]

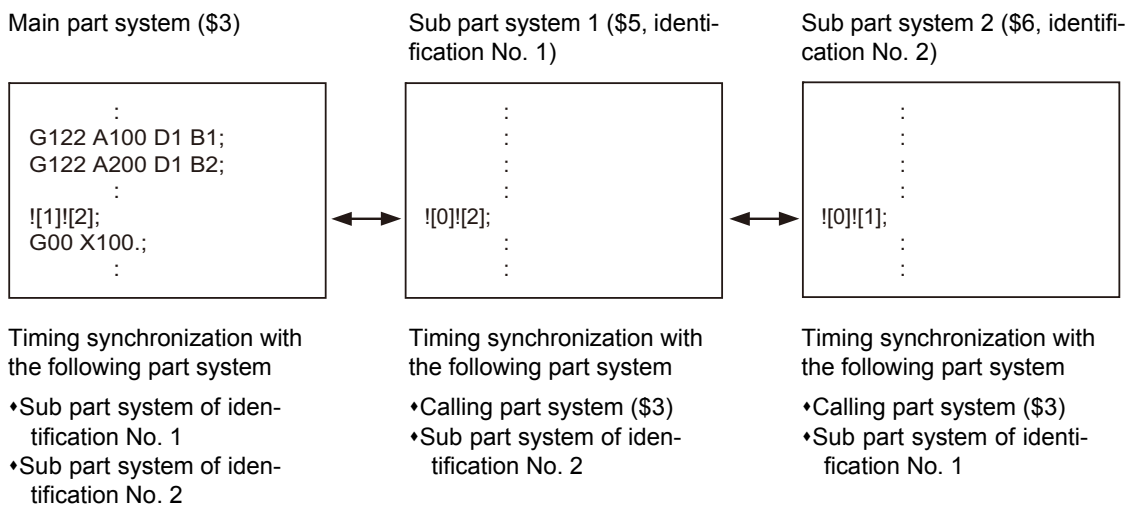
For example, to synchronize timing with the calling part system, command "![0]". Note that, designate the calling part system with "![0]", not the main part system.

(Example 1) and (Example 2) shown below are examples of the timing synchronization operation between the main part system (\$3), sub part system 1 (\$5, identification No. 1), and sub part system 2 (\$6, identification No. 2).

(Example 1) Timing synchronization by designating a part system No.



(Example 2) Timing synchronization by designating a sub part system identification No.



Timing synchronization operation ignore signal

Whether to ignore the "[Sub part system identification No.]" command or not depends on the MTB specifications. (Setting of parameter "#1279 ext15/bit0" and the following PLC signals)

#1279 ext15/bit0	PLC signal for ignoring timing synchronization between part systems	Operation	
		If the other part system is being activated as a sub part system	If the other part system is not being activated as a sub part system
0	ON	The timing synchronization operation is ignored when activation of a sub part system is completed for the other part system.	Program error (P35)
	OFF		
1	ON	Ignore the timing synchronization operation.	
	OFF	Execute the timing synchronization operation between part systems.	

Arbitrary axis exchange control

With the sub part system control I, axes that belong to the sub part system when the sub part system is activated can be controlled. To change the axis to be controlled, exchange axes (to transfer the control rights of the specified axis from other part systems to the own part system) with the arbitrary axis exchange return command (G140).

Tool functions

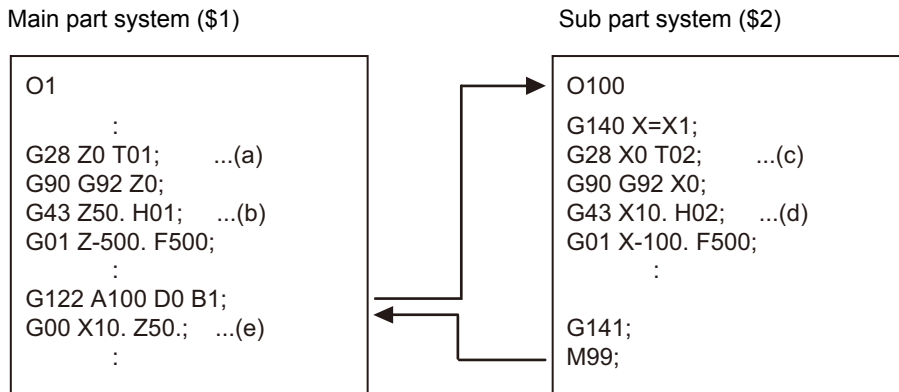
If the tool No. is changed (T command) in the program run of a sub part system, the T code data will be changed for the sub part system only. The T code data will not be changed for the main part system or other sub part systems.

Tool compensation

When an axis in the main part system, for which the tool compensation has been commanded, is moved to a sub part system with the arbitrary axis exchange or other operation, the tool compensation will be maintained. Also, when an axis (*1) in a sub part system, for which tool compensation has been commanded, is moved to the main part system or another sub part system with the arbitrary axis exchange operation, tool compensation will be maintained.

Whether the arbitrary axis exchange function is available depends on the specifications of your machine tool.

(*1) If tools are managed for each part system, the offset data to be referenced when the tool compensation command is issued in a sub part system is used as setting values for the sub part system. (The setting value of the main part system will not be referenced.)



G140: Arbitrary axis exchange command
G141: Arbitrary axis exchange return command

	T code data		Compensation amount	
	\$1	\$3	Z1	X1
(a)	1	-	:	:
(b)	1	-	L(01_\$1)	:
(c)	1	2	L(01_\$1)	:
(d)	1	2	L(01_\$1)	L(02_\$3)
(e)	1	2	L(01_\$1)	L(02_\$3)

L(N_\$M) indicates the compensation amount of compensation No. N in the Mth part system.

User macro

The sub part system control I command does not affect nesting in user macros and subprograms. It can be commanded from a subprogram nested at the deepest level.

Resetting

- (1) If the NC reset signal is input to the main part system, the operation of the main part system will be reset and end immediately. However, the operation of sub part systems will continue. The reset operation of the sub part system follows the NC reset signal of the sub part system.
- (2) If the NC reset signal is input to an operating sub part system, the operation of the sub part system will end immediately. Therefore, if the calling part system is in the sub part system completion standby state, the sub part system is reset, and at the same time, the calling part system cancels the standby state, and the following block will be executed.

Buffer correction

If both of the following conditions (1) and (2) are satisfied, the buffer correction is disabled. (The buffer correction window will not open even if the program correction key is pressed.)

- (1) The next block is G122 command (including "macro statement + G122 command").
- (2) The program designated by G122 is the same as that of the calling part system.

O100	
:	
G00 Z50.;	Buffer correction possible
G00 X100.;	Buffer correction impossible
G122 A100 P77 D0 B1;	Designated program is the program (O100) of its own part system
G00 Y30.;	Buffer correction possible
:	
N77	
:	Program operated in sub part system
M99;	

Machining time computation

The completion wait time of the sub part system control I command (G122) will not be added to the machining time computation for the main part system.

Program restart

If the restart search from the block of the G122 command is attempted, a program error (P49) will occur.

Illegal modal of a sub part system control I command

If the sub part system control I (G122) is commanded during the following G command modal, a program error (P652) will occur.

- User macro modal call (G66, G66.1)
- Fixed cycle modal
- High-speed machining mode (G05P1, G05P2)
- High-speed high-accuracy control (G05.1Q1, G05P10000, G05P20000)

Manual arbitrary reverse run

The sub part system control I (G122) is ignored at the reverse run or the forward run after the reverse run.

Because the sub part systems are in a mode in which reverse run is prohibited, reverse run cannot be carried out in sub part systems.

**Precautions**

- (1) The sub part system control I command (G122) is a G code that must be issued alone. If another G code is commanded in the same block, a program error (P651) or (P32) occurs.
 - If another G code is commanded before G122 (for example, when "G00 G122" is commanded), a program error (P651) occurs.
 - If another G code is commanded after G122 (for example, when "G122 G00" is commanded), a program error (P32) occurs.
- (2) While the sub part system I operation mode is in operation, even if the sub part system is not being activated, automatic operation cannot be started with the automatic operation start signal (ST). The stop code (0146) will be generated. However, when a sub part system is being activated, automatic operation is started with the automatic operation start signal (ST).
- (3) If a sub part system identification No. of its own part system is designated for the B command with the sub part system control I command (G122), a program error (P650) will occur.
- (4) The PLC signal of the sub part system references the state of the sub part system. (The signal state of the main part system will not be taken over.)
- (5) Parameters per part system of the sub part system follow the setting in the sub part system. Therefore, parameters must also be set in the sub part system.
- (6) If the sub part system completion wait cancel command (G145) is issued in the main part system, a program error (P34) occurs.
- (7) The following operations are performed in the M80 series/C80 series. These parameter settings depend on the MTB specifications.
 - Activation of a sub part system is only possible in sub part systems that are reserved using the parameter "#1483 SBS1_sys num". If the sub part system activation command is issued to a main part system (*1), an operation error (M01 1111) occurs.
 - (*1) This refers to a case in which the sub part system I operation mode is established (SBSM: ON) using the PLC signal before G122 is commanded.
 - Operation searches cannot be carried out in sub part systems that are reserved using the parameter "#1483 SBS1_sys num".
 - (M80 series only) If the values set for the parameters "#1483 SBS1_sys num" and "#1474 SBS2_sys num" are both "1" or more, an MCP alarm (Y05 1483) will occur.

High-speed High-accuracy Control

17.1 High-speed Machining Mode

17.1.1 High-speed Machining Mode I, II ; G05 P1, G05 P2



Function and purpose

This function runs a machining program for which a freely curved surface has been approximated by fine segments at high speed.

A higher fine segment processing capability leads to a faster cutting speed, resulting in a shorter cycle time and a better machining surface quality.

The high-speed high-accuracy control I/II/III enable not only the high-speed machining mode but also the high-accuracy control mode. Use the high-speed high-accuracy control I/II/III for machining which needs to make an edge at a corner or reduce an error from an inner route of curved shape.

This function can be used simultaneously for up to two part systems depending on the MTB specifications.

kBPM, the unit for the fine segment processing capability, is an abbreviation of "kilo blocks per minute" and refers to the number of machining program blocks that can be processed per minute.

In the main text, the axis address refers to the address of an axis that exits on the machine.

It corresponds to the address designated in the parameters "#1013 axname" and "#1014 incax".

These parameter settings depend on the MTB specifications.

For one part system

G01 block fine segment capacity for 1mm segment (unit: kBPM)

Mode	Command	Maximum feedrate when 1mm segment G01 block is executed (kBPM)			
		M850/M830	M80W	M80	
				Type A	Type B
High-speed machining mode I	G05 P1	33.7	33.7	33.7	16.8
High-speed machining mode II	G05 P2	168	67.5	67.5	67.5

Note

(1) The above performance applies under the following conditions.

- 6-axis system (including spindle) or less
- 1-part system
- 3 axes or less commanded simultaneously in G01
- The block containing only the axis name and movement amount (Macro and variable command are not included.)
- In the "G61.1" high-accuracy control mode or cutting mode (G64)
- During tool radius compensation cancel (G40) (only in the high-speed machining mode II)

When the above conditions are not satisfied, the given feedrate may not be secured.

(2) The performance in the table may vary depending on the combination with other function.

Multi-part system (high-speed machining mode II)

G01 block fine segment capacity for 1mm segment (unit: kBPM)

	Specified number of part systems (#8040 = 1)	Maximum feedrate when 1mm segment G01 block is executed (kBPM)		
		M850/M830	M80	
			Type A	Type B
1-part system	1 part system	168	67.5	67.5
2-part system	1 part system only	100	67.5	67.5
	Two part systems simultaneously	67.5	33.7	33.7
4-part system Up to 16 axes	1 part system only	- (*1)	- (*1)	- (*1)
	Two part systems simultaneously	- (*1)	- (*1)	- (*1)
5 part systems or more or 17 axes or more	1 part system only	- (*1)	- (*1)	- (*1)
	Two part systems simultaneously	- (*1)	- (*1)	- (*1)

(*1) This system cannot be used for this model.

(*2) There are no high-speed machining mode II specifications.

Note

(1) The above performance applies under the following conditions.

- ♦3 axes commanded simultaneously in G01
- ♦The block containing only the axis name and movement amount (Macro and variable command are not included.)
- ♦Tool radius compensation cancel (G40) mode

When the above conditions are not satisfied, the given feedrate may not be secured.

(2) The performance in the table may vary depending on the combination with other function.

(3) The number of part systems and axes that can be used depends on the specifications of your machine tool.



Command format**High-speed machining mode I ON**

```
G05 P1 ;
```

High-speed machining mode II ON

```
G05 P2 ;
```

High-speed machining mode I/II OFF

```
G05 P0 ;
```

In addition to the G05 P0 command, the high-speed machining mode I is canceled when the high-speed machining mode II (G05 P2) is commanded.

In reverse, the high-speed machining mode II is canceled when the high-speed machining mode I (G05 P1) is commanded.

Command G05 in an independent block. A program error (P33) will occur if a movement or other command is additionally issued in a G05 command block. A program error (P33) will also occur if there is no P command in a G05 command.

In addition to cancel the high-speed machining mode II, a G05 P0 command is also used to cancel the high-speed high-accuracy control II/III.

Refer to "17.3 High-speed High-accuracy Control" for details.



Detailed description

- (1) The override, maximum cutting speed clamp, single block operation, dry run, manual interruption and graphic trace and high-accuracy control mode are valid even during the high-speed machining mode I/II.
For a part system that uses the high-speed machining mode II, "1" must be set for the parameter "#8040 High-SpeedAcc". By default, the high-speed machining mode II can only be used in the first part system.
- (2) When using the high-speed machining mode II, setting to eliminate the speed fluctuation at the seams between the arc and the straight line, or between arcs depends on the MTB specifications (parameter "#1572 Cirorp/bit1").
- (3) Combination with high-accuracy control
The high-speed machining mode and high-accuracy control can be used simultaneously by taking the following steps:
 - (a) Set "1" for the parameter "#8040 High-SpeedAcc".
 - (b) Command "G05 P2" and "G08 P1" or "G61.1" from the machining program.
 The parameter "#8040 High-SpeedAcc" can be set to "1" for up to two part systems. If "0" is set for all part systems, the first and second part systems can use the high-speed machining mode and high-accuracy control simultaneously.

Also refer to the following for the description of each function:

 - ♦High-accuracy control: "17.2 High-accuracy Control"
 - ♦Simultaneous usage of the high-speed machining mode and high-accuracy control: "17.3 High-speed High-accuracy Control"
- (4) If the variable command, variable operation command, or macro control statement is issued while high-speed machining mode II is valid, the fine segment processing capacity is not guaranteed. However, only when the variable commands and variable four-basic-arithmetic operation commands shown below are issued following the axis address or the F address of the cutting feedrate command, the fine segment processing capacity is guaranteed.

- (a) Referencing common variables or local variables

Common variables or local variables can be referenced (example: X#500, Y#1, Z##100, A#[#101], etc.).

- (b) Four basic arithmetic rule

Four basic arithmetic rule (+, -, *, /) operations are available, and also the operation priority can be designated using parentheses () ([#500+1.0]*#501, etc.).

If a common variable or local variable is referenced using the variable number operated with a macro operation instruction, a program error (P232) may occur. In this case, set the operated value to the variable before referencing the variable.

Example that causes an error	F#[FIX[100.1]];
Example that does not cause an error	#500 = FIX[100.1] ; F#[#500] ;



Program example

High-speed machining mode I

G28 X0. Y0. Z0. ;	
G91 G00 X-100. Y-100. ;	
G01 F10000 ;	
G05 P1 ;	High-speed machining mode I ON
:	
X0.1 Y0.01 ;	
X0.1 Y0.02 ;	
X0.1 Y0.03 ;	
:	
G05 P0 ;	High-speed machining mode I OFF
M30 ;	



Relationship with other functions

Relationship between the high-speed machining mode II and G code functions

Column A: Operation when the additional function is commanded while the high-speed machining mode II is enabled
 Column B: Operation when the high-speed machining mode II (G05P2) is commanded while the additional function is enabled

○: The high-speed machining mode II and the additional function are both enabled

△: The high-speed machining mode II is temporarily canceled, while the additional function is enabled

x: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)

-: No combination

□: Others

Group	G code	Function name	A	B
0	G04	Dwell	△	-
	G05P0	High-speed machining mode II OFF High-speed high-accuracy control II OFF High-speed high-accuracy control III OFF	□ (*1)	□ (*2)
	G05P2	High-speed machining mode II ON	□ (*3)	□ (*3)
	G05P10000	High-speed high-accuracy control II ON	□ (*2)	□ (*2)
	G05P20000	High-speed high-accuracy control III ON	□ (*2)	□ (*3)
	G05.1Q0	High-speed high-accuracy control I OFF Spline interpolation OFF	□ (*3)	□ (*2)
	G05.1Q1	High-speed high-accuracy control I ON	□ (*2)	□ (*2)
	G05.1Q2	Spline interpolation ON	○ (*8)	○ (*8)
	G07	Hypothetical axis interpolation	△ (*8)	△ (*8)
	G08P0	High-accuracy control OFF	□ (*3)	□ (*2)
	G08P1	High-accuracy control ON	□ (*4)	□ (*4)
	G09	Exact stop check	△	-
	G10 I_ J_ G10 K_	Parameter coordinate rotation input	△ (*8)	- (*8)
	G10 L2	Compensation data input by program	△	-
	G10 L70 G10 L50	Parameter input by program	△	-
	G27	Reference position check	△	-
	G28	Reference position return	△	-
	G29	Start position return	△	-
	G30	2nd to 4th reference position return	△	-
	G30.1- G30.6	Tool exchange position return	△	-
	G31	Skip Multiple-step skip 2	△	-
	G31.1- G31.3	Multi-step skip	△	-
	G34-G36 G37.1	Special fixed cycle	△	-
	G37	Automatic tool length measurement	△	-
	G38	Tool radius compensation vector designa- tion	△	-
	G39	Tool radius compensation corner circular command	△	-

Group	G code	Function name	A	B
0	G52	Local coordinate system setting	Δ	-
	G53	Machine coordinate system selection	Δ	-
	G60	Unidirectional positioning	Δ	-
	G65	User macro simple call	\square (*5)	\square (*6)
	G92	Coordinate system setting	Δ	-
	G92.1	Workpiece coordinate system preset	Δ (*8)	- (*8)
	G122	Sub part system control I	\times (P652) (*8)	\square (*7) (*8)
	G144	Sub part system control II	\times (P652) (*8)	\square (*7) (*8)
1	G00	Positioning	Δ	Δ
	G01	Linear interpolation	\circ	\circ
	G02 G03	Circular interpolation	\circ	\circ
	G02.1 G03.1	Spiral interpolation	Δ (*8)	Δ (*8)
	G02.3 G03.3	Exponential interpolation	Δ (*8)	Δ (*8)
	G02.4 G03.4	3-dimensional circular interpolation	Δ (*8)	Δ (*8)
	G06.2	NURBS interpolation	\circ (*8)	\circ (*8)
	G33	Thread cutting	Δ	Δ
2	G17 to G19	Plane selection	\circ	\circ
3	G90	Absolute value command	\circ	\circ
	G91	Incremental value command	\circ	\circ
4	G22	Stroke check before travel ON	Δ	Δ
	G23	Stroke check before travel OFF	\circ	\circ
5	G93	Inverse time feed	Δ (*8)	Δ (*8)
	G94	Asynchronous feed (feed per minute)	\circ	\circ
	G95	Synchronous feed (feed per revolution)	Δ	Δ
6	G20	Inch command	\circ	\circ
	G21	Metric command	\circ	\circ
7	G40	Tool radius compensation cancel	\circ	\circ
	G41 G42	Tool radius compensation	\circ	\circ
	8	G43 G44	Tool length offset	\circ
G43.1		Tool length compensation along the tool axis	\circ (*8)	\circ (*8)
G43.4 G43.5		Tool center point control	\circ (*8)	\circ (*8)
G49		Tool length offset cancel	\circ	\circ
9	G80	Fixed cycle cancel	\circ	\circ
	Group 9 Other than G80	Fixed cycle	Δ	Δ
10	G98	Fixed cycle initial level return	\circ	\circ
	G99	Fixed cycle R point return	\circ	\circ
11	G50	Scaling cancel	\circ	\circ
	G51	Scaling ON	Δ	Δ
12	G54 to G59 G54.1	Workpiece coordinate system selection	\circ	\circ

Group	G code	Function name	A	B
13	G61	Exact stop check mode	Δ	Δ
	G61.1	High-accuracy control	○	○
	G61.2	High-accuracy spline	○ (*8)	○ (*8)
	G61.4	Spline interpolation 2	○ (*8)	○ (*8)
	G62	Automatic corner override	Δ	Δ
	G63	Tapping mode	Δ	Δ
	G64	Cutting mode	○	○
14	G66 G66.1	User macro modal call	Δ	Δ
	G67	User macro modal call Cancel	○	○
15	G40.1	Normal line control cancel	○	○
	G41.1 G42.1	Normal line control	× (P29) (*8)	× (P29) (*8)
	G68	Coordinate rotation by program ON	Δ	Δ
16	G68.2 G68.3	Inclined surface machining command	○ (*8)	○ (*8)
	G69	Coordinate rotation cancel	○	○
	17	G96	Constant surface speed control ON	○
G97		Constant surface speed control OFF	○	○
18	G15	Polar coordinate command OFF	○	○
	G16	Polar coordinate command ON	Δ	Δ
19	G50.1	Mirror image OFF	○	○
	G51.1	Mirror image ON	○	○
21	G07.1	Cylindrical interpolation	X (P34)	X (P481)
	G12.1	Polar coordinate interpolation ON	X (P34)	X (P481)
	G13.1	Polar coordinate interpolation OFF	○	○
27	G54.4P0	Workpiece installation error compensation cancel	○	○
	G54.4 P1 to P7	Workpiece installation error compensation	○ (*8)	○ (*8)

(*1) Disables the high-speed machining mode II.

(*2) Enables the high-speed machining mode II.

(*3) High-speed machining mode II continues.

(*4) Enables the high-speed machining mode II and high-accuracy control.

(*5) Enables the high-speed machining mode II in a macro program.

(*6) Enables the high-speed machining mode II if G05P2 is commanded in a macro program.

(*7) Enables the high-speed machining mode II if G05P2 is commanded in a sub part system.

(*8) In M80 Type B, the following program errors occur depending on the G codes.

G code	Program error	G code	Program error
G05.1Q2, G92.1, G122, G61.2, G61.4	P39	G10 I_J_/G10 K_	P260
G144, G54.4 P1-P7	P34	G06.2	P550
G07	P80	G93	P124
G02.1, G03.1	P73	G41.1, G42.1	P900
G02.3, G03.3	P611	G43.1	P930
G02.4, G03.4	P76	G43.4, G43.5	P940
		G68.2, G68.3	P950

Relationship between the high-speed machining mode II and functions other than G codes

Column A: Operation when the additional function is commanded while the high-speed machining mode II is enabled

Column B: Operation when the high-speed machining mode II (G05P2) is commanded while the additional function is enabled

○: The high-speed machining mode II and the additional function are both enabled

Δ: The high-speed machining mode II is temporarily canceled, while the additional function is enabled

x: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)

-: No combination

□: Others

Function name	A	B
SSS ON	-	○
Mirror image by parameter setting ON	-	Δ
PLC mirror image ON	-	Δ
Coordinate rotation by parameter	-	Δ
Subprogram call (M98)	□ (*1)	□ (*2)
Figure rotation (M98 I_J_K_)	□ (*8) (*10)	□ (*9) (*10)
Timing synchronization between part systems	□ (*3)	-
MTB macro	□ (*4)	□ (*5)
Macro interruption	□ (*6)	□ (*7)
Corner chamfering/Corner R	Δ	-
Linear angle command	Δ	-
Geometric command	Δ	-
Chopping	○	○
Fairing/ smooth fairing ON	○	○
Optional block skip	○	-

(*1) Enables the high-speed machining mode II in a subprogram.

(*2) Enables the high-speed machining mode II if G05P2 is commanded in a subprogram.

(*3) Enables timing synchronization.

(*4) Enables the high-speed machining mode II in a MTB program.

(*5) Enables the high-speed machining mode II if G05P2 is commanded in a MTB program.

(*6) Enables the high-speed machining mode II in an interrupt program.

(*7) Enables the high-speed machining mode II if G05P2 is commanded in an interrupt program.

(*8) Disables the high-speed machining mode II in a figure rotation subprogram.

(*9) The high-speed machining mode II is disabled even if G05P2 is commanded in a figure rotation subprogram.

(*10) In M80 Type B, the following program error occurs depending on the function type.

Function	Program error
Figure rotation (M98 I_J_K_)	P250



Precautions

- (1) If "G05 P1(P2)" is commanded when the high-speed machining mode I/II specifications are not provided, a program error (P39) will occur.
- (2) The automatic operation process has priority in high-speed machining mode I/II, and as a result, the screen display may slow down.
- (3) The speed will decelerate once at the G05 command block, so turn ON and OFF when the tool separates from the workpiece.
- (4) When carrying out operations in high-speed machining mode I/II by communication or tape mode, the machining speed may be suppressed depending on the program transmission speed limit.
- (5) Command G05 in an independent block.
- (6) A decimal point is invalid for the P address in the G05 command block.
- (7) The P addresses, which are valid in the G05 command block, are P0, P1 and P2 only.
If other P addresses are commanded, a program error (P35) will occur.
If there is no P command, a program error (P33) will occur.
- (8) The machining speed may be suppressed depending on the number of characters in one block.

17.2 High-accuracy Control

17.2.1 High-accuracy Control ; G61.1, G08



Function and purpose

Machining errors caused by delays in control systems can be inhibited. This function is useful for machining which needs to make an edge at a corner or reduce an error from an inner route of curved shape. In high-accuracy control, acceleration/deceleration is performed not to cause machining error by pre-reading blocks and acceleration/deceleration is automatically performed according to a machining shape so that the machining error is inhibited with minimizing an extension of machining time.

	High-accuracy control OFF	High-accuracy control ON
Corner shape		
Curve shape		

Commands to enable high-accuracy control are as follows:

- High-accuracy control command (G08P1/G61.1)
- High-speed high-accuracy control I command (G05.1Q1)
- High-speed high-accuracy control II/III command (G05P10000/G05P20000)
- High-accuracy spline interpolation command (G61.2)

This function uses the following functions to minimize the increase in machining time while reducing the shape error.

- (1) Acceleration/deceleration before interpolation
- (2) Optimum speed control
- (3) Vector accuracy interpolation
- (4) Feed forward
- (5) S-pattern filter control

In the main text, the axis address refers to the address of an axis that exists on the machine.

It corresponds to the address designated in the parameters "#1013 axname" and "#1014 incax".

These parameter settings depend on the MTB specifications.



Command format**High-accuracy control valid**

G61.1 ;
or, G08 P1;

High-accuracy control invalid

G08 P0 ;
or, G command in G code group 13 except G61.1

High-accuracy control can be canceled with either command regardless of the command that has enabled the control.

Note

- (1) After "G08 P1" is commanded, G code group 13 is automatically switched to the G61.1 modal.
If the high-accuracy control mode is canceled by the "G08 P0" command, G code group 0 is switched to the "G08P0" modal and G code group 13 becomes the "commanded mode".



Detailed description

- (1) Feedrate command F is clamped with the "#2110 Clamp (H-precision)" (Cutting feed clamp speed for high-accuracy control mode) set with parameter.
- (2) Rapid traverse rate enables "#2109 Rapid(H-precision)" (Rapid traverse rate during high-accuracy control mode) set by the parameter.
- (3) When the "#2109 Rapid(H-precision)" is set to "0", the movement follows "#2001 rapid" (rapid traverse rate) set by the parameter. Also, when "#2110 Clamp (H-precision)" is set to "0", the speed will be clamped with "#2002 clamp" (Cutting clamp speed) set with parameter.
- (4) The modal holding state of the high-accuracy control mode depends on the MTB specifications (combination of the parameters "#1151 rstint" (reset initial) and "#1148 I_G611" (initial high-accuracy)).

Parameter		Default state	Resetting		
Reset initial (#1151)	Initial high-accuracy (#1148)	Power ON	Reset 1	Reset 2	Reset & rewind
OFF	OFF	OFF	Hold	OFF	
ON			OFF		
OFF	ON	ON	Hold	ON	
ON			ON		

Parameter		Emergency stop	Emergency stop cancel
Reset initial (#1151)	Initial high-accuracy (#1148)	Emergency stop switch or external emergency stop	Emergency stop switch or external emergency stop
OFF	OFF	Hold	Hold
ON			OFF
OFF	ON	Hold	Hold
ON			ON

Parameter		Block interruption	Block stop	NC alarm	OT
Reset initial (#1151)	Initial high-accuracy (#1148)	Mode changeover (automatic/manual) or feed hold	Single block	Servo alarm	H/W OT
OFF	OFF	Hold			
ON					
OFF	ON	Hold			
ON					

Hold: Modal hold

ON: Switches to the high-accuracy control mode

As for G61.1, the mode is switched to the high-accuracy mode, even if the other modes (G61 to G64) are valid.

OFF: The status of the high-accuracy control mode is OFF.

Acceleration/deceleration before interpolation

Acceleration/deceleration control is carried out for the movement commands to suppress the impact and to smooth out the velocity waveform when the machine starts or stops moving. However, if high-accuracy control is disabled, the corners at the block seams are rounded, and path errors occur regarding the command shape because acceleration/deceleration is performed after interpolation.

In the high-accuracy control function mode, acceleration/deceleration is carried out before interpolation to solve the above problems. This acceleration/deceleration before interpolation enables machining with a faithful path to the commanded shape of the machining program.

Furthermore, the acceleration/deceleration time can be reduced because the constant inclination acceleration/deceleration is performed for the acceleration/deceleration before interpolation.

(1) Basic patterns of acceleration/deceleration control in linear interpolation commands

Acceleration/deceleration waveform pattern	
<p>Normal mode</p>	
<p>High-accuracy control mode</p>	<p style="text-align: center;">(F) Combined speed (T) Time</p>

(a) Because of the acceleration/deceleration that controls the acceleration time to achieve the commanded speed at a constant level (constant time constant acceleration/deceleration), the acceleration/deceleration becomes more gentle as the command speed becomes slower (the acceleration/deceleration time does not change).
 (b) The time to achieve the commanded speed (G1tL) can be set independently for each axis. Note, however, that an arc shape will be distorted if the time constant differs among the base axes.

G1tL: G1 time constant (linear)
 (MTB-specified parameter #2007)

(a) Because of the acceleration/deceleration that controls the acceleration time to achieve the maximum speed (G1bF) set by a parameter at a constant level (constant inclination type linear acceleration/deceleration), the acceleration/deceleration time is reduced as the command speed becomes slower.
 (b) Only one acceleration/deceleration time constant (common for each axis) exists in a system.

G1bF: Maximum speed
 (MTB-specified parameter #1206)
 G1btL: Time constant
 (MTB-specified parameter #1207)

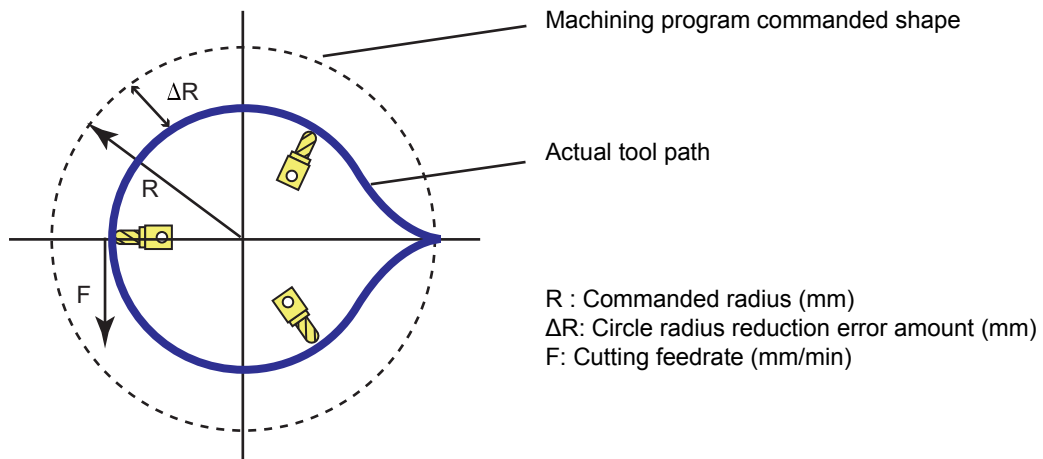
<Note>
 •G1bF and G1btL are values for specifying the inclination of the acceleration/deceleration time. The actual cutting feed maximum speed is clamped by the "#2002 clamp" value.

(2) Path control in circular interpolation commands

When commanding circular interpolation with the conventional post-interpolation acceleration/deceleration control method, the path itself that is output from the NC to the servo runs further inside the commanded path, and the circle radius becomes smaller than that of the commanded circle. This is due to the influence of the smoothing course droop amount for NC internal acceleration/deceleration.

With the pre-interpolation acceleration/deceleration control method, the path error is eliminated and a circular path faithful to the command results, because interpolation is carried out after the acceleration/deceleration control. Note that the tracking lag due to the position loop control in the servo system is not the target here.

The following shows a comparison of the circle radius reduction error amounts for the conventional post-interpolation acceleration/deceleration control and pre-interpolation acceleration/deceleration control in the high-accuracy control mode.



If an arc is commanded by a machining program as shown above, the error ΔR occurs for the commanded shape on the actual tool path. In the normal mode (acceleration/deceleration after interpolation), ΔR is caused by acceleration/deceleration of NC and lag of servo system. High-accuracy control (acceleration/deceleration before interpolation), however, can eliminate errors caused by acceleration/deceleration of NC. By additionally using the feed forward control, it is also possible to reduce errors caused by lag of servo system.

The compensation amount of the circle radius reduction error (ΔR) is theoretically calculated as shown in the following table.

Post-interpolation acceleration/deceleration control (normal mode)	Pre-interpolation acceleration/deceleration control (high-accuracy control mode)
Linear acceleration/deceleration $\Delta R = \frac{1}{2R} \left[\frac{1}{12} T_s^2 + T_p^2 \right] \left(\frac{F}{60} \right)^2$	Linear acceleration/deceleration $\Delta R = \frac{1}{2R} \left\{ T_p^2 \left[1 - K_f^2 \right] \right\} \left(\frac{F}{60} \right)^2$
Exponential function acceleration/deceleration $\Delta R = \frac{1}{2R} \left[T_s^2 + T_p^2 \right] \left(\frac{F}{60} \right)^2$	(a) Because the item T_s can be ignored by using the pre-interpolation acceleration/deceleration control method, the radius reduction error amount can be reduced. (b) Item T_p can be negated by making $K_f = 1$.

T_s : Acceleration/deceleration time constant in the NC (s)

T_p : Servo system position loop time constant (s) (inverse number to "#2203 PGN1")

K_f : Feed forward coefficient

$K_f = \text{fwd_g} / 1000$ (fwd_g: #2010 Feed forward gain)

Optimum speed control

When the moving direction is changed on the corner, arc, etc., acceleration corresponding to the amount of change and the feedrate is generated. When the acceleration is large, there is a possibility of machine vibration and it may leave stripes on the machining surface.

In the high-accuracy control mode, the deceleration control (optimum speed control) is performed to keep the generated acceleration under the allowance that has been designed with the parameter so that the problem mentioned above can be solved. The optimum speed control suppresses the machine vibration and enables highly accurate machining while minimizing the extension of cycle time.

Corner deceleration

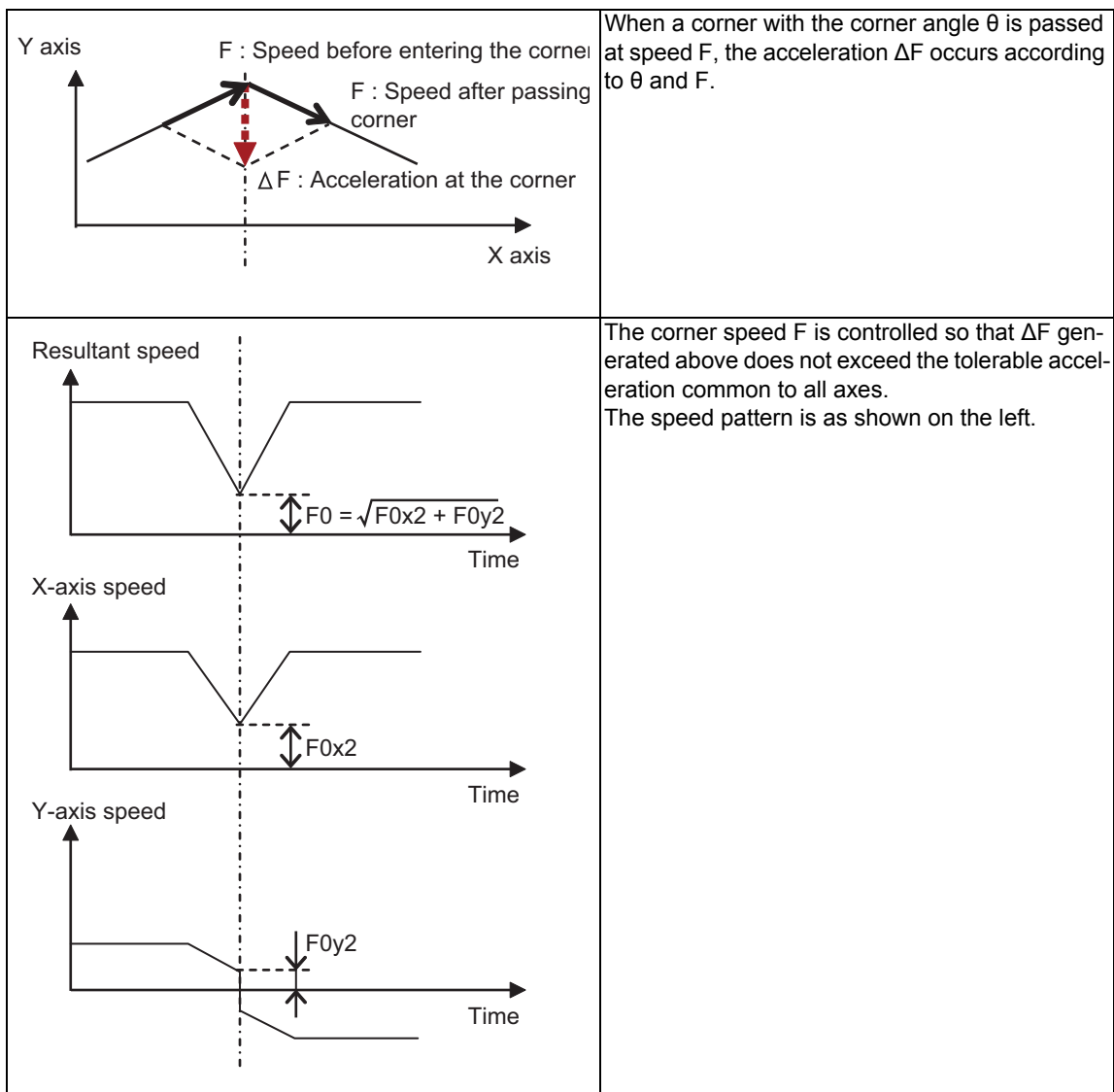
Consists of optimum corner deceleration and tolerable acceleration control for each axis.

Arc speed clamp

Controls deceleration so that the combined acceleration on an arc is kept below the tolerable acceleration common to all axes. This can suppress path errors (circle radius reduction error amount) on an arc to a certain level.

(1) Optimum corner deceleration

Highly accurate edge machining can be achieved by controlling deceleration so that the combined acceleration at the seam between blocks is kept under the tolerable acceleration common to all axes, which is determined by "#1206 G1bF (maximum speed)", "#1207 G1btL (time constant)", and accuracy coefficient. When entering in a corner, optimum speed for the corner (optimum corner speed) is calculated from the angle with the next block (corner angle) and the tolerable acceleration common to all axes. The machine decelerates to the speed in advance, and then accelerates back to the command speed after passing the corner.



Optimum corner deceleration is not carried out when blocks are smoothly connected, because deceleration is not necessary. The criteria for whether the connection is smooth or not can be designated by the machining parameter "#8020 DCC ANGLE". If the corner angle is equal to or less than the corner deceleration angle, the connection is judged to be smooth and optimum corner deceleration is not carried out.

The edge accuracy can be further improved by setting a greater accuracy coefficient. A greater accuracy coefficient, however, reduces the optimum corner speed, which may increase the cycle time. Setting a negative accuracy coefficient can increase the optimum corner speed and reduce the cycle time.

As shown below, different accuracy coefficients can be used depending on the parameter "#8021 COMP_CHANGE", and the tolerable acceleration common to all axes can be obtained with the following formula:

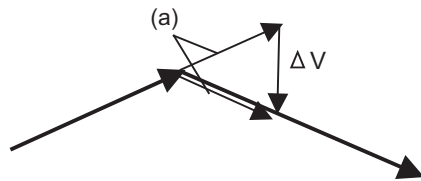
#8021 COMP CHANGE	Accuracy coefficient used
0	#8019 R COMP
1	#8022 CORNER COMP

$$\text{Tolerable acceleration for all axes (mm/s}^2\text{)} = \frac{G1bF(\text{mm/min})}{G1btL(\text{ms})} * 60 * 1000 * \frac{100 - R \text{ COMP}}{100}$$

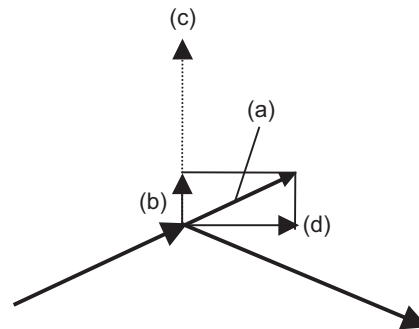
The corner speed V0 can be maintained at more than a certain speed so that the corner speed does not drop too far.

Set "#2096 crncsp (corner deceleration minimum speed)" for each axis, and make a combined speed so that the moving axis does not exceed this setting.

Speed is not clamped



Speed is clamped



- (a) Corner deceleration speed
- (c) Y axis setting value

- (b) Clamp value according to X axis
- (d) X axis setting value

Note that the speed is controlled with the optimum corner deceleration speed in the following cases.

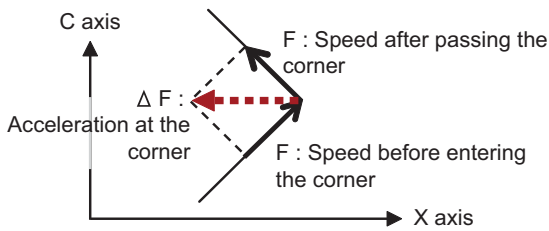
- ♦When the combined corner deceleration speed is equal to or less than the optimum corner deceleration speed
- ♦When the corner deceleration minimum speed parameter setting for the moving axes is set to "0" for even one axis.

(2) Tolerable acceleration control for each axis (optimum acceleration control)

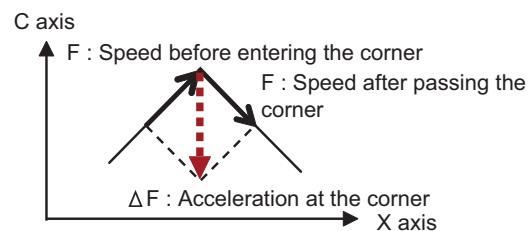
The acceleration to be generated at a seam between blocks is evaluated for each axis to control deceleration so that the seam is passed at the optimum speed. This enables highly accurate edge machining.

The optimum deceleration speed is calculated so that the acceleration of each axis to be generated at the seam is equal to or less than the tolerable acceleration for each axis, which is determined by "#2157 G1bFx" (maximum speed for each axis), "#2158 G1btLx" (time constant for each axis), and the accuracy coefficient. The machine decelerates to the speed in advance, and then accelerates back to the command speed after passing the corner. This control enables deceleration at an appropriate speed for the characteristics of each axis even when machine vibrations may easily occur due to a low tolerable acceleration for a specific axis (rotary axis). This means that the deceleration speed can be raised at a corner where acceleration is generated only for an axis with a high tolerable acceleration, leading to a reduced cycle time.

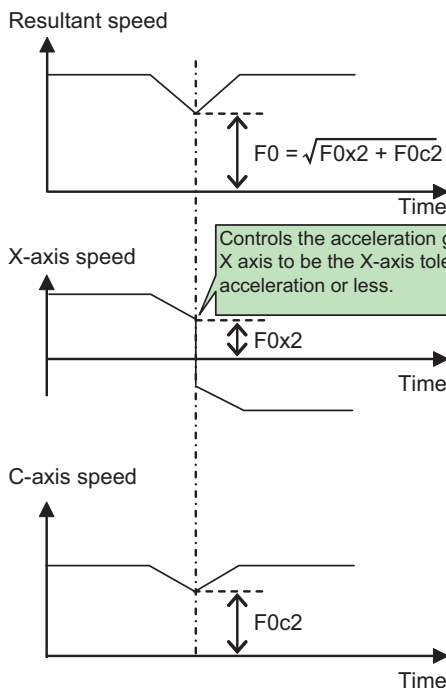
If acceleration is generated for the X axis (linear axis) as shown in Figure (a) below or for the C axis (rotary axis) as shown in Figure (b), the corner speed F is controlled so that the acceleration to be generated at the X or C axis does not exceed the tolerable acceleration for the X or C axis, respectively. If the tolerable acceleration for the X axis is higher than that for the C axis, a higher deceleration speed can be used for a path where acceleration is generated only for the X axis than where acceleration is generated only for the C axis. In this case, the speed patterns are as shown in Figures (c) and (d) below:



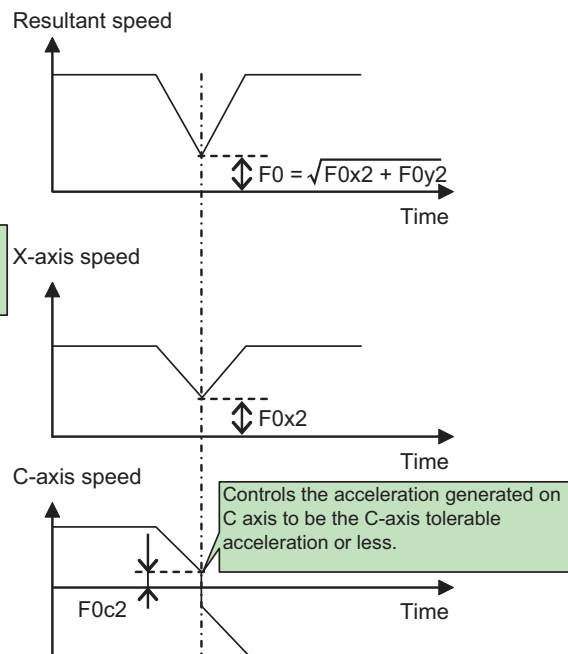
(a) Corner shape which generates the acceleration on X axis (linear axis)



(b) Corner shape which generates the acceleration on C axis (rotary axis)



(c) Speed pattern which generates the acceleration on X axis (linear axis)



(d) Speed pattern which generates the acceleration on C axis (rotary axis)

Deceleration is not carried out when blocks are smoothly connected (when the acceleration to be generated for each axis is equal to or lower than the tolerable acceleration for each axis).

The edge accuracy can be further improved by setting a greater accuracy coefficient. A greater accuracy coefficient, however, reduces the optimum corner speed, which may increase the cycle time. Setting a negative accuracy coefficient can increase the optimum corner speed and reduce the cycle time.

As shown below, different accuracy coefficients can be used depending on the parameter "#8021 COMP_CHANGE". Also, the tolerable acceleration can be adjusted for each axis using "#2159 compx" (accuracy coefficient for each axis), and the tolerable acceleration for each axis can be obtained with the following formula. It is necessary, however, to set the same tolerable acceleration for all base axes because an arc shape is distorted if it differs among them. If G1bFx is 0 (not set), the tolerable acceleration is calculated using "#2001 rapid" (rapid traverse rate). And if G1btLx is 0 (not set), the tolerable acceleration is calculated using "#2004 G0tL" (G0 time constant (linear)).

If G1bFx and G1btLx are 0 for all base axes, the tolerable accelerations for the base axes are unified to the lowest one.

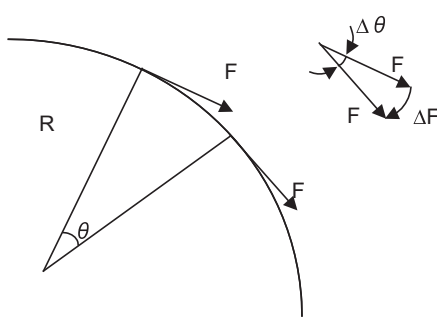
#8021 COMP CHANGE	Accuracy coefficient used
0	#8019 R COMP
1	#8022 CORNER COMP

$$\text{Tolerable acceleration for each axes (mm/s}^2\text{)} = \frac{\text{G1bFx(mm/min)}}{\text{G1btLx(ms)}} * 60 * 1000 * \frac{100 - \text{R COMP}}{100} * \frac{100 - \text{compx}}{100}$$

(3) Arc speed clamp

During circular interpolation, even when moving at a constant speed, acceleration is generated as the advance direction constantly changes. When the arc radius is large enough in relation to the commanded speed, control is carried out at the commanded speed. However, when the arc radius is relatively small, the speed is clamped so that the generated acceleration does not exceed the tolerable acceleration/deceleration speed before interpolation, calculated with the parameters.

This allows arc cutting to be carried out at an optimum speed for the arc radius. The figure below shows the acceleration ΔF (mm/s²) for movement at the constant speed F (mm/min) on an arc shape with the radius R (mm). Here, the arc clamp speed F' (mm/min) that makes the acceleration ΔF lower than the tolerable acceleration common to all axes A_c (mm/s²) can be obtained with the following formula:



F : Commanded speed (mm/min)
 R : Commanded arc radius (mm)
 $\Delta\theta$: Angle change per interpolation unit
 ΔF : Speed change per interpolation unit
 The tool is fed with the arc clamp speed F' so that ΔF does not exceed the tolerable acceleration common to all axes A_c (mm/s²).

$$F' \leq \sqrt{R \cdot A_c \cdot 60}$$

$$\Delta F' = \frac{G1bF(\text{mm/min})}{G1btL(\text{ms})}$$

When the above F' expression is substituted with F in the expression for the maximum logical arc radius reduction error amount ΔR , explained in the section "Pre-interpolation acceleration/deceleration", the commanded radius R is eliminated, and ΔR does not rely on R .

Here, T_p is the servo system position loop time constant (s) and K_f is the feed forward coefficient. T_p is the inverse number to "#2203 PGN1" (position loop gain) ($T_p = 1 / \text{PGN1}$) and K_f is a ratio of "#2010 fws_g" (feed forward gain) ($K_f = \text{fwd}_g / 100$), both of which depend on the MTB specifications.

$$\Delta R = \frac{1}{2R} \left\{ T_p^2 \left[1 - K_f^2 \right] \right\} \left\{ \frac{F}{60} \right\}^2$$

$$= \frac{AC}{2} \left\{ T_p^2 \left[1 - K_f^2 \right] \right\}$$

ΔR : Arc radius reduction error amount
 T_p : Position loop gain time constant of servo system
 K_f : Feed forward coefficient
 F : Cutting feedrate

In other words, with an arc command to be clamped at the arc clamp speed, in logical terms regardless of the commanded radius R , machining can be carried out with a radius reduction error amount within a constant value. The roundness can be further improved by setting a greater accuracy coefficient. A greater accuracy coefficient, however, reduces the arc clamp speed, which may increase the cycle time. Setting a negative accuracy coefficient can increase the arc clamp speed and reduce the cycle time.

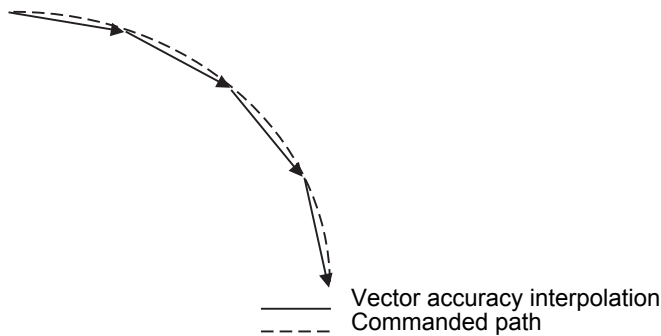
As shown below, different accuracy coefficients can be used depending on the parameter "#8021 COM-P_CHANGE", and the tolerable acceleration common to all axes can be obtained with the following formula:

#8021 COMP CHANGE	Accuracy coefficient used
0	#8019 R COMP
1	#8023 CURVE COMP

$$\text{Tolerable acceleration for all axes (mm/s}^2\text{)} = \frac{G1bF(\text{mm/min})}{G1btL(\text{ms})} * 60 * 1000 * \frac{100 - \text{R COMP}}{100}$$

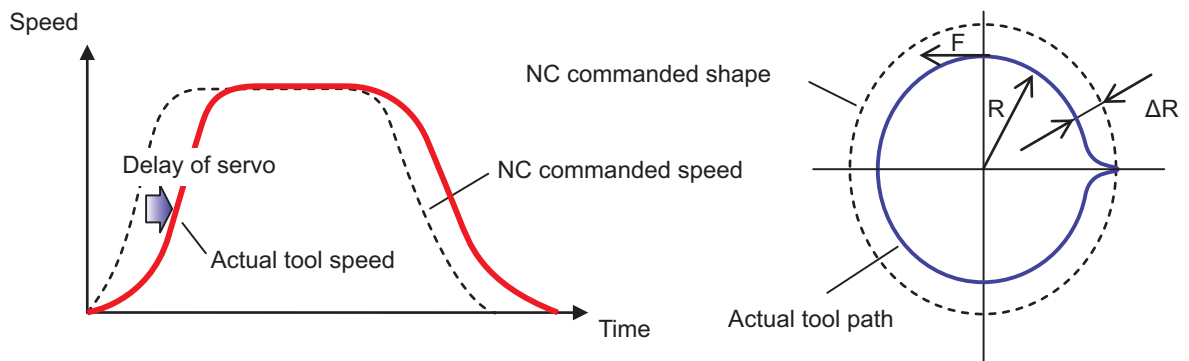
Vector accuracy interpolation

When a fine segment is commanded and the angle between the blocks is extremely small (when not using optimum corner deceleration), interpolation can be carried out more smoothly using the vector accuracy interpolation.

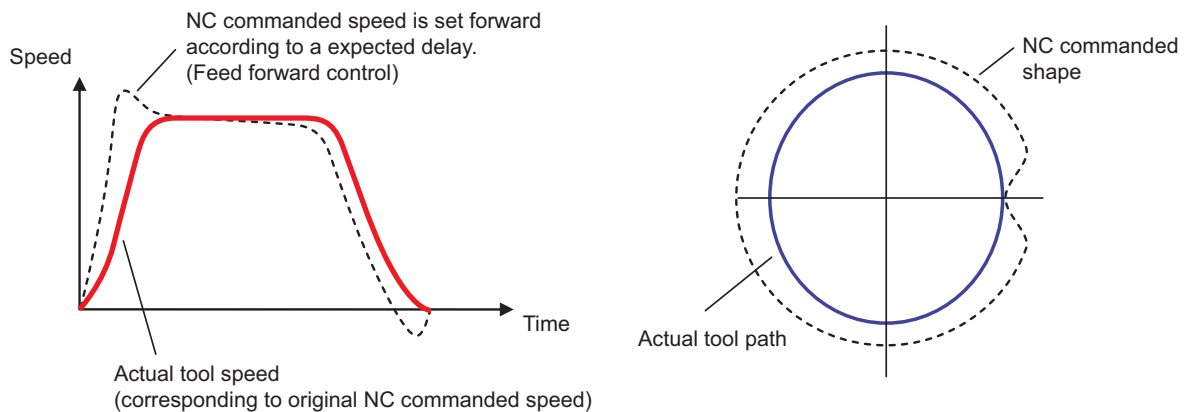


Feed forward control

This function reduces path errors caused by delay of servo systems. Path errors caused by acceleration/deceleration of NC can be eliminated by acceleration/deceleration before interpolation, however errors caused by delay of servo systems cannot be eliminated by acceleration/deceleration before interpolation. Therefore, when the arc shape of radius R (mm) is machined at speed F (mm/min) as the figure (a) below, for instance, the lag time occurs between the NC commanded speed and the actual tool speed in amount of the servo system time constant and the path error ΔR (mm) occurs. Feed forward control generates the command value taking the delay of servo systems as shown in figure (b) below so that the path error caused by delay of servo systems can be inhibited.



(a) NC command and actual tool movement during Feed forward control OFF



(b) NC command and actual tool movement during Feed forward control ON

$$\Delta R = \frac{1}{2R} \left\{ T_p^2 (1 - K_f^2) \right\} \left(\frac{F}{60} \right)^2$$

Here, T_p is the servo system position loop time constant (s) and K_f is the feed forward coefficient. T_p is the inverse number to "#2203 PGN1" (position loop gain) ($T_p = 1 / PGN1$) and K_f is a ratio of "#2010 fws_g" (feed forward gain) ($K_f = fws_g / 100$), both of which depend on the MTB specifications.

Combination with the smooth high gain (SHG) control function

Feed forward control can inhibit path errors more effectively by increasing the feed forward coefficient. In some cases, however, the coefficient cannot be increased because a greater coefficient may cause machine vibrations. In this case, use this function together with the smooth high gain (SHG) control function to stably compensate path errors caused by lag of servo system.

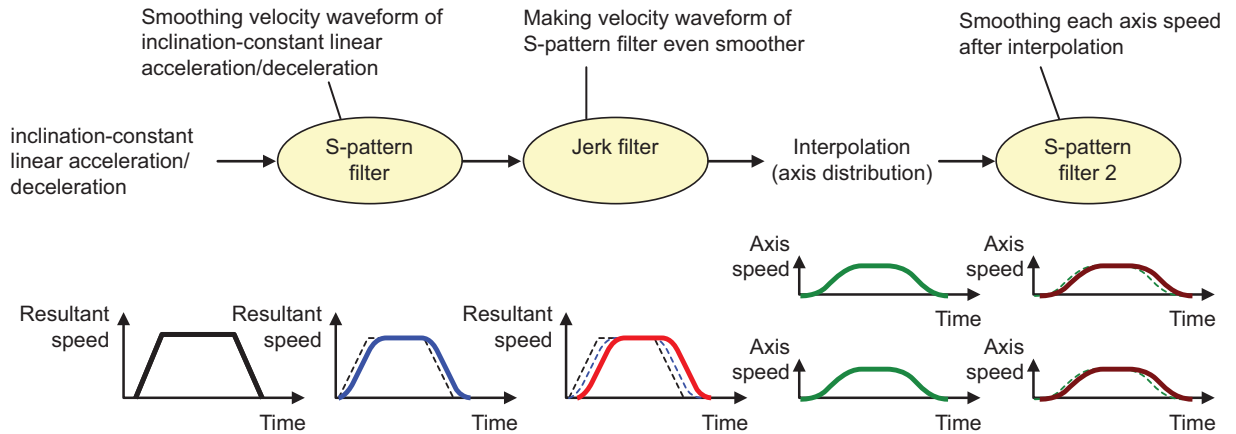
To enable the SHG control, it is also necessary to set "#2204 PGN2" (position loop gain 2) and "#2257 SHGC SHG" (control gain) in addition to "#2203 PGN1" (position loop gain 1), all of which depend on the MTB specifications. By enabling the SHG control, it is possible to inhibit path errors, for example, for an arc shape equivalently as with conventional control (SHG control OFF) using the equivalent feed forward gain fwd_g as shown in the following formula. This means that setting $fwd_g = 50$ (%) for the SHG control is as effective as setting $fwd_g = 100$ (%) for conventional control in inhibiting path errors.

$$fwd_g' = 100 \sqrt{1 - \left\{ 1 - \left(\frac{fwd_g}{100} \right)^2 \right\} \left(\frac{1}{2} \right)}$$

S-pattern filter control

S-pattern filter (soft acceleration/deceleration filter) is the function that inhibits the machine vibration by smoothing a velocity waveform. There are following types of S-pattern filters:

- G01/G00 S-pattern filter
- G01/G00 jerk filter
- S-pattern filter 2



(1) G01/G00 S-pattern filter

This function inhibits the machine vibration by smoothing a velocity waveform generated by inclination-constant linear acceleration/deceleration.

Inclination-constant linear acceleration/deceleration generates continuous velocity waveforms, but makes the acceleration discontinuous. As a result, machine vibrations may easily occur when there are discontinuities in acceleration, which may cause scratches or streaks on the machining surface. The S-pattern filter can make the velocity waveform even smoother and eliminate acceleration discontinuities to inhibit machine vibrations. The S-pattern filter does not impair machining accuracy because it makes the combined speed smoother before interpolation. A greater S-pattern filter time constant, however, may increase the cycle time.

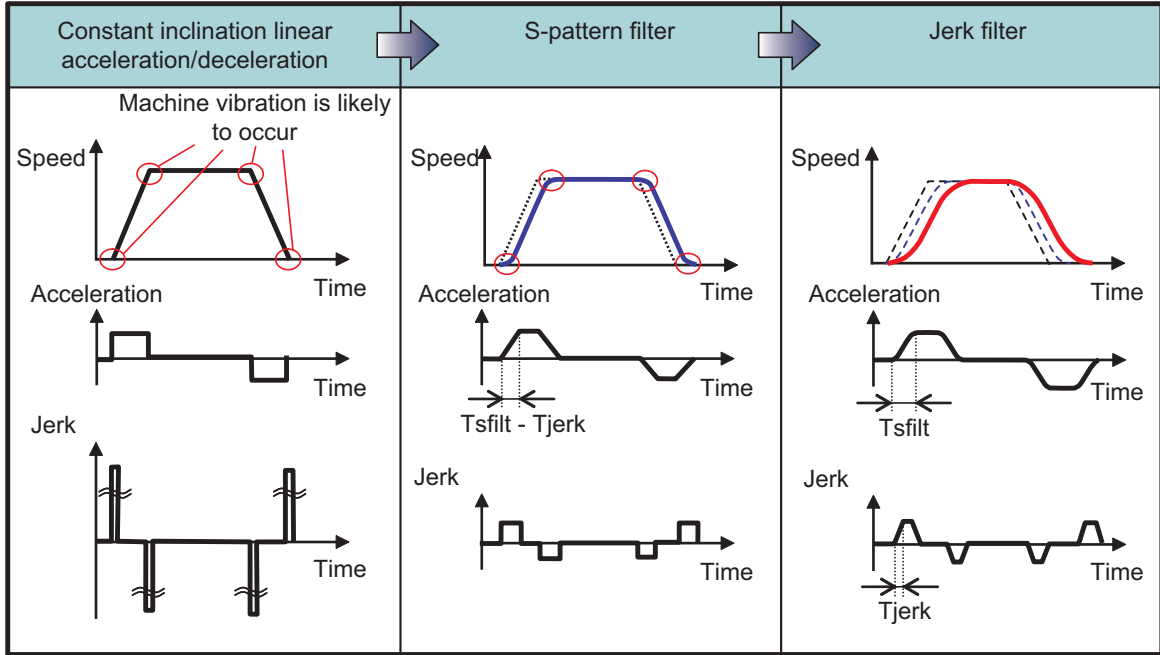
To the S-pattern filter time constant, "#1568 SfiltG1" is applied during cutting feed (G01) or "#1569 SfiltG0" during rapid traverse (G00), each of which can be set in the range of 0 to 200 (ms).

(2) G01/G00 jerk filter

The jerk filter function inhibits machine vibrations by eliminating jerk discontinuities when the S-pattern filter alone cannot inhibit such vibrations.

Through the S-pattern filter, continuous velocity waveforms can be obtained up to acceleration, but jerk discontinuities remain. The jerk filter further filters the velocity waveform smoothed by the S-pattern filter to smooth jerk as well to inhibit machine vibrations. The jerk filter does not impair machining accuracy because it makes the combined speed smoother before interpolation.

To the jerk filter time constant, "#12051 Jerk_filtG1" is applied during cutting feed (G01) or "#12052 Jerk_filtG0" during rapid traverse (G00), each of which can be set in the range of 0 to 50 (ms). Even if a jerk filter time constant is set, the S-pattern filter time constant is the time to achieve the target acceleration. As a result, the time constant for S-pattern filter processing is "S-pattern filter time constant" - "Jerk filter time constant". If the jerk filter time constant is the same as or greater than the S-pattern filter time constant, an MCP alarm (Y51 0030) will occur.



Tsfilt: S-pattern filter time constant
Tjerk: Jerk filter time constant

(3) S-pattern filter 2

This function inhibits machine vibrations by smoothing slight speed fluctuation caused when the combined speed is distributed to each axis element.

S-pattern filter 2 can inhibit machine vibrations by smoothing slight speed fluctuation on each axis. The function, however, may impair machining accuracy because it filters each axis speed after interpolation. A greater S-pattern filter 2 time constant, however, may increase the cycle time.

To the S-pattern filter 2 time constant, "#1570 Sfilt2" is applied, which can be set in the range of 0 to 200 (ms).

(4) How to adjust parameters

(a) The table below shows typical initial values for each filter time constant. If your machine's natural angular frequency f_n (Hz) is known, vibrations can be inhibited effectively by setting the vibration period T_n (ms) obtained with the following formula for the S-pattern filter time constant:

$$T_n = \frac{1000}{f_n} \text{ (ms)}$$

S-pattern filter (SfiltG1/SfiltG0)	Jerk filter (Jerk_filtG1/Jerk_filtG0)	S-pattern filter (Sfilt2)
50ms	0ms	10ms

(b) If vibrations cannot be inhibited properly with the above initial values, increase the S-pattern filter time constant. Or, decrease the S-pattern filter time constant to reduce the cycle time.

(c) If vibrations occur at a corner or other section and stripes remain on the machining surface even after the S-pattern filter time constant is increased, increase the S-pattern filter 2 time constant. The maximum S-pattern filter 2 time constant, however, should be 20 to 25 ms because a greater S-pattern filter 2 time constant may impair machining accuracy.

(d) If high-frequency machine vibrations remain even after the S-pattern filter/S-pattern filter 2 are applied, set the jerk filter time constant.

If a shorter cycle time has a priority over the machining accuracy, it is possible to inhibit vibrations at a corner by reducing the corner accuracy coefficient to increase the corner deceleration speed and increasing the S-pattern filter 2 time constant.



Relationship with other functions

- (1) The modal must be set as shown below when commanding G08 P1/G61.1.

Function	G code
Cylindrical interpolation cancel (*1)	G07.1
Polar coordinate interpolation cancel (*1)	G15
Tool radius compensation mode cancel	G40
Tool length compensation cancel	G49
Normal line control cancel	G40.1
Programmable mirror image OFF	G50.1
Mirror image with settings	Cancel
Mirror image with signals	Cancel
No macro modal call	G67
Feed per revolution cancel	G94
Constant surface speed control mode cancel	G97
Interruption type macro mode cancel	M97

(*1) These functions can be commanded if the tolerable acceleration control for each axis (optimum acceleration control) or variable-acceleration pre-interpolation acceleration/deceleration specifications are valid.

- (2) A program error will occur if high-accuracy control is commanded in the following modes.
- ♦During milling -> Program error (P481)
 - ♦During cylindrical interpolation -> Program error (P481) (*2)
 - ♦During polar coordinate interpolation -> Program error (P481) (*2)
 - ♦During normal line control -> Program error (P29)
- (3) A program error (P29) will occur if the following commands are issued during the high-accuracy control mode.
- ♦Milling
 - ♦Cylindrical interpolation (*2)
 - ♦Polar coordinate interpolation (*2)
 - ♦Normal line control

(*2) An error will not occur if the tolerable acceleration control for each axis (optimum acceleration control) or variable-acceleration pre-interpolation acceleration/deceleration specifications are valid.

Operation when high-accuracy control-related G commands are combined

The table below shows operations when following high-accuracy control-related commands are combined:

G61.1, G8P1	: High-accuracy control
G64	: Cutting mode
G61	: Exact stop check mode
G62	: Automatic corner override
G63	: Tapping mode
G61.2	: High-accuracy spline interpolation
G08P0	: High-accuracy control cancel (cutting mode)
G05.1Q1	: High-speed high-accuracy control I
G05.1Q2	: Spline interpolation
G05P2	: High-speed machining mode II
G05P10000	: High-speed high-accuracy control II
G05P20000	: High-speed high-accuracy control III

A	B	Operation when B is commanded during A command
G61.1/G08P1	G61.1	Continues high-accuracy control.
	G61,G62,G63,G64	Cancels high-accuracy control and operates in the commanded mode.
	G61.2	Operates in the high-accuracy spline interpolation mode.
	G8P1	Continues high-accuracy control.
	G8P0	Cancels high-accuracy control. (Changes G code group 13 to G64.)
	G05.1Q1	Operates in the high-speed high-accuracy control I mode.
	G05.1Q2	A program error (P34) will occur.
	G05P2	Operates in high-accuracy control + high-speed machining mode II.
	G05P10000	Operates in the high-speed high-accuracy control II mode.
	G06.2	A program error (P34) will occur.
G61.2	G61.1	Operates in the high-accuracy control mode.
	G61,G62,G63,G64	Operates in the commanded mode.
	G61.2	Continues high-accuracy spline interpolation.
	G08P1	Operates in the high-accuracy control mode.
	G08P0	A program error (P29) will occur.
	G05.1Q1	A program error (P29) will occur.
	G05.1Q2	A program error (P34) will occur.
	G05P2	Operates in high-accuracy spline interpolation + high-speed machining mode II.
	G05P10000	A program error (P29) will occur.
	G06.2	A program error (P34) will occur.
G05.1Q1	G61.1	Continues the high-speed high-accuracy control I mode.
	G64	Continues the high-speed high-accuracy control I mode.
	G61,G62,G63	Operates in the high-speed high-accuracy control I + commanded mode.
	G61.2	A program error (P29) will occur.
	G08P1	Continues the high-speed high-accuracy control I mode.
	G08P0	Continues the high-speed high-accuracy control I mode.
	G05.1Q1	Continues the high-speed high-accuracy control I mode.
	G05.1Q2	A program error (P34) will occur.
	G05P2	Operates in the high-speed machining mode II.
	G05P10000	A program error (P34) will occur.
G06.2	A program error (P34) will occur.	

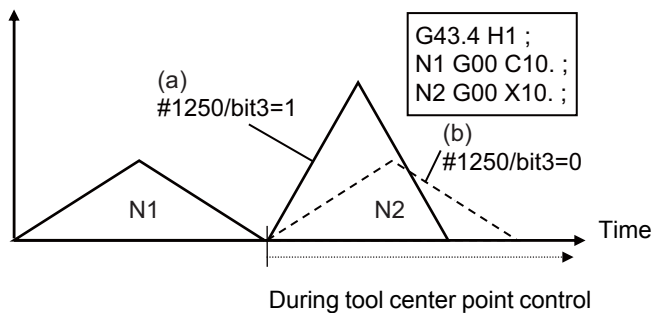
A	B	Operation when B is commanded during A command
G05P10000	G61.1	Continues the high-speed high-accuracy control II mode.
	G64	Continues the high-speed high-accuracy control II mode.
	G61,G62,G63	Operates in the high-speed high-accuracy control II + commanded mode.
	G61.2	A program error (P29) will occur.
	G08P1	Continues the high-speed high-accuracy control II mode.
	G08P0	Continues the high-speed high-accuracy control II mode.
	G05.1Q1	A program error (P34) will occur.
	G05.1Q2	Operates in the high-speed high-accuracy control II mode + spline interpolation
	G05P2	Operates in the high-speed machining mode II.
	G05P10000	Continues the high-speed high-accuracy control II mode.
	G06.2	Operates in the NURBS interpolation mode.

Rapid traverse acceleration switching during tool center point control, workpiece installation error compensation, or inclined surface machining command

The acceleration of either the cutting feed (G01) or the rapid traverse (G00) can be used for the rapid traverse (G00) in the high-accuracy control mode during the tool center point control, workpiece installation error compensation, or inclined surface machining command. Which acceleration is used depends on the MTB specifications (parameter "#1250 set22/bit3").

Normally, the acceleration of the cutting feed (G01) is used and the acceleration in the route direction becomes constant. If the acceleration of the rapid traverse (G00) is used, the acceleration can be changed to suit the responsiveness of the moving axis, and also the cycle time can be reduced compared to that of the cutting feed (G01) acceleration. The figure below shows the relationship between parameters and speed waveforms (when the responsiveness of the linear axis is higher than that of the rotary axis).

Resultant speed



(a) Acceleration of rapid traverse (G00)

(b) Acceleration of cutting feed (G01)

When at least one of the functions shown in the table below is commanded, positioning is performed with the acceleration of the rapid traverse (G00) in the following conditions:

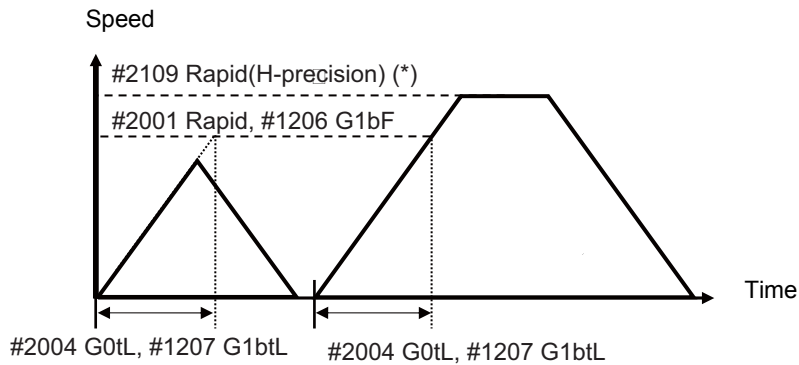
- (1) The parameter "#1250 set22/bit3" is set to "1".
- (2) SSS control is being executed.

If the above conditions are not satisfied, the system runs with the acceleration of the cutting feed (G01).

Function	Instruction (G code)
Tool center point control	G43.4, G43.5
Workpiece installation error compensation	G54.4
Inclined surface machining command	G68.2, G68.3

The table below shows the rapid traverse acceleration/deceleration patterns and acceleration designation methods under the SSS control. The acceleration is calculated based on the designated parameters to perform operations. Designate the parameters to determine the rapid traverse acceleration, referring to the table below.

#1250 bit3	Functions in the table above	Rapid traverse constant inclination multi-step acceleration/deceleration	Acceleration/deceleration pattern	Acceleration designation method
0	Not commanded.	Invalid	Constant inclination acceleration/deceleration	Designated with the parameters "#2001 Rapid" and "#2004 G0tL".
		Valid	Constant inclination multi-step acceleration/deceleration	Follows the rapid traverse constant inclination multi-step acceleration/deceleration specifications.
	Commanded.	Invalid/Valid	Constant inclination acceleration/deceleration	Designated with the parameters "#1206 G1bF" and "#1207 G1btL".
1	Not commanded.	Invalid	Constant inclination acceleration/deceleration	Designated with the parameters "#2001 Rapid" and "#2004 G0tL".
		Valid	Constant inclination multi-step acceleration/deceleration	Follows the rapid traverse constant inclination multi-step acceleration/deceleration specifications.
	Commanded.	Invalid/Valid	Constant inclination acceleration/deceleration	Designated with the parameters "#2001 Rapid" and "#2004 G0tL".

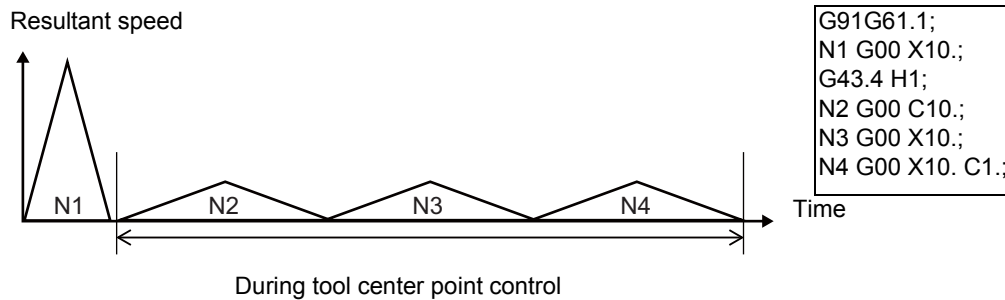


(*) When "#2109 Rapid(H-precision)" (rapid traverse rate for high-accuracy control mode) is set to "0", "#2001 rapid" is used.

The following describes the relationship between the setting values of the parameter "#1250 set22/bit3" and the applied acceleration using two operation examples (resultant speed waveform). The major parameters are as follows.

Parameters	Description	Setting value	
#1205 G0bdcc	G0 acceleration/deceleration before interpolation	1	
#1206 G1bF	Maximum speed (mm/min)	10000	
#1207 G1btL	Time constant (ms)	500	
#8090 SSS ON	SSS control ON/OFF	1	
#2001 Rapid	Rapid traverse rate	(X axis) 30000	(C axis) 20000
#2004 G0tL	G0 time constant (linear)	(X axis) 100	(C axis) 500

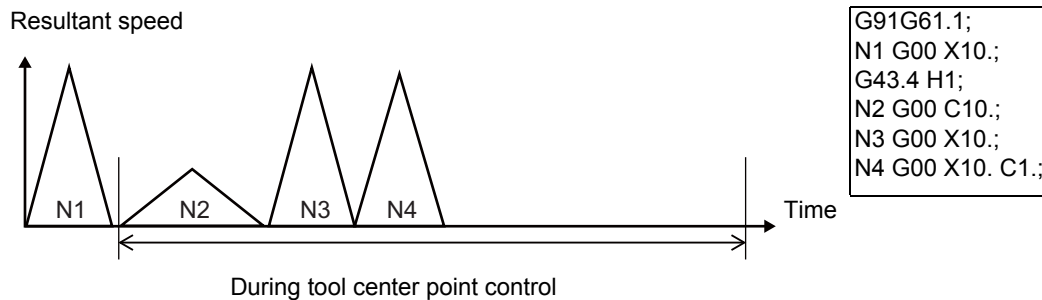
(1) Acceleration/deceleration pattern when "#1250 bit3" is set to "0"



N1: Runs with the acceleration calculated from "#2001" and "#2004" of the X axis.

N2-N4: Runs with the acceleration calculated from "#1206" and "#1207".

(2) Acceleration/deceleration pattern when "#1250 bit3" is set to "1"



N1: Runs with the acceleration calculated from "#2001" and "#2004" of the X axis.

N2-N4: Runs with the acceleration calculated from "#2001" and "#2004" of each axis.

(Calculate the optimum acceleration within the range in which the acceleration of each axis does not exceed the setting.)



Precautions

- (1) The "high-accuracy control" specifications are required to use this function. If G61.1 is commanded when there are no specifications, a program error (P123) will occur.
- (2) The high-accuracy control function is internally enabled by the high-speed high-accuracy I/II/III (G5.1Q1/G5P10000) command. If the high-speed high-accuracy I/II/III is commanded in the high-accuracy control mode, the high-speed high-accuracy I/II/III mode is enabled. Then, if the high-speed high-accuracy I/II/III mode is canceled, the high-accuracy control mode is restored.
- (3) In the high-accuracy control mode, feedrate command F is clamped with the "#2110 Clamp (H-precision)" (Cutting feed clamp speed for high-accuracy control mode) set with parameter. When the cutting feed clamp speed for the high-accuracy control mode is 0, however, it is clamped with the "#2002 clamp" cutting clamp speed set by the parameter.
- (4) In the high-accuracy control mode, rapid traverse rate conforms to "#2109 Rapid(H-precision)" (Rapid traverse rate during high-accuracy control mode) set by the parameter. When the rapid traverse rate during the high-accuracy control mode is set to "0", however, the movement follows "#2001 rapid" set by the parameter.
- (5) If the specifications for the multi-part system simultaneous high-accuracy control are not provided, the "#1205 G0bdcc" (G0 pre-interpolation) can be used with only one part system. If the 2nd or later part system is set to the G0 pre-interpolation acceleration/deceleration, an MCP alarm (Y51 0017) will occur.
- (6) "#1568 SfiltG1", "#1569 SfiltG0" and "#1570 Sfilt2" cannot be changed from the screen during program mode. If these parameters is changed by "parameter input by program", these parameters become valid from the next block.
- (7) If Reset or Emergency signal is input during axis travel, it takes a time equal to the time constant to recover from the reset or emergency stop state.
- (8) When there are high-accuracy acceleration/deceleration time constant expansion specifications, the sampling buffer area may be smaller.
- (9) The high-accuracy control time constant expansion specifications can only be used for a 1-part system. In a multi-part system, the high-accuracy acceleration/deceleration time constant expansion specifications are disabled even when they are set to ON.
- (10) For a part system where high-accuracy control is to be commanded, set the number of axes in the part system to 8 or less. If high-accuracy control is commanded for a part system that has 9 or more axes, an operation error (M01 0135) will occur. The error will not occur, however, if the number of axes in the part system excluding the master axis/slave axis is 8 or less during the synchronous control/control axis synchronization between part systems.
- (11) Even if the parameter "#1210 RstGmd" (modal G code reset setting) is set to "not to initialize group 13 at reset", group 13 is initialized according to the setting of "#1148 I_G611" (Initial hi-precis) if it is enabled. To retain group 13 at reset, set "#1148 I_G611" to "0".
These parameters depend on the MTB specifications.
- (12) If the parameter "#1205 G0bdcc" (G0 acceleration/deceleration before interpolation) is set to "1", the value set with the parameter "#2224 SV024" (in-position detection width) will be used as the in-position width. The setting of the parameter "#2077 G0inps" (G0 in-position width) and the programmable in-position check with "I" address are disabled.

17.2.2 SSS Control

**Function and purpose**

This function runs a machining program that approximates a freely curved surface with fine segment lines at high speed and with high-level accuracy. This function enables machining with less scratches and streaks on the cutting surface compared to the conventional high-accuracy control function.

With conventional high-accuracy control, the angle between two blocks is compared with the corner deceleration angle to determine whether to execute corner deceleration between the blocks. This can cause the speed to suddenly change between the blocks with an angle close to the corner deceleration angle, resulting in scratches or streaks.

The SSS (Super Smooth Surface) control uses information on not only the angle between two blocks but also global paths to provide optimum speed control that is not significantly affected by minute stepping or waviness. The favorable effects of this control include a reduction in the number of scratches or streaks on cutting surfaces.

The SSS control has the following features:

- (1) This function is effective at machining smooth-shaped dies using a fine segment program.
- (2) This function provides speed control that is not susceptible to errors in paths.
- (3) Even if corner deceleration is not required, the speed is clamped if the predicted acceleration is high.
(The clamp speed can be adjusted using the parameter "#8092 ClampCoeff".)

The length of the path direction recognized with SSS control can be adjusted with the machining parameter "#8091 reference length". The range is increased as the setting value increases, and the effect of the error is reduced.

If the multi-part system simultaneous high-accuracy specification is provided, up to two part systems can be used at the same time.

Note

- (1) The use of this function requires the following functions, in addition to the SSS control specifications. Make sure that these specifications are enabled before using this function.
 - High-accuracy control (G61.1/G08P1)
 - High-speed high-accuracy control I (G05.1 Q1)
 - High-speed high-accuracy control II (G05 P10000)
 - High-speed high-accuracy control III (G05 P20000)



Detailed description

When the parameters are set as below, each of the following high-accuracy control commands is activated under SSS control.

<Parameter>

"#8090 SSS ON" ON

<Command format of the modes activated under SSS control>

[High-accuracy control]	
G61.1 ; or G08P1;	High-accuracy control ON
G08P0; or, G command in group 13 except G61.1	High-accuracy control OFF
[High-speed high-accuracy control I]	
G05.1 Q1 ;	High-speed high-accuracy control I ON
G05.1 Q0 ;	High-speed high-accuracy control I OFF
[High-speed high-accuracy control II]	
G05 P10000 ;	High-speed high-accuracy control II ON
G05 P0 ;	High-speed high-accuracy control II OFF
[High-speed high-accuracy control III]	
G05 P20000 ;	High-speed high-accuracy control III ON
G05 P0 ;	High-speed high-accuracy control III OFF

"SSS" is displayed on the modal display screen under SSS control.

However "SSS" is not displayed when a command being executed is out of the scope of SSS control.

Adjustment of accuracy coefficient

The clamp speed at a corner and arc can be adjusted using "#8022 CORNER COMP" and "#8023 CURVE COMP" (If "#8021 COMP_CHANGE" is set to "0", use "#8019 R COMP" to adjust the clamp speed at a corner and arc). When "#8096 Deceler. coeff. ON" is set to "1", "#8097 Corner decel coeff" and "#8098 Arc clamp spd coef" become valid during SSS control. Using these parameters, you can use different corner deceleration speeds and clamp speeds at arcs according to whether or not the SSS control is enabled.

For parameters #8097 and #8098, respectively, set a percentage ratio to the level of the relevant speed that is applied when the SSS control is disabled.

Parameter	Item to be adjusted
#8097 Corner decel coeff	Corner deceleration speed to be applied when the SSS control is enabled
#8098 Arc clamp spd coef	Arc clamp speed to be applied when the SSS control is enabled

(Example) When "#8097 Corner decel coeff" is set to 200 (%), the corner deceleration speed that is applied when the SSS control is enabled becomes twice the corner deceleration speed that is applied when the SSS control is disabled.

When setting the parameters, adjust the values within the range in which the machine does not vibrate.

Parameter standard values

The standard values of the parameters related to SSS control are shown below.

(1) User parameters

#	Item	Standard value
8090	SSS ON	1
8091	StdLength	1.000
8092	ClampCoeff	1
8093	StepLeng	0.005
8094	DccWaitAdd	0
8096	Deceler. coeff. ON	1
8097	Corner decel coeff	300
8098	Arc clamp spd coef	100
8019	R COMP	0
8020	DCC ANGLE	10
8021	COMP CHANGE	1
8022	CORNER COMP	0
8023	CURVE COMP	-20
8034	AccClampt ON	0
8036	CordecJudge	0
8037	CorJudgeL	0

<Note>

- ♦Reference items for adjusting the parameter

The relationship between each parameter, accuracy and speed is shown below.

The accuracy and speed required for machining can be adjusted with these settings.

When setting the parameters, adjust the values within the range in which the machine does not vibrate.

Parameter	Adjustment target	Effect
#8022 CORNER COMP	Accuracy at corner section	Large setting = Accuracy increases, speed drops
#8023 CURVE COMP	Accuracy at curve section	Large setting = Accuracy increases, speed drops
#8092 ClampCoeff	Accuracy at curve section	Large setting = Accuracy drops, speed increases <Note> ♦Usually use the standard value and adjust with "#8023".

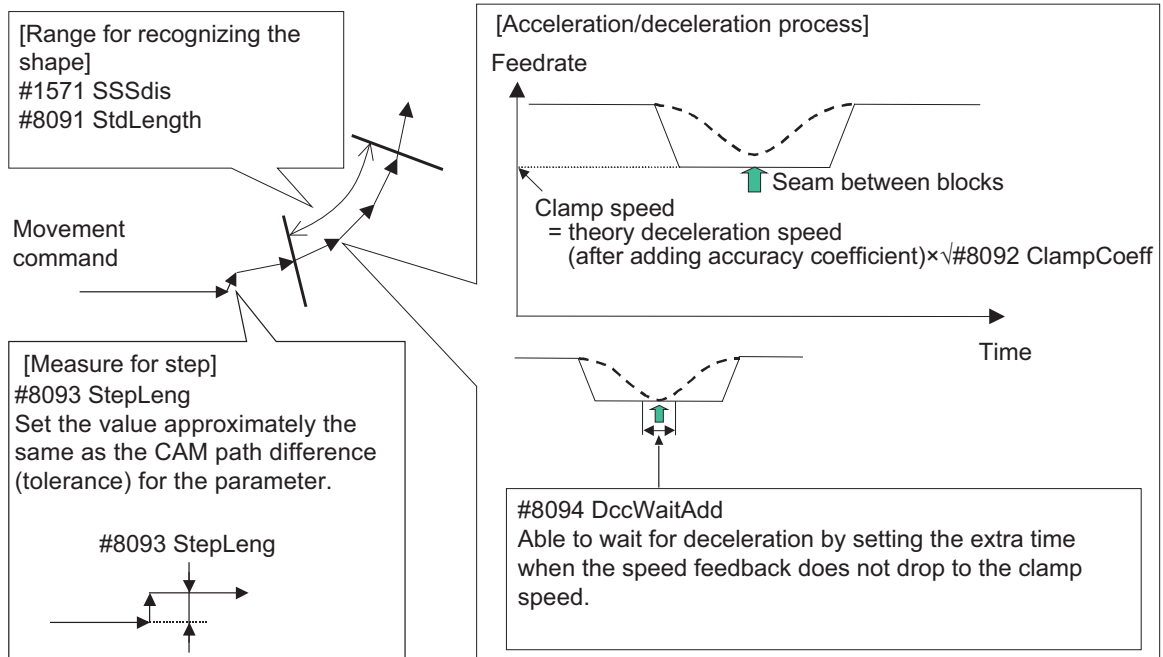
(2) Basic specification parameters (depend on the MTB specifications)

#	Item	Standard value
1148	I_G611 Initial high-accuracy	0
1206	G1bf Acceleration/deceleration before interpolation Maximum speed	-
1207	G1btL Acceleration/deceleration before interpolation Time constant	-
1571	SSSdis SSS control adjustment coefficient fixed value selection	0
1572	Cirorp Arc command overlap	0
1568	SfiltG1 G1 soft acceleration/deceleration filter	0
1569	SfiltG0 G0 soft acceleration/deceleration filter	0
1570	Sfilt2 Soft acceleration/deceleration filter 2	0

(3) Axis specification parameters (depend on the MTB specifications)

#	Item	Standard value
2010	fwd_g Feed forward gain	70
2068	G0fwdg G00 feed forward gain	70
2096	crncsp Minimum corner deceleration speed	0

SSS control parameter



Precautions

- (1) Pre-reading is executed during SSS control, so a program error could occur before the block containing the error is executed.
- (2) Buffer correction is not guaranteed during SSS control.
- (3) If automatic/manual simultaneous or automatic handle feed interrupt are used during SSS control, the machining accuracy will not be guaranteed.
- (4) If a fine arc command is issued during SSS control, it may take longer to machine.
- (5) The same path as single block operation will be used during graphic check.
- (6) The line under the cutting feedrate and arc command block are subjected to the speed control in the SSS control. The command blocks that are not subjected to speed control, decelerate first and automatically switch the SSS control ON and OFF.
- (7) SSS control is temporarily disabled in the following modal:
 - NURBS interpolation
 - Polar coordinate interpolation
 - Cylindrical interpolation
 - User macro interruption enable (M96)
 - Feed per revolution (synchronous feed)
 - Inverse time feed
 - Constant surface speed control
 - Fixed cycle
 - 3-dimensional coordinate conversion
 - Hypothetical axis interpolation
 - Automatic tool length measurement
 - Tool length compensation along the tool axis
- (8) There are some restrictions for each high-accuracy control. Refer to each section for restrictions.
 - "17.2 High-accuracy Control"
 - "17.3 High-speed High-accuracy Control"
- (9) Fairing is disabled during the SSS control.

17.2.3 Tolerance Control

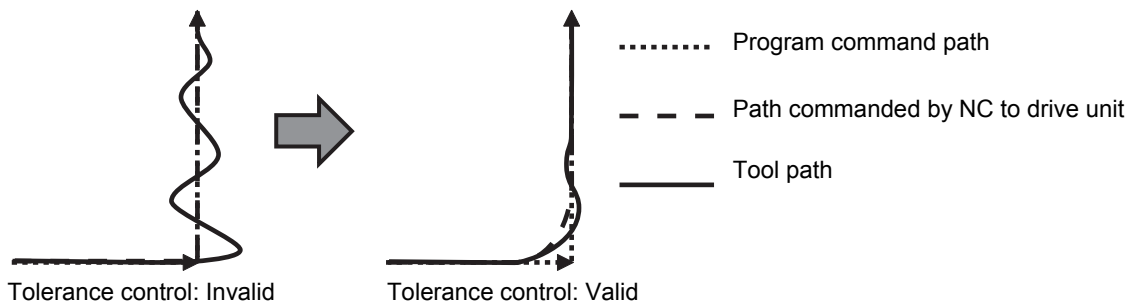
**Function and purpose**

This function obtains the optimum clamp speed for corners or curves based on the designated tolerance to perform operations. It also ensures smooth passing within the tolerance range in corner sections, which suppresses machine vibrations. This means that the clamp speed can be increased to reduce the cycle time.

This function allows the machine to operate with the optimum tool path and speed, simply by specifying the tolerance, so an operator can easily carry out high quality machining.

The tolerance refers to the allowable error amount between the path commanded in the machining program and the path output by NC.

The validity of this function depends on the MTB specifications. This function also requires the SSS control specifications because it can only be used under SSS control.



This function is enabled when the following conditions are satisfied:

- (1) The tolerance control specification is valid. (Based on the MTB specifications.)
 - (2) The parameter "#8090 SSS ON" is set to "1".
 - (3) The parameter "#12066 Tolerance ctrl ON" is set to "1". (*1)(*2)
 - (4) High-accuracy control (G61.1/G08P1), spline interpolation (G61.2/G05.1Q2), spline interpolation 2 (G61.4), or high-speed high-accuracy control I/II/III (G05.1Q1/G05P10000/G05P20000) is valid.
- (*1) Even if conditions (1) and (3) are satisfied, an operation error (M01 0139) will occur and the cycle start cannot be performed automatically if the parameter "#8090 SSS ON" is set to "0". In this case, enable SSS control and reset the alarm to start the cycle automatically.
- (*2) A setting error will occur if "1" is set when this specification is invalid.



Command format

Set the tolerance with the parameter "#2659 tolerance" or the ",K" address following the G code (G61.1 or G61.4 command). When the setting value is "0", this function runs with "0.01(mm)".

Tolerance specification

G61.1 or G61.4 ,K__ ;

,K	Tolerance (mm)
----	----------------

- ♦The range of the command value is 0.000 to 100.000. If a value exceeding the range is commanded, a program error (P35) will occur.
- ♦The tolerance designated by ",K" is applied to all axes in the part system.
- ♦When "0" is designated or ",K" is omitted, the program runs based on the value of the parameter "#2659 tolerance".
- ♦The tolerance designated by ",K" is not held after reset. Therefore, if ",K" is not designated in the G61.1 or G61.4 command after reset, the axis runs based on the value of the parameter "#2659 tolerance".

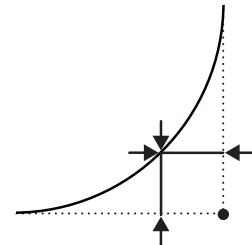
Note

(1) The G61.4 command requires the specifications of spline interpolation 2.



Detailed description

The axis moves in the designated tolerance range during tolerance control. The tolerance on the corner shape is as shown on the right.



Speed control

The clamp speed is obtained from the tolerance in the corner or curve section during tolerance control. As the designated tolerance is lower, the axis speed decelerates.

	Tolerance: High	Tolerance: Low
Command path		
Resultant speed		

Parameters valid during tolerance control

The parameters valid and invalid during tolerance control are as follows. Some parameters depend on the MTB specifications.

(1) Valid parameters

No.	Parameter name	Supplements
1206	G1bF	When combining with the variable-acceleration pre-interpolation acceleration/deceleration or tolerable acceleration control for each axis, specify parameters "#2157 G1bFx" and "#2158 G1btLx".
1207	G1btL	
1568	SfiltG1	
12051	Jerk_filtG1	
2659	tolerance	

(2) Invalid parameters (Parameters with no setting required)

No.	Parameter name	Supplements
1570	Sfilt2	Ignored even if the value is entered.
2159	compX	Ignored even if the value is entered. The clamp speed is obtained from the tolerance during tolerance control; therefore, parameters for adjusting the clamp speed are not required.
8019	R COMP	
8020	DCC ANGLE	
8021	COMP CHANGE	
8022	CORNER COMP	
8023	CURVE COMP	
8096	Deceler. coeff. ON	
8097	Corner decel coeff	
8098	Arc clamp spd coef	

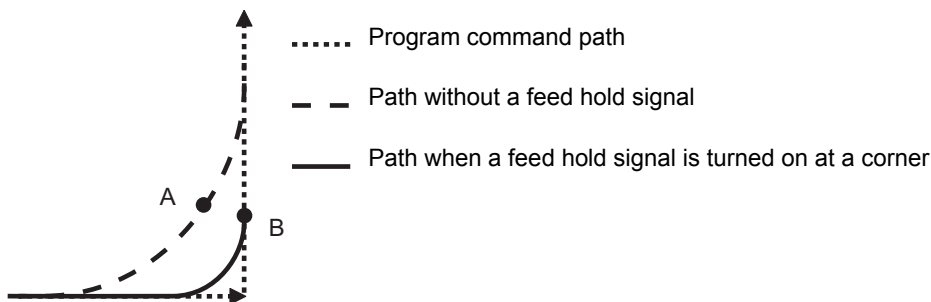
**Program example**

:	
G91 ;	
G61.1 ,K0.02;	Designate tolerance 0.02 (mm).
G01 X0.1 Z0.1 F1000 ; X0.1 Z-0.2 ; Y0.1 ;	Tolerance: 0.02 (mm)
G61.1 ,K0;	Designate tolerance 0 (mm).
X-0.1 Z-0.05 ; X-0.1 Z-0.3 ;	Tolerance: Follows parameter "#2659 tolerance".
G64 ;	
:	



Precautions

- (1) While tolerance control is valid, tolerance control may be canceled temporarily depending on some commands. If tolerance control is canceled temporarily, the axis moves to the commanded position without taking an inner route in a corner section. After this, when a temporary cancel cause is removed, tolerance control restarts. The temporary cancel conditions are as follows:
 - (a) Modal in which the group 1 command is not G01 (linear interpolation) or G02/G03 (circular interpolation).
 - (b) Under single block operation
 - (c) Modal in which SSS control is disabled temporarily (Modal shown below)
 - NURBS interpolation
 - Polar coordinate interpolation
 - Cylindrical interpolation
 - User macro interruption enable (M96)
 - Feed per revolution (Synchronous feed)
 - Inverse time feed
 - Constant surface speed control
 - Fixed cycle
 - 3-dimensional coordinate conversion
 - Hypothetical axis interpolation
 - Automatic tool length measurement
 - Tool length compensation along the tool axis
 - Normal line control
 - Unidirectional positioning
 - Exponential interpolation
 - 3-dimensional circular interpolation
- (2) The stored stroke limit's prohibited range is determined based on the program command path. As a result, machining may not be stopped even if the command moved inward by tolerance control enters the prohibited range.
- (3) If a feed hold signal is turned ON at a corner, machining stops on the program command path. This means that it does not stop at point A in the figure below but at point B.



17.2.4 Variable-acceleration Pre-interpolation Acceleration/Deceleration

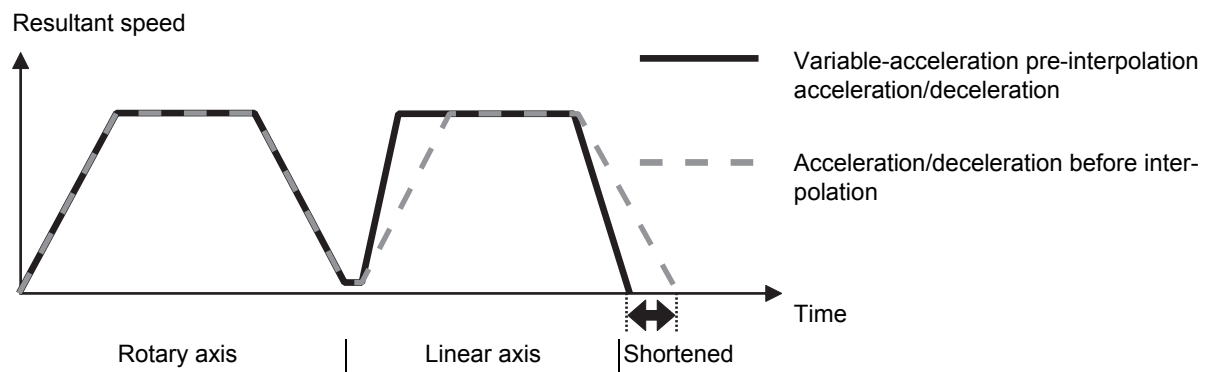


Function and purpose

This function is useful when each axis differs in the characteristics (responsiveness) (4-axis/5-axis machine, etc.). The normal acceleration/deceleration before interpolation performs the acceleration/deceleration by setting acceleration common to all axes. Therefore, if the high responsiveness and low responsiveness coexist in axes, the acceleration needs to be set to suit the axis with low responsiveness.

On the other hand, the variable-acceleration pre-interpolation acceleration/deceleration can perform the acceleration/deceleration by setting diverse acceleration to each axis. This means that it is possible to set a higher acceleration for axes with high responsiveness than before. Therefore, the cycle time can be reduced especially in the indexing machining. (Refer to following figure.)

The validity of this function depends on the MTB specifications. This function also requires the SSS control specifications because it can only be used under SSS control.



This function is enabled when the following conditions are satisfied:

- (1) The variable-acceleration pre-interpolation acceleration/deceleration specification is valid. (Based on the MTB specifications.)
- (2) The MTB-specific parameter has been set (#12060 VblAccPreInt). (*1)
- (3) Under SSS control (*2)

(*1) A setting error will occur if "1" is set when this specification is invalid.

(*2) The validity of the SSS control function depends on the MTB specifications.

To enable SSS control, it is necessary to set the parameter "#8090 SSS ON" to "1" to command high-accuracy control.

(*3) Even if conditions (1) and (2) are satisfied, an operation error (M01 0136) will occur and the cycle start cannot be performed automatically if the parameter "#8090 SSS ON" is set to "0". In this case, enable SSS control and reset the alarm to start the cycle automatically.

"VAC" is displayed on the operation screen and modal display under variable-acceleration pre-interpolation acceleration/deceleration.



Detailed description

The acceleration for each axis is determined in the MTB specifications (parameters "#2157 G1bFx" (maximum speed for each axis) and "#2158 G1btLx" (axis time constant)).

For an axis with G1bFx = 0 (not set), the acceleration is calculated using "#1206 G1bF" (maximum speed).

And for an axis with G1btLx = 0 (not set), the acceleration is calculated using "#1207 G1btL" (time constant).

Therefore, if G1bFx and G1btLx are 0 (not set) for all axes, the normal acceleration/deceleration before interpolation is performed.

The following shows examples of settings.

Set linear axis acceleration for "#1206 G1bF" and "#1207 G1btL".

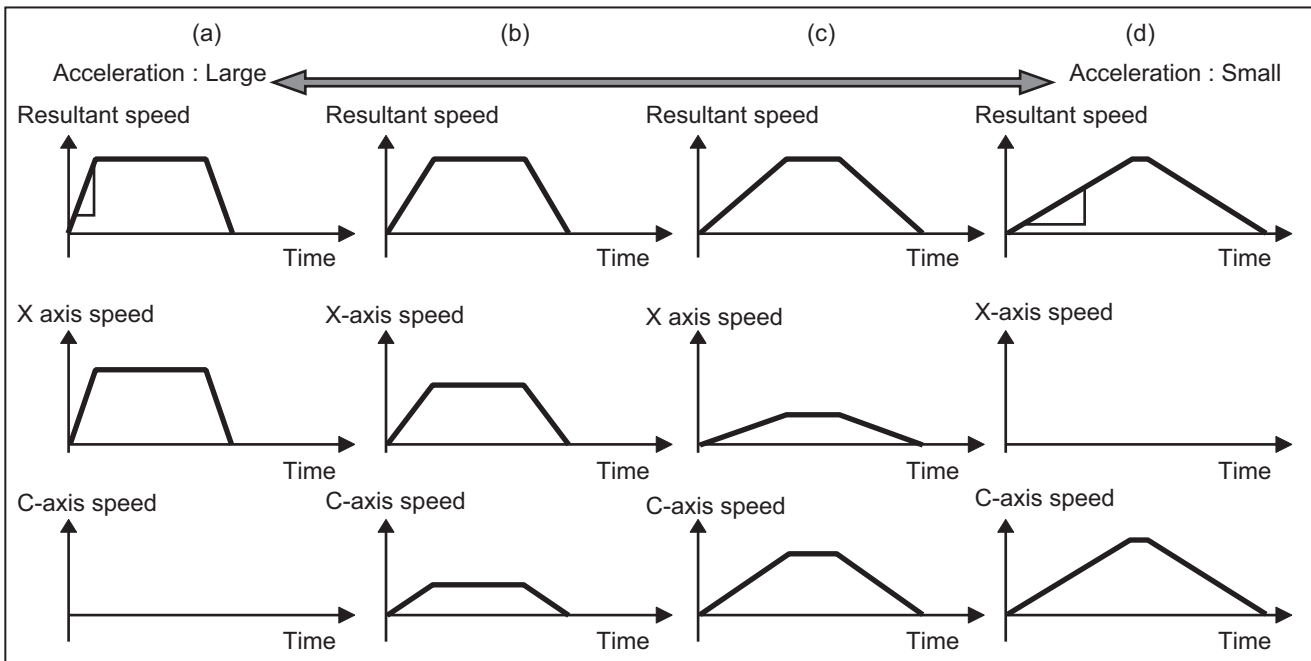
#1206 G1bF	10000 (mm/min)
#1207 G1btL	100 (ms)

It is assumed that only the acceleration for the rotary axis is set for "#2157 G1bFx" and "#2158 G1btLx". ("#1206 G1bF" and "#1207 G1btL" are used by not setting the acceleration for the linear axis.)

	X	Y	Z	C
#2157 G1bFx	0 (not set)	0 (not set)	0 (not set)	10000 (mm/min)
#2158 G1btLx	0 (not set)	0 (not set)	0 (not set)	500 (ms)

The figure below shows movements with the above settings.

- (1) If only the X axis moves, acceleration/deceleration is performed at the acceleration set for the X axis. ... (a)
- (2) If only the C axis moves, acceleration/deceleration is performed at the acceleration set for the C axis. ... (d)
- (3) If both of the X and C axes move, acceleration/deceleration is performed at the optimum acceleration calculated within the range that the acceleration of each axis does not exceed the setting.
 - If the movement of the X axis is dominant ... (b)
 - If the movement of the C axis is dominant ... (c)





Precautions

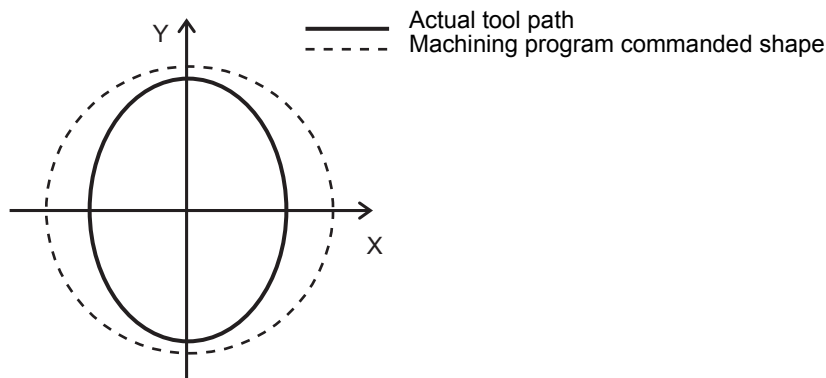
- (1) Under variable-acceleration pre-interpolation acceleration/deceleration, corner deceleration is realized with tolerable acceleration control for each axis.

Corner deceleration patterns and acceleration/deceleration patterns are as follows with each parameter setting:

#12060 VblAccPreInt Variable-acceleration Pre-interpolation Acceleration/Deceleration ON	0	0	1	1
#12053 EachAxAccCntrl Tolerable acceleration control for each axis ON	0	1	0	1
Corner deceleration pattern	Optimum corner deceleration	Tolerable acceleration control for each axis		
Acceleration/deceleration pattern	Acceleration/deceleration before interpolation		Variable-acceleration pre-interpolation acceleration/deceleration	

- (2) This function can only be used under SSS control. This means that variable-acceleration pre-interpolation acceleration/deceleration is also disabled during a modal that temporarily disables SSS control. As a result, the tool is under tolerable acceleration control for each axis. In this mode, the acceleration is determined by "#1206 G1bF" and "#1207 G1btL". Out of #2157 and #2158, set the longer one for #1206 and #1207. (Make a note of the original values and restore them as necessary.)
Refer to "17.2.2 SSS Control" for modals that temporarily disable SSS control.

- (3) Basically, set the same acceleration for base axes I, J, and K. A different acceleration causes a distorted shape against an arc command.
The figure below shows an example where the acceleration in the Y direction is greater than that in the X direction.



17.2.5 Initial High-accuracy Control

If "#1148 L_G611" (Initial high-accuracy) is set by the MTB specifications, high-accuracy control-related functions can be enabled when the power is turned ON. At power ON, the modes set by this parameter are enabled, but each mode can be changed to a different one by commanding as follows in the machining program.

#1148 setting value	Modes enabled at power ON
0	G08P0/G64 (cutting mode) command
1	G08P1/G61.1 (high-accuracy control mode) command
2	G05.1Q1 (high-speed high-accuracy control I mode) command
3	G05P10000 (high-speed high-accuracy control II mode) command
4	G05P20000 (high-speed high-accuracy control III mode) command

It is impossible, however, to shift to the high-speed high-accuracy control II/high-speed high-accuracy control III mode during the high-speed high-accuracy control I. Likewise, it is also impossible to shift to the high-speed high-accuracy control I mode during the high-speed high-accuracy control II/high-speed high-accuracy control III mode. To shift to either mode, cancel the current high-speed high-accuracy control mode using "G05.1 Q0" or "G05 P0" first and then command the target mode.

If any function set by this parameter is not included in your machine's specifications, an available high-accuracy function with a number smaller than the parameter setting is enabled.

17.2.6 Multi-part System Simultaneous High-accuracy

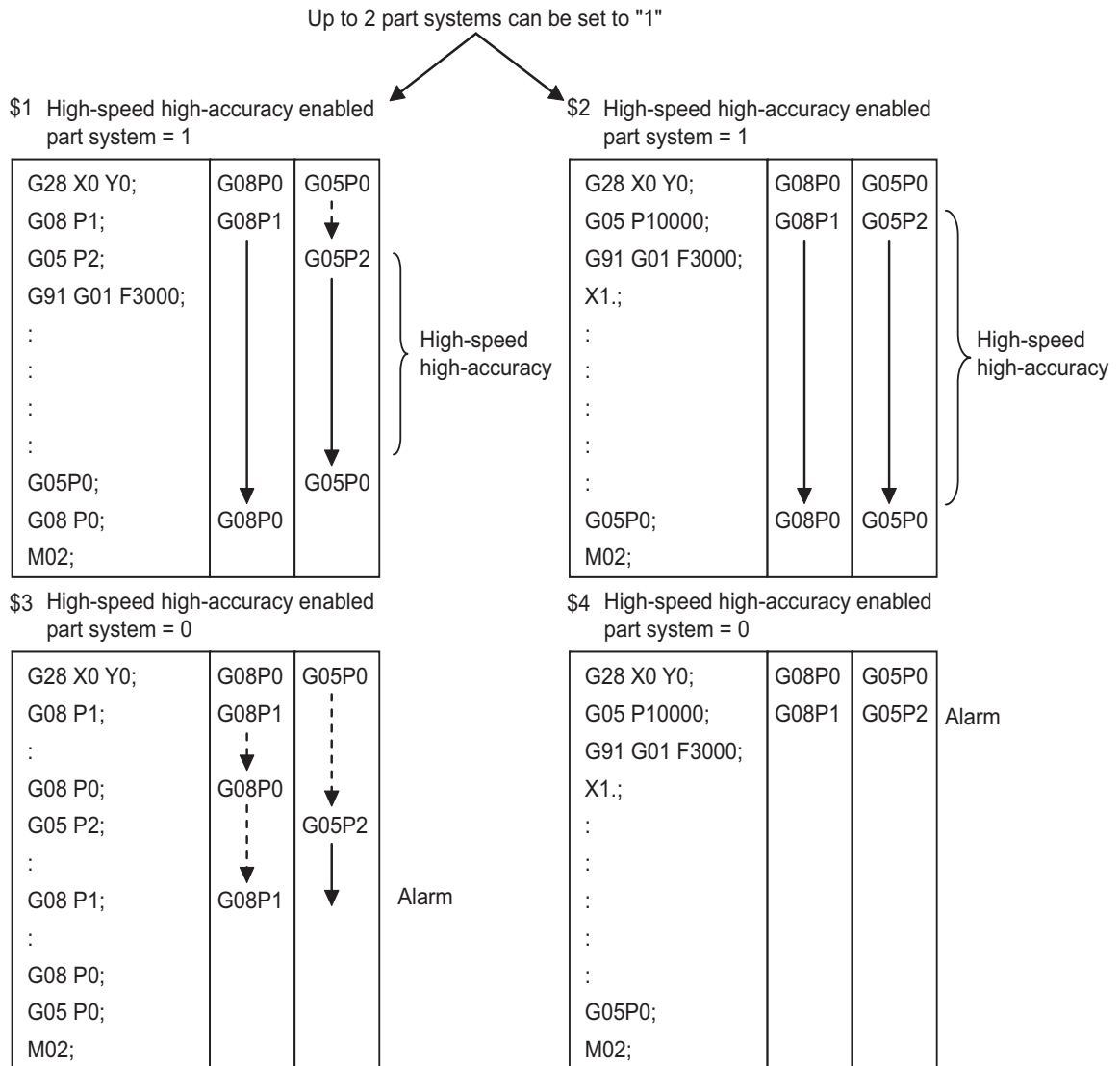


Function and purpose

High-accuracy control and high-speed machining mode are available respectively in all part systems, however, the simultaneous usage of high-accuracy control and high-speed machining mode (including High-speed high-accuracy control I/II/III) are available only in part systems which are limited by the parameter "#8040 High-SpeedAcc". While high-accuracy control and high-speed machining mode are available simultaneously in a part system where this parameter is set to "1", a program error (P129) will occur in those where the parameter is set to "0" when commanded. Also, for part systems where "#8040 High-SpeedAcc" is set to "0", "#1148 I_G611" must be set to "0" (Cutting mode when the power is turned ON) or "1" (High-accuracy control mode when the power is turned ON). If the parameter "#1148 I_G611" is set to a value other than "0" and "1", the parameter is regarded as being set to "1".

Note that up to two part systems can be set to use high-accuracy control and high-speed machining mode simultaneously. If three or more part systems are set as such, an MCP alarm (Y51 0032) will occur.

If the parameter "#8040 High-SpeedAcc" is set to "0" for all part systems, only the first part system is handled as the one with the parameter set to "1".



(Note) It is limited also in G61.1 command.

Although some MTB specifications support the high-accuracy acceleration/deceleration time constant expansion specifications, only one part system can be used. Multi-part systems cannot be used even if the high-accuracy acceleration/deceleration time constant expansion specifications are valid. For multi-part systems, "#1207 G1btL" must be set to a value within the setting range that is applicable when there are no high-accuracy acceleration/deceleration time constant expansion specifications.

Refer to the following chapters for details of each high-accuracy control.

"17.2 High-accuracy Control"

"17.3 High-speed High-accuracy Control"



Detailed description

- (1) When "#1148 I_G611" (Initial hi-precis) is enabled, the initial modal state after power ON will be the high-accuracy control mode. Refer to "17.2.5 Initial High-accuracy Control" for details.
In this case, the high-accuracy control mode is enabled if the multi-part system simultaneous high-accuracy specification is provided. Otherwise, the 1st part system enters the high-accuracy control mode, but the 2nd part system enters the cutting mode.
- (2) If you use the high-accuracy acceleration/deceleration time constant expansion function together with the multi-part system simultaneous high-accuracy function, an MCP alarm (Y51 0020) will occur.
Make sure to disable the high-accuracy acceleration/deceleration time constant extension function when you use the multi-part system simultaneous high-accuracy function.

17.3 High-speed High-accuracy Control

It depends on the MTB specifications whether the modal state at power ON is high-speed high-accuracy control I, II, III, or OFF. It also depends on the specifications whether to hold the modal state at reset.

Refer to the specifications of your machine.

In the main text, the axis address refers to the address of an axis that exists on the machine.

It corresponds to the address designated in the parameters "#1013 axname" and "#1014 incax".

These parameter settings depend on the MTB specifications.

17.3.1 High-speed High-accuracy Control I, II, III ; G05.1 Q1/Q0, G05 P10000/P0, G05 P20000/P0



Function and purpose

This function runs a machining program that approximates a freely curved surface with fine segments at high speed and with high-level accuracy. This is effective in increasing the speed of machining dies of a freely curved surface. This function is useful for machining which needs to make an edge at a corner or reduce an error from an inner route of curved shape.

A higher fine segment processing capability leads to a faster cutting speed, resulting in a shorter cycle time and a better machining surface quality. kBPM, the unit for the fine segment processing capability, is an abbreviation of "kilo blocks per minute" and refers to the number of machining program blocks that can be processed per minute.

Fine segment capacity for 1-part system

G01 block fine segment capacity for 1mm segment (unit: kBPM)

The performance below applies under the following conditions.

- ♦6-axis system (including spindle) or less
- ♦1-part system
- ♦3 axes or less commanded simultaneously in G01
- ♦ The block containing only the axis name and movement amount (Macro and variable command are not included.)
- ♦Tool radius compensation cancel mode (G40)

When the above conditions are not satisfied, the given feedrate may not be secured.

	Fine segment capacity				Restriction in the program
	M850 / M830	M80W	M80		
			Type A	Type B	
High-speed high-accuracy function I mode	67.5	33.7	33.7	33.7	Yes
High-speed high-accuracy function II mode	168 (*1) (*2)	67.5	67.5	67.5	Yes
High-speed high-accuracy function III mode	270 (*1) (*2)	135 (*3)	135 (*3)	-	Yes

(*1) When the fairing is valid (When the parameter "#8033" is set to "1"), and the fairing is executed successively, depending on machining programs, the performance of fine segment execution may decelerate more than the value described in the above table. In the network connection, the value described in the above table may not be guaranteed depending on the state.

(*2) 100 kBPM for a time constant expansion system or while tool center point control, inclined surface machining, or workpiece installation error compensation is in process.

(*3) 67.5 kBPM for a time constant expansion system or while tool center point control, inclined surface machining, or workpiece installation error compensation is in process.

Fine segment capacity for multi-part system

G01 block fine segment capacity for 1mm segment (unit: kBPM)

The fine segment processing capability below applies under the following conditions.

- ♦3 axes or less commanded simultaneously in G01
- ♦The block containing only the axis name and axis movement amount (Macro and variable command are not included.)
- ♦Tool radius compensation OFF (G40)

When the above conditions are not satisfied, the given feedrate may not be secured.

(1) High-speed high-accuracy control I

Number of part systems/ number of axes	Number of part systems (#8040=1)	M850 / M830	M80	
			Type A	Type B
1-part system	1 part system	67.5	33.7	33.7
2-part system	1 part system	67.5	33.7	33.7
	2 part systems	33.7	16.8	16.8
4-part system	1 part system	- (*1)	- (*1)	- (*1)
Up to 16 axes	2 part systems	- (*1)	- (*1)	- (*1)
5 part systems or more or 17 axes or more	1 part system	- (*1)	- (*1)	- (*1)
	2 part systems	- (*1)	- (*1)	- (*1)

(2) High-speed high-accuracy control II

Number of part systems/ number of axes	Number of part systems (#8040=1)	M850 / M830	M80	
			Type A	Type B
1-part system	1 part system	168 (*3)	67.5	67.5
2-part system	1 part system	100	67.5	67.5
	2 part systems	67.5	33.7	33.7
4-part system	1 part system	- (*1)	- (*1)	- (*1)
Up to 16 axes	2 part systems	- (*1)	- (*1)	- (*1)
5 part systems or more or 17 axes or more	1 part system	- (*1)	- (*1)	- (*1)
	2 part systems	- (*1)	- (*1)	- (*1)

(3) High-speed high-accuracy control III

Number of part systems/ number of axes	Number of part systems (#8040=1)	M850 / M830	M80	
			Type A	Type B
1-part system	1 part system	270 (*3)	135 (*4)	- (*2)
2-part system	1 part system	168	135	- (*2)
	2 part systems	100	67.5	- (*2)
4-part system	1 part system	- (*1)	- (*2)	- (*2)
Up to 16 axes	2 part systems	- (*1)	- (*2)	- (*2)
5 part systems or more or 17 axes or more	1 part system	- (*1)	- (*2)	- (*2)
	2 part systems	- (*1)	- (*2)	- (*2)

(*1) This system cannot be used for this model.

(*2) There are no corresponding high-speed high-accuracy control specifications.

(*3) 100 kBPM for a time constant expansion system or while tool center point control, inclined surface machining, or workpiece installation error compensation is in process.

(*4) 67.5 kBPM for a time constant expansion system or while tool center point control, inclined surface machining, or workpiece installation error compensation is in process.

(The time constant expansion system can be used in a system configured with a single part system when the specification is valid.)

High-speed high-accuracy control simultaneously for two part systems

High-speed high-accuracy control I, II, III can be used simultaneously in up to two part systems.

High-speed high-accuracy control I, II, III can be used in a part system where "1" is set for the parameter "#8040 High-SpeedAcc". A program error occurs (P129) if this is commanded for a part system where "0" is set for the parameter.

If the parameter "#8040 High-SpeedAcc" is set to "0" for all part systems, only the first part system is handled as the one with the parameter set to "1". Also, a part system where the parameter "#1148 Initial hi-precis" is set to "2" to "4" is handled as the one with the parameter "#8040 High-SpeedAcc" set to "1".

The parameter "#8040 High-SpeedAcc" can be set to "1" for up to two part systems. If 3 or more part systems are set to "1", an MCP alarm (Y51 0032) occurs. When "1" is set for two part systems, the fine segment processing capability decreases compared to when "1" is set only for one part system.

**Command format**

G05.1 Q1 ;	High-speed high-accuracy control I ON
-------------------	--

G05.1 Q0 ;	High-speed high-accuracy control I OFF
-------------------	---

G05 P10000 ;	High-speed high-accuracy control II ON
---------------------	---

G05 P20000 ;	High-speed high-accuracy control III ON
---------------------	--

G05 P0 ;	High-speed high-accuracy control II/III OFF
-----------------	--

Note

- (1) The high-speed high-accuracy mode II and III cannot be used at the same time.
- (2) These commands are valid regardless of the parameter "#1267 ext03/bit0" setting if the specifications are available.
- (3) High-speed high-accuracy control III can also be used by setting a parameter instead of a G code.
If the parameter "#8131 High speed/accu 3" is set to "1", the high-speed high-accuracy control II command can be handled as the III command. This also enables the high-speed high-accuracy control III mode in the machining program using "G05P10000". Likewise, the G05P2 command issued during a high-accuracy control mode can be handled as the high-speed high-accuracy control III command.
Furthermore, by setting "#1148 Initial hi-precis" to "4", the high-speed high-accuracy control III mode can be set as the initial modal state after power ON.



Detailed description

- (1) The high-speed high-accuracy control I / II / III can be used during tape, MDI, SD card or memory modes.
- (2) The override, maximum cutting speed clamp, single block operation, dry run, handle interrupt and graphic trace are valid even during the high-speed high-accuracy control I / II / III modal.
- (3) The machining speed may drop depending on the number of characters in one block.
- (4) The high-speed high-accuracy control I / II / III function automatically turns the high-accuracy control mode ON. For high-accuracy control function, refer to "17.3 High-speed High-accuracy Control".
- (5) Turn the tool radius compensation command ON and OFF during the high-speed high-accuracy control I/II/III mode.
If the high-speed high-accuracy control I/II/III mode is turned OFF without turning the tool radius compensation OFF, a program error (P34) will occur.
- (6) Turn the high-speed high-accuracy control I / II / III mode OFF before commanding data other than those that can be commanded.
- (7) When using the high-speed high-accuracy control II / III mode, it is necessary to set the parameter "#1572 Cirorp" to eliminate the speed fluctuation at the seams between arc and straight line or arc and arc. This parameter, however, depends on the MTB specifications.
- (8) Feedrate command F is clamped with the "#2110 Clamp (H-precision)" (Cutting feed clamp speed for high-accuracy control mode) set with parameter.
- (9) Rapid traverse rate enables "#2109 Rapid(H-precision)" (Rapid traverse rate during high-accuracy control mode) set by the parameter.
- (10) When the "#2109 Rapid(H-precision)" is set to "0", the movement follows "#2001 rapid" (rapid traverse rate) set by the parameter. Also, when "#2110 Clamp (H-precision)" is set to "0", the speed will be clamped with "#2002 clamp" (Cutting clamp speed) set with parameter.

Enabling conditions

To enable each high-speed high-accuracy control function, it is necessary to satisfy the following conditions respectively:

- (1) The specification of each function is valid. (*1)
- (2) Each function is in a valid modal state. (Refer to "Relationship with other functions".)
- (3) Each function is enabled by one of the following procedures:
 - ♦Command each in the machining program. (*2)
 - ♦Set each for the parameter "#1148 Initial hi-precis". (The modal at power ON corresponds to each high-speed high-accuracy control function.)

	#1148 setting
High-speed high-accuracy control I	2
High-speed high-accuracy control II	3 (*3)
High-speed high-accuracy control III	4 (*4)

- (*1) The following conditions are additionally required to enable high-speed high-accuracy control III.
 - ♦The time constant expansion system is invalid.
 - ♦The SSS control specifications are valid, and the parameter "#8090 SSS ON" is set to "1".

If high-speed high-accuracy control III is commanded when the SSS control mode is set to OFF, high-speed high-accuracy control II is enabled. (However, this is available only when the conditions defined in "Relationship with other functions" are satisfied.)
- (*2) High-speed high-accuracy control III is also enabled by the following commands. (However, this is available only when the conditions defined in "Relationship with other functions" are satisfied.)
While the parameter "#8131 High speed/accu 3" is set to "1", command "G05 P10000" (high-speed high-accuracy control II) from the machining program.
- (*3) When the parameter "#1148 Initial hi-precis" is set to "3" (high-speed high-accuracy control II) and the parameter "#8131 High speed/accu 3" to "1" ("High speed/accu 3" is valid), the initial modal state after power ON changes to the high-speed high-accuracy control II mode.
- (*4) However, when the parameter "#1074 Initial sync feed" is set to "1", the modal state changes to the high-speed high-accuracy control II mode.



Relationship with other functions

Relationship between the high-speed high-accuracy control I and other functions

(1) Relationship between the high-speed high-accuracy control I and the other G code functions

Column A: Operation when the additional function is commanded while the high-speed high-accuracy control I is enabled

Column B: Operation when the high-speed high-accuracy control I (G05.1Q1) is commanded while the additional function is enabled

○: The high-speed high-accuracy control I and the additional function are both enabled

△: The high-speed high-accuracy control I is temporarily canceled, while the additional function is enabled

X: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)

-: No combination

□: Others

Group	G code	Function name	A	B
0	G04	Dwell	△	-
	G05P0	High-speed machining mode II OFF High-speed high-accuracy control II OFF High-speed high-accuracy control III OFF	X (P34)	□ (*2)
	G05P2	High-speed machining mode II ON	□ (*4)	□ (*2)
	G05P10000	High-speed high-accuracy control II ON	X (P34)	X (P34)
	G05P20000	High-speed high-accuracy control III ON	X (P34)	X (P34)
	G05.1Q0	High-speed high-accuracy control I OFF Spline interpolation OFF	□ (*1)	□ (*2)
	G05.1Q1	High-speed high-accuracy control I ON	□ (*3)	□ (*3)
	G05.1Q2	Spline interpolation ON	× (P34) (*10)	× (P34) (*10)
	G07	Hypothetical axis interpolation	△ (*10)	△ (*10)
	G08P0	High-accuracy control OFF	□ (*3)	□ (*2)
	G08P1	High-accuracy control ON	□ (*3)	□ (*2)
	G09	Exact stop check	△	-
	G10 I_ J_ G10 K_	Parameter coordinate rotation input	△ (*10)	- (*10)
	G10 L2	Compensation data input by program	△	-
	G10 L70 G10 L50	Parameter input by program	△	-
	G27	Reference position check	△	-
	G28	Reference position return	△	-
	G29	Start position return	△	-
	G30	2nd to 4th reference position return	△	-
	G30.1- G30.6	Tool exchange position return	△	-
	G31	Skip Multiple-step skip 2	△	-
	G31.1- G31.3	Multi-step skip	△	-
	G34-G36 G37.1	Special fixed cycle	△	-

Group	G code	Function name	A	B
0	G37	Automatic tool length measurement	Δ	-
	G38	Tool radius compensation vector designation	Δ	-
	G39	Tool radius compensation corner circular command	Δ	-
	G52	Local coordinate system setting	Δ	-
	G53	Machine coordinate system selection	Δ	-
	G60	Unidirectional positioning	Δ	-
	G65	User macro simple call	□ (*5)	□ (*6)
	G92	Coordinate system setting	Δ	-
	G92.1	Workpiece coordinate system preset	Δ (*10)	- (*10)
	G122	Sub part system control I	× (P652) (*10)	□ (*7) (*10)
	G144	Sub part system control II	× (P652) (*10)	□ (*7) (*10)
1	G00	Positioning	Δ	Δ
	G01	Linear interpolation	○	○
	G02 G03	Circular interpolation	□ When SSS is enabled: ○ When SSS is disabled: Δ	□ When SSS is enabled: ○ When SSS is disabled: Δ
	G02.1 G03.1	Spiral interpolation	Δ (*10)	Δ (*10)
	G02.3 G03.3	Exponential interpolation	Δ (*10)	Δ (*10)
	G02.4 G03.4	3-dimensional circular interpolation	Δ (*10)	Δ (*10)
	G06.2	NURBS interpolation	× (P34) (*10)	× (P34) (*10)
	G33	Thread cutting	Δ	Δ
2	G17-G19	Plane selection	○	○
3	G90	Absolute value command	○	○
	G91	Incremental value command	○	○
4	G22	Stroke check before travel ON	○	○
	G23	Stroke check before travel OFF	○	○
5	G93	Inverse time feed	× (P125) (*10)	× (P125) (*10)
	G94	Asynchronous feed (feed per minute)	○	○
	G95	Synchronous feed (feed per revolution)	○	○
6	G20	Inch command	○	○
	G21	Metric command	○	○
7	G40	Tool radius compensation cancel	○	○
	G41 G42	Tool radius compensation	○	× (P29)
	8	G43 G44	Tool length offset	○
G43.1		Tool length compensation along the tool axis	○ (*10)	× (P29) (*10)
G43.4 G43.5		Tool center point control	○ (*10)	× (P29) (*10)
G49		Tool length offset cancel	○	○
9	G80	Fixed cycle cancel	○	○
	Group 9 Other than G80	Fixed cycle	Δ	Δ
10	G98	Fixed cycle initial level return	○	○
	G99	Fixed cycle R point return	○	○
11	G50	Scaling cancel	○	○
	G51	Scaling ON	○	X (P34)

Group	G code	Function name	A	B
12	G54-G59 G54.1	Workpiece coordinate system selection	○	○
13	G61	Exact stop check mode	□ (*8)	□ (*9)
	G61.1	High-accuracy control	□ (*3)	□ (*2)
	G61.2	High-accuracy spline	× (P29) (*10)	× (P29) (*10)
	G61.4	Spline interpolation 2	○ (*10)	○ (*10)
	G62	Automatic corner override	□ (*3)	□ (*2)
	G63	Tapping mode	□ (*3)	□ (*2)
	G64	Cutting mode	□ (*3)	□ (*2)
14	G66 G66.1	User macro modal call	□ (*5)	□ (*6)
	G67	User macro modal call cancel	○	○
15	G40.1	Normal line control cancel	○	○
	G41.1 G42.1	Normal line control	× (P29) (*10)	× (P29) (*10)
16	G68	Coordinate rotation by program ON	○	X (P34)
	G68.2 G68.3	Inclined surface machining command	○ (*10)	○ (*10)
	G69	Coordinate rotation cancel	○	○
17	G96	Constant surface speed control ON	○	○
	G97	Constant surface speed control OFF	○	○
18	G15	Polar coordinate command OFF	○	○
	G16	Polar coordinate command ON	× (P34)	× (P34)
19	G50.1	Mirror image OFF	○	○
	G51.1	Mirror image ON	○	× (P34)
21	G07.1	Cylindrical interpolation	× (P485)	Δ
	G12.1	Polar coordinate interpolation ON	× (P485)	Δ
	G13.1	Polar coordinate interpolation OFF	○	○
27	G54.4P0	Workpiece installation error compensation cancel	○	○
	G54.4 P1-P7	Workpiece installation error compensation	○ (*10)	○ (*10)

(*1) Disables the high-speed high-accuracy control I.

(*2) Enables the high-speed high-accuracy control I.

(*3) High-speed high-accuracy control I continues.

(*4) Enables the high-speed machining mode II.

(*5) Enables the high-speed high-accuracy control I in a macro program.

(*6) Enables the high-speed high-accuracy control I if G05.1Q1 is commanded in a macro program.

(*7) Enables the high-speed high-accuracy control I if G05.1Q1 is commanded in a sub part system.

(*8) Enables the exact stop check mode.

(*9) Exact stop check mode continues.

(*10) In M80 Type B, the following program errors occur depending on the G codes.

G code	Program error	G code	Program error
G05.1Q2, G92.1, G122, G61.2, G61.4	P39	G10 I_J_/G10 K_	P260
G144, G54.4 P1-P7	P34	G06.2	P550
G07	P80	G93	P124
G02.1, G03.1	P73	G41.1, G42.1	P900
G02.3, G03.3	P611	G43.1	P930
G02.4, G03.4	P76	G43.4, G43.5	P940
		G68.2, G68.3	P950

- (2) Relationship between the high-speed high-accuracy control I and functions other than G codes
- Column A: Operation when the additional function is commanded while the high-speed high-accuracy control I is enabled
- Column B: Operation when the high-speed high-accuracy control I (G05.1Q1) is commanded while the additional function is enabled
- : The high-speed high-accuracy control I and the additional function are both enabled
 - △: The high-speed high-accuracy control I is temporarily canceled, while the additional function is enabled
 - X: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)
 - : No combination
 - : Others

Function name	A	B
SSS ON	-	○
Mirror image by parameter setting ON	-	X (P34)
PLC mirror image ON	-	X (P34)
Coordinate rotation by parameter	-	△
Subprogram call (M98)	□ (*1)	□ (*2)
Figure rotation (M98 I_J_K_)	□ (*8) (*10)	□ (*9) (*10)
Timing synchronization between part systems	□ (*3)	-
MTB macro	□ (*4)	□ (*5)
Macro interruption	□ (*6)	□ (*7)
PLC interruption	□ (*6)	□ (*7)
Corner chamfering/Corner R	△	-
Linear angle command	○	-
Geometric command	○	-
Chopping	○	○
Optional block skip	○	-

- (*1) Enables the high-speed high-accuracy control I in a subprogram.
- (*2) Enables the high-speed high-accuracy control I if G05.1Q1 is commanded in a subprogram.
- (*3) Enables timing synchronization.
- (*4) Enables the high-speed high-accuracy control I in a MTB program.
- (*5) Enables the high-speed high-accuracy control I if G05.1Q1 is commanded in a MTB program.
- (*6) Enables the high-speed high-accuracy control I in an interrupt program.
- (*7) Enables the high-speed high-accuracy control I if G05.1Q1 is commanded in an interrupt program.
- (*8) Disables the high-speed high-accuracy control I in a figure rotation subprogram.
- (*9) The high-speed high-accuracy control I is disabled even if G05.1Q1 is commanded in a figure rotation subprogram.
- (*10) In M80 Type B, the following program error occurs depending on the function type.

Function	Program error
Figure rotation (M98 I_J_K_)	P250

Relationship between the high-speed high-accuracy control II and other functions

(1) Relationship between the high-speed high-accuracy control II and G code functions

Column A: Operation when the additional function is commanded while the high-speed high-accuracy control II is enabled

Column B: Operation when the high-speed high-accuracy control II (G05P10000) is commanded while the additional function is enabled

○: The high-speed high-accuracy control II and the additional function are both enabled

Δ: The high-speed high-accuracy control II is temporarily canceled, while the additional function is enabled

X: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)

-: No combination

□: Others

Group	G code	Function name	A	B
0	G04	Dwell	Δ	-
	G05P0	High-speed machining mode II OFF High-speed high-accuracy control II OFF High-speed high-accuracy control III OFF	□ (*1)	□ (*2)
	G05P2	High-speed machining mode II ON	□ (*4)	□ (*2)
	G05P10000	High-speed high-accuracy control II ON	□ (*3)	□ (*3)
	G05P20000	High-speed high-accuracy control III ON	□ (*2)	□ (*2)
	G05.1Q0	High-speed high-accuracy control I OFF Spline interpolation OFF	□ (*3)	□ (*2)
	G05.1Q1	High-speed high-accuracy control I ON	X (P34)	X (P34)
	G05.1Q2	Spline interpolation ON	○ (*8)	○ (*8)
	G07	Hypothetical axis interpolation	Δ (*8)	Δ (*8)
	G08P0	High-accuracy control OFF	□ (*3)	□ (*2)
	G08P1	High-accuracy control ON	□ (*3)	□ (*2)
	G09	Exact stop check	Δ	-
	G10 I_ J_ G10 K_	Parameter coordinate rotation input	Δ (*8)	- (*8)
	G10 L2	Compensation data input by program	Δ	-
	G10 L70 G10 L50	Parameter input by program	Δ	-
	G27	Reference position check	Δ	-
	G28	Reference position return	Δ	-
	G29	Start position return	Δ	-
	G30	2nd to 4th reference position return	Δ	-
	G30.1- G30.6	Tool exchange position return	Δ	-
	G31	Skip Multiple-step skip 2	Δ	-
	G31.1- G31.3	Multi-step skip	Δ	-
	G34-G36 G37.1	Special fixed cycle	Δ	-
	G37	Automatic tool length measurement	Δ	-
	G38	Tool radius compensation vector designa- tion	Δ	-
	G39	Tool radius compensation corner circular command	Δ	-
	G52	Local coordinate system setting	Δ	-
	G53	Machine coordinate system selection	Δ	-
	G60	Unidirectional positioning	Δ	-

Group	G code	Function name	A	B
0	G65	User macro simple call	□ (*5)	□ (*6)
	G92	Coordinate system setting	Δ	-
	G92.1	Workpiece coordinate system preset	Δ (*8)	- (*8)
	G122	Sub part system control I	× (P652) (*8)	□ (*7) (*8)
	G144	Sub part system control II	× (P652) (*8)	□ (*7) (*8)
1	G00	Positioning	Δ	Δ
	G01	Linear interpolation	○	○
	G02 G03	Circular interpolation	○	○
	G02.1 G03.1	Spiral interpolation	Δ (*8)	Δ (*8)
	G02.3 G03.3	Exponential interpolation	Δ (*8)	Δ (*8)
	G02.4 G03.4	3-dimensional circular interpolation	Δ (*8)	Δ (*8)
	G06.2	NURBS interpolation	○ (*8)	○ (*8)
	G33	Thread cutting	Δ	Δ
2	G17-G19	Plane selection	○	○
3	G90	Absolute value command	○	○
	G91	Incremental value command	○	○
4	G22	Stroke check before travel ON	Δ	Δ
	G23	Stroke check before travel OFF	○	○
5	G93	Inverse time feed	Δ (*8)	Δ (*8)
	G94	Asynchronous feed (feed per minute)	○	○
	G95	Synchronous feed (feed per revolution)	Δ	Δ
6	G20	Inch command	○	○
	G21	Metric command	○	○
7	G40	Tool radius compensation cancel	○	○
	G41 G42	Tool radius compensation	○	○
	8	G43 G44	Tool length offset	○
G43.1		Tool length compensation along the tool axis	○ (*8)	○ (*8)
G43.4 G43.5		Tool center point control	○ (*8)	○ (*8)
G49		Tool length offset cancel	○	○
9	G80	Fixed cycle cancel	○	○
	Group 9 Other than G80	Fixed cycle	Δ	Δ
10	G98	Fixed cycle initial level return	○	○
	G99	Fixed cycle R point return	○	○
11	G50	Scaling cancel	○	○
	G51	Scaling ON	Δ	Δ
12	G54-G59 G54.1	Workpiece coordinate system selection	○	○

Group	G code	Function name	A	B
13	G61	Exact stop check mode	Δ	Δ
	G61.1	High-accuracy control	□ (*3)	□ (*2)
	G61.2	High-accuracy spline	X (P29)	X (P29)
	G61.4	Spline interpolation 2	○ (*8)	○ (*8)
	G62	Automatic corner override	Δ	Δ
	G63	Tapping mode	Δ	Δ
	G64	Cutting mode	□ (*3)	□ (*2)
14	G66 G66.1	User macro modal call	Δ	Δ
	G67	User macro modal call Cancel	○	○
15	G40.1	Normal line control cancel	○	○
	G41.1 G42.1	Normal line control	× (P29) (*8)	× (P29) (*8)
16	G68	Coordinate rotation by program ON	Δ	Δ
	G68.2 G68.3	Inclined surface machining command	○ (*8)	○ (*8)
	G69	Coordinate rotation cancel	○	○
17	G96	Constant surface speed control ON	○	○
	G97	Constant surface speed control OFF	○	○
18	G15	Polar coordinate command OFF	○	○
	G16	Polar coordinate command ON	Δ	Δ
19	G50.1	Mirror image OFF	○	○
	G51.1	Mirror image ON	○	○
21	G07.1	Cylindrical interpolation	X (P34)	X (P481)
	G12.1	Polar coordinate interpolation ON	X (P34)	X (P481)
	G13.1	Polar coordinate interpolation OFF	○	○
27	G54.4P0	Workpiece installation error compensation cancel	○	○
	G54.4 P1-P7	Workpiece installation error compensation	○ (*8)	○ (*8)

(*1) Disables the high-speed high-accuracy control II.

(*2) Enables the high-speed high-accuracy control II.

(*3) High-speed high-accuracy control II continues.

(*4) Enables the high-speed machining mode II.

(*5) Enables the high-speed high-accuracy control II in a macro program.

(*6) Enables the high-speed high-accuracy control II if G05P10000 is commanded in a macro program.

(*7) Enables the high-speed high-accuracy control II if G05P10000 is commanded in a sub part system.

(*8) In M80 Type B, the following program errors occur depending on the G codes.

G code	Program error	G code	Program error
G05.1Q2, G92.1, G122, G61.2, G61.4	P39	G10 I_J_/G10 K_	P260
G144, G54.4 P1-P7	P34	G06.2	P550
G07	P80	G93	P124
G02.1, G03.1	P73	G41.1, G42.1	P900
G02.3, G03.3	P611	G43.1	P930
G02.4, G03.4	P76	G43.4, G43.5	P940
		G68.2, G68.3	P950

17 High-speed High-accuracy Control

- (2) Relationship between the high-speed high-accuracy control II and functions other than G codes
 Column A: Operation when the additional function is commanded while the high-speed high-accuracy control II is enabled
 Column B: Operation when the high-speed high-accuracy control II (G05P10000) is commanded while the additional function is enabled
 ○: The high-speed high-accuracy control II and the additional function are both enabled
 Δ: The high-speed high-accuracy control II is temporarily canceled, while the additional function is enabled
 X: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)
 -: No combination
 □: Others

Function name	A	B
SSS ON	-	○
Mirror image by parameter setting ON	-	Δ
PLC mirror image ON	-	Δ
Coordinate rotation by parameter	-	Δ
Subprogram call (M98)	□ (*1)	□ (*2)
Figure rotation (M98 I_J_K_)	□ (*8) (*10)	□ (*9) (*10)
Timing synchronization between part systems	□ (*3)	-
MTB macro	□ (*4)	□ (*5)
Macro interruption	□ (*6)	□ (*7)
PLC interruption	□ (*6)	□ (*7)
Corner chamfering/Corner R	Δ	-
Linear angle command	Δ	-
Geometric command	Δ	-
Chopping	○	○
Fairing/smooth fairing ON	○	○
Optional block skip	○	-

- (*1) Enables the high-speed high-accuracy control II in a subprogram.
 (*2) Enables the high-speed high-accuracy control II if G05P10000 is commanded in a subprogram.
 (*3) Enables timing synchronization.
 (*4) Enables the high-speed high-accuracy control II in a MTB program.
 (*5) Enables the high-speed high-accuracy control II if G05P10000 is commanded in a MTB program.
 (*6) Enables the high-speed high-accuracy control II in an interrupt program.
 (*7) Enables the high-speed high-accuracy control II if G05P10000 is commanded in an interrupt program.
 (*8) Disables the high-speed high-accuracy control II in a figure rotation subprogram.
 (*9) The high-speed high-accuracy control II is disabled even if G05P10000 is commanded in a figure rotation subprogram.
 (*10) In M80 Type B, the following program error occurs depending on the function type.

Function	Program error
Figure rotation (M98 I_J_K_)	P250

Relationship between the high-speed high-accuracy control III and other functions

(1) Relationship between the high-speed high-accuracy control III and G code functions

Column A: Operation when the additional function is commanded while the high-speed high-accuracy control III is enabled

Column B: Operation when the high-speed high-accuracy control III (G05P20000) is commanded while the additional function is enabled

○: The high-speed high-accuracy control III and the additional function are both enabled

Δ: The high-speed high-accuracy control III is temporarily canceled, while the additional function is enabled

X: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)

-: No combination

□: Others

Group	G code	Function name	A	B
0	G04	Dwell	Δ	-
	G05P0	High-speed machining mode II OFF High-speed high-accuracy control II OFF High-speed high-accuracy control III OFF	□ (*1)	□ (*2)
	G05P2	High-speed machining mode II ON	□ (*8)	□ (*2)
	G05P10000	High-speed high-accuracy control II ON	□ (*3)	□ (*2)
	G05P20000	High-speed high-accuracy control III ON	□ (*3)	□ (*3)
	G05.1Q0	High-speed high-accuracy control I OFF Spline interpolation OFF	□ (*3)	□ (*2)
	G05.1Q1	High-speed high-accuracy control I ON	X (P34)	X (P34)
	G05.1Q2	Spline interpolation ON	Δ	□ (*4)
	G07	Hypothetical axis interpolation	Δ	Δ
	G08P0	High-accuracy control OFF	□ (*4)	□ (*2)
	G08P1	High-accuracy control ON	□ (*4)	□ (*2)
	G09	Exact stop check	Δ	-
	G10 I _ G10 J _ G10 K _	Parameter coordinate rotation input	Δ	-
	G10 L2	Compensation data input by program	Δ	-
	G10 L70 G10 L50	Parameter input by program	Δ	-
	G27	Reference position check	Δ	-
	G28	Reference position return	Δ	-
	G29	Start position return	Δ	-
	G30	2nd to 4th reference position return	Δ	-
	G30.1- G30.6	Tool exchange position return	Δ	-
	G31	Skip Multiple-step skip 2	Δ	-
	G31.1- G31.3	Multi-step skip	Δ	-
	G34-G36 G37.1	Special fixed cycle	Δ	-
	G37	Automatic tool length measurement	Δ	-
	G38	Tool radius compensation vector designation	Δ	-
	G39	Tool radius compensation corner circular command	Δ	-
	G52	Local coordinate system setting	Δ	-

Group	G code	Function name	A	B
0	G53	Machine coordinate system selection	Δ	-
	G60	Unidirectional positioning	Δ	-
	G65	User macro simple call	□ (*5)	□ (*6)
	G92	Coordinate system setting	Δ	-
	G92.1	Workpiece coordinate system preset	Δ	-
	G122	Sub part system control I	× (P652)	□ (*7)
	G144	Sub part system control II	× (P652)	□ (*7)
1	G00	Positioning	Δ	Δ
	G01	Linear interpolation	○	○
	G02 G03	Circular interpolation	Δ	Δ
	G02.1 G03.1	Spiral interpolation	Δ	□ (*4)
	G02.3 G03.3	Exponential interpolation	Δ	× (P34)
	G02.4 G03.4	3-dimensional circular interpolation	Δ	× (P34)
	G06.2	NURBS interpolation	Δ	-
	G33	Thread cutting	Δ	□ (*4)
2	G17-G19	Plane selection	○	○
3	G90	Absolute value command	○	○
	G91	Incremental value command	○	○
4	G22	Stroke check before travel ON	Δ	□ (*4)
	G23	Stroke check before travel OFF	○	○
5	G93	Inverse time feed	Δ	□ (*4)
	G94	Asynchronous feed (feed per minute)	○	○
	G95	Synchronous feed (feed per revolution)	Δ	□ (*4)
6	G20	Inch command	○	○
	G21	Metric command	○	○
7	G40	Tool radius compensation cancel	○	○
	G41 G42	Tool radius compensation	Δ	□ (*4)
	8	G43 G44	Tool length offset	○
G43.1		Tool length compensation along the tool axis	Δ	□ (*4)
G43.4 G43.5		Tool center point control	Δ	□ (*4)
G49		Tool length offset cancel	○	○
9		G80	Fixed cycle cancel	○
	Group 9 Other than G80	Fixed cycle	Δ	□ (*4)
10	G98	Fixed cycle initial level return	○	○
	G99	Fixed cycle R point return	○	○
11	G50	Scaling cancel	○	○
	G51	Scaling ON	Δ	□ (*4)
12	G54-G59	Workpiece coordinate system selection	○	○
	G54.1			

Group	G code	Function name	A	B
13	G61	Exact stop check mode	<input type="checkbox"/> (*4)	<input type="checkbox"/> (*4)
	G61.1	High-accuracy control	<input type="checkbox"/> (*4)	<input type="checkbox"/> (*2)
	G61.2	High-accuracy spline	X (P29)	X (P29)
	G61.4	Spline interpolation 2	<input type="checkbox"/> (*4)	<input type="checkbox"/> (*4)
	G62	Automatic corner override	<input type="checkbox"/> (*4)	<input type="checkbox"/> (*4)
	G63	Tapping mode	<input type="checkbox"/> (*4)	<input type="checkbox"/> (*4)
	G64	Cutting mode	<input type="checkbox"/> (*4)	<input type="checkbox"/> (*2)
14	G66 G66.1	User macro modal call	Δ	<input type="checkbox"/> (*4)
	G67	User macro modal call cancel	○	○
15	G40.1	Normal line control cancel	X (P29)	○
	G41.1 G42.1	Normal line control	X (P29)	<input type="checkbox"/> (*4)
16	G68	Coordinate rotation by program ON	Δ	<input type="checkbox"/> (*4)
	G68.2 G68.3	Inclined surface machining command	Δ	<input type="checkbox"/> (*4)
	G69	Coordinate rotation cancel	○	○
17	G96	Constant surface speed control ON	○	○
	G97	Constant surface speed control OFF	○	○
18	G15	Polar coordinate command OFF	○	○
	G16	Polar coordinate command ON	Δ	<input type="checkbox"/> (*4)
19	G50.1	Mirror image OFF	○	○
	G51.1	Mirror image ON	Δ	<input type="checkbox"/> (*4)
21	G07.1	Cylindrical interpolation	X (P34)	X (P481)
	G12.1	Polar coordinate interpolation ON	X (P34)	X (P481)
	G13.1	Polar coordinate interpolation OFF	○	○
27	G54.4P0	Workpiece installation error compensation cancel	○	○
	G54.4P1-P7	Workpiece installation error compensation	Δ	<input type="checkbox"/> (*4)

(*1) Disables the high-speed high-accuracy control III.

(*2) Enables the high-speed high-accuracy control III.

(*3) High-speed high-accuracy control III continues.

(*4) Enables the high-speed high-accuracy control II.

(*5) Enables the high-speed high-accuracy control III in a macro program.

(*6) Enables the high-speed high-accuracy control III if G05P20000 is commanded in a macro program.

(*7) Enables the high-speed high-accuracy control III if G05P20000 is commanded in a sub part system.

(*8) Enables the high-speed machining mode II.

(2) Relationship between the high-speed high-accuracy control III and functions other than G codes

Column A: Operation when the additional function is commanded while the high-speed high-accuracy control III is enabled

Column B: Operation when the high-speed high-accuracy control III (G05P20000) is commanded while the additional function is enabled

○: The high-speed high-accuracy control III and the additional function are both enabled

Δ: The high-speed high-accuracy control III is temporarily canceled, while the additional function is enabled

X: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)

-: No combination

□: Others

Function name	A	B
SSS ON	-	○
SSS OFF	-	□ (*1)
Mirror image by parameter setting ON	-	Δ
PLC mirror image ON	-	Δ
Coordinate rotation by parameter	-	Δ
Subprogram call (M98)	□ (*2)	□ (*3)
Figure rotation (M98 I_J_K_)	□ (*9)	□ (*10)
Timing synchronization between part systems	□ (*4)	-
MTB macro	□ (*5)	□ (*6)
Macro interruption	□ (*7)	□ (*8)
PLC interruption	□ (*7)	□ (*8)
Corner chamfering/Corner R	Δ	-
Linear angle command	Δ	-
Geometric command	Δ	-
Chopping	○	○
Fairing/smooth fairing ON	□ (*1)	□ (*1)
Optional block skip	□ (*1)	-

(*1) Enables the high-speed high-accuracy control II.

(*2) Enables the high-speed high-accuracy control III in a subprogram.

(*3) Enables the high-speed high-accuracy control III if G05P20000 is commanded in a subprogram.

(*4) Enables timing synchronization.

(*5) Enables the high-speed high-accuracy control III in a MTB program.

(*6) Enables the high-speed high-accuracy control III if G05P20000 is commanded in a MTB program.

(*7) Enables the high-speed high-accuracy control III in an interrupt program.

(*8) Enables the high-speed high-accuracy control III if G05P20000 is commanded in an interrupt program.

(*9) Disables the high-speed high-accuracy control III in a figure rotation subprogram.

(*10) The high-speed high-accuracy control III is disabled even if G05P20000 is commanded in a figure rotation subprogram.

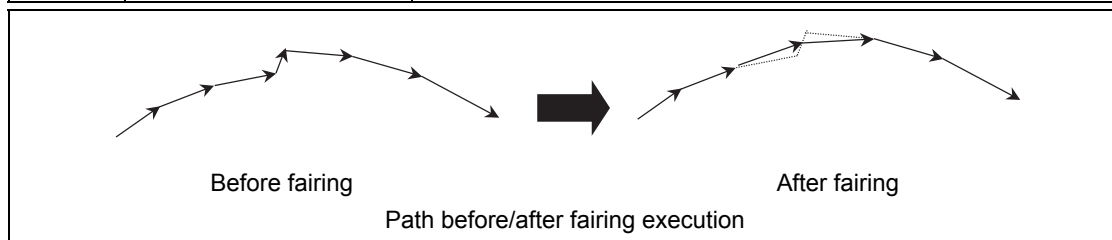
17.3.2 Fairing

**Function and purpose**

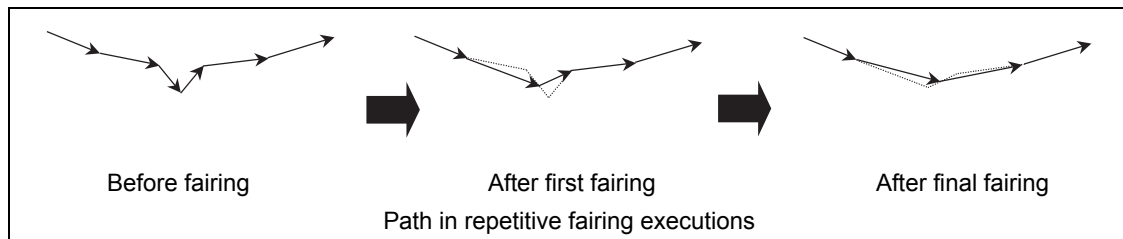
This function is an additional function when the high-speed high-accuracy control II or III mode is ON. If there is a protrusion in a path (zigzagging path) in a machining program generated with a CAM, etc., this function can be used to eliminate the protruding path smaller than the setting value so that the protruding path is smoothly connected with the previous and the next paths.

This function is valid only for continuous linear commands (G01).

Related parameter		Contents
#8033	Fairing ON	0: Fairing invalid 1: Execute fairing for the protruding block. 2: Smooth fairing valid
#8029	Fairing L	Execute fairing for the shorter block than this setting value.



If there is any protruding path after fairing, fairing is repeated.



17.3.3 Smooth Fairing



Function and purpose

This function is an additional function when the high-speed high-accuracy control II or III mode is ON. A path can be smoothed by compensating commanded positions of a machining program.

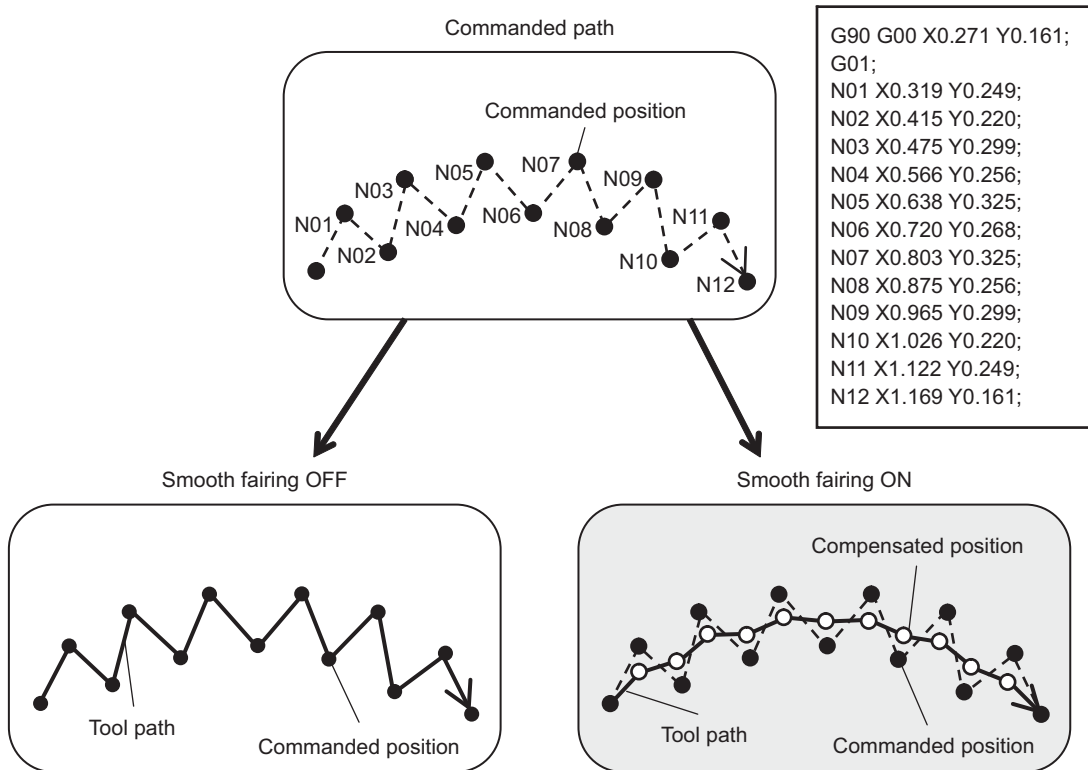
This function is useful when executing a fine segment program to machine smoothly at low speed or a rough machining program with long segment to machine smoothly.

This function is enabled while high-speed high-accuracy control II or III is ON or while high-accuracy control is ON in high-speed machining mode II, and performs compensation on consecutive G01 command during this time.

The validity of this function depends on the MTB specifications. To use this function, the high-speed high-accuracy control II or III specification, or the high-speed machining mode II and high-accuracy control specifications are required.

Note

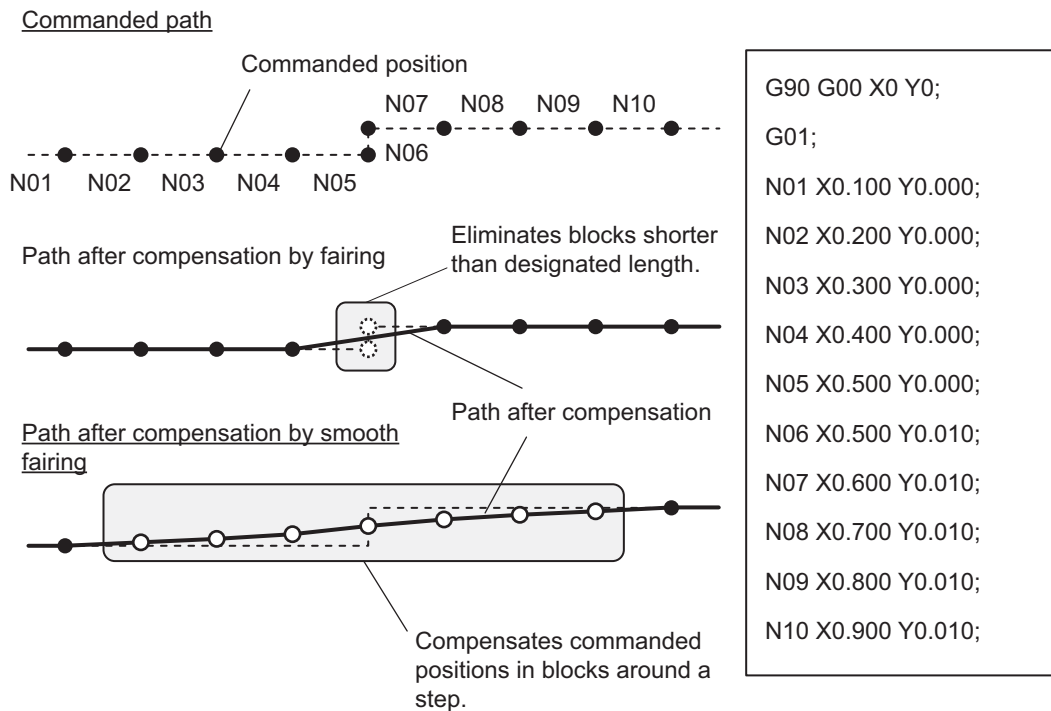
(1) High-speed high-accuracy control III functions as high-speed high-accuracy control II while smooth fairing is ON.



Fairing and smooth fairing differ as follows:

	Fairing	Smooth fairing
Operation	- Eliminating blocks shorter than designated length	- Compensating commanded positions across multiple blocks
Usage	- Eliminating minute steps to occur at fillet and other sections - Eliminating noises on commanded paths	- Smooth machining at low speed for a fine segment program - Smooth machining for a rough machining program

When a minute step exists on a commanded path, for instance, the path after compensation differs between fairing and smooth fairing as follows:



Refer to "Relationship with other functions" for the relationship between smooth fairing and other functions.



Detailed description

Enabling conditions

To enable smooth fairing, it is necessary for the following conditions to be satisfied respectively:

- (a) The smooth fairing specification is set to ON.
- (b) One of the following modes is set to ON.
 - G05 P20000 (*1)
 - G05 P10000
 - G05 P2 and the high-accuracy function (G61.1/G08P1 or G61.2) are used simultaneously.
- (c) At least one of the following conditions is satisfied.
 - The parameter "#8033 Fairing ON" is set to "2".
 - The G05 P20000, R1/G05 P10000, R1/G05 P2, or R1 command is issued.

(*1) This command functions as G05 P10000 while smooth fairing is ON.

Enabling smooth fairing

Two methods are available to enable smooth fairing: "G05 Pp,Rr command" and parameter "#8033 Fairing ON" ((c) of the enabling conditions).

Relationship between ",R" address and parameter "#8033 Fairing ON"

		Parameter "#8033 Fairing ON"		
		0 Both OFF	1 Fairing ON	2 Smooth fairing ON
G05 P20000	No ,R	x	o	•
G05 P10000	,R0	x	x	x
G05 P2 Command	,R1	•	•	•

•: Smooth fairing ON, o: Fairing ON, X: Both OFF

(1) When the ",R" address is set to the G05 command, the operation shown in the table below is performed regardless of the setting value of the parameter "#8033".

Smooth fairing ON	G05 P20000, R1 G05 P10000, R1 G05 P2, R1	Smooth fairing is ON regardless of the setting value of the parameter "#8033".
	G05 P20000, R0 G05 P10000, R0 G05 P2, R0	Both fairing and smooth fairing are OFF regardless of the setting value of the parameter "#8033".
	G05 P0,Rr (r=0,1) G05 P1,Rr (r=0,1)	Program error (P33)
Smooth fairing OFF	G05 P20000,Rr (r=0,1) G05 P10000,Rr (r=0,1) G05 P2,Rr (r=0,1) G05 P1,Rr (r=0,1) G05 P0,Rr (r=0,1)	Program error (P39)

(2) The ",R" address is unmodal information. The ",R" address value designated by previous G05 command is not inherited to the next and subsequent G05 commands. Each time the G05 command is issued, the fairing function is switched as shown in the table above.

Machining program	Operation
N01 G05 P10000, R1;	
...	In this period, the program runs with G05 P10000, R1.
N02 G05 P0; N03 G05 P10000;	The ",R" address of the N01 G05 command is not inherited.
...	In this period, the program runs with G05 P10000 (without the ",R" address).
N04 G05 P0;	

(3) To switch smooth fairing and fairing, insert the G05P0; command between them. If this switching is commanded without inserting the G05P0 command, a program error (P560) will occur.

Machining program	Operation
N01 G05 P10000, R1;	Set the parameter "#8033" to "1".
...	In this period, the program runs with smooth fairing.
N03 G05 P10000;	Issuing this command switches to fairing, which causes an error.

(4) To enable smooth fairing without setting the "R" address to the G05 command, set the parameter "#8033 Fairing ON" to "2". The following operation is performed.

G05 P20000 G05 P10000 G05 P2	Smooth fairing ON
G05 P1 G05 P0	Both fairing and smooth fairing are OFF



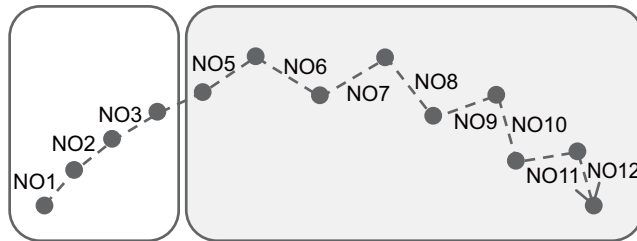
Details of Operation

An operation example is as follows.

(In this figure, symbol ○ indicates the compensated position, and symbol ● indicates the commanded position.)

(1) Smooth fairing smoothens the path by compensating the positions designated by successive G01 commands. This function recognizes the paths before and after each commanded position, and compensates commanded positions that cause a path to become unsmooth.

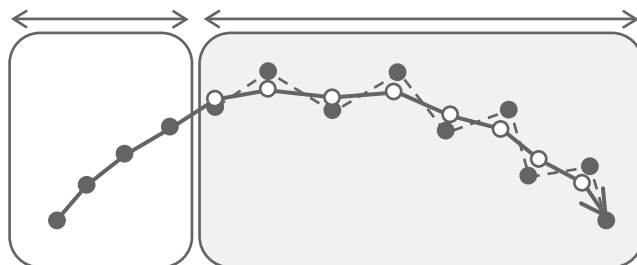
[Commanded path] (indicated by dashed lines)



```
G90 G00 X0.322 Y0.234;
G01;
N01 X0.413 Y0.276;
N02 X0.507 Y0.311;
N03 X0.603 Y0.338;
N04 X0.701 Y0.357;
N05 X0.798 Y0.399;
N06 X0.900 Y0.343;
N07 X1.003 Y0.399;
N08 X1.095 Y0.328;
N09 X1.205 Y0.367;
N10 X1.284 Y0.282;
N11 X1.399 Y0.304;
N12 X1.465 Y0.207;
```

[Path after compensation] (indicated by solid lines)

The smooth parts are not targeted for compensation.	Only the unsmooth parts are compensated.
---	--

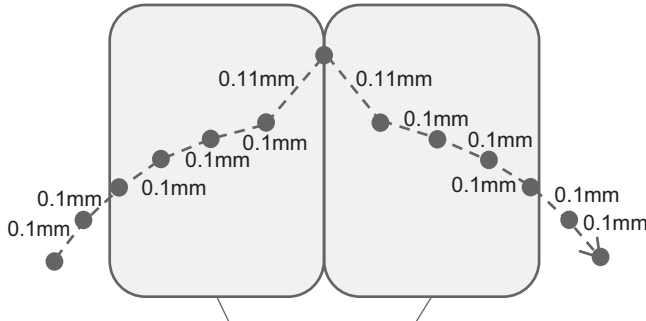


○: Compensated position

●: Commanded position

- (2) The path recognition range is determined by the parameter "#8038 Path recog. range". Determine the setting value to include multiple G01 commands in the path recognition range. When the setting value is "0", the range is set to "1.0 (1 mm)".

When the path recognition range is set to 0.5 mm:

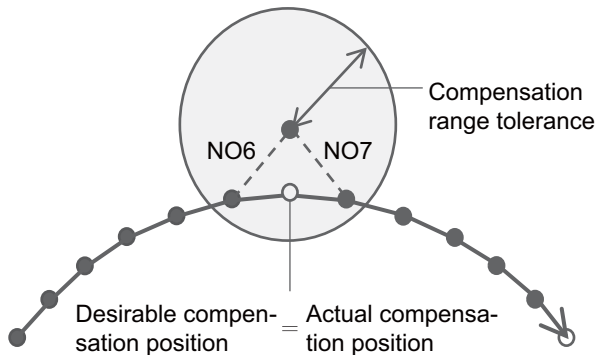


The path is recognized in the range of 0.5 mm forward and 0.5 mm backward of the commanded position.

```
G90 G00 X0.322 Y0.234;
G01;
N01 X0.413 Y0.276;
N02 X0.507 Y0.311;
N03 X0.603 Y0.338;
N04 X0.701 Y0.357;
N05 X0.800 Y0.369;
N06 X0.900 Y0.423;
N07 X1.000 Y0.369;
N08 X1.099 Y0.357;
N09 X1.198 Y0.338;
N10 X1.294 Y0.311;
N11 X1.388 Y0.276;
N12 X1.478 Y0.234;
```

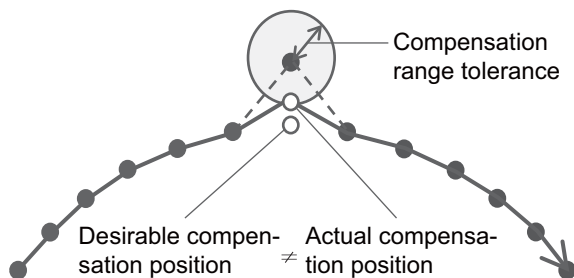
- (3) The upper limit of the compensation distance can be determined so that the compensated position does not deviate from the commanded position significantly. Designate this upper limit in the parameter "#8039 Comp. range limit". Ordinarily, designate the tolerance that is designated when generating the machining program with CAM. When the setting value is "0", the range is set to "0.005 (5 microns)".

(a) When the compensation distance tolerance is high:



```
G90 G00 X0.322 Y0.234;
G01;
N01 X0.413 Y0.276;
N02 X0.507 Y0.311;
N03 X0.603 Y0.338;
N04 X0.701 Y0.357;
N05 X0.800 Y0.369;
N06 X0.900 Y0.423;
N07 X1.000 Y0.369;
N08 X1.099 Y0.357;
N09 X1.198 Y0.338;
N10 X1.294 Y0.311;
N11 X1.388 Y0.276;
N12 X1.478 Y0.234;
```

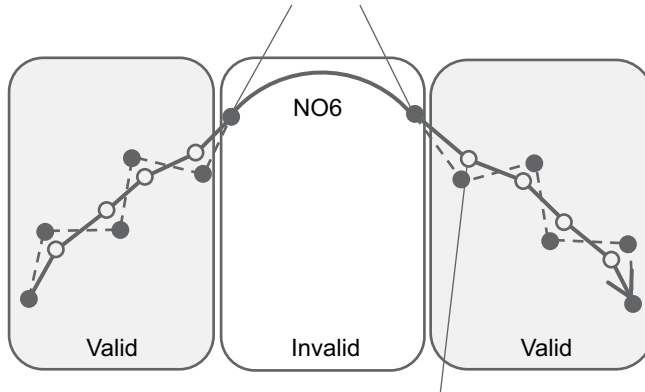
(b) When the compensation distance tolerance is low:



```
G90 G00 X0.322 Y0.234;
G01;
N01 X0.413 Y0.276;
N02 X0.507 Y0.311;
N03 X0.603 Y0.338;
N04 X0.701 Y0.357;
N05 X0.800 Y0.369;
N06 X0.900 Y0.423;
N07 X1.000 Y0.369;
N08 X1.099 Y0.357;
N09 X1.198 Y0.338;
N10 X1.294 Y0.311;
N11 X1.388 Y0.276;
N12 X1.478 Y0.234;
```

- (4) While smooth fairing is ON, the modal or mode status is changed, and smooth fairing may be set to OFF. While smooth fairing is OFF, the commanded position is not compensated, and the axis moves as commanded. For details on the modal or mode status that causes smooth fairing to be set to OFF, refer to "Relationship with other functions".

While smooth fairing is OFF, the axis moves to the commanded position.

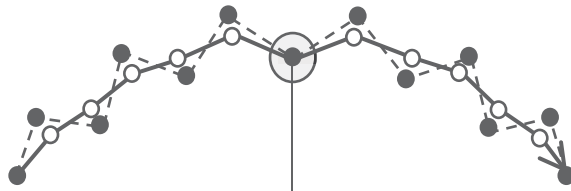


```
G90 G00 X0.0 Y0.0;
G01;
N01 G01 X0.039 Y0.077;
N02 G01 X0.139 Y0.080;
N03 G01 X0.172 Y0.174;
N04 G01 X0.271 Y0.161;
N05 G01 X0.319 Y0.249;
N06 G02 X1.122 Y0.249 R0.5;
N07 G01 X1.169 Y0.161;
N08 G01 X1.268 Y0.174;
N09 G01 X1.301 Y0.080;
N10 G01 X1.401 Y0.077;
N11 G01 X1.441 Y0.000;
```

Compensation restarts from the block in which the enabling conditions are satisfied again.

- (5) While smooth fairing is ON, it may be canceled temporarily depending on commands when:
- there is a block that contains only a sequence number;
 - the modal status of the absolute value/incremental value command is changed by the G90 or G91 command;
- and
- the movement command is issued to an axis other than the three basic axes.

If a command that triggers a temporary cancel is inserted, the axis moves to the commanded position once. For the list of commands that trigger a temporary cancel, refer to "Relationship with other functions".



If a block that triggers a temporary cancel is inserted, the axis moves to the commanded position once.

```
G90 G00 X0.322 Y0.234;
G90 G01;
N01 X0.413 Y0.276;
N02 X0.507 Y0.311;
N03 X0.603 Y0.338;
N04 X0.701 Y0.357;
N05 X0.798 Y0.399;
N06 X0.900 Y0.343;
N07;
N08 X1.003 Y0.399;
N09 X1.095 Y0.328;
N10 X1.205 Y0.367;
N11 X1.284 Y0.282;
N12 X1.399 Y0.304;
N13 X1.465 Y0.207;
```



Relationship with other functions

(1) Relationship between smooth fairing and other G code functions

A	Shows if smooth fairing is valid or not when the G code function on the left is enabled. ○ (valid): Compensates for the commanded position. × (invalid): Does not compensate the commanded position.
B	Shows operation when the G code on the left is commanded together with a movement command (XYZ address command)(*) while smooth fairing is ON. ○ (continuation): Compensates for the commanded position. × (temporary cancel): Temporarily suspends compensation to move to the commanded position. Temporary cancel for blocks with no movement commands (example: When G90; is commanded independently).

G code group	G code	Function name	A	B
0	G05	High-speed machining mode/high-speed high-accuracy control	(*1)	-
	G08	High-accuracy control G command in group 0 except the above	○ -	× ×
1	G01	Linear interpolation	○	○
		G command in group 1 except the above	×	×
2	G17/G18/G19	Plane selection	○	(*2)
3	G90/G91	Absolute value command/incremental value command	○	(*2)
4	G23	Stroke check before travel OFF	○	×
		G command in group 4 except the above	×	×
5	G94	Asynchronous feed (feed per minute)	○	○
		G command in group 5 except the above	×	×
6	G20/G21	Inch/Metric command	○	×
7	G40	Tool radius compensation cancel/3-dimensional tool radius compensation cancel	○	○
		G41/G42	Tool radius compensation/3-dimensional tool radius compensation	○
		G command in group 7 except the above	×	×
8	G43/G44	Tool length offset + /tool length offset -	○	×
	G43.1	Tool length compensation along the tool axis	○	×
	G49	Tool length offset cancel G command in group 8 except the above	○ ×	× ×
9	G80	Fixed cycle cancel	○	×
		G command in group 9 except the above	×	×
10	G98/G99	Fixed cycle initial level return/R point level return	○	×
11	G50	Scaling cancel	○	×
		G command in group 11 except the above	×	×
12	G54-G59/G54.1	Workpiece coordinate system selection	○	×
13	G61.1	High-accuracy control ON	○	×
		G61.2	High-accuracy spline	○
		G command in group 13 except the above	×	×
14	G67	User macro modal call cancel	○	×
		G command in group 14 except the above	×	×
15	G40.1/G150	Normal line control cancel	○	×
		G command in group 15 except the above	×	×

G code group	G code	Function name	A	B
16	G69	Coordinate rotation cancel/3-dimensional coordinate conversion cancel	○	×
		G command in group 16 except the above	×	×
17	G96/G97	Constant surface speed control ON/OFF	○	×
18	G15	Polar coordinate command OFF	○	×
		G command in group 18 except the above	×	×
19	G50.1	Mirror image by G code OFF	○	○
		G command in group 19 except the above	×	×
21	G13.1/G113	Cylindrical interpolation/polar coordinate interpolation OFF	○	○
		G command in group 21 except the above	×	×
27	G54.4	Workpiece installation error compensation	(*3)	×

(*1) ○ (valid) for G05P2/G05P10000/G05P20000 and X (invalid) for the others.

(*2) ○ (continuation) if the modal state does not change before and after the command and X (temporary cancel) otherwise.

(*3) ○ (Valid) for G54.4P0 and X (invalid) for the others.

(2) Relationship between smooth fairing and functions other than G codes

A	Shows if smooth fairing is valid or not when the function on the left is enabled. ○ (valid): Compensates for the commanded position. × (invalid): Does not compensate the commanded position.
B	Shows operation when the function on the left is commanded while smooth fairing is ON. ○ (continuation): Compensates for the commanded position. × (temporary cancel): Temporarily suspends compensation to move to the commanded position.

Function other than G code	A	B
Block containing only EOB(;))	-	(*1)
Block containing only comment	-	○
Block containing only sequence number	-	×
Block containing only MSTB command	-	×
Block containing only F command	-	×
If there is an axis movement command for other than three base axes	-	×
Block without movement command	-	×
During single block operation	×	×
Subprogram call (M98 P_)	○	×
Figure rotation subprogram call (M98 P_I_J_K_)	×	×
Macro interruption (M96, UIT)	○	×
User macro simple call	○	×
User macro modal call	×	×
MTB macro	×	×
PLC interruption (PIT)	(*2)	
Coordinate rotation by parameter (G10 I_J_/K_)	×	×
Mirror image by parameter setting (#8211 Mirror image)	×	×
Mirror image with PLC signals ON	×	×

(*1) When there is a block containing only EOB, compensation is not temporarily canceled. However, in such a case, the path slightly changes compared to when there are no blocks containing only EOB.

(*2) PLC interruption is not allowed during high-speed high-accuracy control II/III.

The table below shows which fairing functions are enabled according to the combination of the parameter "#8033 Fairing ON" setting and G command:

		#8033 Fairing ON		
		0 Both OFF	1 Fairing ON	2 Smooth fairing ON
G05 P0	G61.1	x	x	x
	G61.2	o	o	o
G05 P2	G61.1	x	x	●
	G61.2	o	o	●
G05 P10000	G5.1 Q0	x	o	●
G05 P20000	G5.1 Q2	x	x	●

●: Smooth fairing ON, o: Fairing ON, X: Both OFF

17.3.4 Acceleration Clamp Speed

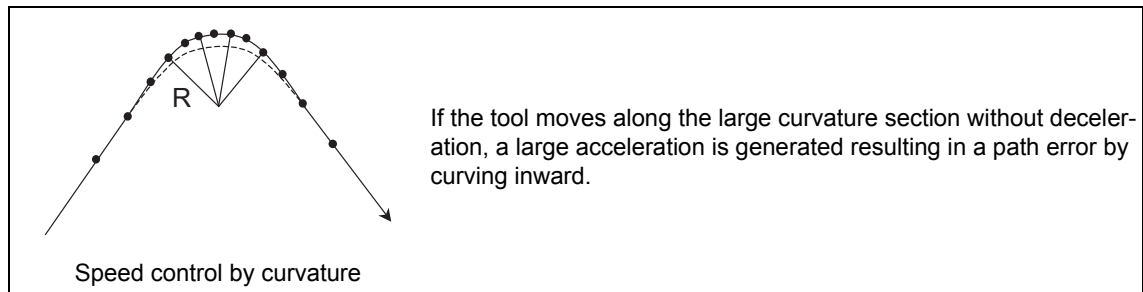
**Function and purpose**

This function is an additional function when the high-speed high-accuracy control II mode is ON.

The cutting feed clamp speed during the high-speed high-accuracy control II / III mode, when the following parameter is set to "1", is clamped so that the acceleration generated by each block movement does not exceed the tolerable value. This function clamps the speed optimally even at a section where "angle change at each block is small but entire curvature is large" such as shown below.

The tolerable acceleration value is calculated from the parameter "#1206 G1bF" and "#1207 G1btL" setting values. (Tolerable acceleration = #1206/#1207)

Related parameter		Details
#8034	AccClamp ON	0 : Clamp the cutting speed with parameter "#2002 clamp" (*1) or the corner deceleration function. 1: Cutting speed clamp determined by acceleration reference is also executed.



(*1) When a speed is set in "#2109 Clamp(H-precision)", clamp is executed at that speed. When the setting value is "0", clamp is executed with "#2002 clamp".

17.3.5 High-speed Mode Corner Deceleration

**Function and purpose**

This function is an additional function when high-speed high-accuracy control II mode is ON.

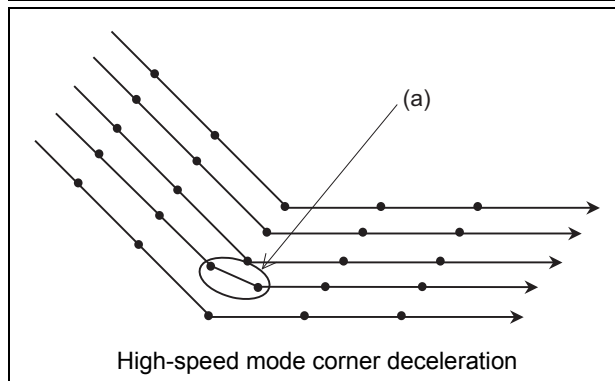
During high-accuracy control, if the angle between the adjacent blocks in the machining program is large, this function, conventionally, automatically decelerates the machining so that the acceleration generated when passing through the corner is maintained within the tolerable value.

If a fine block is inserted at the corner section in the machining program generated with the CAM, etc., the corner passing speed will not match the periphery. This can affect the machining surface.

In the corner deceleration in the high-speed mode, even when this type of fine block is inserted, the corner will be judged from a vantage point by setting the below parameter.

The fine block is excluded at the judgment of an angle, but is not excluded from the actual movement command.

Related parameter		Details
#8036	CordecJudge	0 : Judge the corner from the angle of the neighboring block. 1 : Judge the corner from the angle of the neighboring block, excluding the minute blocks.
#8027	CorJudgeL	Exclude shorter block than this setting value.



(a) When "#8036 CordecJudge" is set to "1", corner deceleration is realized without an influence of fine blocks.

17.3.6 Precautions on High-speed High-accuracy Control



Precautions

Common precautions on high-speed high-accuracy control I/II/III

- (1) The validity of each high-speed high-accuracy control function depends on the MTB specifications. If any of the above is commanded when the corresponding specification is not available on the machine, a program error (P39) will occur.
- (2) The machining speed may drop depending on the number of characters in one block.
- (3) Feedrate command F is clamped with the "#2110 Clamp (H-precision)" (Cutting feed clamp speed for high-accuracy control mode) set with the parameter.
- (4) The rapid traverse rate conforms to "#2109 Rapid(H-precision)" (Rapid traverse rate during high-accuracy control mode) set with the parameter.
- (5) When "#2109 Rapid(H-precision)" (high-accuracy control mode rapid traverse rate) is set to "0", however, the movement follows "#2001 rapid" (Rapid traverse rate) set with the parameter. Also, when "#2110 Clamp (H-precision)" (Cutting feed clamp speed for high-accuracy control mode) is set to "0", the speed will be clamped with "#2002 clamp" (Cutting clamp speed) set with parameter.
- (6) The automatic operation processing has priority in the high-speed high-accuracy control I/II/III modal, so the screen display, etc., may be delayed.
- (7) The speed will decelerate once at the high-speed high-accuracy control I command (G05.1 Q1), high-speed high-accuracy control I OFF command (G05.1 Q0), high-speed high-accuracy control II command (G05P10000), high-speed high-accuracy control III command (G05P20000), and high-speed high-accuracy control II/III OFF command (G05P0), so turn ON and OFF when the tool separates from the workpiece.
- (8) When carrying out high-speed high-accuracy control I/II operation during tape mode, the machining speed may be suppressed depending on the program transmission speed and the number of characters in one block.
- (9) If "#1205 G0bdcc" (G0 acceleration/deceleration before interpolation) is set to "1", the value set with the parameter "#2224 SV024" (in-position detection width) will be used as the in-position width. "#2077 G0inps" (G0 in-position width) and the ",I" command (programmable in-position check) are disabled.

Common precautions on high-speed high-accuracy control II/III

- (1) If the variable command, variable operation command, or macro control statement is commanded while high-speed high-accuracy control II/III is enabled, the fine segment processing capacity is not guaranteed. However, only when the variable commands and variable four-basic-arithmetic operation commands shown below are issued following the axis address or the F address of the cutting feedrate command, the fine segment processing capacity is guaranteed.
 - (a) Referencing common variables or local variables
Common variables or local variables can be referenced (example: X#500, Y#1, Z##100, A#[#101], etc.).
 - (b) Four basic arithmetic rule
Four basic arithmetic rule (+, -, *, /) operations are available, and also the operation priority can be designated using parentheses () ([#500 + #501]*#502, etc.).

If a common variable or local variable is referenced using the variable number operated with a macro operation instruction, a program error (P232) may occur. In this case, set the operated value to the variable before referencing the variable.

Example that causes an error	F#[FIX[100.1]];
Example that does not cause an error	#500 = FIX[100.1] ; F#[#500] ;

Precautions on high-speed high-accuracy control I

- (1) Command G05.1Q0 ; after turning the tool radius compensation OFF. If G05.1Q0; is commanded without turning the tool radius compensation OFF, a program error (P29) will occur.
- (2) G05.1Q1; and G05.1Q0; are independent commands. If a sequence number other than "N" is commanded, the program error (P33) will occur.
- (3) The program error (P33) will occur if the G05.1 command block does not contain a Q command.
- (4) If the high-speed high-accuracy control I command is issued in the high-speed high-accuracy control II modal, a program error (P34) will occur.

Precautions on high-speed high-accuracy control II

- (1) G05P10000; and G05P0; are independent commands. If a sequence number other than "N" is commanded, the program error (P33) will occur.
- (2) The program error (P33) will occur if the G05 command block does not contain a P command.
- (3) Fairing function is valid for the continuous linear command (G01). Fairing is not possible in the case below.



- (4) In a single block mode, operation stops at the end point of each block.
- (5) When using the high-speed high-accuracy control II mode, set parameter "#1572 Cirorp/Bit0" to "1" to eliminate the speed fluctuation at the seams between the arc and the straight line, or between arcs.
- (6) A program error (P33) will occur if the geometric command is issued during the high-speed high-accuracy control II.
- (7) If the high-speed high-accuracy control II command is issued in the high-speed high-accuracy control I modal, a program error (P34) will occur.

Precautions on high-speed high-accuracy control III

- (1) If high-speed high-accuracy control III is commanded while its specifications are invalid, a program error (P39) will occur.
- (2) G05P20000; and G05P0; are independent commands. If a sequence number other than "N" is commanded, the program error (P33) will occur.
- (3) The program error (P33) will occur if the G05 command block does not contain a P command.
- (4) A program error (P33) will occur if the geometric command is issued during the high-speed high-accuracy control III.
- (5) If the high-speed high-accuracy control III command is issued in the high-speed high-accuracy control I modal, a program error (P34) will occur.
- (6) If the high-speed high-accuracy control II mode is valid when high-speed high-accuracy control III is commanded, follow the precautions on high-speed high-accuracy control II.
- (7) High-speed high-accuracy control III can be enabled by commanding the G code from the machining program.

- (a) High-speed high-accuracy control III command with the high-speed high-accuracy control III enable conditions satisfied

If all modal conditions in each G code group and each mode condition shown in "Relationship with other functions" (*1) are satisfied when G05P20000; is commanded, the high-speed high-accuracy control III mode is enabled, and "G05P20000" is displayed on the modal screen. If conditions are not satisfied after G05P20000; has been commanded, the high-speed high-accuracy control III mode is enabled, but the fine segment capacity is not guaranteed.

Machining program		Enable conditions for high-speed high-accuracy control III	Enable mode
G05 P20000;	High-speed high-accuracy control III command	Enabling conditions are satisfied.	G05P20000
G41 XxYyDd;	Tool radius compensation ON	Enabling conditions are not satisfied.	G05P20000 (*2)
G40 XxYy;	Tool radius compensation OFF	Enabling conditions are satisfied.	G05P20000

(*1) For details, refer to "Relationship with other functions" in "17.3.1 High-speed High-accuracy Control I, II, III ; G05.1 Q1/Q0, G05 P10000/P0, G05 P20000/P0".

(*2) High-speed high-accuracy control III is enabled, but the fine segment capacity shown in "Fine segment capacity for multi-part system" is not guaranteed.

- (b) High-speed high-accuracy control III command with no high-speed high-accuracy control III enabling conditions satisfied

If the conditions shown in "Relationship with other functions" (*1) are not satisfied when G05P20000; is commanded, the high-speed high-accuracy control II mode is enabled, and "G05P10000" is displayed on the modal screen. In this case, even if all the conditions shown in "Relationship with other functions" (*1) are satisfied after G05P20000; has been commanded, the high-speed high-accuracy control III mode is not enabled. To enable the high-speed high-accuracy control III mode, command G05P20000; again.

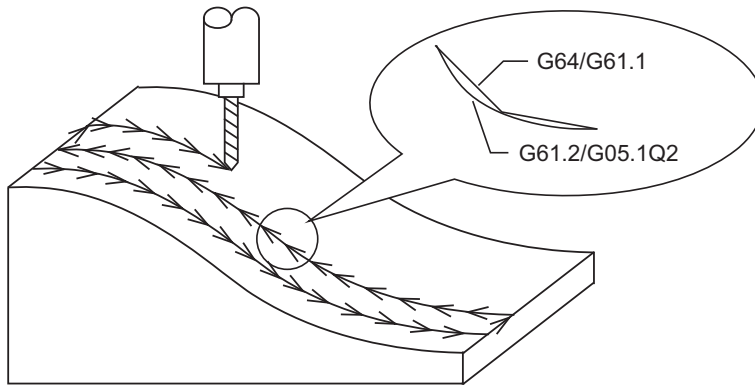
Machining program		Enable conditions for high-speed high-accuracy control III	Enable mode
G41 XxYyDd;	Tool radius compensation ON	Enabling conditions are not satisfied.	G05P10000
G05 P20000;	High-speed high-accuracy control III command		
G40 XxYy;	Tool radius compensation OFF	Enabling conditions are satisfied.	G05P10000

17.4 Spline Interpolation ; G05.1 Q2/Q0



Function and purpose

This function automatically generates a spline curve that passes through a sequence of points commanded by the fine segment machining program, and interpolates the path along this curve. This enables high-speed and high-accuracy machining to be achieved.



There are two types of spline interpolation command format: G61.2 and G05.1Q2. Both formats can be used regardless of the parameter "#1267 ext03/bit0" setting if the spline interpolation specifications are available to the machine. This section describes the G05.1Q2 command. For G61.2, refer to "17.6 High-accuracy Spline Interpolation ; G61.2".

The G05.1Q2 command can be issued when the machining parameter "#8025 SPLINE ON" is set to "1" in the high-speed high-accuracy control function II or III mode. The following explanation is limited to the spline function in the high-speed high-accuracy control function II or III mode.

Note

(1) High-speed high-accuracy control III functions as high-speed high-accuracy control II while spline interpolation is ON.

Difference between G61.2 and G05.1Q2

Conditions under which the command can be issued and functions that are valid during a specific modal differ between G61.2 and G05.1Q2.

Command format	Conditions under which the command can be issued	Functions that become valid		
		Spline interpolation (*2)	Fairing (*3)	High-accuracy control (*4)
G61.2	None	Valid	Valid	Valid
G05.1 Q2	When the system is in the high-speed high-accuracy control II or III mode and "#8025 SPLINE ON" is set to "1" (*1)	Valid	Can be turned ON and OFF using "#8033 Fairing ON"	Valid (Because the system is in the high-speed high-accuracy control II or III mode)

(*1) The validity of the high-speed high-accuracy control II or III function depends on the MTB specifications. A program error (P34) will occur if the conditions under which the command can be issued are not satisfied.

(*2) The spline interpolation smoothly connects a sequence of points commanded by program. As a result, the glossy machining surface can be obtained, and the machining time can be reduced because the frequency of the corner deceleration decreases compared with conventional linear interpolation.

- (*3) Super-fine blocks often included in the data generated with CAM are deleted. Such a super-fine block may scratch the machining surface, and increase machining time because of acceleration/deceleration. This function prevents these problems.
- (*4) The following shows the functions and their operations included in the high-accuracy control described in this section.

Functions of high-accuracy control	Contents
Acceleration/deceleration before interpolation (Constant inclination acceleration/deceleration, S-pattern filter)	The process is the same as that performed in the high-accuracy control mode (G61.1/G08P1).
Optimum corner deceleration	As is done in the high-accuracy control mode (G61.1/G08P1), optimum corner deceleration is performed at points where the angle between blocks exceeds the spline cancel angle or points at the boundary between G01 and G00, because spline interpolation is temporarily canceled to make corners.
Arc speed clamp (For spline interpolation, curvature speed clamp)	Clamp speed is calculated based on the spline curvature radius. The process for arc blocks is the same as that performed in the high-accuracy control mode (G61.1/G08P1).
Curvature radius speed clamp	Clamp speed is calculated based on the spline curvature radius.
Arc entrance/exit deceleration control	The process for arc blocks is the same as that performed in the high-accuracy control mode (G61.1/G08P1).
SSS Control	Optimum speed control is performed so that the process is not affected by steps or reverse runs.
Feed forward control	The process is the same as that performed in the high-accuracy control mode (G61.1/G08P1).

The validity of the SSS control function depends on the MTB specifications.



Command format

Spline interpolation mode ON

G05.1 Q2 X0 Y0 Z0 ;

Spline interpolation mode OFF

G05.1 Q0;



Detailed description

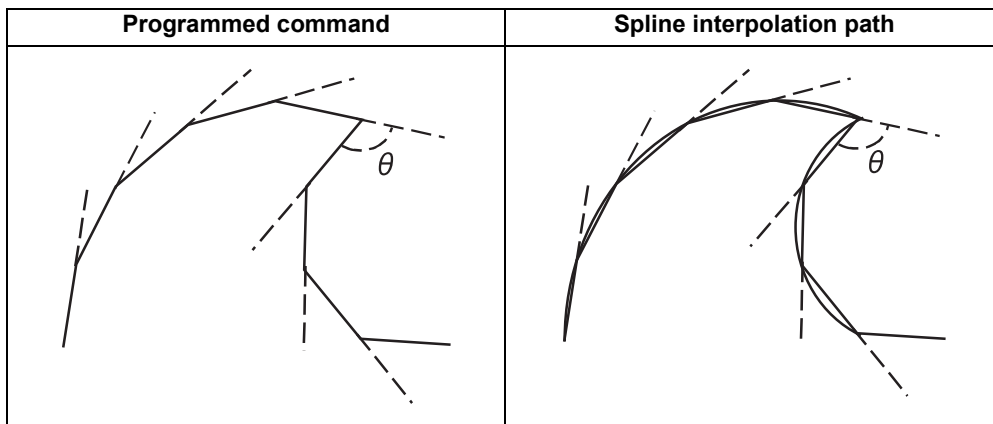
Temporary cancellation of spline interpolation

Normally, once the spline function is activated, one curve is generated by smoothly connecting all points until it is canceled. However, if a corner edge should be created, or if the segment length is long and spline interpolation should not to be carried out, the function can be canceled temporarily with the parameters.

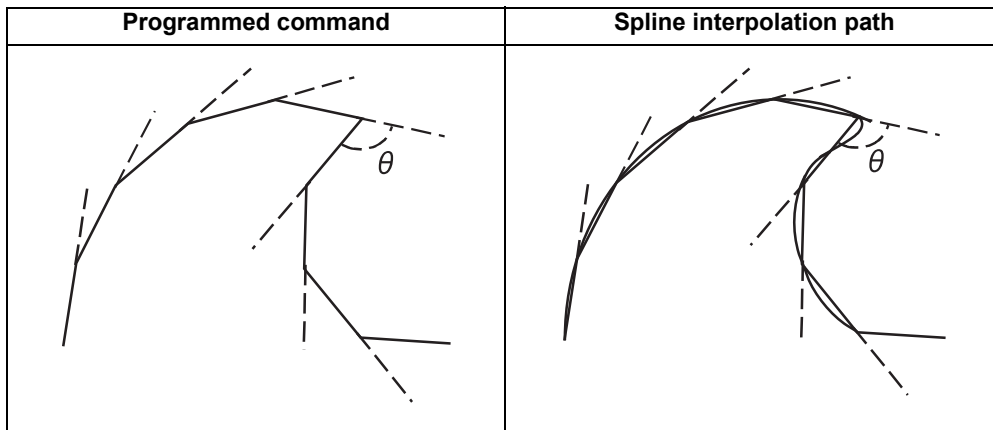
(1) Cancel angle

If the angle θ of two consecutive blocks exceeds the value set in parameter "#8026 CANCEL ANG.", the spline function will be temporarily canceled, and optimum corner deceleration will be applied. When this parameter is not set (=0), the spline interpolation will be constantly applied. The corner deceleration angle of the high-accuracy control function is valid during the temporary cancellation, and the optimum corner deceleration will be applied.

(Example 1) Cancel angle = 60°



(Example 2) Cancel angle = 0°



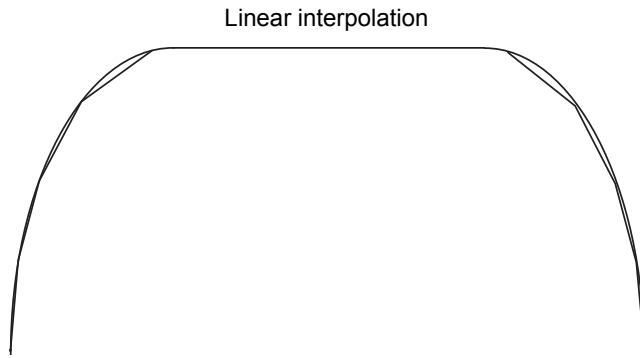
<Note>

- ◆If the section to be a corner is smooth when actual machining is carried out, lower the "CANCEL ANG.". If a smooth section becomes a corner, increase the "CANCEL ANG.".
- ◆If "CANCEL ANG." \geq "DCC ANGLE", the axis will decelerate at all corners where the angle is larger than the "CANCEL ANG.".
- ◆If the "CANCEL ANG." $<$ "DCC ANGLE", corner deceleration will not be applied if the corner angle is equal to or less than "DCC ANGLE" even if the spline interpolation is canceled.

(2) Fine segment length

If the movement amount in a block is longer than the parameter "#8030 MINUTE LENG", the spline function will be temporarily canceled, and the linear interpolation will be executed. When this parameter is not set (= 0), the fine segment length will be 1mm.

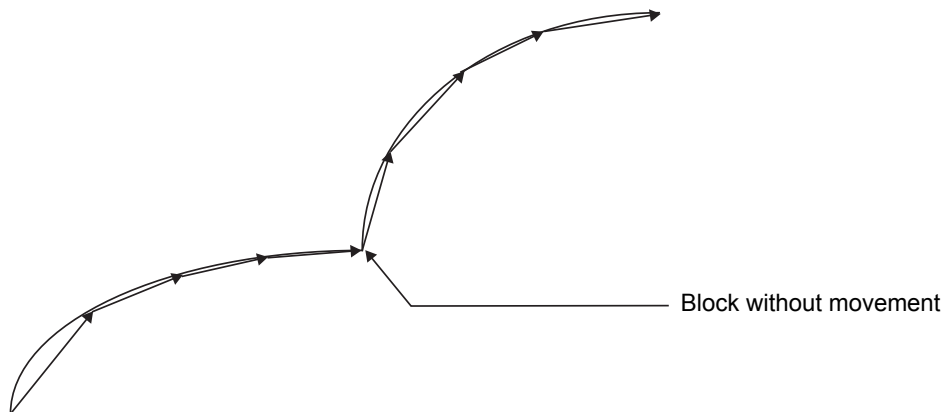
When the segment length in a block > fine segment length (#8030 MINUTE LENG), the linear interpolation will be executed.



If the fine segment length is set to "-1", the spline interpolation will not be canceled according to the block length.

(3) When a block without movement exists

If a block without movement exists during the spline function is operating, the spline interpolation will be canceled temporarily. Note that blocks containing only ";" will not be viewed as a block without movement.



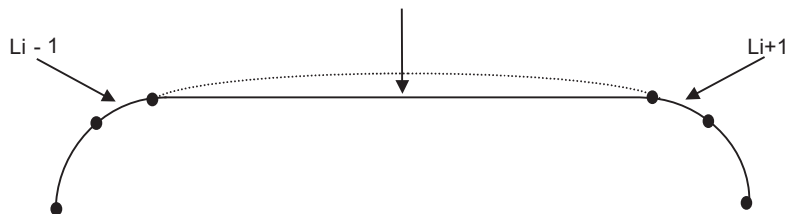
(4) When a block markedly longer than other blocks exists in spline function

Given that the i-th block length is L_i in the spline interpolation mode and if the following condition is met, the block will be interpreted as a linear section, and the spline interpolation mode will be temporarily canceled:

$$L_i > L_{i-1} \times 8 \text{ or } L_i > L_{i+1} \times 8$$

However, if the parameter "#8030 MINUTE LENG" is set to "-1", the mode will not be canceled.

$$L_i > L_{i-1} \times 8 \text{ or } L_i > L_{i+1} \times 8$$



Spline interpolation curve shape correction

Normally, once the spline function is enabled, one curve is generated by connecting all points smoothly until the function is canceled. But if the spline curve shape should be corrected, the spline curve shape can be corrected with the parameters.

(1) Chord error of block containing inflection point

When changing the CAD curve data into fine segments with the CAM, normally, the tolerance (chord error) of the curve is approximated in segments that are approx. 10µm. If there is an inflection point in the curve, the length of the block containing the inflection point may lengthen. (Because the tolerance is applied at both ends near the inflection point.) If the block lengths with this block and the previous and subsequent blocks are unbalanced, the spline curve in this block may have a large error in respect to the original curve.

At sections where the tolerance (chord error) of the fine segment block and spline curve in a block containing this type of inflection point, if the chord error in the corresponding section is larger than the value set in parameter (#8027 Toler-1), the spline curve shape is automatically corrected so that the error is within the designated value. However, if the maximum chord error of the corresponding section is more than five times larger than the parameter "#8027" setting value, the spline function will be temporarily canceled.

The curve is corrected only in the corresponding block.

The corrections are carried out under the following conditions for each block in the spline interpolation mode.

There is an inflection point in the spline curve, and the maximum error of the spline curve and linear block is larger than parameter "#8027".
 (Distance between P3-P4 in Fig. 1)

When the above conditions are satisfied, the spline curve will be corrected so that the error between P3-P4 in Fig. 2 is within the designated value.

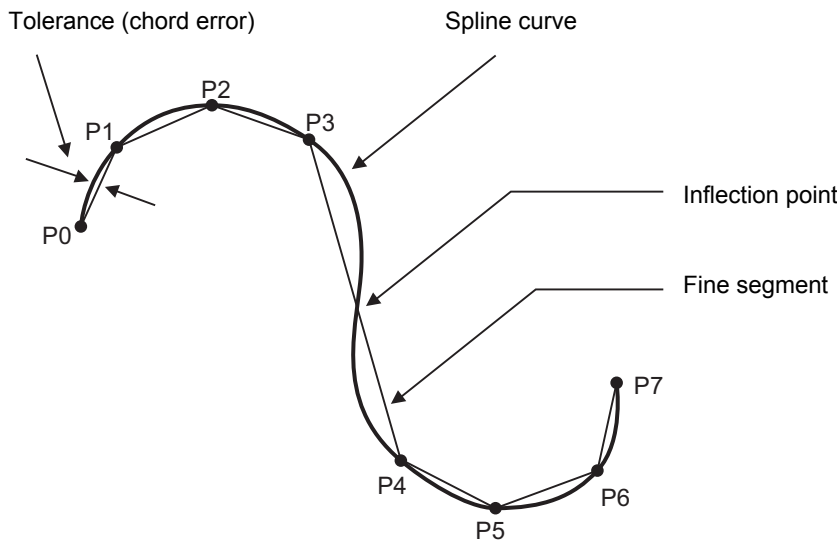


Fig. 1 Spline curve before error correction

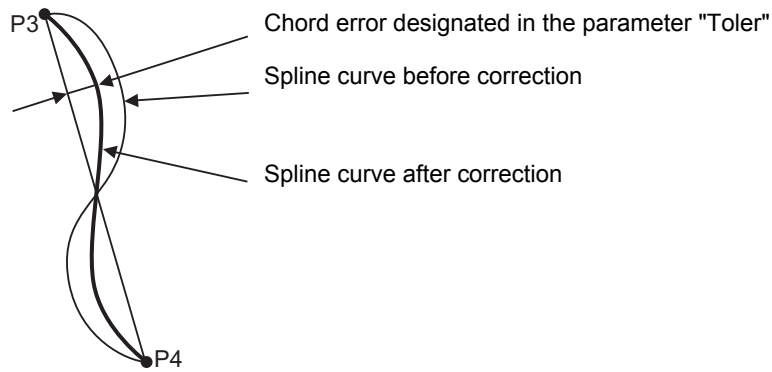


Fig. 2 Spline curve after error correction

In parameter "#8027 Toler-1", set the tolerance when developed into fine segments with the CAM. Set a smaller value if the expansion (indentation) is apparent due to the relationship with the adjacent cutting paths.

(2) Chord error of block not containing inflection point

Even in blocks that do not contain an inflection point, if the block lengths are not matched, the tolerance of the spline curve may increase. The curve may also expand due to the effect of relatively short blocks.

At sections where the tolerance (chord error) between the fine segment block and spline curve in a block without an inflection point becomes large, if the chord error in the corresponding section is larger than the value set in parameter (#8028 Toler-2), the spline curve shape is automatically corrected so that the error is within the designated value. However, if the maximum chord error of the corresponding section is more than five times larger than the parameter "#8028" setting value, the spline function will be temporarily canceled.

The curve is corrected only in the corresponding block.

The corrections are carried out under the following conditions for each block in the spline interpolation mode.

There is no inflection point in the spline curve, and the maximum error of the spline curve and linear block is larger than parameter "#8028".
(Distance between P2-P3 in Fig. 3)

When the above conditions are satisfied, the spline curve will be corrected so that the error between P2-P3 in Fig. 4 is within the designated value.

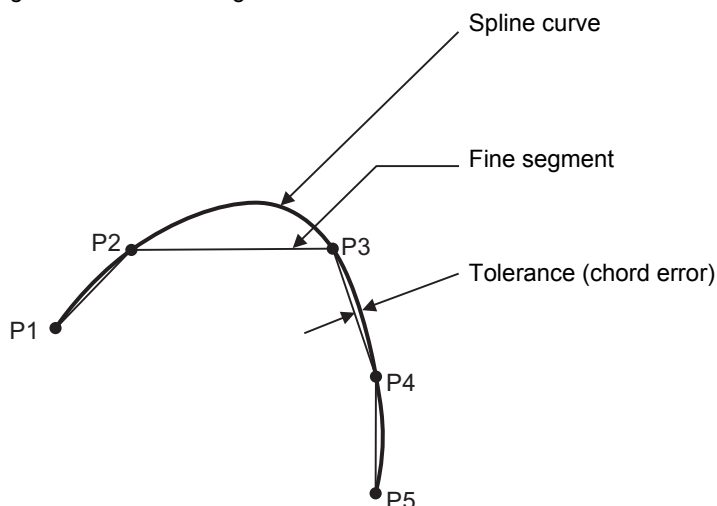


Fig. 3 Spline curve before error correction

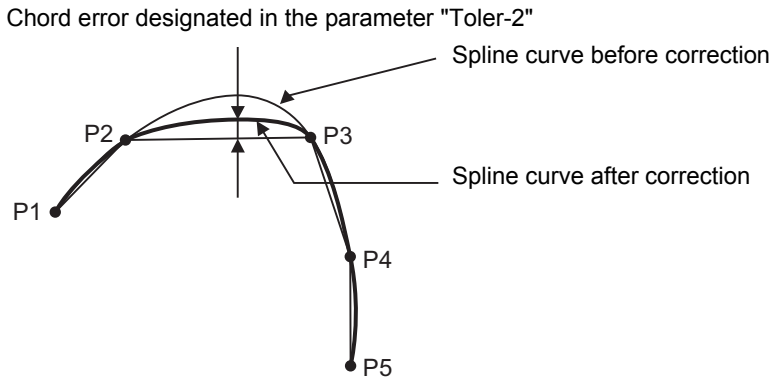
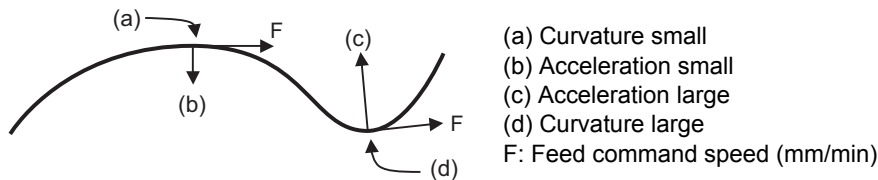


Fig. 4 Spline curve after error correction

In parameter "Toler-2", set the tolerance when developed into fine segments with the CAM.

Curvature speed clamp

The commanded speed F for the spline function during a segment linear arc will be the speed commanded in the previously set modal. However, if the axis is fed with the same speed, excessive acceleration may occur at the sections where the curvature is large (where curvature radius is small) as shown below. Thus, the speed clamp will be applied.

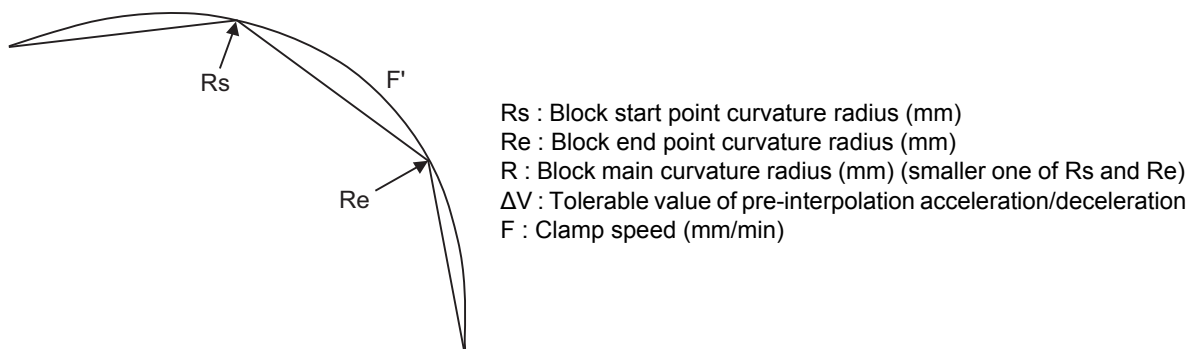


Acceleration and curvature

With the spline function, the high-accuracy control function is always valid. Thus, even if the curvature changes such as in this curve, the speed will be clamped so that the tolerable value for pre-interpolation acceleration/deceleration, which is calculated with the parameters, is not exceeded.

The clamp speed is set for each block, and the smaller of the curvature radius R_s at the curve block start point and the curvature radius R_e at the end point of the block will be used as the main curvature radius R . Using this main curvature radius R , the clamp speed F' will be calculated with expression (1).

The smaller of this clamp speed F' and the commanded speed F will be incorporated for the actual feedrate. This allows cutting with an adequate feedrate corresponding the curvature radius along the entire curve.



$$F' = \sqrt{R \times \Delta V \times 60 \times 1000} \times \frac{100 - K_s}{100} \dots (1)$$

$$\Delta V = \frac{G1bF(\text{mm/min})}{G1btL(\text{ms})}$$

- $G1bF$: Target pre-interpolation acceleration/deceleration
- $G1btL$: Acceleration/deceleration time to reach the target speed
- K_s : Accuracy coefficient



Program example

```

:
G91;
G05 P10000;           High-speed high-accuracy control function II mode ON
:
G05.1 Q2 X0 Y0 Z0;   Spline interpolation mode ON
G01 X1000 Z-300 F1000;
X1000 Z-200;
Y1000;
X-1000 Z-50;
X-1000 Z-300;
G05.1 Q0;           Spline interpolation mode OFF
:
G05 P0 ;           High-speed high-accuracy control function II mode OFF
:

```

- (1) The spline function carries out spline interpolation when the following conditions are all satisfied. If the following conditions are not satisfied, the spline function will be canceled once, and the judgment whether to carry out new spline from the next block will be made.
- ♦It is the movement only of three axes set to the basic axes I, J and K.
 - ♦When the block length is smaller than the value of the machining parameter "#8030 MINUTE LENGHS".
 - ♦When the movement amount is not 0.
 - ♦When one of the following modes is entered.
 - G01: Linear interpolation, G40: Tool compensation cancel, G64: Cutting mode,
 - G80: Fixed cycle cancel, G94: Feed per minute
 - ♦When only an axis commanded with G05.1Q2 is commanded.
 - ♦A single block is not being executed.
- (2) Graphic check will draw the shape of when the spline interpolation OFF.
- (3) During the spline function mode, the command to the axis must be issued after G05.1 Q2 in the same block. For example, if the X axis and Y axis are to be commanded in the spline function mode, command "G05.1 Q2 X0 Y0;". The command block containing an axis not designated with this command (G05.1 Q2 X0 Y0) in the spline function mode will carry out linear interpolation instead of spline interpolation.
- (4) If G05.1 Q2 is commanded when not in the high-speed high-accuracy control function II or III mode, the program error (P34) will occur.
- (5) If the machining parameter "#8025 SPLINE ON" is "0" in the high-speed high-accuracy control function II or III mode and G05.1 Q2 is commanded, the program error (P34) will occur.
- (6) Up to three axes set as the basic axes I, J and K can be commanded for the spline function.



Relationship with other functions

Refer to "Relationship with other functions" in "17.2 High-accuracy Control".



Precautions

- (1) If this function are not provided and "G05.1 Q2" is commanded, the program error (P39) will occur.
- (2) Even if "-1" is set for parameter "#8030 MINUTE LENGHS", the spline function will be temporarily canceled by the cancel conditions (cancel angle, non-movement block, excessive chord error, etc.) other than the block length.
- (3) Command "G05.1 Q2" and "G05.1 Q0" commands in independent blocks.
 - A program error (P33) will occur if not commanded in independent blocks.
- (4) The program error (P33) will occur if the G05.1 command block does not contain a Q command.
- (5) A program error (P34) will occur if the number of axis in the part system does not exceed 3.

17.5 Spline Interpolation 2; G61.4

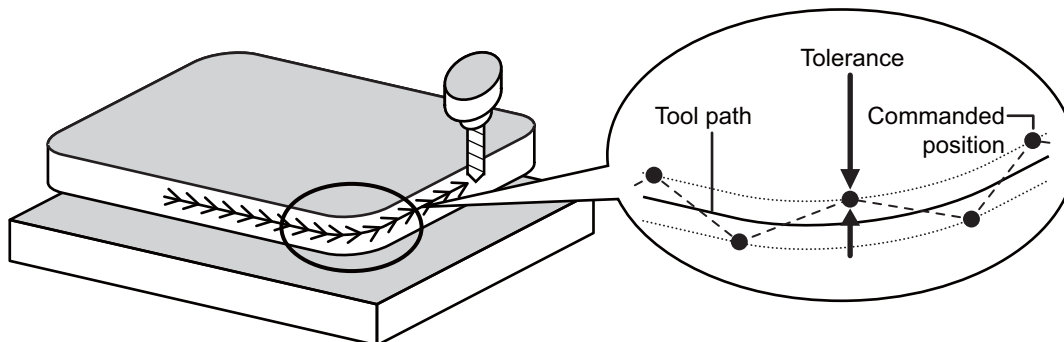


Function and purpose

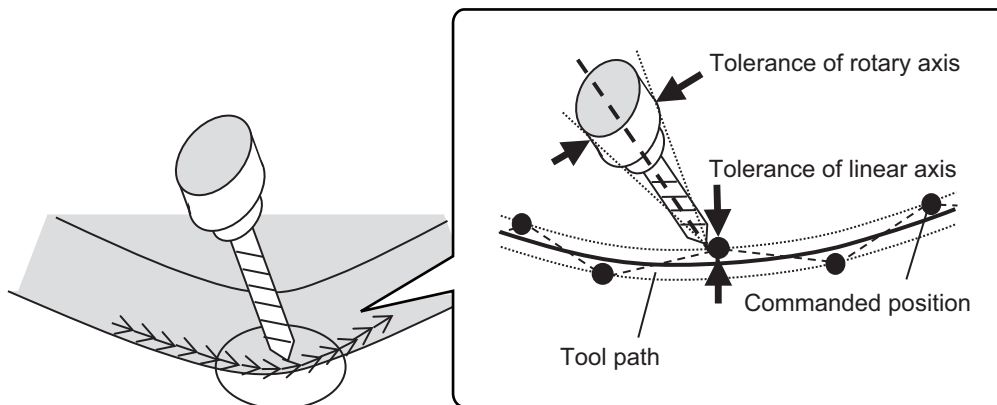
This function automatically generates a curve that smoothly passes through within the tolerable error range. The tool is able to move along the curve, providing smooth machining.

This function allows the machine to operate with the optimum tool path and speed, simply by specifying the tolerance, so an operator can easily attain high quality machining.

This function also requires the tolerance control specifications because it can only be used under tolerance control. The tolerance refers to the allowable error amount between the path commanded in the machining program and the path output by NC.



When spline interpolation 2 is used in combination with tool center point control, spline interpolation 2 is performed with 5 axes. It generates a curve that passes through the tool center point points smoothly within the tolerance, with the rotary axis angle also within the tolerance. The tool moves along the curve.



This function is enabled when the following conditions are satisfied:

- (1) SSS control is enabled.
- (2) Tolerance control is valid.
- (3) The specifications of spline interpolation 2 are valid.
- (4) "G61.4" is commanded from the machining program. (*1) (*2)
- (5) (When spline interpolation 2 is performed using 5 axes) Tool center point control command "G43.4" or "G43.5" is given from within a machining program. (*3)

(*1) If G61.4 is commanded while tolerance control is invalid, a program error (P34) occurs.

(*2) If G61.4 is commanded while the specifications of spline interpolation 2 are not defined, a program error (P39) occurs.

(*3) If G43.4 or G43.5 is given while the tool center point control specification is not provided, a program error (P39) occurs.

For details on spline interpolation 2 using 5 axes, refer to "Tool center point control" in "Details of operation".



Command format

Spline interpolation 2 mode ON

G61.4 (,K_) (,R_);

,K	Tolerance (mm) (For linear axis)
,R	Tolerance (deg) (For rotary axis)

Spline interpolation 2 mode with command G61.4 will be cancelled by designating any one of G code group 13.

- G61 (Exact stop check mode)
- G61.1 (High-accuracy control mode)
- G61.2 (Spline interpolation command)
- G62 (Automatic corner override)
- G63 (Tapping mode)
- G64 (Cutting mode)
- G08P1 (High-accuracy control mode start)
- G08P0 (High-accuracy control mode end)



Detailed description

Tolerance specification method

Designate the tolerance using one of the following methods.

- Designate the tolerance using the parameter "#2659 tolerance". When the setting is "0", the tolerance is handled as "0.01 (mm)" for a linear axis, or "0.01 (deg)" for a rotary axis.
- Designate the numeric value following the ",K" address in the G61.4 command for a linear axis, and following the ",R" address in the G61.4 command for a rotary axis.
 - (a) The tolerance range is 0.000 to 100.000 (mm), or 0.000 to 100.000 (deg). If a value exceeding the range is commanded, a program error (P35) will occur.
 - (b) The tolerance designated by ",K" is applied to all the linear axes in the part system.
The tolerance designated by ",R" is applied to all the rotary axes in the part system.
 - (c) When "0" is set to ",K" or ",R" or when ",K" or ",R" is omitted, the program runs using the setting value of the parameter "#2659 tolerance" as the tolerance.
 - (d) The tolerance designated by ",K" or ",R" is not held after reset. Therefore, if ",K" or ",R" is not designated in the G61.4 command after reset, the setting value of the parameter "#2659 tolerance" is enabled.

[Program example]

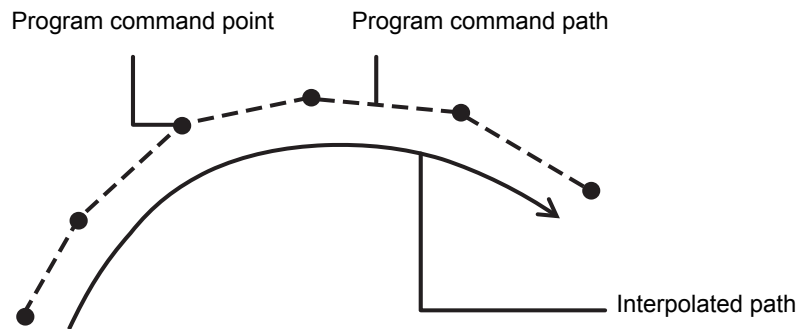
:	
G91 ;	
G61.4 ,K0.02;	Designate tolerance 0.02 (mm).
G01 X0.1 Z0.1 F1000 ; X0.1 Z-0.2 ; Y0.1 ;	Tolerance: 0.02 (mm)
G61.4 ,K0;	Designate the tolerance 0 [mm].
X-0.1 Z-0.05 ; X-0.1 Z-0.3 ;	Tolerance: Follows parameter "#2659 tolerance".
G64 ;	
:	



Details of operation

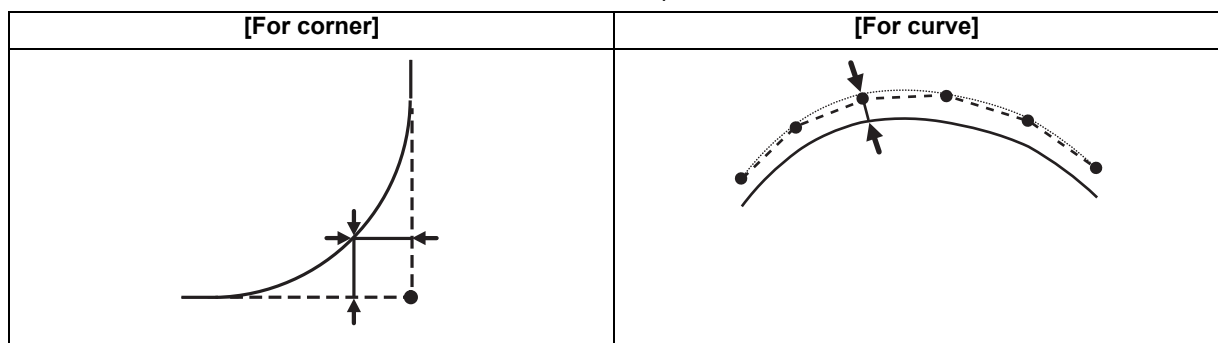
Basic operations

Spline interpolation 2 interpolates a command point row of the machining program with a smooth curve. The following figures show the command points and paths.

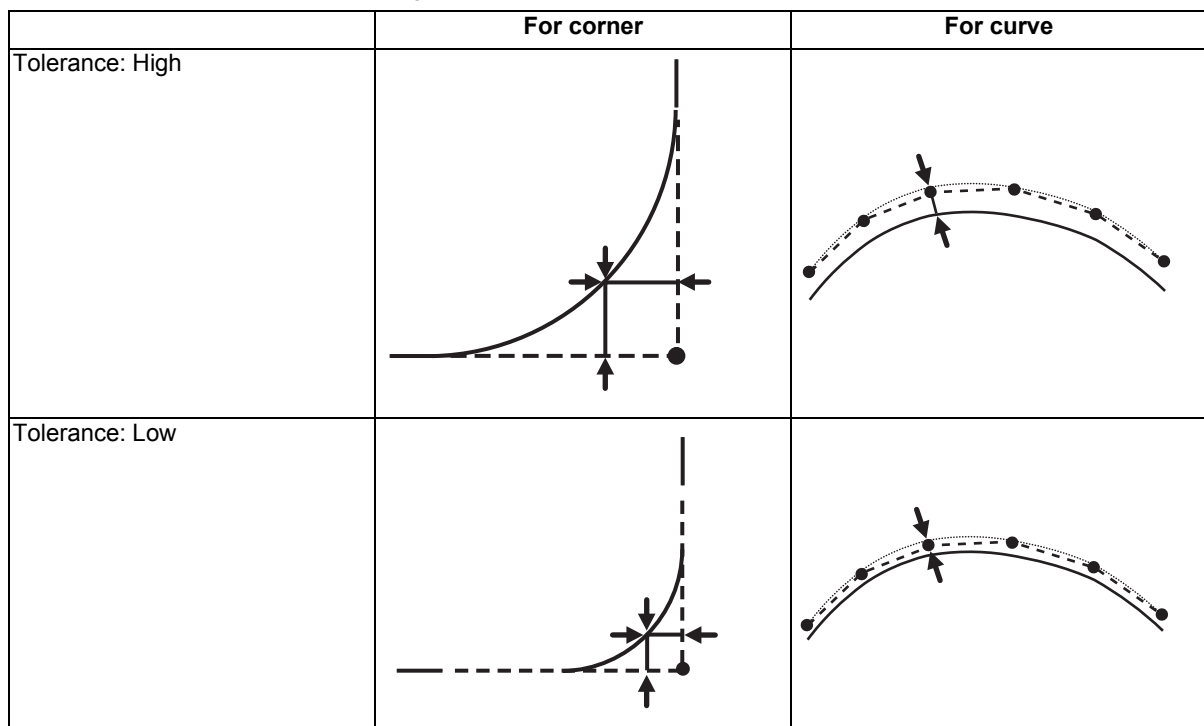


During spline interpolation 2, the control generates a smooth curve within the specified tolerance (tolerable error) and moves the tool along the generated path.

Illustrated below are the tolerances for a corner or curved shape.



The interpolated path varies depending on the tolerance as shown below.

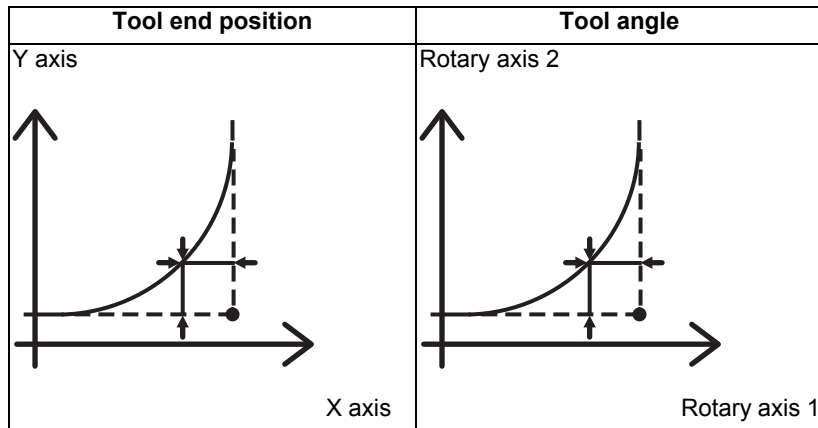


Behavior under tool center point control

Spline interpolation 2 can be used together with tool center point control (G43.4/G43.5). When all the conditions below are satisfied, spline interpolation 2 is performed using five axes.

- During G61.4 mode
- During tool center point control
- The G01 command in SSS control mode was issued.

Spline interpolation 2, coupled with tool center point control, generates a smooth curve along the tool center point points within the tolerance (tolerable error) and moves the tool along the curve. The tool posture is also controlled smoothly within the specified rotary axis angle tolerance to generate a smooth curve.



The points to note when tool center point control is used together with spline interpolation 2 are as follows:

- (1) The tool center point control specification is needed to enable the tool center point control.
- (2) Tool center point control is enabled only when "Joint interpolation method" is selected in "#7910 SLCT_INT_MODE" (Interpolation method selection). If "Single axis rotation interpolation method" is selected, the program error (P29) occurs.
- (3) The rotary axis prefiltering function cannot be used together.
- (4) Any G code that is unavailable under tool center point control cannot be commanded.

[Program example]

:	
G91 ;	
G43.4 H1;	Tool center point control ON
G61.4 ;	Spline interpolation 2 mode ON
G01 X0.1 Z0.1 F1000 ; X0.1 Z-0.2 ; Y0.1 ; X-0.1 Z-0.05 ; X-0.1 Z-0.3 ;	Spline interpolation 2 mode using 5 axes
G61.1 ;	Spline interpolation 2 mode OFF
G43.9;	Tool center point control OFF
:	

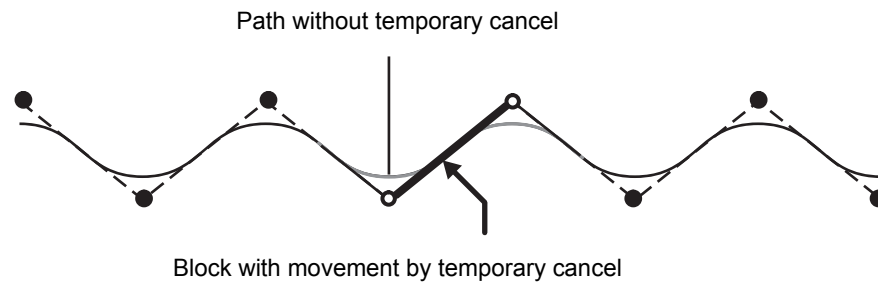
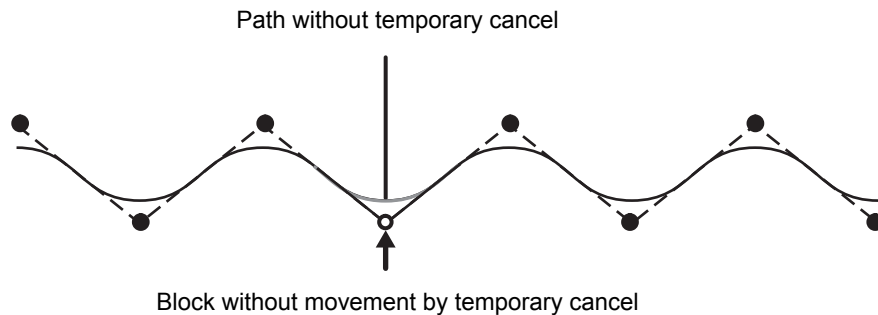
Temporary cancel

While spline interpolation 2 is enabled, it may be canceled temporarily depending on commands.

If spline interpolation 2 is canceled temporarily, the axis moves to the commanded position. After this, when a temporary cancel cause is removed, spline interpolation 2 restarts.

The temporary cancel conditions are as follows.

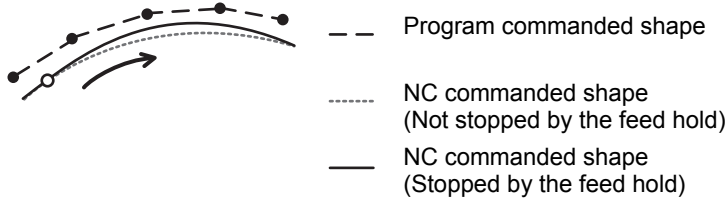
- (1) The group 1 modal is not G01, G02, or G03.
- (2) The block has a G code other than G90, G91, G01, G02, or G03 commanded.
- (3) The block has M (miscellaneous function command value), S (spindle command rotation speed), T (tool command value), or B (2nd miscellaneous function command value) designated.
- (4) Under single block operation (For details, refer to "Single Block Operation".)
- (5) Modal in which SSS control is disabled temporarily (Modal shown below)
 - NURBS interpolation
 - Polar coordinate interpolation
 - Cylindrical interpolation
 - User macro interruption enable (M96)
 - Feed per revolution (synchronous feed)
 - Inverse time feed
 - Constant surface speed control
 - Fixed cycle
 - 3-dimensional coordinate conversion
 - Hypothetical axis interpolation
 - Automatic tool length measurement
 - Tool length compensation along the tool axis
 - Normal line control
 - Unidirectional positioning
 - Exponential interpolation
 - 3-dimensional circular interpolation



Feed hold

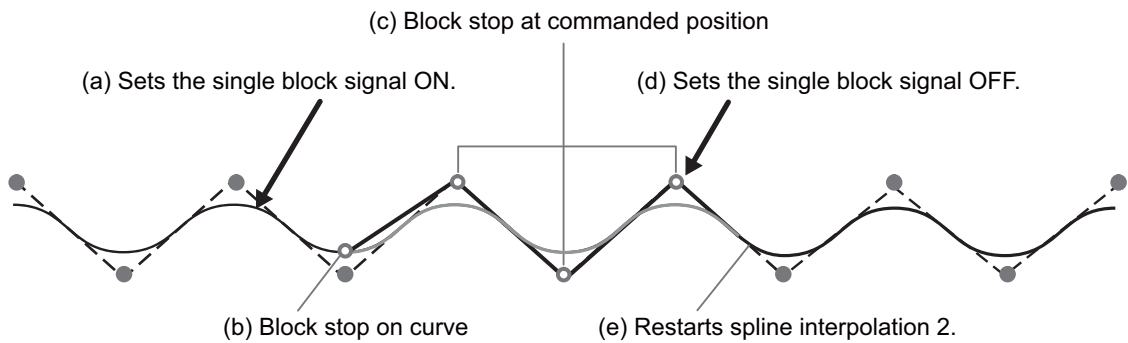
Feed hold allows a deceleration stop in the middle of a curve. However, no interrupt operation can be performed. If the mode is switched to the manual mode or MDI mode during the feed hold, an operation error (M01 0180) will occur and the interrupt operation will be prohibited.

After the program has been stopped by the feed hold, the movement on the curve can be restarted by the cycle start. The tool path specified just after the program has restarted is different from that specified when the program is not stopped by the feed hold, and the tool passes an area near the program-commanded shape.



Single block operation

During single block operation, spline interpolation 2 is canceled temporarily. In this period, linear interpolation is carried out at the commanded position. If single block is set to ON during continuous operation, the currently processed block stops on a curve, and the next and subsequent blocks stop on the commanded points.



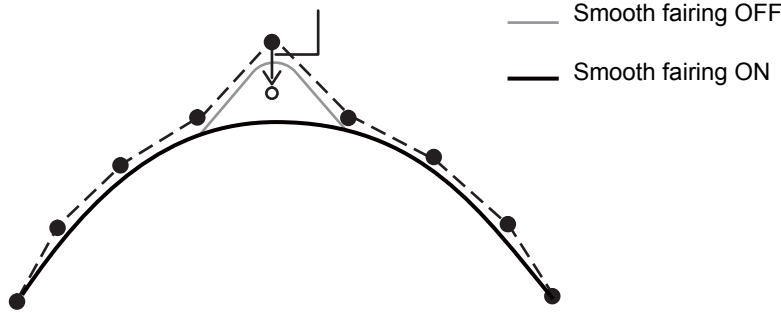


Relationship with other functions

Smooth fairing

Spline interpolation 2 and smooth fairing can be used together. A spline interpolation 2 curve is generated along the points that are compensated by smooth fairing. If spline interpolation 2 is used together with tool center point control, smooth fairing is disabled.

Compensation by smooth fairing



Tool radius compensation

Spline interpolation 2 and tool radius compensation can be combined. A spline curve is generated along the path for which the radius is compensated.

High-speed high-accuracy control III

Spline interpolation 2 and high-speed high-accuracy control III can be combined. However, the fine segment processing capacity is limited.

Spline interpolation 2 used in combination with tool center point control cannot be used together with high-speed high-accuracy control III.

Spline interpolation

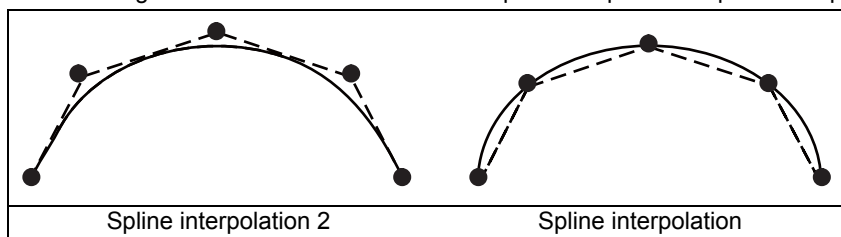
Spline interpolation 2 (G61.4) and spline interpolation (G61.2/G05.1Q2) cannot be combined.

The following differences are between spline interpolation 2 (G61.4) and spline interpolation (G61.2/G05.1Q2).

	Feature of spline curve	Parameter for adjusting the curve shape
Spline interpolation 2 (G61.4)	Passes near the commanded points. (*1)	#2659 tolerance
Spline interpolation (G61.2/G05.1Q2)	Passes on the commanded points.	#8026 CANCEL ANG. #8027 Toler-1 #8028 Toler-2 #8029 FairingL #8030 MINUTE LENG #8033 Fairing ON

(*1) The axis passes through the commanded points at the start and end points.

The following shows differences between the spline interpolation 2 path and spline interpolation path.



Inclined surface machining command and workpiece installation error compensation

Inclined surface machining is available when tool center point control is used together with spline interpolation 2. Workpiece installation error compensation cannot be used together with spline interpolation 2 even when tool center point control is enabled.

The points to note when tool center point control is used together with spline interpolation 2 are as follows:

- (1) If G61.4 is given during inclined surface machining in any mode other than tool center point control mode, the operation is performed in the G61.1 mode. G61.4 mode is activated after tool center point control command is given.
- (2) Any G code that is unavailable under inclined surface machining cannot be commanded.

[Program example]

:	
G68.2 X45. Y-50. I0. J32. K0.	Inclined surface machining command ON
G53.1	
G43.4 H1;	Tool center point control ON
G61.4 ;	Spline interpolation 2 mode ON
G01 X0.Y0. F1000 ; Z0. X50. Y50. ; Y0. ; X0. ;	Spline interpolation 2 mode using 5 axes
G61.1 ;	Spline interpolation 2 mode OFF
G49;	Tool center point control OFF
G69	Inclined surface machining command OFF

[Relationship between spline interpolation 2 and each function]

G code	Function name	When the G codes shown on the left are commanded while spline interpolation 2 is enabled	When spline interpolation 2 is enabled while the functions shown on the left are enabled
G43.4 G43.5	Tool center point control	Operation is performed in spline interpolation 2 mode using five axes.	Operation is performed in spline interpolation 2 mode using five axes.
G68.2 G68.3	Inclined surface machining command	<ul style="list-style-type: none"> ♦Under tool center point control, the program error (P952) occurs. ♦While tool center point control is disabled, the operation with G61.1 commanded is carried out. 	<ul style="list-style-type: none"> ♦While tool center point control is enabled, spline interpolation 2 operation using five axes is carried out. ♦While tool center point control is disabled, the operation with G61.1 commanded is carried out.
G54.4 P1 to P7	Workpiece installation error compensation	Program error (P546)	Program error (P546)



Precautions

- (1) The graphic check drawing is not carried out during spline interpolation 2 (the period from G61.4 to the cancel command).
- (2) PLC interrupt is not available during spline interpolation 2. If an PLC interrupt is performed during spline interpolation 2, the operation error (M01 0180) will occur.
- (3) When the arrangement at the commanded point of the adjacent path of the machining program generated from CAM is different extremely, the tool center path generated by spline interpolation may not be aligned with the adjacent path. In this case, reduce the setting value of the parameter "#2659 tolerance" (tolerance amount) or the commanded value of the "K" address. You can move the tool center path closer to the machining program path. However, the cycle time becomes longer.

	(a) When the tolerance amount is large	(b) When the tolerance amount is small
There is a position at which the commanded point is not arranged in the adjacent path.		
The block segment length of the adjacent path is different extremely.		

(*1) The commanded point is not arranged.

(*2) The arrangement interval between the commanded points is different extremely.

- Program command point
- > Program path
- Tool center path

17.6 High-accuracy Spline Interpolation ; G61.2



Function and purpose

This function automatically generates a spline curve that passes through a sequence of points commanded by the fine segment machining program, and interpolates the path along this curve. This enables high-speed and high-accuracy machining to be achieved.

This function has two functions; fairing function to delete unnecessary fine blocks, and spline interpolation function to connect smoothly a sequence of points commanded by the program.

The high-accuracy control function G61.1 is also valid.

The high-accuracy spline Interpolation is valid only for the first part system.

G61.2 cannot be commanded in the 2nd part system even when the multi-part system simultaneous high-accuracy specifications are available.

There are two types of spline interpolation command format: G61.2 and G05.1Q2. Both formats can be used regardless of the parameter "#1267 ext03/bit0" setting if the spline interpolation specifications are available to the machine. This section describes the G61.2 command. For information about differences between G05.1Q2 and G61.2 or features of spline interpolation, refer to "Spline Interpolation ; G05.1Q2".



Command format

G61.2 X__ Y__ Z__ F__ ; or G61.2 ; ... Spline mode ON

X	X axis end point coordinate
Y	Y axis end point coordinate
Z	Z axis end point coordinate
F	Feedrate

The "G61.2" high-accuracy spline interpolation mode is canceled when any of the functions of G code group 13 is commanded.



Detailed description

(1) Fairing

Refer to "Additional functions when high-speed high-accuracy control II mode is ON" in "High-speed high-accuracy control".

(2) Spline interpolation

Refer to "Detailed description" of "Spline Interpolation".



Program example

```

:
G91 ;
G61.2 ;           ..... Spline interpolation mode ON
G01 X0.1 Z0.1 F1000 ;
X0.1 Z-0.2 ;
Y0.1 ;
X-0.1 Z-0.05;
X-0.1 Z-0.3;
G64 ;           ..... Spline interpolation mode OFF
:

```

- (1) The spline interpolation is available when the following conditions are all satisfied. If the following conditions are not satisfied, the spline function will be canceled once, and the judgment whether to carry out new spline from the next block will be made.
 - It is the movement only of three axes set to the basic axes I, J and K.
 - When the block length is smaller than the value of the machining parameter "#8030 MINUTE LENGHS".
 - When the movement amount is not 0.
 - The group 1 command is G01 (linear interpolation).
 - Operation in fixed cycle modal
 - It is not during hypothetical axis interpolation mode.
 - It is not during 3-dimensional coordinate conversion modal.
 - It is not in a single block mode.
- (2) The spline function is a modal command of group 13. This function is valid from G61.2 command block.
- (3) The spline function is canceled by group 13 commands (G61 to G64).
- (4) The spline function is canceled by NC reset 2, reset & rewind, NC reset 1 (the setting which does not hold modal when NC is reset) or power ON/OFF.



Precautions

- (1) If this function are not provided and G61.2 is commanded, the program error (P39) will occur.
- (2) Even if "-1" is set for parameter "#8030 MINUTE LENGHS", the spline function will be temporarily canceled by the cancel conditions (cancel angle, non-movement block, excessive chord error, etc.) other than the block length.
- (3) Graphic check will draw the shape of when the spline interpolation OFF.
- (4) A program error (P34) will occur if the number of axis in the part system does not exceed 3.

17.7 Machining Condition Selection I ; G120.1,G121



Function and purpose

After initializing the machining condition parameter groups with the machining condition selection I function, the machining condition parameter groups can be switched by G code command.

Switching is also possible on the machining condition selection screen. In that case, however, the machining conditions selected on the screen are applied to all part systems.



Command format

Machining condition selection I

G120.1 P_ Q_ ;

P	Machining purpose 0: Reference parameter 1: Usage 1 2: Usage 2 3: Usage 3
Q	Condition 1: Condition 1 2: Condition 2 3: Condition 3 When omitted, Q1 will be applied

Machining condition selection I cancel

G121 ;



Detailed description

- (1) G120.1 and G121 commands are unmodal commands of G code group 0.
- (2) Switching of the machining condition parameter group using the G120.1 or G121 command is only applied to the commanded part system.
- (3) Command G120.1 and G121 in an independent block. If not, a program error (P33) will occur.
- (4) Address P in G120.1 command cannot be omitted. If omitted, a program error (P33) will occur.
- (5) Address Q in G120.1 command can be omitted. If omitted, it will be handled as "Q1 (condition 1)" is commanded.
- (6) When address P and Q in G120.1 command is commanded with a decimal point, the digit after the decimal point is ignored.
- (7) If other than "0 to 3" is set to address P in G120.1 command or other than "1 to 3" is set to address Q, a program error (P35) will occur.
- (8) When address P is set to "0" and address Q is omitted or set between "1" and "3" in G120.1 command, it will be switched to the reference parameter.
- (9) It will be switched to the machining condition parameter group selected in "Machining cond" screen by the G121 command.
- (10) When the emergency stop and reset (reset 1, reset 2, and reset & rewind) are performed while running the machining program whose machining condition parameter group is switched by G120.1 command, it will be switched to the selected condition parameter group machining in "Machining cond" screen.

- (11) Because the parameters are switched after being decelerated by G120.1 and G121 commands, the workpiece may be damaged. Make sure to keep the tool away from the workpiece when commanding G120.1 and G121.
- (12) When the machining condition parameter group is switched by G120.1 command more than once, the parameter group commanded last becomes valid.
- (13) It is switched to the selected machining condition parameter group in the "Machining cond" screen by program end (M02 and M30).
- (14) If G120.1 and G121 are commanded without initializing the machining condition parameter group, a program error (P128) will occur.



Program example

"Machining cond" (setting) screen

The displayed machining condition parameter group is switched depending on whether tolerance control is enabled or disabled (parameter "#12066 Tolerance ctrl ON").

Machining cond					
Application1 PROCESS1					
No.	Name	Ref. param	Machining condition parameter set Condition1	Condition2	Condition3
1206	G1bF	100000	100000	100000	100000
1207	G1btL	100	100	100	100
	Cutting feed Acc	1.700	1.700	1.700	1.700
1568	SfiltG1	0	0	0	0
	Notch frequency Hz	0.000	0.000	0.000	0.000
2659	tolerance	0.000	0.000	0.000	0.000
12070	Sfilt2_tol	0	0	0	0

High-speed setting
(for rough cutting machining)

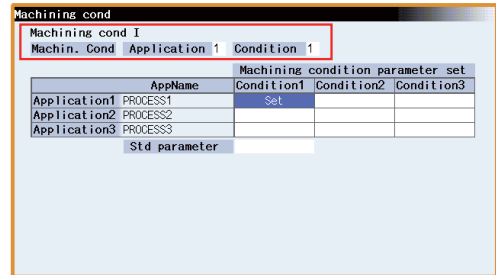
Standard setting
(for medium finishing machining)

High-accuracy setting
(for finishing machining)

(1) When "Application1" and "Condition1" from the machining condition parameter group are selected in "Machining cond" (selecting) screen before running the program.

N1 G91; G28 Z0;
N2 G28; X0 Y0;

Operate with the machining condition parameter group (machining usage 1/condition 1)



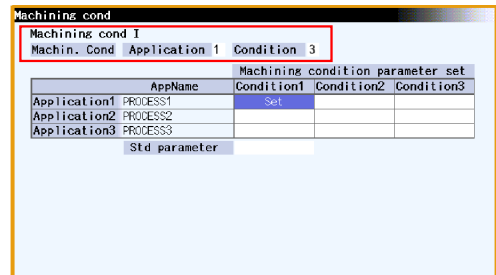
N3 G90 G54 G00 X2. Y2.;;
N4 G43 H1 Z50.;;
N5 G90 G01 Z-5. F3000;
N6 M3 S10000;
N7 F2000;
N8 G05 P10000;
N9 G01; X2.099 Y1.99;
N10 X2.199 Y1.990;
:

N1499 G05 P0;
N1500 G91; G28 Z0;
N1501 G28; X0 Y0;
N1502 M5;
N1503 G120.1 P1 Q3;

The machining condition parameter groups are switched.

N1504 G90 G54 G00 X2. Y2.;;
N1505 G43 H1 Z50.;

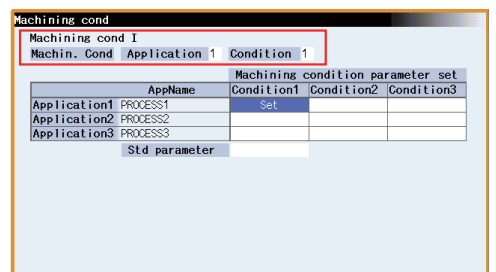
Operate with the machining condition parameter group (machining usage 1/condition 3)



N1506 G90 G01 Z-8. F3000;
N1507 M3 S10000;
N1508 F1200;
N1509 G05 P10000;
N1510 G01; X2.099 Y1.997;
N1511 X2.199 Y1.990;
:

N2999 G05 P0;
N3000 G91; G28 Z0;
N3001 G28; X0 Y0;
N3002 M5;
N3003 M30;

Return to the selected machining condition parameter group in "Machining cond" screen at the program end.





Relationship with other functions

- (1) G code modal that cause a program error when commanding G120.1 and G121 are listed below.

G code	Function	Program error when G120.1 and G121 are commanded
G02.3, G03.3	Exponential interpolation	P128
G06.2	NURBS interpolation	P32
G07.1	Cylindrical interpolation	P128
G12.1	Polar coordinate interpolation	P128
G10	Parameter input by program	P421
	Tool compensation input by program	
G33	Thread Cutting	P128
G38	Tool radius compensation (vector designation)	P128
G39	Tool radius compensation (corner arc)	P128
G41, G42	Tool radius compensation	P128
	3-dimensional tool radius compensation	
G41.1/G151	Normal line control Left	P128
G42.1/G152	Normal line control Right	P128
G43	Tool length compensation (+)	P128
G44	Tool length compensation (-)	P128
G43.1	Tool length compensation along the tool axis	P128
G43.4, G43.5	Tool center point control	P942
G66, G66.1	User macro (modal call A, B)	P128
G68.2, G68.3	Inclined surface machining	P951
G73/G74/G76/G81/G82/G83/ G84/G85/G86/G87/G88/G89	Fixed cycle	P33 (When G120.1 command is issued)
		P128 (When G121 command is issued)



Precautions

- (1) Because the parameters are switched after being decelerated once G120.1 or G121 is commanded, the workpiece may be damaged. Make sure to keep the tool away from the workpiece when commanding G120.1 and G121.
- (2) For the parameters "#8033 Fairing ON" and "#8090 SSS ON", the switched machining condition parameter group is effective only after it has been switched on the machining condition selection screen.
- (3) It is switched to the reference parameter by turning the power ON again.
- (4) When G120.1 and G121 are commanded, parameters are switched when smoothing for NC axes in all part systems become "0".
- (5) The machining condition parameter group neither set the parameter setting from the program by G10 command nor read the parameters by system variables (from #100000).
- (6) When the machining condition parameter group is switched, the same values are used for all NC axes which belong to the switched part system to the parameter "#2010 Feed forward gain" and "#2659 Tolerance".
- (7) The machining condition parameters are not switched for the operation search. The machining condition parameters are switched for the restart search.

Advanced Multi-Spindle Control Function

18.1 Spindle Synchronization



Function and purpose

In a machine having two or more spindles, this function controls the rotation speed and phase of one spindle (reference spindle) in synchronization with the rotation of the other spindle (synchronized spindle).

This function provides, for example, an effect that re-grasps the workpiece grasped by the 1st spindle to the 2nd spindle while maintaining the rotation speed of the 1st spindle to reduce the cycle time by the 1st spindle deceleration time and the 2nd spindle acceleration time in the next process during re-grasping.

Furthermore, this function carries out turning or phase control while grasping both edges of a longer workpiece using the 1st and 2nd spindles, preventing a twist or bow from occurring in the workpiece under machining and enabling the machining accuracy.

The spindle synchronous multi-step acceleration/deceleration of the reference spindle is applied to the acceleration/deceleration of the spindle-synchronization relation spindle under spindle synchronization. (Only for C80 series)

The following control methods are available. Which mode is valid depends on the MTB specifications (parameter "#1300 ext36/bit7"). This section describes spindle synchronization control I that is executed with G commands.

Spindle synchronization I

The designation of the synchronized spindle and start/stop of the synchronization are executed by commanding G codes in the machining program.

Spindle synchronization II

The selections of the synchronized spindle and synchronization start, etc., are controlled from PLC based on the MTB specifications. Refer to the instruction manual issued by the MTB for details.

18.1.1 Spindle Synchronization I; G114.1

**Function and purpose**

With the spindle synchronization I, the designation of the spindle and start/stop of the synchronization are executed by commanding G codes in the machining program.

**Command format****Spindle synchronization command**

G114.1 H__D__R__A__ ;	Spindle synchronization control start command
------------------------------	--

G114.1 D__R__ ;	Phase shift amount change during spindle synchronization (*1) [C80]
------------------------	--

(*1) This command changes the phase shift amount of the synchronized spindle of two spindles which have already synchronized by spindle synchronization command.

H	Reference spindle specification
D	Synchronized spindle specification
R	Phase shift amount of synchronized spindle
A	Spindle synchronization acceleration/deceleration time constant

Spindle synchronization control ON (G114.1) command designates the reference spindle and synchronized spindle, and synchronizes the two designated spindles. By commanding the synchronized spindle phase shift amount, the phases of the reference spindle and synchronized spindle can be aligned.

Address	Meaning of address	Command range (unit)	Remarks
H	Reference spindle specification Select the number or name of the spindle to be used as the reference spindle from the two spindles to be synchronized. (*1)	For spindle number: 1 to n (n: Maximum number of available spindles) For spindle name: 1 to 9	<ul style="list-style-type: none"> ♦If a value exceeding the command range or spindle No. without specifications is commanded, a program error (P35) will occur. ♦If there is no command, a program error (P33) will occur. ♦If an analog-connected spindle is commanded, a program error (P700) will occur. (*2)
D	Synchronized spindle specification Select the number or name of the spindle to be synchronized with the reference spindle from the two spindles to be synchronized. (*1)	For spindle number: 1 to n or -1 to -n (n: Maximum number of available spindles) For spindle name: 1 to 9 or -1 to -9	<ul style="list-style-type: none"> ♦If a value exceeding the command range is commanded, a program error (P35) occurs. ♦If there is no command, a program error (P33) will occur. ♦If the same spindle as that commanded for the reference spindle selection is designated, a program error (P33) will occur. ♦The rotation direction of the synchronized spindle in respect to the reference spindle is commanded with the D sign. ♦If an analog-connected spindle is commanded, a program error (P700) will occur. (*2)

Address	Meaning of address	Command range (unit)	Remarks
R	Phase shift amount of synchronized spindle Set the shift amount from the synchronized spindle's reference position (one rotation signal).	0 to 359.999 (°) or 0 to 359999 (° * 10 ⁻³)	<ul style="list-style-type: none"> •If a value exceeding the command range is commanded, a program error (P35) occurs. •The commanded shift amount will be effective in the clockwise direction of the reference spindle. •Minimum resolution of commanded shift amount <ul style="list-style-type: none"> Semi-close case (Gear ratio: 1:1 only) 360/4096 [°] Full-close case (360/4096) * K [°] (K: Gear ratio of spindle and encoder) •If there is no R command, phase alignment will not be carried out.
A	Spindle synchronization acceleration/deceleration time constant Command the acceleration/deceleration time constant for when the spindle synchronization command rotation speed changes. (Command this to accelerate or decelerate at a speed slower than the time constant set in the parameters.)	0.001 to 9.999 (s) or 1 to 9999 (ms)	<ul style="list-style-type: none"> •If a value exceeding the command range is commanded, a program error (P35) occurs. •If the commanded value is smaller than the acceleration/deceleration time constant set with the parameters, the value set in the parameters will be applied.

(*1) There are two spindle command methods: spindle number method and spindle name method.

Command with the spindle names, only when all spindles are set with the spindle name parameter (#3077 Sname) (from 1 to 9). For others, command with the spindle number. These settings depend on the MTB specifications.

(*2) The available spindle type and connection method depend on the specifications of your machine tool.

Canceling Spindle Synchronization

```
G113.1;
```

Note

- (1) An axis that involves any travel cannot be put in the same block as the Spindle synchronization cancel command. Doing so causes the program error (P33) when the cancel command is issued, which causes automatic operation to pause.

Spindle synchronization cancel command (G113.1) cancels the synchronous state of the two spindles rotating in synchronization with the spindle synchronization command.



Detailed description

Rotation speed and rotation direction

- (1) The rotation speed and rotation direction of the reference spindle and synchronized spindle during spindle synchronization are the rotation speed and rotation direction commanded for the reference spindle. Note that the rotation direction of the synchronized spindle can be reversed from the reference spindle through the program.
- (2) The reference spindle's rotation speed and rotation direction can be changed during spindle synchronization.
- (3) If spindle stop is commanded for the synchronized spindle during spindle synchronization, the synchronized spindle rotation will stop.
- (4) The rotation speed command (S command) and constant surface speed control are invalid for the synchronized spindle during spindle synchronization. Note that the modal is updated, so these will be validated when the spindle synchronization is canceled.
- (5) The constant surface speed can be controlled by issuing a command to the reference spindle even during spindle synchronization.

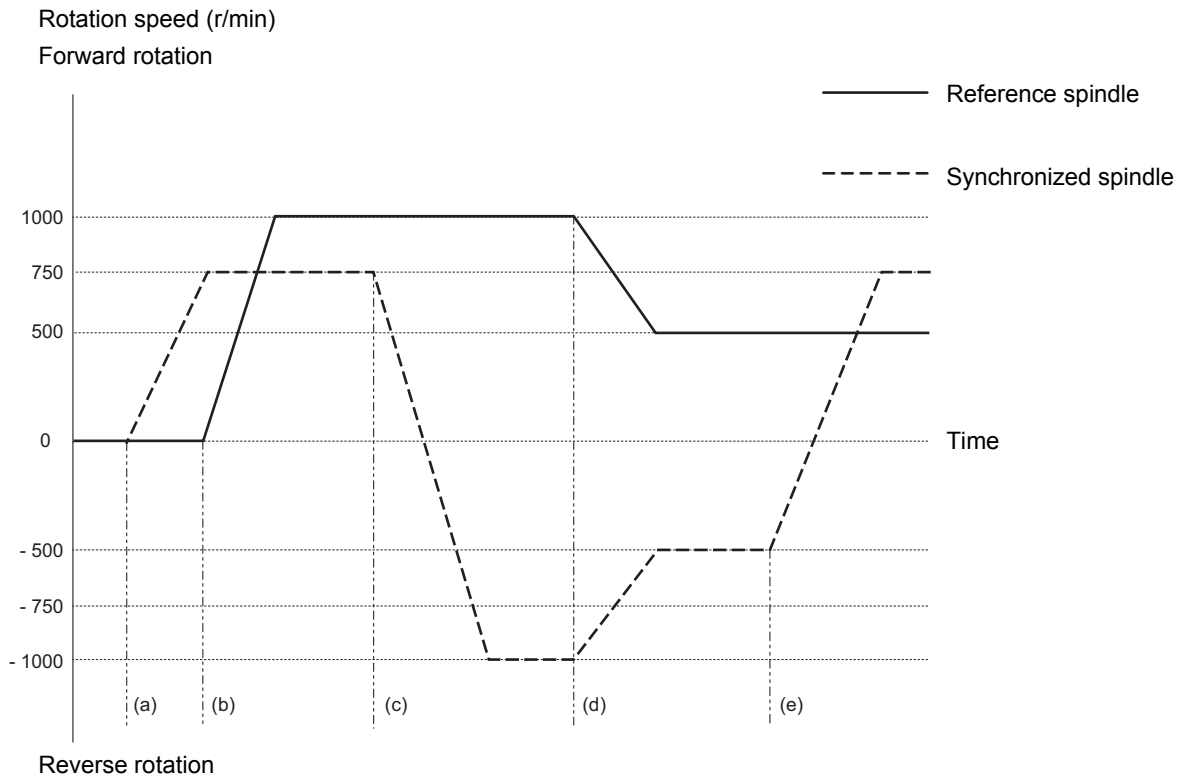
Rotation synchronization

- (1) When rotation synchronization control (command with no R address) is commanded with the G114.1 command, the synchronized spindle rotating at an arbitrary rotation speed will accelerate or decelerate to the rotation speed commanded beforehand for the reference spindle, and will enter the rotation synchronization state.
- (2) If the reference spindle's commanded rotation speed is changed during the rotation synchronization state, acceleration/deceleration will be carried out while maintaining the synchronization state following the spindle acceleration/deceleration time constants set in the parameters, and the commanded rotation speed will be achieved.
- (3) In the rotation synchronization state, the reference spindle can be controlled at a constant surface speed even when two spindles are grasping one workpiece.
- (4) The following type of operation will take place.

```

M23 S2=750 ;      Forward rotate the 2nd spindle (synchronized spindle) at 750 r/min (speed command). (a)
:
M03 S1=1000 ;    Forward rotate the 1st spindle (reference spindle) at 1000 r/min (speed command). (b)
:
G114.1 H1 D-2 ;  Synchronize the 2nd spindle (synchronized spindle) with the 1st spindle (reference spindle) by reverse run. (c)
:
S1=500 ;        Change the rotation speed of the 1st spindle (reference spindle) to 500 r/min. (d)
:
G113.1;        Cancel the spindle synchronization. (e)
    
```

<Operation>



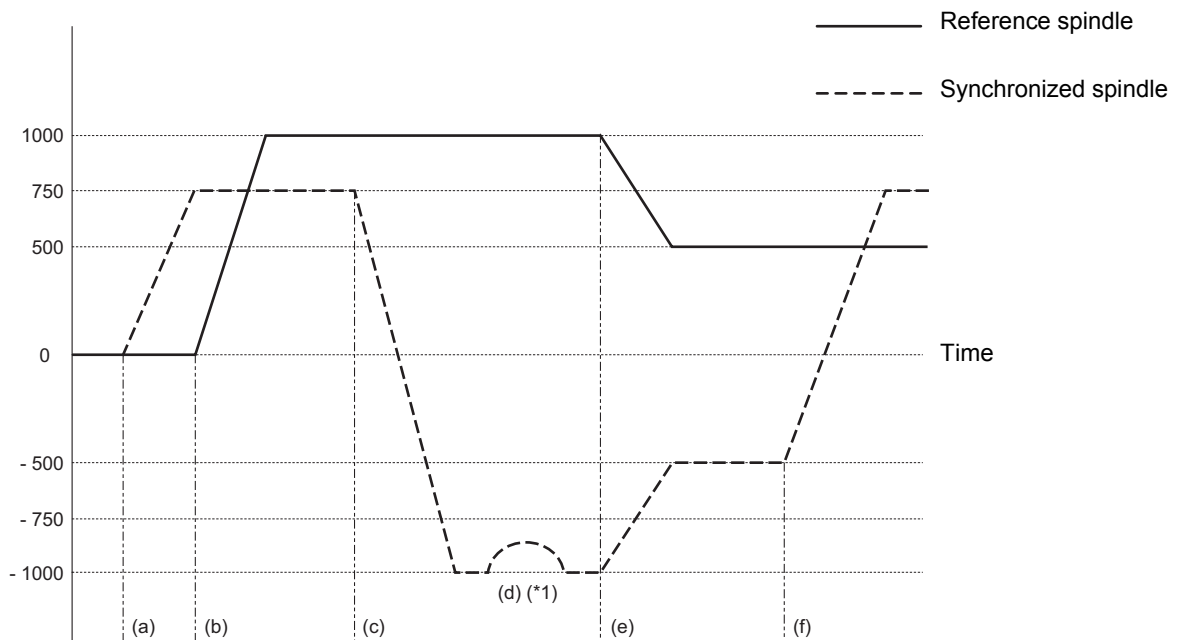
Phase synchronization

- (1) When phase synchronization (command with R address) is commanded with the G114.1 command, the synchronized spindle rotating at an arbitrary rotation speed will accelerate or decelerate to the rotation speed commanded beforehand for the reference spindle, and will enter the rotation synchronization state. Then, the phase is aligned so that the rotation phase commanded with the R address is reached, and the phase synchronization state is entered.
- (2) If the reference spindle's commanded rotation speed is changed during the phase synchronization state, acceleration/deceleration will be carried out while maintaining the synchronization state following the spindle acceleration/deceleration time constants set in the parameters, and the commanded rotation speed will be achieved.
- (3) In the phase synchronization state, the reference spindle can be controlled at the constant surface speed even when two spindles are grasping one workpiece.
- (4) The following type of operation will take place.

M23 S2=750 ;	Forward rotate the 2nd spindle (synchronized spindle) at 750 r/min (speed command). (a)
:	
M03 S1=1000 ;	Forward rotate the 1st spindle (reference spindle) at 1000 r/min (speed command). (b)
:	
G114.1 H1 D-2 R0;	Synchronize the 2nd spindle (synchronized spindle) with the 1st spindle (reference spindle) by reverse run. (c)
	Shift the phase of synchronized spindle by the value commanded with "R". (d)
:	
S1=500 ;	Change the rotation speed of the 1st spindle (reference spindle) to 500 r/min. (e)
:	
G113.1;	Cancel the spindle synchronization. (f)

<Operation>

Rotation speed (r/min)
Forward rotation



Reverse rotation

- (*1) Phase synchronization is performed with the step alignment method (without acceleration/deceleration) when "#3130 syn_spec/bit1" = "0", and with the multi-step acceleration/deceleration method (described later) when "#3130 syn_spec/bit1" = "1".

Operation when the "phase shift amount change during spindle synchronization" command is issued [C80]

(1) When the "phase shift amount change during spindle synchronization" command (G114.1 D_ R_) is issued to change the phase shift amount of the spindle, for which the rotation or phase has already synchronized by the G114.1 command, the phase is aligned so that the rotation phase commanded with the R address is reached, and the phase synchronization state is entered.

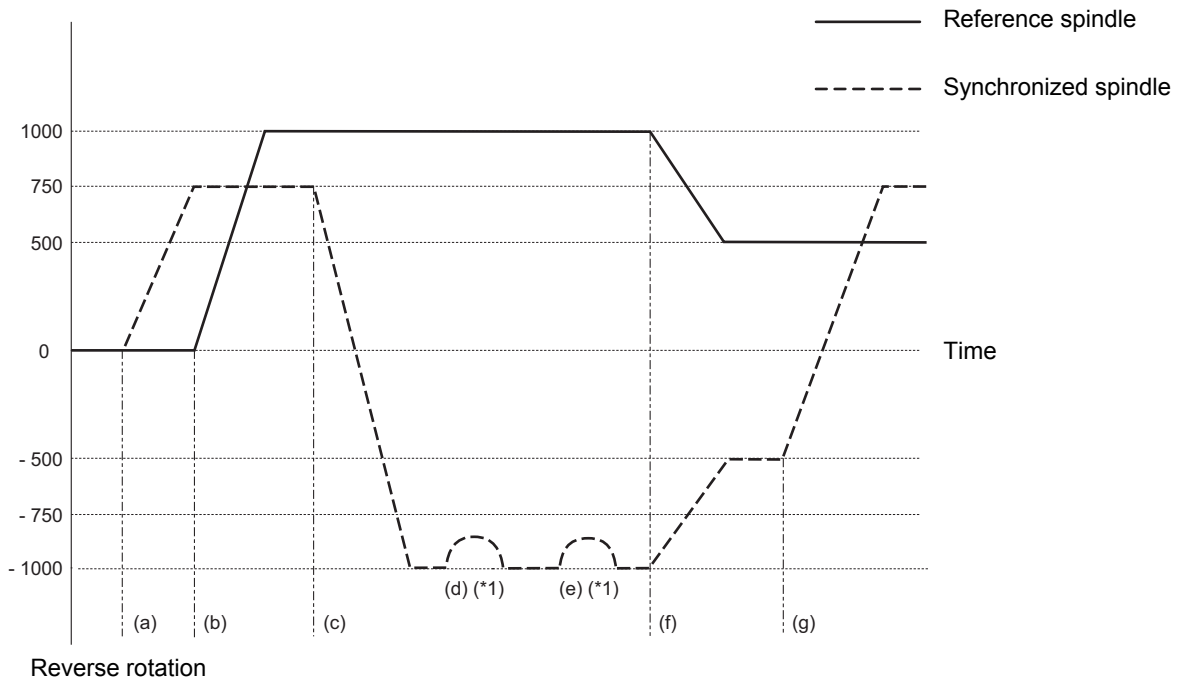
The following type of operation will take place.

```

N01 M23 S2=750;      Forward rotate the 2nd spindle (synchronized spindle) at 750 r/min (speed
                    :                               command). (a)
:
N10 M03 S1=1000;    Forward rotate the 1st spindle (reference spindle) at 1000 r/min (speed com-
                    :                               mand). (b)
:
N20 G114.1 H1 D-2 Rxx; Synchronize the 2nd spindle (synchronized spindle) with the 1st spindle (refer-
                    :                               ence spindle) by reverse run. (c)
:                               Shift the phase of synchronized spindle by the value commanded with "R". (d)
N25 Mzz;           Wait until phase synchronization is completed.
                    :                               (An M code is used to check the completion of spindle phase synchronization)
:
N30 G114.1 D2 Ryy; Shift the phase of the 2nd spindle (synchronized spindle) by the value com-
                    :                               manded with "R". (e)
:
N40 S1=500;        Change the rotation speed of the 1st spindle (reference spindle) to 500 r/min.
                    :                               (f)
:
N50 G113.1;        Cancel the spindle synchronization. (g)
    
```

<Operation>

Rotation speed (r/min)
Forward rotation



Reverse rotation

(*1) Phase synchronization is performed with the step alignment method when "#3130 syn_spec/bit1" = "0", and with the multi-step acceleration/deceleration method (described later) when "#3130 syn_spec/bit1" = "1".

- (2) The operation to be performed when the "phase shift amount change during spindle synchronization" command (G114.1 D_ R_) is issued varies depending on conditions, for example, whether spindle synchronization has started.

		Phase shift amount (When "start" command (*1) is issued)		State in which the spindle synchronization "start" command is not executed.
		"R" is omitted. (G114.1 H_ D_)	"R" is specified. (G114.1 H_ D_ R_)	
Phase shift amount (When "change" command (*2) is issued)	"R" is omitted. (G114.1 D_)	Unchanged as specified when the start command (*1) is issued.	The phase remains unchanged with the shift amount specified when the start command (*1) is issued.	Program error (P33)
	"R" is specified. (G114.1 D_ R_)	The phase shift is carried out in accordance with the change command (*2).		

(*1) This is "Spindle synchronization control start" command.

(*2) This is "Phase shift amount change during spindle synchronization" command.

Spindle synchronization phase shift amount calculation function

The spindle phase shift amount calculation function obtains and saves the phase difference of the reference spindle and synchronized spindle by turning the PLC signal ON when the phase synchronization command is executed. When the phase is positioned to the automatically saved phase difference before executing the phase synchronization control command, phases can be aligned easier when re-grasping profile materials.

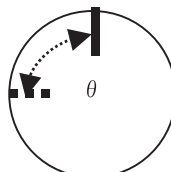
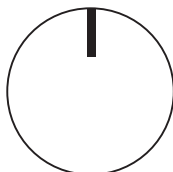
[Saving the reference spindle and synchronized spindle phase difference]

- (1) Set a profile material in the reference spindle.
- (2) Set the profile material in the synchronized spindle.
- (3) Turn the phase shift calculation request signal (SSPHM) ON.
- (4) Input a rotation command, with 0 speed, for the reference spindle and synchronized spindle.
<Example> M3 S1=0 M24 S2 = 0;
- (5) Execute the rotation synchronization signal (with no R address command).
<Example> G114.1 H1 D-2;
- (6) Rotate the reference spindle at the speed actually used when re-grasping.
<Example> S1 = 3000;
- (7) Check that the phase difference has been saved by looking at the spindle speed synchronization complete signal.
- (8) Stop both spindles.
- (9) Turn the phase shift calculation request signal OFF.

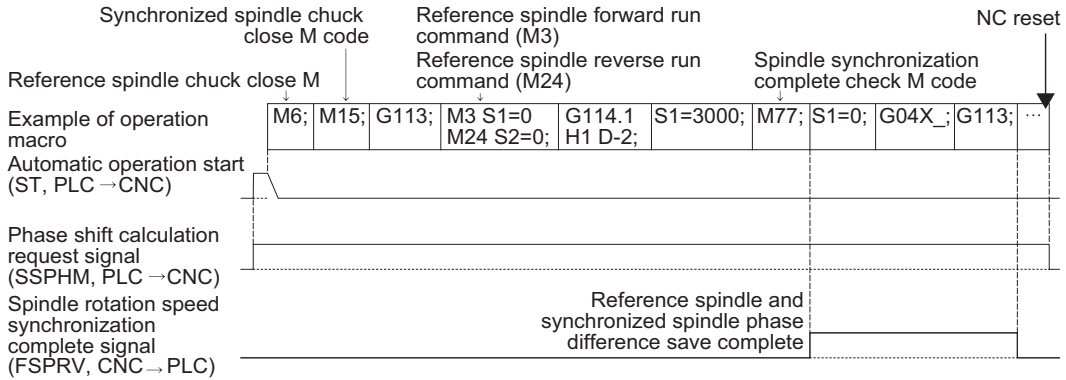
Reference spindle

Synchronized spindle

θ : Saved phase difference

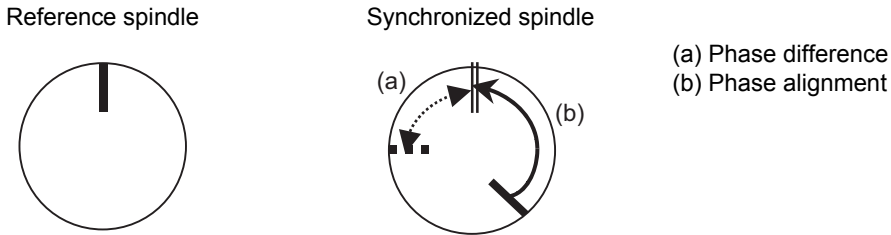


<Example of operation>

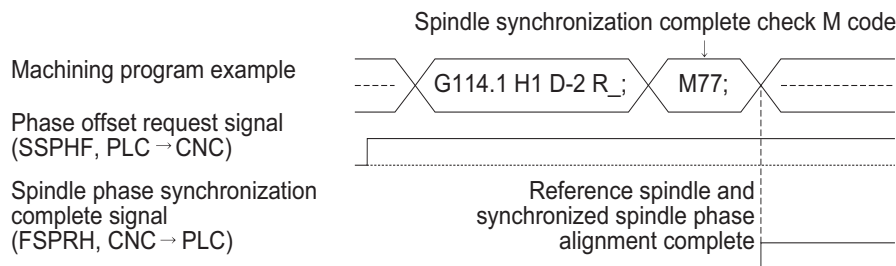


[Automatic phase alignment of reference spindle and synchronized spindle]

- (1) Turn the phase offset request signal ON.
- (2) Issue the phase synchronization command (with R command).
<Example> G114.1 H1 D-2 R0;
- (3) The phase is aligned by offsetting the phase synchronization command by the phase difference obtained with the spindle synchronization phase shift calculation function. The state in which the synchronized spindle phase shift amount designation R value is 0 is the same as the reference state (state obtained with phase shift calculation request signal).



<Example of operation>



Multi-step acceleration/deceleration

Acceleration/deceleration time constants for up to eight steps can be selected according to the spindle rotation speed for the acceleration/deceleration during spindle synchronization.

The acceleration/deceleration in each step is as follows.

Time required from minimum rotation speed to maximum rotation speed in each step

$$= [\text{Time constant without multi-step acceleration/deceleration}] * [\text{magnification of time constant in each step}] \\ * [\text{Rate of rotation speed width in each step respect to rotation speed width up to limit rotation speed}]$$

Note

(1) When the "A" address is designated at G114.1 command, the time is obtained with the "A" address instead of "spt" in the formula below.

Time required to rotate to sptc1 set rotation speed from stopped state (a)

$$= \text{spt} * \text{sptc1} / \text{slimit}$$

Time required to reach sptc2 set rotation speed from sptc1 (b)

$$= \text{spt} * \text{spdiv1} * (\text{sptc2} - \text{sptc1}) / \text{slimit}$$

Time required to reach sptc3 set rotation speed from sptc2 (c)

$$= \text{spt} * \text{spdiv2} * (\text{sptc3} - \text{sptc2}) / \text{slimit}$$

Time required to reach sptc4 set rotation speed from sptc3 (d)

$$= \text{spt} * \text{spdiv3} * (\text{sptc4} - \text{sptc3}) / \text{slimit}$$

Time required to reach sptc5 set rotation speed from sptc4 (e)

$$= \text{spt} * \text{spdiv4} * (\text{sptc5} - \text{sptc4}) / \text{slimit}$$

Time required to reach sptc6 set rotation speed from sptc5 (f)

$$= \text{spt} * \text{spdiv5} * (\text{sptc6} - \text{sptc5}) / \text{slimit}$$

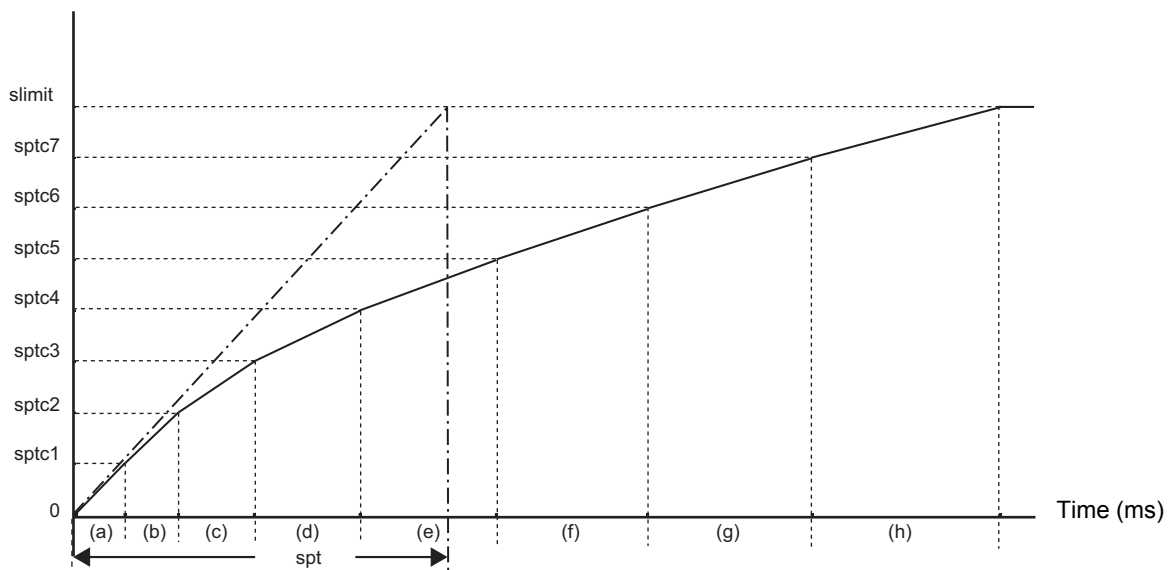
Time required to reach sptc7 set rotation speed from sptc6 (g)

$$= \text{spt} * \text{spdiv6} * (\text{sptc7} - \text{sptc6}) / \text{slimit}$$

Time required to reach slimit set rotation speed from sptc7 (h)

$$= \text{spt} * \text{spdiv7} * (\text{slimit} - \text{sptc7}) / \text{slimit}$$

Rotation speed (r/min)



To decrease the number of acceleration/deceleration steps during spindle synchronization, set one of the following for the unnecessary step.

- Magnification for time constant changeover speed (spdiv7 to spdiv1) = 0 (or 1)
- Spindle synchronous multi-step acceleration/deceleration changeover speed (sptc7 to sptc1) = Limit rotation speed (slimit) or higher



Relationship with other functions

Spindle orientation signal (ORC)

- ♦The spindle orientation is carried out with the spindle orientation command for the reference spindle while the spindle synchronization status remains kept. The spindle orientation command is ignored for the synchronized spindle. The multi-step orientation command or indexing command is also enabled.
- ♦The spindle orientation command (ORC) for the reference spindle in the C axis mode is ignored during spindle synchronization. Also, the spindle position control command (C axis mode switch command) under spindle orientation is ignored.

The spindle orientation signal operation depends on the MTB specifications. Refer to the instruction manual issued by the MTB for details.

Switching the spindle gear

- ♦Gear switching is enabled while the reference spindle is in the spindle mode.
- ♦Gear switching is disabled while the reference spindle is in the C axis mode or in process of spindle orientation. Switching to the C axis mode or the spindle orientation cannot be carried out during gear switching. After gear switching has been completed, the mode is switched to the C axis mode.

Spindle override

- ♦The reference position return operation at C axis mode switch command for the reference spindle and the spindle override in the C axis mode are invalid.
- ♦The cutting feed override or rapid traverse override of the NC axis is valid in the C axis mode.
- ♦The spindle override is invalid while the reference spindle is in process of spindle orientation or spindle indexing.

Spindle clamp speed setting

- ♦The maximum clamp rotation speed specified with the address S following G92 is valid for the reference or synchronized spindle.
- ♦The minimum clamp rotation speed specified with the address Q following G92 is valid for the reference spindle, but invalid for the synchronized spindle. If the rotating spindle is set to the synchronized spindle at the minimum clamp rotation speed, the minimum rotation speed clamp is canceled, and the spindle rotates at the commanded rotation speed. If the spindle synchronization control state is canceled, the minimum clamp rotation speed is enabled.

Synchronous tapping cycle

- ♦The synchronous tap spindle cannot be commanded as the reference spindle or synchronized spindle of the spindle synchronization I. The operation error (M01 1007) will occur, which causes automatic operation to pause.
- ♦You cannot command a synchronous tapping that uses the reference spindle or synchronized spindle of spindle synchronization I. The operation error (M01 1139) will occur, which causes automatic operation to pause.



Precautions

- (1) The spindle rotating with spindle synchronization control will stop when emergency stop is applied.
- (2) The rotation speed clamp during spindle synchronization will follow the smaller clamp value set for the reference spindle or synchronized spindle.
- (3) Orientation of the reference spindle and synchronized spindle is not possible during the spindle synchronization mode. To carry out orientation, cancel the spindle synchronization mode first.
- (4) The rotation speed command (S command) is invalid for the synchronized spindle during the spindle synchronization mode. Note that the modal will be updated, so this command will be validated when spindle synchronization is canceled.
- (5) The constant surface speed control is invalid for the synchronized spindle during the spindle synchronization mode. Note that the modal will be updated, so this will be validated when spindle synchronization is canceled.
- (6) The rotation speed command (S command) and constant surface speed control for the synchronized spindle will be validated when spindle synchronization is canceled. Thus, the synchronized spindle may carry out different operations when this control is canceled.
- (7) If the phase difference is not obtained with the phase shift calculation request signal and the phase synchronization command is executed by turning the phase offset request signal ON, the phase shift amount will not be calculated correctly.
- (8) The spindle Z phase encoder position parameter "#3035 sppst" is invalid when using the spindle synchronization phase shift amount calculation function. (It is ignored.)
The spindle Z phase encoder position parameter "#3035 sppst" is valid when the phase offset request signal is OFF.
- (9) If the phase synchronization command (command with R address) is issued while the phase shift calculation request signal is ON, an operation error (1106) will occur.
- (10) If the phase shift calculation request signal is ON and the reference spindle or synchronized spindle is rotation while rotation synchronization is commanded, an operation error (1106) will occur.
- (11) If the phase synchronization command R0 (<Ex.> G114.1 H1 D-2 R0) is commanded while the phase offset request signal is ON, the reference spindle and synchronized spindle phases will be aligned to the phase difference of the reference spindle and synchronized spindle saved in the NC memory.
- (12) If a value other than the phase synchronization command R0 (<Ex.> G114.1 H1 D-2 R100) is commanded while the phase offset request signal is ON, the phase difference obtained by adding the value commanded with the R address command to the phase difference of the reference spindle and synchronized spindle saved in the NC memory will be used to align the reference spindle and synchronized spindle.
- (13) The phase offset request signal will be ignored when the phase shift calculation request signal is ON.
- (14) The phase difference of the reference spindle and synchronized spindle saved in the NC is valid only when the phase shift calculation signal is ON and for the combination of the reference spindle selection (H_) and synchronized spindle selection (D_) commanded with the rotation synchronization command (no R address). For example, if the reference spindle and synchronized spindle phase difference is saved as "G114.1 H1 D-2 ;", the saved phase difference will be valid only when the phase offset request signal is ON and "G114.1 H1 D-2 R*** ;" is commanded. If "G114.1 H2 D-1 R*** ;" is commanded in this case, the phase shift amount will not be calculated correctly.
- (15) The reference spindle and synchronized spindle phase difference saved in the NC is held until the next spindle synchronous phase shift calculation (rotation synchronization command is completed with phase shift calculation request signal ON).
- (16) When the spindle synchronization commands are being issued with the PLC I/F method (#1300 ext36/bit7 OFF), a program error (P610) will occur if the spindle synchronization is commanded with G114/G113.1.
- (17) When the spindle-mode servo is used for the reference spindle or the synchronized spindle, the spindle parameter "#13003 SP003" (PGS) and spindle-mode servo parameter "#52203 SV003" (PGN) must be set to the same value between the reference and synchronized spindles. (These settings depend on the MTB specifications.)
- (18) (Only for M800/M80 series) When the spindle synchronization control is commanded and if the start signal of the synchronized spindle is not input, turn the servo ON for the synchronized spindle and accelerate or decelerate the spindle rotation up to the commanded speed of the reference spindle. Whether the linear acceleration/deceleration or multi-step acceleration/deceleration is used as the acceleration/deceleration depends on the MTB specifications (parameter "#1255 set27/bit6").
However, when the synchronized spindle is the reference spindle of guide bushing spindle synchronization or spindle-mode servo, multi-step acceleration/deceleration is applied regardless of the parameter.

18.1.2 Spindle Position Control (Spindle/C Axis Control) under Spindle Synchronization Control



Function and purpose

This function enables the spindle position control (spindle/C axis control) by the reference spindle in spindle synchronization control mode.

The reference spindle can be controlled as the rotary axis while the spindle synchronization status remains kept, and also positioning or interpolation with another NC axis is enabled by issuing the position command (movement command) in the same way as for the NC axis.

There are two methods: PLC signal method and program command method to switch the spindle and rotary axis during spindle synchronization control. The method that is applied for switching depends on the MTB specifications (parameter "#3129 cax_spec/bit0").

This section describes the program command method.

In this manual, the state to control an axis as a spindle is referred to as "spindle mode", and the state to control an axis as a rotary axis as "C axis mode".

For details about the spindle position control (spindle/C axis control) function, refer to "10.4 Spindle Position Control (Spindle/C Axis Control)".

This section also describes considerations to perform the spindle position control (spindle/C axis control) under spindle synchronization, and the status of various PLC signals and restrictions. The status, control method, and operation of the PLC signal depend on the MTB specifications.



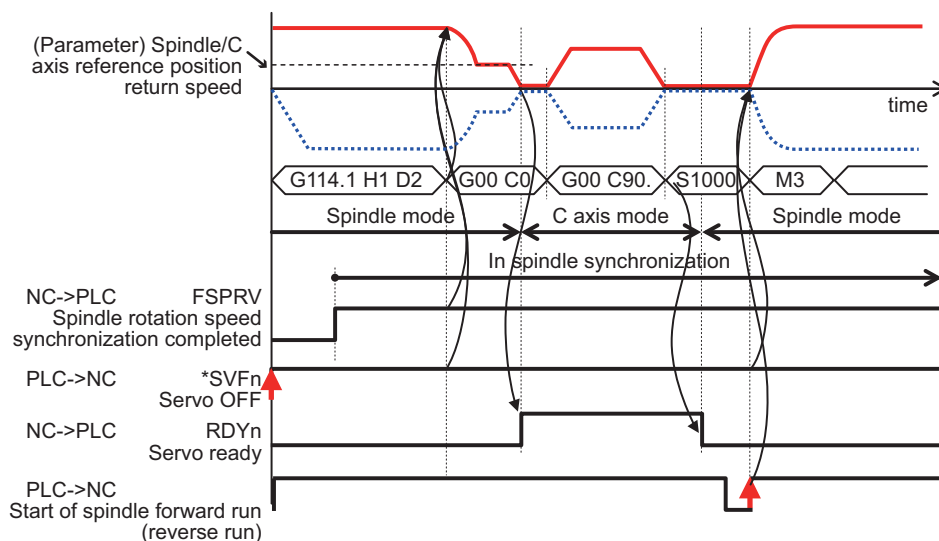
Detailed description

Program command method

The machining program switches the reference spindle to the C axis mode with G00 command, and to the spindle mode with S command. The C axis servo OFF signal (*SVFn) must be kept ON while the program command method is selected.

When the program command method is selected, switching operation is performed only with the reference position return type.

The following shows the C axis switching sequence under spindle synchronization.



<Spindle control mode under spindle synchronization to C axis control mode switching>

- (1) If "G00 C_" is commanded on the program when the reference spindle under spindle synchronization is in the spindle mode, the spindle is directly located at the position commanded by the reference spindle while the synchronous status remains kept.
- (2) Only the G00 command is valid to switch the mode. If the C axis movement is commanded with the G code other than G00, it causes a program error (P430).
- (3) The spindle position control (spindle/C axis control) axis must be commanded with the absolute value address or absolute value command (G90). If the incremental value address or incremental value command (G91) is used, it causes a program error (P32).
- (4) In the switching specifications, only the reference position return type (equivalent to "#3106 zrn_typ/bit8" = 0) is valid, and the direction to return from the rotation mode to the reference position follows the rotation direction (equivalent to "#3106 zrn_typ/bitB" = 1). The direction to return from the spindle stop state to the reference position ("#3106 zrn_typ/bitA-bit9") and interpolation mode selection ("#3106 zrn_typ/bitD-bitE ") follow the appropriate parameters.

[C axis mode switching conditions]

- (1) The C axis servo OFF signal (*SVFn) of the reference spindle is set to ON when switching is commanded.
- (2) The spindle rotation speed synchronization completion signal (FSPRV) is set ON for rotation synchronization, and the spindle phase synchronization completion signal (FSPPH) is set ON for phase synchronization.

<Switching from C axis control mode under spindle synchronization to spindle control mode>

- (1) The switching is performed with the spindle forward run signal (SRN) ON or the spindle reverse run signal (SRI) ON and the S command.
- (2) The switching is performed with the startup of the spindle forward run signal (SRN) or the spindle reverse run signal (SRI).

[Spindle mode switching condition]

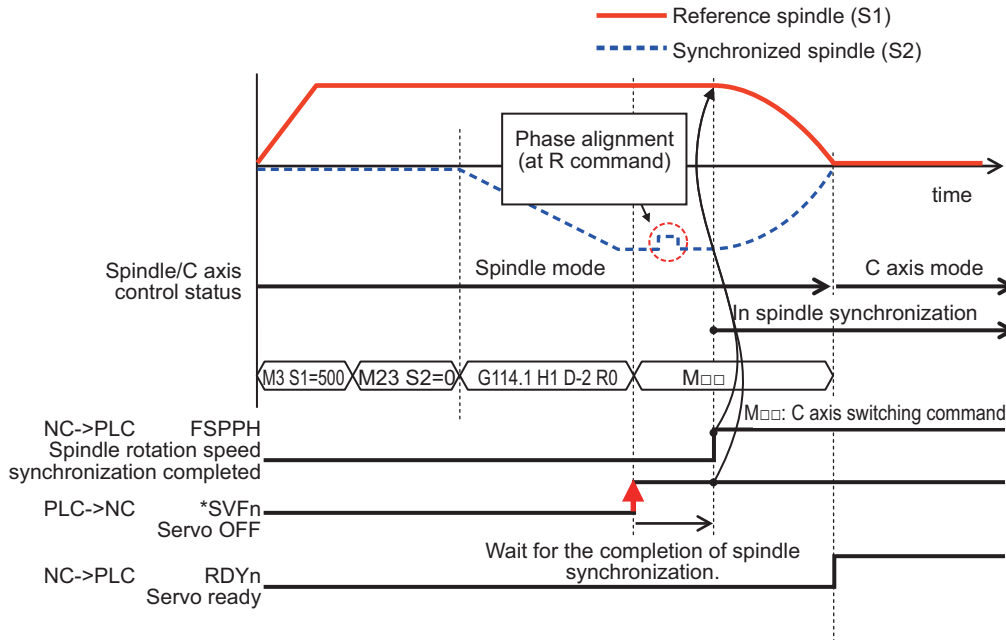
- (1) The C axis servo OFF signal (*SVFn) of the reference spindle is set to ON and the C axis selection signal (CMD) is set to OFF when switching is commanded.

<Operation when the servo OFF signal (*SVFn) of the reference spindle is set off>

- (1) The mode cannot be switched from the spindle mode to the C axis mode or from the C axis mode to the spindle mode.
- (2) In the spindle mode, the axis does not run as a spindle even if the forward run command (SRN) or reverse run command (SRI) is executed.
- (3) In the C axis mode, an operation error (M01 0005) occurs if the movement command is executed. In the servo OFF mode, switching follows the setting of the spindle specification parameter "#1064 svof " (error correction).

C axis mode selection command in spindle synchronization incomplete state

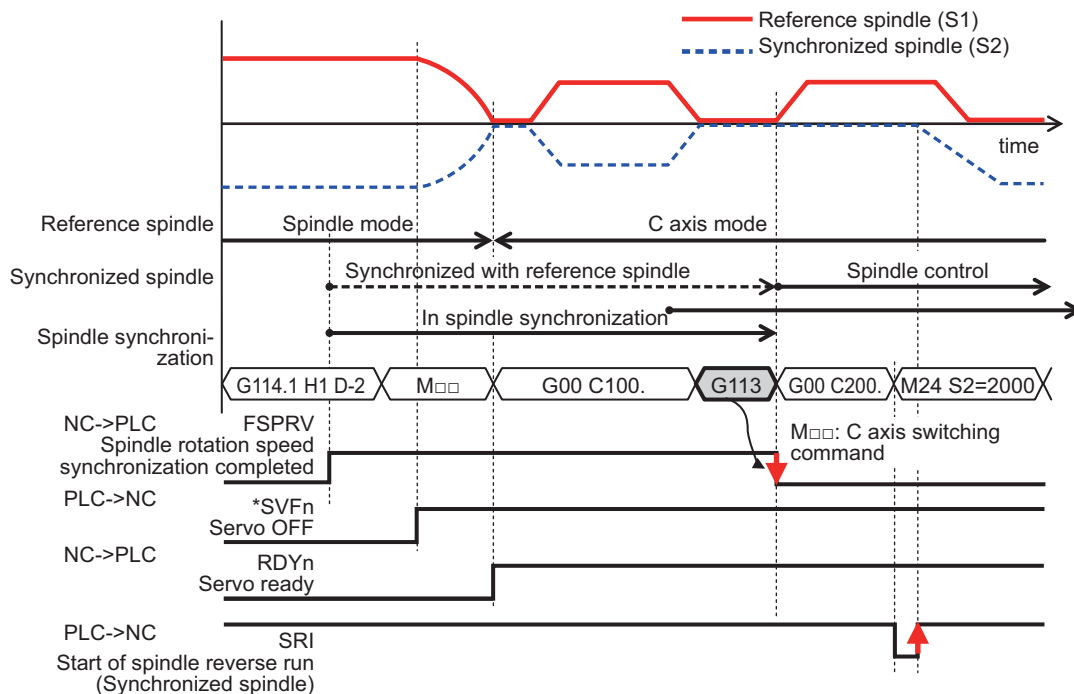
If the C axis mode selection command is issued until the spindle synchronization completion signal (spindle rotation speed synchronization completion (FSPRV) for rotation synchronization and spindle phase synchronization completion (FSPPH) for phase synchronization) is set ON after the spindle synchronization command has been issued, the mode is switch to the C axis mode after the spindle synchronization completion signal has been set ON. The following shows the C axis mode selection command from after the spindle synchronization command has been issued to before the spindle synchronization is completed.



Spindle synchronization cancel in C axis mode

The spindle synchronization is canceled while the reference spindle remains set in the C axis mode by issuing the spindle synchronization cancel command in the C axis mode or axis stop state.

The following shows the spindle synchronization cancel operation in the C axis mode.



- (1) If the spindle is stopped by feed hold or cutting override zero during C axis movement, the spindle synchronization control is canceled with the spindle synchronization cancel command.
- (2) If the C axis is in process of movement when the spindle synchronization cancel command is issued, it causes an operation error (M01 1135), and the spindle synchronization cancel operation is not completed. When the movement of the C axis is completed and the C axis is set to the smoothing zero, the operation error is canceled, and the spindle synchronization cancel operation is completed.

Spindle synchronization command using the spindle in the C axis mode as the reference spindle

- (1) When a spindle that is not in the spindle synchronization state is set in the C axis control mode, if the spindle synchronization command is issued using the spindle as the reference spindle, it causes an operation error (M01 1026).



Relationship with other functions

Function that can be commanded by the spindle position control (spindle/C axis control) under spindle synchronization control

When the reference spindle is in the C axis control state, the synchronized spindle synchronizes with the reference spindle in the spindle control state without being setting in the C axis control state.

Contents of command	Operation
C axis selection command OFF	•The reference spindle is switched to the C axis mode, but the synchronized spindle is kept in the synchronous state.
Spindle synchronization cancel command (G113.1)	•Cancels the spindle synchronization control. •If the spindle synchronization cancel command is issued during movement of the C axis, it causes an operation error (M01 1135), and the spindle synchronization is not canceled. When the operation error is canceled after the C axis has stopped, the spindle synchronization control is also canceled.
Emergency stop	•When the spindle is set in the emergency stop state, the spindle synchronization control is canceled immediately. •If emergency stop occurs in the C axis mode, the reference or synchronized spindle decelerates and stops based on the parameter (spindle parameter "#13056 SP056 EMGt " deceleration time constant at emergency stop) that is set to each spindle.
Tool center point control (G43.4)	•When the reference spindle is in C axis mode, the functions shown on the left can be commanded.
Inclined surface machining command (G68.2)	•If the function shown on the left is commanded when the reference spindle is in spindle mode, a program error (P934) or operation error (M01 0186) occurs.
3-dimensional tool radius compensation (Tool's vertical-direction compensation) (G41.2/G42.2/G40)	
Workpiece installation error compensation (G54.4)	
3-dimensional manual feed	
Rotation center error compensation	

Cautions on spindle position control under spindle synchronization control

- (1) The upper limit of the rotation speed in the C axis mode is controlled by the rapid traverse rate (#2001 rapid) or cutting feed clamp speed (#2002 clamp) of the C axis set to the axis specification parameter; therefore, the rotation speed of the spindle motor may exceed the highest spindle rotation speed depending on the setting of the C axis rapid traverse rate or clamp speed.
The rapid traverse speed (#2001 rapid) and cutting feed clamp speed (#2002 clamp) of the C axis must not be below the highest spindle rotation speed (#3001 slimt1 to #3004 slimt4) of the reference or synchronized spindle (which depends on the MTB specifications).

$$\text{C axis rapid traverse rate (°/min)} \leq \text{Highest spindle rotation speed (r/min)} \times 360(^{\circ})$$

- (2) If the spindle in the C axis mode is set to the reference or synchronized spindle, it causes an operation error (M01 1026). When the control mode at power-on is set to the C axis mode, switch it to the spindle mode once to command the spindle synchronization, and switch it to the C axis mode again.
The control mode at power-on depends on the MTB specifications (parameter "#3129 cax_spec/bit2").
- (3) If the C axis servo OFF signal (*SVFn) of the synchronized spindle is set to ON during spindle synchronization, it causes an operation error (M01 1026).

18.2 Multiple Spindle Synchronization Set Control [C80]



Function and purpose

By this function, multiple sets of spindle synchronization I control can be performed simultaneously. Spindle synchronization I cannot be combined with spindle synchronization II. This section describes spindle synchronization I.

List of available combinations

G114.1: Spindle synchronization I command
G84: Synchronous tapping

[Ref] indicates "reference axis" or "reference spindle".

[Sync] indicates "synchronized axis" or "synchronized spindle".

This is also applied to spindle synchronization II.

	Spindle synchronization command for the 2nd set		Spindle synchronization command for the 1st set	
	[Ref]	[Sync]	G114.1 S1 [Ref] - S2 [Sync]	G84 S1
G114.1	S3	S4	○	○
	S1	S4	○	×(*3)
	S3	S2	×(*1)	○
	S2	S3	×(*2)	○
	S3	S1	×(*2)	×(*3)
G84	S1		×(*3)	×(*3)
	S2		×(*3)	○
	S3		○	○

(*1) The synchronized spindle of the 1st set overlaps with that of the 2nd set.

(*2) The reference spindle of the 1st set overlaps with the synchronized spindle of the 2nd set. Or the synchronized spindle of the 1st set overlaps with the reference spindle of the 2nd set.

(*3) The reference spindle or synchronized spindle of the 1st or 2nd set overlaps with the synchronous tap spindle.

Enabling conditions

- (1) This function can be applied to a lathe system equipped with two or more spindles.
- (2) The validity of this function depends on the MTB specifications. (Parameter "#1440 multi_sp_syn")
If this function is invalid, multiple spindle synchronization commands cannot be issued. (If two or more spindle synchronization commands are issued, the operation error (M01 1005) occurs, which causes automatic operation to pause.)



Command format

For details about the command format to start each function, refer to the relevant function's section.

Canceling Spindle Synchronization

(1) Command to cancel all synchronization modes

Cancels all the synchronized sets. The format varies depending on the MTB specifications (parameter "#1242 set14/bit6").

[When "#1242 set14/bit6" = 0]

G113.1 D0; Cancels all the currently executed spindle synchronization I commands.

Note

- If one set of spindle synchronization is active, G113.1 (without D_ command) is able to cancel the spindle synchronization control. However, if two or more sets of Spindle synchronization are active, the command causes the operation error (M01 1135) to occur.

[When "#1242 set14/bit6" = 1]

G113.1 ;

Note

- If "G113.1 D0;" is commanded, a program error (P35) occurs.

(2) Spindle synchronization I cancel command

G113.1 D_;

Address	Meaning of address	Command range	Remarks
D	Synchronization axis to be canceled Specify the number or name of the spindle that serves as the synchronized spindle in the Spindle synchronization I. (*1)	Spindle No.: 1 to number of spindles Spindle name: 1 to 9	<ul style="list-style-type: none"> •If the command range is exceeded, the program error (P35) will occur. •If you specify a non-existent spindle, the program error (P35) occurs. •If you specify a spindle that is not under synchronization, the operation error (M01 005) occurs. •If there is no address D, the synchronization status of all spindle synch sets is cancelled.

(*1) There are two types of spindle designation methods: Spindle number method and spindle name method. When any name (1 to 9) is set to the spindle name parameter "#3077 Sname" of all the spindles, the spindle name method takes effect. In other cases, the spindle number is used.

(*2) Spindle synchronization I can be canceled by the G113.1 D_ command regardless of whether the multiple spindle synchronization set control is valid or invalid.



Operation example

The function combination table in the operation example shows each function as follows.

G114.1: Spindle synchronization I command

G84: Synchronous tap

- (1) If there is no overlap among the spindles of each spindle synchronization set, the multiple synchronization sets can be controlled simultaneously.

Example: Combinations when the commanded reference spindle or synchronized spindle is not included in other control sets

Spindle synchronization command for the 2nd set	Spindle synchronization command for the 1st set
	G114.1 (S1-S2)
G114.1 (S3-S4)	○

S1, S2, S3, S4: S command name

C1, C3: Name of rotary axis

- (2) If the reference spindle of one spindle synchronization set overlaps with that of another synchronization set, the operation error (M01 1005) occurs. Note, however, that the reference spindle in spindle synchronization mode can be commanded as that for spindle synchronization in another synchronization set.

Example: Combinations when the reference spindle of the 2nd set is controlled as the reference spindle of another set (When S1 is duplicated as the reference spindle)

Spindle synchronization command for the 2nd set	Spindle synchronization command for the 1st set
	G114.1 (S1-S2)
G114.1 (S1-S3)	○

S1,S2,S3,S4: S command name

C1, C3: Name of rotary axis

- (3) If the synchronized spindle to be controlled in each spindle synchronization set overlaps with that of another synchronization set, an operation error (M01 1005) occurs.
- (4) If the reference spindle of one spindle synchronization set overlaps with the synchronized spindle of another synchronization set, the operation error (M01 1005) occurs.



Relationship with other functions

Spindle synchronization II

[M800/M80]

Spindle synchronization II does not support the multiple spindle synchronization set control. Thus, output the PLC output signal of the 1st spindle regardless of whether the multiple spindle synchronization set control is valid or invalid.

[C80]

- (1) Multiple spindle synchronization set control can be used with spindle synchronization II. However, Spindle synchronous control I and II cannot be used simultaneously.
- (2) For Spindle synchronous control II, if the parameter "#1440 multi_sp_syn" is set to "0", the synchronization is performed by using the setting value of the 1st spindle of the PLC interface. In this case, the values set to spindles other than the 1st spindle are not used.

Synchronous tapping cycle

In the following cases, the operation error will occur, which causes automatic operation to pause.

- (1) A synchronous tapping spindle cannot be commanded as the reference spindle of Spindle synchronization I. (Operation error (M01 1007))
- (2) A synchronous tapping spindle cannot be commanded as the synchronized spindle of Spindle synchronization I. (Operation error (M01 1007))
- (3) You cannot command a synchronous tapping that uses the reference spindle of spindle synchronization I mode. (Operation error (M01 1139))
- (4) You cannot command a synchronous tapping that uses the synchronized spindle of spindle synchronization I mode. (Operation error (M01 1139))



Precautions

- (1) If any of the following commands is issued to a spindle for which the cancel operation is being processed by the (G113.1) command or the spindle synchronization/superimposition cancel signal (SPSYC), the operation error (M01 1005) occurs.
 - Spindle synchronization
 Note that the commanded spindle synchronization operation starts after the cancel process is completed.
- (2) If the cancel command (G113.1D_) is issued to the spindle that is not under synchronous control, an operation error (M01 1005) occurs.
- (3) If the "spindle synchronization/superimposition cancel" signal (SPSYC) turns ON for a spindle that is not under synchronization control, the control ignores the cancel signal.
- (4) An axis that involves any travel cannot be put in the same block as the spindle synchronization cancel command (G113.1). Doing so causes a program error (P33) when the spindle synchronization cancel command is issued, which causes automatic operation to be paused.

Advanced Machining Control

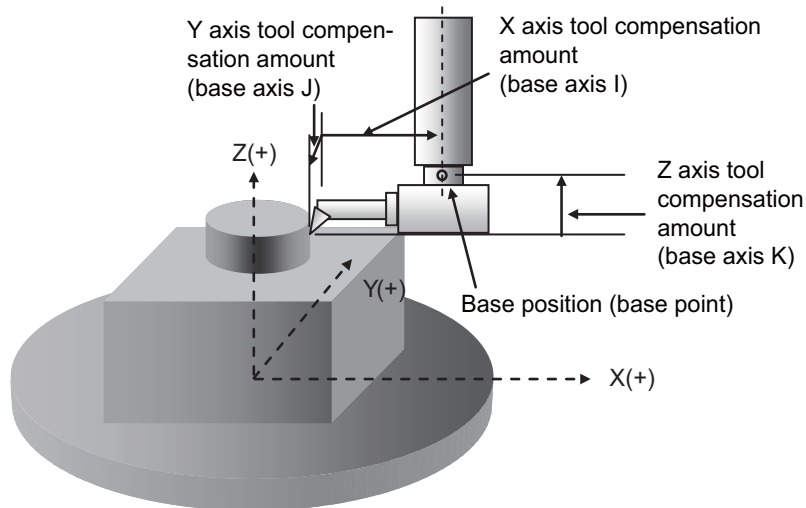
19.1 Tool Position Compensation; G43.7/G49



Function and purpose

The position compensation of a turning tool is executed when turning is performed in a machine of machining center system. Use of the tool position compensation enables the three base axes (X, Y and Z axes) to be compensated from the tool base position (base point).

To set the compensation amount of the three base axes, switch the tool compensation display type to tool compensation type III. The validity of this parameter depends on the MTB specifications (parameter "#1046 T-ofs disp type").



The tool position compensation function is valid for machining center compensation type II. This setting depends on the MTB specifications (parameter "#1037 cmdtyp").



Command format

Tool position compensation start

`G43.7 H__;`

H	Compensation No. (H0 cancels tool position compensation.)
---	---

Tool position compensation cancel

`G49;`

The valid range of the compensation No. will differ according to the specifications (No. of compensation sets). If the commanded compensation No. exceeds the specification range, the program error (P170) will occur. The H address can be omitted. If omitted, the previously specified compensation No. is used.

Note

- (1) Do not omit the H address. If the H address is omitted, an unintended operation may be performed by the H address that is input using a command other than G43.7.
- (2) Even if the H command is issued independently, the compensation amount corresponding to the compensation No. does not become valid. The compensation amount designated by the previous command is applied continuously.
- (3) If G43.7 is commanded with a type other than tool compensation type II of the machining center, the program error (P39) will occur.



Detailed description

Three base axes

The tool position compensation function compensates the tool position for the axis specified in the parameter using the offset specified by the compensation No. The three base axes are determined by the following parameters.

- #1026 base_I (Base axis I)
- #1027 base_J (Base axis J)
- #1028 base_K (Base axis K)

Differences between tool length compensation and tool position compensation

[Tool length compensation (G43/G44)]	[Tool position compensation (G43.7)]
<p>The H address is used to compensate only one axis.</p>	<p>The H address is used to compensate three axis directions.</p>

Start-up and cancel operations

When G43.7 is commanded in the program, tool position compensation is enabled, and the axis moves using the coordinate position, which is obtained by adding the compensation amount specified by the compensation No. to the end point coordinates specified in the movement command of the block, as the end point.

This process is executed regardless of the absolute value command or incremental value command. Then, the compensation amount is added to the end point coordinates specified in the program until tool position compensation is canceled with a G49 command.

Even except when the power is turned ON, G49 mode is set after M02 and M30 have been executed or after resetting has been performed.

When no movement command is included in the same block as for G43.7 or G49, the operation depends on the MTB specifications

(parameter "#1247 set19/bit0" (Movement by tool length offset)). For details, refer to "Movement by tool length compensation".

<p>For absolute command</p> <pre>N1 G91 G28 X0 Y0 Z0 ; N2 G00 G90 ; N3 G43.7 X-20. Y0. Z-40. H01 ; N4 Z-80. N5 G01 X-50. F500 ;</pre>	
<p>For incremental command</p> <pre>N1 G91 G28 X0 Y0 Z0 ; N2 G00 G91 ; N3 G43.7 X-20. Y0. Z-40. H01 ; N4 Z-40. N5 G01 X-30. F500 ;</pre>	

Compensation No.

(1) The compensation No. commanded in the same block as G43.7 will be valid for the following modals.

G43.7 Hh1 ;	Used as the tool compensation amount of (lh1).
:	
G49;	Tool length compensation is canceled.
:	
G43.7;	Used again as the tool compensation amount of (lh1).
:	

(2) When G43.7 is further commanded in G43.7 mode, the compensation is applied by the tool compensation amount commanded later.

G43.7 Hh1 ;	Used as the tool compensation amount of (lh1).
:	
G43.7 Hh2 ;	Used as the tool compensation amount of (lh2).
:	

(3) When the H command is issued independently during G43.7 modal, the compensation amount in modal mode is applied continuously.

G43.7 Hh1 ;	Used as the tool compensation amount of (lh1).
:	
G43.7 Hh2 ;	Used as the tool compensation amount of (lh2).
:	
Hh3 ;	The compensation amount designated in (lh2) is applied continuously.
:	

Tool compensation cancel at reference position return operation

If the reference position return operation is performed, the tool length compensation amount is canceled when the reference position return is completed. However, for the manual high-speed reference position return, the axis can be returned to the coordinates that are shifted by the tool length compensation amount again when the axis is moved after it has reached the reference position, using the parameter. (Parameter "#8122 Keep G43 MDL M-REF")

	Automatic reference position return (G28/G30)	Manual reference position return	
		Dog type	High-speed type
When the axis reached the reference position:	Cancel	Cancel	Cancel
When the axis moves after the above:	Cancel	Cancel	#8122 = 0: Cancel #8122 = 1: Reactivates tool length compensation amount that is applied before the axis reaches the reference position.

(Example 1) Automatic reference position return operation

G43.7 Xx1 Zz1 Hh1 ; :	
G28 Xx2 Zz2 ; :	Canceled when reference position is reached. (Same as when G49 is commanded.)
G01 Xx3 Zz3 Ff3 ; :	Performs the same operation as that in G49 mode.

(Example 2) Manual dog-type reference position return operation (The same operation is also performed when "#8122" is set to "0" and manual high-speed reference position return is valid.)

G43.7 Xx1 Zz1 Hh1 ; :	
(Interrupted by manual dog-type reference position return.)	Canceled when reference position is reached.
G01 Xx2 Zz2 Ff2 ; :	Performs the same operation as that in G49 mode.

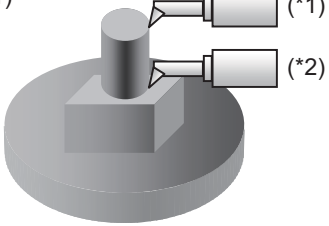
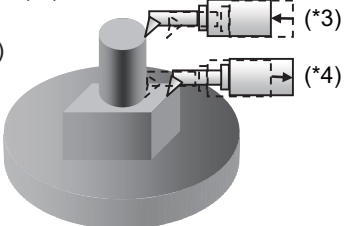
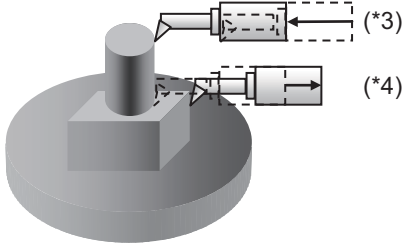
(Example 3) When "#8122" is set to "1" and manual high-speed reference position return is valid:

G43.7 Xx1 Zz1 Hh1 ; :	
(Interrupted by manual high-speed reference position return.)	Canceled when reference position is reached.
G01 Xx2 Zz2 Ff2 ; :	The end point is set for the coordinates that are shifted by the compensation amount specified by compensation No. h1.

The movement is commanded to the G53 machine coordinate system, the axis will move to the machine position when the tool compensation amount is canceled. If the movement command is issued first after G53, the axis returns to the coordinates that are shifted by the tool length compensation amount.

Movement by tool length compensation

If no movement command is included in the same block as for the G43.7 or G49 command, whether the axis moves to the current position by the specified compensation amount when the G43.7 command block is executed is determined depending on the MTB specifications (parameter "#1247 set19/bit0").

G43.7/G49	Not moved. (#1247 set19/bit0=1)	Moved. (#1247 set19/bit0=0)
Independent command : G00 Xx Yy Zz ; G43.7 H1 ; (*1) : G49 ; (*2) :		: G00 Xx Yy Zz ; G43.7 H1 ; (*3) : G49 ; (*4) : 
Movement command included : G00 Xx Yy Zz ; G43.7 H1 X10. ; (*3) : G49 X5. ; (*4) :		
	(*3) Movement by (+) compensation amount (*4) Movement by (-) compensation amount If the tool position compensation and axis movement command are issued in the same block, the axis moves to the end point that is obtained by adding the tool length compensation amount to the movement command.	



Relationship with other functions

Relationship between tool position compensation command and G code function

- Column A : Operation to be performed when the tool position compensation command (G43.7/G49) and another G command are issued to the same block
- Column B : Operation to be performed when another command is issued in G43.7 mode
- Column C : Operation to be performed when G43.7 is commanded in non-G43.7 mode
- : Can be executed.
- : The G43.7 command is ignored.
- P(xx) : The program error will occur.

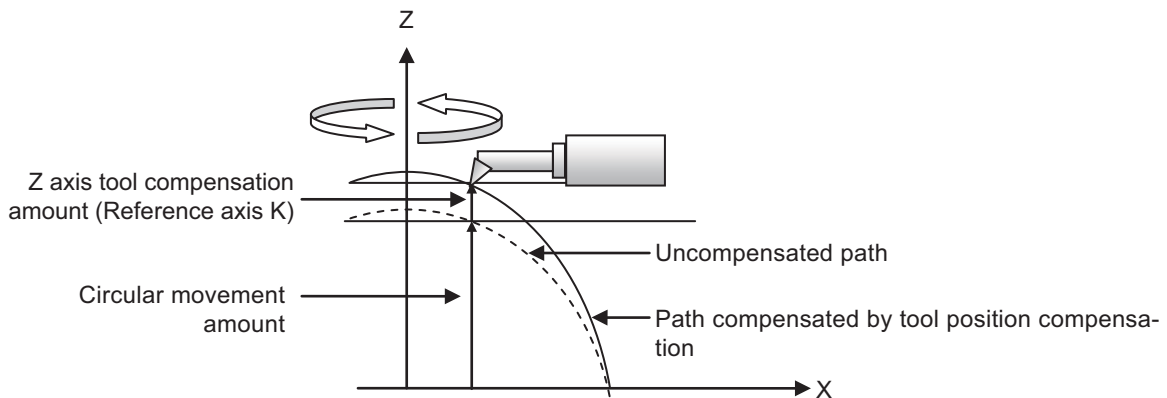
Modal group	G code	Function	A	B	C
0/1	G04	Dwell	P45 (*1)	○	○
	G05	High-speed high-accuracy II	P33	○	○
	G05.1	High-speed high-accuracy I	P34	P34	○
	G07	Hypothetical axis interpolation	P33	○	○
	G08	High-accuracy control	P33	○	○
	G10	Parameter input by program / Compensation data input by program	P45 (*1)	○	○
	G11	Parameter input by program cancel	-	○	○
	G12/G13	Circular cut	P32 (H command only)	○	○
	G27	Reference position check	P45 (*1)	○	○
	G28	Reference position return	P45 (*1)	○ (*5)	○
	G29	Start position return	P45 (*1)	○	○
	G30	2nd to 4th reference position return	P45 (*1)	○	○
	G30.1 - G30.6	Tool exchange position return	-	○	○
	G37	Automatic tool length measurement	P801	P801	○
	G52	Local coordinate system setting	P45 (*1)	○	○
	G53	Machine coordinate system selection	P45 (*1)	○	○
	G53.1/G53.6	Tool axis direction control	P953	○	○
	G65	User macro simple call	P231 (*1)	○	○
	G115/G116	Start point timing synchronization	P32	○	○
G120.1/G121	Machining condition selection I	P33	○	○	
G122	Activate sub part system I	P651, P32 (*2)	○	○	
1	G02/G03	Circular interpolation	P33 (*1)	○	○
	G2.3/G3.3	Exponential interpolation	○	○	P33
	G2.4/G3.4	3-dimensional circular interpolation	P75	P75	P75
	G06.2	NURBS interpolation	P33	○	P32
7	G41.2/G42.2	3-dimensional tool radius compensation (Tool's vertical-direction compensation)	P163	○	P162
8	G43	Tool length compensation (+)	○ (*3)	P801	P801
	G43.1	Tool length compensation along the tool axis ON	○ (*3)	P801	P930
	G43.4/G43.5	Tool center point control	○ (*3)	P941	P942
	G44	Tool length compensation (-)	○ (*3)	P801	P801
	G49	Tool length compensation cancel	○ (*3)	○	○
9	G73 - G76 G81 - G89	Fixed cycle for drilling	P801	○	P801
14	G66	User macro modal call	- (*4)	○	○

Modal group	G code	Function	A	B	C
16	G68	3-dimensional coordinate conversion mode ON	P923	○	○
	G68.2/G68.3	Inclined surface machining command	P954	○	○
19	G50.1	G command mirror image cancel	P801	○	○
	G51.1	G command mirror image ON	P801	P801	P801
21	G7.1/G107	Cylindrical interpolation	P33	○	P481
	G12.1/G112	Polar coordinate interpolation ON	P33	○	P481
	G13.1/G113	Polar coordinate interpolation cancel	P33	○	○
27	G54.4	Workpiece installation error compensation	P546	P546	○

- (*1) When the parameter "#1241 set13" is set to "1", G43.7 is ignored.
- (*2) If G122 or G144 is called before G43.7, the program error (P651) will occur. If it is called after G43.7, the program error (P32) will occur.
- (*3) When "G43.7 G43 H1;" is commanded, the G43 commanded later is enabled.
- (*4) Only the modal is updated.
- (*5) If the reference position return (G28) is commanded during the G43.7 modal, the G43.7 modal is canceled when the return is completed.

Circular interpolation

When the compensation by tool position compensation command G43.7/ G49 is applied to the circular movement axis, compensation movement is superimposed with circular movement if the axis moves by the specified compensation amount in the circular command block.



19.2 Tool Length Compensation Along the Tool Axis; G43.1/G49



Function and purpose

(1) Changes in the tool length compensation in the tool axis direction and compensation amount

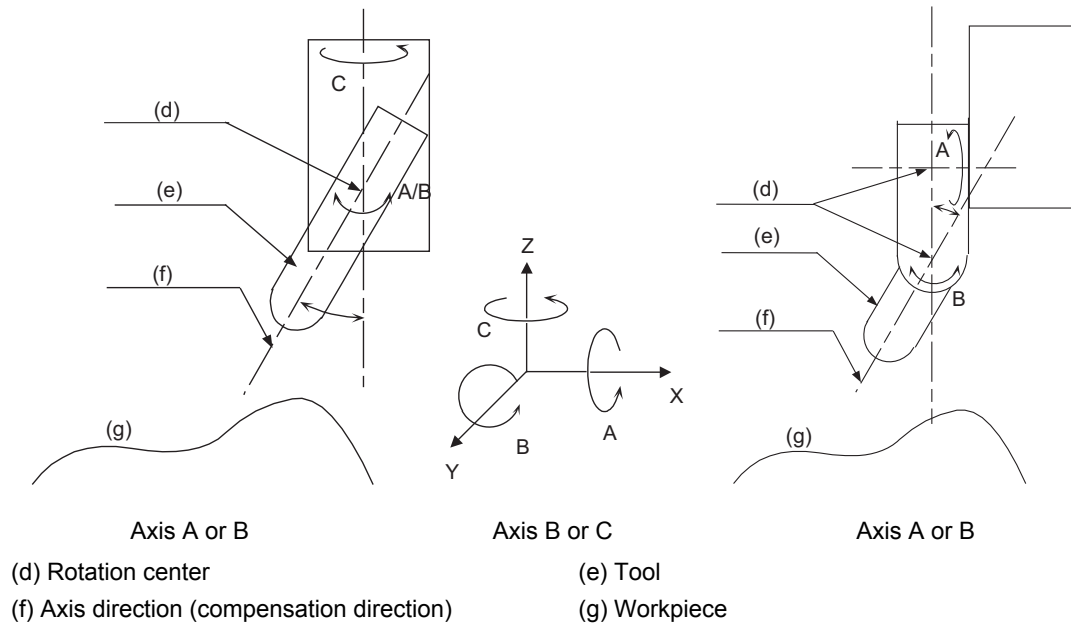
The tool length can be compensated for in the tool axis direction even when the rotary axis rotates and the tool axis direction becomes other than the Z axis direction. By using this function, and setting the deviation between the tool length amount set in the program and the actual tool length as the compensation amount, a more flexible program can be created. This is especially valid for programs in which many axis movement commands are present.

The tool length compensation amount in the tool axis direction can be changed by rotating the manual pulse generator when the tool length compensation amount in the tool axis direction is being changed during the tool length compensation in the tool axis direction mode.

(2) Machine configuration

The compensation using the tool length compensation in the tool axis direction function is applied to the direction of the tool tip axis (rotary axis).

As for the axes that determine the compensation direction, a combination of the C axis (spindle) for Z axis rotation and the A axis for X axis rotation or B axis for Y axis rotation is designated using a parameter.





Command format

Tool length compensation along the tool axis ON

```
G43.1 X__ Y__ Z__ H__ ;
```

Tool length compensation cancel

```
G49 X__ Y__ Z__ ;
```

X, Y, Z	Movement data
H	Tool length compensation No. (If the compensation No. exceeds the specification range, a program error (P170) will occur.)



Detailed description

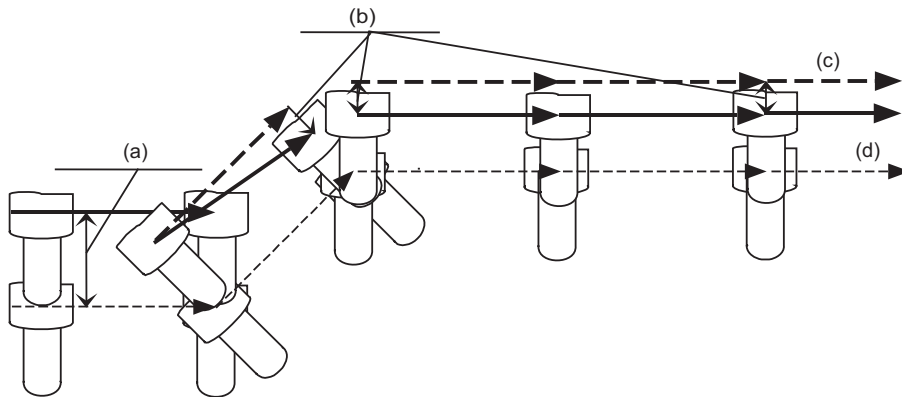
- (1) G43, G44 and G43.1 are in the same G code group. Therefore, it is not possible to designate more than one of these commands simultaneously for compensation. G49 is used to cancel the G43, G44 and G43.1 commands.
- (2) If the G43.1 command is designated when the specification for the tool length compensation in the tool axis direction is not provided, the program error (P930) will occur.
- (3) If reference position has not been completed for any of the X, Y, Z, A or B and C axes in the G43.1 block, the program error (P430) will occur. However, the error does not apply to the following cases.
 - When mechanical axes have been selected
The error does not apply to the A, B and C axes.
 - When "1" has been set for the "#2031 noref" zero point return parameter
The error does not apply to the axis for which "noref" is set to "1" because it is considered that the reference position return of the axis has already completed.

Changing the amount of tool length compensation in the tool axis direction

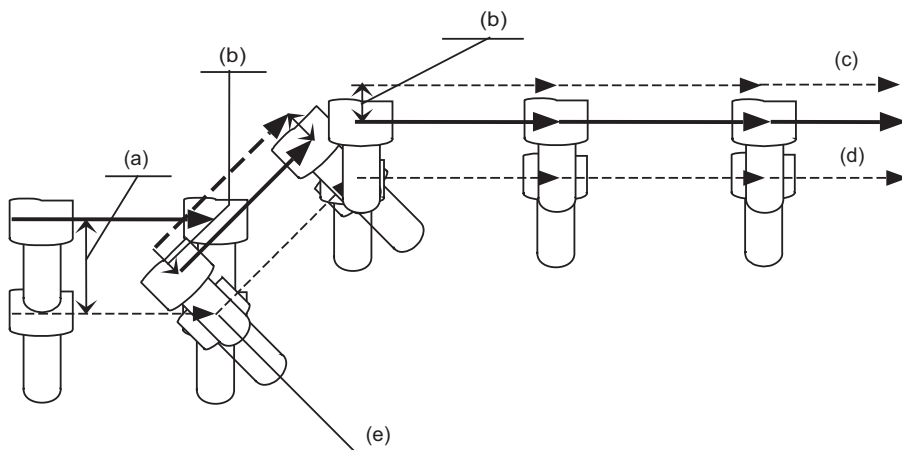
- (1) When the following conditions have been met, the handle movement amount is added to the tool length compensation amount in the tool axis direction by rotating the manual pulse generator.
- When the operation mode is MDI, memory or tape operation mode and the state is "during single block stop", "during feed hold" or "during cutting feed movement". Note that compensation amount cannot be changed during error or warning.
 - During tool length compensation in the tool axis direction (G43.1).
 - In the tool length compensation amount in the tool axis direction changing mode (YC92/1).
 - In the tool handle feed & interruption mode (YC5E/1).
 - The 3rd axis (tool axis) is selected for the handle selection axis.
- (2) The change amount is canceled when the compensation No. is changed.

Note

- The coordinate value in the tool length compensation amount in the tool axis direction change mode operates in the same manner as that when the manual ABS is ON, regardless of manual ABS switch (YC28) or base axis specification parameter "#1061 intabs".
 - If compensation amount is changed during continuous operation, single block stop, or feed hold, the compensation amount will be effective immediately in the next block.
- (Example) When changing compensation amount during continuous operation.



(Example) When changing compensation amount during single block stop.



- (a) Compensation amount before change
 (c) Path after compensation
 (e) Single block stop

- (b) Changed compensation amount
 (d) Program path

- When changing compensation amount, the compensation amount corresponding to the actual compensation No. will be changed. However, when executing the NC reset or tool length compensation in the direction of tool axis cancel (G49), the compensation amount will be returned to the original.

Tool length compensation in the tool axis direction vector

The vectors representing the tool length compensation in the tool axis direction are as follows.

- (1) When the A and C axes are set as the rotary axes:

$$\begin{aligned} V_x &= L * \sin(A) * \sin(C) \\ V_y &= -L * \sin(A) * \cos(C) \\ V_z &= L * \cos(A) \end{aligned}$$

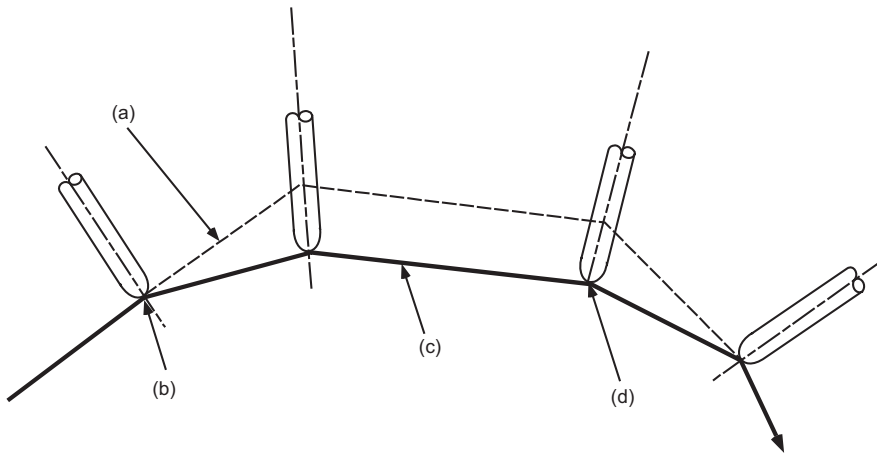
- (2) When the B and C axes are set as the rotary axes:

$$\begin{aligned} V_x &= L * \sin(B) * \cos(C) \\ V_y &= L * \sin(B) * \sin(C) \\ V_z &= L * \cos(B) \end{aligned}$$

V_x, V_y, V_z : Tool length compensation along the tool axis vectors for X, Y and Z axes

L: Tool length compensation amount (1h)

A, B, C: Rotation angle (machine coordinate position) of A, B and C axes



(a) Path after tool length compensation in the tool axis direction

(b) G43.1 command

(c) Program path

(d) G49 command

- (3) Rotary axis angle command

The value used for the angle of the rotary axis (tool tip axis) differs according to the type of rotary axis involved.

When servo axes are used:

The machine coordinate position is used for the rotation angles of the A, B and C axes.

When mechanical axes are used:

Instead of the machine coordinate position of the axes, the values read out from the R registers (R2628 to R2631) are used for the rotation angles of the A, B and C axes.

Compensation amount resetting

Tool length compensation in the tool axis direction is cleared in the following cases.

- (1) When manual reference position return is completed.
- (2) When reset 1, reset 2 or reset & rewind has been executed.
- (3) When the G49 command has been designated.
- (4) When the offset No. 0 command has been executed.
- (5) When NC reset has been executed with "1" set for the basic system parameter "#1151 rstint".
- (6) When the G53 command is designated while the compensation status is still established, the compensation is temporarily canceled, and the tool moves to the machine position designated by G53.



Program example

Example of arc machining

Shown below is an example of a program for linear -> arc -> arc -> linear machining using the B and C rotary axes on the ZX plane.

Machining program

```

N01 G91 G28 X0 Y0 Z0 ;
N02 G28 B0 C0 ;
N03 G90 G54 G00 X400. Y0 ;
N04 Z-150. ;
N05 B90. ;
N06 G18 ;
N07 G43.1 X250. H01 ;
N08 G01 Z0 F200 ;
N09 G02 X0 Z250. I-250. K0 B0 ;
N10 G02 X-250. Z0 I0 K-250. B-90. ;
N11 G01 Z-150. ;
N12 G00 G49 X-400. ;
N13 G91 G28 B0 C0 ;
N14 G28 X0 Y0 Z0 ;
N15 M02 ;

```

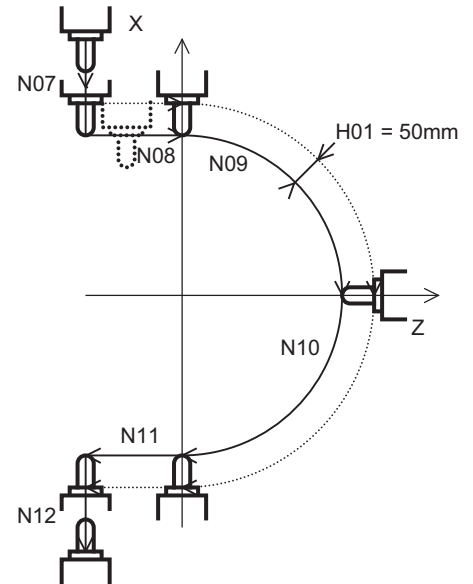
Compensation amount H01
= 50 mm




B axis: 90°

Tool length compensation in
the tool axis direction ON

Top right arc, B axis: 0°
Bottom right arc, B axis: -90°

Tool length compensation in
the tool axis direction OFF



-  Tool with no compensation
-  Program path
-  Path after compensation



Relationship with other functions

Relationship with 3-dimensional coordinate conversion

- (1) A program error (P931) will occur if 3-dimensional coordinate conversion is carried out during tool length compensation in the tool axis direction.
- (2) A program error (P921) will occur if the tool length is compensated for in the tool axis direction during 3-dimensional coordinate conversion.
- (3) A program error (P923) will occur if the tool length compensation in the tool axis direction is commanded in the same block as the 3-dimensional coordinate conversion.

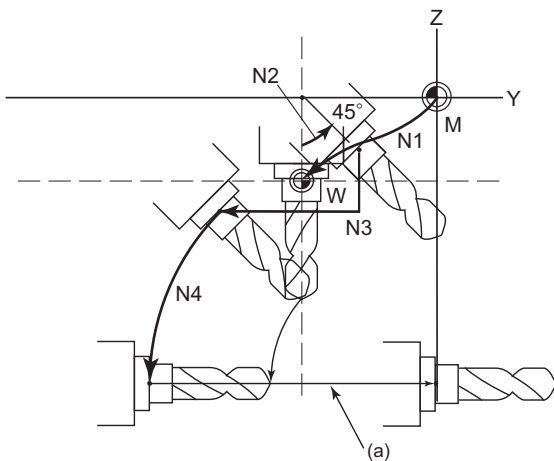
Relationship with automatic reference position return

- (1) A program error (P931) will occur if a command from G27 to G30 is issued during tool length compensation in the tool axis direction.

Relationship with manual reference position return

- (1) Reference position return of orthogonal axis
Tool length compensation along the tool axis will be canceled, as well as the dog-type reference position return and the high-speed reference position return.

<Y axis Manual reference position return>



```

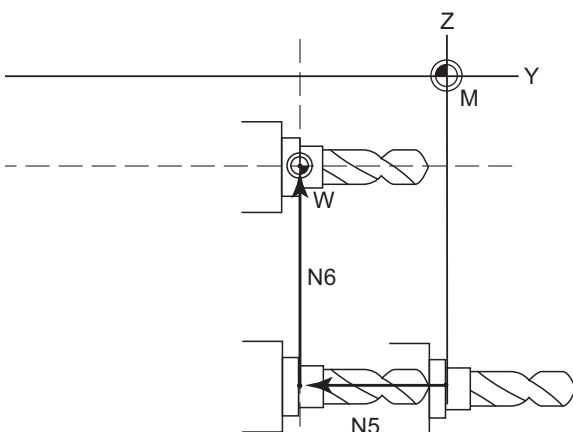
N1 G90 G00 G54 X0 Y0 Z0 ;
    Positioning to the workpiece origin
N2 G00 A45. ;
    Rotating the rotary axis by 45°
N3 G43.1 H1 ;
    Tool length compensation along the tool axis ON
N4 G19 G03 Y-5.858 Z-14.142 J14.142 K-14.142 A90. ;
    Circular cutting
    
```

Manual dog-type reference position return (a)

```

N5 G00 Y0. ;
N6 Z0. ;
:
:
    
```

<Movement after Y axis Manual reference position return>



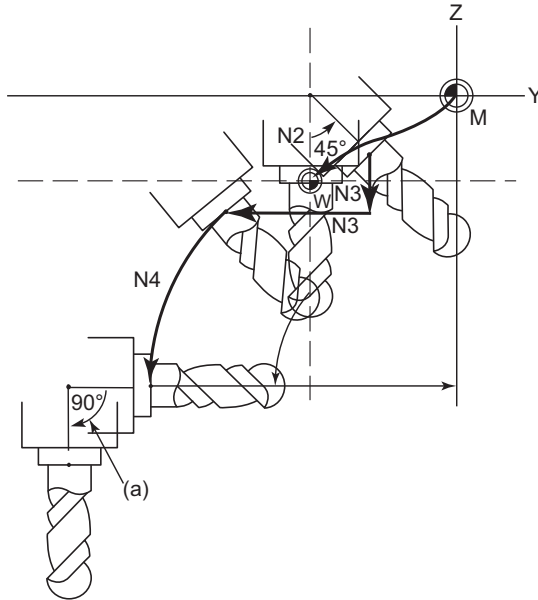
```

N5 G00 Y0. ;
    Positioning to the position where tool length compensation
    along the tool axis was canceled.
N6 Z0. ;
    Positioning to the position where tool length compensation
    along the tool axis was canceled.
:
:
    
```

(2) Reference position return of rotary axis

Tool length compensation along the tool axis will be canceled, as well as the dog-type reference position return and the high-speed reference position return.

<A axis Manual reference position return>



```

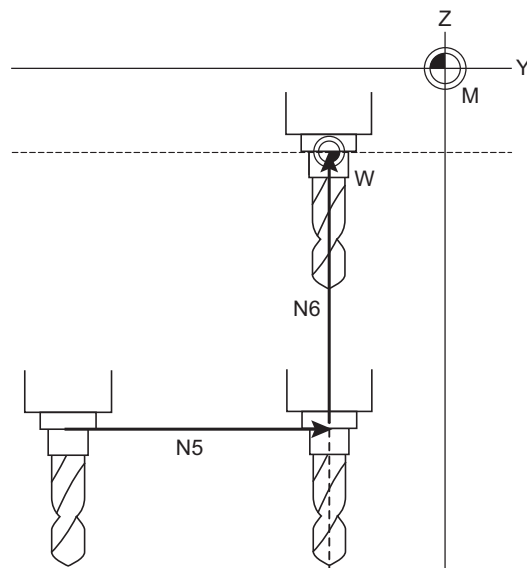
N1 G90 G00 G54 X0 Y0 Z0 ;
    Positioning to the workpiece origin
N2 G00 A45. ;
    Rotating the rotary axis by 45°
N3 G43.1 H1 ;
    Tool length compensation along the tool axis ON
N4 G19 G03 Y-5.858 Z-14.142 J14.142 K-14.142 A90. ;
    Circular cutting
    
```

Manual dog-type reference position return (a)

```

N5 G00 Y0. ;
N6 Z0. ;
:
:
    
```

<Movement after A axis Manual reference position return>



```

N5 G00 Y0. ;
    Positioning to the position where tool length compensation in the tool axis direction was canceled.
N6 Z0. ;
    Positioning to the position where tool length compensation in the tool axis direction was canceled.
:
:
    
```

Relationship with graphic check

(1) Graphic check draws a path after compensation.

19.3 Tool Center Point Control; G43.4, G43.5/G49



Function and purpose

The tool center point control function controls a commanded position described in the machining program to be the tool center point in the coordinate system that rotates together with a workpiece (table coordinate system). This function can be applied for the three types of machine as below.

- (1) A tool tilt type: a machine with two rotary axes set on the head.
- (2) A table tilt type: a machine with two rotary axes set on the table.
- (3) A combined type: a machine with one rotary axis set on the tool and another on the table.

With this function, in the case of using tool tilt type, the tool center point is controlled so that it moves on the programmed path specified on the workpiece coordinate system. In the case of using the table tilt type, the tool center point is controlled so that it moves on the programmed path specified on the table coordinate system (a coordinate system which rotates together with a workpiece).

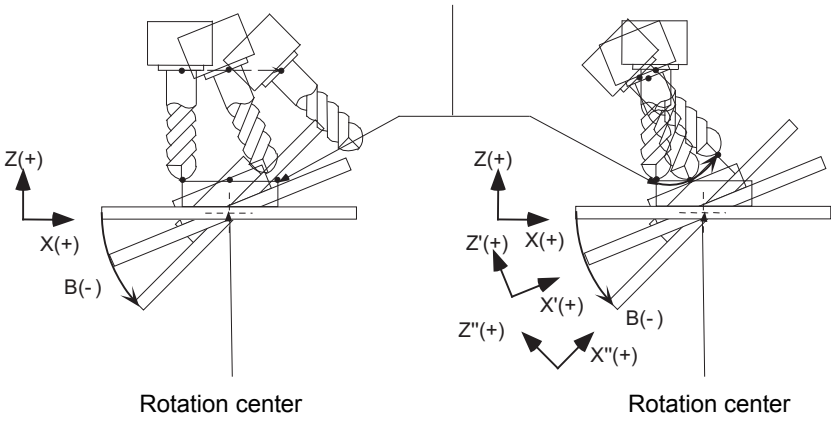
(1) Tool tilt type

Tool center point control OFF and tool length compensation along the tool axis ON	Tool center point control ON
<div style="display: flex; justify-content: space-around;"> Rotation center Program path Rotation center </div>	
<p>Path of the tool center point</p>	
Controls so that the path of the tool holder center point draws a straight line.	Controls so that the tool center point draws a straight line.

(2) Table tilt type

Tool center point control OFF and tool length compensation along the tool axis ON	Tool center point control ON
<div style="display: flex; justify-content: space-around;"> Rotation center Rotation center </div>	
<p>Path of the tool center point</p>	
Controls so that the tool holder center point positions on the workpiece coordinate system.	Controls so that the tool center point positions on the table coordinate system.

(3) Combined type

Tool center point control OFF and tool length compensation along the tool axis ON	Tool center point control ON
Path of the tool center point 	
Controls so that the tool holder center point positions on the workpiece coordinate system.	Controls so that the tool center point positions on the table coordinate system.

If the tool center point control is commanded without the specifications of this function, a program error (P940) will occur.

In addition, 3 orthogonal axes must be commanded first and 2 rotary axes after.

Note

- Set the short-cut valid or linear-type rotary axis to the rotary axis type (parameter "#8213 Rotation axis type") of two rotary axes. When short-cut valid is set, the rotary axis always moves in the positive (+) direction, and the tool center point moves following the movement of the rotary axis, which may cause a machining failure.

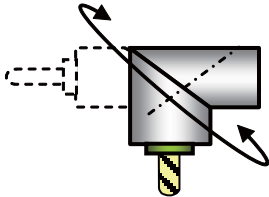
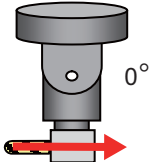
Axis configuration of applicable machines

In the machine coordinate system (right-hand orthogonal coordinate system) defined in ISO standard, the rotation turning to the right-handed screw is handled as ISO polarity and the one turning to the left-handed screw is handles as reverse ISO polarity.

This function can be used on the machine whose rotary axis is installed with ISO polarity and the machine whose axis is installed with reverse ISO polarity. However, the orthogonal axis needs to be installed with ISO polarity. Refer to "Operation when the rotary axis is set with reverse ISO polarity" described later for details.

Note

(1) This function is not applicable to machines as in the table below:

Description	Example of machines
The machine where the rotation center axis of the rotary axis is not parallel to any orthogonal axes.	
When the machine position of all rotary axes is 0°, the machine where the direction from tool's tip to the bottom is not parallel to Z axis (positive direction of Z axis). (The arrow shown in the right figure indicates the direction of the tool axis.)	

When the number of simultaneous contouring control axes is 4

When the number of simultaneous contouring control axes is 4, the tool center point control is only available for the simultaneous movement of 4 axes or less.

Using this function allows you to designate the tool center point position on the table coordinate system (which rotates as the workpiece rotates); therefore, you can easily create a machining program without calculating the workpiece rotation or spindle end point position.

[Restrictions]

	When the number of simultaneous contouring control axes is 5 or more	When the number of simultaneous contouring control axes is 4
Command type	G43.4/G43.5	G43.4 only (*1)
Limitations when commanded	None (Format error, etc. only)	A single rotary axis can be commanded in the same block. (*2)
Interpolation method	Joint interpolation / Single axis rotation interpolation (*3)	Joint interpolation only
Type of passing singular point	Type 1/type 2 (*3)	Invalid (*4)
Program coordinate system selection	Table coordinate system / workpiece coordinate system (selected by parameter) (*3)	
Rotary axis reference position selection	Zero degree position basis / start position basis (*3)	
Rotary axis prefiltering	Select whether this function is valid or invalid with a parameter. Designate the time constant with a parameter.	

(*1) If G43.5 is commanded, a program error (P34) occurs.

(*2) If two rotary axes are commanded, a program error (P10) occurs. However, if a single rotary axis only moves even when two rotary axes are commanded, it is not judged to be erroneous.

(*3) Selected by a parameter.

(*4) Only the joint interpolation is available in G43.4; therefore, the singular point type is invalid.

[Restrictions in movement command]

o: Can be commanded, x: Alarm

3 orthogonal axes or less	1 rotary axis	2 rotary axes	3 orthogonal axes or less + 1 rotary axis	3 orthogonal axes or less + 2 rotary axes
o	o	x	o	x



Command format

There are two command formats: <Type 1>, where tool angle is commanded by the rotary axis; and <Type 2>, where tool angle is commanded by the vectors of the workpiece surface, I, J, and K.

Tool center point control ON

G43.4 (X__ Y__ Z__ A__ C__) H__ ; Type 1 ON

G43.5 (X__ Y__ Z__) I__ J__ K__ H__ ; Type 2 ON (*1)

X,Y,Z	Orthogonal coordinate axis movement command
A,C	Rotary axis movement command
I,J,K	Workpiece surface angle vector
H	Tool length compensation No.

(*1) Can only be commanded when the number of simultaneous contouring control axes is 5 or more.

Note

- (1) When orthogonal coordinate axis movement command or rotary axis movement command is not issued in the same block, start-up will be applied without axis movement (No movement for the compensation amount).
- (2) Commands to I, J, and K will be ignored during the tool center point control type 1.
- (3) Rotary axis movement command cannot be issued during the tool center point control type 2. If commanded, a program error (P33) occurs.
- (4) If I, J, or K is omitted when issuing the tool center point control type 2 command, the omitted address will be considered as "0".

Tool center point control cancel

G49 (X__ Y__ Z__ A__ C__);

Note

- (1) Instead of using G49, other G codes in G code group 8 can be used for canceling.
- (2) If orthogonal coordinate axis command and rotary axis command are issued in the same block as G49, the tool center point control modal will be canceled on the spot. Then, commanded axis movement will be performed. If the cancel command is issued alone, the modal will be canceled on the spot, and yet no axis movement (movement for the compensation amount) will be performed.



Detailed description

Programming coordinate system

Specify the end position of each block looking from the programming coordinate system in the tool center point control mode. In the program, specify the position of the tool center point.

The programming coordinate system is a coordinate system used for the tool center point control, and whether to use the table coordinate system or the workpiece coordinate system depends on the MTB specifications (parameter "#7908 SLCT_PRG_COORD").

(1) Table coordinate system

When "0" is set to the programming coordinate system selection parameter, the table coordinate system, which is the valid workpiece coordinate system at that time fixed to the table, is specified as the programming coordinate system. Table coordinate system rotates along the table rotation. And it does not rotate along the tool axis rotation. The X,Y,Z addresses are considered to have been issued on the table coordinate system.

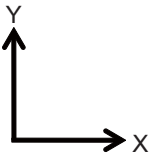
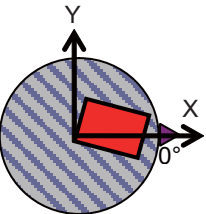
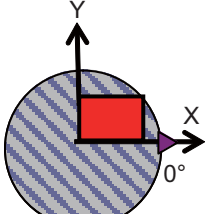
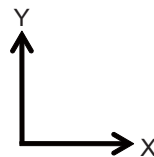
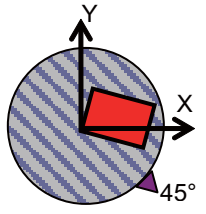
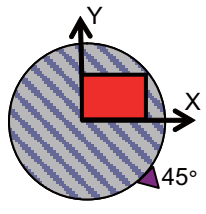
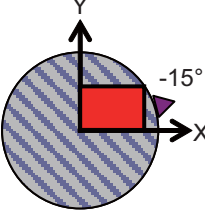
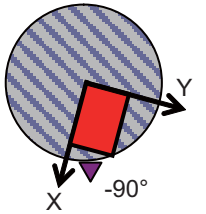
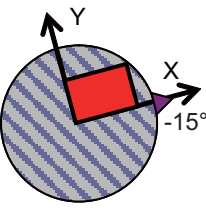
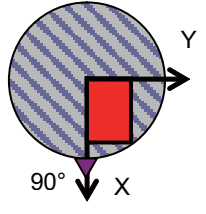
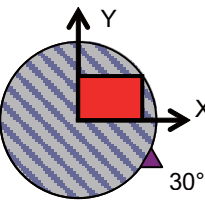
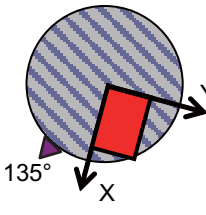
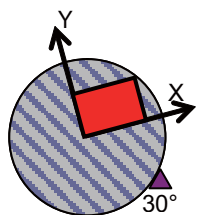
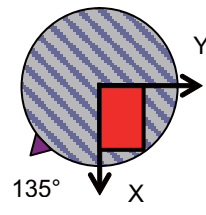
When a rotary axis movement is commanded in a block prior to G43.4/G43.5 command, the angle generated by rotary axis movement is regarded as an initial setting at G43.4/G43.5 command.

(2) Workpiece coordinate system

When "1" is set to the programming coordinate system selection parameter, the valid workpiece coordinate system at that time is specified as the programming coordinate system. The coordinate system in this case does not rotate along the table rotation. A linear movement is carried out for the table (workpiece) when the X,Y,Z addresses are issued. The end position looking from the workpiece coordinate system after table rotation is specified to the X, Y and Z.

Rotary axis reference position selection

When the table coordinate system, the workpiece coordinate system fixed to the table, is to be defined as a programming coordinate system, the appropriate rotary axis angle for fixing the workpiece coordinate system to the table depends on the MTB specifications (parameter "#7911 SLCT_STANDARD_POS").

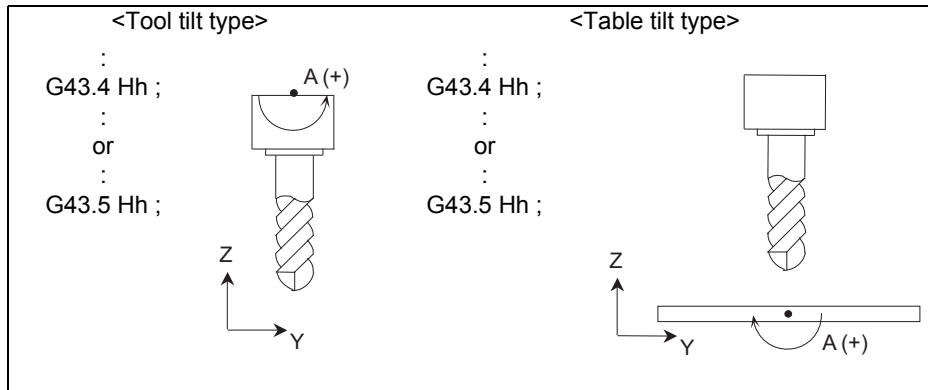
Rotary axis reference position selection	Start position basis (#7911=1)		Zero degree position basis (#7911=0)	
The timing to be fixed to the table	The position of rotary axis holding the workpiece at the start of tool center point control		When the position of rotary axis holding the workpiece is at 0 degree on the workpiece coordinate system	
<p>Operation example 1 (Workpiece coordinate offset 0°)</p>  <p>Machine coordinate system</p>	<p>: G90G54G0C0 C-15 ; G43.4 Hh ; : C90 ; :</p>	<p>Workpiece coordinate 0°</p> 	<p>: G90G54C0 C-15 ; G43.4 Hh ; : C90 ; :</p>	<p>Workpiece coordinate 0°</p> 
<p>Operation example 2 (Workpiece coordinate offset 45°)</p>  <p>Machine coordinate system</p>	<p>: G90G54G0C0 C-15 ; G43.4 Hh ; : C90 ; :</p>	<p>Workpiece coordinate 0°</p> 	<p>: G90G54C0 C-15 ; G43.4 Hh ; : C90 ; :</p>	<p>Workpiece coordinate 0°</p> 
<p>Machine coordinate position fixed at -15°</p> 	<p>Machine coordinate position 90° by C90. command</p> 	<p>Machine coordinate position -15° by C-15. command</p> 	<p>Machine coordinate position 90° by C90. command</p> 	
<p>Machine coordinate position fixed at 30°</p> 	<p>Machine coordinate position 135° by C90. command</p> 	<p>Machine coordinate position 30° by C-15. command</p> 	<p>Machine coordinate position 135° by C90. command</p> 	

Start-up

(1) Independent start-up command

(a) Tool center point control type 1, type 2

When the tool center point control is ON, no axis movement is performed (including movement for the compensation amount).



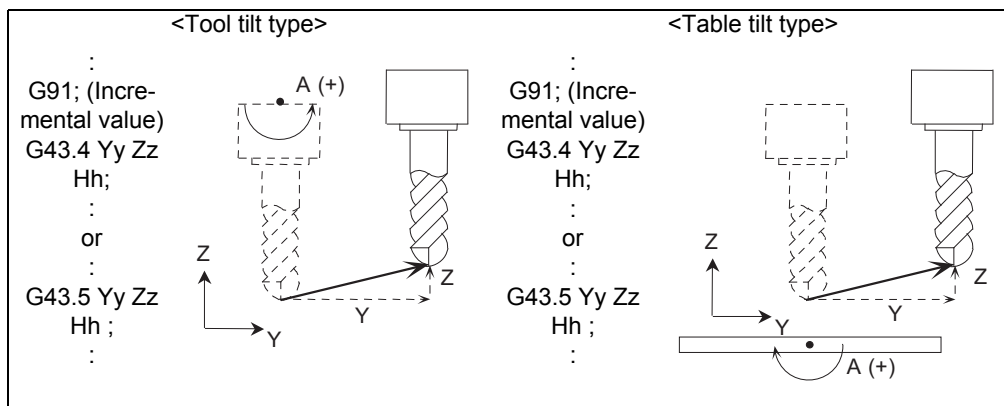
(b) Tool center point control type 2

"G43.5 li Jj Kk Hh ;" performs the same movement as the tool center point control type 1 in (2).

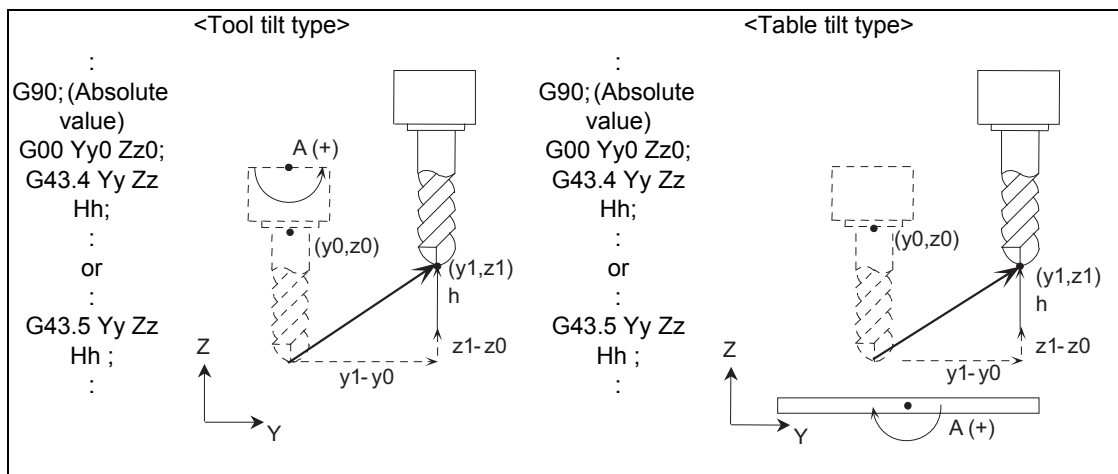
(2) Start-up with movement command (When orthogonal coordinate axis command is issued in the same block)

(a) Tool center point control type 1, type 2

When the tool center point control is ON, the tool center point moves only as much as it is ordered under the incremental value command.

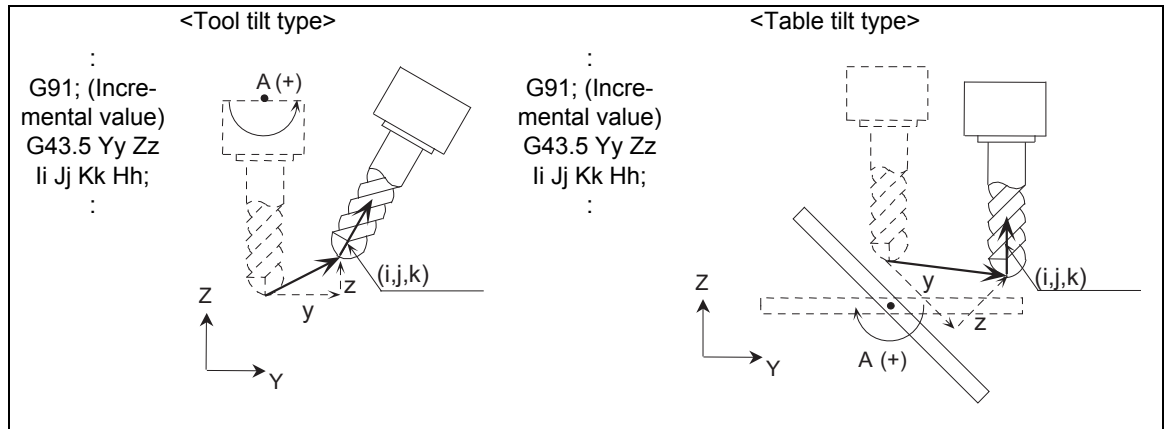


Under the absolute value command, the tool center point moves to y1, z1.



(b) Tool center point control type 2

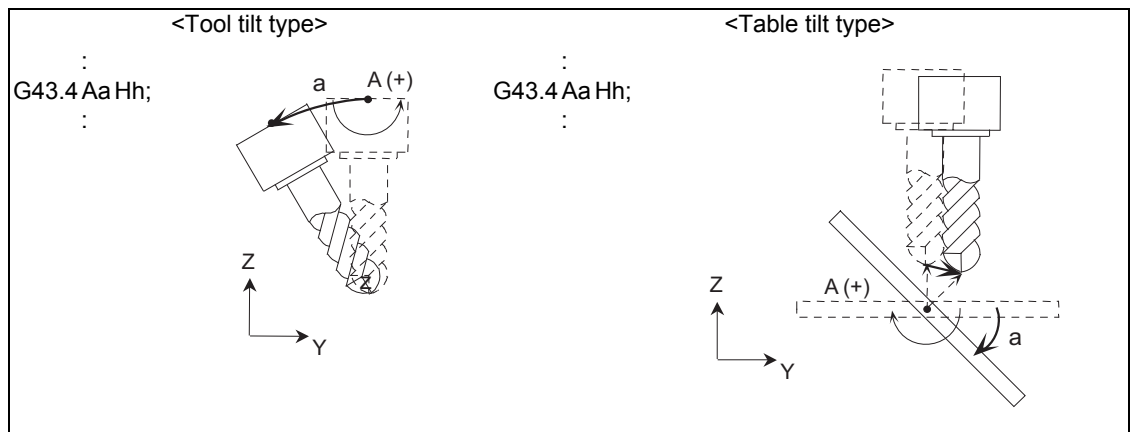
The rotary axis moves toward the commanded workpiece surface vector (I,J,K) direction along the movement command issued.



(3) Start-up with movement command (When rotary axis command is issued in the same block)

(a) Tool center point control type 1

In the case of using the tool tilt type, the orthogonal axis moves according to the rotary axis angle while fixing the tool center point to the center. In the case of using the table tilt type, the orthogonal axis moves so that the tool center point locates on the rotated table workpiece coordinate system.



(b) Tool center point control type 2

A program error (P33) will occur.

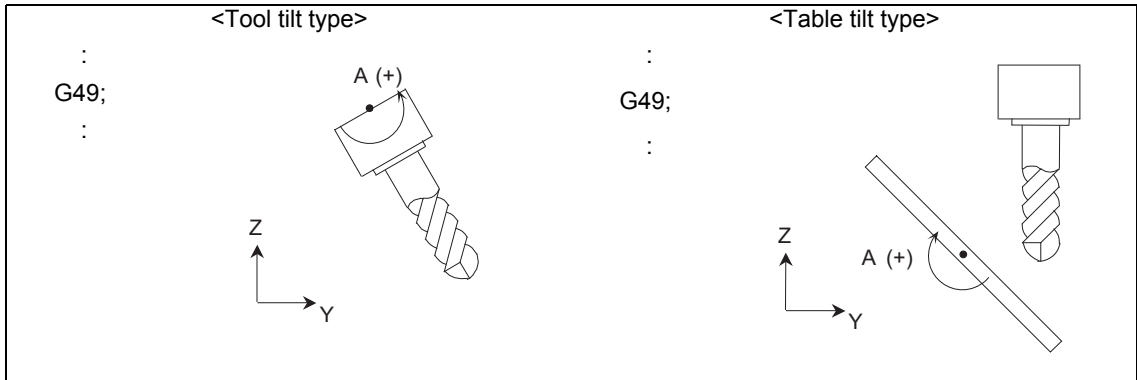
Cancel

(1) Independent cancel command

(a) Tool center point control type 1, type 2

Canceling the movement for the compensation amount is not performed regardless of absolute/incremental value command.

On the other hand, the tool center point control modal will be canceled.



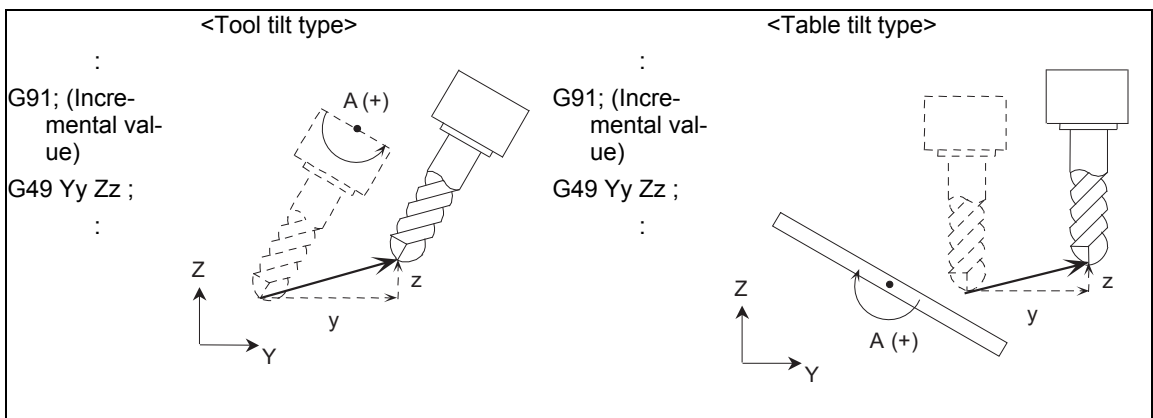
The tool will not move.

(2) Cancellation with movement command (When orthogonal coordinate axis command is issued in the same block)

(a) Tool center point control type 1, type 2

Canceling the movement for the compensation amount is not performed regardless of absolute/incremental value command.

Orthogonal coordinate axis movement command is executed upon cancellation of the tool center point control modal.

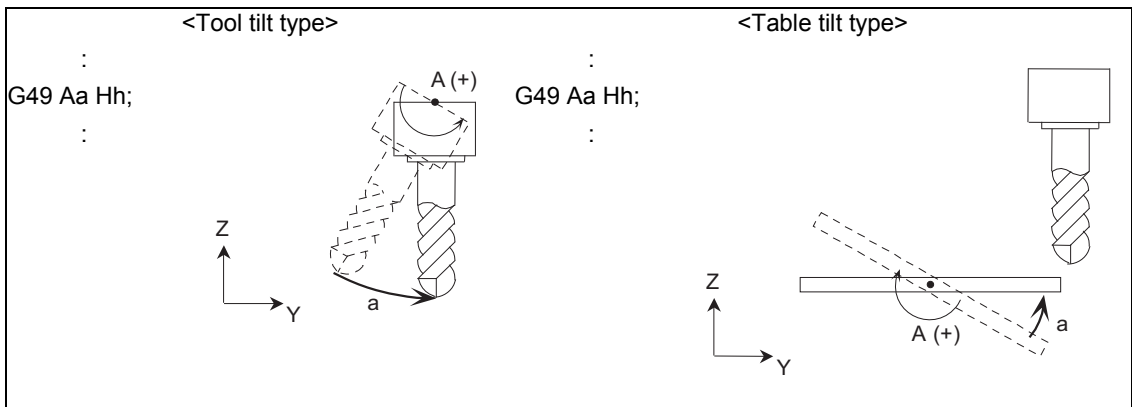


(3) Cancellation with movement command (When rotary axis command is issued in the same block)

(a) Tool center point control type 1, type 2

Canceling the movement for the compensation amount is not performed regardless of absolute/incremental value command.

Rotary axis movement command is executed upon cancellation of the tool center point control modal.



During tool center point control

(1) Tool center point control type 1

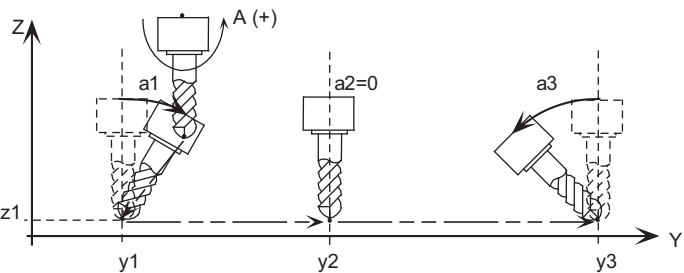
(a) When executing the movement command to the orthogonal coordinate axis and rotary axis.

```

:
G90 ;
G43.4 Yy1 Zz1 Aa1 Hh ;
Yy2 Aa2 ;
Yy3 Aa3 ;
:

```

The tool center point moves along the programmed path.



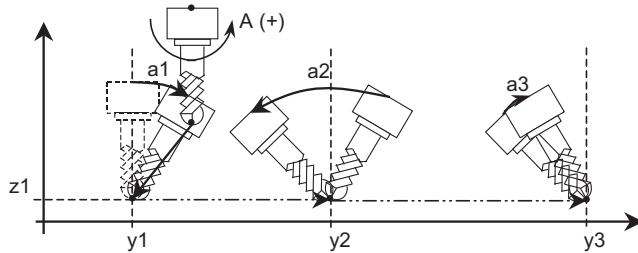
(b) When executing the movement command to the rotary axis only.

```

:
G90 ;
G43.4 Yy1 Zz1 Aa1 Hh ;
Yy2 ;
Aa2 ;
Yy3 Aa3 ;
:

```

When executing the movement command to the rotary axis only, the orthogonal axis moves without moving the tool center point.



(2) Tool center point control type 2

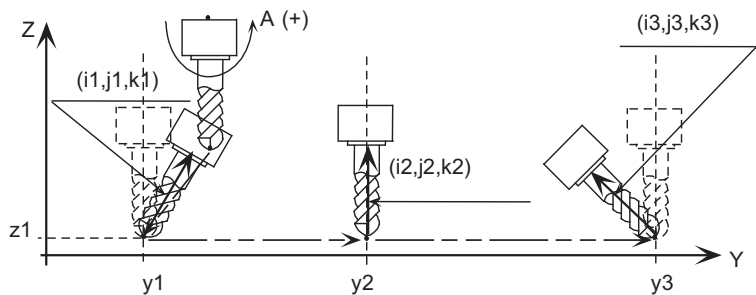
(a) When executing the movement command to the orthogonal coordinate axis and the workpiece surface angle vector command.

```

:
G43.5 Yy1 Zz1
li1 Jj1 Kk1 Hh ;
Yy2 li2 Jj2 Kk2 ;
Yy3 li3 Jj3 Kk3 ;
:

```

Tool center point moves along the program path.



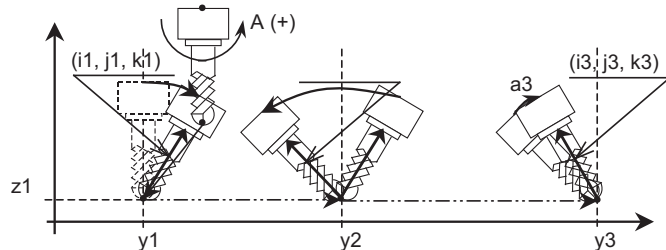
(b) When executing workpiece surface angle vector command only.

```

:
G43.5 Yy1 Zz1
li1 Jj1 Kk1 Hh ;
Yy2 ;
li2 Jj2 Kk2 ;
Yy3 li3 Jj3 Kk3 ;
:

```

When executing the workpiece surface angle vector command only, the orthogonal axis moves without moving the tool center point.



Feedrate during tool center point control

Feedrate during the tool center point control is controlled so that the tool center point moves according to the commanded speed.

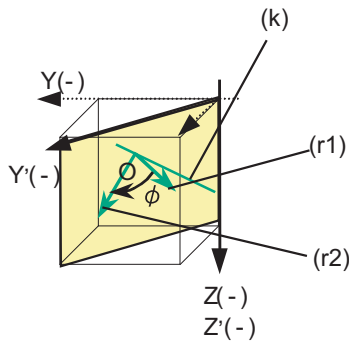
Interpolation method

There are two methods of interpolation: single axis rotation interpolation and joint interpolation, which are selected by parameter.

(1) Single axis rotation interpolation

When transforming from a start-point angle vector "r1" into an end-point angle vector "r2", interpolate so that the angular rate of the rotary " ϕ " around the vector "k" axis, which is vertical to "r1"-r2" plane, will be constant.

- (r1) Start-point command vector "r1"
- (r2) End-point command vector "r2"
- (k) Unit vector vertical to r1-r2 plane



(a) Features

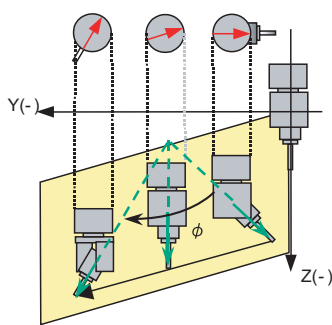
- Tool angle vector always exists on the plane consisting of "O", "r1" and "r2".
- The angular rates of each rotary axis will not be constant.

(b) Operations

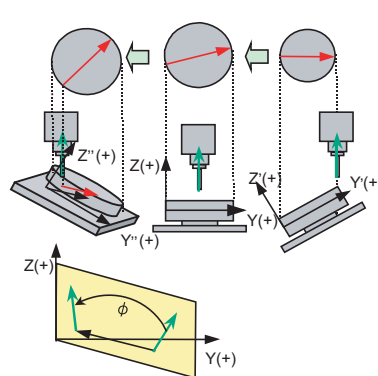
(Example) Current position: Aa°, C0°

When commanding "G90 Yy A-a. C45. ;" or "G90 Yy li Jj Kk ;"

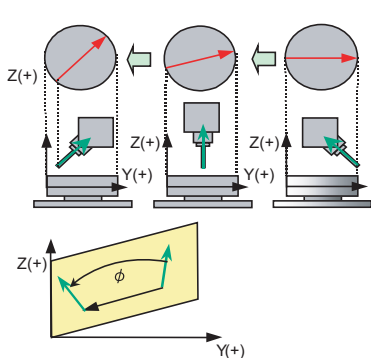
<Tool tilt type>



<Table tilt type>



<Combined type>

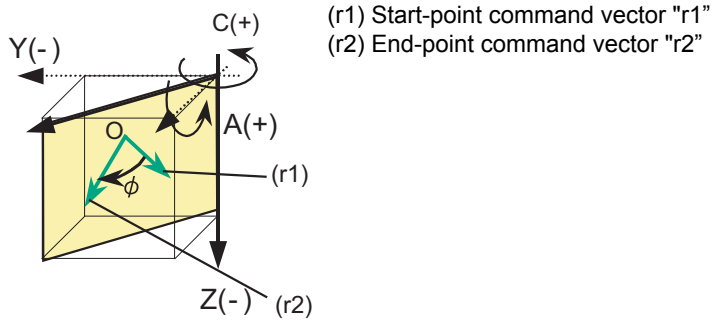


(2) Joint interpolation

A movement from a start-point angle vector "r1" to an end-point angle vector "r2" is interpolated to keep the angular rates of each axis constant.

(a) Features

- The angular rates of each rotary axis become constant.
- As this control aims to keep the angular rates of each rotary axis constant, a tool angle vector may not exist on the plane consisting of "O", "r1" and "r2".



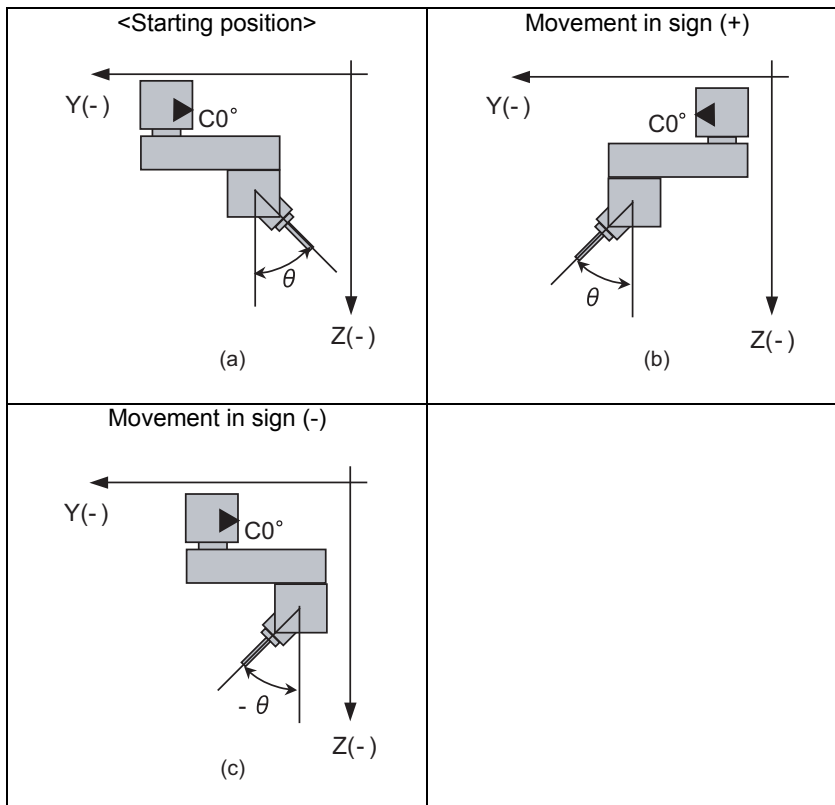
Passing singular point

When passing the singular point (singular position (*1)), there are two kinds of movements to be followed from the singular point.

When using an A-C axis tilt type machinery, there are two different movements (Fig. b, c) to be followed. In those movements, the rotation angles of the A axis are the same absolute value but different in signs (+/-). The rotation angles of the C axis corresponding the two movements are differed by 180 degrees one another.

Determine which one of the two movements are to be selected with parameter.

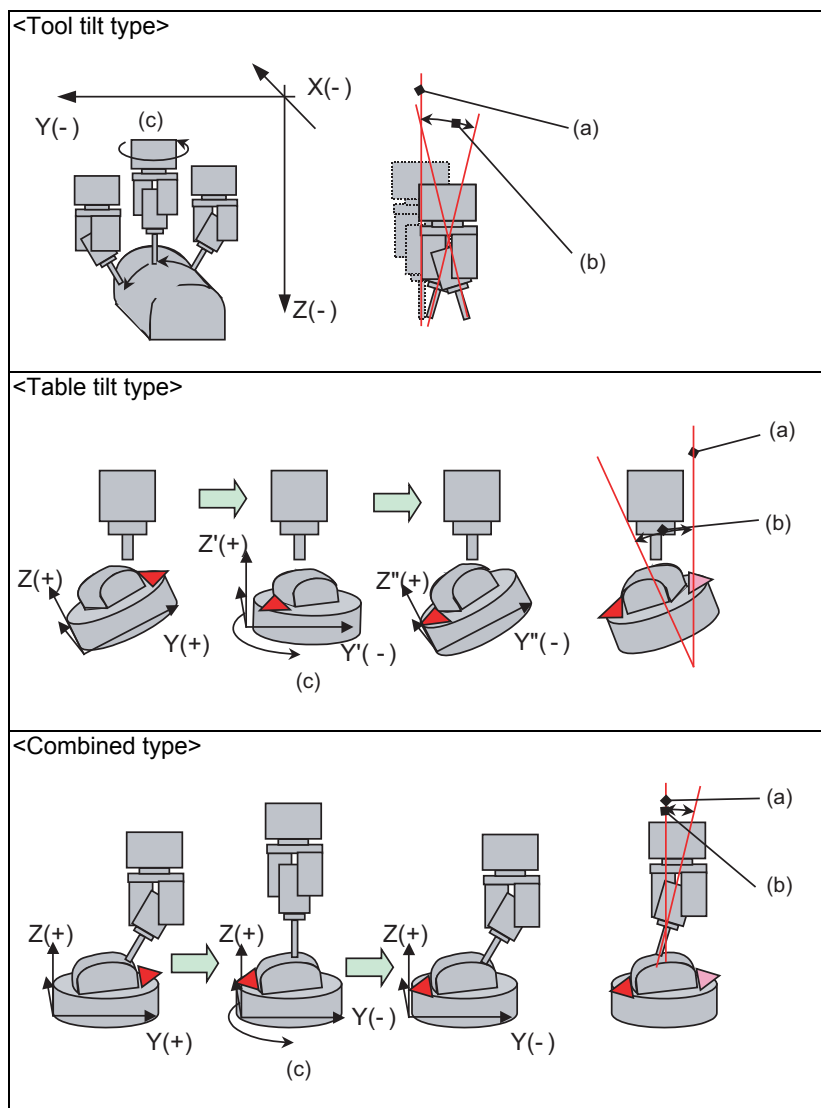
The figures below are the example of movements seen during tool center point control type 2. When the tool-center-point-side rotary axis moves in the sign (+) direction from the starting position (Fig. a), (Fig. b) is representing "passing singular point type 1". When the tool-center-point-side rotary axis moves in the sign (-) direction from the starting position (Fig. a), (Fig. c) is representing "passing singular point type 2".



(*1) The position in which the tool-center-point-side rotary axis or the table-base-side rotary axis is 0.

(1) Passing singular point type 1

Select the same direction as the start point of the tool-base-side rotary axis or table-workpiece-side rotary axis in the block where a singular point passing is carried out. When the rotation angle of the start point is 0°, select the wider stroke limit. When the stroke limits are the same, select the one with a minus-coded rotation angle.



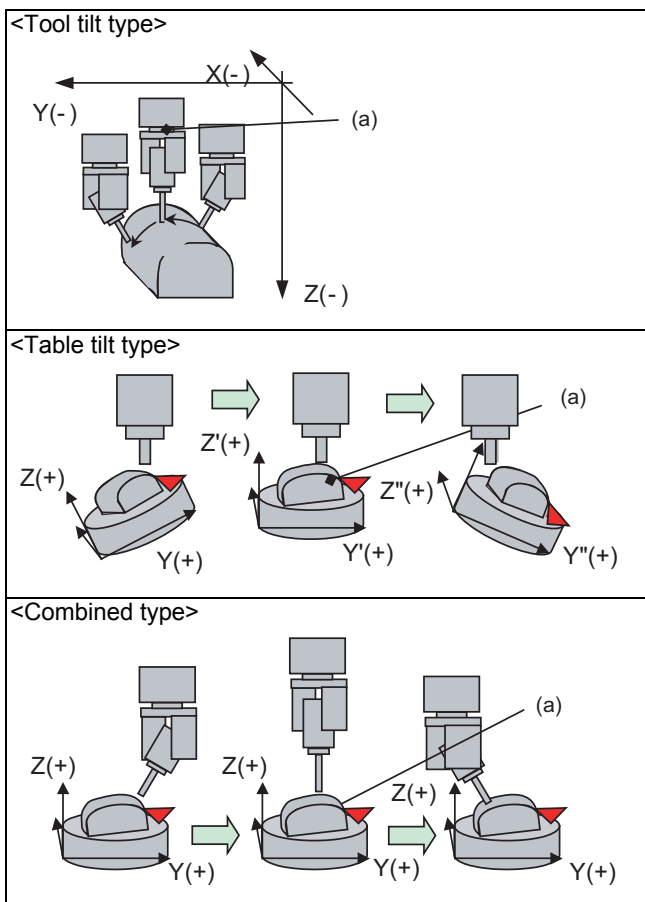
(a) Singular point

(b) When passing near the singular point, C axis rotates 180° within the parameter "#7907 CHK_ANG" (Near the singular judgment angle).

(c) C axis rotates 180°

(2) Passing singular point type 2

Select the one with the smaller rotary movement amount of the tool-base-side rotary axis or the table-workpiece-side rotary axis on the singular point. When the tool-base-side rotary axis and the table-workpiece have the same rotary movement amount, select the one with the tool-base-side rotary axis or the table-workpiece-side rotary axis that are to be rotated in the minus-coded direction.



(a) C axis does not rotate 180° when passing near the singular point.

(3) Operation near the singular point neighborhood in each interpolation method

Interpolation method	Command	Passing singular point type	Command from a singular point to a non-singular point	Command to pass a singular point
Single axis rotation interpolation	G43.4 (Rotary axis command)	Type 1	As designated in the command value. However, in the case where the signs at the start point and end point of either tool-center-point-side rotary axis or table-base-side rotary axis differ, if tool-base-side rotary axis or table-workpiece-side rotary axis rotates in the same block, the tool will not pass the singular point, resulting in a program error (P943).	
		Type 2		
	G43.5 (I/J/K command)	Type 1	Select the one with the wider stroke range. When the stroke range is the same, select a minus direction of the tool-center-point-side rotary axis or the table-base-side rotary axis.	Select the one with the same-coded end point as the start point of the tool-center-point-side rotary axis or the table-base-side rotary axis.
		Type 2	Select the one with the smaller movement amount of the tool-base-side rotary axis or the table-workpiece-side rotary axis.	

Interpolation method	Command	Passing singular point type	Command from a singular point to a non-singular point	Command to pass a singular point
Joint interpolation	G43.4 (Rotary axis command)	Type 1	As designated in the command value.	
		Type 2		
	G43.5 (I/J/K command)	Type 1	Select the one with the wider stroke range. When the stroke range is the same, select a minus direction of the tool-center-point-side rotary axis or the table-base-side rotary axis.	Select the one with the same-coded end point as the start point of the tool-center-point-side rotary axis or the table-base-side rotary axis.
		Type 2	Select the one with the smaller movement amount of the tool-base-side rotary axis or the table-workpiece-side rotary axis.	

Machine speed fluctuation suppression during high-accuracy control

In tool center point control during high-accuracy control, if there is no change in a center point speed command (F command), and also if a rotary axis moves with each block's segment length short, the machine end speed (speed of a motor that drives the tool/table) may fluctuate sharply. By enabling the parameter "#7913 MCHN_SPEED_CTRL" (Machine speed fluctuation suppression), fluctuation can be suppressed.

(1) When "#7913 = 0", the machine end speed is awaited to decelerate down to the machine end speed command (*1).

Select this setting when a machining is desired to closely follow the movement commands.

(2) When "#7913 = 1", the next block movement command is output to the machine immediately after a movement command output of the currently processed block is completed.

Select this setting in such a case as an execution of a machining program with non-continuous rotary axis movement commands, where a smooth movement is desired preventing a sudden deceleration of the machine end speed between blocks.

Nevertheless, if any of the conditions below is satisfied, deceleration is awaited regardless of the parameter setting.

- When judged to be a corner
- When the machining program's F command is changed
- When the speed is clamped
- When the override is changed

(*1) A machine end speed command value means a speed command value that is output to the machine end so that the center point speed becomes the F command value.

Nevertheless, depending on the machining program, enabling the parameter "#7913 MCHN_SPEED_CTRL" (Machine speed fluctuation suppression) may generate a machine vibration without deceleration.

<Machining program example>

Discontinuous rotary axis movement commands (a block is skipped between the movement commands)

:

G61.1;

G43.4 Hh;

G1 Ff;

:

N10 Xx1 Yy1 Zz1 Aa1;

N20 Xx2 Yy2 Zz2;

N30 Xx3 Yy3 Zz3 Aa3;

N40 Xx4 Yy4 Zz4;

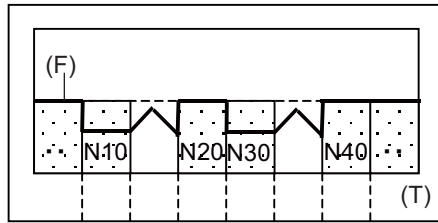
:

<Note>

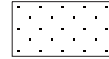
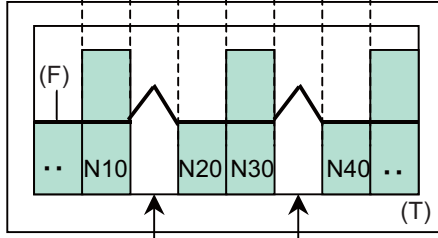
- Center point block lengths are even.
- A machine end block length is longer when it has a rotary axis movement command. (In this case, the machine end speed is faster in a block with rotary axis movement than in a block without rotary axis movement.)
- When SSS control is enabled, a machine speed fluctuation suppression is disabled.

(1) Speed when "#7913 = 0"

Tool center point speed



Machine end speed



Center point command speed



Machine end command speed

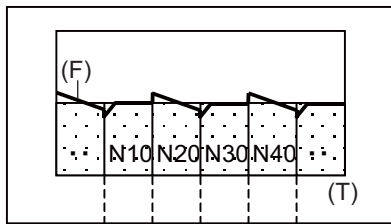
(F) Actual speed

(T) Time

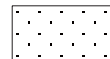
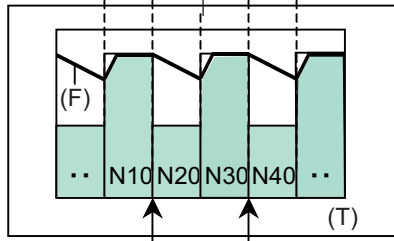
Awaited to decelerate down to the machine end speed of the next block. Thus, the speed changes sharply.

(2) Speed when "#7913 = 1"

Tool center point speed



Machine end speed



Center point command speed



Machine end command speed

(F) Actual speed

(T) Time

Not awaited to decelerate down to the machine end command speed of the next block. Thus, the speed does not change significantly, and the movement is smooth.

In (2), because the control does not wait for the deceleration to the machine end command speed of the next block, the actual center point speed exceeds the command speed. In such a case, by adjusting (increasing) the setting value of "#1570 Sfilt2" (Soft acceleration/deceleration filter 2), a range of the excess of the center point speed can be suppressed even when it exceeds the command speed.

<Note>

- ◆When SSS control is enabled, a machine speed fluctuation suppression is disabled.

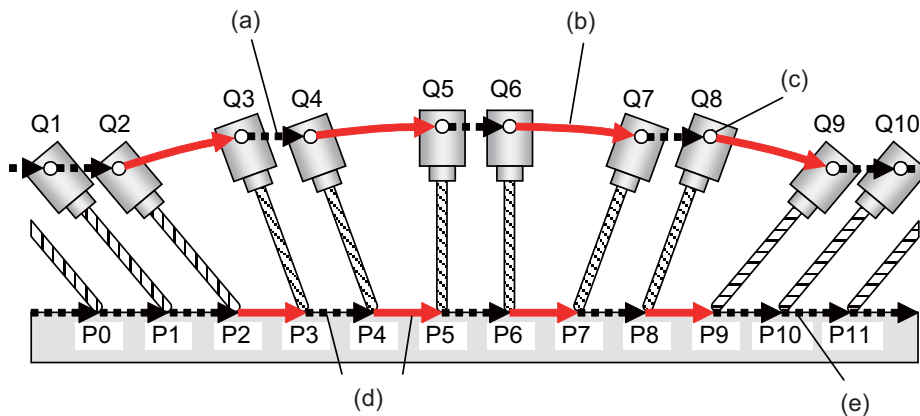
Rotary axis prefiltering

Rotary axis prefiltering means smoothing (prefiltering) the rotary axis command (tool angle shift) process, which moves the rotary axis smoothly and produces smoother cutting surface. Tool center point moves on the tracks as programmed by the rotary axis command while the command process is smoothed with this function.

This function is available for the programs which have intermittent rotary axis commands (tool angle shifts) or the programs with inconstant shift amount of rotary axis angle (or tool angle) per unit time.

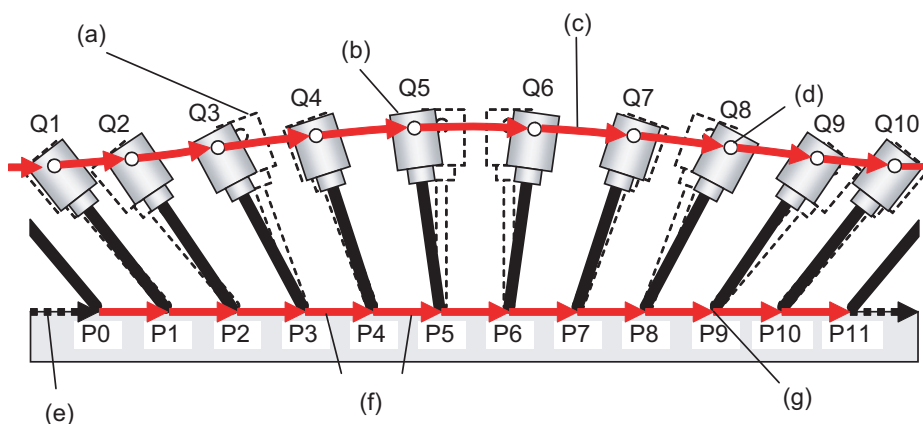
Set the filter time constant for this function with parameters.

When the rotary axis prefiltering is disabled, the tool center point shift speed may be sharply fluctuated due to the intermittent rotary axis command, as the figure below.



- (a) Without tool angle shift
- (b) With tool angle shift
- (c) Machine position (rotation center)
- (d) Tool center point needs to be shifted at constant speed in spite of the tool angle shift.
- (e) Tool center point

As shown below, the rotary axis prefiltering reduces speed fluctuation of tool center point by smoothing the rotary axis command process.



- (a) Tool angle before smoothing
- (b) Tool angle after smoothing
- (c) With tool angle shift
- (d) Machine position (rotation center)
- (e) Without tool angle shift
- (f) Tool center point needs to be shifted at constant speed
- (g) Tool center point

Note

- (1) This function is available only when SSS control is enabled. (Not available together with a machine speed fluctuation suppression.)
- (2) This function is disabled at G00 command.
- (3) The actual angle of the tool may be deviated from the commanded one in the program.
- (4) Even if a rotary axis prefiltering is enabled for a program without an angle shift, it does not affect the machining quality. However, it may extend cycle time, so when executing such a machining, it is recommended that the rotary axis prefiltering is disabled.

Mounting the rotary axis on the left-hand orthogonal coordinate system

When tool center point control is used on the machine in which the rotary axis is mounted on the left-hand orthogonal coordinate system, all the following three conditions must be satisfied.

- (1) Use tool center point control type 1 (G43.4).
(Normal operation is not assured in tool center point control type 2 (G43.5).)
- (2) Set the parameter "#7910 SLCT_INT_MODE" (interpolation mode selection) to the joint interpolation method.
(Normal operation is not assured in single axis interpolation mode.)
- (3) Set the rotary axis configuration parameter "rotation direction" of the rotary axis mounted on the left-hand orthogonal coordinate system to CCW.
The target "rotation direction" parameters are as follows.
 "#7923 DIR_T1" (Rotation direction of the tool rotating type base-side rotary axis)
 "#7933 DIR_T2" (Rotation direction of the tool rotating type / composite type tool axis)
 "#7943 DIR_W1" (Rotation direction of the table rotating type base-side rotary axis)
 "#7953 DIR_W2" (Rotation direction of the table rotating type / composite type workpiece axis)

Circular command in tool center point control (G43.4/G43.5)

For details, refer to "19.3.1 Circular Command in Tool Center Point Control (G43.4/G43.5)".

Operation when the rotary axis is set with reverse ISO polarity

The tool center point control type 1 (G43.4) can be used on the machine where the rotary axis is installed with reverse ISO polarity. In this case, all of the following conditions need to be satisfied.

- (1) Combined type 5-axis machine
- (2) Set the parameter "#1450 5axis_Spec/bit3" (Select specifications of rotation direction parameter) to "1".
- (3) Set the rotation direction parameter of the rotary axis installed with reverse ISO polarity (either "#7923 DIR_T1", "#7933 DIR_T2", "#7943 DIR_W1" or "#7953 DIR_W2") to "1".

		Setting value of rotation direction parameter (#7923, #7933, #7943, #7953)	
		0	1
Setting value of Select specifications of rotation direction parameter (#1450 5axis_Spec/bit3)	0	Rotation turning to the right-handed screw (direction switch is disabled)	
	1	Rotation turning to the right-handed screw	Rotation turning to the left-handed screw

Note

- (1) When the conditions in (2) and (3) above are satisfied, if the tool center point control type 2 (G43.5) is commanded or if the tool center point control is commanded with machine type other than combined type, the program error (P934) occurs.



Relationship with other functions

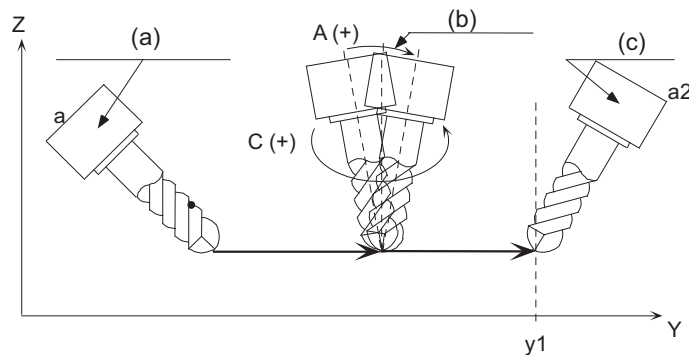
- (1) F1-digit Feed
Controls so that the tool center point moves at the commanded speed. Note that speed cannot be changed with the manual handle.
- (2) Buffer correction
Buffer correction cannot be performed during tool center point control.
- (3) Miscellaneous function (MSTB)
Miscellaneous function (MSTB) command can be executed during tool center point control.
(When passing the singular point, strobe signal is output at the block start and the completion wait at the block end.)

(Example)

```

:
G90 Aa1 ;
G43.4 Yy1 Aa2 Mm Hh ;
:

```



(a) M strobe output

(b) Passing singular point

(c) M completion wait

- (4) Spindle/C Axis Control
Axes unrelated to the tool tilt or table tilt can be controlled.
- (5) Manual reference position return
Do not perform manual reference position return during tool center point control. If performed, the tool moves off the programmed track.
- (6) Machining time computation
Machining time calculation is not performed accurately on the machining program in which the tool center point control mode is commanded.
- (7) Graphic trace
Graphic trace during the tool center point control is always traced with the tool center point.
- (8) Graphic check
Graphic check during the tool center point control is always check the graphic with the tool center point.
- (9) Program restart
Restart search cannot be performed during the tool center point control. If the restart search is attempted, the program error (P49) occurs.
- (10) Reset modal retention
Canceled during the tool center point control.
- (11) Collation and stop
Position in the tool center point control can be collated and stopped.
- (12) Automatic operation handle interruption
Do not perform the automatic operation handle interruption during the tool center point control. If performed, the tool moves off the programmed track.
- (13) Manual / Automatic simultaneous
Manual / Automatic simultaneous cannot be executed to the axes related to the tool center point control during the tool center point control.
- (14) Tool handle feed & interruption
Do not perform the tool handle feed & interruption during the tool center point control. If performed, the tool moves off the programmed track.
- (15) Corner chamfering/Corner R
When the corner chamfering/corner R is performed during the tool center point control, the tool center point control becomes valid to the track after the corner chamfering/corner R.

- (16) Mirror image by parameter setting / external mirror image input
When the tool center point control command is issued during the mirror image by parameter/external Input, a program error (P941) occurs. Also, do not turn the mirror image by parameter/external input ON during the tool center point control.
- (17) Linear angle command
When A axis is used as a rotary axis, the linear angle command cannot be executed. When A axis is not used as a rotary axis, tool center point control becomes valid to the shape after the linear angle command.
- (18) Geometric command
When A axis is used as a rotary axis, the geometric command cannot be executed. When A axis is not used as a rotary axis, tool center point control becomes valid to the shape after the geometric command.
- (19) Figure rotation
The tool center point control becomes valid to the shape after the figure rotation.
- (20) Coordinate rotation by parameter
When the tool center point control command is issued during the coordinate rotation by parameter, a program error (P941) occurs. Also, do not turn the coordinate rotation by parameter ON during the tool center point control.
- (21) Chopping
Chopping operation for the 3 orthogonal axes and 2 rotary axes cannot be performed during the tool center point control.
- (22) Macro interruption
If the macro interruption command is executed during the tool center point control, a program error (P942) occurs.
- (23) Tool life management
The compensation amount of the tool center point control during the tool life management is equal to the compensation amount of the tool subjected to the tool life management.
- (24) G00 non-interpolation
Functions as "G00 interpolation".
- (25) Actual feedrate display
The final combined feedrate is displayed here.
- (26) Manual interruption
When the manual interruption is executed during the feed hold or single block stop, the movement will be the one to be observed when the manual ABS is OFF when rebooting regardless of whether an absolute/incremental value command is selected.
- (27) Machine lock
The each axis machine lock becomes valid to the motor axis.
- (28) Remaining distance counter
Remaining distance at the tool center point on the programming coordinate system is displayed.
- (29) Interlock
Interlock is applied for the motor axis.
- (30) Cutting feed / Rapid traverse override
Override is applied to the feedrate at the tool center point. When the feedrate is clamped, the override is applied to the clamp speed.
- (31) Manual reference position return
If the manual reference position return is performed during the tool center point control, the tool moves off the programmed track after that.
- (32) Dry run
Dry run is applied to the speed at the tool center point.
- (33) NC reset
Immediately decelerates to stop when the NC reset is executed during the tool center point control. The tool center point control will be canceled even if NC reset 1 and the modal retention.
- (34) Emergency stop
Immediately stops if the emergency stop is applied during the tool center point control.
- (35) Stored stroke limit
limit Stored stroke limit will be valid at the motor axis for all IB, IIB and IC.
- (36) MDI interruption
When the MDI interruption is performed during the tool center point control, an operation error (M01 0170) occurs.

(37) High-accuracy control function

The acceleration with the rapid traverse (G00) in high-accuracy control mode depends on the MTB specifications (parameter "#1250 set22/bit3").

0	The acceleration is the same as that for cutting feed (G01). This is determined based on the parameters "#1206 G1bF" (maximum speed) and "#1207 G1btL" (time constant).
1	The acceleration is the same as that for rapid traverse (G00). This is determined based on the parameters "#2001 Rapid" (rapid traverse rate) and "#2004 G0tL" (G0 time constant (linear)). (*1)

(*1) Even when the parameter "#1250 set22/bit3" is set to "1", this bit is assumed to be set to "0" to perform operation while SSS control is disabled.

(38) Spindle position control (Spindle/C axis)

If the tool center point control is commanded in spindle mode when the axis set to the spindle/C axis is used as a rotary axis of tool center point control, a program error (P934) occurs. When the mode is changed to the spindle mode during the tool center point control, an operation error (M01 0186) occurs. An operation error (M01 0186) can be reset by NC reset.

Combination with arbitrary axis exchange

When performing tool center point control in combination with an arbitrary axis exchange (G140) command, you need to set the rotary axis configuration parameters using the 2nd axis name. Set the parameter "#1450 5axis_Spec/bit0" to "1" (setting by the 2nd axis name), and assign the axis configuration for executing tool center point control to the rotary axis configuration parameter (#7900 or later) using the 2nd axis name (example: A1, B2). If the G43.4/G43.5 command is issued after the arbitrary tool exchange has been completed while the parameter "#1450 5axis_Spec/bit0" is not designated, a program error (P941) will occur.

You can set the configurations up to the number of valid part systems (up to four part systems) in the rotary axis configuration parameter. With multiple configurations set, you can perform tool center point control in different axis configurations.

Tool center point control can be performed using the axis configuration in the part system with axis exchange completed by applying the rotary axis configuration parameter in the configuration in which all axes included in the part system are set.

Relation with other G codes

Pxxx in the list indicates the program error Nos.

Column A: Operation to be carried out when the G command in the list is issued while this function is modal

Column B: Operation to be carried out when this function is commanded while the G command in the list is modal

Column C: Operation to be carried out when the G command in the list and this function are commanded for the same block

All the G codes not listed above cannot be used.

Format		Function	A	B	C
G00		Positioning	Switched to a rapid traverse, and then perform tool center point control.	Perform tool center point control with rapid traverse.	Perform tool center point control with rapid traverse.
G01		Linear interpolation	Switched to a cutting feed, and then perform tool center point control.	Perform tool center point control with cutting feed.	Perform tool center point control with cutting feed.
G02/G03		Circular interpolation	Perform circular under interpolation tool center point control.	P941	P941
		Helical interpolation	P942	P941	P941
G02.1/G03.1		Spiral Interpolation	P942	P941	P941
G02.3/G03.3		Exponential interpolation	P942	P941	P941
G04		Dwell	Dwelling is performed.	-	Tool center point control is ignored as dwell function takes precedence over the tool center point control function.
G05	P1 (*1)	High-speed machining mode	Max. feedrate is 16.8 m/min when 1 mm segment G1 block is commanded with 5 axes simultaneously	Max. feedrate is 16.8 m/min when 1 mm segment G1 block is commanded with 5 axes simultaneously	P33
	P2 (*1)		Max. feedrate is 100m/min when 1 mm segment G1 block is commanded with 5 axes simultaneously	Max. feedrate is 100m/min when 1 mm segment G1 block is commanded with 5 axes simultaneously	
	P1000 (*2)	High-speed high-accuracy control II	Max. feedrate is 100m/min when 1 mm segment G1 block is commanded with 5 axes simultaneously	Max. feedrate is 100m/min when 1mm segment G1 block is commanded with 5 axes simultaneously	P33
G05.1 (*2)		High-speed high-accuracy control I	Max. feedrate is 33.7m/min when 1 mm segment G1 block is commanded with 5 axes simultaneously	Max. feedrate is 33.7m/min when 1 mm segment G1 block is commanded with 5 axes simultaneously	P33
G06.2		NURBS interpolation	P942	P*** NURBS general error	P941
G07.1 G107		Cylindrical interpolation	P942	P941	P941
G08 (*2)	P0	High-accuracy control	Perform tool center point control in cutting mode.	Perform tool center point control in cutting mode.	P33
	P1		Tool center point control is performed in the high-accuracy control mode.	Tool center point control is performed in the high-accuracy control mode.	

Format	Function	A	B	C
G09	Exact stop check	Deceleration check is performed at the block end.	-	Deceleration check is performed at the block end.
G10/G11	Parameter input by program	P942	-	P941
G10	Compensation data input by program	P942	-	P941
G12/G13	Circular cut	P942	-	Tool center point control is ignored as the circular cutting takes precedence over the tool center point control function.
G12.1/G13.1 G112/G113	Polar coordinate interpolation	P942	P941	P941
G15/G16	Polar coordinate command	P942	P941	P941
G17 to G19	Plane selection	The modal is switched to the specified plane.	-	The modal is switched to the specified plane.
G20/G21	Inch/ Metric	P942	Tool center point control is performed according to the inch / metric modal.	P941
G22/G23	Stroke check before travel	P942	P941	P941
G27	Reference position Check	P942	-	The tool center point control is ignored as the reference position check becomes valid.
G28	Reference position Return	P942	-	The tool center point control is ignored as the reference position return becomes valid.
G29	Start position return	P942	-	The tool center point control is ignored as the start position return becomes valid.
G30	2nd to 4th reference position return	P942	-	The tool center point control is ignored as the 2nd, 3rd, 4th reference position return becomes valid.
G30.1 to G30.6	Tool exchange position return 1 to 6	P942	-	P941
G31	Skip	P942	-	P941
G31.1 to G31.3	Multi-step skip	P942	-	P941
G33	Thread cutting	P942	P941	P941
G34 to G36/ G37.1	Special fixed cycle	P942	-	P941
G37	Automatic tool length measurement	P942	-	P941
G38	Tool radius compensation vector designation	P942	-	P941

Format	Function	A	B	C
G39	Tool radius compensation corner circular command	P942	-	P941
G40/G41/G42	Tool radius compensation	P942	P941	P941
G40.1/G41.1/ G42.1/G150/ G151/G152	Normal line control	P942	P941	P941
G43/G44/G49	Tool length compensation	Tool length compensation can be performed upon tool center point control cancellation.	Tool center point control can be performed upon tool length compensation cancellation.	The subsequently commanded modal takes precedence.
G43.1/G49	Tool length compensation along the tool axis	Tool length compensation along the tool axis can be performed upon tool center point control cancellation.	Tool center point control can be performed upon tool length compensation along the tool axis cancellation.	The subsequently commanded modal takes precedence.
G45/G46/ G47/G48	Tool position offset	P942	-	P941
G50/G51	Scaling	P942	P941	P942
G50.1/G51.1	Mirror image	P942	P941	P941
G52	Local coordinate system Setting	P942	-	The tool center point control is ignored as the local coordinate system setting becomes valid.
G53	Machine coordinate system selection	P942	-	The tool center point control is ignored as the machine coordinate system selection becomes valid.
G54 to G59/ G54.1	Workpiece coordinate system selection	P942	Tool center point control is performed in the currently selected workpiece coordinate system.	P941
G60	Unidirectional positioning	P942	-	The tool center point control is ignored as the unidirectional positioning becomes valid.
G61	Exact stop check mode	Deceleration check is performed at the block end.	Deceleration check is performed at the block end.	Deceleration check is performed at the block end.
G61.1	High-accuracy control	Tool center point control is performed in the high-accuracy control mode.	Tool center point control is performed in the high-accuracy control mode.	Tool center point control is performed in the high-accuracy control mode.
G61.2	High-accuracy spline interpolation 1	P942	P941	P941
G62	Automatic corner override	P942	P941	P941
G63	Tapping mode	P942	P941	P941
G64	Cutting mode	Perform tool center point control in cutting mode.	Perform tool center point control in cutting mode.	Perform tool center point control in cutting mode.
G65 to G67/ G66.1	User macro	Tool center point control becomes valid even in the user macro program.	Tool center point control becomes valid even in the user macro program.	Tool center point control is ignored as the user macro takes precedence over the tool center point control function.

Format	Function	A	B	C
-	User macro subprogram termination	User macro subprogram is terminated.	-	Tool center point control will be ignored.
-	End point error check cancellation	The end position error check cancellation becomes valid.	-	The end position error check cancellation, too, becomes valid.
G68/G69	Coordinate rotation	P942	P941	P941
G68iJjKk/ G69	3-dimensional coordinate conversion	P922	P941	P923
G70 to G89	Fixed cycle	P942	The tool center point control is ignored as the start fixed cycle becomes valid.	The tool center point control is ignored as the start fixed cycle becomes valid.
G90/G91	Absolute/Incremental value command	The modal is switched to the specified absolute / incremental value command, and then tool center point control is performed.	Tool center point control is performed following the absolute / incremental modal.	Tool center point control is performed under the specified absolute / incremental value command.
G92	Machine coordinate system setting	P942	-	P941
G94	Feed per minute	Tool center point control is performed in the feed-per-minute mode.	Tool center point control is performed in the feed-per-minute mode.	Tool center point control is performed in the feed-per-minute mode.
G95	Feed per revolution	P942	P941	P941
G96/G97	Constant surface speed control	P942	P941	P941
G98	Fixed cycle Initial level return	The modal is switched to G98 and tool center point control becomes valid.	The modal is switched to G98 and tool center point control becomes valid.	The modal is switched to G98 and tool center point control becomes valid.
G99	Fixed cycle (R point level return)	The modal is switched to G99 and tool center point control becomes valid.	The modal is switched to G99 and tool center point control becomes valid.	The modal is switched to G99 and tool center point control becomes valid.
G114.1	Spindle synchronization	P942	Tool center point control is also available while spindle synchronization is enabled. (*3)	P941

(*1) It is valid when the parameter "#1267 ext03/bit0" is "0". If it is commanded when this parameter is "1", the program error (P34) will occur.

(*2) It is valid when the parameter "#1267 ext03/bit0" is "1". If it is commanded when this parameter is "0", the program error (P34) will occur.

(*3) If the axis set to the spindle/C axis is used as a rotary axis for tool center point control, the following program errors will occur.

- ◆If spindle synchronization is commanded for the spindle in the C axis mode, an operation error (M01 1026) will occur.
- ◆If tool center point control is commanded in the spindle mode, a program error (P934) will occur.



Precautions

- (1) If axis movement is commanded for any axis not specified in the rotary axis configuration parameters during tool center point control, a program error (P942) will occur.

19.3.1 Circular Command in Tool Center Point Control (G43.4/G43.5)

**Function and purpose**

The circular command during tool center point control mode interpolates the tool center point with an arc in a table coordinate system.

The circular command during tool center point control mode enables the circular interpolation under the following conditions.

Circular interpolation during tool center point control	
Command type	G43.4 / G43.5
Limitations when commanded	None (Format error, etc. only)
Interpolation method	Joint interpolation (*1)
Program coordinate system selection	Table coordinate system/workpiece coordinate system (*1)
Rotary axis reference position selection	Zero degree position basis/start position basis (*1)
Rotary axis prefiltering	Select Invalid with the parameter. (*2)

(*1) Selected with the parameter.

(*2) Designate the time constant to "0" with the parameter.

Tool center point control type 1 (G43.4)

G02(G03) X__ Y__ I__ J__ A__ C__ F__ ; Format with I address
--

G02(G03) X__ Y__ R__ A__ C__ F__ ; Format with R address
--

G02/G03	Arc rotation direction
X, Y	End point coordinates of orthogonal coordinate axis
A, C	End-point angle of rotary axis
I, J	Coordinates for arc center
R	Circular radius
F	Feedrate

Note

- (1) Designate the arc rotation direction with G02 (CW) or G03 (CCW).
- (2) Commands to I, J, and K are the arc center coordinates during the tool center point control type 1.
- (3) Either an absolute value or incremental value can be used for the arc end point coordinate command, but the arc center coordinate value must always be designated with an incremental value from the start point.
- (4) The circular arc center coordinate value of the axis must be commanded in the input setting unit. Caution is required for the circular arc command for which the input command unit differs. Command with a decimal point to avoid confusion.
- (5) The feedrate is controlled to always move along the circumference.

Tool center point control type 2 (G43.5)

G02(G03) X__Y__R__I__J__K__F__ ; Format with R address

G02/G03	Arc rotation direction
X, Y	End point coordinates of orthogonal coordinate axis
I, J, K	Workpiece surface angle vector
R	Circular radius
F	Feedrate

Note

- (1) Designate the arc rotation direction with G02 (CW) or G03 (CCW).
- (2) Commands to I, J, and K are issued as the tool angle vector command during tool center point control type 2.
- (3) Rotary axis movement command cannot be issued during the tool center point control type 2. If commanded, a program error (P33) occurs.
- (4) If I, J, or K is omitted when the tool center point control type 2 command is issued, the omitted address will be considered as "0".
- (5) When the R address is omitted, a program error (P33) occurs.
- (6) The feedrate is controlled to always become the rate along the circumference.

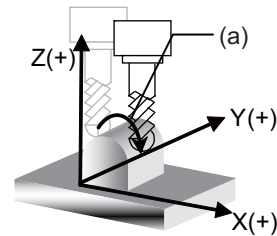
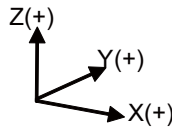


Detailed description

Operation for each type

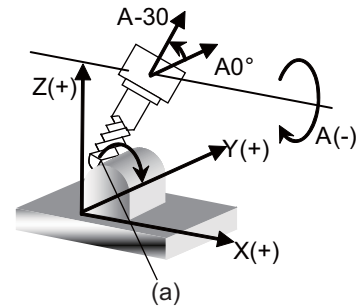
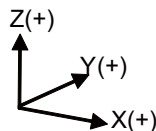
- (1) Tool tilt type (Rotary axis machine coordinate 0°)

```
<Machining program>
:
G18;
G43.4 H1;
:
G02 Xx Zz Ii Kk;
:
```



- (2) Tool tilt type (Rotary axis machine coordinate - 30°)

```
<Machining program>
:
G18;
G43.4 H1;
A-30.;
:
G02 Xx Zz Ii Kk;
:
```

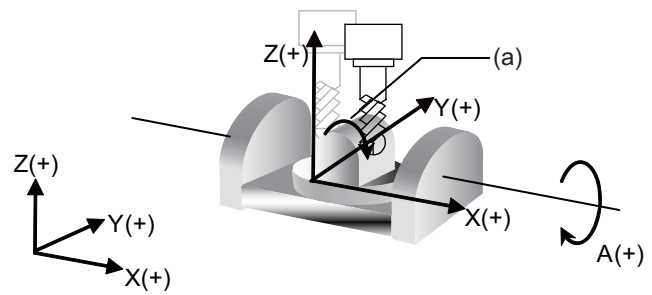


(3) Table tilt type (0° position standard)

<Machining program>

```

:
G18;
G43.4 H1;
:
G02 Xx Zz Ii Kk;
:
    
```



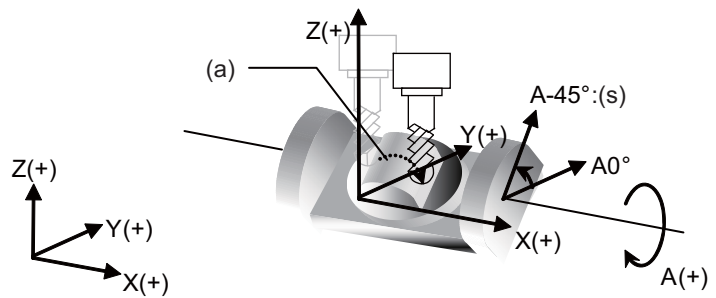
(a) Arc operations

(4) Table tilt type (Start position standard)

<Machining program>

```

:
G19 A-45.;
G43.4 H1;
:
G02 Xx Yy Ii Jj;
:
    
```

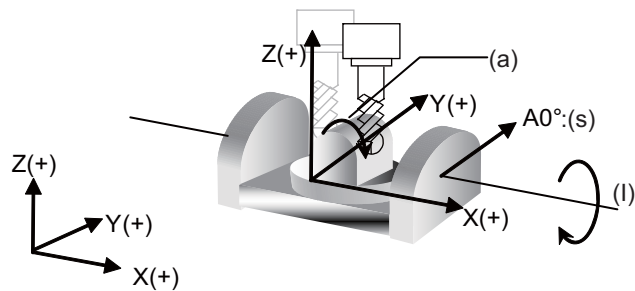


(5) Table tilt type (Programming coordinate system = workpiece coordinate system)

<Machining program>

```

:
G18;
G43.4 H1;
:
G02 Xx Zz Ii Kk;
:
    
```



(a) Arc operations

(s) Start position

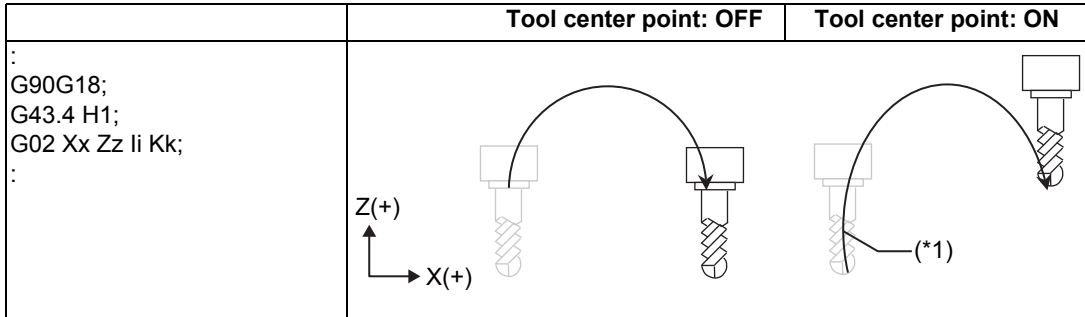
(I) About I axis



Precautions

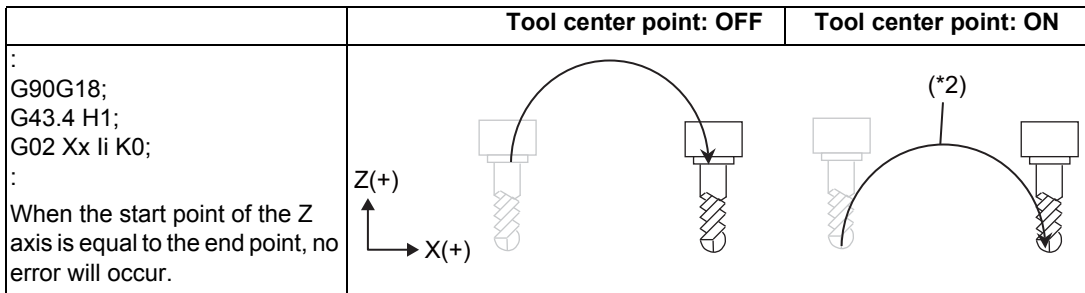
(1) If the circular command is issued without positioning three orthogonal axes after tool center point control has been started independently, a program error (P70) or (P71) may occur.

(a) Tool center point control commanded independently:



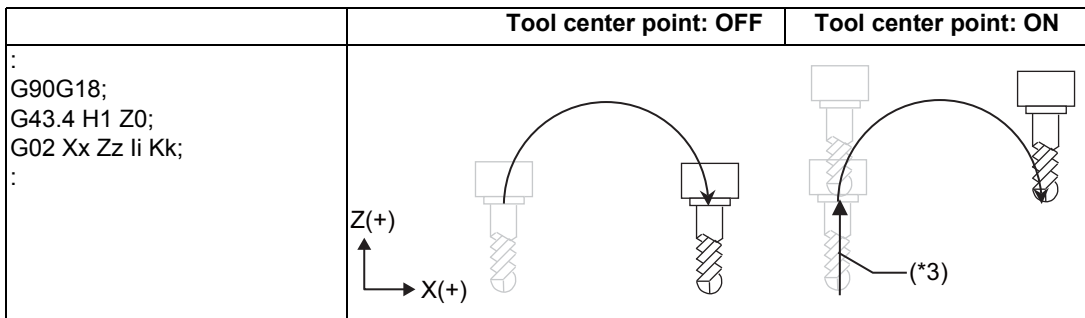
(*1) The current position is used as the tool center point, and the end point of the arc as the tool center point, so the arc radius cannot be obtained.

(b) Tool center point control commanded independently:



(*2) The tool center point is not set as a program path.

(c) Tool center point control commanded with movement:



(*3) The tool center point is used as the Z0 position.

(2) When G49 (tool center point control cancel) is commanded in circular interpolation modal during tool center point control, the circular interpolation modal remains enabled. Therefore, command the linear interpolation (G00 or G01) before canceling tool center point control.

(3) If the spiral interpolation or helical interpolation command is issued during tool center point control, a program error (P942) occurs.

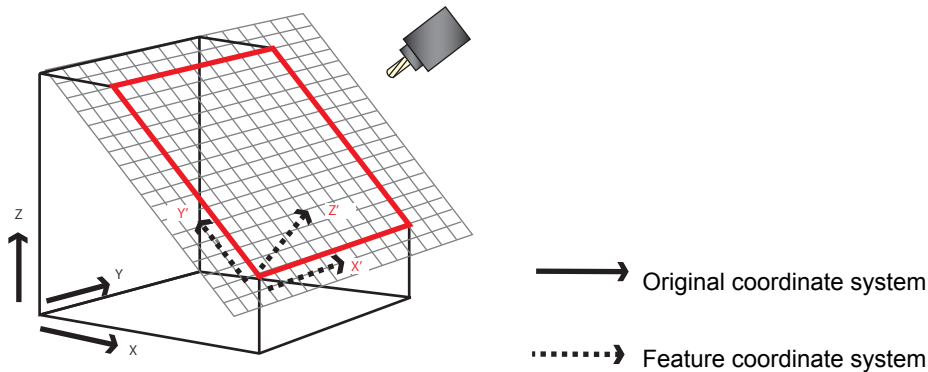
19.4 Inclined Surface Machining; G68.2, G68.3/G69



Function and purpose

Inclined surface machining function enables defining a new coordinate system (feature coordinate system) which is obtained by rotating and parallel translating the origin of the present coordinate system (X, Y, Z) (a coordinate system that existed before the inclined surface machining command was issued). With this function, you can define an arbitrary plane in a space and issue normal program commands to this plane in machining.

It's possible to automatically control the tool axis to be in the + Z direction of the newly defined feature coordinate system. The feature coordinate system is redefined in accordance with the tool axis direction, thus there is no need to mind the feature coordinate system's direction and tool axis' rotation direction in making machining programs. If the inclined surface machining is commanded while this function is not defined in the specifications, it causes a program error (P950).



When workpiece installation error compensation is valid, the workpiece coordinate system is set to the workpiece installation coordinate system.

If the linear axis and two rotary axes are commanded to the same block in a model for which the number of simultaneous contouring control axes is 4 or less, a program error (P10) occurs.

(Example) When the following machining program is executed with machine configuration X-Y-Z-A-C

:	
G68.2 X10. Y20. I0. J-45. K0.;	
:	
X20. A10 C20;	Program error (P10)
:	
G69;	

The feature coordinate system is defined using the following method.

G code	Command method
G68.2 P0	Define using Euler angles
G68.2 P1	Define using roll-pitch-yaw angles
G68.2 P2	Define using three points in a plane
G68.2 P3	Define using two vectors
G68.2 P4	Define using projection angles
G68.2 P10	Define by selecting the registered machining surface
G68.3	Define using tool axis direction
G69	Cancel inclined surface machining command

- If address P is omitted when G68.2 is commanded, it is assumed that G68.2 P0 is designated (define using Euler angles).
- If address P is not set to "0" to "4" or "10" when G68.2 is commanded, a program error (P954) will occur.
- If address P or Q of the G68.2 command includes a decimal point, it is rounded to an integer.
- Make sure to command G68.2, G68.3, and G69 in an independent block. If they are commanded in the same block as for other G codes or a motion command, etc., a program error (P954) will occur.
- The G69 command cannot be issued during circular interpolation or fixed cycle mode. If issued, a program error (P952) will occur.

19.4.1 How to Define Feature Coordinate System Using Euler Angles



Command format

Inclined surface machining mode ON (define using Euler angles) (P0 can be omitted.)

```
G68.2 P0 X__ Y__ Z__ I__ J__ K__ ;
```

X, Y, Z	Feature coordinate system's origin Command by the absolute values with respect to the coordinate system before issuing the inclined surface machining command.
I, J, K	Euler angles (-360.0° to 360.0°)

Note

- (1) If the address X, Y or Z is omitted, the address will be regarded as zero.
When all of addresses X, Y, and Z are set to "0", the feature coordinate system's origin will be the same as that of the coordinate system before the inclined surface machining command is issued.
- (2) If the address I, J or K is omitted, the address will be regarded as zero.
- (3) If any address other than P, X, Y, Z, I, J and K is included, a program error (P954) will occur.



Detailed description

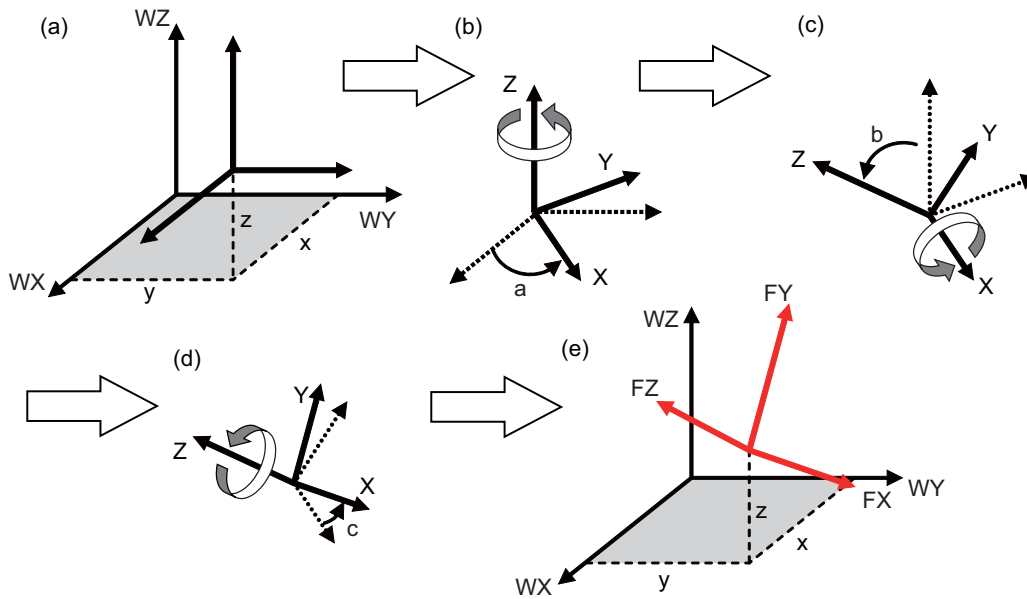
By commanding G68.2 P0 (define using Euler angles), the feature coordinate system (a coordinate system made by rotating and shifting the origin of the coordinate system before inclined surface machining) is defined.

Coordinate system rotation is commanded using the Euler angles.

(Example) When "G68.2 Xx Yy Zz Ia Jb Kc;" is commanded, the feature coordinate system is established as below.

- (a) Define a point (x, y, z) in the coordinate system before issuing the inclined surface machining command, as the feature coordinate system's origin.
- (b) Rotate the coordinate system, which was defined by shifting the origin in (a), by angle a about its Z axis.
- (c) Rotate the coordinate system, which was defined by rotation in (b), by angle b about its X axis.
- (d) Rotate the coordinate system, which was defined by rotation in (c), by angle c about its Z axis.
- (e) The coordinate system created in the above steps is the feature coordinate system.

If coordinate system's rotation is counter clockwise when viewing from the positive ends of the rotation center axis, this rotation will be considered as forward rotation. The relationship between the coordinate system before issuing the inclined surface machining command and the feature coordinate system is as shown below.



19.4.2 How to Define Feature Coordinate System Using Roll-Pitch-Yaw Angles



Command format

Inclined surface machining mode ON (define using roll-pitch-yaw angles)

```
G68.2 P1 Q_ X_ Y_ Z_ I_ J_ K_ ;
```

X, Y, Z	Feature coordinate system's zero point Command by the absolute values with respect to the coordinate system before issuing the inclined surface machining command.																												
Q	Rotation order (q: Setting value for address "Q") <table border="1"> <thead> <tr> <th>q</th> <th>First</th> <th>Second</th> <th>Third</th> </tr> </thead> <tbody> <tr> <td>123</td> <td>X</td> <td>Y</td> <td>Z</td> </tr> <tr> <td>132</td> <td>X</td> <td>Z</td> <td>Y</td> </tr> <tr> <td>213</td> <td>Y</td> <td>X</td> <td>Z</td> </tr> <tr> <td>231</td> <td>Y</td> <td>Z</td> <td>X</td> </tr> <tr> <td>312</td> <td>Z</td> <td>X</td> <td>Y</td> </tr> <tr> <td>321</td> <td>Z</td> <td>Y</td> <td>X</td> </tr> </tbody> </table> If address Q is omitted, "q" will be handled as "123".	q	First	Second	Third	123	X	Y	Z	132	X	Z	Y	213	Y	X	Z	231	Y	Z	X	312	Z	X	Y	321	Z	Y	X
q	First	Second	Third																										
123	X	Y	Z																										
132	X	Z	Y																										
213	Y	X	Z																										
231	Y	Z	X																										
312	Z	X	Y																										
321	Z	Y	X																										
I	Rotation angle about the X axis (roll angle) (the setting range is from -360.0° to 360.0°)																												
J	Rotation angle about the Y axis (pitch angle) (the setting range is from -360.0° to 360.0°)																												
K	Rotation angle about the Z axis (yaw angle) (the setting range is from -360.0° to 360.0°)																												

Note

- (1) If the address X, Y or Z is omitted, the address will be regarded as zero.
When all of addresses X, Y, and Z are set to "0", the feature coordinate system's zero point will be the same as that of the coordinate system before the inclined surface machining command is issued.
- (2) If the address I, J or K is omitted, the address will be regarded as zero.
- (3) If any address other than P, Q, X, Y, Z, I, J and K is included, a program error (P954) will occur.
- (4) A program error (P954) will occur if "q" is a value other than those listed above.



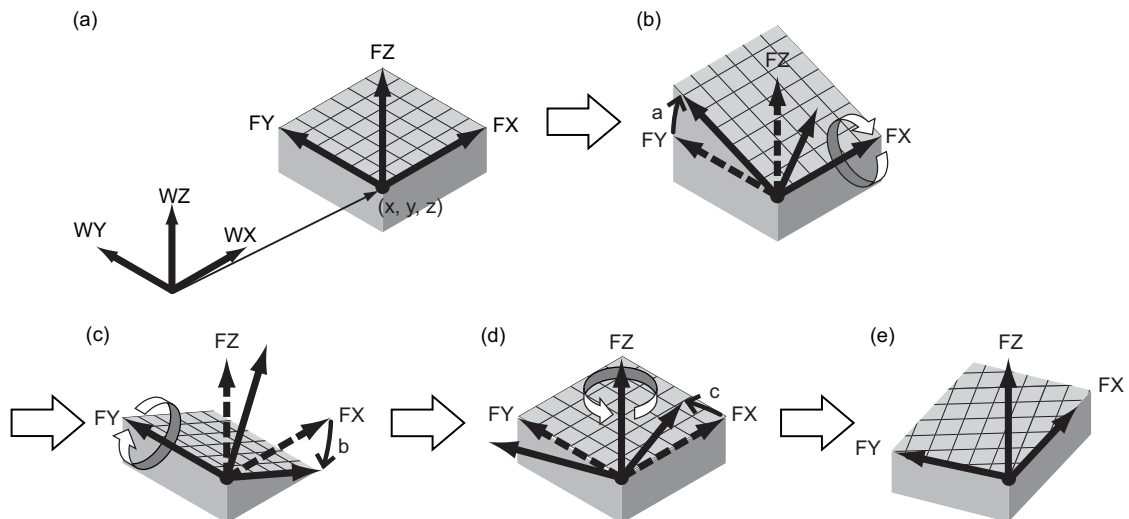
Detailed description

(Example) The feature coordinate system is established by the machining program as shown below.

G68.2 P1 Q123 Xx Yy Zz Ia Jb Kc;

(When q=123 [rotations in the order of WX, WY and WZ])

- (a) Designate the feature coordinate system's origin by x, y and z (coordinates that existed before the inclined surface machining command was issued).
- (b) Rotate the shifted coordinate system by angle "a" about the X axis of the coordinate system before issuing the inclined surface machining command. (Roll angle)
- (c) Rotate the coordinate system, which was defined after rotation in (b), by angle "b" about the Y axis of the coordinate system before issuing the inclined surface machining command. (Pitch angle)
- (d) Rotate the coordinate system, which was defined after rotation in (c), by angle "c" about the Z axis of the coordinate system before issuing the inclined surface machining command. (Yaw angle)
- (e) The coordinate system created in the above steps is the feature coordinate system.



19.4.3 How to Define Feature Coordinate System Using Three Points in a Plane



Command format

Inclined surface machining mode ON (define using three points in a plane)

G68.2 P2 Q0 X__ Y__ Z__ R__ ;	Shift amount setting
G68.2 P2 Q1 X__ Y__ Z__ ;	1st point coordinate setting
G68.2 P2 Q2 X__ Y__ Z__ ;	2nd point coordinate setting
G68.2 P2 Q3 X__ Y__ Z__ ;	3rd point coordinate setting

Q	Designate the points Designate from the 1st to the 3rd points, or specify by the shift distance. 0: Shift distance 1: The 1st point 2: The 2nd point 3: The 3rd point
X, Y, Z (Shift amount setting)	Shift amount between the 1st point and the feature coordinate system's origin Command by the incremental value with respect to the feature coordinate system before parallel shift.
R	The angle to rotate the feature coordinate system about the Z axis (the setting range is from -360.0° to 360.0°)
X, Y, Z (The 1st point)	Designate the feature coordinate system's origin using with the workpiece coordinate system's position. (*1)
X, Y, Z (The 2nd point)	Designate a point on the feature coordinate system's X axis (+ direction) using with the workpiece coordinate system's position. (*1)
X, Y, Z (The 3rd point)	Designate a point on the feature coordinate system's Y axis using with the workpiece coordinate system's position. (*1)

(*1) Command by the absolute values with respect to the coordinate system before issuing the inclined surface machining command.

Note

- (1) If the address Q is omitted, the address will be regarded as zero.
- (2) If the address X, Y or Z in Q0 to Q3 is omitted, the address will be handled as zero.
- (3) If the address R is omitted, the address will be regarded as zero.
- (4) If any address other than P, Q, X, Y, Z and R is included, a program error (P954) will occur.
- (5) A program error (P954) will occur in the following cases.
 - When any other command is included among G68.2 P2 Q0 to Q3.
 - When any of G68.2 P2 Q1 to Q3 is lacked.
 - When G68.2 P2 Q0 to Q3 are overlapped.
 - When a value other than 0 to 3 is commanded in the address Q.
 - When R is commanded in more than one block.
- (6) A program error (P955) will occur in the following cases.
 - When the same point was designated for two or more points among the 1st to the 3rd points.
 - When the three points exist on a straight line.
 - The distance between one of the three points and the straight line connecting the other two points is less than 0.1 (mm).

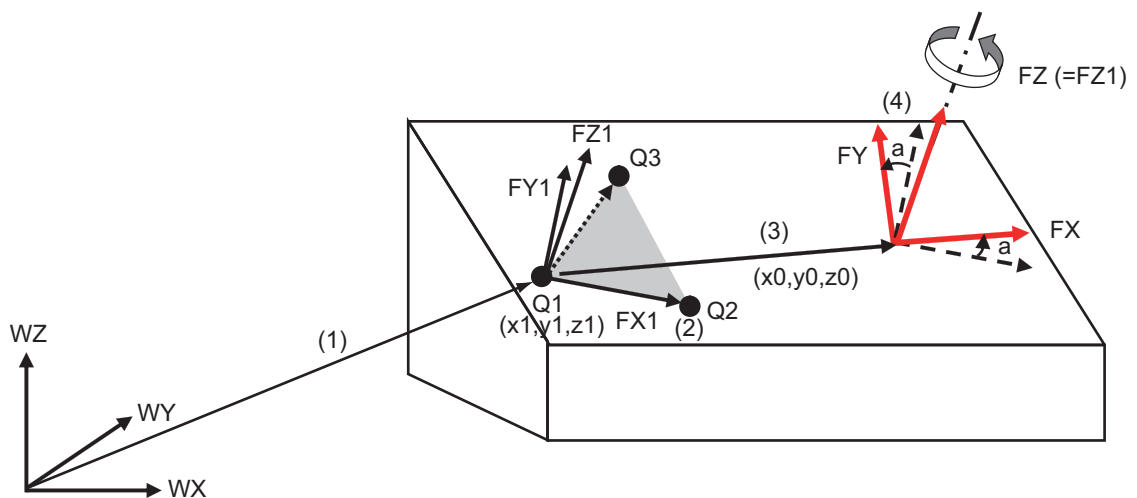


Detailed description

- (1) Designate the points Q1, Q2 and Q3 with respect to the coordinate system before issuing the inclined surface machining command. The point Q1 will be the origin of the feature coordinate system.
- (2) Define the X, Y and Z axes' directions of the feature coordinate system in the following procedure.
 - ♦ Feature coordinate system's X axis is in the direction from the 1st point (Q1) to the 2nd point (Q2). Normally, designate a point on Y axis (+ direction) as Q3. (If the commanded X axis and Y axis are not at perfect right angles, the Y axis will be automatically compensated to be at right angles to the X axis.)
 - ♦ Feature coordinate system's Z axis is in the direction of the cross product of $(Q2-Q1) \times (Q3-Q1)$.
 - ♦ Feature coordinate system's Y axis is determined with respect to the right-handed system.
- (3) When shift distance (x_0, y_0, z_0) of the feature coordinate system's origin is commanded, the feature coordinate system's origin is further parallel translated by (x_0, y_0, z_0) . Command the parallel translation distance with respect to the feature coordinate system before parallel translation. Always specify x_0, y_0 and z_0 by incremental value.
- (4) When the rotation angle a is commanded in the address R, the feature coordinate system is rotated by the angle "a" about the Z axis of the feature coordinate system.

(Example) The feature coordinate system is established by the machining program as shown below.

```
G68.2 P2 Q0 Xx0 Yy0 Zz0 Ra ;
G68.2 P2 Q1 Xx1 Yy1 Zz1 ;
G68.2 P2 Q2 Xx2 Yy2 Zz2 ;
G68.2 P2 Q3 Xx3 Yy3 Zz3 ;
```



Coordinate system before issuing the inclined surface machining command
(Workpiece coordinate system)

19.4.4 How to Define Feature Coordinate System Using Two Vectors



Command format

Inclined surface machining mode ON (define using two vectors)

```
G68.2 P3 Q1 X__ Y__ Z__ I__ J__ K__ ;
G68.2 P3 Q2 I__ J__ K__ ;
```

Q	Designate vectors Select the X axis direction vector or the Z axis direction vector. 1: X axis direction vector 2: Z axis direction vector
X, Y, Z	Feature coordinate system's origin Command by the absolute values with respect to the coordinate system before issuing the inclined surface machining command.
I, J, K	The feature coordinate system's X or Z axis direction vector When Q1 is commanded, the X axis direction vector is set. When Q2 is commanded, the Z axis direction vector is set. Command the direction with respect to the coordinate system before issuing the inclined surface machining command. The setting range is the same as the axis setting range, and the unit is dimensionless.

Note

- (1) If the address X, Y or Z is omitted, the address will be regarded as zero.
When all of addresses X, Y, and Z are set to "0", the feature coordinate system's origin will be the same as that of the coordinate system before the inclined surface machining command is issued.
- (2) If the address I, J or K in G68.2 P3 Q1 and Q2 is omitted, the omitted value will be handled as zero.
- (3) If any address other than P, Q, I, J and K is included, a program error (P954) will occur. (X, Y and Z are possible to command in G68.2 P3 Q1)
- (4) A program error (P954) will occur in the following cases.
 - When any other command is included between G68.2P3 Q1 and Q2.
 - When either G68.2 P3 Q1 or Q2 is lacked.
 - When G68.2 P3 Q1 and Q2 are overlapped.
 - When a value other than 1 to 2 is commanded in the address Q.
 - When the address Q is omitted
- (5) A program error (P955) will occur in the following cases.
 - When all of addresses I, J, and K are set to "0":
 - When the angle formed by the feature coordinate system's X and Z vectors is not a right angle, and the deviation is 5 degrees or more.



Detailed description

- (1) Designate the feature coordinate system's origin by x , y and z (coordinates that existed before the inclined surface machining command was issued).
- (2) Define the X , Y and Z axes' directions of the feature coordinate system in the following procedure.
 - ♦ Feature coordinate system's X axis positive direction is $rx = (ix, jx, kx)$.
 - ♦ Feature coordinate system's Y axis positive direction is that of the cross product of $(iz, jz, kz) \times (ix, jx, kx)$.
 - ♦ The feature coordinate system's Z axis is determined with respect to the right-handed system.

The direction of $rx=(ix, jx, kx)$ is the X axis of the feature coordinate system.

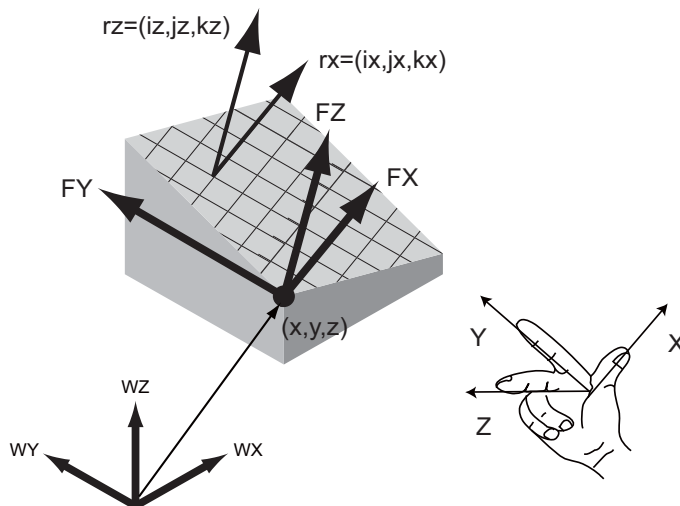
Normally, the direction of $rz=(iz, jz, kz)$ is the Z axis (positive direction) of the feature coordinate system.

(If rx and rz are not at perfect right angle to each other, they will be automatically compensated so that they are at right angle to the X axis.)

(Example) The feature coordinate system is established by the machining program as shown below.

```
G68.2 P3 Q1 Xx Yy Zz Iix Jjx Kkx ;
```

```
G68.2 P3 Q2 Iiz Jjz Kkz ;
```



Coordinate system before issuing the inclined surface machining command
(Workpiece coordinate system)

19.4.5 How to Define Feature Coordinate System Using Projection Angles

**Command format****Inclined surface machining mode ON (define using projection angles)**

```
G68.2 P4 X__ Y__ Z__ I__ J__ K__ ;
```

X, Y, Z	Feature coordinate system's origin Command by the absolute values with respect to the coordinate system before issuing the inclined surface machining command.
I	The angle to rotate the X axis about the Y axis of the coordinate system before issuing the inclined surface machining command (the setting range is from -360.0° to 360.0°)
J	The angle to rotate the Y axis about the X axis of the coordinate system before issuing the inclined surface machining command (the setting range is from -360.0° to 360.0°)
K	The rotation angle about the Z axis of the feature coordinate system (the setting range is from -360.0° to 360.0°)

Note

- (1) If the address X, Y or Z is omitted, the address will be regarded as zero.
When all of addresses X, Y, and Z are set to "0", the feature coordinate system's origin will be the same as that of the coordinate system before the inclined surface machining command is issued.
- (2) If the address I, J or K is omitted, the omitted value will be handled as zero.
- (3) If any address other than P, X, Y, Z, I, J and K is included, a program error (P954) will occur.
- (4) A program error (P955) will occur when the angle formed by the X axis after rotating by the angle designated with address I about the Y axis, and the Y axis after rotating by the angle designated with address J about the X axis is 1 degree or less.

**Detailed description**

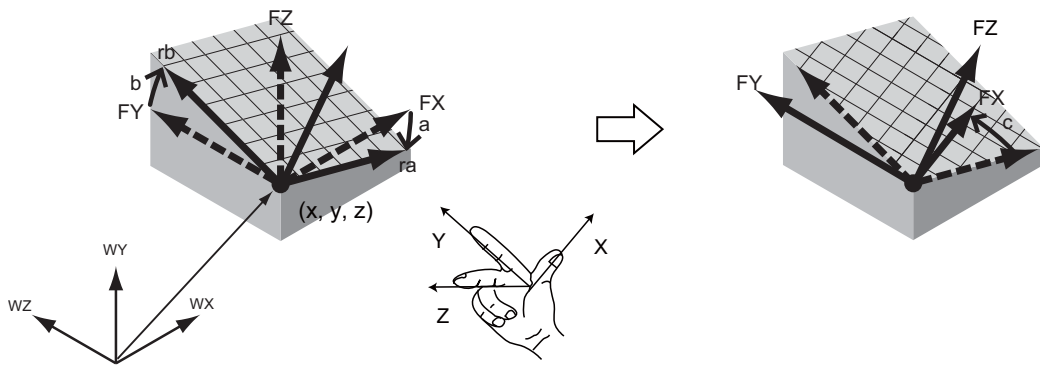
- (1) Designate the feature coordinate system's origin by x, y and z (coordinates that existed before the inclined surface machining command was issued).
- (2) Define the X, Y and Z axes' directions of the feature coordinate system in the following procedure.
 - The direction in which the X axis of the coordinate system before issuing the inclined surface machining command is rotated by angle a about the Y axis is defined as "ra".
 - The direction in which the Y axis of the coordinate system before issuing the inclined surface machining command is rotated by angle b about the X axis is defined as "rb".
 - Feature coordinate system's Z axis is in the direction of the cross product of (ra × rb).
 - Feature coordinate system's X axis is in the direction determined by rotating "ra" by the angle "c" about the feature coordinate system's Z axis.
 - Feature coordinate system's Y axis is determined with respect to the right-handed system.

Note

- If "ra" and "rb" are considered to be parallel (or if the angle formed by the two vectors is 1 degree or less), a program error (P955) will occur.
- Except XZ and YZ plane, it is not possible to designate a plane that is in parallel with Z axis.

(Example) The feature coordinate system is established by the machining program as shown below.

G68.2 P4 Xx Yy Zz Ia Jb Kc ;



Coordinate system before issuing the inclined surface machining command
(Workpiece coordinate system)

19.4.6 Define by Selecting the Registered Machining Surface

**Command format****Inclined surface machining mode ON (define R-Navi machining surface)**

G68.2 P10 Q__ D__ ;

Q	Workpiece No. designated by R-Navi or workpiece name designated by R-Navi
D	Machining surface No. designated by R-Navi or machining surface name designated by R-Navi

Note

- (1) When address Q is omitted, select workpiece No. 1 if Q0,Q1 is commanded.
- (2) When address D is omitted, select machining surface No. 1 (BASE-SURFACE) if D0,D1 is commanded.
- (3) If any address other than P, Q, and D is designated when G68.2 P10 is commanded, a program error (P954) will occur.
- (4) A program error (P954) will occur when:
 - a value other than 0 to 10 is commanded in address Q or an undefined workpiece name is designated;
 - a value other than 0 to 17 is commanded in address D or an undefined machining surface name is designated;
 - the workpiece name does not include a character string (command represented by "Q<>"); and
 - the machining surface name does not include a character string (command represented by "D<>").
- (5) If no feature coordinate system can be defined for the selected machining surface, a program error (P956) will occur.
- (6) If there are multiple workpieces or machining surfaces of the same name when the workpiece name or machining surface name is designated, a lower number is selected.
- (7) When the machining surface is called from the program, [SEL] or [*] is not displayed on the screen. Various PLC signals are not set to ON. (R-Navi machining surface selecting signal (XD28), R-Navi selecting workpiece No. signal (R660), R-Navi selecting machining surface No. signal (R661))
- (8) When the machining surface is called from the program, the basic coordinate system designated by R-Navi for each workpiece is not selected. Before "G68.2 P10" is commanded, select a workpiece coordinate system from the program.
- (9) If you have defined a workpiece No. in address Q, define a machining surface No. in address D.
If you have defined a workpiece name in address Q, define a machining surface name in address D.

19.4.7 How to Define Feature Coordinate System Using Tool Axis Direction

**Command format**

Inclined surface machining mode ON (define using tool axis direction)

G68.3 X__ Y__ Z__ R__;

X, Y, Z	Feature coordinate system's origin Command by the absolute values with respect to the coordinate system before issuing the inclined surface machining command.
R	The angle to rotate the feature coordinate system about Z axis (the setting range is from -360.0° to 360.0°)

Note

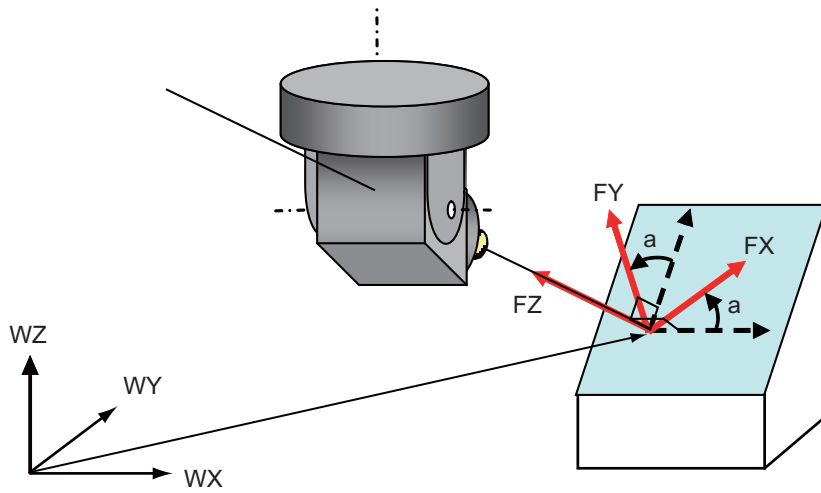
- (1) If the address X, Y or Z is omitted, the address will be regarded as zero.
When all of addresses X, Y, and Z are set to "0", the feature coordinate system's origin will be the same as that of the coordinate system before the inclined surface machining command is issued.
- (2) If the address R is omitted, the omitted value will be handled as zero.
- (3) If any address other than X, Y, Z and R is included, a program error (P954) will occur.

**Detailed description**

- (1) Designate the feature coordinate system's origin by x, y and z (coordinates that existed before the inclined surface machining command was issued).
- (2) Define the X, Y and Z axes' directions of the feature coordinate system in the following procedure.
 - ♦Feature coordinate system's Z axis is in the tool axis direction.
 - ♦Feature coordinate system's X axis is in the direction of the X axis of the coordinate system before issuing the inclined surface machining command after rotating with the tool. (When all the tool-side rotary axes are at 0 degrees (machine value), the feature coordinate system's X axis will be in the same direction as the X axis of the coordinate system before issuing the inclined surface machining command.)
 - ♦Feature coordinate system's Y axis is in the direction of the Y axis of the coordinate system before issuing the inclined surface machining command after rotating with the tool. (When all the tool-side rotary axes are at 0 degrees (machine value), the feature coordinate system's Y axis will be in the same direction as the Y axis in the coordinate system before issuing the inclined surface machining command.)
 - ♦Feature coordinate system is finally established by rotating the commanded angle with address R about the Z axis.

(Example) The feature coordinate system is established by the machining program as shown below.

G68.3 Xx Yy Zy Ra;



Coordinate system before issuing the inclined surface machining command
(Workpiece coordinate system)

19.4.8 Tool Axis Direction Control; G53.1/G53.6

**Function and purpose**

A rotary axis is automatically moved so that the tool axis direction (direction from the tool's tip to the bottom) will be the feature coordinate system's +Z axis direction. For table tilt type and combined type machines, feature coordinate system may change in accordance with the rotation of the table rotary axis.

The following two types of tool axis direction control can be utilized:

Type 1 (G53.1 command)	Only the rotary axis is moved.
Type 2 (G53.6 command)	The rotary axis and the orthogonal axis are moved by fixing the tool center point position in the view from the workpiece.

**Command format**

Tool axis direction control (type 1): Only the rotary axis is moved.

G53.1 P__ ;

Tool axis direction control (type 2): The rotary axis and the orthogonal axis are moved by fixing the tool center point position in the view from the workpiece.

G53.6 P__ Q__ H__ ;

P	Select a solution for the rotary axis 0: Select a default solution for each machine type 1: Select a solution so that the primary rotary axis rotation is positive. 2: Select a solution so that the primary rotary axis rotation is negative.
Q	Select the rotation order of the rotary axes when the number of simultaneous contouring control axes is 4 axes or less and the operation at the time of G53.6 command is limited to simultaneous 4 axes (3 orthogonal axes + 1 rotary axis) or less. (In the following example, there are 2 rotary axes.) 0: The axes operate in the order set in the "#7917 SLCT_G53_6_ROTAX" parameter. 1: The axes operate in the order of primary rotary axis and secondary rotary axis. 2: The axes operate in the order of secondary rotary axis and primary rotary axis. Even if the number of simultaneous contouring control axes is 5 axes or more, the number can be limited to the simultaneous 4 axes (3 orthogonal axes + 1 rotary axis) or fewer by address Q command. However, when address Q command is "0", 5 axes operate simultaneously regardless of the parameter settings.
H	Tool length offset No.

G53.1/G53.6 are group 00.

Note

- (1) Command G53.1/G53.6 during inclined surface machining mode. If commanded in any other mode, a program error (P953) will occur.
- (2) Make sure to command G53.1/G53.6 surely in a block. If this command is issued in the same block as of other G codes or travel command etc., a program error (P953) will occur.
- (3) The travel speed when G53.1 is commanded follows the G group 1 modal (such as G00/G01) during the tool axis direction control command.
- (4) The travel speed on the feature coordinate system when G53.6 is commanded follows G group 1 modal (such as G00/G01). The travel speed of each axis may exceed the command speed as the tool tip position is fixed in the view from workpiece. However, rapid traverse (G00) is clamped by the parameter "#2001 rapid", and the cutting feed (G01) is clamped by the parameter "#2002 clamp". (These parameters depend on the MTB specifications.)

- (5) The behavior when the address P is omitted depends on the setting of parameter "#7918 SLCT_ROTAX_ANS" (MTB specifications). If any other value than 0, 1, and 2 is command, a program error (P35) will occur.
- (6) If any address other than P/N is commanded while commanding G53.1, a program error (P953) will occur.
- (7) When address Q is omitted at the time of G53.6 command, it is regarded that 0 is commanded. If any other value than 0, 1, and 2 is command, a program error (P35) will occur.
- (8) If any address other than P/H/N is commanded while commanding G53.6, a program error (P953) will occur.
- (9) When the address H is omitted, H modal commanded before G53.6 command will be applied. If H modal is not commanded, a program error (P953) will occur.

(Example 1)

```

:
G43 H1
:
G53.6 ← Use H1
:

```

(Example 2)

```

:
G53.6 ← Error (P953)
:

```

- (10) If the tool length offset No. is changed by address H command, a program error (P953) will occur.

(Example 1)

```

:
G43 H1 ← Command the tool offset No.1
:
G53.6 H2 ← If tool length offset No.2 is commanded, a program error (P953) will occur.
:

```

- (11) If the offset amount for the tool No. that the address H is commanded is "0", a program error (P957) will occur.

(Example 1) When "H1 = 0"

```

:
G43 H1
:
G53.6 ← Error (P957)
:

```

(Example 2) When "H1 ≠ 0"

```

:
G43 H1
:
H0 ← Tool length offset is "0"
: while it remains G43 modal.
G53.6 ← Error (P957)
:

```

(Example 3) When "0" is commanded to address H

```

:
G53.6 ← Error (P957)
:

```



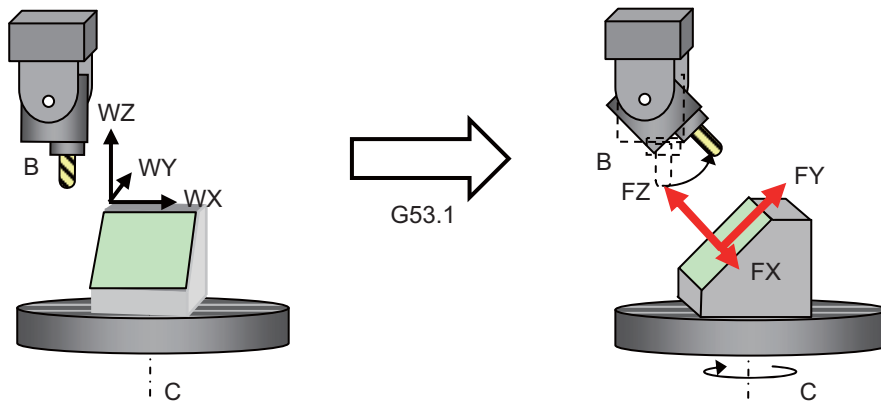
Detailed description

The operation of type 1 (G53.1)

For the G53.1 command, the 3 orthogonal axes (X, Y, and Z axes) do not move, however, only the 2 rotary axes rotate simultaneously so that the tool axis direction is in line with the +Z direction of the feature coordinate system.

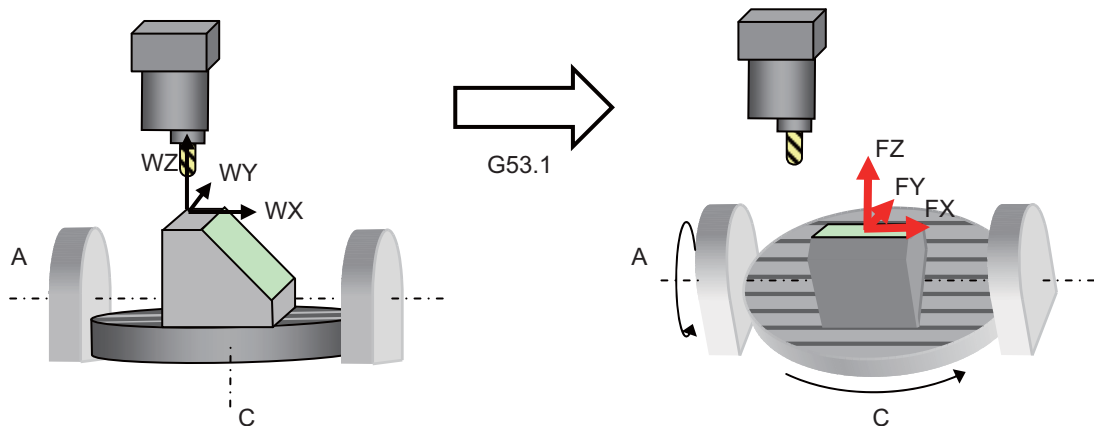
(1) For combined type B-C axes

When the G53.1 command is issued for a combined type (B-C axes) machine, the B axis of the tool and the C axis of the table rotate simultaneously.



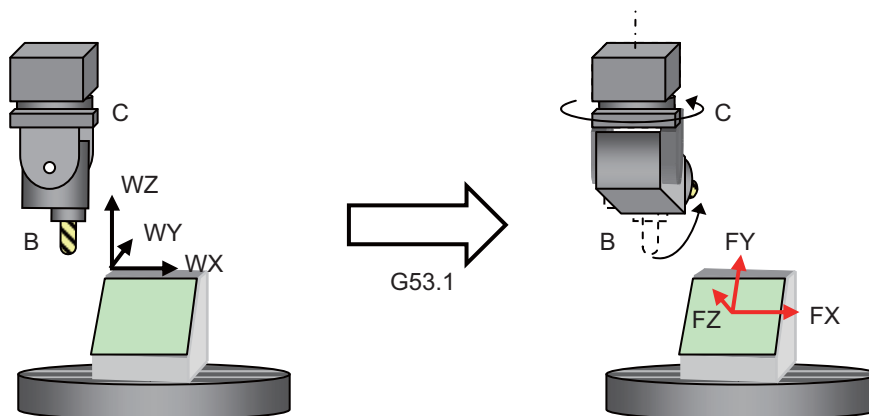
(2) For table tilt type A-C axes

When the G53.1 command is issued for a table tilt type (A-C axes) machine, the A and C axes of the table rotate simultaneously.



(3) For tool tilt type B-C axes

When the G53.1 command is issued for a tool tilt type (B-C axes) machine, the B and C axes of the tool rotate simultaneously.



Type 2 (G53.6) command

For G53.6 command, the tool axis end position of the tool axis direction seen from workpiece is fixed to be in +Z direction in the feature coordinate system, and up to 3 orthogonal axes (x axis, Y axis, and Z axis) and 2 rotary axes moves simultaneously.

The number of axes to be moved simultaneously is limited by the number of simultaneous contouring control axes. If the number of simultaneous contouring control axes is 4 axes or fewer, the rotary axes move separately.

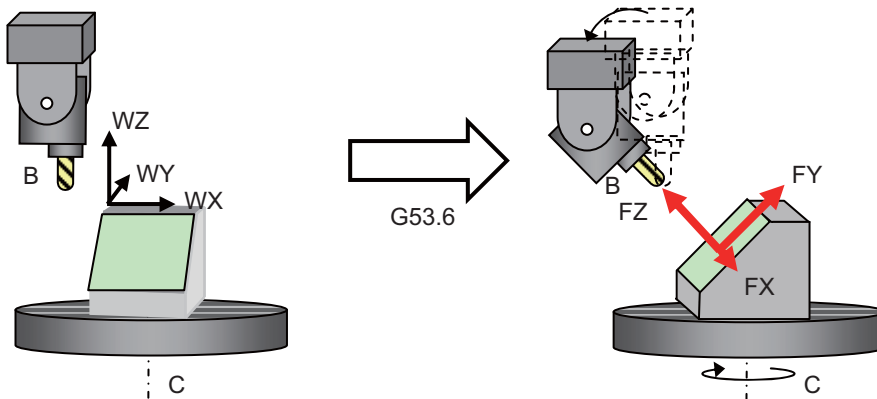
The order of rotary axes moving separately can be specified with address Q command. When the number of simultaneous contouring control axes is 5 axes or more, the rotary axes can also be moved separately by issuing the address Q command.

[When the number of simultaneous contouring control axes is 5 axes or more]

For G53.6 command, up to 3 orthogonal axes(X axis, Y axis, and Z axis) and 2 rotary axes can be moved simultaneously.

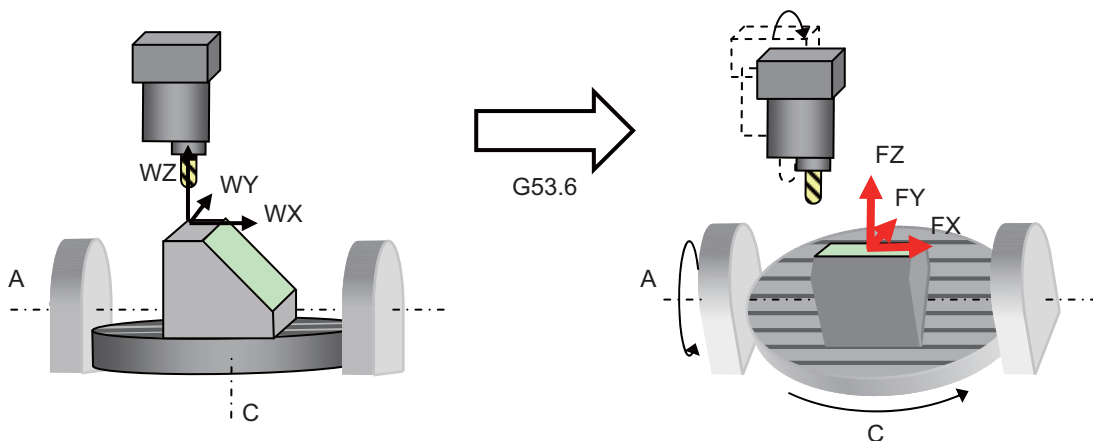
(1) For combined type B-C axes

When the G53.6 command is issued for a combined type (B-C axes) machine, the X, Y, Z, and B axes of the tool and the C axis of the table move simultaneously.



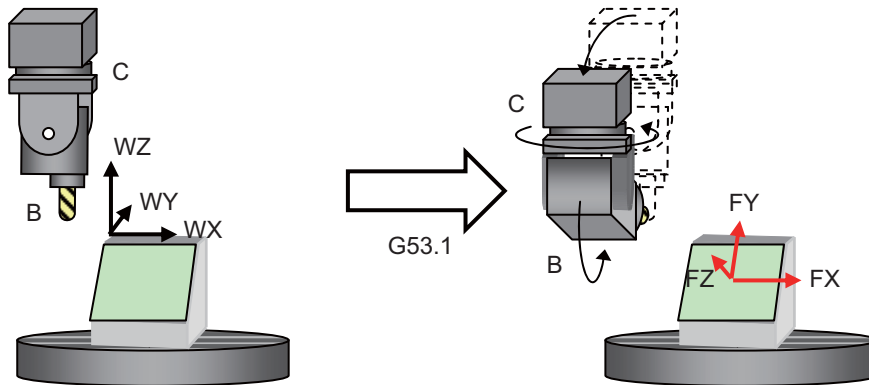
(2) For table tilt type A-C axes

When the G53.6 command is issued for a table tilt type (A-C axes) machine, the X, Y, and Z axes of the tool and the A and C axes of the table move simultaneously.



(3) For tool tilt type B-C axes

When the G53.6 command is issued for a tool tilt type (B-C axes) machine, the X, Y, Z, B, and C axes of the tool rotate simultaneously.



[When the number of simultaneous contouring control axes is 4 axes or fewer]

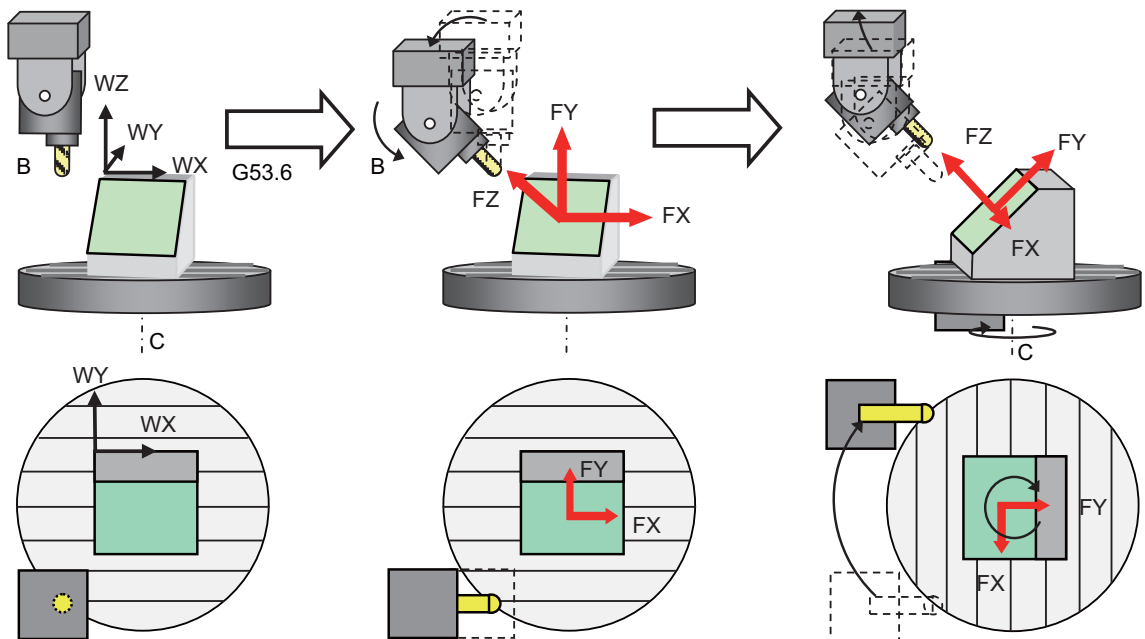
Up to 3 orthogonal axes (X, Y, Z axes) and a rotary axis move simultaneously with G53.6 command. When 2 rotary axes move, the order of rotary axes to move is specified with the parameter "#7917 SLCT_G53_6_ROTAX" or address Q. When specifying "1" to the address Q, the axis moves in the order of primary rotary axis and secondary rotary axis. When setting "2" to the address Q, the axis moves in the order of the secondary axis and the primary axis. When executing the single block operation, either performing the block stop or not at the movement completion for each rotary axis can also be specified with parameter "#8132 G53.6 block stop".

(1) For combined type B-C axis

(a) When moving in the order of primary rotary axis and secondary rotary axis

First, the B axis ("B" in the figure below) of the tool rotates, and then the X, Y, and Z axes of the tool also move to fix the tool center position.

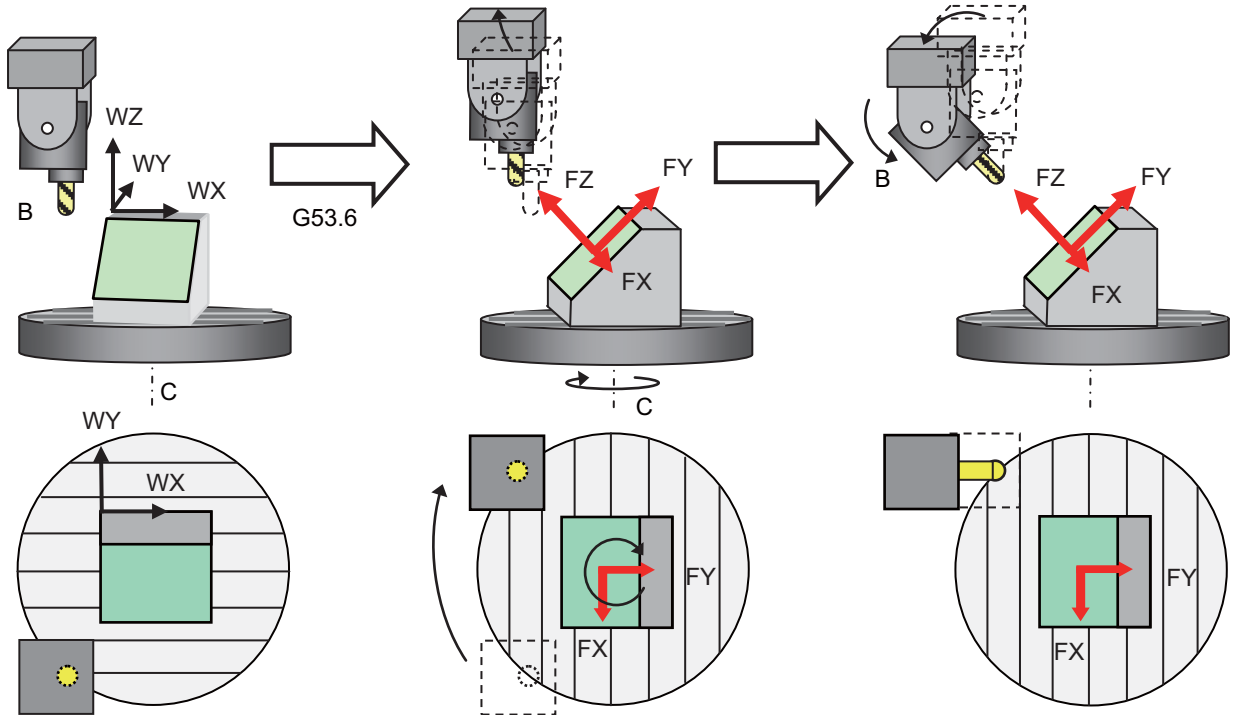
Next, the C axis ("C" in the figure below) of the table rotates, and then the X, Y, and Z axes of the tool also move as if the tool follows the workpiece movement.



(b) When moving in the order of secondary rotary axis and primary rotary axis

First, the C axis ("C" in the figure below) of the table rotates, and then the X, Y, and Z axes of the tool also move as if the tool follows the workpiece movement.

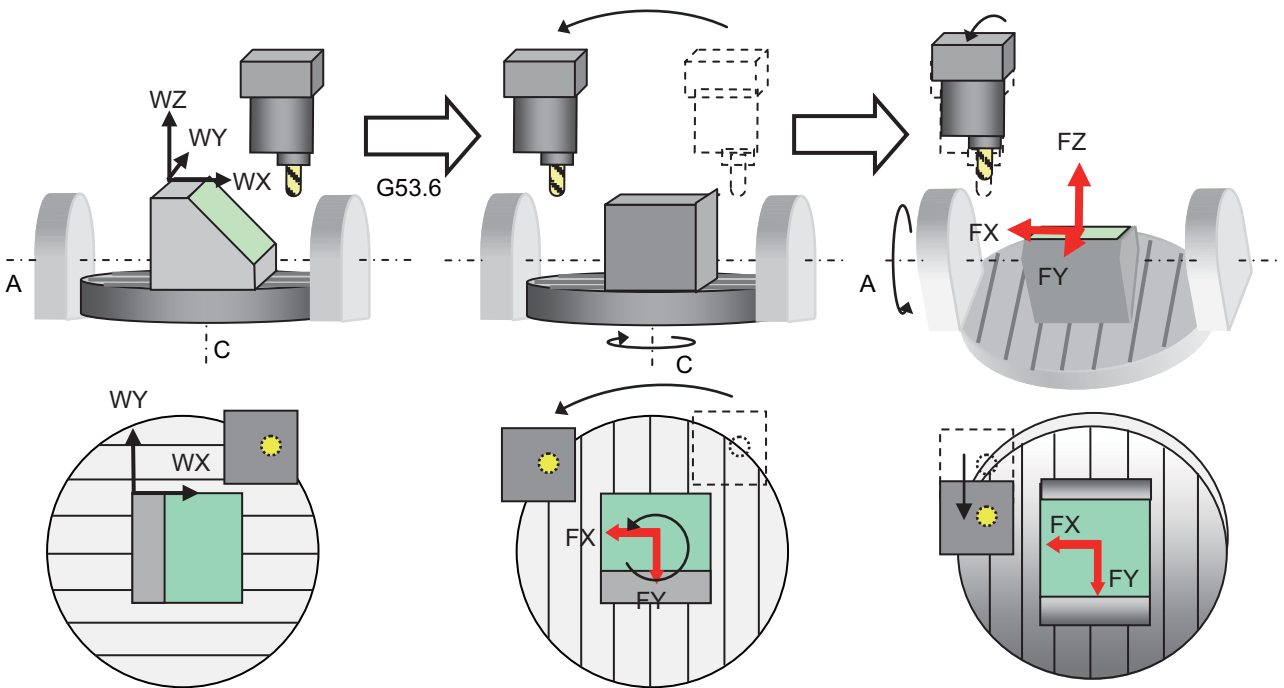
Next, the B axis ("B" in the figure below) of the tool rotates, and then the X, Y, and Z axes of the tool also move to fix the tool center position.



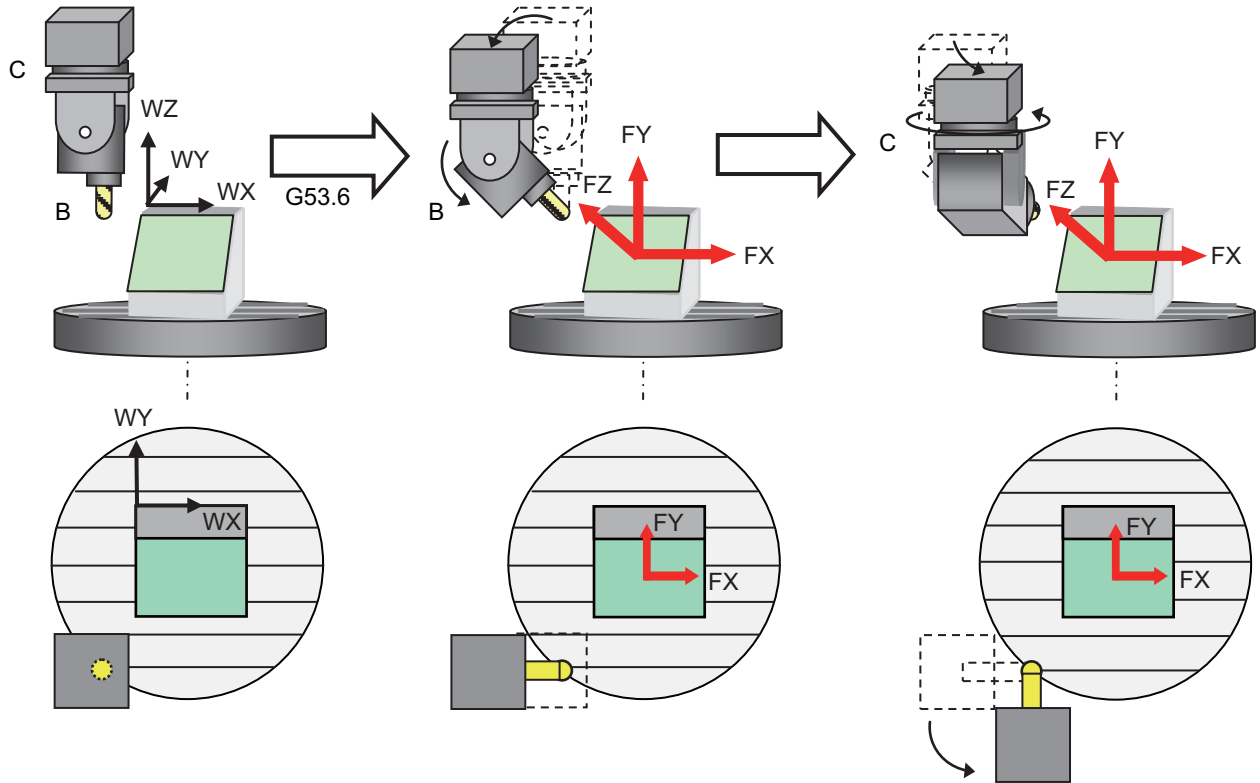
(2) For table tilt type A-C axes (When moving in the order of secondary rotary axis and primary rotary axis)

First, the C axis ("C" in the figure below) of the table rotates, and then the X, Y, and Z axes of the tool also move as if the tool follows the workpiece movement.

Next, the A axis ("A" in the figure below) of the table rotates, and then the X, Y, and Z axes of the tool also move as if the tool follows the workpiece movement.



(3) For tool tilt type B-C axis (When moving in the order of primary rotary axis and secondary rotary axis)
 First, the B axis ("B" in the figure below) of the tool rotates, and then the X, Y, and Z axes of the tool also move to fix the tool center position.
 Next, the C axis ("C" in the figure below) of the tool rotates, and then the X, Y, and Z axes of the tool also move to fix the tool center position.



Select rotary axis' solution

When G53.1 is commanded, there are normally two types of solutions for the rotary axis' calculated angle; one is to rotate the primary rotary axis positively, and the other negatively. Use the address P (P=0, 1 or 2) in G53.1 command to select either one of the solutions.

These are the default solutions for each machine type.

When P is "0": Selects a default solution for each machine type

When P is "1": Selects a solution so that the primary rotary axis rotation is positive

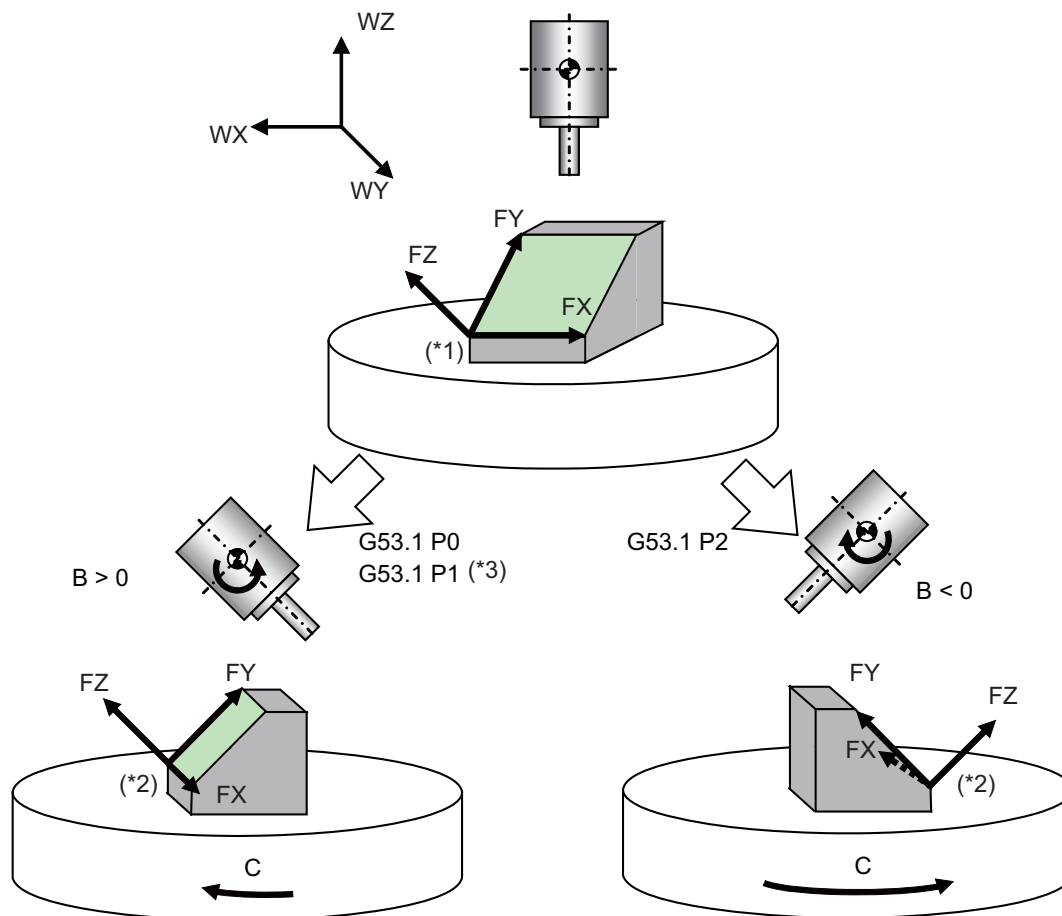
When P is "2": Selects a solution so that the primary rotary axis rotation is negative

The behavior when the address P is omitted depends on the setting of parameter "#7918 SLCT_ROTAX_ANS" (MTB specifications). If any other value than 0,1, and 2 is command, a program error (P35) will occur.

These are the default solutions for each machine type.

Machine type	Primary rotary axis	Solution selected by default
Tool tilt type	Tool-side 2nd rotary axis	Selects a solution so that the primary rotary axis rotation is positive (same as when P is "1")
Table tilt type	Table-side 2nd rotary axis	Selects a solution so that the primary rotary axis rotation is negative (same as when P is "2")
Combined type	Tool-side rotary axis	Selects a solution so that the primary rotary axis rotation is positive (same as when P is "1")

Primary rotary axis is the rotary axis which serves as the criteria for selecting the solution in G53.1 command.



(*1) Indicates the 1st feature coordinate system.

(*2) Indicates the 2nd feature coordinate system.

(*3) For combined type machines, a solution that makes the primary rotary axis rotation positive is selected as a result of issuing the G53.1P0 command.

19.4.9 Details of Inclined Surface Machining Operation

**Detailed description****Operation during inclined surface machining mode**


When inclined surface machining is commanded, the above-mentioned feature coordinate system is defined. By setting the parameters #8901 to #8906 to "23", you can display the coordinates of the feature coordinate system on the counter (no machine motion). The travel commands during inclined surface machining mode are handled with respect to the feature coordinate system.

In the counter display of the feature coordinate system, whether the machining position on the program command that does not include the tool length compensation/tool radius compensation can be selected depends on the MTB specifications (parameter "#1287 ext23/bit1, bit2 (inclined surface coordinate display)").

Tool axis direction control

When G53.1 is commanded, the rotary axis moves so that the tool axis direction will be + Z direction of the feature coordinate system. At this time, the rotary axis moves, but X, Y and Z axes won't move. The rotary axis' travel speed is determined based on the modal when G53.1 is commanded.


CAUTION

 Depending on the feature coordinate system setting, rotary axis may move greatly in response to G53.1 command. Thus, before commanding G53.1, move the tool far enough away from the table.

Cancel inclined surface machining mode

The command G69 cancels the inclined surface machining. When this mode is canceled, the feature coordinate system setting will be canceled, the coordinate system will change back to the workpiece coordinate system when inclined surface machining was commanded, and workpiece coordinate position counter will change back to the previous workpiece coordinate system's coordinates (no machine motion). By inputting Reset, the inclined surface machining is also canceled.

(If the parameter "#1151 rstint" or "#1210 RstGmd/bitF" is set to "0", however, the inclined surface machining mode is kept even when Reset is input.)



Program example

Program example 1

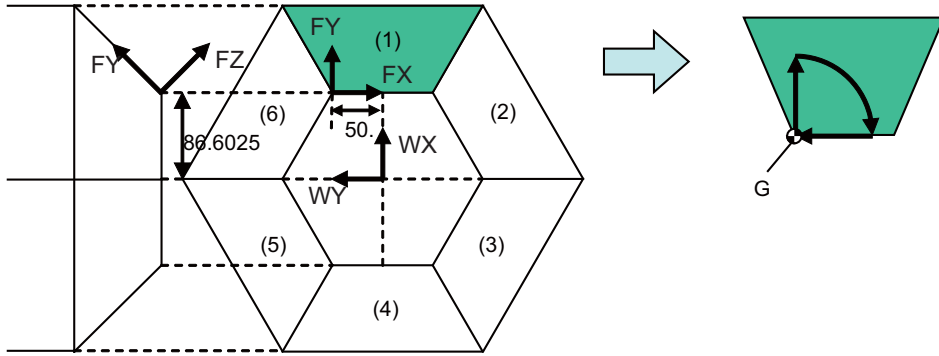
The machining program #10 is for machining an identical shape on each face of the hexagonal column using a combined type machine. Feature coordinate systems on each face are defined in the blocks N1 to N6, and then same machining is performed using the subprogram (Machining program #100). The workpiece origin is deemed to be at the center of the hexagonal column's end-face.

Machining program #10

N1 G68.2 X86.6025 Y50. Z0. I-90. J-45. K0.;	Machining on the face (1)
M98 P100;	
G69;	
G00 Z200.;	
N2 G68.2 X86.6025 Y-50. Z0. I-150. J-45. K0.;	Machining on the face (2)
M98 P100;	
G69;	
G00 Z200.;	
N3 G68.2 X0. Y-100. Z0. I-210. J-45. K0.;	Machining on the face (3)
M98 P100;	
G69;	
G00 Z200.;	
N4 G68.2 X-86.6025 Y-50. I-270. J-45. K0.;	Machining on the face (4)
M98 P100;	
G69;	
G00 Z200. ;	
N5 G68.2 X-86.6025 Y50. I-330. J-45. K0.;	Machining on the face (5)
M98 P100;	
G69;	
G00 Z200. ;	
N6 G68.2 X0. Y100. I-30. J-45. K0.;	Machining on the face (6)
M98 P100;	
G69;	
G00 Z200. ;	
M30	

Machining program #100

```
G53.1;
G90 G00 X0. Y0. Z5.;
G01 Z-5.F500;
G01 Y20. F1000;
G02 X20. Y0. R20. F1000;
G01 X0.F1000;
M99;
```



G: Feature coordinate system's origin

Program example 2

The machining program #10 to #15 are for machining a shape on an inclined surface of the cube as shown in the figure next page. The feature coordinate system is defined by designating the inclined surface in each main program, and then same machining is done using the subprogram (Machining program #100).

Machining program #10 Euler angles

```
N1 G28XYZBC;  
  G54X0Y0Z0;  
  M200;  
  G68.2 X33.3333 Y 33.3333 Z66.6666 I-45 J54.7356 K0;  
  M98 P100;  
  G69;
```

Machining program #11 Roll-pitch-yaw angles

```
N2 G28XYZBC;  
  M200;  
  G68.2 P1 Q321 X33.3333 Y 33.3333 Z66.6666 I45 J-35.2644 K-30;  
  M98 P100;  
  G69;  
  M30;
```

Machining program #12 Three points in a plane

```
N3 G28XYZBC;  
  G54X0Y0Z;  
  M200;  
  G68.2 P2 Q0 X0 Y-18.7503 Z0 R0;  
  G68.2 P2 Q1 X50 Y50 Z100;  
  G68.2 P2 Q2 X50 Y0 Z50;  
  G68.2 P2 Q3 X50 Y50 Z100;  
  M98 P100;  
  G69;  
  M30;
```

Machining program #13 Two vectors

```
N4 G28XYZBC;  
  G54X0Y0Z0;  
  M200;  
  G68.2 P3 Q1 X33.3333 Y 33.3333 Z66.6666 J-100 K0;  
  G68.2 P3 Q2 I-100 J-100 K100;  
  M98 P100;  
  G69;  
  M30;
```

Machining program #14 Projection angles

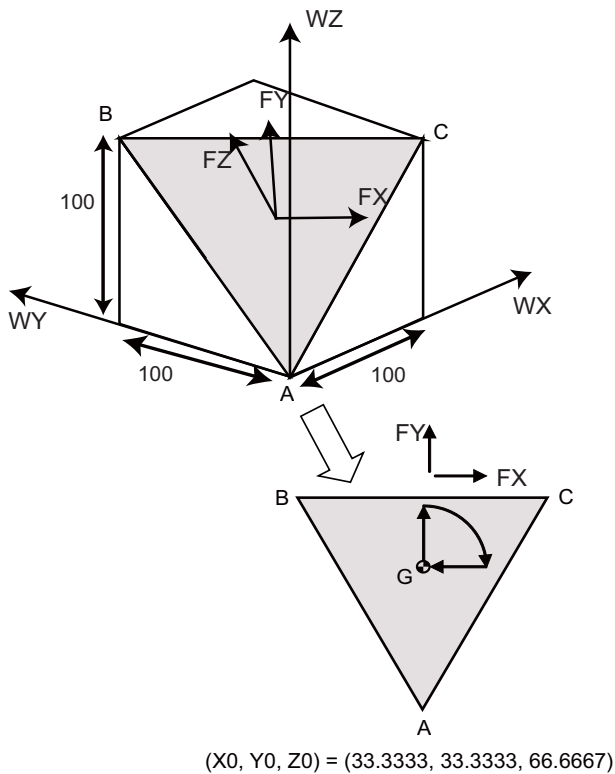
```
N5 G28XYZBC;
  G54X0Y0Z0;
  M200;
  G68.2 P4 X33.3333 Y 33.3333 Z66.6666 I-45 J45 K-60;
  M98 P100;
  G69;
  M30;
```

Machining program #15 Tool axis direction basis

```
N6 G28XYZBC;
  G54X0Y0Z0;
  M200;
  B-45. C45.;
  G68.3 X33.3333 Y33.3333 Z66.6667 R0.;
  M98 P100;
  M69;
  M30;
```

Machining program #100

```
G53.1;
  G90G00X0.Y0.Z0.B0.C0.;
  G00X0Y0Z0;
  G01 Y50.F1000;
  G02 X50. Y0. R50. F1000;
  G01 X0.F1000;
  M99;
```



G: Feature coordinate system's origin

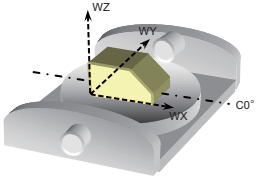
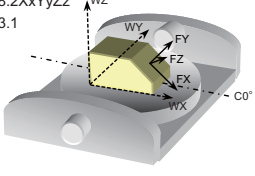
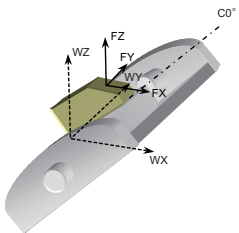
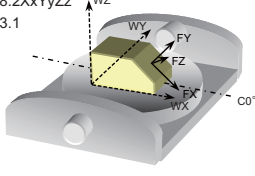
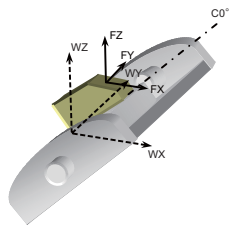
19.4.10 Rotary Axis Basic Position Selection

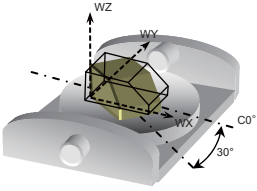
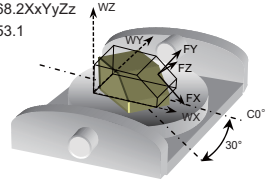
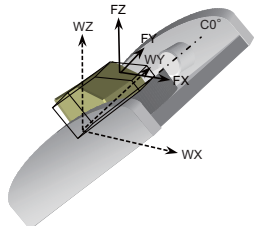
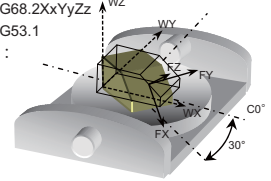
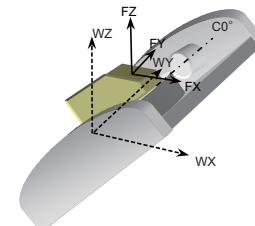
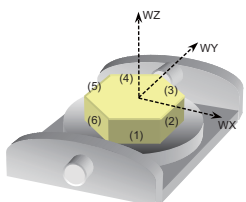


Detailed description

When inclined surface machining is commanded, the basic position for establishing the feature coordinate system's origin can be set with the parameter ("#7915 SLCT_SLOPE_CRD_MOD" (Rotary axis basic position in inclined surface machining)).

There are two types of basic position selection. One is to set the feature coordinate system in view from the workpiece coordinate system independently of the rotary axis' position when inclined surface machining is commanded (Start position basis), and the other is to set the feature coordinate system in view from a workpiece coordinate system which is determined regardless of the rotary axis position when inclined surface machining is commanded (Zero degree position basis).

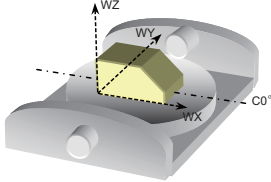
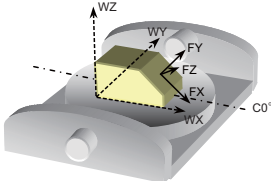
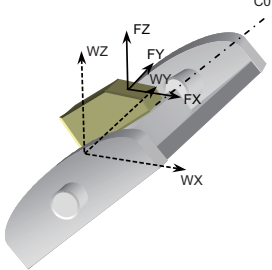
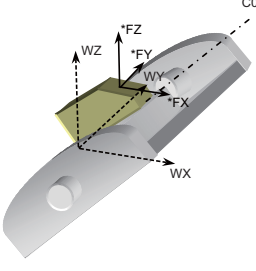
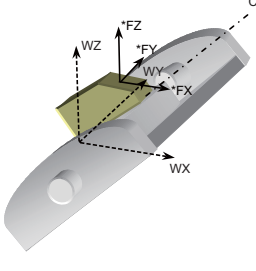
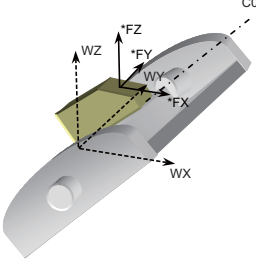
	Rotary axis basic position in inclined surface machining	
	Start position basis (#7915=1)	Zero degree position basis (#7915=0)
<p>When workpiece is placed in the workpiece coordinate system direction: Workpiece coordinate offset A0. C0.</p> 	<p>: G90 G54 A0. C0. G68.2XxYyZz G53.1 :</p>  <p>When G68.2 is commanded: Feature coordinate system is defined at a position in view from a workpiece coordinate system.</p>  <p>When G53.1 is commanded: The tool axis direction matches the Z axis direction of the feature coordinate system, which has been defined with G68.2.</p>	<p>: G90 G54 A0. C0. G68.2XxYyZz G53.1 :</p>  <p>When G68.2 is commanded: Feature coordinate system is defined at a position in view from a workpiece coordinate system regardless of rotary axis position.</p>  <p>When G53.1 is commanded: The tool axis direction matches the Z axis direction of the feature coordinate system, which has been defined with G68.2.</p>

Rotary axis basic position in inclined surface machining		
	Start position basis (#7915=1)	Zero degree position basis (#7915=0)
<p>When workpiece is placed deviated from the workpiece coordinate system: Workpiece coordinate offset A0. C30.</p> 	<p>: G90 G54 A0. C30. G68.2XxYyZz G53.1 :</p>  <p>When G68.2 is commanded: Feature coordinate system is defined at a position in view from a workpiece coordinate system.</p>  <p>When G53.1 is commanded: The tool axis direction does not match the Z axis direction of the feature coordinate system, which has been defined with G68.2 command.</p>	<p>: G90 G54 A0. C30. G68.2XxYyZz G53.1 :</p>  <p>When G68.2 is commanded: Feature coordinate system is defined at a position in view from a workpiece coordinate system regardless of rotary axis position.</p>  <p>When G53.1 is commanded: The tool axis direction matches the Z axis direction of the feature coordinate system, which has been defined with G68.2.</p>
<p>Example) Polygon machining: Machining an identical shape on each of the six surfaces.</p> 	<p>(Subprogram) : G68.2 Xx Yy Zz li Jj Kk G53.1 G01 Xx Ff : G69 M99 Create a machining shape in the subprogram using inclined surface machining and tool axis direction control. Need to consider the rotary axis' angle before calling the subprogram.</p>	<p>(Main program) : G68.2 Xx Yy Zz li Jj Kk G53.1 M98 Pp G69 : (Subprogram) G01 Xx Ff : M99 Carry out inclined surface machining and tool axis direction control in the main program and create a machining shape in the subprogram. No need to consider the rotary axis' angle before calling the subprogram.</p>

Combination with tool center point (TCP) control

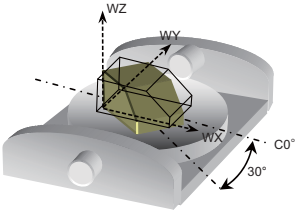
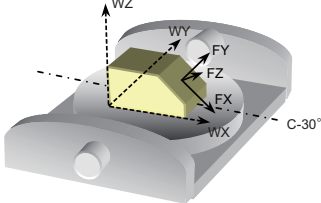
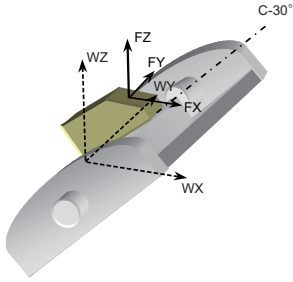
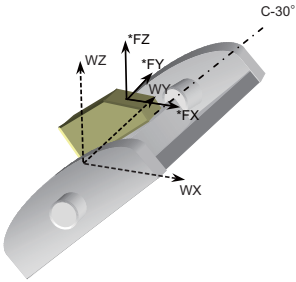
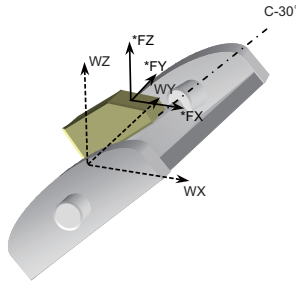
When inclined surface machining control is commanded together with the TCP control (G43.4), define the table-interlocked feature coordinate system by setting the parameter (#7911 Rotary axis basic position selection) for TCP control. When zero degree position basis (#7911=0) is selected, it is possible to define the table-interlocked feature coordinate system at an arbitrary rotary axis angle. For the start position basis (#7911=1), the table-interlocked feature coordinate system can be defined on an inclined surface only when the TCP control is commanded at the same rotary axis angle as of the inclined surface machining command (G68.2) or tool axis direction control command (G53.1).

When workpiece is placed in the workpiece coordinate system direction:

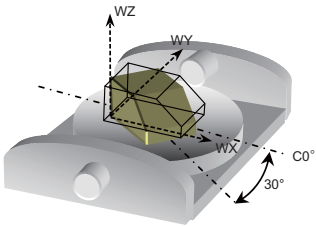
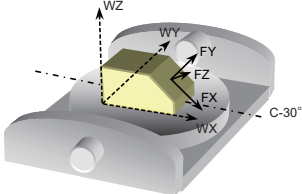
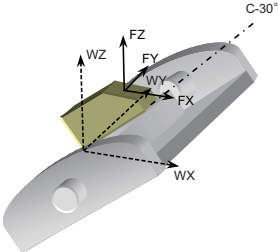
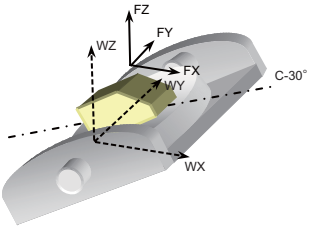
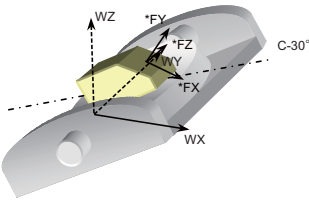
	Rotary axis basic position selection	
	Workpiece coordinate system zero for a basis (#7911 = 0)	The position when the tool center point is commanded for a basis (#7911 = 1)
<p>Inclined surface machining command Zero degree position basis (#7911=0) Workpiece coordinate offset A 0. C0.</p> 	<pre> : G90 G54 A0. C0. G68.2XxYyZz G53.1 G43.4 : </pre> 	
	<p>When G68.2 is commanded: Feature coordinate system is defined at a position in view from a workpiece coordinate system regardless of rotary axis position.</p> 	
	<p>When G53.1 is commanded: The tool axis direction matches the Z axis direction of the feature coordinate system, which has been defined with G68.2.</p> 	<p>When G43.4 is commanded: Feature coordinate system is fixed to the table at a position in view from a workpiece coordinate system regardless of rotary axis position.</p> 
	<p>When G43.4 is commanded: Feature coordinate system is fixed to the table at 0 degree of the coordinate system.</p> 	

When workpiece is placed deviated from workpiece coordinate system

(1) When TCP control (G43.4) is commanded at the same angle as of tool axis direction control (G53.1)

		Rotary axis basic position selection		
		Workpiece coordinate system zero point for a basis (#7911 = 0)	The position when the tool center point is commanded for a basis (#7911 = 1)	
<p>Inclined surface machining command Zero degree position basis (#7915 = 0) Workpiece coordinate offset A 0. C30.</p> 	<pre> : G90 G54 A0. C0. G68.2XxYyZz G53.1 G43.4 : </pre>			
		<p>When G68.2 is commanded: Feature coordinate system is defined at a position in view from a workpiece coordinate system regardless of rotary axis position.</p> 		
		<p>When G53.1 is commanded: The tool axis direction matches the Z axis direction of the feature coordinate system, which has been defined with G68.2.</p>		
		<p>When G43.4 is commanded: Feature coordinate system is fixed to the table at a position in view from a workpiece coordinate system regardless of rotary axis angle.</p>		<p>When G43.4 is commanded: Feature coordinate system is fixed to the table at the position of the present feature coordinate system.</p>

(2) When TCP control (G43.4) is commanded at a different angle from tool axis direction control (G53.1).

		Rotary axis basic position selection	
		Workpiece coordinate system zero for a basis (#7911 = 0)	The position when the tool center point is commanded for a basis (#7911 = 1)
<p>Inclined surface machining command Zero degree position basis (#7915 = 0) Workpiece coordinate offset A 0. C30. Rotary axis is rotated before tool center point control command</p> 	<pre> : G90 G54 A0. C0. G68.2XxYyZz G53.1 G00 A30. G43.4 : </pre>		
	<p>When G68.2 is commanded: Feature coordinate system is defined at a position in view from a workpiece coordinate system regardless of rotary axis position.</p>		
	<p>When G53.1 is commanded: The tool axis direction matches the Z axis direction of the feature coordinate system, which has been defined with G68.2.</p>		
	<p>When G00 A30 is commanded: The tool axis direction does not match the Z axis direction of the feature coordinate system, which has been defined with G68.2 command.</p>		
	<p>When G43.4 is commanded: Feature coordinate system is fixed to the table at a degree determined based on the rotary axis position.</p>		<p>When G43.4 is commanded: Feature coordinate system is fixed to the table at the position of the present feature coordinate system.</p>

Start position based inclined surface machining command (#7915=1)

If the Z axis of the feature coordinate system defined on an inclined surface matches the tool axis direction, the table-interlocked feature coordinate system can be defined by commanding the tool center point (TCP) control.

However, if the TCP control start position basis is selected (#7911=1), the table-interlocked feature coordinate system can be defined on an inclined surface only when TCP control is commanded at the same angle as of the inclined surface machining command (G68.2) or tool axis direction control command (G53.1).

19.4.11 Relationship between Inclined Surface Machining and Other Functions



Relationship with other functions

Commands Available in Inclined Surface Machining Mode

If commanded in any other mode, a program error (P951) occurs.

Command	Function
G00, G01	Positioning, Linear interpolation
G02, G03	Circular interpolation, Helical interpolation
G02.1, G03.1	Spiral interpolation
G04	Dwell
G05 P0, P1, P2, P10000	High-speed machining mode, high-speed high-accuracy control II
G05.1 Q0, Q1	High-speed high-accuracy control I
G08 P1	High-accuracy control
G09	Exact stop check
G10, G11	Parameter input by program / cancel, Compensation data input by program
G12, G13	Circular cut
G17, G18, G19	Plane selection
G22/G23	Stroke check before travel ON / cancel
G28	Automatic 1st reference position return
G29	Start position return
G30	2nd to 4th reference position return
G30.1 to G30.6	Tool exchange position return
G31	Skip (*1)
G31.1 to G31.3	Multi-step skip (*1)
G34, G35, G36, G37.1	Special fixed cycle
G40, G41, G42	Tool radius compensation cancel/left/right
G43, G44, G49	Tool length compensation plus/minus/cancel
G43.1	Tool length compensation along the tool axis
G43.4, G43.5	Tool center point control types I/II
G45, G46, G47, G48	Tool position offset
G50, G51	Scaling cancel/ON
G50.1, G51.1	G command mirror image cancel/ON
G53	Machine coordinate system selection
G53.1	Tool axis direction control
G61	Exact stop check mode
G61.1	High-accuracy control
G62	Automatic corner override
G64	Cutting mode
G65	User macro simple call
G66, G66.1, G67	User macro modal call A/B/cancel
G69	Coordinate rotation cancel, Inclined surface machining cancel
G70 to G76, G80 to G89	Fixed cycle for drilling (Including synchronous tapping)
G90, G91	Absolute value command, Incremental value command
G93	Inverse time feed
G94	Feed per minute
G95	Feed per revolution
G98, G99	Fixed cycle initial level return, R point level return
G127	Reverse run prohibition in all part systems

Command	Function
G162, G163	Spindle speed fluctuation detection
G186	Interference check III
M98, M99	Subprogram call, main program return
F	Feedrate command
M,S,T,B	M, S, T, B command
Macro command	Local variable, Common variable, Arithmetic commands (such as four basic arithmetic rule, trigonometric functions, square root), Control commands (IF-GOTO- and WHILE-DO-)

(*1) Only the three orthogonal axes designated by the rotary axis configuration parameter can be commanded. If two rotary axes are commanded, a program error (P951) will occur

Modes where inclined surface machining (including cancel command) is available

If inclined surface machining (G68.2 or G68.3) is commanded in a mode other than those listed below, a program error (P952) occurs.

Mode	Function
G00, G01	Positioning, Linear interpolation
G05 P0, P1, P2	High-speed machining mode
G05.1 Q0, Q1	High-speed high-accuracy control I
G08 P1	High-accuracy control
G10.9	Diameter/Radius designation switch
G13.1	Polar coordinate interpolation cancel
G15	Polar coordinate command cancel
G17, G18, G19	Plane selection
G20, G21	Inch command, Metric command
G22/G23	Stroke check before travel ON/cancel
G40	Tool radius compensation cancel
G40.1	Normal line control cancel
G43, G44 G49	Tool length compensation Tool length compensation cancel
G50	Scaling cancel
G50.1	Mirror image by G code OFF
G54 to G59, G54.1	Workpiece coordinate system selection, Extended workpiece coordinate system selection
G54.4 P	Workpiece installation error compensation
G61	Exact stop check mode
G61.1	High-accuracy control
G64	Cutting mode
G67	User macro modal call OFF
G69	Coordinate rotation cancel, 3-dimensional coordinate conversion cancel
G80	Fixed cycle cancel
G90, G91	Absolute value command, Incremental value command
G93	Inverse time feed
G94	Feed per minute
G95	Feed per revolution
G97	Constant surface speed control OFF
G98, G99	Fixed cycle initial level return, R point level return
G114.1	Spindle synchronization I (*1)
G113.1	Spindle synchronization cancel (*1)
G162, G163	Spindle speed fluctuation detection

(*1) The inclined surface machining can be commanded using the C axis in spindle synchronization.

Modes where tool axis direction control (G53.1/G53.6) is available

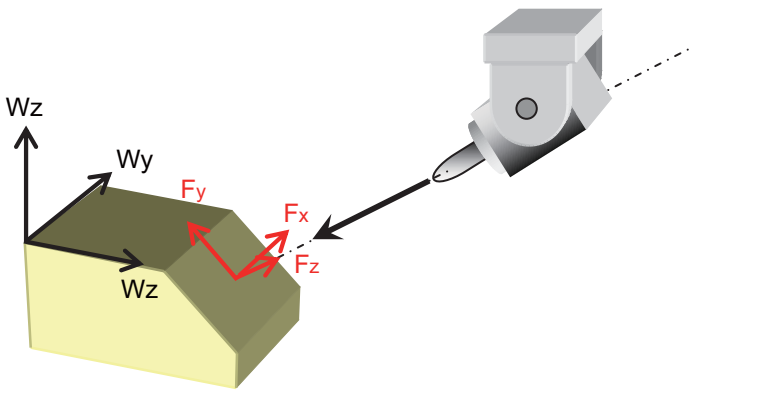
If tool axis direction control is commanded in a mode other than those listed below, a program error (P953) occurs.

Mode	Function
G00, G01	Positioning, Linear interpolation
G05 P0, P1, P2	High-speed machining mode
G05.1 Q0, Q1	High-speed high-accuracy control I
G08	High-accuracy control
G10.9	Diameter/Radius designation switch
G13.1	Polar coordinate interpolation cancel
G15	Polar coordinate command cancel
G17, G18, G19	Plane selection
G20, G21	Inch command, Metric command
G23	Stroke check before travel OFF
G40	Tool radius compensation cancel
G40.1	Normal line control cancel
G43, G44	Tool length compensation
G49	Tool length compensation cancel
G50	Scaling cancel
G50.1	Mirror image by G code OFF
G54 to G59, G54.1	Workpiece coordinate system selection, Extended workpiece coordinate system selection
G54.4 P	Workpiece installation error compensation
G61	Exact stop check mode
G61.1/G08P1	High-accuracy control
G64	Cutting mode
G67	User macro modal call OFF
G68.2 to G68.9	Inclined surface machining
G80	Fixed cycle cancel
G90, G91	Absolute value command, Incremental value command
G93	Inverse time feed
G94	Feed per minute
G95	Feed per revolution
G97	Constant surface speed control OFF
G98, G99	Fixed cycle initial level return, R point level return
G114.1	Spindle synchronization I (*1)
G113.1	Spindle synchronization cancel (*1)
G162, G163	Spindle speed fluctuation detection

(*1) The inclined surface machining can be commanded using the C axis in spindle synchronization.

Skip during inclined surface machining command

A skip operation during the inclined surface machining command is the same as the normal skip operation. The axis moves on the feature coordinate system.

Machining program		
N1 G68.2 Xx Yy Zz li Jj Zz ;		
N2 G53.1;		
N3 G90 G31 Z0. F100;		

The axis moves to the Z axis direction of the feature coordinate system in the N3 block.

For the skip function, refer to each chapter in "22 Measurement Support Functions".

Combination with arbitrary axis exchange

When performing inclined surface machining in combination with an arbitrary axis exchange (G140) command, you need to set the rotary axis configuration parameters using the 2nd axis name. Set the parameter "#1450 5axis_Spec/bit0" to "1" (setting by the 2nd axis name), and assign the axis configuration for executing inclined surface machining to the rotary axis configuration parameter (#7900 or later) using the 2nd axis name (example: A1, B2). If the inclined surface machining is commanded after arbitrary axis exchange has been completed while the parameter "#1450 5axis_Spec/bit0" is not designated, a program error (P952) will occur.

You can set the configurations up to the number of valid part systems (up to four part systems) in the rotary axis configuration parameter. With multiple configurations set, you can perform inclined surface machining in different axis configurations.

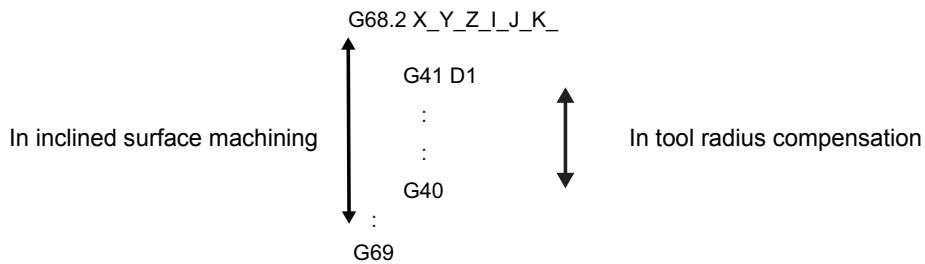
Inclined surface machining can be performed using the axis configuration in the part system with axis exchange completed by applying the rotary axis configuration parameter in the configuration in which all axes included in the part system are set.

19.4.12 Precautions for Inclined Surface Machining

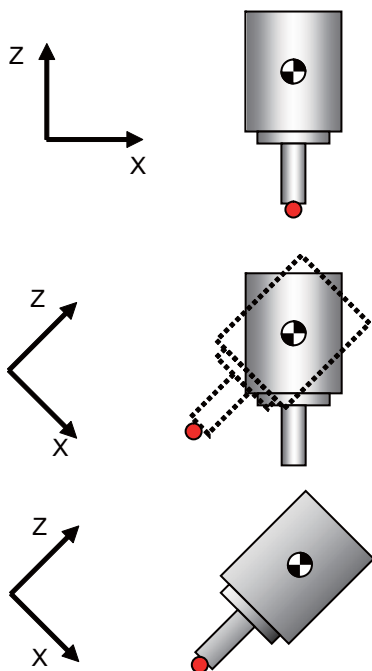


Precautions

- (1) A rotary axis moves at G53.1 command. Thus, move the tool far enough away from the table before commanding G53.1.
- (2) When inclined surface machining is commanded, the coordinates of the feature coordinate system are set in the system variables #5001 to #5100+n (excluding #5021 to #5021+n), which are used to read the position information. But, the coordinates of the machine coordinate system are set in the variables #5021 to #5021+n (machine coordinate values) even when inclined surface machining is commanded.
n: Varies depending on the number of control axes.
- (3) When Reset signal is input during inclined surface machining command, the inclined surface machining mode will be canceled, and the modal G code will be G69. (If the parameter "#1151 rsint" or "#1210 RstGmd/bitF" is set to "0", however, the inclined surface machining mode is kept even when Reset is input.)
- (4) When the external deceleration signal is input, the signal is not input to the axes of the feature coordinate system, but to the axes of the machine coordinate system that is actually operating.
- (5) If G28 or G30 is commanded after the inclined surface machining command has been issued, the control is carried out with respect to the inclined surface coordinate system up to the intermediate point, and then carried out with respect to the machine coordinate system from the intermediate point.
- (6) Tool radius compensation, mirror image by G code, fixed cycle command, tool center point control, scaling, and tool length compensation along the tool axis should be nested in the inclined surface machining command. Thus, these commands need to be commanded between the inclined surface machining command (G68.2, etc.) and G69.



- (7) If inclined surface machining (G68.2) is commanded while tool length compensation is active, the actual tool tip position does not match the current position. In such case, command G53.1 to align the tool axis direction with the Z axis of the feature coordinate system, which will make the tool tip position the same as the current position.



Before commanding G68.2, the current position and actual tool tip are the same.

When feature coordinate system is defined in G68.2, a point obtained by compensating the tool length direction to be in the feature coordinate axis' Z direction is deemed as the current position. Thus, the current position doesn't match with the actual tool tip position.

The actual tool tip point becomes the same as the current position by commanding G53.1 to align the tool axis direction with the feature coordinate system's Z direction.

- (8) Coordinate conversion is not carried out upon manual interruption, so travel, by a distance equivalent to the manual interruption amount, is carried out with respect to the machine coordinate system. When manual interruption with ABS ON, or tool center point control has been performed during inclined surface machining, return to the position before interruption, and then restart automatic operation. Do not perform the manual interruption for the rotary axis while the inclined surface machining command is active.
- (9) MDI interruption, PLC interruption and macro interruption are disabled during inclined surface machining. If MDI interruption or PLC interruption is attempted during inclined surface machining, an operation error (M01 0185) will occur. If macro interruption is enabled during inclined surface machining, a program error (P951) occurs. Also when inclined surface machining is commanded while macro interruption is active, a program error (P952) occurs.
- (10) When inclined surface machining is commanded during MDI interruption, PLC interruption, or macro interruption, a program error (P952) will occur.
- (11) When a circular command is graphically traced under the inclined surface machining command, circular tracing is performed if the feature coordinate system matches the machine coordinate system. If the systems are unmatched, a linear tracing is performed instead.
- (12) Tracing is carried out using the machine coordinate values.
- (13) When this function is used together with tool center point control or the workpiece installation error compensation function, inclined surface machining is subject to the restraints of each function. For details, refer to each chapter.
- (14) Program restart from the block after the inclined surface machining command is issued cannot be implemented. If commanded, a program error (P49) occurs.

Program example

```
N10 G00 X_Y_Z_;
N11 G00 X_Y_Z_B_C;
```

Restart from the block N10 or N11 is possible.

:

```
N20 G68.2 X_Y_Z_I_J_K;
N21 G01 X_Y_Z_F_;
N22 G01 X_Y_Z_F_;
N23 G69

N30 G90 G00 X_Y_;
N31 G90 G00 Z_;
```

Restart from the block N20 or later is not possible.

Attempting to do so will cause an alarm.

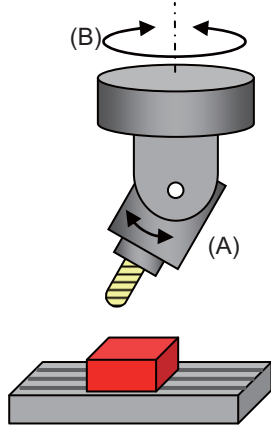
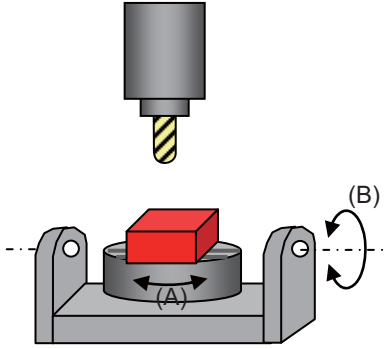
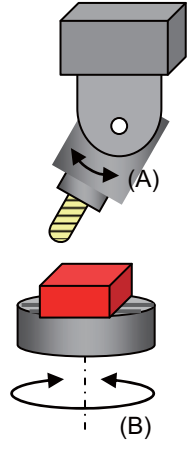
- (15) If you want to display the coordinates on the position screen during inclined surface machining, enter "23" in the parameters #8901 to #8906. Then, the corresponding counter is shown with respect to the feature coordinate system.
In the inclined surface coordinate counter display, whether the machining position on the program command that does not include the tool length compensation/tool radius compensation can be selected depends on the MTB specifications (parameter "#1287 ext23/bit1, bit2 (inclined surface coordinate display)").
When tool tip coordinate display is enabled, inclined surface coordinates' counter can be displayed on the window by selecting the inclined surface for the counter selections 1, 2 and 3.
- (16) The movement that occurs in response to the G00 command is always the interpolation type. (Non-interpolation type is not available.)
- (17) In the case of table rotation type machines, the tool axis direction is not changed in G68.3. Thus, a feature coordinate system is defined with respect to the Z axis of the coordinate system before the inclined surface machining command is issued. But, designation of feature coordinate system's zero point, and the rotation R about Z axis are enabled.
- (18) When inclined surface machining is commanded during inclined surface machining, a program error (P951) will occur.
- (19) In the parameters #7900 to #7902, #7922, #7932, #7942, and #7952, designate the axes of the first part system. If you command inclined surface machining in a part system where any of the designated axes is not ready, a program error (P932) will occur.
- (20) The feature coordinate system is defined with respect to the coordinate system (workpiece coordinate system), which is independent of the table rotary axis' rotation angle, so it is dependent on the table rotary axis' angle before the inclined surface machining command is issued.
- (21) A linear axis command during inclined surface machining is carried out using the coordinates of the feature coordinate system. And a rotary axis command is done using the coordinates (machine values) of the workpiece coordinate system.

- (22) If the address R, I, J or K has exceeded the setting range, a program error (P35) will occur.
- (23) Buffer correction cannot be implemented during inclined surface machining command.
- (24) If the operation mode is switched to "reference position return" during inclined surface machining command, an operation error (M01 0185) will occur.
- (25) If a linear angle command, geometric command, or figure rotation command is issued during inclined surface machining command, a program error (P951) occurs.
- (26) If arbitrary axis exchange (G140) is issued during inclined surface machining modal, the program error (P951) occurs.
- (27) When the axis during inclined surface machining control is the target for axis exchange, the operation error (M01 1101) will occur. The alarm will be canceled by reset.
- (28) When the inclined surface modal is retained at resetting, the arbitrary axis exchange status is also retained even if the parameter is set so as to cancel the arbitrary axis exchange by reset.
If an axis in a part system in which inclined surface machining is being carried out is specified as the axis to be exchanged in the part system, the axis exchange will not be possible and an operation error (M01 1101) occurs regardless of whether the automatic operation mode has been established.
- (29) When the number of simultaneous contouring control axes is four or less and the indexing is performed with the indexing type 2 of R-navi on the selected machining surface, the block stop due to the completion of each axis travel will not be performed regardless of whether the parameter "#8132 G53.6 block stop" has been set.
- (30) If axis movement is commanded for any axis not specified in the rotary axis configuration parameters during inclined surface machining when SSS is set to ON, a program error (P951) occurs.
- (31) If the inclined surface machining is commanded in spindle mode when the axis set for the spindle/C axis is used as a rotary axis of the inclined surface machining, a program error (P934) occurs.
When the mode is changed to the spindle mode during the inclined surface machining, an operation error (M01 0186) occurs. This error can be reset by NC reset.
When the inclined surface machining modal is retained by resetting the modal retention (#1151 rstint=0), an operation error (M01 0186) occurs at the timing of the next automatic operation start.
To reset the inclined surface machining modal, issue the inclined surface machining cancel command (G69) to the head (first block) of the machining program.
- (32) This function is not compatible with the automatic handle interruption that is in use of R-NAVI.
- (33) The rapid traverse block overlap is temporarily canceled while the inclined surface machining command is active. When the rapid traverse block overlap (G00.5) is commanded while the inclined surface machining command is active, it becomes valid after the inclined surface machining command is canceled.
- (34) Do not combine the inclined surface machining command with the following functions.
- ♦Sub part system control I (G122/G145)
 - ♦Automatic error detection (AUTED:YCFB)
 - ♦NC axis/PLC axis switch (NPCHGREQn:YBC0)
 - ♦Torque limitation skip (G160)
- (35) While the inclined surface machining command is active, any axis not specified in the rotary axis configuration parameter can be commanded only in the G00 modal state. Do not designate the axis not specified in the rotary axis configuration parameter when the G code modal of group 01 is other than G00.
- (36) The acceleration is the same as that at rapid traverse (G00) (*1) in the inclined surface machining command mode when high-accuracy control is enabled.
(*1) Acceleration determined by "#1206 G1bF" (maximum speed) and "#1207 G1btL" (time constant).

(37) The axis configuration of applicable machines is as follows:

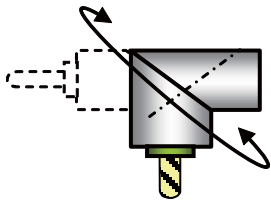
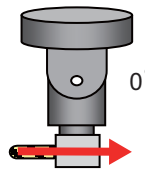
The function is effective for the machine configuration with the right-hand orthogonal coordinate system defined in ISO standard.

(a) This function applies to three types of machine configurations as below:

Type	Tool tilt type	Table tilt type	Combined type
Description	Two rotary axes on tool head side	Two rotary axes on table side	One rotary axis each on tool head side and table side
Example of machines			
Primary rotary axis (A)	Tool-side 2nd rotary axis	Table-side 2nd rotary axis	Tool-side rotary axis
Secondary rotary axis (B)	Tool-side 1st rotary axis	Table-side 1st rotary axis	Table-side rotary axis

In this manual, the following axes are called as primary rotary axis: the tool-side 2nd rotary axis (tool tilt type), the table-side 1st rotary axis (table tilt type), and the tool side rotary axis (combined type).

(b) This function is not applicable to machines as below:

Description	Example of machines
A machine whose rotary axis's rotation center axis is not parallel to any orthogonal coordinate axis.	
A machine whose direction from the tool tip to the tool base is not parallel to Z axis (Z axis positive direction) when machine positions of the rotary axes are all 0°.	
A machine in which three linear axes do not form a right-handed orthogonal coordinate system.	Tool axis direction

When the inclined surface machining is commanded with the following conditions satisfied, the program error (P934) occurs.

- The parameter "#1450 5axis_Spec/bit3" (Select specifications of rotation direction parameter) is set to "1".
- Any of the parameters "#7923 DIR_T1", "#7933 DIR_T2", "#7943 DIR_W1" or "#7953 DIR_W2" is set to "1" (left-hand screw direction).

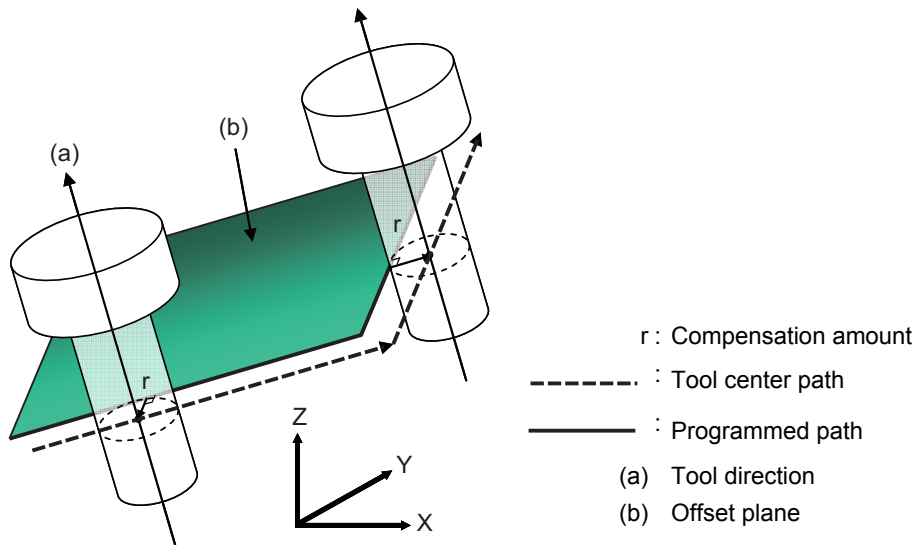
(38) If any of the orthogonal axes of all the active part systems is under machine lock during inclined surface machining, normal synchronous tapping is applied even though the high-speed tapping specification is provided.

19.5 3-dimensional Tool Radius Compensation (Tool's Vertical-direction Compensation) ; G40/G41.2,G42.2



Function and purpose

This function realizes the tool radius compensation for the machine with 2 rotary axes by calculating the change in the direction of a workpiece and the inclination of the tool caused by the move of the rotary axes. Tool radius compensation is carried out three-dimensionally by calculating the tool path on the surface of a workpiece from the program command and obtaining the compensation vector on a plane (compensation plane) perpendicular to the tool direction (offset plane).



If the specification is not provided, when 3-dimensional tool radius compensation (tool's vertical-direction compensation) is commanded, a program error (P161) will occur.



Command format

3-dimensional tool radius compensation (Tool's vertical-direction compensation) left

G41.2 (X_ Y_ Z_ A_ B_ C_) D_;

3-dimensional tool radius compensation (Tool's vertical-direction compensation) right

G42.2 (X_ Y_ Z_ A_ B_ C_) D_;

3-dimensional tool radius compensation (Tool's vertical-direction compensation) cancel

G40 (X_ Y_ Z_ A_ B_ C_); or D0;

X, Y, Z	Orthogonal coordinate axis movement command (can be omitted)
A, B, C	Rotary axis movement command (can be omitted)
D	Compensation No. "D0;" refers to a D command of compensation number "0".

Note

(1) All the G codes in the above command format belong to the modal group 7.



Detailed description

This function calculates the change in the direction of a workpiece and the inclination of the tool caused by the move of the rotary axes, and converts the programmed tool path onto the offset plane (perpendicular to the tool direction at the compensation point) to conduct the tool radius compensation for 5-axis machining. (Refer to "How to calculate the compensation vector" for the details of offset plane.)

The operations at the start/cancel and in compensation mode on the offset plane conform to the normal tool radius compensation. Refer to Chapter "12.3 Tool Radius Compensation; G38,G39/G40/G41,G42" for the operations which are not explained in this section.

Tool radius compensation start (startup)

The type of compensation start can be selected from type A and type B by the parameter "#8157 Radius comp type B", like the conventional tool radius compensation. Refer to "12.3 Tool Radius Compensation; G38,G39/G40/G41,G42" for the descriptions of type A/type B.

The startup must be carried out in the G code modal listed in "Modes in which G41.2/G42.2 command is issued" in "Relation with other functions". If commanded in an unlisted modal, a program error (P163) will occur.

Tool radius compensation operation

For usable functions during the compensation, refer to "Commands which can be issued while G41.2/G42.2 is executed" in "Relation with other functions". If an unavailable function is commanded, a program error (P162) will occur. Interference check is not available for this function.

Tool radius compensation cancel

When any of the following condition is met, the tool radius compensation for 5-axis machining will be canceled.

- (1) After the compensation cancel command (G40) is executed
- (2) A command of offset number D00 is issued
- (3) NC reset 1 (*1)
- (4) NC reset 2 or Reset & Rewind is commanded

The type of compensation cancel can be selected from type A and type B by the parameter "#8157 Radius comp type B", as well as the startup.

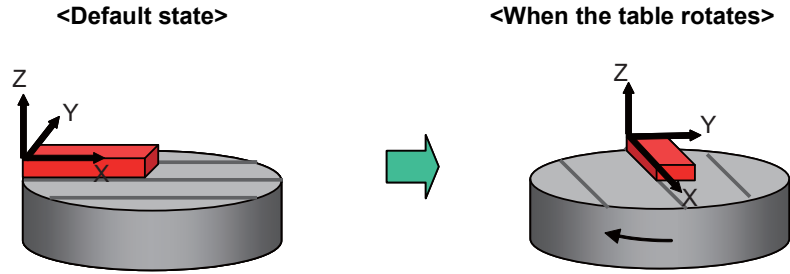
(*1) The compensation is canceled when "#1151 rstint" is ON.

How to calculate the compensation vector

The compensation vectors for tool radius compensation are obtained as shown below.

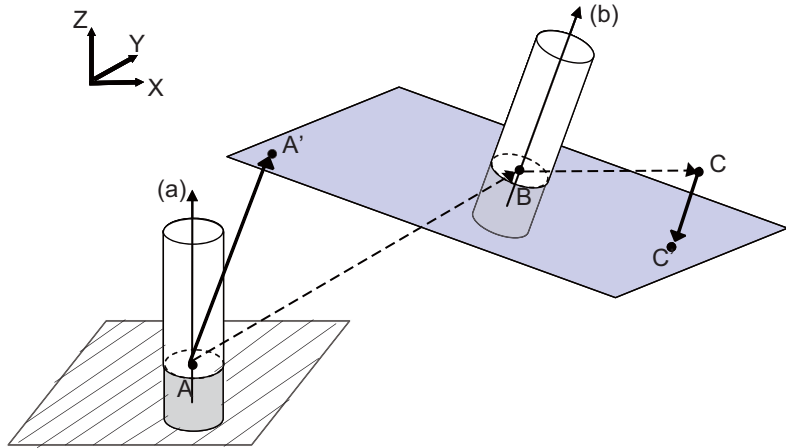
(1) Table coordinate system

Convert the program command into the path on the surface of the table coordinate system. Table coordinate system rotates with the workpiece (the figure below) as the table rotates. The command path on this coordinate system is the relative command path of the tool against the workpiece.






(2) Conversion into the points on the offset plane

Reflect the obtained path on the table coordinate system onto the offset plane (vertical to the tool axis direction at the compensation point) and calculate the points (A' and C' in the figure below) on the offset plane.

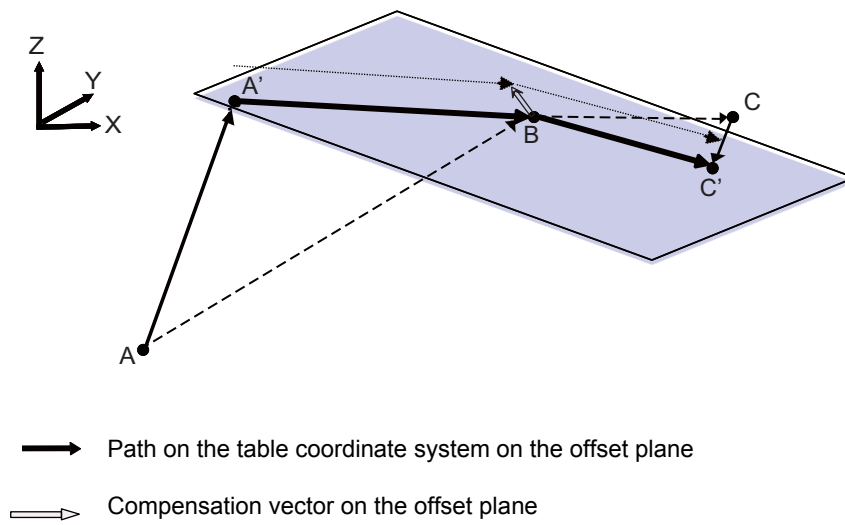


(a) Tool direction at point A
 (b) Tool direction at point B

-  Offset plane at point A
-  Offset plane at point B
-  Path on the table coordinate system

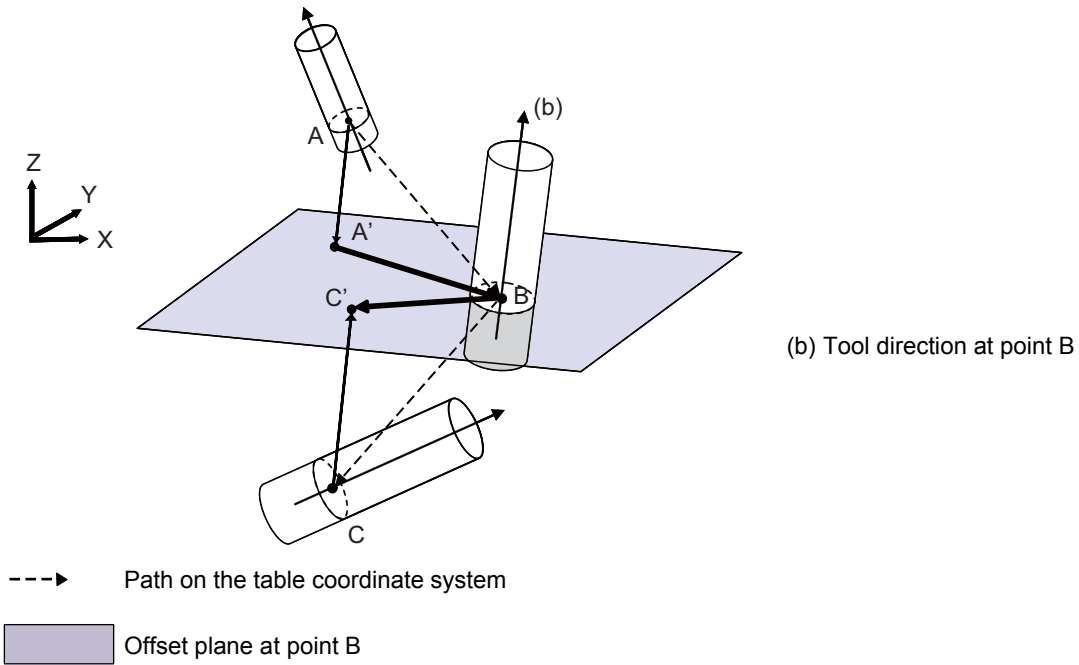
(3) Compensation on the offset plane

Perform the conventional tool radius compensation on the offset plane and calculate the compensation vector on the offset plane.

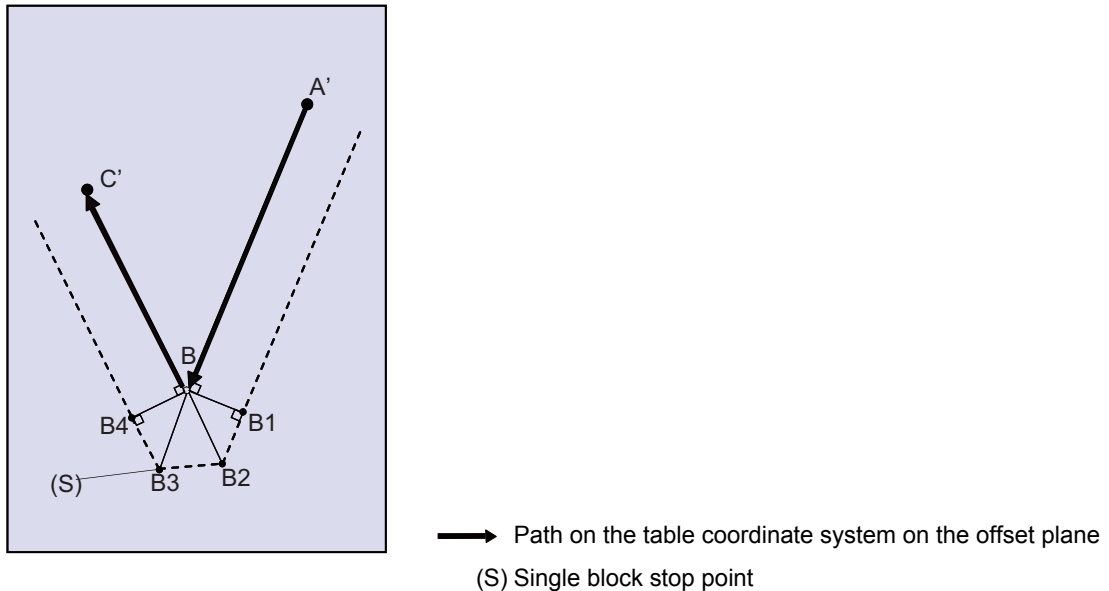


When a block is inserted

When a block is inserted while cutting a corner, the direction of the tool at the single block stop is equal to that of the previous block. (Like a feedrate and other modal data, the rotation angle of the previous block is kept.)



If the program moves from A, B to C as shown in the figure above, the offset plane at point B is as the figure below. The block between the points B2 and B3 is inserted, the tool direction between B2-B3 is the same as that at point B2 and the tool moves on the offset plane which is created at point B.





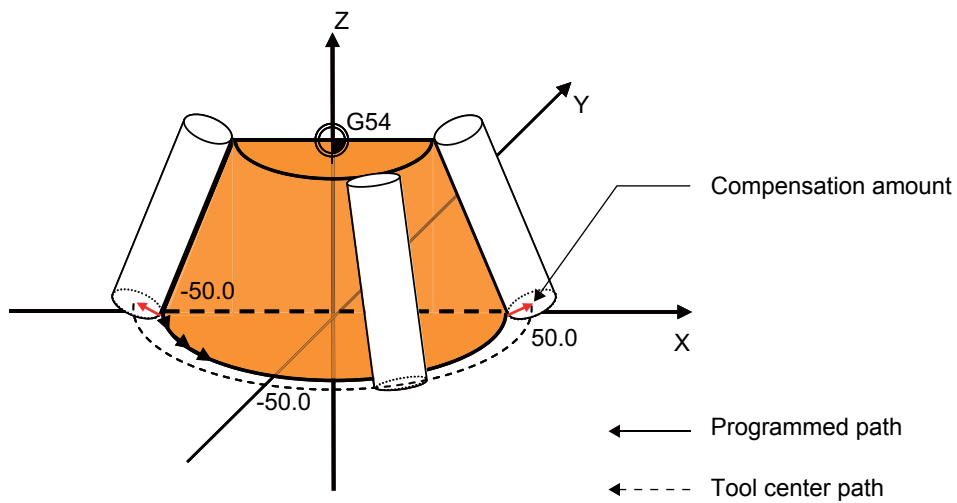
Program example

```

N1    G28 Z
N2    G28 BC
N3    G28 XY
N4    G90 G54 G00 X-60. Y0.
N5    G00 B30.
N6    G43.4 H1 Z-50.
N7    G42.2 G01 X-50. D1
N8        G01 X-49.990 Y-1.000 C 1.15
N9        G01 X-49.960 Y-1.999 C 2.29
N10       G01 X-49.910 Y-2.998 C 3.44
:
:
N200      G01 X50. Y0. C180.
N201      G01 Z0.
N202    G40
N203    G49
N204    G28 Z
N205    G28 BC
N206    G28 X
M30

```

(D1 =5.0, H1=50.0)





Relationship with other functions

Commands which can be issued in a same block as G41.2/G42.2

	Function
G00	Positioning
G01	Linear interpolation
G90	Absolute value command
G91	Incremental value command
F	Feedrate command
N	Sequence No.

If unlisted commands are issued in a same block with 3-dimensional tool radius compensation (tool's vertical-direction compensation) (G41.2/G42.2), a program error (P163) will occur.

Commands which can be issued while G41.2/G42.2 is executed

	Function
G00	Positioning
G01	Linear interpolation
G04	Dwell
G05 P0, G05 P1, G05 P2	High-speed Machining Mode
G05 P0, G05 P10000	High-speed high-accuracy control II
G08 P0, G08 P1	High-accuracy control
G09	Exact stop
G20, G21	Inch/metric command (*1)
G22, G23	Stroke check before travel ON/OFF
G40	Tool radius compensation cancel
G41.2, G42.2	3-dimensional tool radius compensation (tool's vertical-direction compensation) right/left
G61	Exact stop check mode
G61.1	High-accuracy control ON
G64	Cutting mode
G65	User macro simple call
G66	User macro modal call A
G66.1	User macro modal call B
G67	User macro modal call cancel
G90, G91	Absolute value command, Incremental value command
G93	Inverse time feed
G94	Feed per minute
G95	Feed per revolution
G96, G97	Constant surface speed control ON/OFF
M98, M99	Subprogram call, main program return
F	Feedrate command
M, S, T, B	M, S, T, B command
Macro command	Local variables and common variables Operation commands (four basic arithmetic rule, trigonometric functions, square root) Control commands (IF - GOTO -, WHILE - DO -)
N	Sequence No.

(*1) If the inch/metric command switches during 3-dimensional tool radius compensation (tool's vertical-direction compensation), a program error (P162) will occur.

Modes in which G41.2/G42.2 command can be issued

	Function
G00, G01	Positioning, Linear interpolation
G17, G18, G19	Plane selection
G20, G21	Inch/Metric command
G22, G23	Stroke check before travel ON/OFF
G40	Tool radius compensation cancel
G40.1, G150	Normal line control cancel
G41.2, G42.2	3-dimensional tool radius compensation (tool's vertical-direction compensation) left/right
G43, G44	Tool length compensation (+/-)
G43.1	Tool length compensation along the tool axis
G43.4, G43.5	Tool center point control type I/II
G49	Tool length compensation cancel
G50	Scaling cancel
G50.1	G command mirror image cancel
G54, G55, G56, G57, G58, G59, G54.1	Workpiece coordinate system selection, extended workpiece coordinate system selection
G54.4Pn	Workpiece installation error compensation
G61	Exact stop check mode
G61.1	High-accuracy control ON
G64	Cutting mode
G67	User macro modal call cancel
G68.2	Inclined surface machining
G68.3	Inclined surface machining command (Define using tool axis direction)
G69	3-dimensional coordinate conversion cancel
G80	Fixed cycle cancel
G90, G91	Absolute value command, Incremental value command
G93	Inverse time feed
G94	Feed per minute
G95	Feed per revolution
G96, G97	Constant surface speed control ON/OFF
G98, G99	Fixed cycle initial level return, R point level return
G15, G13.1, G113	Polar coordinate command cancel

Combination with arbitrary axis exchange control

When performing 3-dimensional tool radius compensation (tool's vertical-direction compensation) in combination with an arbitrary axis exchange (G140) command, you need to set the rotary axis configuration parameters using the 2nd axis name. Set the parameter "#1450 5axis_Spec/bit0" to "1" (setting by the 2nd axis name), and assign the axis configuration for executing 3-dimensional tool radius compensation (tool's vertical-direction compensation) to the rotary axis configuration parameter (#7900 or later) using the 2nd axis name (example: A1, B2). If the G41.2/G42.2 command is issued after arbitrary tool exchange has been completed while the parameter "#1450 5axis_Spec/bit0" is not designated, a program error (P163) will occur.

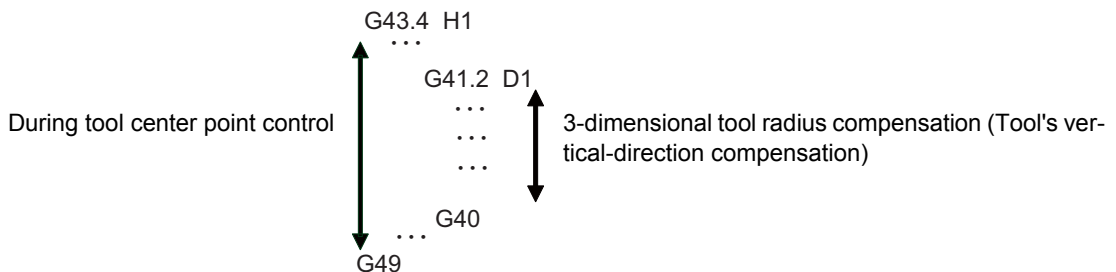
You can set the configurations up to the number of valid part systems (up to four part systems) in the rotary axis configuration parameter. With multiple configurations set, you can perform 3-dimensional tool radius compensation (tool's vertical-direction compensation) in different axis configurations.

3-dimensional tool radius compensation (tool's vertical-direction compensation) can be performed using the axis configuration in the part system with axis exchange completed by applying the rotary axis configuration parameter in the configuration in which all axes included in the part system are set.



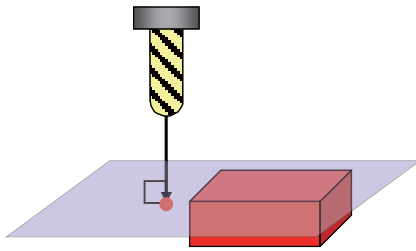
Precautions

- (1) Interference check is not available for 3-dimensional tool radius compensation (tool's vertical-direction compensation). The parameter "#8103 COLL. CHK OFF" to turn ON/OFF the interference check, available for conventional tool radius compensation, is invalid in 3-dimensional tool radius compensation (tool's vertical-direction compensation).
- (2) Tool radius compensation vector designation (G38) and tool radius compensation corner arc (G39) are not available. If commanded, a program error (P162) will occur.
- (3) Corner chamfering/corner rounding, linear angle command and geometric command are not available. If commanded, a program error (P162) will occur.
- (4) Manual interruption, automatic operation handle interruption, manual/automatic simultaneous, manual speed command, manual reference position return, tool handle feed & interruption, and manual arbitrary feed mode will cause an operation error (M01 0232) when the manual mode is ON.
- (5) Macro interruption cannot be used. If 3-dimensional tool radius compensation (tool's vertical-direction compensation) is commanded when macro interruption is valid, a program error (P163) will occur. If macro interruption valid (M96) is commanded in 3-dimensional tool radius compensation (tool's vertical-direction compensation), a program error (P162) will occur.
- (6) Tool escape and return is not available. Turning ON the tool escape and return transit point designation signal and the manual mode will cause an operation error (M01 0232).
- (7) Switching from a mode to MDI mode or from MDI mode to another mode in 3-dimensional tool radius compensation (tool's vertical-direction compensation) will cause an operation error (M01 0232).
- (8) Turning ON on the PLC interruption signal in 3-dimensional tool radius compensation (tool's vertical-direction compensation) will cause an operation error (M01 0232).
- (9) Mirror image by the external input is not available for the target axis (*1) of 5-axis machining. If mirror image by the external input is set to ON in the 3-dimensional tool radius compensation (tool's vertical-direction compensation), a program error (P162) will occur. Also, if the 3-dimensional tool radius compensation (tool's vertical-direction compensation) is commanded during mirror image by the external input, a program error (P163) will occur.
 (*1) Axes here are the axes designated with the parameters "#7900 RCDAX_I", "#7901 RCDAX_J", "#7902 RCDAX_K", "#7922 ROTAXT1", "#7932 ROTAXT2", "#7942 ROTAXW1", and "#7952 ROTAXW2". These settings depend on the MTB specifications.
- (10) If 3-dimensional tool radius compensation (tool's vertical-direction compensation) is commanded in the reverse run control mode, or if the reverse run control mode signal is set to ON in 3-dimensional tool radius compensation (tool's vertical-direction compensation), a program error (P163) will occur.
- (11) This function can be combined with the tool center point control (G43.4, G43.5/G49). However, the ON/OFF of the 3-dimensional tool radius compensation (tool's vertical-direction compensation) must be nested in the ON/OFF of the tool center point control and it must be commanded in the tool center point control. If the tool center point control is commanded in 3-dimensional tool radius compensation (tool's vertical-direction compensation), a program error (P162) will occur.
 This function can also be combined with tool length compensation along the tool axis (G43.1/G49) in the same conditions as the above.

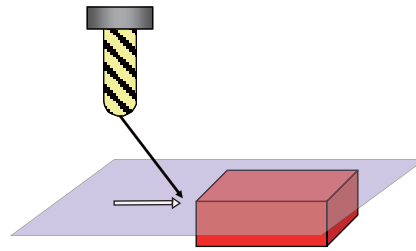


- (12) When used with the tool center point control, the compensation is applied to the tool center point path.
- (13) The restart search from the block in 3-dimensional tool radius compensation (tool's vertical-direction compensation) is possible while the restart search from the block concurrently using the tool center point control is impossible.
- (14) Fairing in high-speed machining mode/high-speed high-accuracy control is not available. The parameter "#8033 Fairing ON" to turn ON/OFF the fairing function in high-speed machining mode/high-speed high-accuracy control is invalid in 3-dimensional tool radius compensation (tool's vertical-direction compensation).


- (15) As shown in the figure below, we recommend that the tool approach to the surface of the workpiece at an angle. The tool radius compensation amount may not be correctly reflected on the cutting when the direction of the approach is opposite to the cutting direction. So the tool must shift to the surface of the workpiece at the start of the cutting at an angle to the tool axis direction.



- ×: Tool radius compensation amount may not be correctly reflected because there is no movement on the offset plane and the tool radius compensation amount is not re-calculated.



- : Tool radius compensation amount is correctly reflected.

 Offset plane (perpendicular to the tool axis direction)

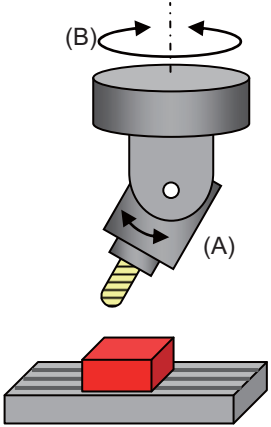
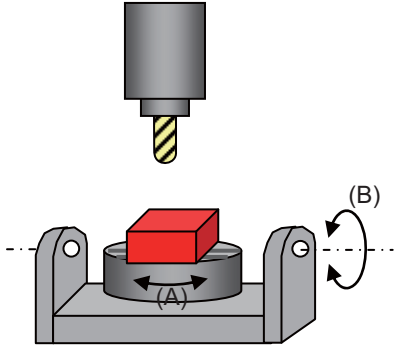
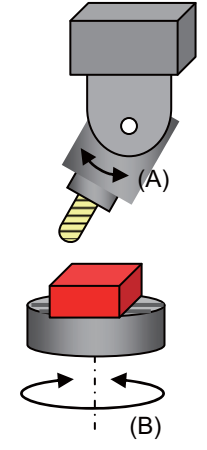
 Movement on the offset plane.

- (16) The buffer correction is not available in 3-dimensional tool radius compensation (tool's vertical-direction compensation). Pressing the menu [Prg correct] will display an error message.
- (17) Also, if 3-dimensional tool radius compensation (tool's vertical-direction compensation) is commanded during mirror image by parameter setting or coordinate rotation by parameter, a program error (P163) will occur. If the parameter is turned on in 3-dimensional tool radius compensation (tool's vertical-direction compensation), a program error (P162) will occur at the next start.

(18) The axis configuration of applicable machines is as follows:

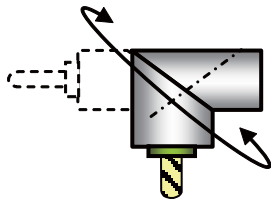
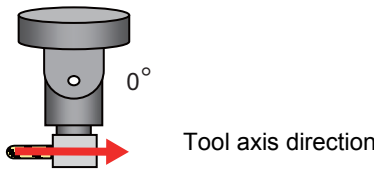
The function is effective for the machine configuration with the right-hand orthogonal coordinate system defined in ISO standard.

(a) This function applies to three types of machine configurations as below:

Type	Tool tilt type	Table tilt type	Combined type
Description	Two rotary axes on tool head side	Two rotary axes on table side	One rotary axis each on tool head side and table side
Example of machines			
Primary rotary axis (A)	Tool-side 2nd rotary axis	Table-side 2nd rotary axis	Tool-side rotary axis
Secondary rotary axis (B)	Tool-side 1st rotary axis	Table-side 1st rotary axis	Table-side rotary axis

In this manual, the following axes are called as primary rotary axis: the tool-side 2nd rotary axis (tool tilt type), the table-side 1st rotary axis (table tilt type), and the tool side rotary axis (combined type).

(b) This function is not applicable to machines as below:

Description	Example of machines
A machine whose rotary axis's rotation center axis is not parallel to any orthogonal coordinate axis.	
A machine whose direction from the tool tip to the tool base is not parallel to Z axis (Z axis positive direction) when machine positions of the rotary axes are all 0°.	
A machine in which three linear axes do not form a right-handed orthogonal coordinate system.	

<Note>

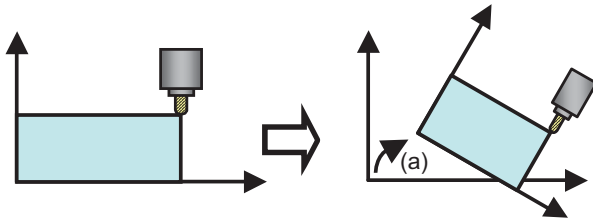
- This cannot be applied to a machine on which the rotary axis's rotation center axis is not parallel to any orthogonal coordinate axis.
- This cannot be applied to a machine of which the direction from the tool tip to the tool base is not parallel to Z axis (Z axis positive direction) when machine positions of the rotary axes are all 0°.

19.6 Workpiece Installation Error Compensation; G54.4



Function and purpose

This compensates the error when a workpiece is placed off the workpiece coordinate system to enable machining according to the program. While this function is enabled, a new coordinate system with the workpiece as its reference position will be defined (called "workpiece installation coordinate system") and the program will be executed in this new coordinate system.



(A) Error at the workpiece installation

According to the workpiece installation error, the coordinate system is automatically shifted.

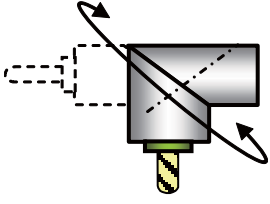
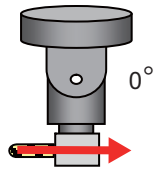
Axis configuration of applicable machines

The function is effective for the machine configuration with the right-hand orthogonal coordinate system defined in ISO standard. This function applies to three types of machine configurations as below:

Type	Tool tilt type	Table tilt type	Combined type
Description	Two rotary axes on tool head side	Two rotary axes on table side	One rotary axis each on tool head side and table side
Example of machines			
Primary rotary axis (A)	Tool-side 2nd rotary axis	Table-side 2nd rotary axis	Tool-side rotary axis
Secondary rotary axis (B)	Tool-side 1st rotary axis	Table-side 1st rotary axis	Table-side rotary axis

Note

(1) This function is not applicable to machines as below:

Description	Example of machines
A machine whose rotary axis's rotation center axis is not parallel to any orthogonal coordinate axis.	
A machine whose direction from the tool tip to the tool base is not parallel to Z axis (Z axis positive direction) when machine positions of the rotary axes are all 0°. (The arrow shown in the right figure indicates the direction of the tool axis.)	

(2) When the following conditions are satisfied, if the workpiece installation error compensation is commanded, the program error (P934) occurs.

- The parameter "#1450 5axis_Spec/bit3" (Select specifications of rotation direction parameter) is set to "1".
- Any of the parameters "#7923 DIR_T1", "#7933 DIR_T2", "#7943 DIR_W1" or "#7953 DIR_W2" is set to "1" (Rotation turning to the left-handed screw).



Command format

G54.4 Pn ;

P	Workpiece installation error compensation No. (n = 0 to 7) 0: Cancel workpiece installation error compensation 1 to 7: Compensate using the corresponding workpiece installation error compensation amount
---	--

Workpiece installation error compensation is modal command of group 27.

Note

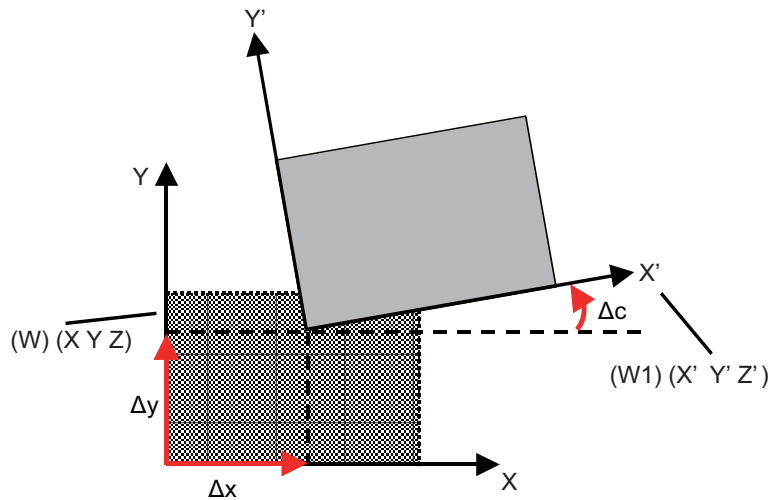
- (1) Make sure to command G54.4 in an independent block. If it is commanded together with another G code, travel command, or miscellaneous command in the same block, a program error (P546) occurs.
- (2) If address P is not commanded, the program error (P33) occurs. If any other value than 0 to 7 is command, a program error (P35) occurs.



Detailed description

Definition of workpiece installation coordinate system

A workpiece installation coordinate system is defined based on the following three workpiece installation error data.



Original workpiece installation position



Workpiece which is installed off the original position

(W) Workpiece coordinate system

(W1) Workpiece installation coordinate system

(1) Error in XYZ direction (Δx , Δy , Δz)

Set the zero point error amount of the workpiece installation coordinate system seen from the original workpiece coordinate system at (3).

(3): Machine coordinate values of table rotary axis when measuring the errors (Δx , Δy , Δz)

(2) Error in the rotation direction of each axis (Δa , Δb , Δc)

Rotate the workpiece coordinate system where all machine coordinate values of table rotary axis are zero in the following direction.

Rotate around X axis by Δa° -> rotate around Y axis of the original workpiece coordinate system by Δb° -> rotate around Z axis of the original workpiece coordinate system by Δc°

Set the Δa , Δb , and Δc directions of the coordinate system after the rotation to correspond with the X'Y'Z' directions of the workpiece installation coordinate system.

The positive direction of the rotational angle is the right-handed rotation.

(3) Table rotary axis position when measuring the errors

Set the machine coordinate values of the table rotary axis when Δx , Δy , Δz and Δa , Δb , Δc are measured.

- In the case of tool tilt type, this setting is not required.
- In the case of table tilt type, set both axes.
- In the case of combined type, set only the table side rotary axis.

Setting workpiece installation error

For workpiece installation errors, it is possible to set seven sets of error compensation amounts (workpiece installation error compensation amount No.01 to No.07) corresponding to compensation numbers commanded by address P.

The common workpiece installation error compensation amounts are added commonly to seven workpiece installation error compensation amount sets.

It is possible to set errors in XYZ directions (Δx , Δy , Δz) and the table side rotary axis position when these errors were measured, however, it is not possible to set an error in rotation direction of each axis (Δa , Δb , Δc).

Carry out these settings in the workpiece installation error setting screen, or using system variables. Refer to "23.24 System Variables (Workpiece Installation Error Compensation Amount)" for details on the system variables.

Addition of error compensation amounts

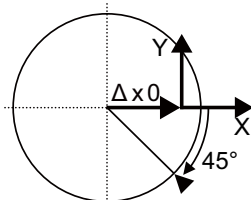
If the rotary axis positions are different between the common workpiece installation error compensation amount and the workpiece installation error compensation amount from No.01 to No.07, the total error compensation amounts are converted to those of a position of 0°, then they are added.

(Example) In the case of combined type machine with a table side rotary axis (C axis), if error compensation amounts are set in Common (A) and No.01 as right, the error amounts to be compensated are as below.

(A)	
Δx	10.0
Δy	0.0
Δz	0.0
Δa	
Δb	
Δc	
B	
C	45.0

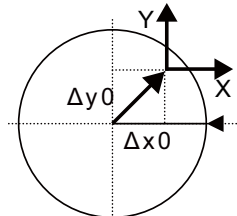
No.01	
Δx	0.0
Δy	5.0
Δz	0.0
Δa	0.0
Δb	0.0
Δc	45.0
B	
C	-90.0

Error at C=45° of common



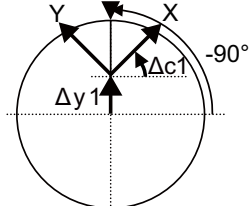
$\Delta x0$	10.0
$\Delta y0$	0.0
$\Delta z0$	0.0
$\Delta a0$	
$\Delta b0$	
$\Delta c0$	

Error at C=0° of common



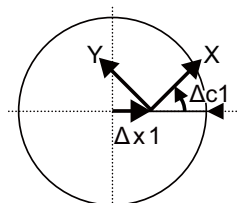
$\Delta x0$	7.071
$\Delta y0$	7.071
$\Delta z0$	0.0
$\Delta a0$	
$\Delta b0$	
$\Delta c0$	

Error at C=-90° of No.01

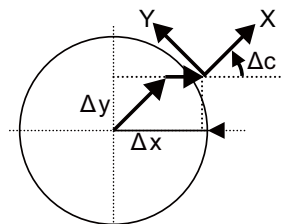


$\Delta x1$	0.0
$\Delta y1$	5.0
$\Delta z1$	0.0
$\Delta a1$	0.0
$\Delta b1$	0.0
$\Delta c1$	45.0

Error at C=0° of No.01



$\Delta x1$	5.0
$\Delta y1$	0.0
$\Delta z1$	0.0
$\Delta a1$	0.0
$\Delta b1$	0.0
$\Delta c1$	45.0



Δx	12.071
Δy	7.071
Δz	0.0
Δa	0.0
Δb	0.0
Δc	45.0

Error amounts to be actually compensated (C=0°)

- (a) Convert to the error at C = 0°
- (b) The errors of Common (A) and No.01 are added
- (c) Convert to the error at C = 0°

Note

(1) Even if C axis rotates, errors in the rotation directions (Δa , Δb , Δc) do not change.

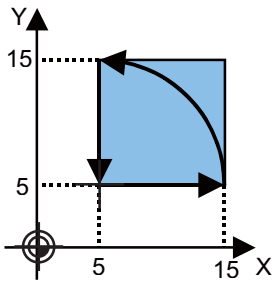
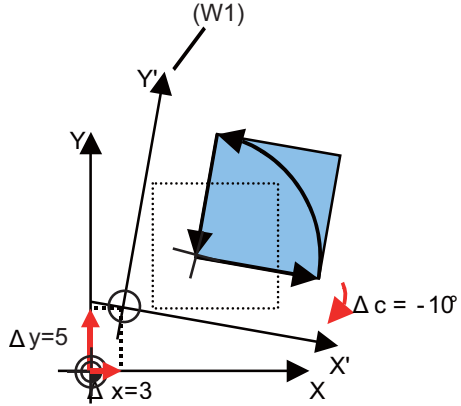
Details of operation

(1) Operation in the mode

- ◆ Workpiece installation error compensation start
 With a command G54.4 Pn (n = 1 to 7), the workpiece installation error compensation mode is started. The workpiece installation coordinate system is set according to the workpiece installation error compensation No. (No.01 to No.07) selected by n and the rotary axis position at the command, and the workpiece installation error counter changes to indicate the coordinate values of the workpiece installation coordinate system. (The machine does not make any movement.) Travel commands during the workpiece installation error compensation mode are dealt as those on the workpiece installation coordinate system.
- ◆ Workpiece installation error compensation cancel
 With a command G54.4 P0, the workpiece installation error compensation mode is canceled. The setting of the workpiece installation coordinate system is canceled, the original workpiece coordinate system is validated, and the workpiece installation error counter changes to indicate the coordinate values of the original workpiece coordinate system. (The machine does not make any movement.) The workpiece installation error compensation is canceled also at resetting.

When attempting to execute the program (A), the workpiece is placed 3mm off the original position in the X direction, 5mm off in the Y direction, and -10° off the Z axis angle. In this case, set the compensation amounts below as the workpiece installation error compensation amounts, and command a workpiece installation error compensation. The machining is carried out as intended by the original program.

<p>Program (A) G90 G0 X5. Y5. G1 X15. Y5. F500 G3 X5. Y15. R10. F500 G1 X5. Y5. F500</p>	<p>Workpiece installation error compensation No.01 Δx = 3. Δy = 5. Δz = 0. Δa = 0. Δb = 0. Δc = -10. Rotary axis position 1 = 0° Rotary axis position 2 = 0°</p>	<p>G54.4P1 G90 G0 X5. Y5. G1 X15.Y5. F500 G3 X5. Y15. R10. F500 G1 X5. Y5. F500 G54.4P0</p>
---	--	---

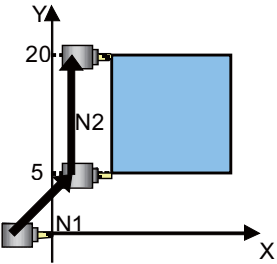
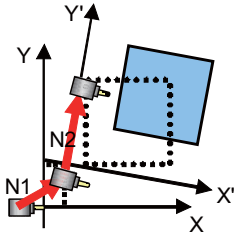
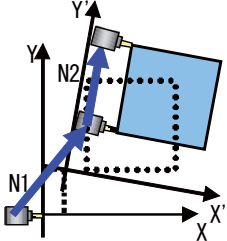

➔


(W1) Workpiece installation coordinate system

[Precautions in commanding and canceling G54.4]

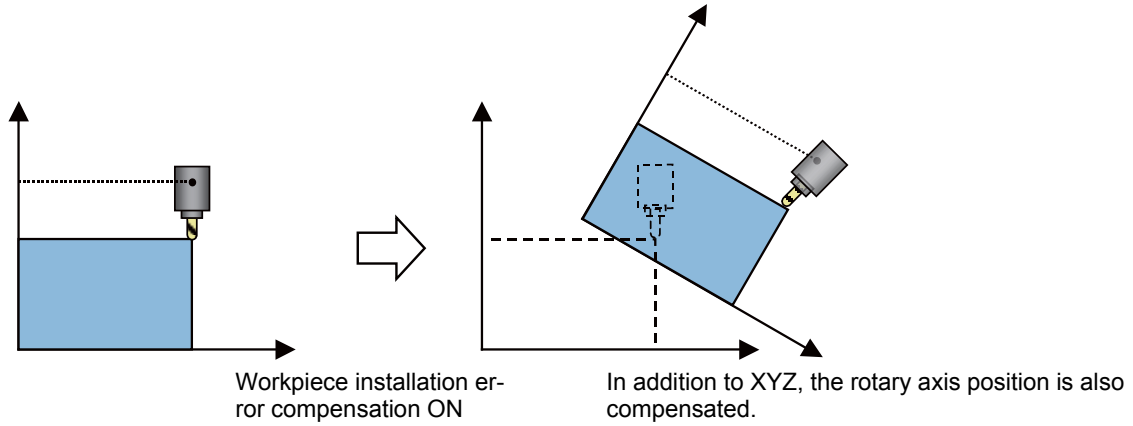
In the first travel command after commanding G54.4, make sure to issue an absolute value command. In the G54.4 command block, the machine does not make any movement. Therefore, if an incremental value command is issued immediately after having commanded G54.4, the operation may result in what is not intended by the program. (Once an absolute value command is issued, the operations of blocks thereafter will be as intended by the program even with incremental value commands.)

This precaution also applies to a cancel operation, so make sure to issue an absolute value command immediately after commanding G54.4 P0.

Operation with no installation error	If an incremental value command is issued immediately after having commanded G54.4
 <p>(Incremental value command) N1 G91 G0 X5. Y5. N2 G91 G1 Y15.</p>	 <p>If an incremental value is commanded in the next block of G54.4, an incremental travel is made from the current machine position. The operation does not result in what is intended by the program.</p> <p>G54.4 P1 N1 G91 G0 X5. Y5. N2 G91 G1 Y15. F500</p>
<p>(Absolute value command) N1 G90 G0 X5. Y5. N2 G90 G1 Y20.</p>	<p>If an absolute value command is issued immediately after having commanded G54.4</p>  <p>If an absolute value is commanded in the next block of G54.4, the operation results in what is intended by the program.</p> <p>G54.4 P1 N1 G90 G0 X5. Y5. N2 G90 G1 Y20. F500</p>

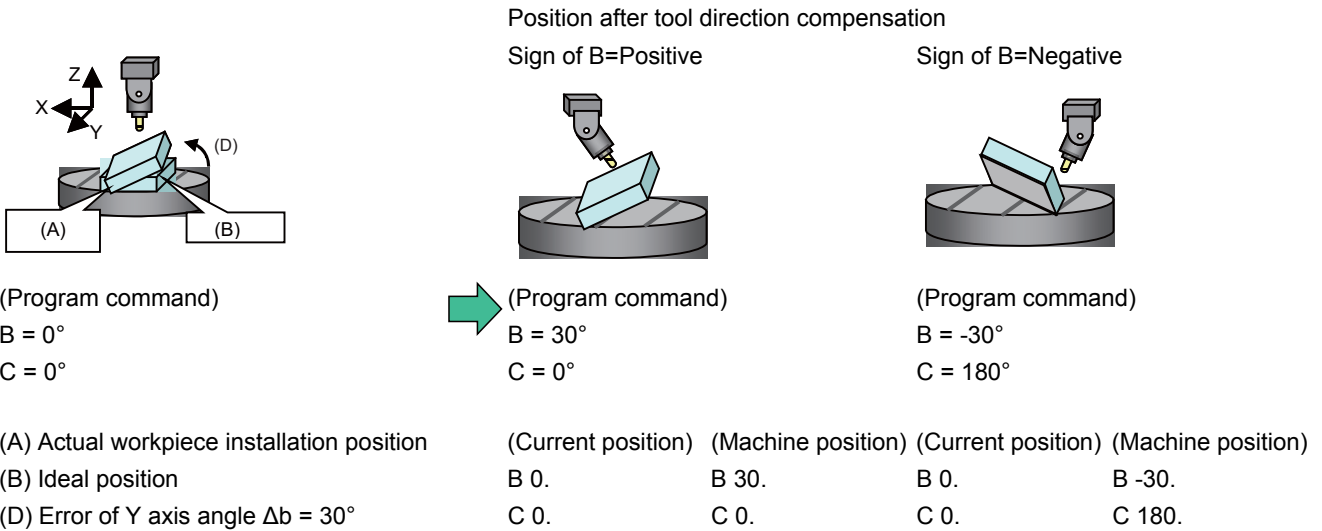
(2) Compensating the tool direction

With workpiece installation error compensation function, in addition to position compensations of XYZ, the tool angle against the workpiece is also compensated compared to the rotary axis position so that it is as intended by the program.



Generally there are two types of combinations of rotary axis angles after compensation.

(Example) Combined type (The tool side rotary axis is B, the table side rotary axis is C)



<Note>

♦The workpiece installation error counter indicates the program command position, and the machine position counter indicates the actual machine position.

Select one of these above according to the rules below.

- ♦In a G54.4 command block
Select the combination whose travel distance of the secondary rotary axis is shorter than in the other combination.
When selecting this, the machine does not move, and the workpiece installation error counter is updated.
- ♦Not in a G54.4 command block
Select the combination according to the parameter "#7906 PASSTYP".
(This parameter is the same parameter as "Singular point passage type" of tool center point control.)

Type	Type 1	Type 2
Parameters	#7906 = 0	#7906 = 1
Operation	Select the combination whose primary rotary axis sign is the same as of the block start point.	Select the combination whose travel distance of the secondary rotary axis is shorter.



Relationship with other functions

Commands available in workpiece installation error compensation mode

If a command other than those listed below is issued, a program error (P545) occurs.

Command	Function
G00, G01	Positioning, Linear interpolation
G02, G03	Circular interpolation, Helical interpolation
G04	Dwell
G05 P0, P1, P2, P10000	High-speed machining mode, high-speed high-accuracy control II
G05.1 Q0, Q1	High-speed high-accuracy control I
G08 P0 P1	High-accuracy control
G09	Exact stop check
G10, G11 (G11.1)	Parameter input by program / cancel, Compensation data input by program
G12, G13	Circular cut
G17, G18, G19	Plane selection
G20, G21	Inch/metric command
G28	Automatic 1st reference position return
G29	Start position return
G30	2nd to 4th reference position return
G30.1 to G30.6	Tool exchange position return
G31	Skip (*2)
G31.1 to G31.3	Multi-step skip (*3)
G34, G35, G36, G37.1	Special fixed cycle
G40, G41, G42	Tool radius compensation cancel/left/right
G41.2, G42.2	3-dimensional tool radius compensation (tool's vertical-direction compensation) right/left
G43, G44, G49	Tool length compensation plus/minus/cancel
G43.4, G43.5	Tool center point control type I/II
G45, G46, G47, G48	Tool position offset
G50, G51	Scaling cancel/ON
G50.1, G51.1	Mirror image by G code cancel/ON
G53	Machine coordinate system selection
G53.1	Tool axis direction control
G54 to G59, G54.1Pn	Workpiece coordinate system selection (*2), extended workpiece coordinate system selection
G54.4 P0	Workpiece installation error compensation cancel
G61	Exact stop check mode
G61.1	High-accuracy control I ON
G64	Cutting mode
G65	User macro simple call
G66, G66.1, G67	User macro modal call A/B/cancel
G68.2, G68.3 G69 (G69.1)	Inclined surface machining Coordinate rotation cancel, Inclined surface machining cancel
G73 to G76, G80 to G89	Fixed cycle for drilling Fixed cycle for drilling (Including synchronous tapping)
G90, G91	Absolute value command, Incremental value command
G93	Inverse time feed
G94	Feed per minute
G95	Feed per revolution

Command	Function
G98, G99	Fixed cycle initial level return, R point level return
G114.1, G113 (G113.1)	Spindle synchronization control I, Spindle synchronization cancel command (*4)
G120.1, G121	Machining condition selection I
G122	Sub part system control I
G127	Prohibits the reverse run in all part systems
G145	Wait for completion of sub part system OFF
G180	Interactive cycle insertion
M98, M99	Subprogram call, main program return
M,S,T,B	M, S, T, B command
Macro command	Local variable, Common variable, Arithmetic commands (such as four basic arithmetic rule, trigonometric functions, square root), Control commands (IF-, GOTO- and WHILE-DO-)

(*1) If inch and metric are changed over during workpiece installation error compensation, an alarm occurs.

(*2) If the workpiece coordinate system is changed during workpiece installation error compensation, an alarm occurs.

(*3) Only the three orthogonal axes designated by the rotary axis configuration parameter can be commanded. If two rotary axes are commanded, a program error (P951) will occur.

(*4) The workpiece installation error compensation can be commanded using the C axis in spindle synchronization.

Modes where workpiece installation error compensation (including cancel command) is available

If workpiece installation error compensation is commanded in a mode other than those listed below, a program error (P546) occurs.

Mode	Function
G00, G01	Positioning, Linear Interpolation
G05 P0, P1, P2, P10000	High-speed machining mode
G05.1 Q0, Q1	High-speed high-accuracy control I
G08 P0, P1	High-accuracy control
G13.1	Polar coordinate interpolation cancel
G15	Polar coordinate command cancel
G17, G18, G19	Plane selection
G20, G21	Inch command, Metric command
G23 (G23.2)	Stroke check before travel OFF
G40	Tool radius compensation cancel
G40.1	Normal line control cancel
G49	Tool length compensation cancel
G50	Scaling cancel
G50.1	G command mirror image cancel
G54 to G59, G54.1	Workpiece coordinate system selection, Extended workpiece coordinate system selection
G54.4 P0	Workpiece installation error compensation cancel
G61 G61.1 G64	Exact stop check mode High-speed high-accuracy control I ON Cutting mode
G67	User macro modal call OFF
G69 (G69.1)	Coordinate rotation cancel, 3-dimensional coordinate conversion cancel
G80	Fixed cycle cancel
G90, G91	Absolute value command, Incremental value command
G93	Inverse time feed
G94	Feed per minute
G95	Feed per revolution
G97	Constant surface speed control OFF
G98, G99	Fixed cycle/initial point return, Fixed cycle/R point return
G114.1, G113 (G113.1)	Spindle synchronization control I, Spindle synchronization cancel command (*1)

(*1) The workpiece installation error compensation can be commanded using the C axis in spindle synchronization.

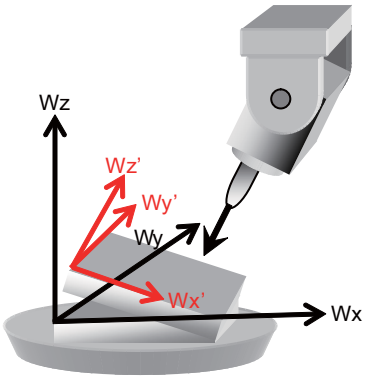
Parameters and PLC signal status to which the workpiece installation error compensation command can not be issued

When the following functions are enabled, workpiece installation error compensation cannot be commanded. If it is commanded, a program error (P546) occurs.

Function name	Related parameter	Related PLC signal
Parameter coordinate conversion	#8621 to #8627	-
Mirror image by parameter setting	#8211	-
Mirror image by external input	-	Y7C0 to Y7DF

Skip during workpiece installation error compensation command

For the skip operation during workpiece installation error compensation, the movement direction is on the workpiece installation error coordinate system in the same way as normal skip operation.

Machining program	
N1 G54.4 Pp ;	
N2 G90 G31 Z0. F100;	

The axis moves on the workpiece installation coordinate system in the N2 block.

Refer to the chapter "22 Measurement Support Functions" for skip function.

Combination with arbitrary axis exchange

In a compounded machine, a single rotary tool may be shared among multiple part systems. In this case, use the arbitrary axis exchange control function that allows you to freely exchange any axis between part systems.

When the workpiece installation error compensation is performed by combining with the arbitrary axis exchange control, set the parameter "#1450 5axis_Spec/bit0" to "1" (Set by 2nd axis name). Also, set the axis configuration in the part system when the workpiece installation error compensation is executed to the rotary axis configuration parameters #7900 or later, #7902, #7922, #7932, #7942, #7952 with 2nd axis name (E.g. A1, B2). When the parameter "#1450 5axis_Spec/bit0" is set to "0" (Set by axis name), if the workpiece installation error compensation is commanded after the arbitrary axis exchange, the program error (P546) occurs.

You can set the configurations up to the number of valid part systems in the rotary axis configuration parameter. With multiple configurations set, you can perform the workpiece installation error compensation in different axis configurations.

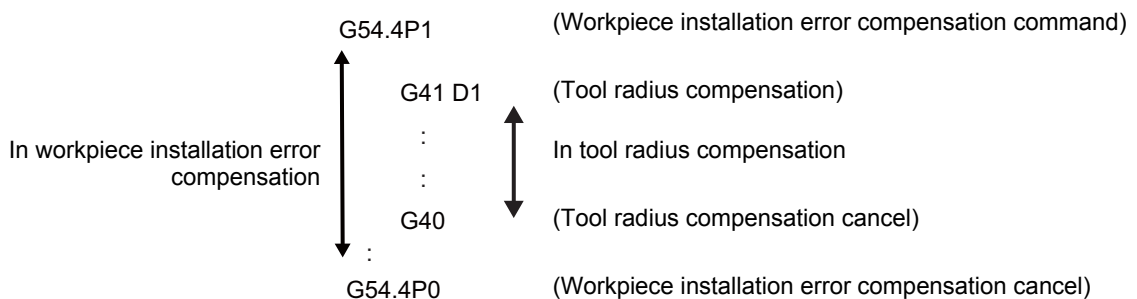
Note

(1) When the number of valid part system is "5" or more, the number of settable configurations is "4".



Precautions

- (1) During workpiece installation error compensation command, the coordinates on the workpiece installation coordinate system are set to the system variables for position information (#5001 to #5116 (excluding #5021 to #5036 and #5101 to #5116)). However, coordinate values of the machine coordinate system are set to the variables #5021 to #5036 and #5101 to #5116 even when the workpiece installation error compensation is commanded.
- (2) Workpiece installation error compensation will be canceled if reset is executed during workpiece installation error compensation.
- (3) In order to enable machine lock, interlock, and external deceleration, input their signals not to the axes of the workpiece installation coordinate system but to the axes of the machine coordinate system which are in actual operation.
- (4) Tool radius compensation, 3-dimensional tool radius compensation (Tool's vertical-direction compensation), tool length compensation, tool center point control, mirror image by G code, scaling, inclined surface machining, and fixed cycle command must be nested in workpiece installation error compensation. They must be commanded during workpiece installation error compensation mode.



- (5) A coordinate conversion is not applied to a manual interruption, it is operated on the machine coordinate system. During workpiece installation error compensation, if a manual interruption or tool center point control is executed with the manual absolute switch ON, make sure to return to the position before the interruption to restart an automatic operation.
An operation error (M01 0170) occurs in the following cases.
 - ♦When you restart automatic operation in a position different from the one before interruption
 - ♦When an interruption to a rotary axis is made during workpiece installation error compensation
 - ♦When an automatic operation handle interruption is made during workpiece installation error compensation
- (6) It is not possible to execute an MDI interruption, PLC interruption, or macro interruption during workpiece installation error compensation. If any of these is executed during workpiece installation error compensation, an operation error (M01 0170) occurs. If macro interruption is enabled during workpiece installation error compensation, a program error (P545) occurs. Also when workpiece installation error compensation is commanded while macro interruption is active, a program error (P546) occurs.
- (7) When workpiece installation error compensation is commanded during MDI interruption, PLC interruption, or macro interruption, a program error (P546) occurs.
- (8) In graphic check, the path of the original workpiece coordinate system is displayed.
- (9) Tracing is done using the machine coordinate values.
- (10) Program restart from the block after the workpiece installation error compensation command is issued cannot be implemented. If commanded, a program error (P49) occurs.
- (11) G0 commands during workpiece installation error compensation are always interpolation type. (Non-interpolation type is not available.)
- (12) During workpiece installation error compensation, a buffer correction is not possible. If you attempt buffer correction, an error message appears.
- (13) When the parameter "#1450 5axis_Spec/bit0" is set to "0" (Set by axis name), set the axis in the 1st part system with the axis name (E.g. X, C) in the parameters #7900 to #7902, #7922, #7932, #7942 and #7952. If you command workpiece installation error compensation in a part system where any of the designated axes is not ready, a program error (P932) occurs.
- (14) If a manual arbitrary feed is performed during workpiece installation error compensation, an operation error (M01 0170) occurs.
- (15) If a manual speed command is issued during workpiece installation error compensation, an operation error (M01 0170) occurs.

- (16) If a figure rotation is performed during workpiece installation error compensation, a program error (P545) occurs. In addition, if workpiece installation error compensation is commanded during figure rotation, a program error (P546) occurs.
- (17) If a coordinate rotation by parameter is executed during workpiece installation error compensation, a program error (P545) occurs. In addition, if workpiece installation error compensation is commanded during coordinate rotation by parameter, a program error (P546) occurs.
- (18) If a linear angle command is issued during workpiece installation error compensation, a program error (P545) occurs.
- (19) If a geometric command is issued during workpiece installation error compensation, a program error (P545) occurs.
- (20) Chopping is not possible while an axis in workpiece installation error compensation is moving. In addition, if a movement command of workpiece installation error compensation is issued during chopping, an operation error (M01 0151) occurs.
- (21) If a command to rotate a rotary axis by 180° or more in a block is issued, a program error (P547) occurs. In such a case, divide the block into multiple ones so that axis rotation angle is within 180°.
- (22) If workpiece installation error compensation is commanded during user macro modal call B (G66.1), a program error (P33) occurs.
- (23) Do not change the workpiece offset amounts during workpiece installation error compensation.
- (24) Even if the workpiece installation error compensation amounts are changed during workpiece installation error compensation, the offset at startup will be enabled.
- (25) If axis movement is commanded for any axis not specified in the rotary axis configuration parameters during workpiece installation error compensation when SSS control is set to ON, a program error (P545) occurs.
- (26) If the workpiece installation error compensation is commanded in spindle mode when the axis set to the spindle/C axis is used as a rotary axis of workpiece installation error compensation, a program error (P934) occurs. When the mode is changed to the spindle mode during the workpiece installation error compensation, an operation error (M01 0186) occurs. This operation error can be cleared by NC reset.
- (27) The acceleration with the rapid traverse (G00) in high-accuracy control mode during the workpiece installation error compensation is as below depending on the value of the parameter "#1250 set22/bit3" (Select how to determine the acceleration for rapid traverse (G0) in high-accuracy control).
 - When "#1250 set22/bit3" = "0"
 - Same as the acceleration for cutting feed (G01) (the acceleration determined with "#1206 G1bF" (Maximum speed) and "#1207 G1btL" (Time constant))
 - When "#1250 set22/bit3" = "1"
 - Same as the acceleration for rapid traverse (G00) (the acceleration determined with "#2001 Rapid" (Rapid traverse rate) and "#2004 G0tL" (G0 time constant (linear))) However, while SSS control is disabled, the operation is same as when the parameter "#1250 set22/bit3" is set to "0".
- (28) If any of the orthogonal axes of all the active part systems is under machine lock during workpiece installation error compensation, normal synchronous tapping is applied even though the high-speed synchronous tapping specification is enabled.

Coordinate System Setting Functions

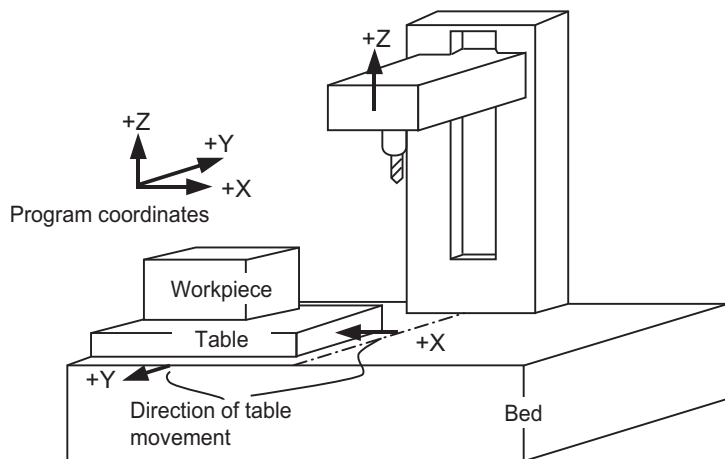
20.1 Coordinate Words and Control Axes



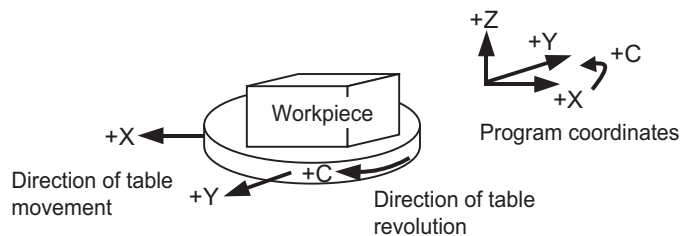
Function and purpose

The number of control axes is set to "3" in the standard specifications; however, up to eight axes can be controlled if an additional axis is added. To specify each machining direction, use alphabetical coordinate words that are pre-defined appropriately.

X-Y table



X-Y and rotating table



20.2 Types of Coordinate Systems

20.2.1 Basic Machine, Workpiece and Local Coordinate Systems



Function and purpose

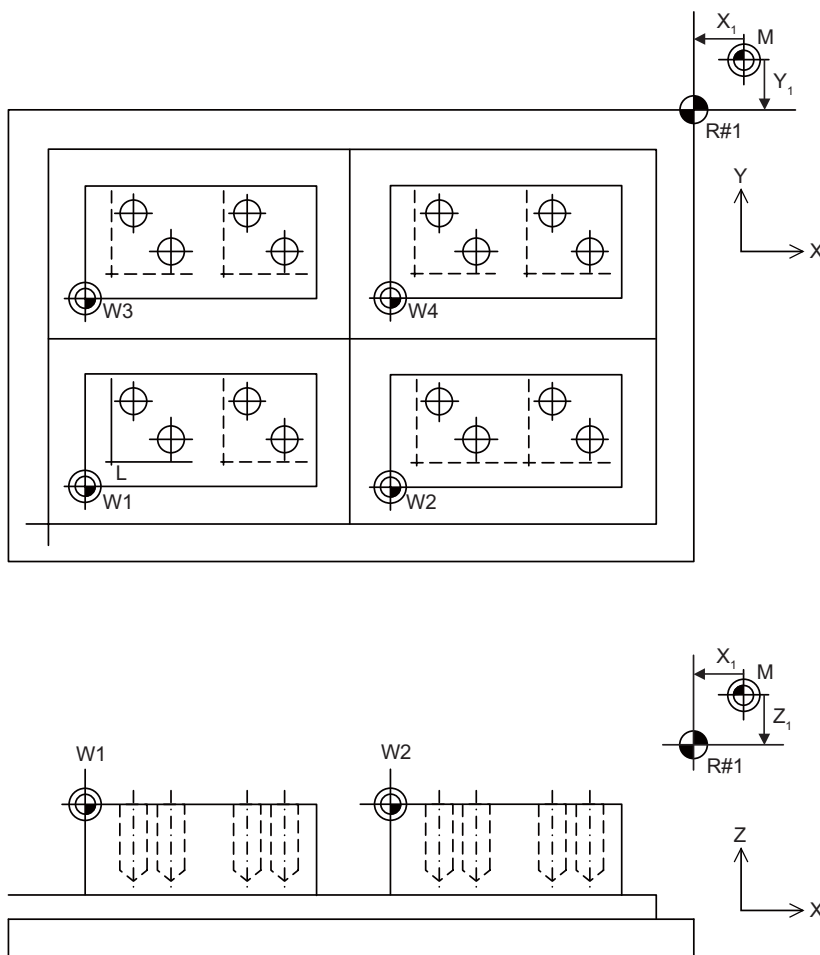
The basic machine coordinate system is fixed in the machine and it denotes that position which is determined inherently by the machine.

The workpiece coordinate systems are used for programming and in these systems the basic point on the workpiece is set as the coordinate zero point.

The local coordinate systems are created on the workpiece coordinate systems and they are designed to facilitate the programs for parts machining.

Upon completion of the reference position return, the basic machine coordinate system and workpiece coordinate systems (G54 to G59) are automatically set with reference to the parameters.

The basic machine coordinate system is set so that the first reference position is brought to the position specified by the parameter from the basic machine coordinate zero point (machine zero point).



M: Basic machine coordinate system

W: Workpiece coordinate system

L: Local coordinate system

The local coordinate systems (G52) are valid on the coordinate systems designated by workpiece coordinate systems 1 to 6.

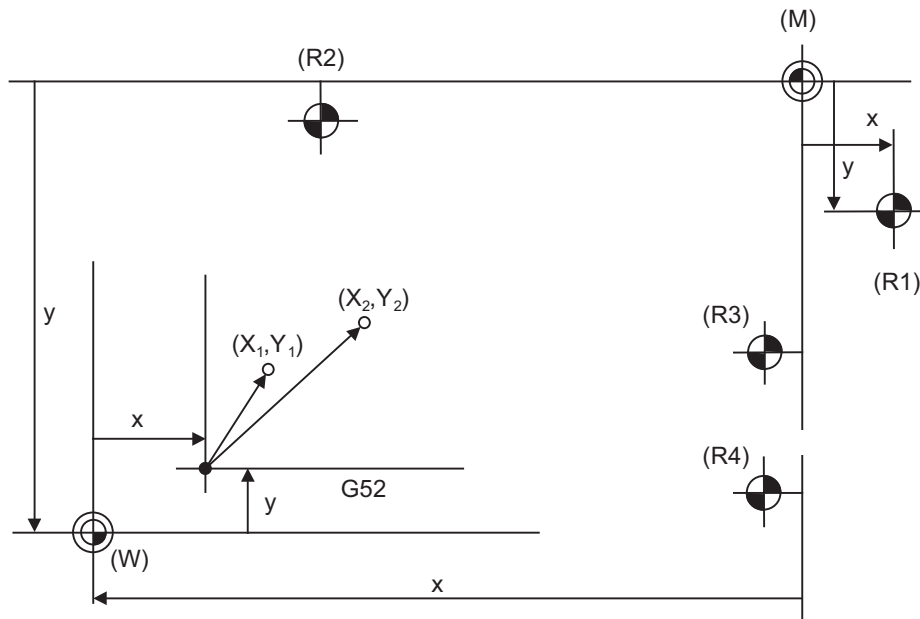
The hypothetical machine coordinate system can be set on the basic machine coordinate system using a G92 command. At this time, the workpiece coordinate system 1 to 6 is also simultaneously shifted.

Also refer to "Coordinate Systems and Coordinate Zero Point symbols".

20.2.2 Machine Zero Point and 2nd, 3rd, 4th Reference Position (Zero Point)

**Function and purpose**

The machine zero point serves as the reference for the basic machine coordinate system. It is inherent to the machine and is determined by the reference (zero) position return. 2nd, 3rd and 4th reference positions relate to the position of the coordinates that have been set beforehand by parameter from the zero point of the basic machine coordinate system.



- (M) Basic machine coordinate system
- (W) Workpiece coordinate systems (G54 to G59)
- (R1) 1st reference position
- (R3) 3rd reference position

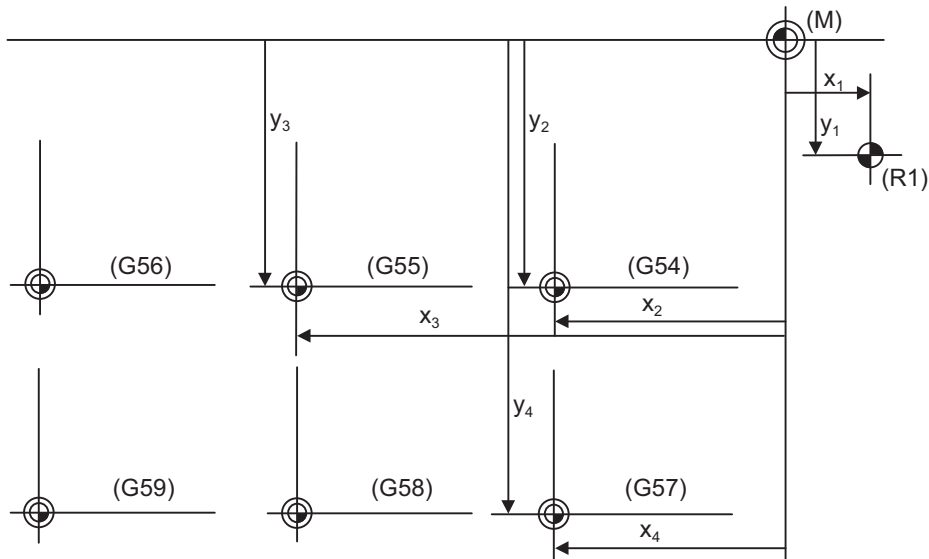
- (G52) Local coordinate system
- (R2) 2nd reference position
- (R4) 4th reference position

20.2.3 Automatic Coordinate System Setting



Function and purpose

This function creates each coordinate system according to the parameter values input beforehand from the setting and display unit when the first manual reference position return or the reference position is reached with the dog-type reference position return when the NC power is turned ON. The actual machining program is programmed over the coordinate systems that have been set above.



(M) Basic machine coordinate system
 (G54) Workpiece coordinate system 1
 (G56) Workpiece coordinate system 3
 (G58) Workpiece coordinate system 5

(R1) 1st reference position
 (G55) Workpiece coordinate system 2
 (G57) Workpiece coordinate system 4
 (G59) Workpiece coordinate system 6



Detailed description

- (1) The coordinate systems created by this function are as follow:
 - Basic machine coordinate system
 - Workpiece coordinate systems (G54 to G59)
- (2) The parameters related to the coordinate system all provide the distance from the zero point of the basic machine coordinate system. Therefore, after deciding at which position the first reference position should be set in the basic machine coordinate system and then set the zero point positions of the workpiece coordinate systems.
- (3) When the automatic coordinate system setting function is executed, shifting of the workpiece coordinate system with G92, setting of the local coordinate system with G52, shifting of the workpiece coordinate system with origin set, and shifting of the workpiece coordinate system with manual interrupt will be canceled.
- (4) The dog-type reference position return will be executed when the first time manual reference position return or the first time automatic reference position return is executed after the power has been turned ON. It will be also executed when the dog-type is selected by the parameter for the manual reference position return or the automatic reference position return for the second time onwards.

CAUTION

If the workpiece coordinate offset amount is changed during automatic operation (including during single block operation), it will be validated from the next block or after multiple blocks of the command.

20.2.4 Coordinate System for Rotary Axis



Function and purpose

The axis designated as the rotary axis with the parameters is controlled with the rotary axis' coordinate system.

The rotary axis includes the rotating type (short-cut valid/invalid) and linear type (workpiece coordinate position linear type and all coordinate position linear type).

The workpiece coordinate position range is 0 to 359.999° for the rotating type, and 0 to ± 99999.999° for the linear type.

The machine coordinate value and relative position differ according to the parameters.

The rotary axis is commanded with a degree (°) unit regardless of the inch or metric designation.

The rotary axis type can be set with the parameter "#8213 rotation axis type" for each axis.

	Rotary axis				Linear axis
	Rotating type rotary axis		Linear type rotary axis		
	Short-cut invalid	Short-cut valid	Workpiece coordinate position linear type	All-coordinate position linear type	
#8213 setting value	0	1	2	3	-
Workpiece coordinate position	Displayed in the range of 0° to 359.999°.		Displayed in the range of 0° to ± 99999.999°.		
Machine coordinate position/relative position	Displayed in the range of 0° to 359.999°.			Displayed in the range of 0° to ± 99999.999°.	
ABS command	The incremental amount from the end point to the current position is divided by 360 degrees, and the axis moves by the remainder amount according to the sign.	Moves with a short-cut to the end point.	In the same manner as the normal linear axis, it moves according to the sign by the amount obtained by subtracting the current position from the end point (without rounding up to 360 degrees).		
INC command	Moves in the direction of the commanded sign by the commanded incremental amount starting at the current position.				
Reference position return	Depends on the absolute command or the incremental command during the movement to the intermediate point.				
	Returns with movement within 360 degrees.			Moves and returns in the R point direction for the difference from the current position to the R point.	



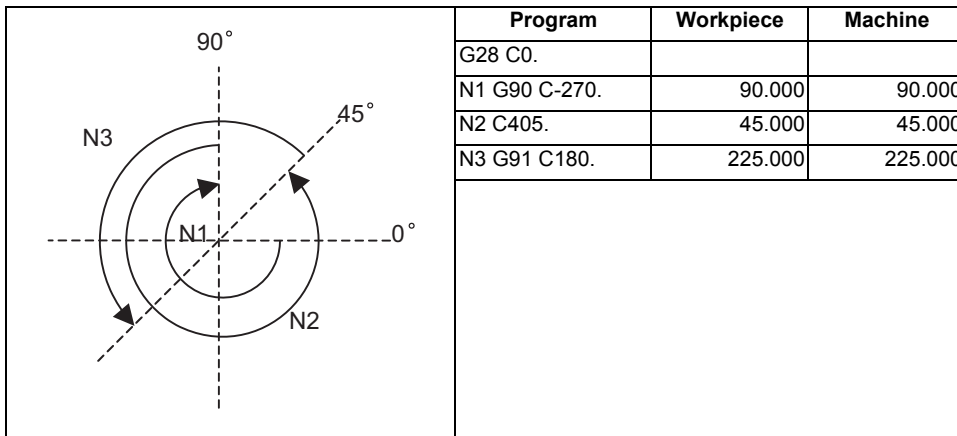
Operation example

Examples of differences in the operation and counter displays according to the type of rotation coordinate are given below.

(The workpiece offset is set as 0°.)

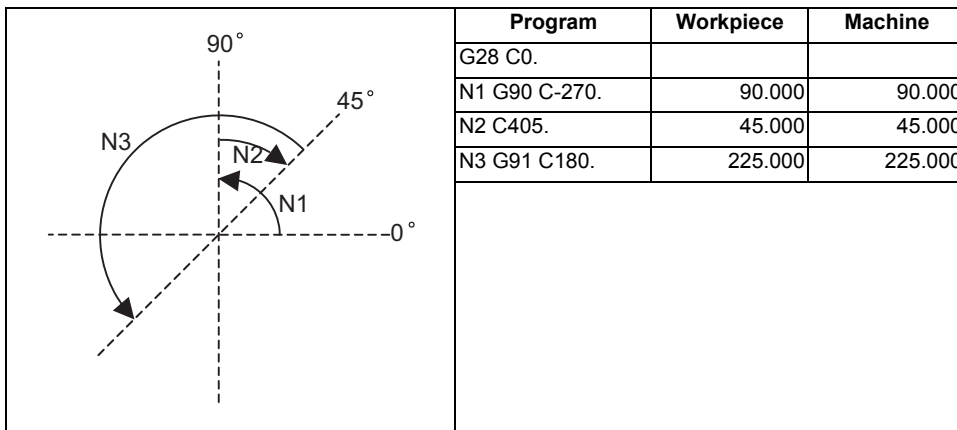
Rotary type (short-cut invalid)

- (1) The machine coordinate position, workpiece coordinate position and relative position are displayed in the range of 0° to 359.999°.
- (2) For the absolute position command, the axis moves according to the sign by the remainder amount obtained by dividing by 360°.



Rotary type (short-cut valid)

- (1) The machine coordinate position, workpiece coordinate position and relative position are displayed in the range of 0° to 359.999°.
- (2) For the absolute position command, the axis rotates to the direction having less amount of movement to the end point.



Linear type (workpiece coordinate position linear type)

- (1) The coordinate position counter other than the workpiece coordinate position is displayed in the range of 0° to 359.999°.
The workpiece coordinate position is displayed in the range of 0 to ±99999.999°.
- (2) The movement is the same as the linear axis.
- (3) During reference position return, the axis moves in the same manner as the linear axis until the intermediate point is reached. The axis returns with a rotation within 360° from the intermediate point to the reference position.
- (4) During absolute position detection, even if the workpiece coordinate position is not within the range of 0 to 359.999°, the system will start up in the range of 0 to 359.999° when the power is turned ON again.

	Program	Workpiece	Machine	Relative position
	G28 C0.			
	N1 G90 C-270.	-270.000	90.000	90.000
	N2 C405.	405.000	45.000	45.000
	N3 G91 C180.	585.000	225.000	225.000
	After the power is turned ON again			
	Workpiece	Machine		
	225.000	225.000		

Linear type (all coordinate position linear type)

- (1) The workpiece coordinate position counter is displayed in the range of 0 to ±99999.999°.
- (2) The movement is the same as the linear axis.
- (3) During reference position return, the axis moves in the same manner as the linear axis until the intermediate point is reached.
The axis rotates by the difference from the intermediate point to the reference position and returns to the reference position.
- (4) During absolute position detection, the system starts up at the position where the power was turned OFF when the power is turned ON again.

	Program	Workpiece	Machine	Relative position
	G28 C0.			
	N1 G90 C-270.	-270.000	-270.000	-270.000
	N2 C405.	405.000	405.000	405.000
	N3 G91 C180.	585.000	585.000	585.000
	After the power is turned ON again			
	Workpiece	Machine		
	585.000	585.000		

20.3 Basic Machine Coordinate System Selection; G53



Function and purpose

The basic machine coordinate system is the coordinate system that expresses the position (tool change position, stroke end position, etc.) that is characteristic to the machine. The tool is moved to the position commanded on the basic machine coordinate system with the G53 command and the coordinate command that follows.



Command format

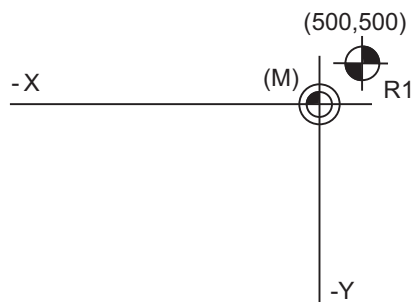
```
(G90)G53 X__ Y__ Z__ α__ ;
```

α	Additional axis
---	-----------------



Detailed description

- (1) When the power is switched on, the basic machine coordinate system is automatically set as referenced to the reference (zero) position return position, which is determined by the automatic or manual reference (zero) position return.
- (2) The basic machine coordinate system is not changed by the G92 command.
- (3) The G53 command is valid only in the designated block.
- (4) In the incremental value command mode (G91), the G53 command provides movement with the incremental value in the coordinate system being selected.
- (5) Even if G53 is commanded, the tool radius compensation amount for the commanded axis will not be canceled.
- (6) The 1st reference coordinate position indicates the distance from the basic machine coordinate system zero point to the reference position (zero point) return position.
- (7) The G53 command will move with cutting feedrate or rapid traverse following command modal.
- (8) If the G53 command and G28 command (reference position return) are issued in the same block, the command issued last will be valid.



(M) Basic machine coordinate system
(R1) 1st reference position

1st reference position coordinate value: X=+500 and Y=+500

- (9) If the G53 command and G28 command (reference position return) are issued in the same block, the command issued last will be valid.
- (10) Even if G53 is commanded, the tool radius compensation amount for the commanded axis will not be canceled.

(11) Depending on the MTB specifications, all movement commands in the G53 command block may run in rapid traverse mode (parameter "#1253 set/bit5").

(a) When the movement method of the G53 command block follows the command modal [Example in which the G53 block is executed during G01 modal]

Program	G group 01 modal	Cutting or Rapid traverse
N01 G01 X100. Z100. F1000;	G01	Cutting
N02 G53 X200. Z200.;	G01	Cutting
N03 X300. Z300.;	G01	Cutting

[Example in which the G53 block is executed during G00 modal]

Program	G group 01 modal	Cutting or Rapid traverse
N01 G00 X100. Z100.;	G00	Rapid traverse
N02 G53 X200. Z200.;	G00	Rapid traverse
N03 X300. Z300.;	G00	Rapid traverse

(b) When all the movement methods of the G53 command block are set to rapid traverse [Example in which the G53 block is executed during G01 modal]

Program	G group 01 modal	Cutting or Rapid traverse
N01 G01 X100. Z100. F1000;	G01	Cutting
N02 G53 X200. Z200.;	G01	Rapid traverse
N03 X300. Z300.;	G01	Cutting

The G group 01 modal does not change in the G53 command block; only the operation is set to rapid traverse.



Relationship with other functions

(1) Tool Compensation Functions

When the G53 command is issued, the tool compensation amount of the axis with the movement command designated is canceled temporarily.

(2) Machine coordinate system selection, Feedrate designation

If an "F" command is specified when there are no specifications for the feedrate command for G53, a program error (P39) will occur.

(3) Inclined surface machining

When the G53 command is issued during inclined surface machining, a program error (P951) occurs.

(4) Polar coordinate interpolation

Do not issue the G53 command during the polar coordinate interpolation mode.

(5) Polar coordinate command

The axis command with the G53 command is not interpreted as the polar coordinate command during the polar coordinate command mode.

(6) Tool length compensation along the tool axis

When the G53 command is designated while the compensation status is still established, the compensation is temporarily canceled, and the tool moves to the machine position designated by G53.

(7) G command mirror image

The mirror image will not be applied to the G53 command.

(8) High-speed High-accuracy Control

A program error will occur if the G53 command is issued during the high-speed high-accuracy control II mode.

(9) 3-dimensional coordinate conversion

Coordinate conversion will not be carried out for the machine coordinate system even if G53 command is issued in the 3-dimensional coordinate conversion modal.

(10) Tool center point control

A program error (P942) will occur if the G53 command is issued during the tool center point modal.

(11) Workpiece installation error compensation

A program error (P545) will occur if the G53 command is issued during workpiece installation error compensation.



Precautions

- (1) In the machine with the specifications in which all the movement commands of the G53 command block run in rapid traverse mode, if the G53 and G01 commands are issued in the same block, the block is set to rapid traverse. However, the G group 01 modal is switched; therefore, the movement commands in the next and subsequent blocks run in cutting feed mode.

[Example in which the G53 and G01 commands are issued in the same block]

Program	G group 01 modal	Cutting or Rapid traverse
N01 G00 X100. Z100.;	G00	Rapid traverse
N02 G53 G01 X200. Z200. F1000;	G01	Rapid traverse
N03 X300. Z300.;	G01	Cutting

- (2) The travel amount of G53 block will be analyzed after the movement by the previous block is completed. Thus update of the coordinate counter such as "Next cmd" or "Remain cmd" may not follow the actual movement.

20.4 Coordinate System Setting; G92



Function and purpose

By commanding G92, the absolute value coordinate system and current position display value can be newly preset according to the command value without moving the machine.



Command format

G92 X__ Y__ Z__ α__ ;

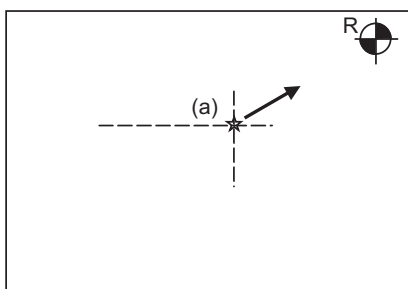
α Additional axis



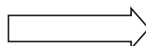
Detailed description

After the power is turned on, the first reference position return will be done with dog-type, and when completed, the coordinate system will be set automatically.

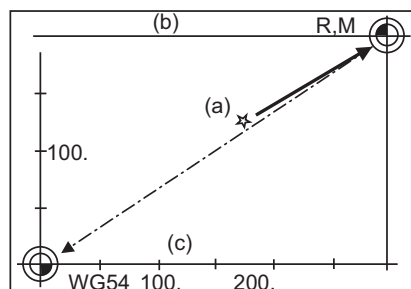
(Automatic coordinate system setting)



Reference position return completed



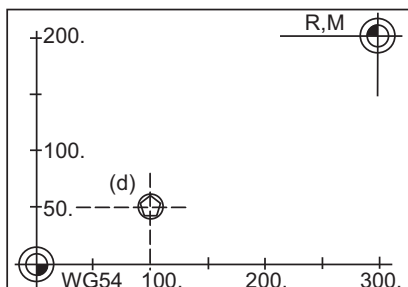
The basic machine coordinate system and workpiece coordinate system are created at the preset position.



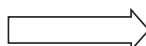
- (a) Power ON position
- (b) Basic machine coordinate system
- (c) Workpiece coordinate system

[Relative position]	[Workpiece]
X 0.000	X 300.000
Y 0.000	Y 200.000

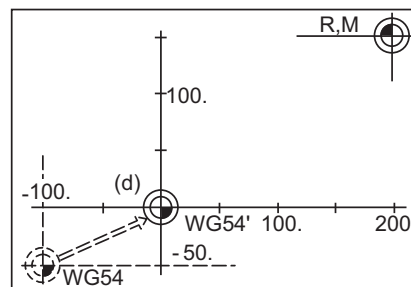
By commanding G92, the absolute value (workpiece) coordinate system and current position display value can be preset in the command value without moving the machine.



Coordinate system setting



For example, if G92 X0 Y0; is commanded, the workpiece coordinate system will be newly created.



(d) Tool position

[Relative position]	[Workpiece]
X -200.000	X 100.000
Y -150.000	Y 50.000

[Relative position]	[Workpiece]
X 0.000	X 0.000
Y 0.000	Y 0.000

Note

- (1) If the workpiece coordinate system deviated because the axis is moved manually when the manual absolute position switch is OFF, etc., the workpiece coordinate system can be corrected with the following steps.
- ♦Execute reference position return while the coordinate system is deviated.
 - ♦After that, command G92G53X0Y0Z0;. With this command, the workpiece coordinate position and current position will be displayed, and the workpiece coordinate system will be preset to the offset value.

**Precautions**

- (1) If the parameter "#1279 ext15/bit5" is set to "1", the coordinate systems setting (G92) shift amount is cleared when the axis reaches to the manual reference position.

20.5 Local Coordinate System Setting; G52



Function and purpose

The local coordinate systems can be set on the G54 through G59 workpiece coordinate systems using the G52 command so that the commanded position serves as the programmed zero point.
 The G52 command can also be used instead of the G92 command to change the deviation between the zero point in the machining program and the machining workpiece zero point.



Command format

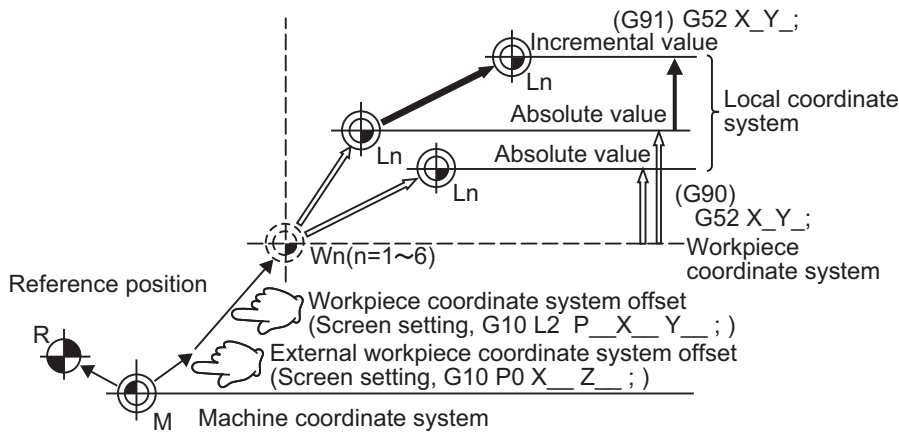
G54 (G54 to G59) G52 X__ Y__ Z__ α__ ;

α	Additional axis
---	-----------------



Detailed description

- (1) The G52 command is valid until a new G52 command is issued, and the tool does not move. This command, G52, comes in handy for employing another coordinate system without changing the zero point positions of the workpiece coordinate systems (G54 to G59).
- (2) The local coordinate system offset will be cleared by the dog-type manual reference (zero) point return or reference (zero) point return performed after the power has been switched ON.
- (3) The local coordinate system is canceled by (G54 to G59) G52 X0 Y0 Z0 α0;.
- (4) Coordinate commands in the absolute value (G90) cause the tool to move to the local coordinate system position.



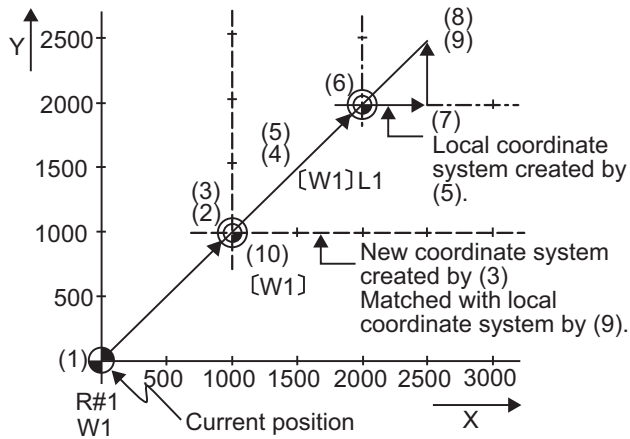
<Note>

- If the program is executed repeatedly, the workpiece coordinate system will deviate each time. Thus, when the program is completed, the reference position return operation must be commanded.

20 Coordinate System Setting Functions

(Example 1) Local coordinates for absolute value mode (The local coordinate system offset is not cumulated)

- (1) G28 X0 Y0 ;
- (2) G00 G90 X1. Y1. ;
- (3) G92 X0 Y0 ;
- (4) G00 X500 Y500 ;
- (5) G52 X1. Y1. ;
- (6) G00 X0 Y0 ;
- (7) G01 X500 F100 ;
- (8) Y500 ;
- (9) G52 X0 Y0 ;
- (10) G00 X0 Y0 ;



The local coordinate system is created by (5), canceled by (9) and matched with the coordinate system for (3).

<Note>

- If the program is executed repeatedly, the workpiece coordinate system will deviate each time. Thus, when the program is completed, the reference position return operation must be commanded.

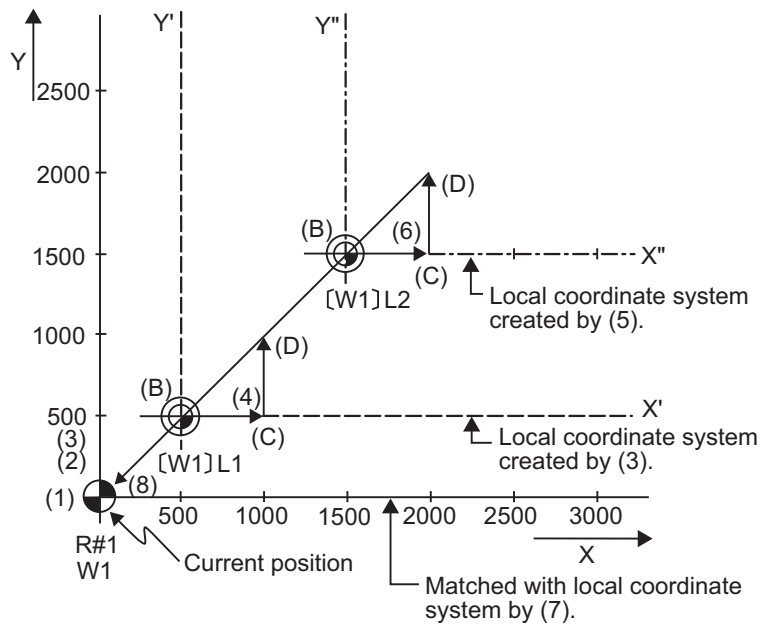
(Example 2) Local coordinates for incremental value mode (The local coordinate system offset is cumulated.)

<Main program>

- (1) G28 X0 Y0 ;
- (2) G92 X0 Y0 ;
- (3) G91 G52 X500 Y500 ;
- (4) M98 P100 ;
- (5) G52 X1. Y1. ;
- (6) M98 P100 ;
- (7) G52 X-1.5 Y-1.5 ;
- (8) G00 G90 X0 Y0 ;
- M02 ;

<Subprogram>

- (A) O100 ;
- (B) G90 G00 X0 Y0 ;
- (C) G01 X500 ;
- (D) Y500 ;
- (E) G91 ;
- (F) M99 ;

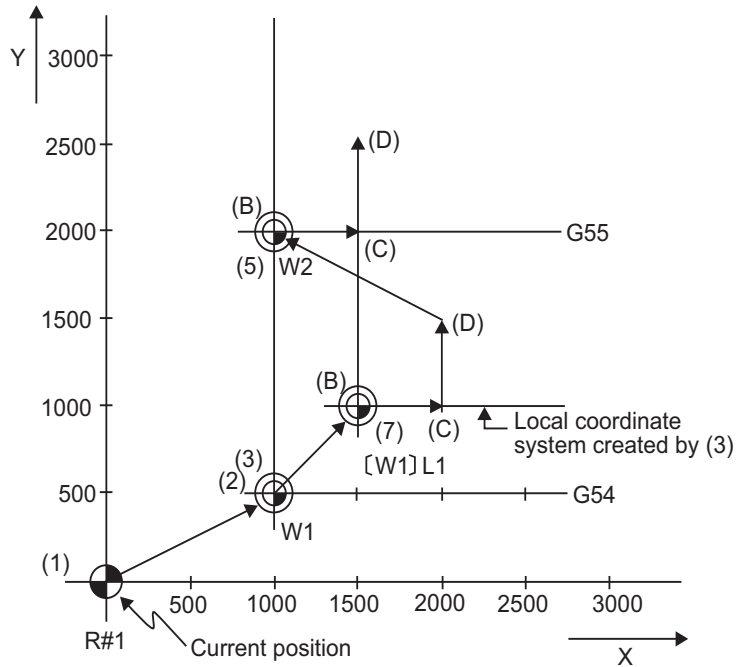


The local coordinate system X'Y' is created at the XY coordinate system (500,500) position by (3).
 The local coordinate system X''Y'' is created at the X'Y' coordinate system (1000,1000) position by (5).
 The local coordinate system is created at the X'''Y''' coordinate system (-1500, -1500) position by (7). In other words, the same occurs as when the local coordinate system and XY coordinate system are matched and the local coordinate system is canceled.

20 Coordinate System Setting Functions

(Example 3) When used together with workpiece coordinate system

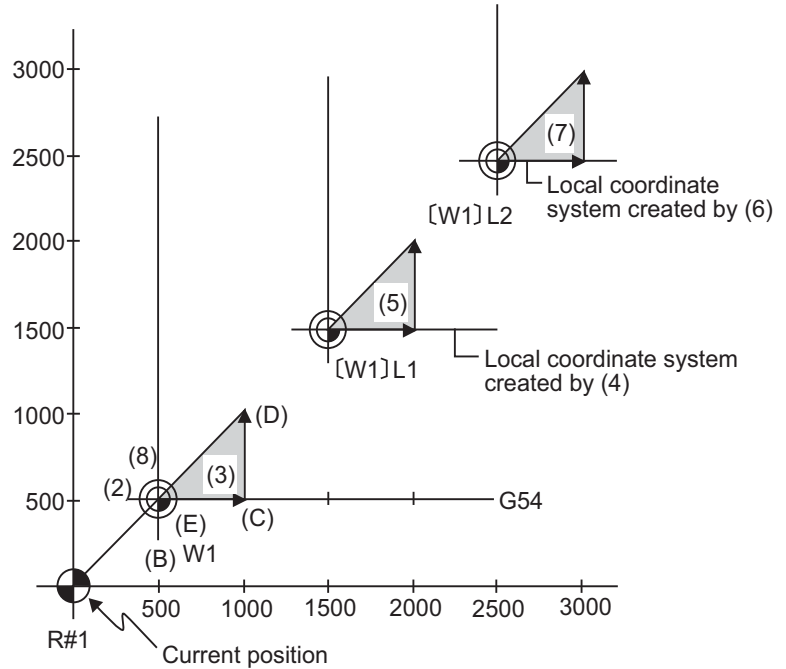
- (1) G28 X0 Y0 ; G54 G55
- (2) G00 G90 G54 X0 Y0 ; X 1000 1000 } Workpiece coordinate system offset
- (3) G52 X500 Y500 ; Y 500 2000 } (parameter setting value)
- (4) M98 P200 ;
- (5) G00 G90 G55 X0 Y0 ;
- (6) M98 P200 ;
- (7) G00 G90 G54 X0 Y0 ;
- :
- M02 ;
- (A) O200 ;
- (B) G00 X0 Y0 ;
- (C) G01 X500 F100 ;
- (D) Y500 ;
- (E) M99 ;



The local coordinate system is created at the G54 coordinate system (500,500) position by (3), but the local coordinate system is not created for the G55 coordinate system. During the movement for (7), the axis moves to the G54 local coordinate system's reference position (zero point). The local coordinate system is canceled by "G90 G54 G52 X0 Y0;".

(Example 4) Combination of workpiece coordinate system G54 and multiple local coordinate systems

- (1) G28 X 0 Y 0 ; G54
- (2) G00 G90 G54 X0 Y0 ; X 500 } Workpiece coordinate system offset
- (3) M98 P300; Y 500 } (parameter setting value)
- (4) G52 X1. Y1. ;
- (5) M98 P300 ;
- (6) G52 X2. Y2. ;
- (7) M98 P300 ;
- (8) G52 X0 Y0 ;
- :
- (A) O300 ;
- (B) G00 X0 Y0 ;
- (C) G01 X500 F100 ;
- (D) Y500 ;
- (E) X0 Y0 ;
- (F) M99 ;
- %



The local coordinate system is created at the G54 coordinate system (1000,1000) by (4).
 The local coordinate system is created at the G54 coordinate system (2000,2000) by (6).
 The G54 coordinate system and local coordinate system are matched by (8).

20.6 Workpiece Coordinate System Setting and Offset; G54 to G59 (G54.1)



Function and purpose

- (1) The workpiece coordinate systems facilitate the programming on the workpiece, serving the reference position of the machining workpiece as the zero point.
- (2) These commands enable the tool to move to the positions in the workpiece coordinate system. There are extended workpiece coordinate systems (G54.1) in addition to 6 workpiece coordinate systems, which are used by the programmer for programming (G54 to G59). The number of the extended workpiece coordinate systems varies depending on the MTB specifications.
- (3) Among the workpiece coordinate systems currently selected by these commands, any workpiece coordinate system with coordinates that have been commanded by the current position of the tool is reset. (The "current position of the tool" includes the compensation amounts for tool radius, tool length and tool position compensation.)
- (4) A hypothetical machine coordinate system with coordinates that have been commanded by the current position of the tool is set by these commands.
(The "current position of the tool" includes the compensation amounts for tool radius, tool length and tool position compensation.) (G54, G92)

Workpiece coordinate system

(G90) G54 to G59 ; Workpiece coordinate system selection

(G54 to G59) G92 X__ Y__ Z__ α__ ; Workpiece coordinate system setting

G90(G91) G10 L2 Pn X__ Y__ Z__ ; Workpiece coordinate system offset amount setting

α	Additional axis
---	-----------------

Note

- (1) The compensation amount in the G91 will be an incremental amount and will be accumulated each time the program is executed. To avoid such an error, command G90 or G91 before G10 as much as possible.

Extended workpiece coordinate system

G54.1 Pn ; Extended workpiece coordinate system selection (P1 to P300) (*1)

**G54.1 Pn ;
G92 X__ Y__ Z__ ; Workpiece coordinate system setting (P1 to P300) (*1)**

G10 L20 Pn X__ Y__ Z__ ; Extended workpiece coordinate system offset amount setting (P1 to P300) (*1)

When the offset amount of the currently designated workpiece coordinate system is rewritten

G10 G54.1 Pn X__ Y__ Z__ ; Extended workpiece coordinate system offset amount setting (P1 to P300) (*1)

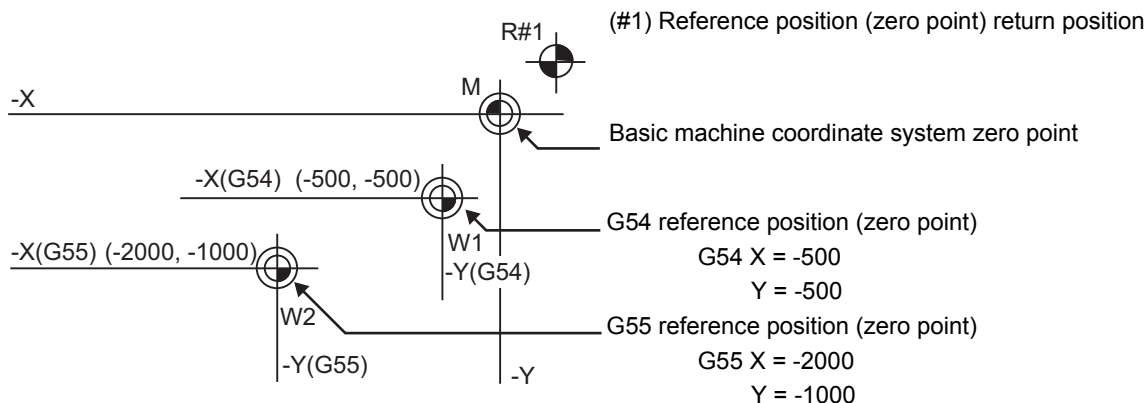
When the extended workpiece coordinate system is selected, and the offset amount is rewritten.

(*1) The maximum number of coordinate systems depends on the specifications.



Detailed description

- (1) With any of the G54 through G59 commands or G54.1P1 through G54.1P300 commands, the tool radius compensation amounts for the commanded axes will not be canceled even if workpiece coordinate system selection is commanded.
- (2) The G54 workpiece coordinate system is selected when the power is turned ON.
- (3) Commands G54 through G59 and G54.1P1 through G54.1P300 are modal commands (group 12).
- (4) The coordinate system will move with G92 in a workpiece coordinate system.
- (5) The offset setting amount in a workpiece coordinate system denotes the distance from the basic machine coordinate system zero point.



- (6) The offset settings of workpiece coordinate systems can be changed any number of times. (They can also be changed by G10 L2 Pp1 Xx1 Yy1 Zz1.)

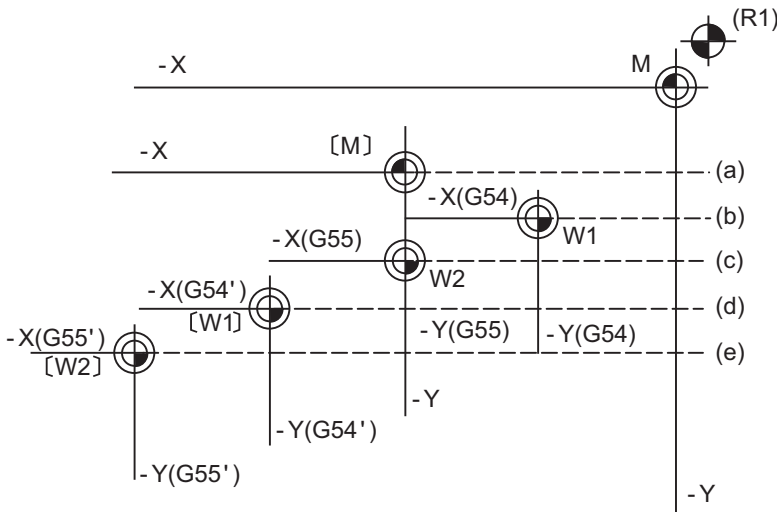
[Handling when L or P is omitted]

G10 L2 Pn Xx Yy Zz ;	n=0: Set the offset amount in the external workpiece coordinate system. n=1 to 6: Set the offset amount in the designated workpiece coordinate system. Others: The program error (P35) will occur.
G10 L2 Xx Yy Zz ;	Set the offset amount in the currently selected workpiece coordinate system. When in G54.1 modal, the program error (P33) will occur.
G10 L20 Pn Xx Yy Zz ;	n=1 to maximum number of coordinate systems: Set the offset amount in the designated workpiece coordinate system. (The number of extended workpiece coordinate systems under the specifications) Others: Program error (P35) will occur.
G10 L20 Xx Yy Zz ;	Set the offset amount in the currently selected workpiece coordinate system. When in G54 to G59 modal, the program error (P33) will occur.
G10 Pn Xx Yy Zz ;	Set the offset amount in the designated coordinate system No. by P code. When the currently selected coordinate system is G54 to G59, P1 to P6 corresponds to G54 to G59 respectively. When the external coordinate system is selected, P No. corresponds to G54.1 P1 to P300. If other values are set, the program error (P35) will occur.
G10 Xx Yy Zz ;	Set the offset amount in the currently selected coordinate system.
G10 G54.1 Xx Yy Zz ;	When there is no P code in the same block as G54.1, the program error (P33) will occur.

- (7) A new workpiece coordinate system 1 is set by issuing the G92 command in the G54 (workpiece coordinate system 1) mode. At the same time, the other workpiece coordinate systems 2 to 6 (G55 to G59) will move in parallel and new workpiece coordinate systems 2 to 6 will be set.

20 Coordinate System Setting Functions

(8) A hypothetical machine coordinate system is formed at the position that deviates from the new workpiece reference position (zero point) by an amount equivalent to the workpiece coordinate system offset amount.



(R1) Reference position 1

(a) Hypothetical machine coordinate system based on G92

(b) Old workpiece 1 (G54) coordinate system

(c) Old workpiece 2 (G55) coordinate system

(d) New workpiece 1 (G54) coordinate system

(e) New workpiece 2 (G55) coordinate system

After the power has been switched on, the hypothetical machine coordinate system is matched with the basic machine coordinate system by the first automatic (G28) or manual reference position (zero point) return.

(9) By setting the hypothetical machine coordinate system, the new workpiece coordinate system will be set at a position that deviates from that hypothetical machine coordinate system by an amount equivalent to the workpiece coordinate system offset amount.

(10) When the first automatic (G28) or manual reference position (zero point) return is completed after the power has been turned ON, the basic machine coordinate system and workpiece coordinate systems are set automatically in accordance with the parameter settings.

(11) If G54 X- Y-; is commanded after the reference position return (both automatic or manual) executed after the power is turned ON, the program error (P62) will occur. (A speed command is required as the movement will be controlled with the G01 speed.)

(12) Do not command a G code for which a P code is used in the same block as G54.1 or G10L20. If a G code is commanded, a P code is used for a prior G command or the program error occurs (P33).

(13) If there are no specifications for the extended workpiece coordinate system selection, a program error (P35) will occur when the G54.1 command is executed. This error will also occur when one of P49 to P300 is commanded although the specifications allow up to the 48 sets.

Command	6 sets (Standard)	48 sets	96 sets	300 sets	
6 sets	○	○	○	○	○ : Movable × : P35 Setting value range over
48 sets	×	○	○	○	
96 sets	×	×	○	○	
300 sets	×	×	×	○	

(14) If there are no specifications for the extended workpiece coordinate system selection, the program error (P172) will occur when the G10 L20 command is executed.

(15) A new workpiece coordinate system P1 can be set by commanding G92 in the G54.1 P1 mode. However, the workpiece coordinate system of the other workpiece coordinate systems G54 to G59, G54.1, and P2 to P96 will move in parallel with it, and a new workpiece coordinate system will be set.

- (16) The offset amount of the extended workpiece coordinate system is assigned to system variables #7001 onwards.
 The system variables #7001 to #890n are available up to the valid number of sets. (You can use them for the 300-set specification also, but there are system variables corresponding to up to 96 sets only.)
 The system variables #101001 to #11595n are available when the 300-set specification is enabled.
 If you use the system variables #101001 to #11595n when the 300-set specification is disabled, the program error (P241) will occur.

CAUTION

If the workpiece coordinate system offset amount is changed during single block stop, the new setting will be valid from the next block.

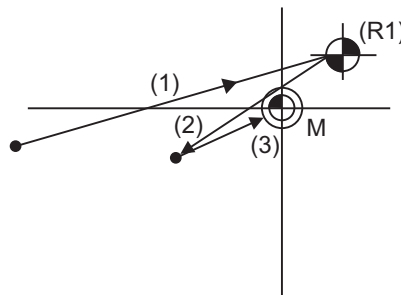
- (17) When the "#1151 Reset ini" parameter is OFF, the modal of G54.1 command will be retained even if the reset 1 is carried out.
 (18) The P address of the G54.1 command cannot be commanded alone even in G54.1 modal. Even if commanded, the designated extended workpiece coordinate system cannot be selected.
 (Example)
 P54.1 P5 ; Changed to P5 workpiece coordinate system.
 P3 ; Ignored.
 G92 X0 Y0 Z0 ; The current position becomes the zero point of P5 workpiece coordinate system.
 (19) When G92 is commanded in the extended workpiece coordinate system, the coordinate system will be sifted.



Program example

(Example 1)

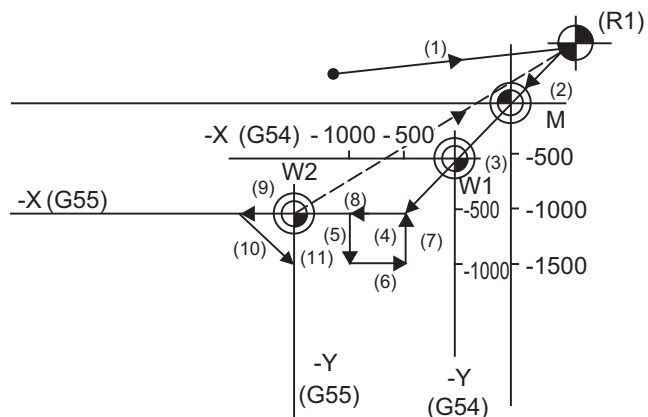
- (1) G28 X0 Y0 ;
- (2) G53 X-1000 Y-500 ;
- (3) G53 X0 Y0 ;



When the coordinate value of the 1st reference position (R1) is zero, the basic machine coordinate system zero point (M) and reference position (zero point) return position (#1) will coincide.

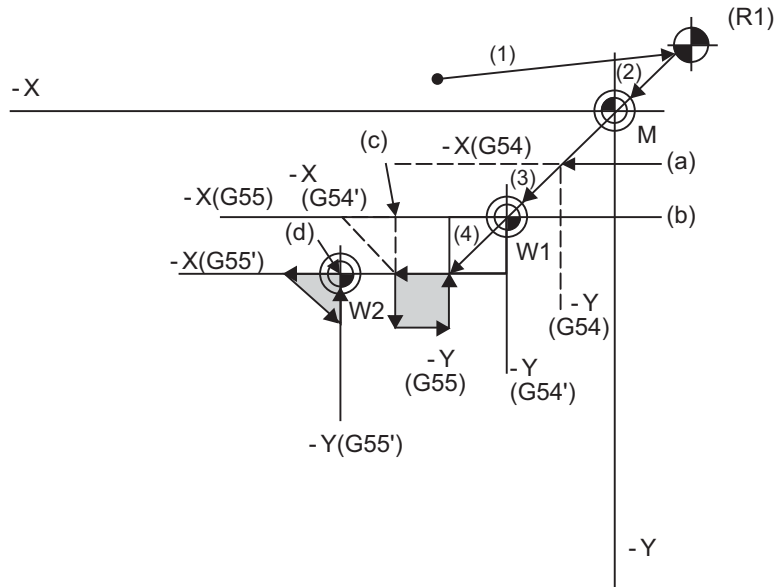
(Example 2)

- (1) G28 X0 Y0 ;
- (2) G90 G00 G53 X0 Y0 ;
- (3) G54 X-500 Y-500 ;
- (4) G01 G91 X-500 F100 ;
- (5) Y-500 ;
- (6) X+500 ;
- (7) Y+500 ;
- (8) G90 G00 G55 X0 Y0 ;
- (9) G01 X-500 F200 ;
- (10) X0 Y-500 ;
- (11) G90 G28 X0 Y0 ;



(Example 3) When workpiece coordinate system G54 (-500, -500) has deviated in Example 2. (It is assumed that (3) to (10) in Example 2 have been entered in subprogram 1111.)

- (1) G28 X0 Y0 ;
- (2) G90 G00 G53 X0 Y0 ; (This is not required when there is no G53 offset.)
- (3) G54 X-500 Y-500 ; Amount by which workpiece coordinate system deviates
- (4) G92 X0 Y0 ; New workpiece coordinate system is set.
- (5) M98 P1111 ;



- (R1) Reference position return position
- (a) Old G54 coordinate system
- (b) New G54 coordinate system
- (c) Old G55 coordinate system
- (d) New G55 coordinate system

Note

(1) The workpiece coordinate system will deviate each time steps (3) to (5) shown in the above figure are repeated. The reference position return (G28) command should therefore be issued upon completion of the program.

(Example 4) When six workpieces are placed on the same coordinate system of G54 to G59, and each is to be machined with the same machining.

(1) Setting of workpiece offset data

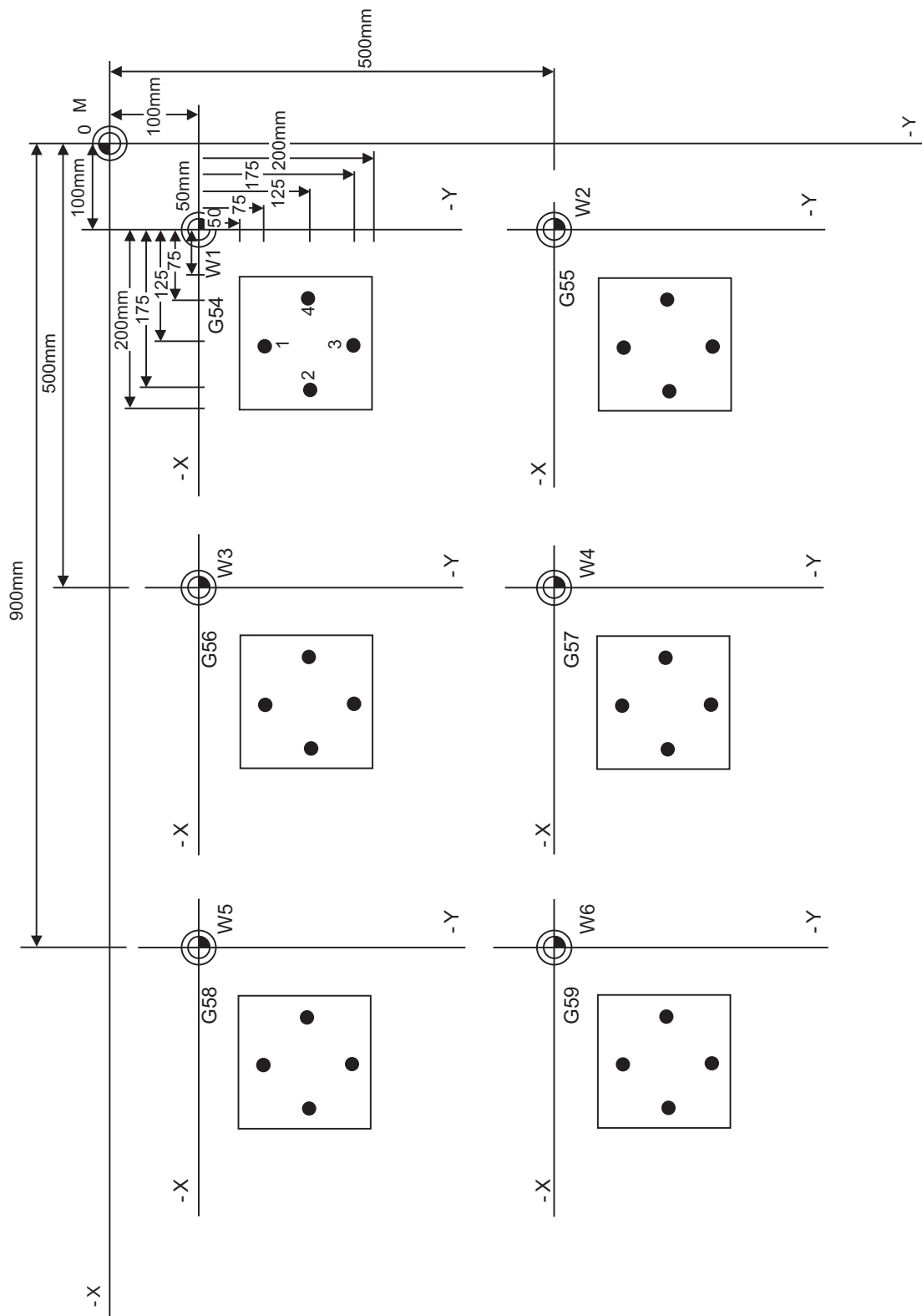
Workpiece 1 X = -100.000 Y = -100.000.....G54
Workpiece 2 X = -100.000 Y = -500.000.....G55
Workpiece 3 X = -500.000 Y = -100.000.....G56
Workpiece 4 X = -500.000 Y = -500.000.....G57
Workpiece 5 X = -900.000 Y = -100.000.....G58
Workpiece 6 X = -900.000 Y = -500.000.....G59

(2) Machining program (subprogram)

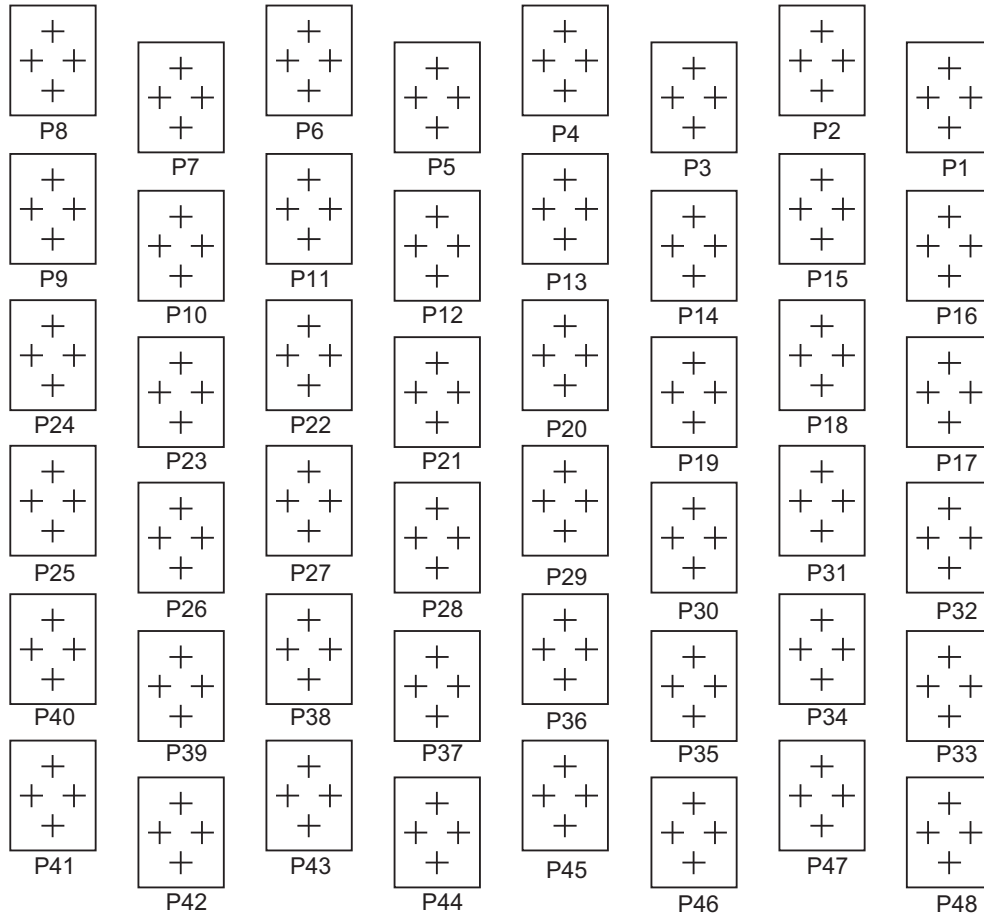
O100;	
N1 G90 G00 G43 X-50. Y-50. Z-100. H10 ;	Positioning
N2 G01 X-200. F50 ;	Surface cutting
Y-200. ;	Surface cutting
X-50. ;	Surface cutting
Y-50. ;	Surface cutting
N3 G28 X0 Y0 Z0 ;	
:	
N4 G98 G81 X-125. Y-75. Z-150. R-100. F40 ;	Drilling 1
X-175. Y-125. ;	Drilling 2
X-125. Y-175. ;	Drilling 3
X- 75. Y-125. ;	Drilling 4
G80 ;	
N5 G28 X0 Y0 Z0 ;	
:	
N6 G98 G84 X-125. Y-75. Z-150. R-100. F40 ;	Tapping 1
X-175. Y-125. ;	Tapping 2
X-125. Y-175. ;	Tapping 3
X- 75. Y-125. ;	Tapping 4
G80 ;	
M99 ;	

(3) Positioning program (main)

G28 X0 Y0 Z0 ;	At power ON
N1 G90 G54 M98 P100 ;	
N2 G55 M98 P100 ;	
N3 G57 M98 P100 ;	
N4 G56 M98 P100 ;	
N5 G58 M98 P100 ;	
N6 G59 M98 P100 ;	
N7 G28 X0 Y0 Z0 ;	
N8 M02 ;	
%	



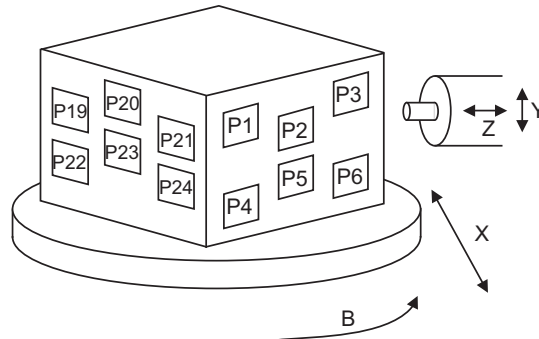
(Example 5) Program example when continuously using 48 sets of added workpiece coordinate system offsets.
 In this example, the offsets for each workpiece are set beforehand in P1 to P48 when 48 workpieces are fixed on a table, as shown in the drawing below.



O1000		O1001	
G28 XYZ ;	Reference position return	G43 X-10.Y-10.Z-100.H10.;	Contour
#100=1 ;	Initialize added workpiece coordinate system P No.	G01 X-30.;	
G90 ;	Absolute value mode	Y-30.;	
WHILE [#100LE48]D01	Repeat P No. to 48	X-10.;	
;		Y-10.;	
G54.1 P#100 ;	Set workpiece coordinate system	G00 G40 Z10.;	
M98 P1001 ;	Call sub-program	G98 G81 X-20.Y-15.Z-150.R5.F40; Drilling	
#100=#100+1 ;	P No. +1	X-25.Y-20.;	
END1 ;		X-20.Y-25.;	
G28 Z ;	Return to reference position	X-15.Y-20.;	
G28 XY ;		G80 ;	
M02 ;		M99 ;	

(Example 6) Program example when the added workpiece coordinate system offsets are transferred to the standard workpiece coordinate system offsets and used.

In this example, the workpiece coordinate system offsets for each workpiece are set beforehand in P1 to P24 when the workpiece is fixed on a rotating table, as shown in the drawing below.



O20000 (Main)

G28 XYZB ;	Reference position return
G90 ;	Absolute value mode
G00 B0 ;	Position table to face 1
G65 P2001 A1 ;	Load workpiece offsets
M98 P2002 ;	Drilling
G00 B90. ;	Position table to face 2
G65 P2001 A7 ;	
M98 P2002 ;	
G00 B180. ;	Position table to face 3
G65 P2001 A13 ;	
M98 P2002 ;	
G00 B270. ;	Position table to face 4
G65 P2001 A19 ;	
M98 P2002 ;	
G28 XYB ;	Return to reference position
M02 ;	
%	

O2001 (Transmission of workpiece offsets)

#2=5221 ;	Leading No. of workpiece coordinate system variables
#3=[#1-1]*20+7001 ;	Leading No. of added workpiece coordinate system variables
#5=0 ;	No. of sets counter clear
WHILE [#5 LT 6] DO1 ;	Check No. of sets
#6=#6+1 ;	Set transmission source 1st axis variable No.
#7=#7+1 ;	Set transmission destination 1st axis variable No.
#4=#4+1 ;	Clear No. of axes counter
WHILE [#4 LT 6] DO2 ;	Check No. of axes
#[#6]=#[#7] ;	Transmit variable data
#6=#6+1 ;	Set transmission source next axis
#7=#7+1 ;	Set transmission destination next axis
#4=#4+1 ;	No. of axes counter +1
END2 ;	
#2=#2+20 ;	Transmission source Set lead of next variable set.
#3=#3+20 ;	Transmission destination Set lead of next variable set.
#5=#5+1 ;	No. of sets counter +1
END1 ;	
M99 ;	
%	

O2002 (Drilling)

G54 M98 H100 ;	Drilling in G54 coordinate system
G55 M98 H100 ;	G55
G56 M98 H100 ;	G56
G57 M98 H100 ;	G57
G58 M98 H100 ;	G58
G59 M98 H100 ;	G59
G28 Z0 ;	
M99 ;	
N100 G98 G81 X-20. Y-15. Z-150. R5. F40 ;	Fixed cycle for drilling
X-25. Y-20. ;	
X-20. Y-25. ;	
X-15. Y-20. ;	
G80 ;	
G28 Z ;	
M99 ;	
%	

20.7 Workpiece Coordinate System Preset; G92.1



Function and purpose

This function presets the workpiece coordinate system shifted with the program command during manual operation to the workpiece coordinate system offset from the machine zero point by the workpiece coordinate offset amount by the program command (G92.1).

The workpiece coordinate system, which is set when the following type of operation or program command is executed, will be shifted from the machine coordinate system.

- When manual interrupt is executed while manual absolute is OFF
- When movement command is issued in machine lock state
- When axis is moved with handle interrupt
- When operation is carried out with mirror image
- When local coordinate system is set with G52
- Shifting the workpiece coordinate system with G92

This function presets the shifted workpiece coordinate system to the workpiece coordinate system offset from the machine zero point by the workpiece coordinate offset amount. This takes place in the same manner as manual reference position return. Whether to preset the relative coordinate depends on the MTB specifications (parameter "#1228 aux12/bit6").



Command format

G92.1 X0. Y0. Z0. α0 ;

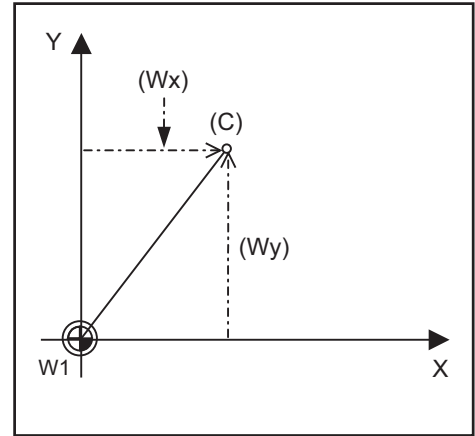
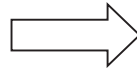
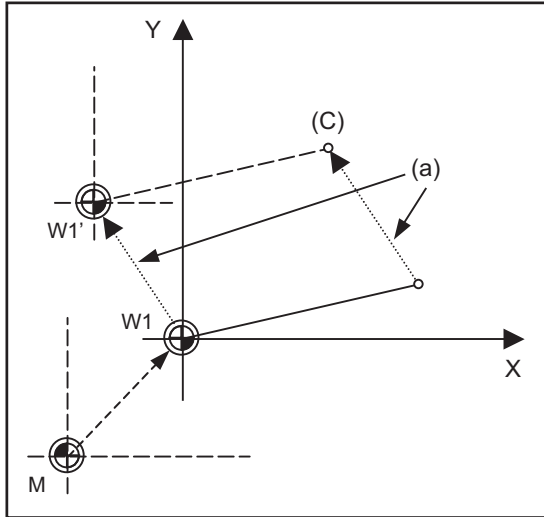
α0	Additional axis
----	-----------------

- (1) Command the address of the axis to be preset. The axis will not be preset unless commanded.
- (2) A program error (P35) will occur if a value other than "0" is commanded.
- (3) Command G92.1 in an independent block.
- (4) Whether to conduct an error check when the coordinate system preset command (G92.1) is independently issued depends on the MTB specifications (parameter "#1242 set14/bit1").



Detailed description

(1) When manual operation is carried out when manual absolute is set to OFF, or if the axis is moved with handle interrupt.



(a) Manual movement amount

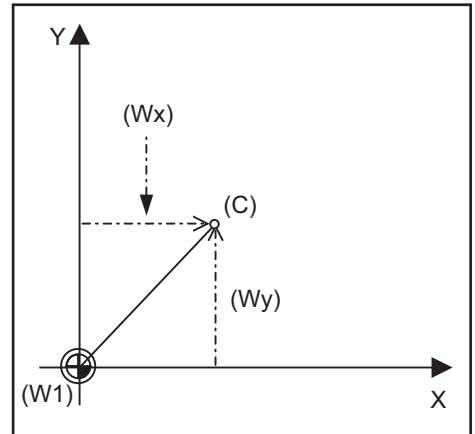
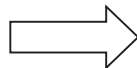
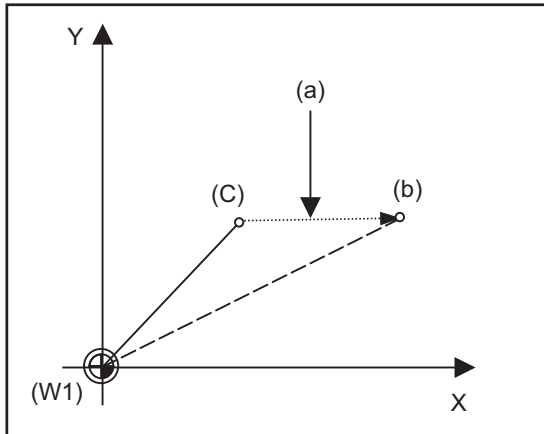
(C) Current position

(Wx) (Wy) Workpiece coordinate (x, y) after preset

If manual operation is carried out when manual absolute is set to OFF, or if the axis is moved with handle interrupt, the workpiece coordinate system will be shifted by the manual movement amount.

This function returns the shifted workpiece coordinate zero point W1' to the original workpiece coordinate zero point W1, and sets the distance from W1 to the current position as the workpiece coordinate system's current position.

(2) When movement command is issued in machine lock state



(a) Movement amount during machine lock

(b) Workpiece coordinate system coordinate value

(Wx) (Wy) Workpiece coordinate (x, y) after preset

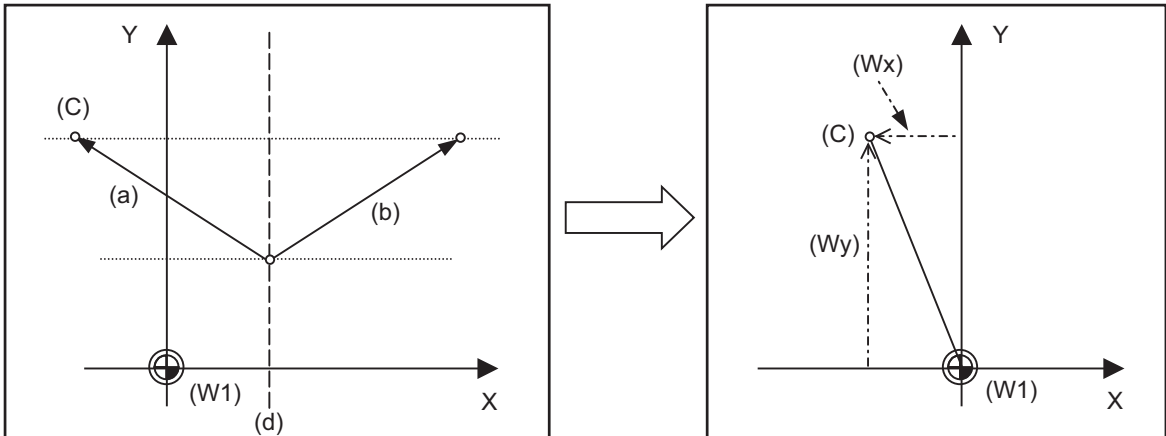
(C) Current position

(W1) Workpiece coordinate zero point

If the movement command is issued in the machine lock state, the current position will not move, and only the workpiece coordinates will move.

This function returns the moved workpiece coordinates to the original current position, and sets the distance from W1 to the current position as the workpiece coordinate system's current position.

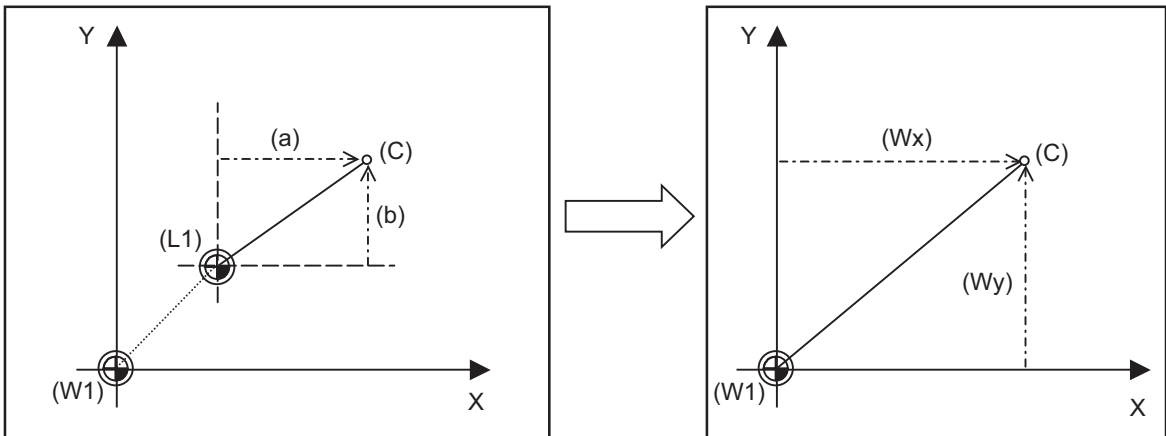
(3) When operation is carried out with mirror image



- (a) Actual operation
- (b) Program command
- (C) Current position
- (d) Mirror image center
- (Wx) (Wy) Workpiece coordinate (x, y) after preset
- (W1) Workpiece coordinate zero point

If operation is carried out with mirror image, only the NC internal coordinates are used as the program command coordinates. The other coordinates are the current position coordinates. This function sets the NC internal coordinates as the current position coordinates.

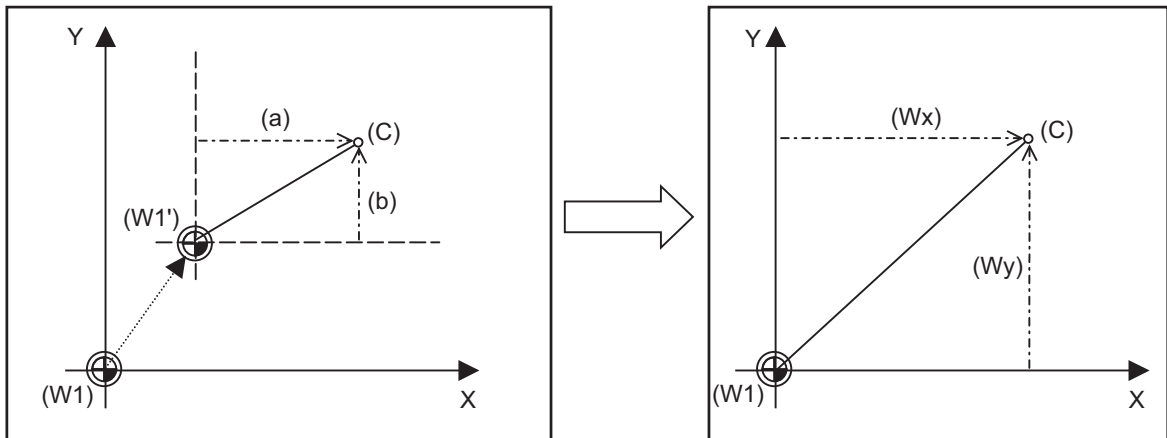
(4) Setting local coordinate system with G52



- (a) Local coordinates x
- (b) Local coordinates y
- (Wx) (Wy) Workpiece coordinate (x, y) after preset
- (C) Current position
- (L1) Local coordinate zero point
- (W1) Workpiece coordinate zero point

The local coordinate system is set with the G52 command, and the program commands, etc., are issued with the local coordinate system. With this function, the set local coordinate system is canceled, and the program commands, etc., use the workpiece coordinate system which has W1 as the zero point. The canceled local coordinate system is only the selected workpiece coordinate system.

(5) Shifting the workpiece coordinate system with G92



(a) Local coordinates x

(b) Local coordinates y

(Wx) (Wy) Workpiece coordinate (x, y) after preset

(C) Current position

(W1) Workpiece coordinate zero point

(W1') Workpiece zero point after G92 command

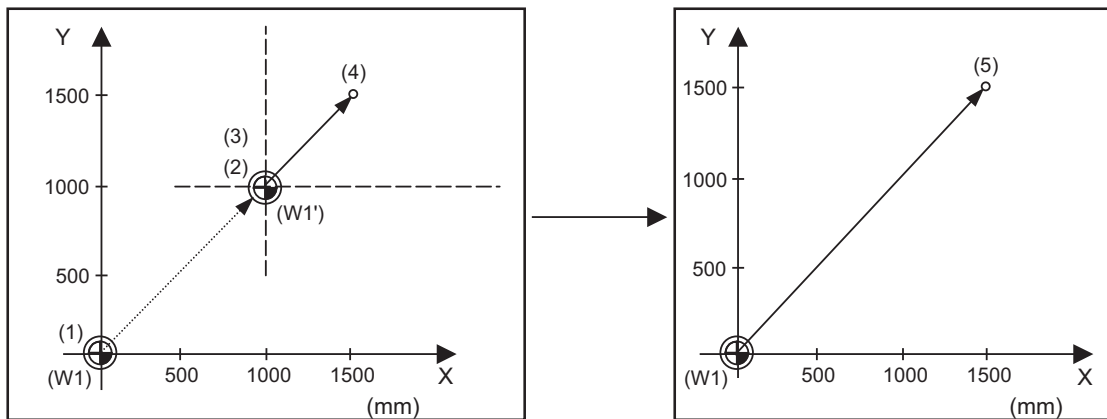
The workpiece coordinate system shifts with the G92 command, and the distance between W1' and the current position is set as the current position of the workpiece coordinate system.

This function returns the shifted workpiece coordinate zero point to W1, and sets the distance from W1 to the current position as the workpiece coordinate system's present position. This is valid for all workpiece coordinate systems.



Program example

The workpiece coordinate system shifted with G92 is preset with G92.1.



(W1) Workpiece coordinate zero point

(W1') Workpiece zero point after G92 command

(Example)

```
G28 X0 Y0 ; ... (1)
G00 G90 X1. Y1. ; ... (2)
G92 X0 Y0 ... (3)
G00 X500 Y500 ; ... (4)
G92.1 X0 Y0 ; ... (5)
```



Relationship with other functions

Tool No./Tool Compensation No. (T Code)/tool length compensation

If the error check is enabled when the workpiece coordinate system preset is independently commanded (*1), command all the tool compensation axes when commanding "G92.1" during the tool compensation. When commanding "G92.1" during the tool length compensation, designate the tool length compensation axis.

If those axes are not commanded, a program error (P29) will occur.

(*1) The setting depends on the MTB specifications (parameter "#1242 set14/bit1").

Tool nose radius compensation / Tool radius compensation

Cancel the tool nose radius compensation or the tool radius compensation, and command the workpiece coordinate system preset (G92.1). When the workpiece coordinate system preset (G92.1) is commanded during the tool nose radius compensation or the tool radius compensation, a program error (P29) will occur if none of the tool compensation axes are commanded.

3-dimensional coordinate conversion

If the workpiece coordinate system preset (G92.1) is commanded in 3-dimensional coordinate conversion, a program error (P921) will occur.

Other G code commands

If the workpiece coordinate system preset (G92.1) is commanded during the modal shown below, a program error (P34) will occur.

- (1) Scaling
- (2) Coordinate rotation by program
- (3) G command mirror image
- (4) Tool length compensation along the tool axis



Precautions

- (1) Cancel tool length compensation, tool nose radius compensation, tool radius compensation, and tool position offset when using this function. If this function is executed without canceling them, the workpiece coordinates will be at a position obtained by subtracting the workpiece coordinate offset amount from the machine value. Thus, the compensation vector will be temporarily canceled.
- (2) This function cannot be executed while the program is being resumed.

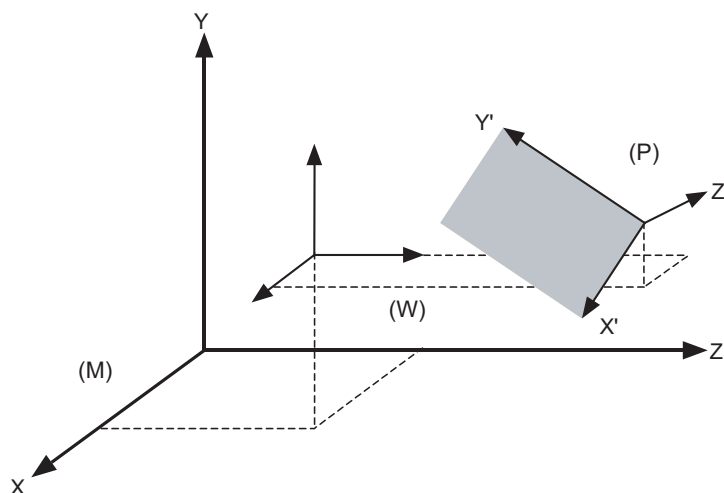
20.8 3-dimensional Coordinate Conversion; G68/G69



Function and purpose

With the 3-dimensional coordinate conversion function, a new coordinate system can be defined by shifting the zero point and rotating in respect to the X, Y and Z axes of the currently set workpiece coordinate system. By using this function, an arbitrary spatial plane can be defined, and machining on that plane can be carried out with normal program commands.

The validity of this function depends on the MTB specifications. Refer to the specifications of your machine tool.



(M) Machine coordinate system (P) G68 Program coordinate system (W) Workpiece coordinate system

When the G68 command is issued, the zero point is shifted by the command value (x, y, z) in respect to the current local coordinate system. A new G68 program coordinate system rotated by the designated rotation angle "r" in respect to the commanded rotation center direction (i, j, k) is created.

The local coordinate system is the same as the workpiece coordinate system when the local coordinate system offset is not ON.



Command format

3-dimensional coordinate conversion mode command

G68 X__Y__Z__I__J__K__R__ ;

X, Y, Z	Rotation center coordinates Designate with the absolute position of the local coordinate system.
I, J, K	Rotation center axis direction (1: Designated 0: Not designated) Note that "1" is designated for only one of the three axes. "0" is designated for the other two axes.
R	Rotation angle The counterclockwise direction looking at the rotation center from the rotation center axis direction is positive (+). The setting range is -360 to 360°, and the unit follows the minimum command unit.

3-dimensional coordinate conversion mode cancel command

G69



Detailed description

- (1) Command the rotation center coordinates with absolute values.
- (2) If the rotation center coordinates are omitted, the zero point of the currently set coordinate system will be the rotation center coordinates.
- (3) Designate values for all of I, J and K.
If any of the I, J, K is not designated, program coordinate rotation command will be valid.
- (4) Set "1" in only one of I, J and K, and set "0" for the other two.
The program error (P33) will occur if "1" is set in two or more.
- (5) The program error (P33) will also occur if "0" is set for all I, J and K.
- (6) When addresses I, J and K are not designated, this will be handled as the program coordinate rotation.
- (7) If a number other than "0" (including numbers of two or more digits) is designated for addresses I, J and K, this will be handled as "1".
If a blank is designated, this will be handled as "0".
- (8) If a G code that cannot be commanded in the 3-dimensional coordinate conversion modal is issued, the program error (P921) will occur.
When 3-dimensional coordinate conversion is commanded during the modal where 3-dimensional coordinate conversion cannot be carried out, the program error (P922) or program error (P923) will also occur. For details, refer to "Relationship with Other Functions".
- (9) Command G68 in an independent block. If another G code command is issued to the same block as for the G68 command, a program error (P923) will occur.
- (10) The 3-dimensional coordinate conversion command for the rotary axis will result in the program error (P32).
- (11) If a 3-dimensional coordinate conversion command is issued when there are no specifications for 3-dimensional coordinate conversion, the program error (P920) will occur.

Coordinate system

- (1) By issuing the 3-dimensional coordinate conversion command, a new coordinate system (G68 program coordinate system) will be created on the local coordinate system.
- (2) The coordinate system for the 3-dimensional coordinate conversion rotation center coordinates is the local coordinate system.
Thus, these coordinate systems are affected by the following coordinate system offset and coordinate system shift amount.
 - When local coordinate system is set with G52
 - G92 shift amount by G92 command
 - Coordinate system offset corresponding to the workpiece coordinate system selected with the command
 - External workpiece coordinate offset
 - Manual interruption amount or manual feed amount when manual ABS is OFF
- (3) If 3-dimensional coordinate conversion is commanded again during the 3-dimensional coordinate conversion modal, a G68 program coordinate system is created on the current G68 program coordinate system, and is used as a new G68 program coordinate system.
- (4) The local coordinate system cannot be created (G52) on the G68 program coordinate system. (If G52 is issued, the program error (P921) will occur.)
- (5) G68 program coordinate system can be reset either by G69 command or reset inputting. (Exclude reset 1 when "0" is set to the parameter "#1151 rsint")
- (6) Whether to run the manual operation during the 3-dimensional coordinate conversion modal in the G68 program coordinate system can be designated by switching the manual feed coordinate for 3-dimensional coordinate conversion.
- (7) Even if the 3-dimensional coordinate conversion modal state is canceled by reset, etc., the manual operation is possible in the G68 program coordinate system before the 3-dimensional coordinate conversion modal is canceled, until the G69 command is issued.
In the same way as during the 3-dimensional coordinate conversion modal, the target coordinate can be designated by switching the manual feed coordinate for 3-dimensional coordinate conversion.

Rotation angle

- (1) For the rotation angle, the counterclockwise direction looking at the rotation center from the plus direction of the rotation center axis is the plus (+) direction.
- (2) The rotation angle command unit with no decimal point depends on the parameter "#1078 Decpt2" (Decimal pnt type 2).
- (3) If the rotation angle is omitted, the rotation angle will be handled as 0°.

Rotation center coordinates

- (1) The G68 rotation center coordinate system is commanded with the local coordinate system (G68 program coordinate system during the 3-dimensional coordinate conversion modal).
- (2) The rotation center coordinate designation is handled as an absolute value command whether or not an absolute/incremental modal (G90/G91) is currently being executed.
- (3) If the rotation center coordinate is omitted, it will be handled as if the zero point of the current local coordinate (G68 program coordinate system during the 3-dimensional coordinate conversion modal) is designated for the omitted address's axis. (The same as when "0" is just set.)

G68 multiple commands

By commanding 3-dimensional coordinate conversion during the 3-dimensional coordinate conversion modal, two or more multiple commands can be issued.

- (1) The 3-dimensional coordinate conversion command in the 3-dimensional coordinate conversion modal is combined with the conversion in the modal.
- (2) If 3-dimensional coordinate conversion is overlapped during the 3-dimensional coordinate conversion modal, the overlapped 3-dimensional coordinate conversion will be created on the coordinate system (G68 program coordinate system) created with 3-dimensional coordinate conversion in the modal.

Thus, the rotary axis and coordinates must be designated with this G68 program coordinate system.

If creating a 90° rotated coordinate system for X axis and Y axis each, commands must be issued as in Example 2, not Example 1.

<Example 1>

G68 X0. Y0. Z0. I1 J0 K0 R90.;	X axis rotation 90°
G68 X0. Y0. Z0. I0 J1 K0 R90.;	Y axis rotation 90° (The Y axis designated here is the same as the Z axis in the original coordinate system.)

<Example 2>

G68 X0. Y0. Z0. I1 J0 K0 R90.;	X axis rotation 90°
G68 X0. Y0. Z0. I0 J0 K1 R-90.;	Z axis rotation 90° (The Z axis -90 rotation designated here is the same as the Y axis +90 rotation in the original coordinate system.)

Conversion method for 3-dimensional coordinate conversion

The coordinate values (Xp, Yp, Zp) in the newly set G68 program coordinate system and the coordinate values (Xm, Ym, Zm) in the reference workpiece coordinate system are converted as shown below.

First G68 command	
$[Xm, Ym, Zm, 1]=[Xp, Yp, Zp, 1]R1 T1$	(Forward row)
$[Xp, Yp, Zp, 1]=[Xm, Ym, Zm, 1](T1^{-1})(R1^{-1})$	(Reverse row)
Second G68 command	
$[Xm, Ym, Zm, 1]=[Xp, Yp, Zp, 1]R2 T2 R1 T1$	
$[Xp, Yp, Zp, 1]=[Xm, Ym, Zm, 1](T1^{-1})(R1^{-1})(T2^{-1})(R2^{-1})$	

R1, R2 : Rotation row calculated from first and second G68 parameter

T1, T2 : Movement row calculated from first and second G68 parameter

The conversion rows Rn and Tn (n = 1, 2) are as follow.

Rn conversion row

I designation (rotation around X axis)	J designation (rotation around Y axis)	K designation (rotation around Z axis)
$\begin{pmatrix} 1 & 0 & 0 & 0 \\ 0 & \cos R & \sin R & 0 \\ 0 & -\sin R & \cos R & 0 \\ 0 & 0 & 0 & 1 \end{pmatrix}$	$\begin{pmatrix} \cos R & 0 & -\sin R & 0 \\ 0 & 1 & 0 & 0 \\ \sin R & 0 & \cos R & 0 \\ 0 & 0 & 0 & 1 \end{pmatrix}$	$\begin{pmatrix} \cos R & \sin R & 0 & 0 \\ -\sin R & \cos R & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{pmatrix}$

Tn conversion row

$\begin{pmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ x & y & z & 1 \end{pmatrix}$	x, y, z: Rotation center coordinates (parallel movement amount) I, J, K: Rotation axis selection R: Rotation angle
--	--

Manual operation in G68 program coordinate system

Whether to run manual operations (jog feed, incremental feed, and manual handle feed) during the 3-dimensional coordinate conversion modal in the coordinate system (G68 program coordinate system) after the 3-dimensional coordinate conversion command was issued can be designated by switching the manual feed coordinate for 3-dimensional coordinate conversion.

When the axis stops during machining, operations such as a pulling operation by manual feed can be performed in the G68 program coordinate system.

(1) Coordinate switching enable conditions

A manual operation coordinate change by switching the manual feed coordinate for 3-dimensional coordinate conversion is available only when the output signal that enables the manual feed for 3-dimensional coordinate conversion is set to ON. (The operation of the PLC signal depends on the MTB specifications.)

The manual operation coordinate change by switching the manual feed coordinate for 3-dimensional coordinate conversion becomes valid after three basic axes have stopped. When the manual feed coordinate for 3-dimensional coordinate conversion is switched while even one of three basic axes is moving, a coordinate change is performed after three basic axes have stopped.

The output signal that enables the manual feed for 3-dimensional coordinate conversion is set to ON when all of the following conditions are satisfied.

- (a) One of the jog, incremental, or handle feed modes is selected.
- (b) G68 (3-dimensional coordinate conversion command) is commanded once. However, if the signal is canceled by the G69 command, it is not turned ON until G68 is commanded again.

(2) Operation example

The absolute coordinate positions are displayed in the G68 program coordinate system with a parameter. This depends on the MTB specifications (parameter "#1561 3Dcdc").

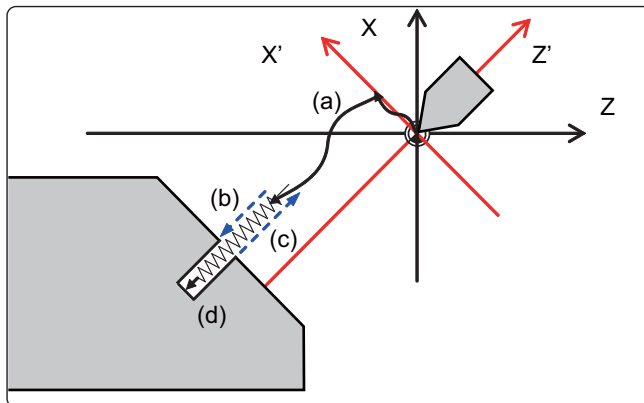
The machine zero point is used as the workpiece coordinate zero point. Also, the manual ABS is set to ON to return to the position commanded by the machining program with the absolute command after a manual interruption.

<Operation procedure>

- (1) Set mode selection to automatic operation (memory, MDI, tape, etc.) mode.
- (2) Execute the following machining program, then perform single block stop after the N03 block has been completed... (a)

N01 G68 X0. Z0. Y0. I0 J1 K0 R45. ;	Set the G68 program coordinate system (X'Y'Z') which has been rotated +45° in the Y axis direction around the (X0, Z0).
N02 G00 X1.; N03 G00 Z-10.;	Position the axis near the hole position in the G68 program coordinate system.
N04 G01 Z-20.;	Cutting in G68 program coordinate system.

- (3) Set the handle mode, then select the Z axis with the 1st handle. Then, check that the output signal that enables the manual feed for 3-dimensional coordinate conversion is set to ON.
- (4) Set to ON the signal that switches the manual feed coordinate for 3-dimensional coordinate conversion.
- (5) Move the axis by -5. with the handle in the Z' direction of the G68 program coordinate, then check the hole position... (b)
- (6) Move the axis by +7. with the handle in the Z' direction to retract the tool... (c)
- (7) After executing an automatic start, execute the N04 block... (d)



		Absolute coordinate	Machine coordinate
(a) Positioning	X	1.000	-6.364
	Z	-10.000	-7.778
(b) Hole position check	X	1.000	-9.900
	Z	-15.000	-11.314
(c) Retract	X	1.000	-4.950
	Z	-8.000	-6.364
(d) Cutting	X	1.000	-13.435
	Z	-20.000	-14.849

(3) Precautions

- (a) When the automatic and manual operation modes are selected simultaneously, the manual feed for 3-dimensional coordinate conversion cannot be commanded. However, manual operation is available only for the axis for which the manual automatic simultaneous valid axis signal is set to ON.
- (b) The speed limit is applied so that the speed distributed to the movement amount in the machine axis direction does not exceed the clamp speed of each axis.
- (c) If even one axis satisfies the external deceleration conditions, the speed limit is applied so that the movement speed in each axis direction does not exceed the external deceleration speed.
- (d) The manual feed operation for 3-dimensional coordinate conversion is not available in manual reference position return mode. If it is started, an operation error (M01 0060) will occur. To use the manual reference position return mode, set to OFF the signal that switches the manual feed coordinate for 3-dimensional coordinate conversion.
- (e) The manual feed operation for 3-dimensional coordinate conversion is not available in tool retract and return mode. If it is started, an operation error (M01 0060) will occur. To use the tool escape mode, set to OFF the signal that switches the manual feed coordinate for 3-dimensional coordinate conversion.
- (f) This function is not compatible with the manual tool length measurement function, workpiece position measurement function, and manual skip based on the manual feed for 3-dimensional coordinate conversion. If it is started, an operation error (M01 014) will occur. While the manual tool length measurement function or workpiece position measurement function is active or when manual skip is valid, set to OFF the signal that switches the manual feed coordinate for 3-dimensional coordinate conversion.
- (g) When the manual automatic simultaneous valid axis signal is set to ON for any of three basic axes, operation is performed in the same way as when the manual automatic simultaneous valid axis signal for three basic axes is set to ON.
- (h) When the manual machine lock signal is set to ON for any of three basic axes, operation is performed in the same way as when the manual machine lock signal for three basic axes is set to ON.
- (i) When a factor such as manual interlock that triggers the stop of the axis under manual movement occurs at any of three basic axes, execute deceleration stop on the three basic axes.
- (j) When the 3-dimensional coordinate conversion modal state is canceled by reset, etc., the manual feed for 3-dimensional coordinate conversion is possible; however, the absolute coordinate position display function and other functions conform to the 3-dimensional coordinate conversion modal state and parameter setting. The coordinate cannot be changed by switching the manual feed coordinate for 3-dimensional coordinate conversion.
- (k) If the G69 command exists up to the block to be restart-searched at program restart, it also cancels the state that enables the manual feed in 3-dimensional coordinate conversion as the G69 command does.



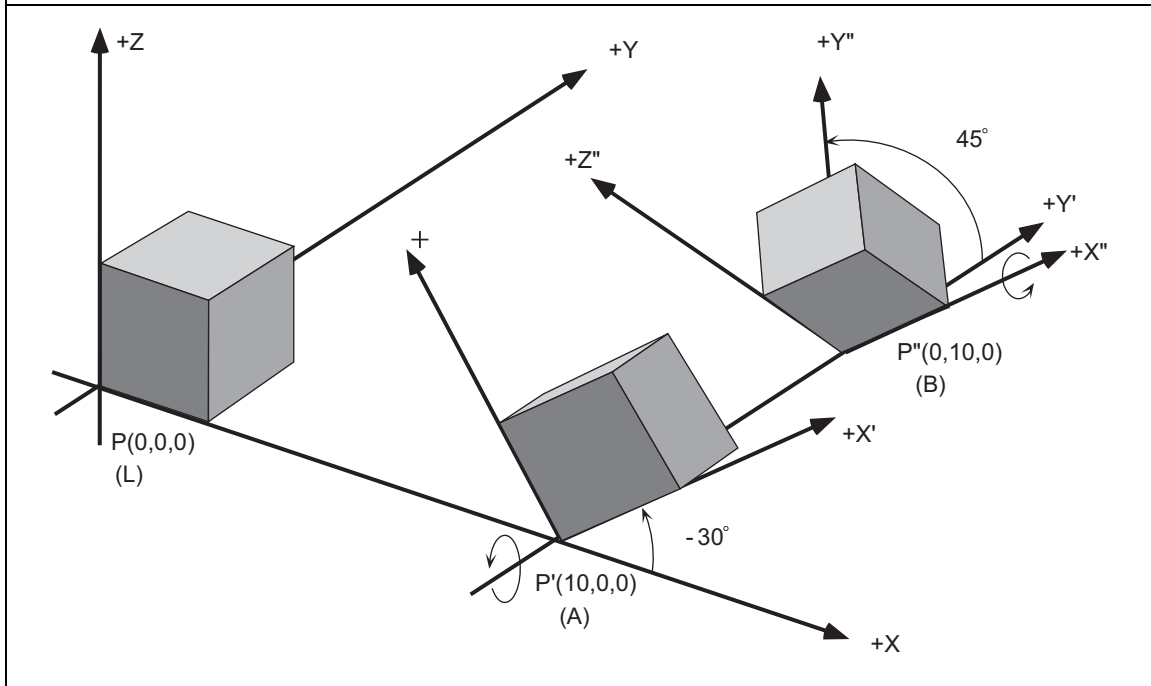
Program example

Program example 1

```

N1 G68 X10.Y0. Z0. I0 J1 K0 R-30.;
N2 G68 X0. Y10. Z0. I1 J0 K0 R45.;
:
N3 G69;

```



- (1) With N1, the zero point is shifted by $(x, y, z) = (10, 0, 0)$ in respect to the currently set local coordinate system (L). The new G68 program coordinate system (A) rotated -30° in the counterclockwise direction using the Y axis as the center, is set.
- (2) With N2, the zero point is shifted by $(x, y, z) = (0, 10, 0)$ in respect to the newly set G68 program coordinate system (A). The new G68 program coordinate system (B) rotated $+45^\circ$ in the counterclockwise direction using the X axis as the center, is set.
- (3) With N3, the G68 program coordinate systems that have been set are all canceled, and the state prior to where the first G68 has been commanded is resumed.

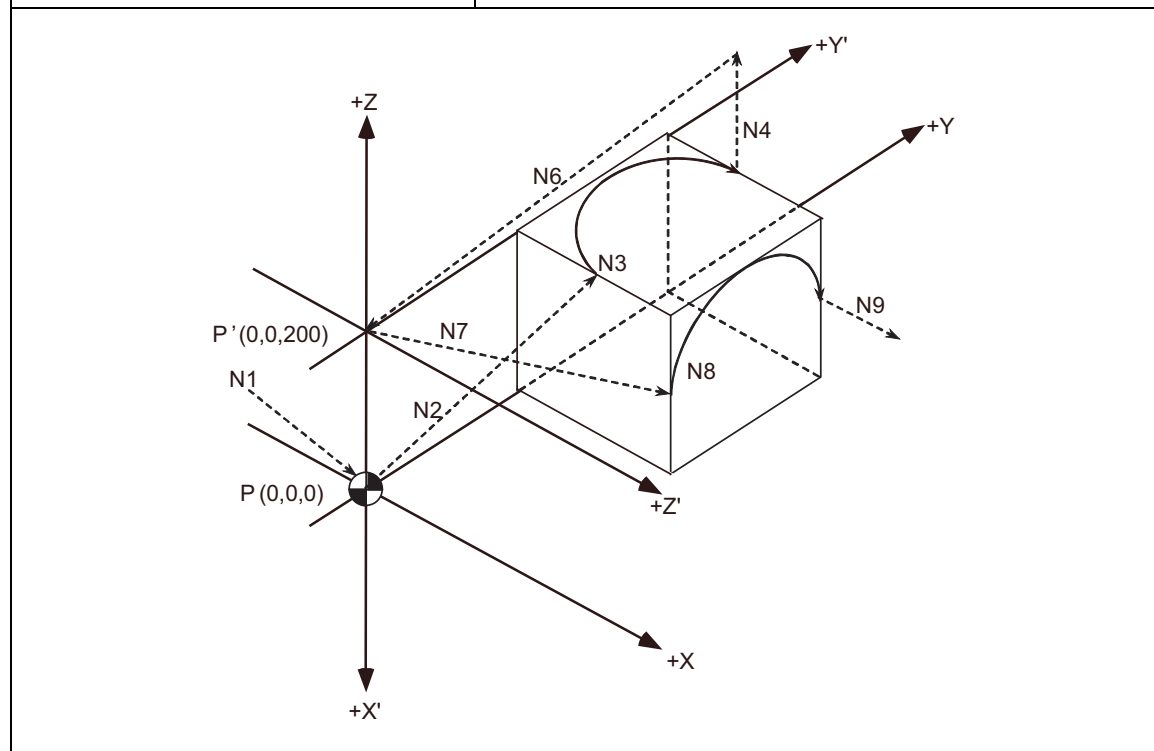
Program example 2

This is only a sample program to explain about the operations. (To actually proceed with the machining by using this program, the dedicated tools and the tool change functions are required.)

(1) Example of machining program using arc cutting

In the following program example, the arc cutting (N3 block) carried out on the top of the workpiece is also carried out on the side of the workpiece. By using 3-dimensional coordinate conversion, the side can be cut using the same process (N8 block).

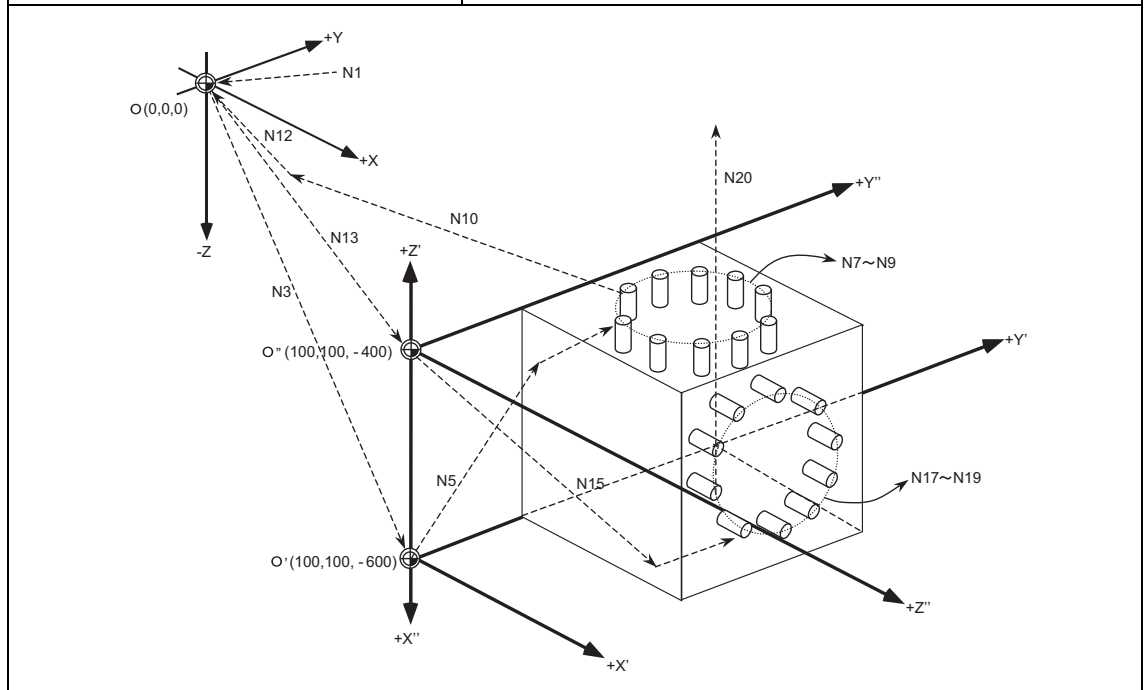
N01 G17 G90 G00 X0 Y0 Z0;	Position to the workpiece zero point P.
N02 G00 X100. Y200. Z200.;	Move to (100, 200, 200) with rapid traverse.
N03 G02 X100. Y400. J100. F1000;	Carry out arc cutting on workpiece top.
N04 G00 Z300.;	Escape +100 in +Z direction at rapid traverse rate.
N05 G68 X0 Y0 Z200. I0 J1 K0 R90.;	Set the G68 program coordinate system (X'Y'Z') which has been rotated +90° in the Y axis direction around the (0,0,200).
N06 G17 G90 G00 X0 Y0 Z0;	Position to the new program zero point P'.
N07 G00 X100. Y200. Z200.;	Move to G68 program coordinate system (100, 200, 200) and workpiece coordinate system (200, 200, 100) at rapid traverse rate.
N08 G02 X100. Y400. J100. F1000;	Carry out arc cutting on workpiece side.
N09 G00 Z300.;	Move +100 in +Z' direction of G68 program coordinate system at rapid traverse rate.
N10 G69;	
N11 M02;	



(2) Example of machining program using fixed cycle

In the following program, the bolt hole circle (N08 block) executed on the top of the workpiece is also carried out on the side of the workpiece. By using 3-dimensional coordinate conversion, the side can be cut using the same process (N18 block).

N01 G90 G00 X0 Y0 Z0;	Position to the workpiece coordinate system's 1st workpiece zero point.
N02 F2000;	
N03 G00 X100. Y100. Z-600.;	Move to (100, 100, -600) with rapid traverse.
N04 G52 X100. Y100. Z-600.;	Set the local coordinate system (X'Y'Z') to the (100, 100, -600).
N05 G00 X100. Y10. Z 200.;	Move to (100, 10, 200) position in the local coordinate system at a rapid traverse.
N06 G91;	Incremental value command
N07 G81 Z-10. R5. L0 F2000;	Drilling cancel
N08 G34 X100. Y200. I90. J270. K10.;	Bolt hole circle
N09 G80;	Drilling cancel
N10 G91 G00 X-200.;	Move -200 from machining end point in X axis direction at rapid traverse rate.
N11 G90 G52 X0 Y0 Z0;	Cancel local coordinate system.
N12 G90 G00 X0 Y0 Z0;	Position to workpiece zero point.
N13 G00 X100. Y100. Z-400.;	Move to (100, 100, -400) with rapid traverse.
N14 G68 X100. Y100. Z-400. I0 J1 K0 R90.;	Set G68 program coordinate system (X'',Y'',Z'') rotated +90° in Y axis direction using (100, 100, -400) position as center.
N15 G00 X100.Y10. Z200.;	Move to (100, 10, 200) position in the G68 program coordinate system at a rapid traverse rate.
N16 G91;	Incremental value command
N17 G81 Z-10. R5. L0 F200;	Drilling cancel
N18 G34 X100.Y200. I90. J270. K10.;	Bolt hole circle
N19 G80;	Drilling cancel
N20 G91 G00 X-200.;	Move -200 from machining end point in X axis direction at rapid traverse rate.
N21 G69;	Cancel 3-dimensional coordinate conversion modal.
N22 M02;	End program.





Relationship with other functions

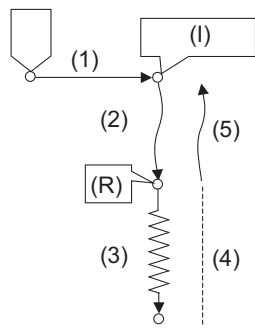
- (1) Circular interpolation
Circular interpolation in the 3-dimensional coordinate conversion modal operates according to the coordinate value resulting from 3-dimensional coordinate conversion. With G17, G18 and G19 commands, circular interpolation functions normally for all the planes in which 3-dimensional coordinate conversion has been executed.
- (2) Fine spline
Designation of a spline axis should be done to the movement axis after the 3-dimensional coordinate conversion. When a movement occurs to the axis in which spline cannot be designated, spline will be in the pause status.
- (3) Reference position check
The 3-dimensional coordinate conversion is applied for the positioning commanded by G27 in the 3-dimensional coordinate conversion modal.
- (4) Reference point return
The 3-dimensional coordinate conversion is applied for the mid-point commanded by G28 and G30 in the 3-dimensional coordinate conversion modal. However, reference position return will be carried out without the 3-dimensional coordinate conversion.
- (5) Tool change position return
3-dimensional coordinate conversion is not carried out for the tool change position even if a command from G30.1 to G30.6 is issued in the 3-dimensional coordinate conversion modal. The returning order and position will be on the machine coordinate system.
- (6) Tool compensation
When executing the tool length/radius/position compensation in the 3-dimensional coordinate conversion modal, the 3-dimensional coordinate conversion is carried out after the compensation value has been applied.
- (7) Machine coordinate system selection
Coordinate conversion will not be carried out for the machine coordinate system even if G53 command is issued in the 3-dimensional coordinate conversion modal.
- (8) Mirror image
When issuing the mirror image command in the 3-dimensional coordinate conversion modal, as well as when executing the 3-dimensional coordinate conversion in the mirror image modal, 3-dimensional coordinate conversion will be executed for the coordinate value, which is calculated by the mirror image.
- (9) User macro
When a user macro call command is issued in the 3-dimensional coordinate conversion modal, the 3-dimensional coordinate conversion will be valid after the macro execution.
- (10) Fixed cycle for drilling
The fixed cycle in the 3-dimensional coordinate conversion can be executed in an oblique direction for the orthogonal coordinate system. In the same manner, synchronous tapping cycle can also be executed. However, the fixed cycle hole drilling rapid traverse speed during the 3-dimensional coordinate conversion modal is switched as shown below by the settings of the parameters "#15663 DselctDrillaxMode" and "#1564 3Dspd". (This depends on the MTB specifications.)

Fixed cycle rapid traverse speed during 3-dimensional coordinate conversion

#1566	0 (Rapid traverse mode)	1 (Cutting mode)	
#1564	-	0	1 to 1000000
Rapid traverse speed	The "#2001 rapid" value for each machine axis is converted to the speed in the composite movement direction, and the slowest speed is applied.	The "#2002 clamp" value for each machine axis is converted to the speed in the composite movement direction, and the slowest speed is applied.	The value (mm/min) set to "#1564 3Dspd" is applied.

<Note>

- ♦The speed of operation 1 in the table above conforms to the "#2001 rapid" value regardless of the parameter setting above.
- ♦When a macro interruption, MDI interruption, or PLC interruption is carried out in the fixed cycle during 3-dimensional coordinate conversion, the rapid traverse speed in the interrupt machining program conforms to the "#2001 rapid" value regardless of the parameter setting above.



[Operation]

- (1) Position to the initial position at the rapid traverse rate.
- (2) Position to the R point at the rapid traverse rate.
- (3) Hole machining is conducted by cutting feed.
- (4) Escape to the R point.
Cutting feed or rapid traverse depending on the fixed cycle mode.
- (5) The tool is returned to the initial point at the rapid traverse rate.

(I) Initial point

(R) R point

Parameter "#1566" becomes valid when each fixed cycle is set to rapid traverse mode. The operation numbers in the table below correspond to those in the figure above.

Relationships between fixed cycles and parameter "#1566"

G code	[Operation] (2)	[Operation] (3)	[Operation] (4)	[Operation] (5)
G73	o	x	o	o
G74	o	x	x	o
G75	o	x	o	o
G76	o	x	o	o
G81	o	x	o	o
G82	o	x	o	o
G83	o	x	o	o
G84	o	x	x	o
G85	o	x	x	o
G86	o	x	o	o
G87	o	o	x	o
G88	o	x	o	o
G89	o	x	x	o

o: "#1566" is valid (rapid traverse).

x: "#1566" is irrelevant (cutting feed).

For G87, the movement completion position in operation 3 is set to R point.

Parameter "#1566" is also valid for rapid traverse operation at G76 or G87 shift.

Parameter "#1566" is also valid for G73 or G83 return operation.

(11) Synchronous tapping cycle

The synchronous tapping cycle in the 3D coordinate conversion can be executed in an oblique direction for the orthogonal coordinate system.

The Synchronous tapping cycle in the 3-dimensional coordinate conversion modal will not function even if "#1223 aux07/bit3" (synchronous tapping in-position check expansion valid) is valid. Set the synchronous tapping cycle to invalid. (This parameter setting depends on the MTB specifications.)

The rapid traverse rate in synchronous tapping cycle always follows the value of #2001(rapid traverse rate) during the 3D coordinate conversion mode, regardless of the values of #1566(switch drill axis's mode from rapid traverse during 3D coordinate conversion) and #1564(hole drilling cycle during 3D coordinate conversion).

If any of the orthogonal axes of all the active part systems is under machine lock during 3-dimensional coordinate conversion, normal synchronous tapping is applied even though the high-speed synchronous tapping specification is enabled.

(12) Geometric command

Geometric command can be issued in the 3-dimensional coordinate conversion modal. However, if the geometric command is issued in the same block as in the 3-dimensional coordinate conversion command (G68, G69), "the program error (P32) will occur.

(13) Init const sur spd

When the 3-dimensional coordinate conversion command is issued while the parameter initial constant surface speed is valid, the program error (P922) will occur. This is the same consequence as in the case where the 3-dimensional coordinate conversion command is issued in the constant surface speed (G96) modal.

(14) Machine lock

The machine lock in the 3-dimensional coordinate conversion modal will be valid for the movement axis for the coordinate value after executing the 3-dimensional coordinate conversion.

(15) Interlock

The interlock in the 3-dimensional coordinate conversion modal will be valid for the movement axis for the coordinate value after executing the 3-dimensional coordinate conversion.

(16) Coordinate read variable

When reading the workpiece coordinate system/skip coordinate system during the 3-dimensional coordinate conversion modal, local coordinate system and G68 program coordinate system can be switched with the parameter "#1563 3Dcdrc".

(17) Workpiece coordinate display

Whether to display the workpiece coordinate system position in the 3-dimensional coordinate conversion modal, in the workpiece coordinate system or in the G68 program coordinate system can be switched with the parameter "#1561 3Dcdc". In the same manner, absolute value can be displayed on the special display.

<Note>

- ♦1μm of display deviation may occur during the 3-dimensional coordinate conversion; however, this is normal.

(18) Remaining command display

Whether to display the remaining commands in the 3-dimensional coordinate conversion modal, in the workpiece coordinate system or in the G68 program coordinate system can be switched with the parameter "#1562 3Dremc".

<Note>

- ♦1μm of display deviation may occur during the 3-dimensional coordinate conversion; however, this is normal.

(19) Graphic check

Linear tracing is applied to circular interpolation (including corner R) during 3-dimensional coordinate conversion in graphic check mode.

(20) Manual operation in G68 program coordinate system

Refer to "Manual Operation in G68 Program Coordinate System" in Detailed description.

(21) Others

G41, G42, and the fixed cycle commands G73 to G89 have to be nested inside the G68/G69 commands.

For the block next to G68, a movement command in the G90 (Absolute value command) mode has to be issued.

(Example)

```

G68 X50. Y100. Z150. I1 J0 K0 R60. ;
G90 G00 X0 Y0 Z0 ;
G41 D01 ;
      . . . . .
G40 ;
G69 ;

```

- ♦G00 command during 3-dimensional coordinate conversion modal is the interpolation type regardless of settings of the basic parameter "#1086 G0Intp (G00 non-interpolation)"
- ♦Origin zero cannot be executed during the 3-dimensional coordinate conversion modal.
- ♦When in a G68/G69 block during tool compensation, the program position counter indicates a position shifted by the tool length offset.

Relation with other G codes

Pxxx in the list indicates the program error Nos.

Column A: Operation to be carried out when the G command in the list is issued during 3-dimensional coordinate conversion

Column B: Operation to be carried out when 3-dimensional coordinate conversion is commanded while the G command modal in the list is established

Column C: Operation to be carried out when the G command in the list and 3-dimensional coordinate conversion are commanded for the same block

G command	Function	A	B	C
G00	Positioning	○	○	P923
G01	Linear interpolation	○	○	P923
G02	Circular interpolation CW	○	○	P923
	Helical interpolation CW	P921	P922	P923
G03	Circular interpolation CCW	○	○	P923
	Helical interpolation CCW	P921	P922	P923
G02.3	Exponential interpolation CW	P921	P922	P923
G02.4	3-dimensional circular interpolation CW	P921	P922	P923
G03.3	Exponential interpolation CCW	P921	P922	P923
G03.4	3-dimensional circular interpolation CCW	P921	P922	P923
G04	Dwell	○	-	G04 valid, G68 ignored
G05 P0	High-speed machining mode cancel	○	-	P923
G05 P1, 2	High-speed machining mode I, II	P34	P34	P923
G05 P10000	High-speed high-accuracy control II	P34	P34	P923
G05.1 Q0	High-speed machining mode/High-speed high-accuracy control cancel	○	○	P923
G05.1 Q1	High-speed high-accuracy control I	○	○	P923
G05.1 Q2	Fine spline	P34	P34	P923
G07.1/ G107	Cylindrical interpolation	P921	P481	P923
G09	Exact stop check	○	-	P923
G10	Parameter input by program	○	P421	P923
	Program tool compensation input	○	-	G10 valid, G68 ignored
G11	Parameter input by program cancel	○	-	P923
G12	Circular cutting CW	○	-	P923
G12.1	Polar coordinate interpolation	P921	P481	P923
G13	Circular cutting CCW	○	-	P923
G13.1	Polar coordinate interpolation cancel	○	-	P923

20 Coordinate System Setting Functions

G command	Function	A	B	C
G15	Polar coordinate command cancel	○	-	P923
G16	Polar coordinate command	○	○	P923
G17	Plane selection X-Y	○	○	○
G18	Plane selection Z-X	○	○	○
G19	Plane selection Y-Z	○	○	○
G20	Inch command	○	○	○
G21	Metric command	○	○	○
G27	Reference position check	○	-	G27 valid, G68 ignored
G28	Reference position return	○	-	G28 valid, G68 ignored
G29	Start position return	○	-	G29 valid, G68 ignored
G30	2nd to 4th reference position return	○	-	G30 valid, G68 ignored
G30.1	Tool change position return 1	○	-	G30.1 valid, G68 ignored
G30.2	Tool change position return 2	○	-	G30.2 valid, G68 ignored
G30.3	Tool change position return 3	○	-	G30.3 valid, G68 ignored
G30.4	Tool change position return 4	○	-	G30.4 valid, G68 ignored
G30.5	Tool change position return 5	○	-	G30.5 valid, G68 ignored
G30.6	Tool change position return 6	○	-	G30.6 valid, G68 ignored
G31	Skip	○	-	P923
G31.1	Multi-step skip 1	○	-	P923
G31.2	Multi-step skip 2	○	-	P923
G31.3	Multi-step skip 3	○	-	P923
G33	Thread cutting	P921	P922	P923
G34	Special fixed cycle (bolt hole circle)	○	-	P923
G35	Special fixed cycle (line at angle)	○	-	P923
G36	Special fixed cycle (arc)	○	-	P923
G37.1	Special fixed cycle (grid)	○	-	P923
G37	Automatic tool length measurement	P921	-	G37 valid, G68 ignored
G38	Tool radius compensation (vector designation)	○	-	P923
G39	Tool radius compensation (corner arc)	○	-	P923
G40	Tool radius compensation cancel	○	-	○
G41	Tool radius compensation	○	P922	P923
	3-dimensional tool radius compensation	○	P922	P923

G command	Function	A	B	C
G42	Tool radius compensation	○	P922	P923
	3-dimensional tool radius compensation	○	P922	P923
G40.1/ G150	Normal line control cancel	P921	-	P923
G41.1/ G151	Normal line control Left	P921	P922	P923
G42.1/ G152	Normal line control Right	P921	P922	P923
G43	Tool length compensation (+)	○	○	P923
G44	Tool length compensation (-)	○	○	P923
G45	Tool position compensation increase	○	-	P923
G46	Tool position compensation decrease	○	-	P923
G47	Tool position compensation 2-fold increase	○	-	P923
G48	Tool position compensation 2-fold decrease	○	-	P923
G49	Tool length compensation cancel	○	-	P923
G43.1	Tool length compensation along the tool axis	P927	P931	P923
G43.4	Tool center point control type1 ON	P941	P922	P923
G43.5	Tool center point control type2 ON	P941	P922	P923
G50	Scaling cancel	P921	-	P923
G51	Scaling ON	P921	○	P923
G50.1	Mirror image cancel	○	-	P923
G51.1	Mirror image ON	○	○	P923
G52	Local coordinate system setting	P921	-	G52 valid, G68 ignored
G53	Machine coordinate system setting	○	-	G53 valid, G68 ignored
G54	Workpiece coordinate system 1 selection	P921	○	P923
G55	Workpiece coordinate system 2 selection	P921	○	P923
G56	Workpiece coordinate system 3 selection	P921	○	P923
G57	Workpiece coordinate system 4 selection	P921	○	P923
G58	Workpiece coordinate system 5 selection	P921	○	P923
G59	Workpiece coordinate system 6 selection	P921	○	P923
G54.1	Extended workpiece coordinate system selection	P921	○	P923

20 Coordinate System Setting Functions

G command	Function	A	B	C
G60	Unidirectional positioning	P921	-	G60 valid, G68 ignored
	Unidirectional positioning (Modal designation)	P921	P922	P923
G61	Exact stop check mode	○	○	P923
G61.1	High-accuracy control	○	○	P923
G62	Automatic corner override	○	○	P923
G63	Tapping mode	P921	P922	P923
G64	Cutting mode	○	○	○
G65	User macro simple call	○	-	Update modal only (Coordinate rotation by program)
G66	User macro Modal call A	○	○	Update modal only (Coordinate rotation by program)
G66.1	User macro modal call B	○	Update modal only (Coordinate rotation by program)	Update modal only (Coordinate rotation by program)
G67	User macro modal call cancel	○	○	Update modal only after macro (Coordinate rotation by program)
G68	Coordinate rotation by program ON	P921	P922	-
	3-dimensional coordinate conversion ON	○	○	-
G69	Coordinate rotation by program cancel	○ (3-dimensional coordinate conversion cancel)	-	-
	3-dimensional coordinate conversion cancel	○	-	-
G73	Fixed cycle (step)	○	P922	P923
G74	Fixed cycle (reverse tapping) (including synchronous tapping)	○	P922	P923
G76	Fixed cycle (fine boring)	○	P922	P923
G80	Fixed cycle cancel	○	-	P923
G81	Fixed cycle (drill/spot drill)	○	P922	P923
G82	Fixed cycle (drill/counter boring)	○	P922	P923
G83	Fixed cycle (deep drilling)	○	P922	P923
G84	Fixed cycle (tapping) (including synchronous tapping)	○	P922	P923
G85	Fixed cycle (boring)	○	P922	P923
G86	Fixed cycle (boring)	○	P922	P923
G87	Fixed cycle (back boring)	○	P922	P923
G88	Fixed cycle (boring)	○	P922	P923
G89	Fixed cycle (boring)	○	P922	P923
G90	Absolute value command	○	○	○
G91	Incremental value command	○	○	○

G command	Function	A	B	C
G92	Coordinate system setting	P921	-	P923
G94	Asynchronous feed (feed per minute)	○	○	○
G95	Synchronous feed (feed per revolution)	○	○	○
G96	Constant surface speed control ON	P921	P922	P923
G97	Constant surface speed control OFF	P921	-	P923
G98	Fixed cycle (Initial level return)	○	○	○
G99	Fixed cycle (R point level return)	○	○	○

Note

(1) None of the G codes not listed above can be used.



Precautions

Precautions related to arc command

If the first command after the 3-dimensional coordinate conversion command is an arc shape, and the center of the arc did not change before and after the 3-dimensional coordinate conversion, an arc is drawn. However, an error will occur in the following cases:

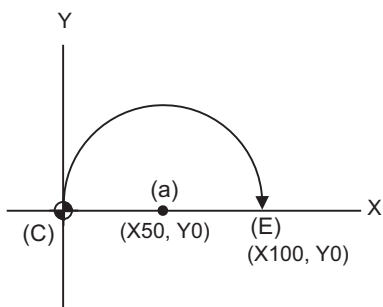
- (1) For the arc in which the arc center is specified with I and J, if the center coordinate has deviated by 3-dimensional coordinate conversion, a program error (P70 Major arc end position deviation) will occur.

G90 G28 X0 Y0 Z0 ;

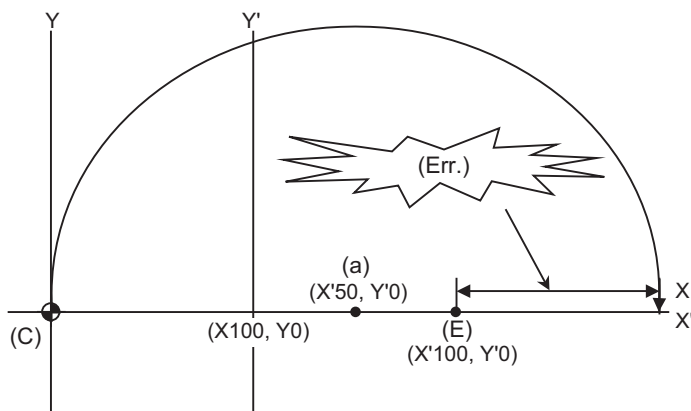
F3000 G17 ;

G68 X100. Y0. Z0. I0 J0 K1 R0. ;

G02 X100. I50. ;



No 3-dimensional coordinate conversion



In 3-dimensional coordinate conversion

(a) Arc center (C) Current position (E) End point (Err.) Program error

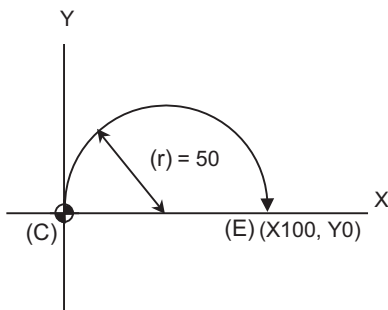
- (2) For the arc in which the arc radius is specified with R, if the center coordinate has deviated by 3-dimensional coordinate conversion, a program error (P71 Arc center calculation disabled) will occur.

G90 G28 X0 Y0 Z0 ;

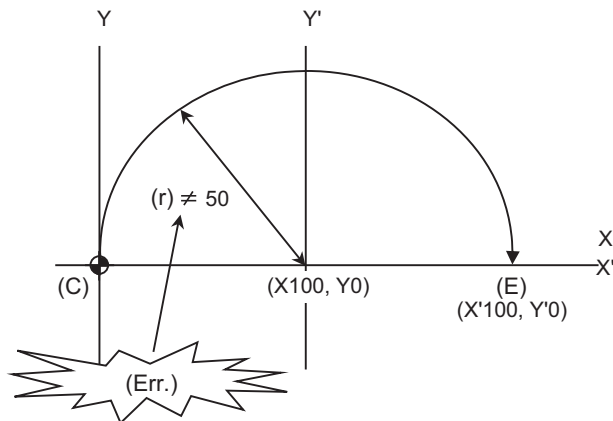
F3000 G17 ;

G68 X100. Y0. Z0. I0 J0 K1 R0. ;

G02 X100. R50. ;



No 3-dimensional coordinate conversion



In 3-dimensional coordinate conversion

(a) Arc center (C) Current position (E) End point (r) Radius (Err.) Program error

20.9 Coordinate Rotation by Program; G68/G69



Function and purpose

When machining a complicated shape located in a rotated position in respect to the coordinate system, this function enables to machine the rotated shape with the program for the shape before rotation on the local coordinate system and with the rotation angle designated by the program coordinate rotation command.



Command format

Coordinate rotation ON

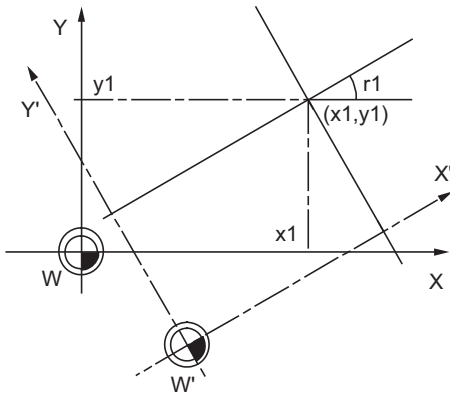
G68 X__ Y__ R__;

X,Y	Rotation center coordinates Two axes (X, Y or Z) corresponding to the selected plane are designated with absolute positions.
R	Rotation angle The counterclockwise direction is +.

Coordinate rotation cancel

G69;

Select the command plane with G17 to G19.



W: Original local coordinate
r1: Rotation angle

W': Rotated local coordinate system
(x1, y1) Rotation center

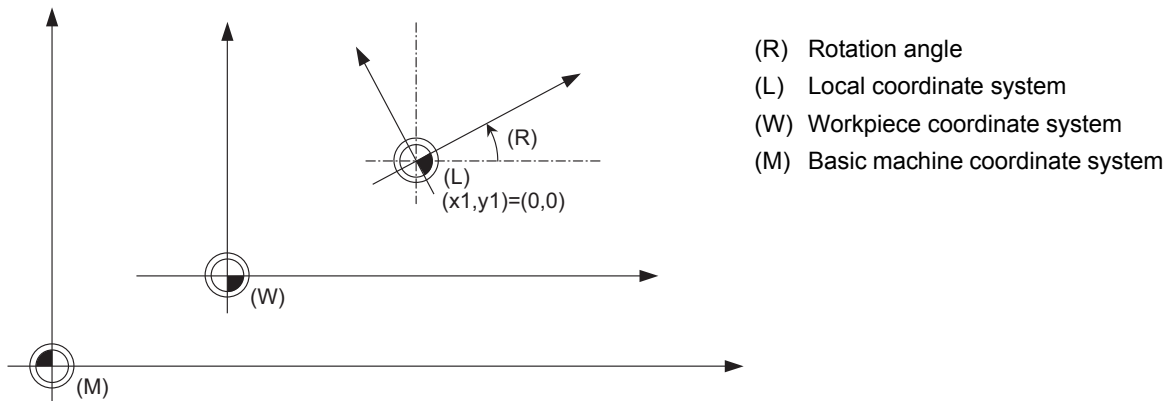


Detailed description

- (1) Always command the rotation center coordinate (x1, y1) with an absolute value. Even if commanded with an incremental address, it will not be handled as an incremental value. The rotation angle "r1" depends on the G90/G91 modal.
- (2) If the rotation center coordinates (x1, y1) are omitted, the position where the G68 command was executed will be the rotation center.
- (3) The rotation takes place in the counterclockwise direction by the angle designated in rotation angle r1.
- (4) The rotation angle r1 setting range is -360.000 to 360.000. If a command exceeding 360 degrees is issued, the remainder divided by 360 degrees will be the command.
- (5) Since the rotation angle "r1" is modal data, if once commanded, it will not be changed until the new angle is commanded. Thus, the command of rotation angle "r1" can be omitted.

If the rotation angle is omitted in spite that G68 is commanded for the first time, "r1" will be regarded as "0".

- (6) The program coordinate rotation is a function used on the local coordinate system. The relation of the rotated coordinate system, workpiece coordinate system and basic machine coordinate system is shown below.



- (7) The coordinate rotation command during coordinate rotation is processed as the changes of center coordinates and rotation angle.
- (8) If M02 or M30 is commanded or the reset signal is input during the coordinate rotation mode, the coordinate rotation mode will be canceled.
- (9) G68 is displayed on the modal information screen during the coordinate rotation mode. When the mode is canceled, the display changes to G69. (The modal value is not displayed for the rotation angle command R.)
- (10) The program coordinate rotation function is valid only in the automatic operation mode.



Program example

Program coordinate rotation by absolute command

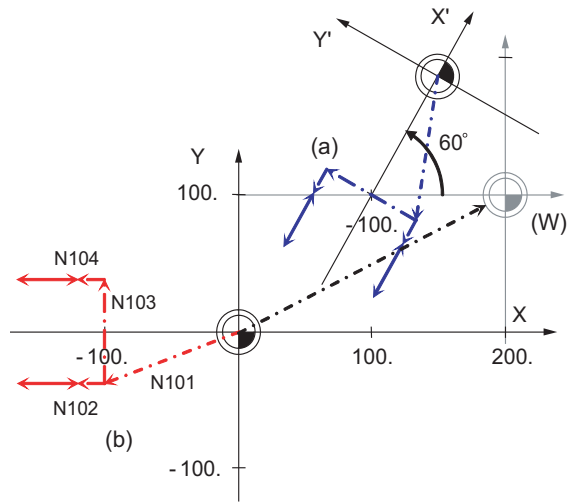
```

N01 G28 X0. Y0.;
N02 G54 G52 X200. Y100.;      Local coordinate designa-
                               tion
N03 T10 ;
N04 G68 X-100. Y0. R60.;      Coordinate rotation ON
N05 M98 H101 ;                Subprogram execution
N06 G69 ;                     Coordinate rotation cancel
N07 G54 G52 X0 Y0 ;           Local coordinate system
                               cancel
N08 M02;                       End
Subprogram
(Shape programmed with original coordinate system)
    
```

```

N101 G00 X-100. Y-40.;
N102 G83 X-150. R-20. Q-10.F100 ;
N103 G00 Y40. ;
N104 G83 X-150. R-20. Q-10.F100 ;
N105 M99;
    
```

(a) Actual machining shape
(W) Local coordinates (before rotation)



(b) Program coordinate

Operation when only one axis was commanded by the first movement command

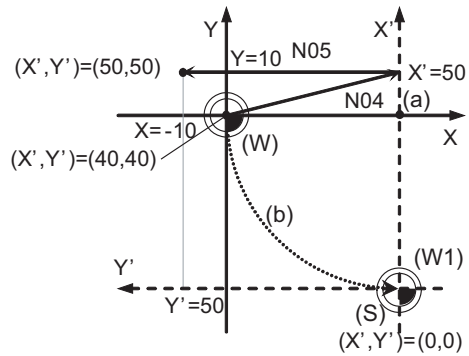
Command basically two axes in the rotation plane by an absolute value immediately after the coordinate rotation command.

When commanding one axis only, the following two kinds of operations can be selected by the parameter "#19003 PRG coord rot type".

(1) When "#19003 PRG coord rot type" is "1", the operation is the same as when "N04" is "X50.Y0.". The end point is calculated on the assumption that the start point rotates along with the coordinates' rotation.

```

N01 G17 G28 X0. Y0.;
N02 G90 G92 G53 X0. Y0.;
N03 G68 X40. Y0. R90.;    Coordinate rotation ON
N04 X50.;
N05 Y50.;
N06 G69;                  Coordinate rotation cancel
N07 M02;                  End
    
```

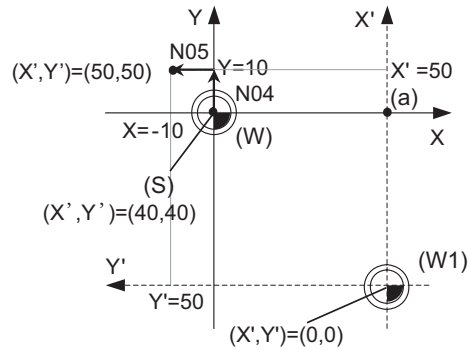


- > Machine movement path
- (a) Center of rotation
- (W) Local coordinate system before rotation
- (S) Start point
- (b) The start point is rotated virtually
- (W1) Local coordinate system after rotation

(2) When "#19003 PRG coord rot type" is "0", only axis commanded in N04 (X' Axis) is moved. The start point does not rotate along with the coordinate rotation; therefore the end position is calculated based on the current position on the local coordinate system before rotation.

```

N01 G17 G28 X0. Y0.;
N02 G90 G92 G53 X0. Y0.;
N03 G68 X40. Y0. R90.;    Coordinate rotation ON
N04 X50.;
N05 Y50.;
N06 G69;                  Coordinate rotation cancel
N07 M02;                  End
    
```



- > Machine movement path
- (a) Center of rotation
- (W) Local coordinate system before rotation
- (S) Start point
- (b) The start point is rotated virtually
- (W1) Local coordinate system after rotation

Local coordinate designation during program coordinate rotation

- (1) When "#19003 PRG coord rot type" is "0", the position commanded on the rotated coordinate system is set as the local coordinate zero point.
- (2) When "#19003 PRG coord rot type" is "1", the position commanded on the coordinate system before it is rotated, is set as the local coordinate zero point and the local coordinate will be rotated.

```

N01 G17 G28 X0. Y0.;
N02 G90 G92 G53 X0. Y0.;
N03 G68 X20. Y0. R90.;   Coordinate rotation ON
N04 G52 X10. Y10.;      Local coordinate setting
N05 X20.;
N06 Y10.;
N07 G69 ;               Coordinate rotation cancel
N08 M02;                End
    
```

W: Workpiece coordinate system
L: Local coordinate system

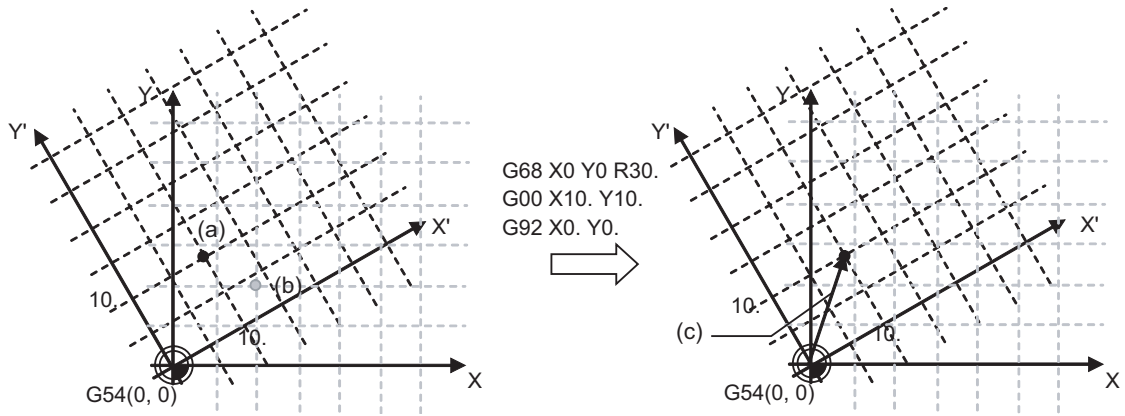
	(1) Operation of #19003 = 0	(2) Operation of #19003 = 1
N03	<p>(Workpiece coordinate system is rotated virtually.)</p> <p>(Rotation center) (Workpiece coordinate system after rotation)</p>	
	Workpiece coordinate system is rotated virtually.	Workpiece coordinate system is not rotated.
N04	<p>(Local coordinate designation)</p> <p>(X, Y)=(0,0) (Local coordinate designation)</p> <p>(X, Y)=(10,10)</p>	<p>(Local coordinate designation)</p> <p>(Rotation center)</p> <p>(X, Y)=(0,0)</p>
	The workpiece coordinate zero point after rotation is considered as (X,Y)=(0,0). The position after shifted by 10 each in the X and Y directions is set as the local coordinate zero point. The direction of the shift is not the direction of X' and Y'.	Designate the local coordinate system on the workpiece coordinate system.
N05	<p>(X'', Y'')=(20,30)</p> <p>Start point:(X'', Y'')=(10,30)</p>	<p>(Rotation center)</p> <p>(X'', Y'')=(10,30)</p> <p>(X'', Y'')=(20,-10)</p> <p>(The start point is rotated virtually.)</p> <p>Start point: (X'', Y'')=(-10,-10)</p>
	The commanded axis moves on the rotation coordinate system. Axis without movement command does not move.	The commanded axis moves on the rotation coordinate system. Axis without movement command moves to the position on rotation coordinate system.
N06	<p>(X'', Y'')=(20,30)</p> <p>(X'', Y'')=(20,10)</p>	<p>(X'', Y'')=(20,10)</p> <p>(X'', Y'')=(20,-10)</p>

Coordinate system designation during program coordinate rotation

When the coordinate system setting (G92) is executed during program coordinate rotation (G68), this program operates same as "Local coordinate designation during program coordinate rotation".

- (1) When "#19003 PRG coord rot type" is "0", the position is preset to the current position commanded on the rotated coordinate system.

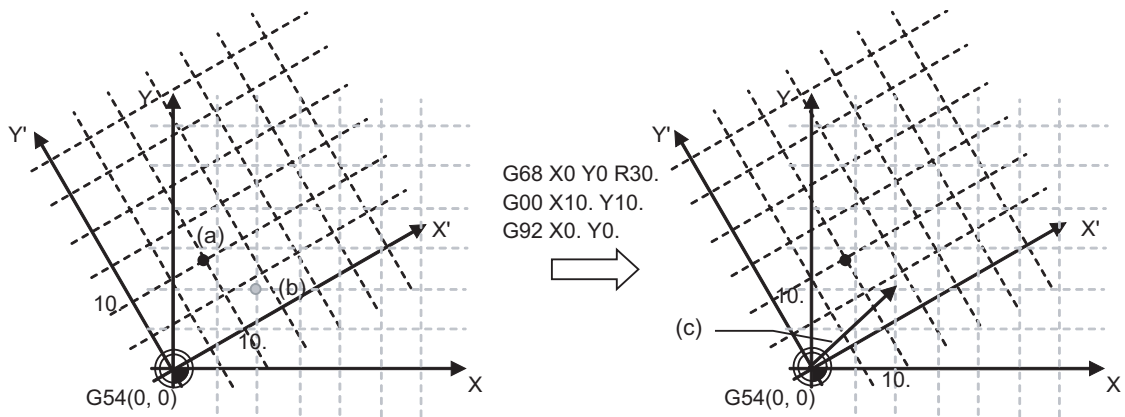
(Example) Designation on the coordinate system (X'-Y') after rotation



(a) Position after rotation (b) Commanded position (c) G92 shift amount

- (2) When "#19003 PRG coord rot type" is "1", the position is preset to the current position commanded on the coordinate system before rotation. The coordinate system is rotated after the position is commanded.

(Example) Setting on the coordinate system (X-Y) after rotation



(a) Position after rotation (b) Commanded position (c) G92 shift amount

<Note>

- When "#19003 PRG coord rot type" is "1" and the coordinate system setting (G92) is executed during coordinate rotation mode, the rotation center of the program coordinate rotation is not shifted. (It stays at the same position in respect to the basic machine coordinate system.)



Relationship with other functions

- (1) Program error (P111) will occur if the plane selection code is commanded during the coordinate rotation mode.
- (2) Program error (P485) will occur if polar coordinate interpolation is commanded during the coordinate rotation mode.
- (3) Program error (P481) will occur if coordinate rotation is commanded during the polar coordinate interpolation mode.
- (4) Program error (P485) will occur if cylindrical interpolation is commanded during the coordinate rotation mode.
- (5) Program error (P481) will occur if coordinate rotation is commanded during the cylindrical interpolation mode.
- (6) Program error (P34) will occur if the workpiece coordinate system preset (G92.1) is commanded during the coordinate rotation mode.
- (7) Program error (P34) will occur if high-accuracy control mode, high-speed machining mode, high-speed high-accuracy I or II is commanded during the coordinate rotation mode.
- (8) Program coordinate rotation and figure rotation cannot be carried out simultaneously. If the coordinate rotation is commanded during the figure rotation and vice versa, a program error (P252) will occur.
- (9) If the tool position offset is commanded during the coordinate rotation mode, a program error (P141) will occur.



Precautions

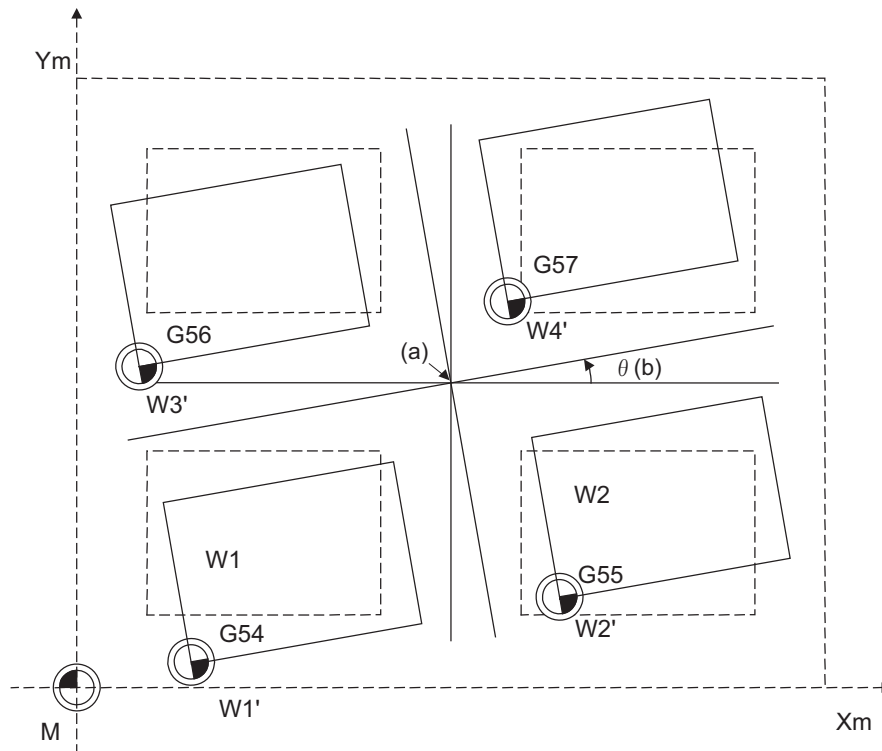
- (1) Always command an absolute value for the movement command immediately after G68 and G69.
- (2) If the manual absolute is ON and interrupted the coordinate rotary axis, then, do not use automatic operation for the following absolute value command.
- (3) The intermediate point during reference position return is the position after the coordinates are rotated.
- (4) If the workpiece coordinate system offset amount is changed during the coordinate rotation mode, the rotation center for the program coordinate rotation will be shifted. (The center will follow the coordinate system.)
- (5) If the workpiece coordinates are changed during the coordinate rotation mode (ex. from G54 to G55), the rotation center of the program coordinate rotation will be the position on the coordinate system which the command was issued. (It stays at the same position in respect to the basic machine coordinate system.)
- (6) If coordinate rotation is executed to the G00 command for only one axis, two axes will move. If G00 non-interpolation (parameter "#1086 G0Intp" = 1) is set, each axis will move independently at the respective rapid traverse rates. If the axis must be moved linearly (interpolated) from the start point to the end point (such as during the hole machining cycle), always turn G00 non-interpolation OFF (parameter "#1086 G0Intp" = 0). The feedrate in this case is the composite speed of each axis' rapid traverse rate, so the movement speed will be faster than when moving only one axis (before coordinate rotation).
- (7) If the coordinate rotation specifications are not provided, a program error (P260) will occur when coordinate rotation is commanded.
- (8) The compensation during the coordinate rotation mode is carried out to the local coordinate system after coordinate rotation. The compensation direction is the coordinate system before rotation.
- (9) Mirror image during the coordinate rotation mode is applied to the local coordinate system after coordinate rotation.
- (10) On the display, the positions after rotation is always displayed on the local coordinate system before rotation.
- (11) When the coordinate value variables are read, the positions are all on the coordinate system before rotation.
- (12) The coordinates can also be rotated for the parallel axis. Select the plane that contains the parallel axis before issuing the G68 command. The plane cannot be selected in the same block as the G68 command.
- (13) The coordinates can be rotated for the rotary axis. The angle will be interpreted as the length when rotating.

20.10 Coordinate Rotation Input by Parameter; G10 I_ J_/K_



Function and purpose

If a deviation occurs between the workpiece alignment line and machine coordinate system's coordinate axis when the workpiece is mounted, the machine can be controlled by rotating the machining program coordinates according to the workpiece alignment line deviation. The coordinate rotation amount is set with the parameters. The parameters can also be set with the G10 command.



(a) Center of rotation

(b) Rotation angle

To enable this function, the following conditions must be satisfied:

- (1) The parameter "#8116 CoordRotPara invd" is set to "0".
- (2) The parameter "#8627 Coord rot angle" is set. Alternatively, "#8625 Coord rot vctr(H)" and "#8626 Coord rot vctr(V)" are set.



Command format

G10 I__ J__ ;

G10 K__ ;

I	Horizontal vector. Command a value corresponding to the parameter "#8625 Coord rot vctr(H)". Command range: -99999.999 to 99999.999 The value of "#8627 Coord rot angle" is automatically calculated when commanding vector contents.
J	Vertical vector. Command a value corresponding to the parameter "#8626 Coord rot vctr(V)". Command range: -99999.999 to 99999.999 The value of "#8627 Coord rot angle" is automatically calculated when commanding vector contents.
K	Rotation angle. Command a value corresponding to the parameter "#8627 Coord rot angle". Command range: -360.000 to 360.000 "#8625 Coord rot vctr(H)" and "#8626 Coord rot vctr(V)" are set to "0" when commanding the coordinate rotation angle.

Parameters specified in the parameter setting screen can be changed from the machining program.

Refer to the "Instruction Manual" for settings and contents of the parameters.

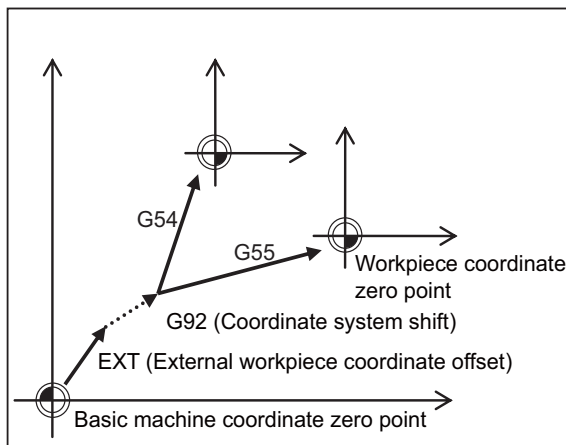


Detailed description

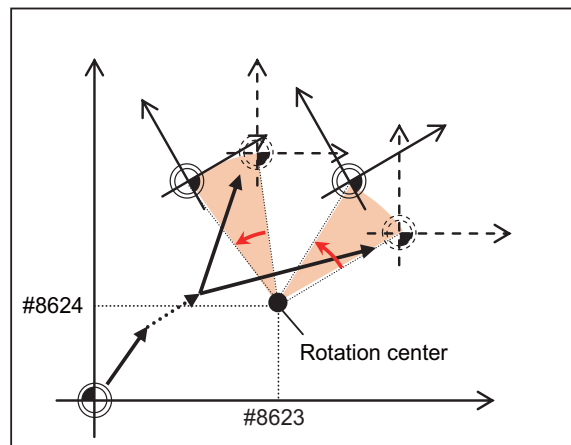
- (1) As for the coordinate rotation center position, designate the position on the machine coordinate system.
- (2) All workpiece coordinate systems from G54 to G59, G54.1 rotate with the rotation command. While the machine coordinate system does not rotate, it can be understood that there is a hypothetical machine coordinate system in the coordinate system after rotation.
- (3) When parameter settings are configured, their setting values become valid at the following timings.
 - Automatic operation: The setting values become valid from the next block after parameter settings have been configured.
 - Manual operation: The setting values become valid if the PLC signal (manual feed coordinate switching for coordinate rotation by parameter) is set to ON after parameter settings have been configured.

Concept of coordinate system

- (1) Set the parameters "#8623 Coord rot centr(H)" and "#8624 Coord rot centr(V)" at the position of the machine coordinate system.
- (2) The workpiece coordinate system set on the orthogonal coordinate system rotates around the rotation center.
- (3) The machine coordinate system does not rotate.



Workpiece coordinate system setting



Workpiece coordinate system after coordinate rotation by parameter

Coordinate rotation start

The coordinate rotation starts when the following parameters are changed. (When the same value is reset to the parameter, it is not considered as change.)

When the parameter "#8116 CoordRotPara invd" is "1" or the parameter "#8627 coordinate rotation angle" is "0", coordinate rotation will not start.

- #8621 Coord rot plane (H)
- #8622 Coord rot plane (V)
- #8623 Coord rot centr (H)
- #8624 Coord rot centr (V)
- #8625 Coord rot vctr (H)
- #8626 Coord rot vctr (V)
- #8627 Coord rot angle
- #8116 CoordRotPara invd (*1)

(*1) The parameter "#8116 CoordRotPara invd" is common to all part systems. Therefore, before designating this parameter, set "0" to the parameter "#8627 Coord rot angle" in part systems in which "coordinate rotation by parameter" is not used.

Coordinate rotation stop (cancel)

When coordinate rotation is enabled, setting the parameter "#8116 CoordRotPara invd" to "1" or "#8627 Coord rot angle" to "0", and then issuing the following movement command cancels the parameter coordinate rotation.

Coordinate rotation temporary cancel

The coordinate rotation by parameter is temporarily canceled when in (1) or (2) as follows.

(1) Reference position return command (G28, G30)

If reference position return is performed on any of the axes in the rotated coordinate system (horizontal axis or vertical axis), both of the two axes will temporarily cancel the coordinate rotation.

However moving to the intermediate point will not be temporarily canceled, but it will keep operating.

(2) Basic machine coordinate system selection (G53)

Only the commanded axis of basic machine coordinate system selection (G53) will be temporarily cancel the coordinate rotation.

In items (1) and (2) above, when the coordinate rotation by parameter is canceled temporarily, the counter display follows the setting of the parameter "#11086 PRM coordinate rotation counter".



Operation example

First movement command after coordinate rotation start/end/temporary cancel

When issuing a movement command for the first time after coordinate rotation has been started, ended, reset, or returned from temporary cancel, issue the command in G00 or G01 mode by designating the two axes configuring the rotation plane using absolute values.

[Parameters]

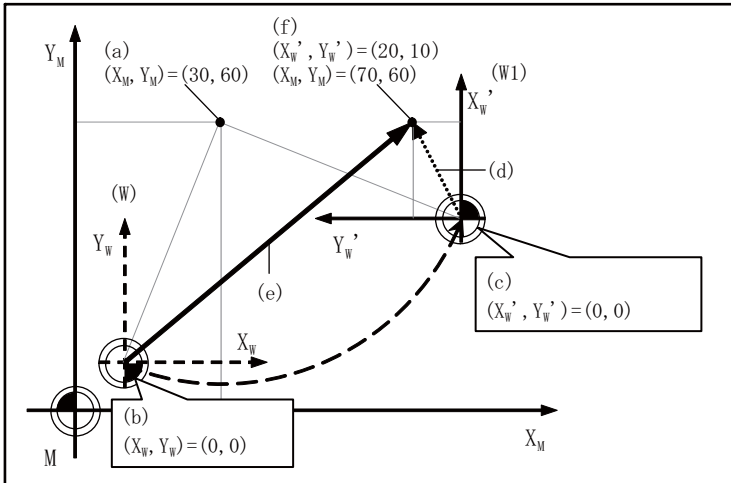
- #8621 Coord rot plane (H) = X
- #8622 Coord rot plane (V) = Y
- #8623 Coord rot centr (H) = 30.0
- #8624 Coord rot centr (V) = 60.0
- #8627 Coord rot angle = 0.0

[Machining program]

- N01 G17 G28 X0. Y0.;
- N02 G54 G90 X0. Y0.;
- N03 G10 K90.; (Coordinate rotation start)
- N04 G54 G90 G00 X20. Y10.; (Absolute value command to two axes)
- :

[G54 workpiece coordinate system offset]

- X = 10.0
- Y = 10.0



- (W): Workpiece coordinate system before rotation
- (W1): Workpiece coordinate system after rotation
- (a): Rotation center
- (b): Actual axis position
- (c): Workpiece coordinate system zero point after coordinate rotation
- (d): N04 Commanded path
- (e): N04 Actual movement path
- (f): N04 End point

Note that, if the command that is issued for the first time after coordinate rotation has been started, ended, or returned from temporary cancel is any of the following, the operation differs depending on the setting of the parameter "#19008 PRM coord rot type".

- (1) Command to an axis configuring the rotation plane by the absolute value
- (2) Command by incremental value
- (3) Circular interpolation command

Operation when an axis configuring the rotation plane is commanded by the absolute value

The operation can be selected from the following 2 types by setting of the parameter "#19008 PRM coord rot type".

(1) When the parameter "#19008 PRM coord rot type" is "0".

The end point is calculated by virtually rotating the start point along with the coordinate rotation.

For that reason, the operation is the same as when "N04" is "G00 X20. Y0." in the following example.

[Parameters]

#8621 Coord rot plane (H) = X
 #8622 Coord rot plane (V) = Y
 #8623 Coord rot centr (H) = 30.0
 #8624 Coord rot centr (V) = 60.0
 #8627 Coord rot angle = 0.0

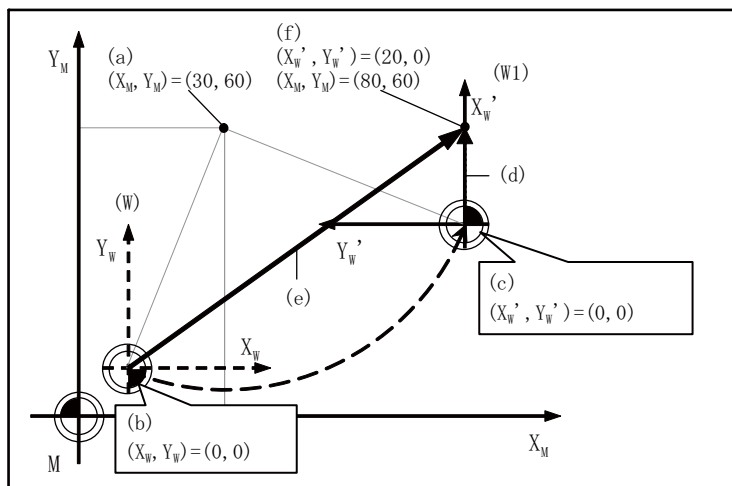
[Machining program]

N01 G17 G28 X0. Y0.;
 N02 G54 G90 X0. Y0.;
 N03 G10 K90.; (Coordinate rotation start)
 N04 G54 G90 G00 X20.; (Absolute value command
 to an axis)
 :

[G54 workpiece coordinate system offset]

X = 10.0

Y = 10.0



(W): Workpiece coordinate system before rotation

(W1): Workpiece coordinate system after rotation

(a): Rotation center

(b): Actual axis position

(c): Start point rotated virtually along with the coordinate rotation

(d): N04 Commanded path

(e): N04 Actual movement path

(f): N04 End point

(2) When the parameter "#19008 PRM coord rot type" is "1".

The start point does not rotate along with the coordinate rotation; therefore the end point position is calculated based on the current position on the local coordinate system before rotation.

For that reason, only the axis commanded in N04 (X' axis) is moved.

[Parameters]

#8621 Coord rot plane (H) = X

#8622 Coord rot plane (V) = Y

#8623 Coord rot centr (H) = 30.0

#8624 Coord rot centr (V) = 60.0

#8627 Coord rot angle = 0.0

[Machining program]

N01 G17 G28 X0. Y0.;

N02 G54 G90 X0. Y0.;

N03 G10 K90.; (Coordinate rotation start)

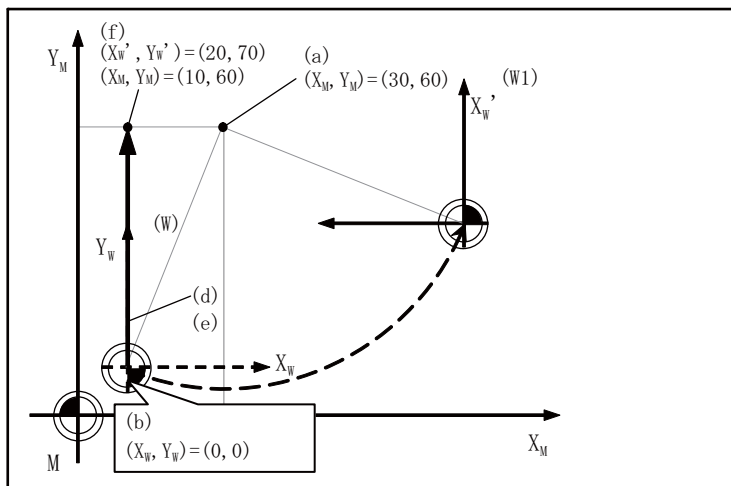
N04 G54 G90 G00 X20.; (Absolute value command to an axis)

:

[G54 workpiece coordinate system offset]

X = 10.0

Y = 10.0



(W): Workpiece coordinate system before rotation

(W1): Workpiece coordinate system after rotation

(a): Rotation center

(b): The actual axis position and start point position are the same

(d): N04 Commanded path

(e): N04 Actual movement path

(f): N04 End point

Operation when incremental value commands are given

The operation can be selected from the following 2 types by setting of the parameter "#19008 PRM coord rot type".

(1) When the parameter "#19008 PRM coord rot type" is "0".

The end point is calculated by virtually rotating the start point along with the coordinate rotation.
For that reason, the commanded path and actual movement path differs in N04.

[Parameters]

#8621 Coord rot plane (H) = X
#8622 Coord rot plane (V) = Y
#8623 Coord rot centr (H) = 30.0
#8624 Coord rot centr (V) = 60.0
#8627 Coord rot angle = 0.0

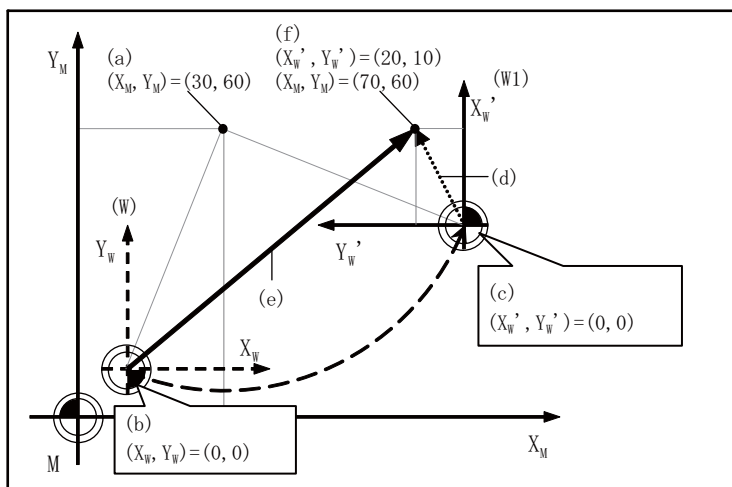
[Machining program]

N01 G17 G28 X0. Y0.;
N02 G54 G90 X0. Y0.;
N03 G10 K90.; (Coordinate rotation start)
N04 G54 G91 G00 X20. Y10.; (Incremental value
command to two axes)
:

[G54 workpiece coordinate system offset]

X = 10.0

Y = 10.0



(W): Workpiece coordinate system before rotation

(W1): Workpiece coordinate system after rotation

(a): Rotation center

(b): Actual axis position

(c): Start point rotated virtually along with the coordinate rotation

(d): N04 Commanded path

(e): N04 Actual movement path

(f): N04 End point

(2) When the parameter "#19008 PRM coord rot type" is "1".

The start point does not rotate along with the coordinate rotation; therefore the end point position is calculated based on the current position on the local coordinate system before rotation.

For that reason, the commanded path and actual movement path are the same in N04.

[Parameters]

#8621 Coord rot plane (H) = X

#8622 Coord rot plane (V) = Y

#8623 Coord rot centr (H) = 30.0

#8624 Coord rot centr (V) = 60.0

#8627 Coord rot angle = 0.0

[Machining program]

N01 G17 G28 X0. Y0.;

N02 G54 G90 X0. Y0.;

N03 G10 K90.; (Coordinate rotation start)

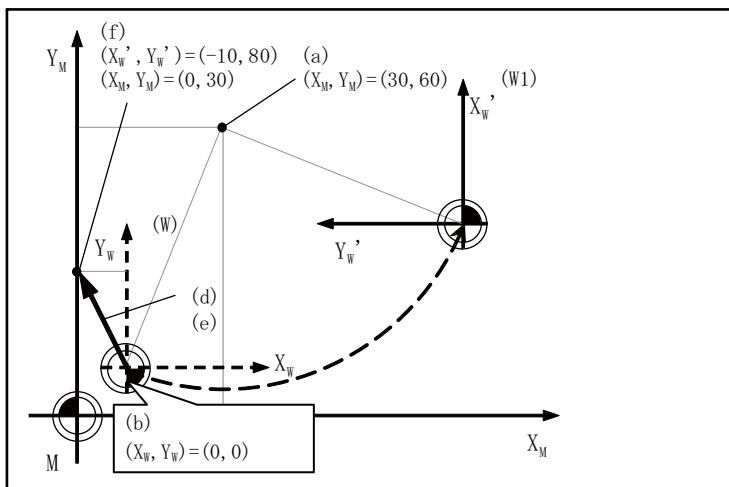
N04 G54 G91 G00 X20. Y10.; (Incremental value command to two axes)

:

[G54 workpiece coordinate system offset]

X = 10.0

Y = 10.0



(W): Workpiece coordinate system before rotation

(W1): Workpiece coordinate system after rotation

(a): Rotation center

(b): The actual axis position and start point position are the same

(d): N04 Commanded path

(e): N04 Actual movement path

(f): N04 End point

Operation when circular interpolation is commanded

The operation can be selected from the following 2 types by setting of the parameter "#19008 PRM coord rot type".

(1) When the parameter "#19008 PRM coord rot type" is "0".

The end point of an arc is calculated from the position to which the start point is virtually rotated along with the coordinate rotation.

In this case, the start point of an arc is not rotating but the end point of an arc is rotating, so it may cause "P70: Arc end point deviation large" due to the difference in radius between the start and end points.

[Parameters]

#1084 Arc error = 0.1

#8621 Coord rot plane (H) = X

#8622 Coord rot plane (V) = Y

#8623 Coord rot centr (H) = 30.0

#8624 Coord rot centr (V) = 60.0

#8627 Coord rot angle = 0.0

[Machining program]

N01 G17 G28 X0. Y0.;

N02 G54 G90 X0. Y0.;

N03 G10 K90.; (Coordinate rotation start)

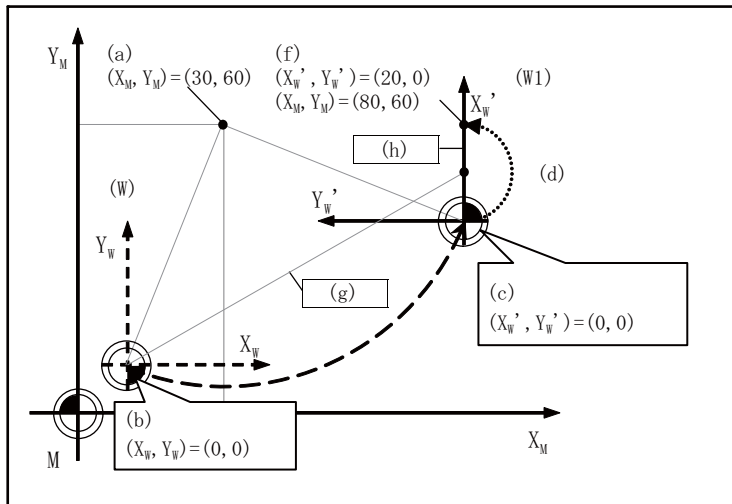
N04 G54 G91 G03 X20. R10. F500; (Circular interpolation command)

:

[G54 workpiece coordinate system offset]

X = 10.0

Y = 10.0



(W): Workpiece coordinate system before rotation

(W1): Workpiece coordinate system after rotation

(a): Rotation center

(b): Actual axis position

(c): Start point rotated virtually along with the coordinate rotation

(d): N04 Commanded path

(f): End point calculated from the virtually rotated start point

(g): Start point radius

(h): End point radius

As the difference in radius between the start and end points is bigger than "#1084 RadErr", it causes program error (P70).

(2) When the parameter "#19008 PRM coord rot type" is "1".

The start point and end point of an arc are calculated from the current position on the workpiece coordinate system before the coordinate rotation and the circular interpolation is performed from the current position to the end point.

[Parameters]

#1084 Arc error = 0.1

#8621 Coord rot plane (H) = X

#8622 Coord rot plane (V) = Y

#8623 Coord rot centr (H) = 30.0

#8624 Coord rot centr (V) = 60.0

#8627 Coord rot angle = 0.0

[Machining program]

N01 G17 G28 X0. Y0.;

N02 G54 G90 X0. Y0.;

N03 G10 K90.; (Coordinate rotation start)

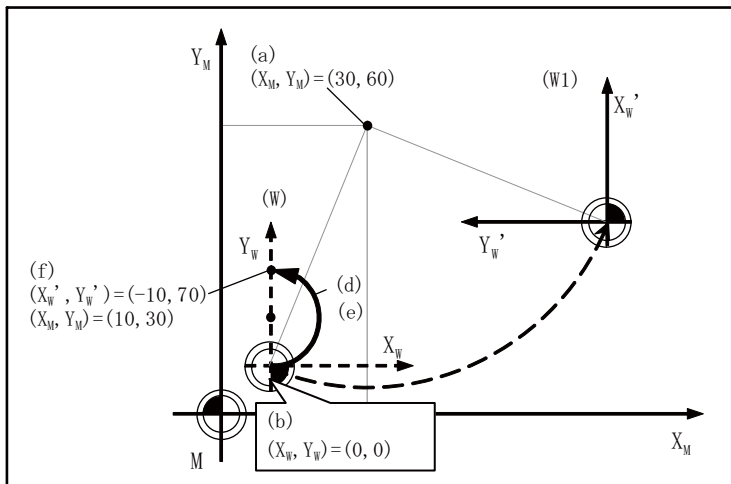
N04 G54 G91 G03 X20. R10. F500; (Circular interpolation command)

:

[G54 workpiece coordinate system offset]

X = 10.0

Y = 10.0



(W): Workpiece coordinate system before rotation

(W1): Workpiece coordinate system after rotation

(a): Rotation center

(b): The actual axis position and start point position are the same

(d): N04 Commanded path

(e): N04 Actual movement path

(f): N04 End point

First movement command after coordinate rotation cancel

When rotation angle "0" is commanded during coordinate rotation, it will be canceled by next movement command regardless of G90 and G91.

The calculation of the end point will be different by setting of the parameter "#19008 PRM coord rot type".

(1) When the parameter "#19008 PRM coord rot type" is "0".

The end point is calculated on the assumption that the start point rotates along with the coordinate rotation cancel.

Program the first movement command after coordinate rotation cancel either G00 or G01 mode.

[Parameters]

#8621 Coord rot plane (H) = X
 #8622 Coord rot plane (V) = Y
 #8623 Coord rot centr (H) = 30.0
 #8624 Coord rot centr (V) = 60.0
 #8627 Coord rot angle = 90.0

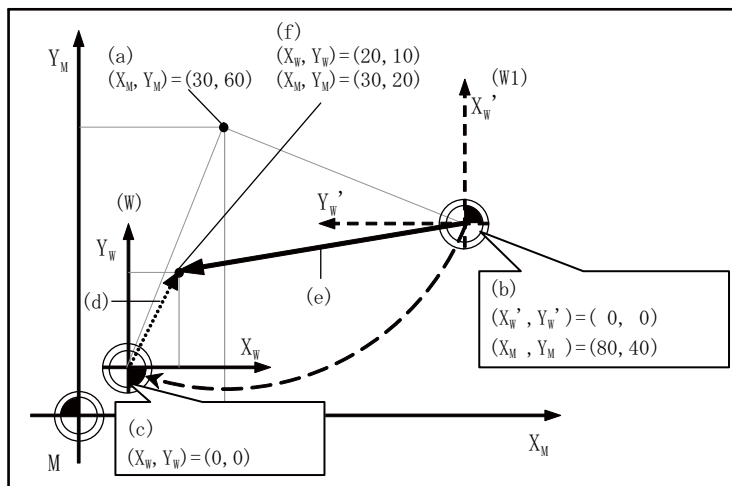
[Machining program]

N01 G54 G90 X50.Y50.;
 N02 G54 G90 X0. Y0.;
 N03 G10 K0.; (Coordinate rotation cancel)
 N04 G91 G00 X20. Y10.; (Incremental value command to two axes)
 :

[G54 workpiece coordinate system offset]

X = 10.0

Y = 10.0



(W): Workpiece coordinate system before rotation

(W1): Workpiece coordinate system after rotation

(a): Rotation center

(b): Actual axis position

(c): Start point rotated virtually along with the coordinate rotation cancel

(d): N04 Commanded path

(e): N04 Actual movement path

(f): N04 End point

(2) When the parameter "#19008 PRM coord rot type" is "1".

The start point does not rotate along with the coordinate rotation cancel; therefore the end point position is calculated based on the current position on the local coordinate system before rotation.

[Parameters]

#8621 Coord rot plane (H) = X

#8622 Coord rot plane (V) = Y

#8623 Coord rot centr (H) = 30.0

#8624 Coord rot centr (V) = 60.0

#8627 Coord rot angle = 90.0

[Machining program]

N01 G54 G90 X50.Y50.;

N02 G54 G90 X0. Y0.;

N03 G10 K0.; (Coordinate rotation cancel)

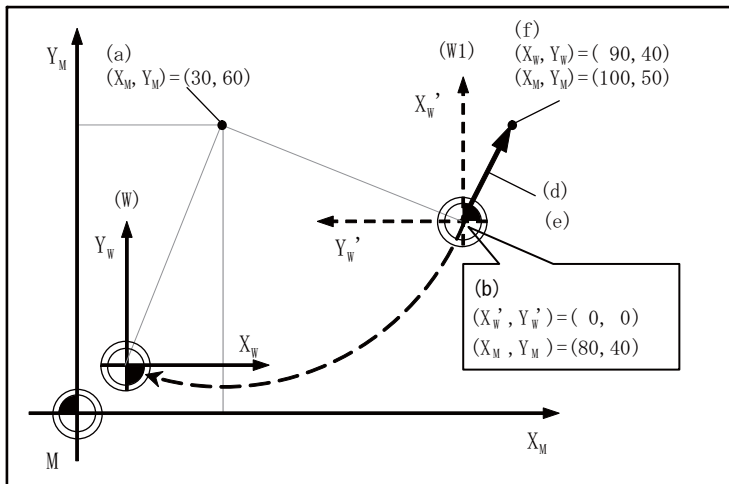
N04 G91 G00 X20. Y10.; (Incremental value command to two axes)

:

[G54 workpiece coordinate system offset]

X = 10.0

Y = 10.0



(W): Workpiece coordinate system before rotation

(W1): Workpiece coordinate system after rotation

(a): Rotation center

(b): The actual axis position and start point position are the same

(d): N04 Commanded path

(e): N04 Actual movement path

(f): N04 End point

First movement command after temporary coordinate rotation cancel

The operation of the first movement command issued after the program coordinate rotation is returned from temporary cancel is the same as the operation that occurs when the parameter "#19008 PRM coord rot type" is set to "0" in "First movement command after coordinate rotation".

Presetting the workpiece coordinate and counter in the rotation coordinate system

The workpiece coordinate and counter can be preset in the same way as for the orthogonal coordinate system by commanding G92/G92.1 in the rotation coordinate system.

Figures (1) to (3) show the operations to be performed when the machining program is executed while the parameters and workpiece coordinate system offset are set as follows:

[Parameters]

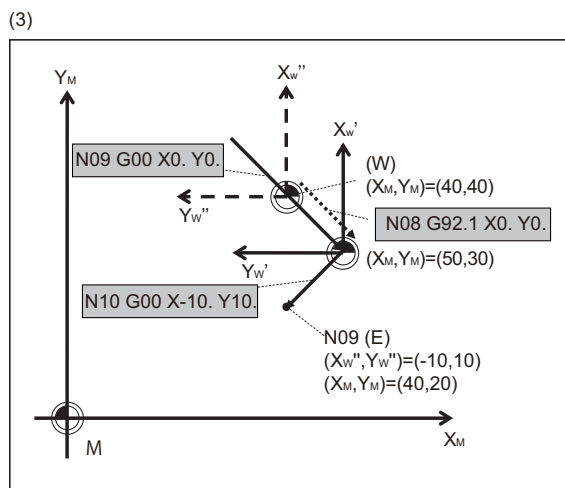
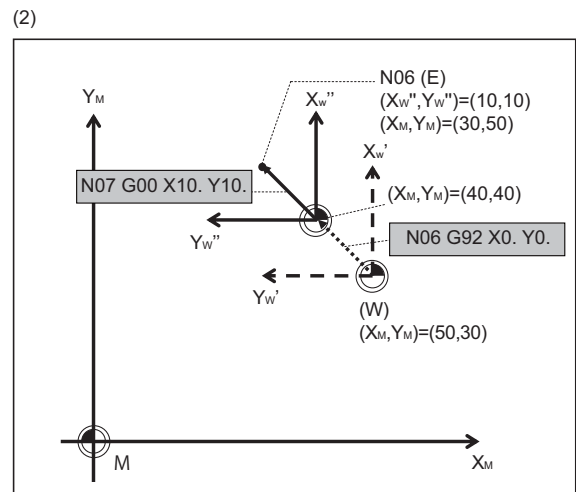
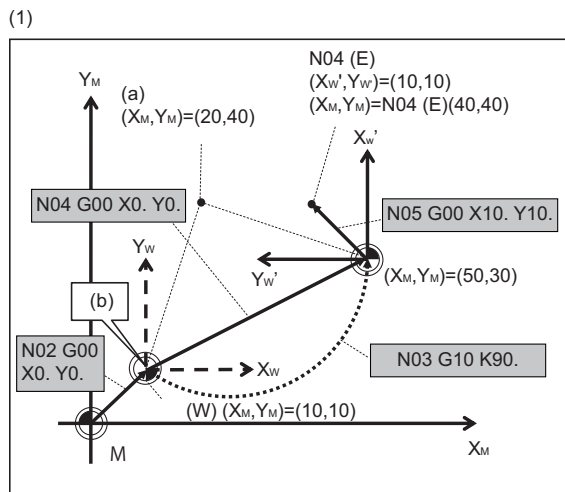
- #19008 PRM coord rot type = 0
(Case where the start point is virtually rotated with the coordinate rotation)
- #8621 Coord rot plane (H) = X
- #8622 Coord rot plane (V) = Y
- #8623 Coord rot centr (H) = 20.0
- #8624 Coord rot centr (V) = 40.0
- #8627 Coord rot angle = 0.0

[Machining program]

- (1) N01 G54 G17 G28 X Y;
N02 G90 G00 X0. Y0.;
- N03 G10 K90. ;
N04 G00 X0. Y0. ;
N05 G00 X10. Y10. ;
- (2) N06 G92 X0. Y0. ;
N07 G00 X10. Y10. ;
- (3) N08 G92.1 X0. Y0. ;
N09 G00 X0. Y0. ;
N10 G00 X-10. Y10. ;

[G54 workpiece coordinate system offset]

- X = 10.0
- Y = 10.0

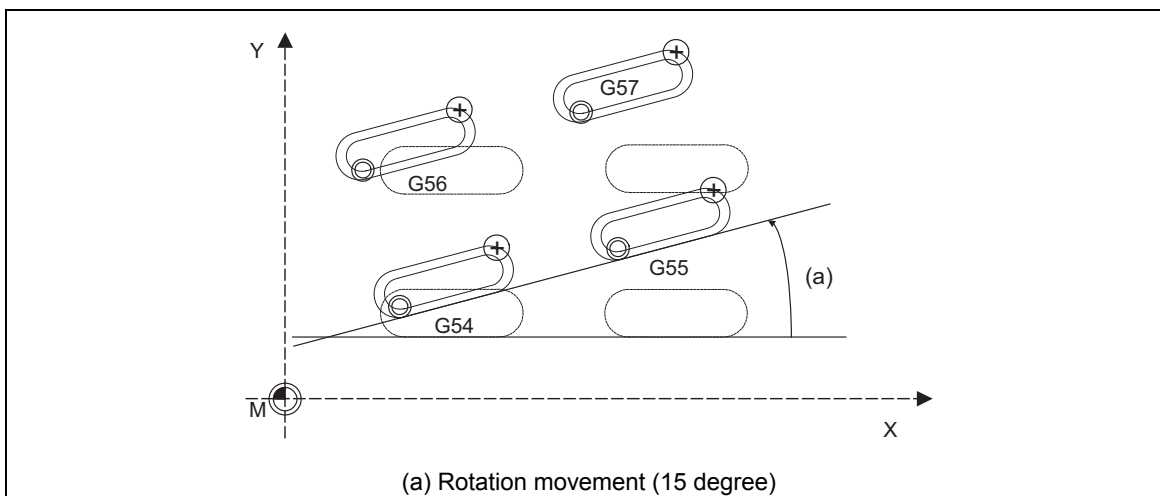


(W): Workpiece coordinate zero point
(a): Rotation center
(b): Axis position before coordinate rotation
(E): End point ("N04 (E)" refers to the end point of the N04 block.)



Program example

(1) When used for compensating positional deviation of pallet changer.



```

N01 G28 X0 Y0 Z0 ;
N02 M98 P9000 ;           Pallet deviation measurement
N03 G90 G53 X0 Y0 ;     Parallel movement amount shift
N04 G92 X0 Y0 ;         Parallel movement amount definition
N05 G10 K15. ;          Rotation amount definition
N06 G90 G54 G00 X0 Y0 ; G54 workpiece machining
N07 M98 H101 ;
N08 G90 G55 G00 X0 Y0 ; G55 workpiece machining
N09 M98 H101 ;
N10 G90 G56 G00 X0 Y0 ; G56 workpiece machining
N11 M98 H101 ;
N12 G90 G57 G00 X0 Y0 ; G57 workpiece machining
N13 M98 H101 ;
N14 G27 X0 Y0 Z0 ;
N15 M02 ;

Machining shape program
N101 G91 G01 G42 D01 F300 ;
N102 X100 ;
N103 G03 Y50. R25. ;
N104 G01 X-100. ;
N105 G03 Y-50. R25. ;
N106 G01 G40 ;
N107 M99 ;

```



Relationship with other functions

- (1) To use any of the following functions together with the coordinate rotation by parameter, start the coordinate rotation by parameter first and command the following function later.
 - ◆ Tool radius compensation
 - ◆ Mirror image
- (2) The coordinate rotation by parameter cannot be used together with the coordinate rotation by program and the 3-dimensional coordinate conversion. If they are used at the same time, the coordinate position will not be calculated right.
- (3) The following high-speed high-accuracy processes are temporarily canceled during coordinate rotation by parameter.

No program errors occur; however, the processing capacity is the same as when the high-speed high-accuracy processes are set off.

 - ◆ High-speed machining mode I / high-speed machining mode II
 - ◆ High-speed high-accuracy control I / High-speed high-accuracy control II / High-speed high-accuracy control III
- (4) If the figure rotation is commanded during the coordinate rotation by parameter, a program error (P252) will occur.
- (5) If the inclined surface machining command (G68.2/G68.3) is commanded during the coordinate rotation by parameter, a program error (P952) will occur.
- (6) The following functions can be used together with the coordinate rotation by parameter.

Classification	Function
Control axes	Number of basic control axes (NC axes) Memory mode
Input command	Inch/Metric changeover
Positioning/Interpolation	Positioning Linear interpolation Circular interpolation (Center/Radius designation)
Feed	Manual rapid traverse Jog feed Incremental feed Handle feed Manual feedrate B Manual speed clamp Dwell (Time-based designation)
Tool compensation	Tool length offset Tool position offset Tool radius compensation Tool radius compensation diameter designation
Coordinate system	Coordinate system setting Workpiece coordinate system selection External workpiece coordinate offset Workpiece coordinate system preset (G92.1) Plane selection
Operation support functions	Single block Graphic trace Manual interruption Automatic operation handle interruption Manual absolute switch
Program support functions	Subprogram control High-accuracy control (G61.1/G08) Multi-part system simultaneous high-accuracy control
Machine support functions	Custom API library



Precautions

- (1) If rotation angle zero is commanded while carrying out coordinate rotation, it will be canceled at the next movement command regardless of the G90 or G91.
- (2) Command the first movement after this command with the G00 or G01 mode. If an arc command is issued, the arc start point will not be rotated. However, only the arc end point will rotate. This will cause the start point radius and end point radius to differ, and the program error (P70) will occur.
- (3) When data has been input using the data input/output function, it is recognized that the parameter "#8627 Coord rot angle" has been input, and automatic calculation from the values of "#8625 Coord rot vctr(H)" and "#8626 Coord rot vctr(V)" is not carried out.
- (4) Do not use this command with G54 to G59 and G90, G91. If used, the command will not be reflected correctly.
- (5) If both vertical / horizontal vectors (I,J) and rotation angle are commanded, the rotation angle will be given the priority.

20.11 Scaling; G50/G51



Function and purpose

By multiplying the moving axis command values within the range specified under this command by the factor, the shape commanded by the program can be enlarged or reduced to the desired size.

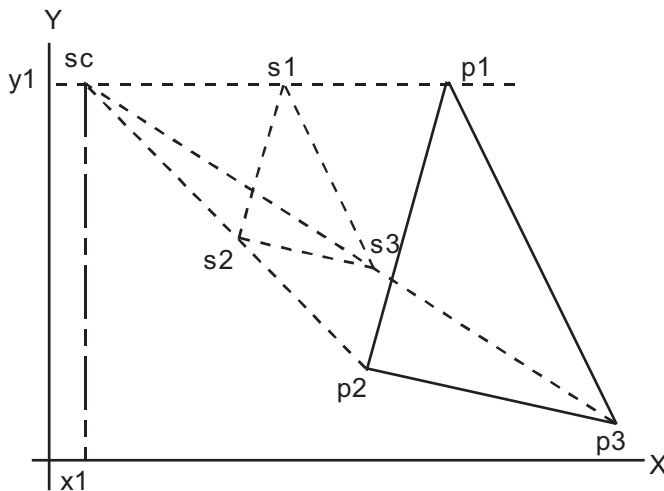


Command format

Scaling ON (set the common scaling factor to the three basic axes)

```
G51 X__ Y__ Z__ P__ ;
```

X,Y,Z	Scaling center coordinates
P	Scaling factor



sc : Scaling center

p1,p2,p3: Program shape

s1,s2,s3: Shape after scaling

Scaling ON (When setting the scaling factor to each of the three basic axes)

```
G51 X__ Y__ Z__ I__ J__ K__ ;
```

X,Y,Z	Scaling center coordinates
I	Scaling factor of basic 1st axis
J	Scaling factor of basic 2nd axis
K	Scaling factor of basic 3rd axis

Scaling cancel

```
G50 ;
```



Detailed description

Specifying the scaling axis, scaling center and its factor

Commanding G51 selects the scaling mode. The G51 command only specifies the scaling axis, its center and factor, and does not move the axis.

Though the scaling mode is selected by the G51 command, the axis actually valid for scaling is the axis where the scaling center has been specified.

(1) Scaling center

- Specify the scaling center in accordance with the then absolute/incremental mode (G90/G91).
- The scaling center must be specified also when the current position is defined as a center.
- As described above, the axis valid for scaling is only the axis whose center has been specified.

(2) Scaling factor

- Use the address P or I, J, K to specify the scaling factor.
- Minimum command unit : 0.000001
- Command range: Both -99999999 to 99999999 (-99.999999 to 99.999999 times) and -99.999999 to 99.999999 is valid, but the decimal point command is valid only after the G51 command.
- When the factor is not specified in the same block as G51, the factor set with the parameter "#8072 SCALING P" is used.
- When the address P and the address I, J, K are commanded in a same block, a factor specified by the address I, J, K is applied for the basic three axes. And a factor specified by the address P is applied for other axes.
- If changed during the scaling mode, the value of this parameter will not become valid. Scaling is performed with the setting value that was used when G51 was commanded.
- When the factor is not specified in either the program nor parameter, it is calculated as 1.

(3) A program error will occur in the following cases.

- Scaling was commanded though there was no scaling specification. (P350) - The upper limit of the factor command range was exceeded in the same block as G51. (P35) (When using the machining parameter scaling factor, the factor is calculated as 1, when $-0.000001 < \text{factor} < 0.000001$, or the factor is more than 99.999999 or less than -99.999999.)

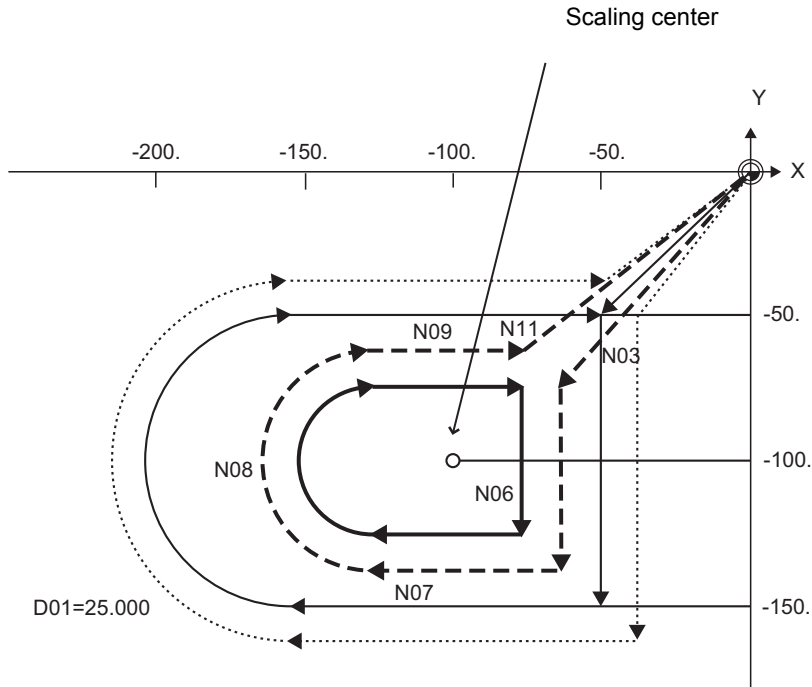
Scaling cancel

When G50 is commanded, scaling is canceled.



Program example

(Example 1)



———Program path after 1/2 scaling

-----Tool path after 1/2 scaling

- - - -Program path when scaling is not applied

.....Tool path when scaling is not applied

<Program>

```

N01 G92 X0 Y0 Z0;
N02 G90 G51 X-100. Y-100. P0.5;
N03 G00 G43 Z-200. H02;
N04 G41 X-50. Y-50. D01;
N05 G01 Z-250. F1000;
N06 Y-150. F200;
N07 X-150.;
N08 G02 Y-50. J50.;
N09 G01 X-50.;
N10 G00 G49 Z0;
N11 G40 G50 X0 Y0;
N12 M02;
    
```




Relationship with other functions

- (1) G27 reference position check command
When G27 is commanded during scaling, scaling is canceled at completion of the command.
- (2) Reference position return command (G28, G29, G30)
When the G28, G30 or reference position return command is issued during scaling, scaling is canceled at the intermediate point and the axis returns to the reference position. When the midpoint is to be ignored, the axis returns to the reference point directly.
When G29 is commanded during scaling, scaling is applied to the movement after the midpoint.
- (3) G60 (unidirectional positioning) command
If the G60 (unidirectional positioning) command is given during scaling, scaling is applied to the final positioning point and is not applied to the creep amount. Namely, the creep amount is uniform regardless of scaling.
- (4) Workpiece coordinate system switching
When the workpiece coordinate system is switched during scaling, the scaling center is shifted by the difference between the offset amounts of the new and old workpiece coordinate systems.
- (5) During figure rotation
When figure rotation is commanded during scaling, scaling is applied to the center of the figure rotation and the rotation radius.
- (6) Scaling command in figure rotation subprogram
By commanding the scaling in the subprogram of the figure rotation, scaling can be applied only to the shape designated by the subprogram, not to the rotation radius of the figure rotation.
- (7) During coordinate rotation
When scaling is commanded during coordinate rotation, the scaling center rotates. Scaling is executed at that rotated scaling center.
- (8) G51 command
When the G51 command is issued during the scaling mode, the axis whose center was newly specified is also made valid for scaling. Also, the factor under the latest G51 command is made valid.



Precautions

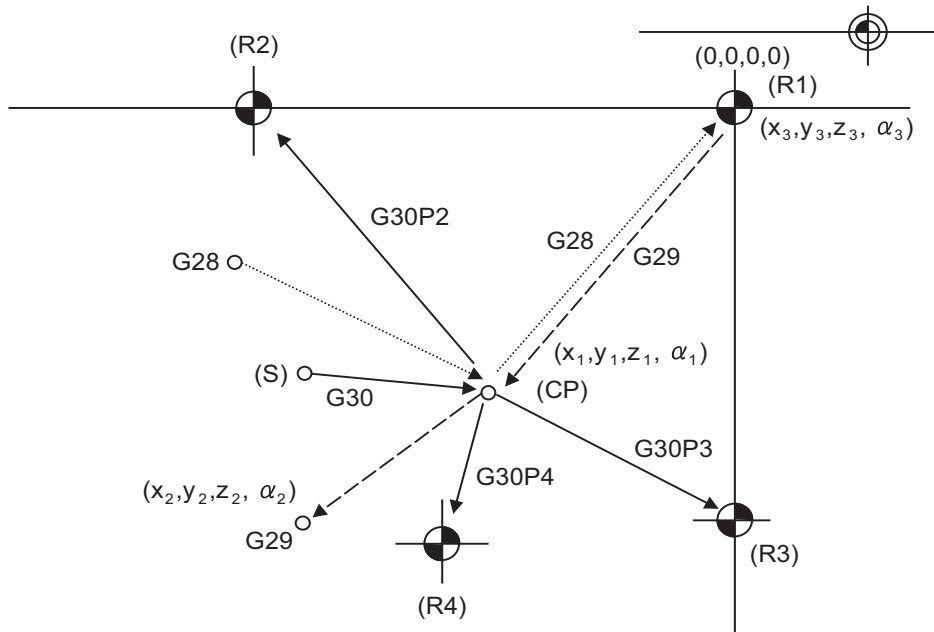
- (1) Scaling is not applied to the compensation amounts of tool radius compensation, tool position compensation, tool length compensation and the like. (Compensation is calculated for the shape after scaling.)
- (2) Scaling is valid for only the movement command in automatic operation. It is invalid for manual movement.
- (3) For X, Y and Z, scaling is valid for only the specified axes and is not applied to unspecified axes.
- (4) When an arc is commanded and scaling is valid for one of the two axes configuring the arc plane, a program error (P70) will occur.
- (5) When M02 or M30 is commanded, or when NC reset is carried out during the scaling mode, the mode switches to a cancel mode.
- (6) When the coordinate system is shifted (G92, G52 command) during scaling, the scaling center is also shifted by the difference amount.
- (7) If manual interruption is made during scaling, manual ABS selection is ignored for the movement followed by an incremental value command and operation performed is the same as in manual ABS OFF.

20.12 Reference Position (Zero Point) Return; G28, G29



Function and purpose

After the commanded axes have been positioned by G00, they are returned respectively at rapid traverse to the first reference position when G28 is commanded. By commanding G29, the axes are first positioned independently at high speed to the G28 or G30 intermediate point and then positioned by G00 to the commanded position.



(R1) 1st reference position

(R3) 3rd reference position

(S) Start point

(R2) 2nd reference position

(R4) 4th reference position

(CP) Intermediate point



Command format

G28 Xx1 Yy1 Zz1 αα1; ... Automatic reference position return

X, Y, Z, α

Coordinate value of the intermediate point (α is an additional axis)

G29 Xx2 Yy2 Zz2 αα2; ... Start point return

X, Y, Z, α

Coordinate value of the end point (α is an additional axis)



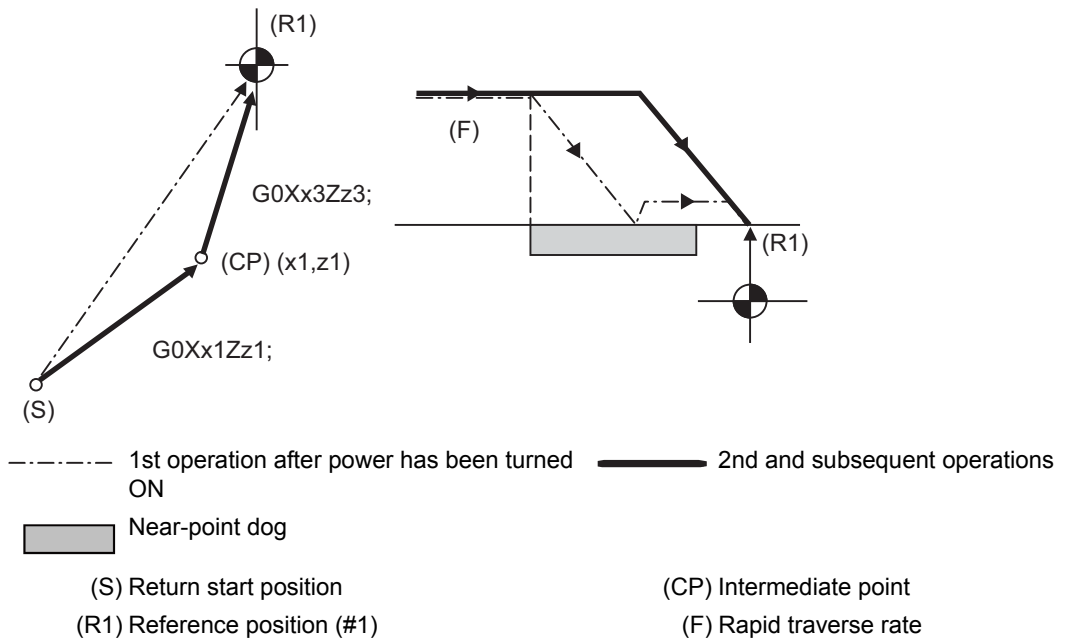
Detailed description

- (1) The G28 command is equivalent to the following:
 G00 Xx1 Yy1 Zz1 α α1;
 G00 Xx3 Yy3 Zz3 α α3;
 where Xx3 Yy3 Zz3 α3 are the coordinate values of the reference position that are set in parameters "#2037 G53ofs" for the distance from the basic machine coordinate system zero point as specified by the MTB.
- (2) After the power has been switched on, the axes that have not been subject to manual reference position return are returned by the dog type of return just as with the manual type. In this case, the return direction is regarded as the command sign direction. If the return type is straight-type return, the return direction will not be checked. For the second and subsequent returns, the return is made at high speed to the reference (zero) position that was stored at the first time and the direction is not checked.
- (3) When reference position return is completed, the zero point arrival output signal is output and also #1 appears at the axis name line on the setting and display unit screen.
- (4) The G29 command is equivalent to the following:
 G00 Xx1 Yy1 Zz1 α α1;
 G00 Xx2 Yy2 Zz2 α α2;
 Rapid traverse (non-interpolation type) applies independently to each axis for the positioning from the reference position to the intermediate point.
 In this case, x1 y1 z1 and α1 are the coordinate value of the G28 or G30 intermediate point.
- (5) Program error (P430) occurs when G29 is executed without executing automatic reference position (zero point) return (G28) after the power has been turned ON.
- (6) When the Z axis is canceled, the movement of the Z axis to the intermediate point will be ignored, and only the position display for the following positioning will be executed. (The machine itself will not move.)
- (7) The intermediate point coordinates (x1, y1, z1, α1) of the positioning point are assigned by the position command modal. (G90, G91).
- (8) G29 is valid for either G28 or G30 but the commanded axes are positioned after a return has been made to the latest intermediate point.
- (9) The tool compensation will be canceled during reference position return unless it is already canceled, and the compensation amount will be cleared.
- (10) The intermediate point can be ignored by parameter "#1091 Ignore middle point" setting.
- (11) Control from the intermediate point to the reference position is ignored for reference position return in the machine lock status. When the designated axis reaches as far as the intermediate point, the next block will be executed.
- (12) Mirror image is valid from the start point to the intermediate point during reference position return in the mirror image mode and the tool will move in the opposite direction to that of the command. However, mirror image is ignored from the intermediate point to the reference position and the tool will move to the reference position.
- (13) When G28/G29/G30 is commanded in single block mode, if "#1279 ext15/bit6 Enable single block stop at middle point" is set to "1", single block stop at middle point will be performed; single block stop at middle point will not be performed if set to "0".
- (14) If the mode is switched to MDI mode or reference position return mode while in a single block stop at the intermediate point, an operation error (M01 0013) occurs.
- (15) If the NC is reset while in a single block stop at middle point, the intermediate point for G29 start position return will not be updated.
- (16) If a miscellaneous function is commanded in the same block, the miscellaneous function completion waiting point will be the end of commanded movement, instead of the intermediate point.
- (17) If the PLC interrupt operation is operated while in a single block stop at the intermediate point, an operation error (M01 0129) occurs.

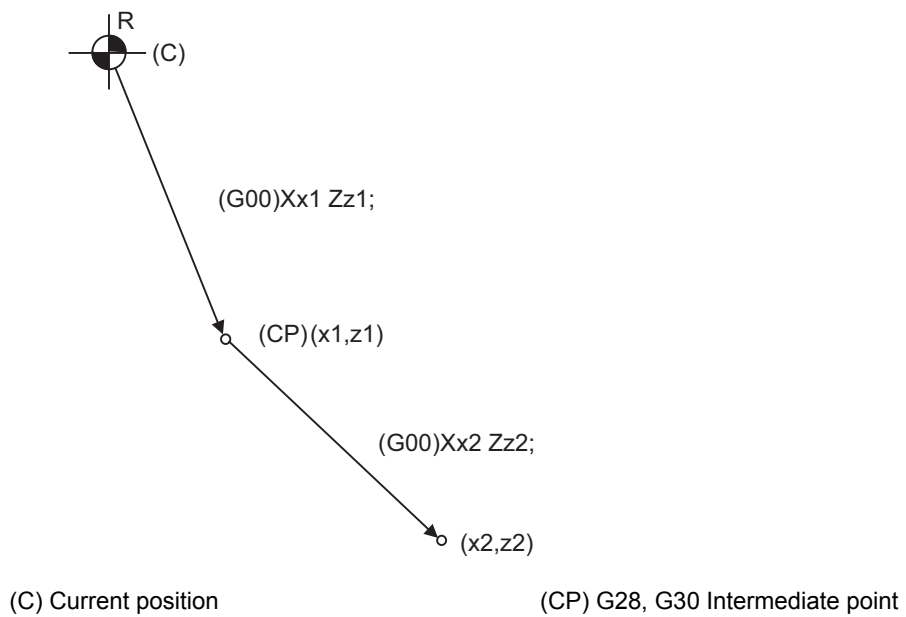


Program example

(Example 1) G28 Xx1 Zz1 ;

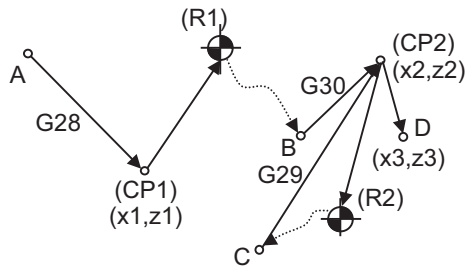


(Example 2) G29 Xx2, Zz2 ;



20 Coordinate System Setting Functions

(Example 3) G28 Xx1 Zz1 ;
 : (From point A to 1st reference position)
 :
 G30 Xx2 Zz2 ;
 : (From point B to 2nd reference position)
 :
 G29 Xx3 Zz3 ;
 (From point C to point D)



(CP1) Old intermediate point
 (CP2) New intermediate point
 (R1) Reference position (#1)

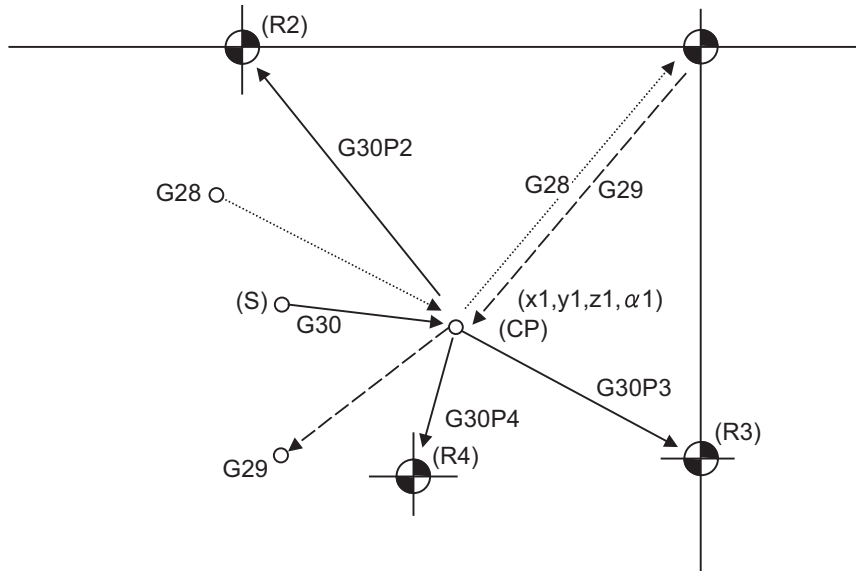
(R2) 2nd reference position (#2)

20.13 2nd, 3rd, and 4th Reference Position (Zero Point) Return ; G30



Function and purpose

The tool can return to the second, third, or fourth reference position by specifying G30 P2 (P3 or P4).



(S) Start point

(R2) 2nd reference position

(R4) 4th reference position

(CP) Intermediate point

(R3) 3rd reference position



Command format

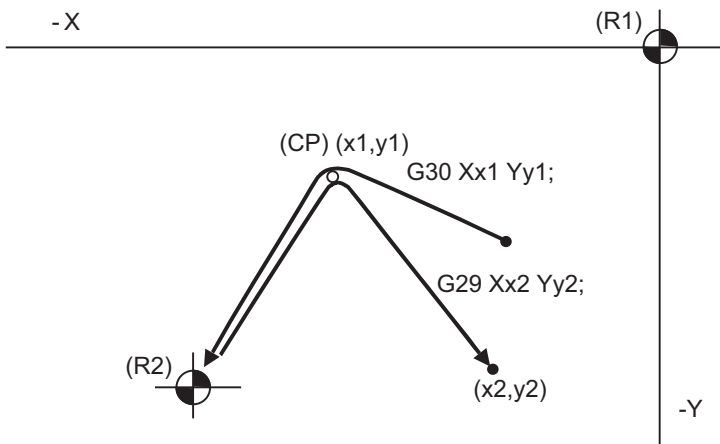
G30 P2(P3,P4)Xx1 Yy1 Zz1 αα1;

X, Y, Z, α	Coordinate value of the intermediate point (α is an additional axis)
P	Reference position No. P2: 2nd reference position return P3: 3rd reference position return P4: 4th reference position return



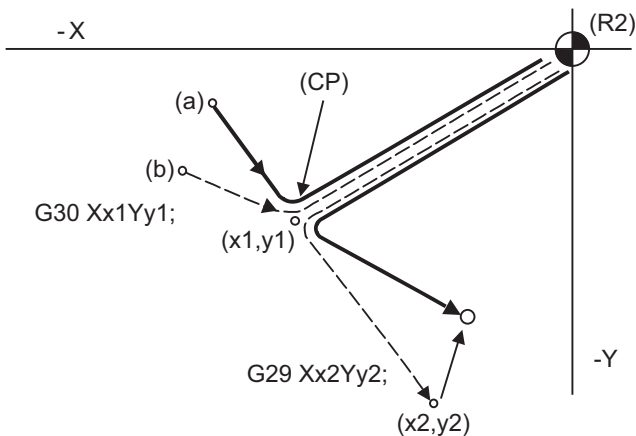
Detailed description

- (1) The 2nd, 3rd, or 4th reference position return is specified by P2, P3, or P4.
A command without P or with other designation method will return the tool to the 2nd reference position.
- (2) In the 2nd, 3rd, or 4th reference position return mode, as in the 1st reference position return mode, the tool returns to the 2nd, 3rd, or 4th reference position via the intermediate point specified by G30.
- (3) The 2nd, 3rd, and 4th reference position coordinates refer to the positions specific to the machine, and these can be checked with the setting and display unit.
- (4) If G29 is commanded after completion of returning to the 2nd, 3rd, and 4th reference position, the intermediate position used last is used as the intermediate position for returning by G29.



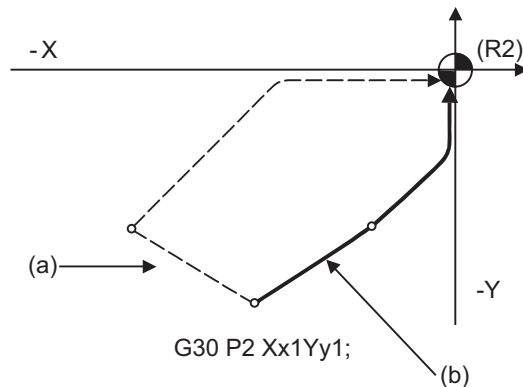
(CP) Intermediate point
(R2) 2nd reference position
(R1) 1st reference position

- (5) With reference position return on a plane during compensation, the tool moves without tool radius compensation from the intermediate point as far as the reference position. With a subsequent G29 command, the tool move without tool radius compensation from the reference position to the intermediate point and it moves with such compensation until the G29 command from the intermediate point.



(a) Tool nose center path
(b) Program path
(CP) Intermediate point
(R2) 2nd reference position
(R1) 1st reference position

- (6) The tool length compensation amount for the axis involved is canceled after the 2nd, 3rd and 4th reference position return.
- (7) With second, third and fourth reference position returns in the machine lock status, control from the intermediate point to the reference position will be ignored. When the designated axis reaches as far as the intermediate point, the next block will be executed.
- (8) With second, third and fourth reference position returns in the mirror image mode, mirror image will be valid from the start point to the intermediate point and the tool will move in the opposite direction to that of the command. However, mirror image is ignored from the intermediate point to the reference position and the tool moves to the reference position.



(a) X-axis mirror image

(b) No mirror image

(R2) 2nd reference position

- (9) If the 2nd, 3rd or 4th reference position is changed while G30 zero point return operation is in pause due to an interlock, "M01 Operation Error" occurs.
- (10) When G28/G29/G30 is commanded in single block mode, if "#1279 ext15/bit6 Enable single block stop at middle point" is set to "1", single block stop at middle point will be performed; single block stop at middle point will not be performed if set to "0".
- (11) If the mode is switched to MDI mode or reference position return mode while in a single block stop at the intermediate point, an operation error (M01 0013) occurs.
- (12) If the NC is reset while in a single block stop at middle point, the intermediate point for G29 start position return will not be updated.
- (13) If a miscellaneous function is commanded in the same block, the miscellaneous function completion waiting point will be the end of commanded movement, instead of the intermediate point.
- (14) If the PLC interrupt operation is operated while in a single block stop at the intermediate point, an operation error (M01 0129) occurs.

20.14 Tool Change Position Return; G30.1 - G30.6



Function and purpose

By specifying the tool change position in a parameter "#8206 tool change" and also specifying a tool change position return command in a machining program, the tool can be changed at the most appropriate position. The axes that are going to return to the tool change position and the order in which the axes begin to return can be changed by commands.



Command format

Tool change position return

G30.n ;

n = 1 to 6 Specify the axes that return to the tool change position and the order in which they return.



Detailed description

Commands and return order are given below.

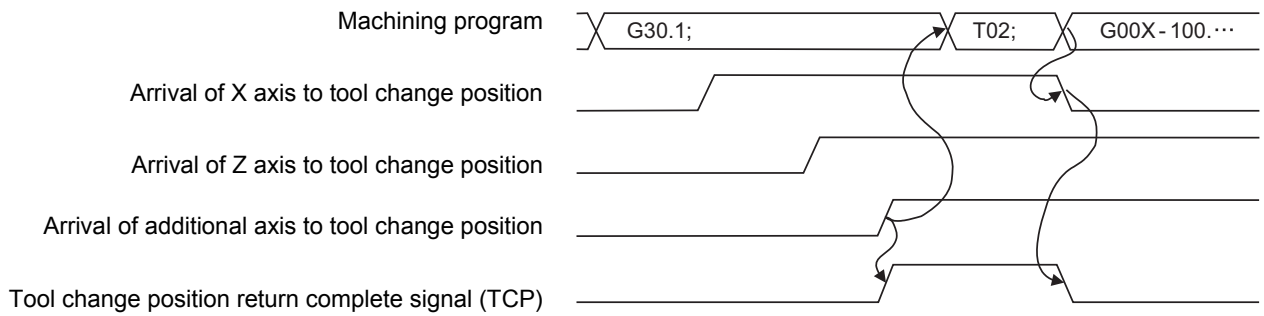
Command	Return order
G30.1	Z axis -> X axis - Y axis (-> additional axis)
G30.2	Z axis -> X axis -> Y axis (-> additional axis)
G30.3	Z axis -> Y axis -> X axis (-> additional axis)
G30.4	X axis -> Y axis - Z axis (-> additional axis)
G30.5	Y axis -> X axis - Z axis (-> additional axis)
G30.6	X axis - Y axis - Z axis (-> additional axis)

<Note>

•An arrow (->) indicates the order of axes that begin to return. A hyphen (-) indicates that the axes begin to return simultaneously. (Example: "Z axis -> X axis - Y axis" indicates that the Z axis returns to the tool change position, then the X axis and Y axis do at the same time.)

- (1) Whether the tool exchange position return for the additional axis is enabled or disabled depends on the MTB specifications (parameter "#1092 Tchg_A").
For the order for returning to the tool change position, the axes return after the standard axis completes the return to the tool change position (refer to above table). The additional axis alone cannot return to the tool change position.
- (2) If the axis address is commanded in the same block as the tool change position return command, a program error (P33) will occur.
- (3) After all necessary tool change position return is completed by a G30.n command, tool change position return complete signal TCP (XC93) is turned ON. When an axis out of those having returned to the tool change position by a G30.n command leaves the tool change position, the TCP signal is turned OFF. With a G30.1 command, for example, the TCP signal is turned on when the Z axis has reached the tool change position after the X and Y axes have reached the tool change position (in addition, after the additional axis has reached the tool change position if additional axis tool change position return is valid). The TCP signal is then turned OFF when the X, Y, or Z axis leaves the position. If tool change position return for added axes is on with parameter "#1092 Tchg_A", the TCP signal is turned ON when the added axis or axes have reached the tool change position after the standard axes did. It is then turned OFF when one of the X, Y, Z, and added axes leaves the position.

[TCP signal output timing chart] (G30.3 command with tool change position return for additional axes set ON)

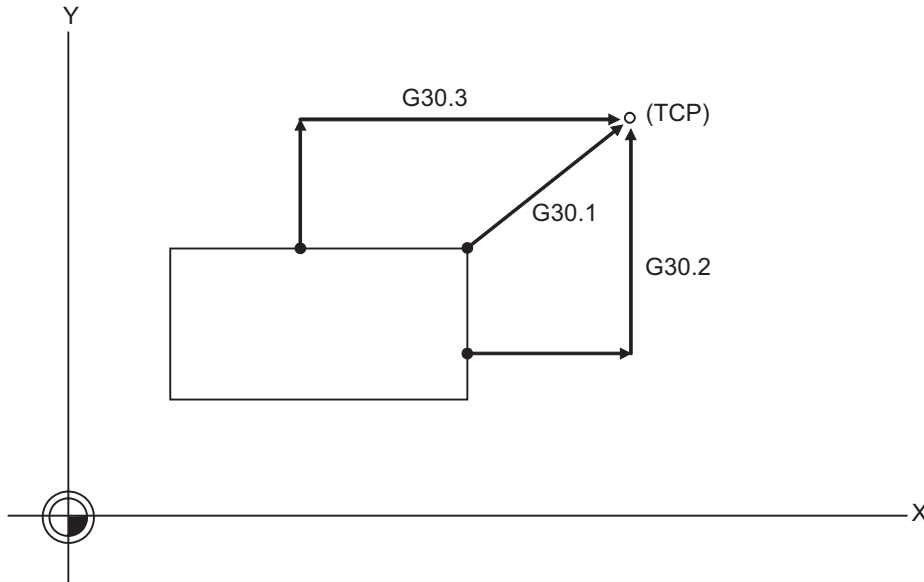


- (4) When a tool change position return command is issued, tool offset data such as for tool length offset and tool radius compensation for the axis that moved is canceled.
- (5) This command is executed by dividing blocks for every axis. If this command is issued during single-block operation, therefore, a block stop occurs each time one axis returns to the tool change position. To make the next axis tool change position return, therefore, a cycle start needs to be specified.



Operation example

The figure below shows an example of how the tool operates during the tool change position return command. (Only operations of X and Y axes in G30.1 to G30.3 are figured.)



TCP : Tool change position

- (1) G30.1 command: The Z axis returns to the tool change position, then the X and Y axes simultaneously do the same thing. (If tool change position return is on for an added axis, the added axis also returns to the tool change position after the X, Y and Z axes reach the tool change position.)
- (2) G30.2 command: The Z axis returns to the tool change position, then the X axis does the same thing. After that, the Y axis returns to the tool change position. (If tool change position return is on for an added axis, the added axis also returns to the tool change position after the X, Y and Z axes reach the tool change position.)
- (3) G30.3 command: The Z axis returns to the tool change position, then the Y axis does the same thing. After the Y axis returns to the tool change position, the X axis returns to the tool change position. (If tool change position return is on for an added axis, the added axis also returns to the tool change position after the X, Y and Z axes reach the tool change position.)
- (4) G30.4 command : The X axis returns to the tool change position, then the Y axis and Z axis simultaneously do the same thing. (If tool change position return is on for an added axis, the added axis also returns to the tool change position after the X, Y and Z axes reach the tool change position.)
- (5) G30.5 command : The Y axis returns to the tool change position, then the X and Z axes return to the tool change position simultaneously. (If tool change position return is on for an added axis, the added axis also returns to the tool change position after the X, Y and Z axes reach the tool change position.)
- (6) G30.6 command : The X, Y and Z axes return to the tool change position simultaneously. (If tool change position return is on for an added axis, the added axis also returns to the tool change position after the X, Y and Z axes reach the tool change position.)

20.15 Reference Position Check; G27



Function and purpose

This command first positions the tool at the position assigned by the program and then, if that positioning point is the 1st reference position, it outputs the reference position arrival signal to the machine in the same way as with the G28 command. Therefore, when a machining program is prepared so that the tool will depart from the 1st reference position and return to the 1st reference position, it is possible to check whether the tool has returned to the reference position after the program has been run.



Command format

X__Y__Z__P__ ; ... Check command

X Y Z	Return control axis
P	Check No. P1: 1st reference position check P2: 2nd reference position check P3: 3rd reference position check P4: 4th reference position check



Detailed description

- (1) If the P command has been omitted, the 1st reference position will be checked.
- (2) The number of axes whose reference positions can be checked simultaneously depends on the number of axes which can be controlled simultaneously.
Note that the display shows one axis at a time from the final axis.
- (3) An alarm will occur if the reference position is not reached after the command is completed.

Protection Function

21.1 Stroke Check before Travel; G22/G23



Function and purpose

By commanding the boundaries from the program with coordinate values on the machine coordinate system, machine entry into that boundary can be prohibited. This can be set only for the three basic axes.

While the normal stored stroke limit function stops entry just before the prohibited area, this function causes a program error (P452) to occur before the actual travel starts if a program block includes such a command as follows:

- the start or end point of the travel command is within the prohibited area.
- the linear or circular travel path enters the prohibited area.

There are the following two methods for stroke check before travel. Whether or not this function specification is provided and either method is valid varies depending on the MTB specifications (parameter "#12057 OT_prechkON").

(1) Stroke check before travel by G code command (G22/G23)

Designate the entry prohibited areas (X, Y, and Z) with the G code G22, and enable the stroke check before travel.

A program error (P452) occurs if there is a block in which start or end point of the travel is within the prohibited area, or the linear or circular travel path enters the prohibited area.

The check function is turned "OFF" when the G code G23 is commanded.

(2) "Stroke check before travel in stored stroke limit area" by parameter setting

A program error (P452) occurs in the block in which the travel command end point is within the prohibited area specified by the stored stroke limit I, IB, IC, II, or IIB function even if the G code G22 is not commanded.

For details, refer to "21.1.1 Stroke Check before Travel in Stored Stroke Limit Area".



Command format

Stroke check before travel ON

G22 X_ Y_ Z_ I_ J_ K_ ;

Stroke check before travel cancel

G23 ;

X Y Z	Coordinates of upper point (basic axis name and its coordinate position)
I J K	Coordinates of lower point (I, J, K address and its coordinate position)

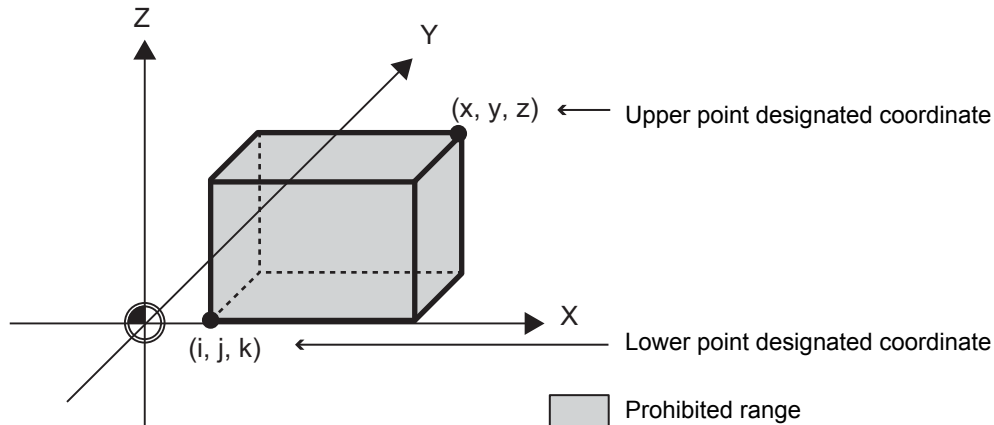
Note

- (1) In the following command format, the basic axes are X, Y and Z. Command the base axis designated in the parameters #1026 to #1028.



Detailed description

- (1) The inner side of the boundary commanded with the upper position coordinate and the lower position coordinate is the prohibited area.
- (2) If the command is omitted, "0" will be set for the address.
- (3) The area designated with this function is different from the area designated with the stored stroke limit. However, the area enabled by both functions will be the actual valid movement range.



<Note>

- The upper point and lower point are commanded with coordinate on the machine coordinate system.



Precautions

- (1) This function is valid only when starting the automatic operation. When interrupted with manual absolute OFF, the prohibited area will also be shifted by the interrupted amount.
- (2) A program error (P452) occurs if there is a block in which start or end point of the travel is within the prohibited area, or the linear or circular travel path enters the prohibited area.
- (3) Stroke check before travel will not be carried out for the axes having the same coordinates set for the upper point and the lower point.
- (4) The stroke check before travel is carried out with the tool center coordinate values.
- (5) If "G23X_Y_Z_;"etc., is commanded, the command will be interpreted as "G23;X_Y_Z;"(2 blocks). Thus, the stroke check before travel will be canceled, then movement will take place with the previous movement modal.
- (6) During automatic reference position return, the check will not be carried out from the intermediate point to the reference position. With G29, when moving from the start point to intermediate point, the check will not be carried out.
- (7) If there is an address that is not used in one block, a program error (P32) will occur.
- (8) The rotary axis is not available because the correct judgment is not performed.
Do not designate the prohibited area for the rotary axis.
- (9) When using a circular command, of which the radius is different at the start and end points, such as the spiral interpolation command, perform the stroke check before travel for the start-point radius path.
- (10) The command by this function is not applied to the parameters of stored stroke limit II and IIB.
- (11) When the axis addresses other than three base axes are commanded to the same block as for the stroke check before travel ON command (G22), the commanded axes will travel.

21.1.1 Stroke Check before Travel in Stored Stroke Limit Area



Function and purpose

When "#12057 OT_prechkON" is set to "1", stroke check before travel is carried out for the stored stroke limit prohibited area.

The stroke check before travel is available only for the axes for which the stored stroke limit prohibited area is designated. Stroke check before travel will not be carried out for the axes with no prohibited area designated.

When a command is issued to move to the stored stroke limit prohibited area, a program error (P452) occurs before the block movement starts, and the axis travel is not carried out.

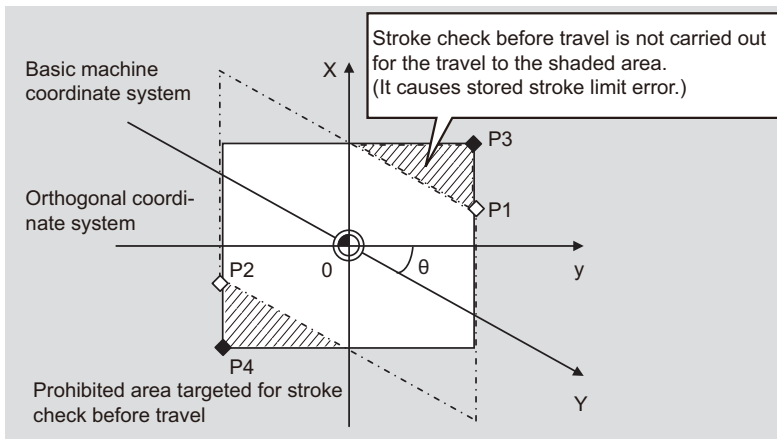


Relationship with other functions

- (1) Stroke check before travel will not be carried out during the following modes.
The axis travel stop by the stored stroke limit function is available.

G code	Function	G code	Function
G07.1	Cylindrical interpolation	G05 P4	Direct command mode
G12.1/G112	Polar coordinate interpolation	G05.1 Q1	High-speed high-accuracy control I
G12.1	Milling interpolation	G05 P10000	High-speed high-accuracy control II
G54.4	Workpiece installation error compensation	G05 P20000	High-speed high-accuracy control III
G43.4/G43.5	Tool center point control	G05.1 Q2	Spline interpolation
G68.2/G68.3	Inclined surface machining command	G06.2	NURBS interpolation
G05 P1	High-speed machining mode I	G30.1 to G30.6	Tool exchange position return
G05 P2	High-speed machining mode II	-	PLC axis control
G68	3-dimensional coordinate conversion		

- (2) For the axis for inclined axis control (inclined axis or base axis), stroke check before travel is carried out only when it is designated in the orthogonal coordinate system (*1).
Stroke check before travel is not carried out in the prohibited area on the machine coordinate system.
The axis travel stop by the stored stroke limit function is available.



(*1) To carry out the stored stroke limit of the axis for inclined axis control in the orthogonal coordinate system, the following settings are required.

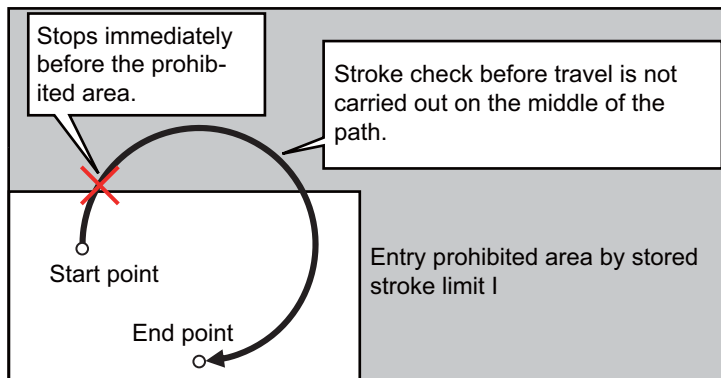
Range setting parameter	Enabling conditions	Remarks
#2061 OT_1B- #2062 OT_1B+	♦#2063 OT_1Btype = 3 ♦#2061 OT_1B- ≠ #2062 OT_1B+ ♦Zero point return complete	The stored stroke limit IC specification is required.

- (3) Whether stroke check before travel is enabled by the skip function (G31 or G31.n) or the automatic tool length measurement (G37) depends on the MTB specifications (parameter "#12058 OT_prechkTYPE"). When #12058=1, stroke check before travel is carried out for the skip function or the automatic tool length measurement.



Precautions

- (1) If the restarting point is in the prohibited area when the program restarts, a program error (P452) occurs by the automatic restart position return operation.
- (2) Issuing a command (fixed cycle, block inserted by tool radius compensation, etc.) that is divided into multiple operations in a single block causes a program error (P452) before the program starts a travel by which the end point enters the prohibited area.
- (3) Stroke check before travel is carried out also during machine lock.
- (4) The tool path of the travel command is not targeted for stroke check before travel. An operation error (M01 0007) occurs and processing stops by the stored stroke limit function just before the prohibited area.



- (5) If the end point at block restart enters the prohibited area by the manual interruption after the axis travel has stopped by field hold, it does not cause a program error (P452) by the stroke check before travel.
- (6) A travel from the intermediate point to the reference point in the automatic reference position return G28/G30 and a travel from the reference point to the intermediate point in G29 are not targeted for stroke check before travel.
- (7) If the stroke check before travel specification by parameter is not provided, a program error (P451) occurs at the timing of the automatic operation start.
- (8) To rewrite the parameter #12057 to "1" with the parameter input by program (G10L70), the stroke check before travel specification is required. If the specification is not provided, a program error (P451) occurs when an attempt is made to rewrite the parameter #12057 to "1" with the parameter input by program (G10L70).

21.2 Enable Interfering Object Selection Data; G186



Function and purpose

Sixteen interfering objects to be checked in the interference check III are preset by the MTB (R register or system variables).

The interfering object selection is enabled by the "Interference check III: Enable interfering object selection data" signal (Y769) or the "Enable interfering object selection data" command (G186) after the target interfering object has been selected.

When the "Interference check III mode" signal (Y76A) is set to ON after the interfering object selection has been enabled, the interference check starts.

Refer to the "PLC Interface Manual" for the R register and commands issued by the PLC device.

For details on the system variables, refer to "23.29 System Variables (Interfering Object Selection)".

This section describes the "Enable interfering object selection data" command (G186).



Command format

"Enable interfering object selection data" command

G186;



Detailed description

Consistency check between interfering object definition and interfering object selection

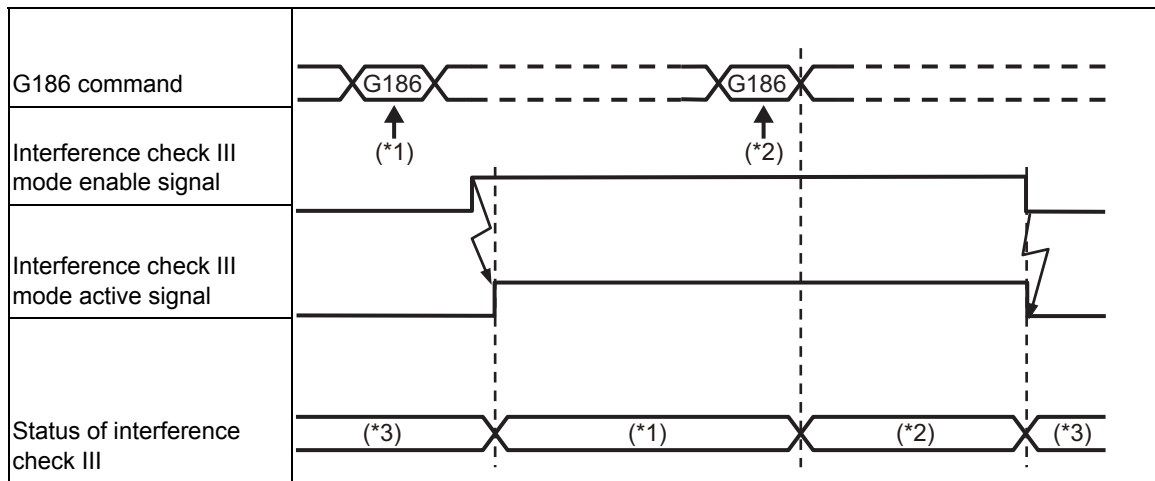
- (1) When the "Enable interfering object selection data" command (G186) or the "interference check III: Enable interfering object selection data" signal is set to ON, the consistency between the interfering object definition and interfering object selection is checked.
- (2) If the consistency check causes an operation error, all axes in all part systems will stop.
An operation error can be remedied by redefining the interfering object data (*1) or resetting all part systems (except for sub part system 2).
(*1) After correcting the interfering object data, issue the "Enable interfering object selection data" signal or "Enable interfering object selection data" command (G186).
- (3) The manual operation and automatic operation are not available until all the part systems (except for subpart system 2) are reset.
- (4) In the case the alarm occurs due to the consistency check, the interfering data will not be updated. For the interference check between interfering objects, the interfering data enabled last time is continuously used.

Interference check III mode enable command

While the interference check III mode signal is set to ON after the "Enable interfering object selection data" signal or the "Enable interfering object selection data" command (G186) has been executed, the interference between interfering objects is checked. While the interference check III is being executed, the interference check III mode active signal is turned ON.

After the NC power is turned ON, if the interference check III mode signal is turned ON without executing the "Enable interfering object selection data" signal or the "Enable interfering object selection data" command (G186) even once, an operation error (M03 1001) will occur.

Timing chart at execution of G186



(*1) The first interfering data pattern is set by the G186 command.

The interference check III function executes check processing based on the first data pattern setting.

(*2) The second interfering data pattern is set by the G186 command.

The interference check III function executes check processing based on the second data pattern setting.

(*3) The interference check III function is not executed.



Relationship with other functions

Manual arbitrary reverse run

The program cannot be run backward prior to the "Enable interfering object selection data" command (G186).

Arbitrary reverse run

If the "Enable interfering object selection data" command (G186) is run backward, the interference data at the reverse run is enabled, instead of returning to the interference data at forward run.



Precautions

- (1) The high-speed high-accuracy control function (high-speed machining mode, high-accuracy control, spline interpolation, etc.) generates a path appropriate for the tolerance amount to execute a machining program commanded with fine segments at high speed and smoothly. Thus, a difference arises between the path on which the interference check III is performed and the path on which the tool actually passes. When using the interference check III together with the high-speed high-accuracy control (high-speed machining mode, high-accuracy control, spline interpolation, etc.), define an interfering object (solid) with the clearance amount to suit the path difference that occurs depending on the tolerance amount.
- (2) The axis that is stopped when an operation error (M03 0001) or (M03 0002) is detected depends on the MTB specifications (parameter "#1444 otsys" (OT all-part-system stop enable/disable selection).
When "0" is set, all the axes in the part system which controls the axes set to "interfering object I/J/K control axis" and "I/J/K rotary control axis" in the interfering object definition will stop.
When "1" is set, all axes in all part systems will stop.
- (3) If an operation error (M03 0002) is detected between the fixed interfering objects (*1), an alarm will be output to part system 1.
(*1) These refer to the interfering objects for which "interfering object I/J/K control axis" and "I/J/K rotary control axis" are not set in the interfering object definition.
- (4) If you perform the interference check III during the high-speed simple program check, an operation error (M03 0001) may occur at a position different from the actual operation.
- (5) If multiple interfering objects including the rotary axis setting are set as one interfering object using the interfering check III: designation of disabled interference object, only the interfering object in which a rotary axis is set will be in rotating operation, checking the interference between the interfering objects.
- (6) If an operation error (M03 0001) occurs, cancel the alarm by moving the interfering object to the retracting direction with the linear axis.
- (7) The PLC axis is not available for the interference check III. However, it is available when NC axis/auxiliary axis switching is enabled.
- (8) In the interference check III, the interference is checked in 0.1 μ m units regardless of the control unit.
- (9) At the occurrence of the operation error (M03 0001), all the axes in the part system in which the alarm has occurred will stop. If the entry to the interference alarm area is not detected by the subsequent axis travel command (manual operation/automatic operation), the operation error (M03 0001) will be cancelled and the axes will travel. Depending on the relative positional relation between the interfering objects or the feedrate of axes, the axis can travel further to the interfering direction from the stopped position (a direction to which the interfering objects interfere).
Even if the axis moves toward the interfering direction, it will stop before entering the interference alarm area.

Measurement Support Functions

22.1 Automatic Tool Length Measurement; G37



Function and purpose

These functions issue the command values from the measuring start position as far as the measurement position, move the tool in the direction of the measurement position, stop the machine once the tool has arrived at the sensor, cause the NC system to calculate automatically the difference between the coordinate values at that time and the coordinate values of the commanded measurement position and provide this difference as the tool offset amount. When offset is already being applied to a tool, it moves the tool toward the measurement position with the offset still applied, and if a further offset amount is generated as a result of the measurement and calculation, it provides further compensation of the present compensation amount.

If there is one type of offset amount at this time, and the offset amount is distinguished between tool length offset amount and wear offset amount, the wear amount will be automatically compensated.



Command format

Automatic tool length measurement command

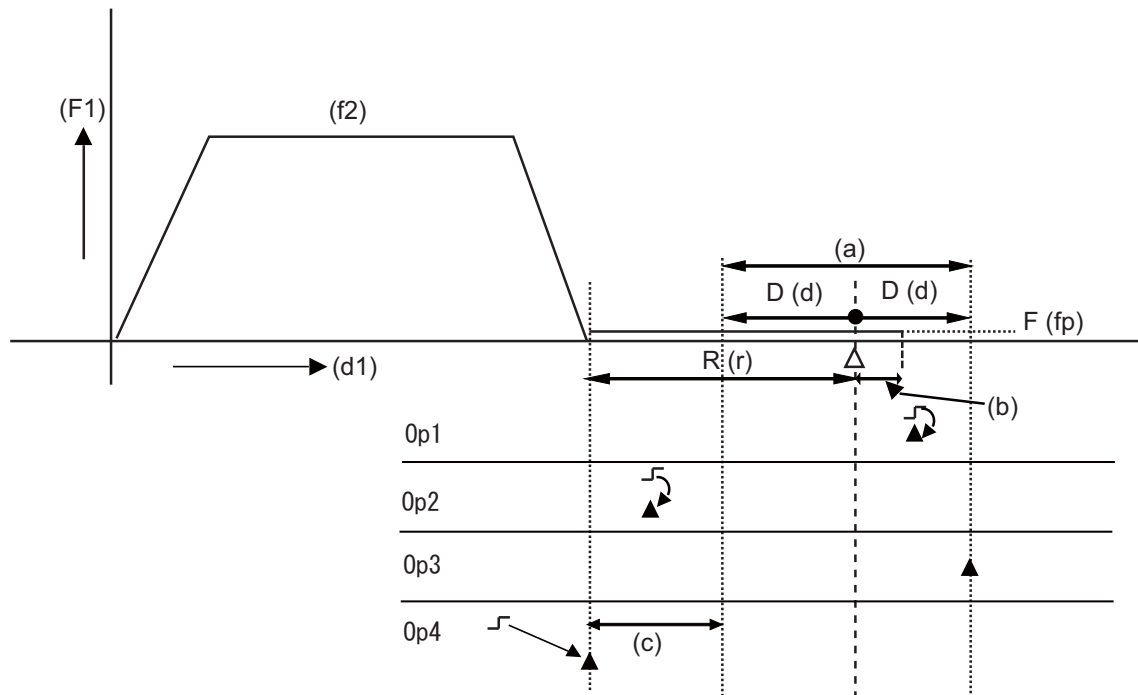
```
G37 Z__ R__ D__ F__ ;
```

Z	Measuring axis address and coordinates of measurement position ----- X,Y,Z, α (α is the additional axis.)
R	This commands the distance between the measurement position and point where the movement is to start at the measuring speed.
D	This commands the range within which the tool is to stop.
F	This commands the measuring feedrate. When R_, D_ or F_ is omitted, the value set in the parameter is used instead. <Parameter> ("Automatic tool length measurement" on the machining parameter screen) <ul style="list-style-type: none"> ◆ #8004 SPEED: 0 to 10000: 0 to 1000000 [mm/min] ◆ #8005 ZONE r: 0 to 99999.999 [mm] ◆ #8006 ZONE d: 0 to 99999.999 [mm]



Detailed description

(1) Operation with G37 command



Op1: Normal completion as it is measurement within the allowable range.

Op2: Alarm stop (P607) as it is outside of the measurement allowable range.

Op3: Alarm stop (P607) as the sensor is not detected.

Op4: Alarm stop (P607) as it is outside of the measurement allowable range. However if there is no (c) area, normal completion will occur.

(a) Measurement allowable range

(b) Compensation amount

(d1) Distance

(F1) Speed

(f2) Feedrate

(d) Measurement range

(r) Deceleration range

△ Measuring position

▲ Stop point

┌ Sensor output

(2) The sensor signal (measuring position arrival signal) is used in common with the skip signal.

(3) The feedrate will be 1mm/min if the F command and parameter measurement speed are 0.

(4) An updated offset amount is valid unless it is assigned from the following Z axis (measurement axis) command of the G37 command.

(5) Excluding the delay at the PLC side, the delay and fluctuations in the sensor signal processing range from 0 to 0.2ms.

As a result, the measuring error shown below is caused.

$$\text{Maximum measuring error [mm]} = \text{Measuring speed [mm/min]} * 1/60 * 0.2 \text{ [ms]}/1000$$

(6) The machine position coordinates at that point in time are read by sensor signal detection, and the machine will overtravel and stop at a position equivalent to the servo droop.

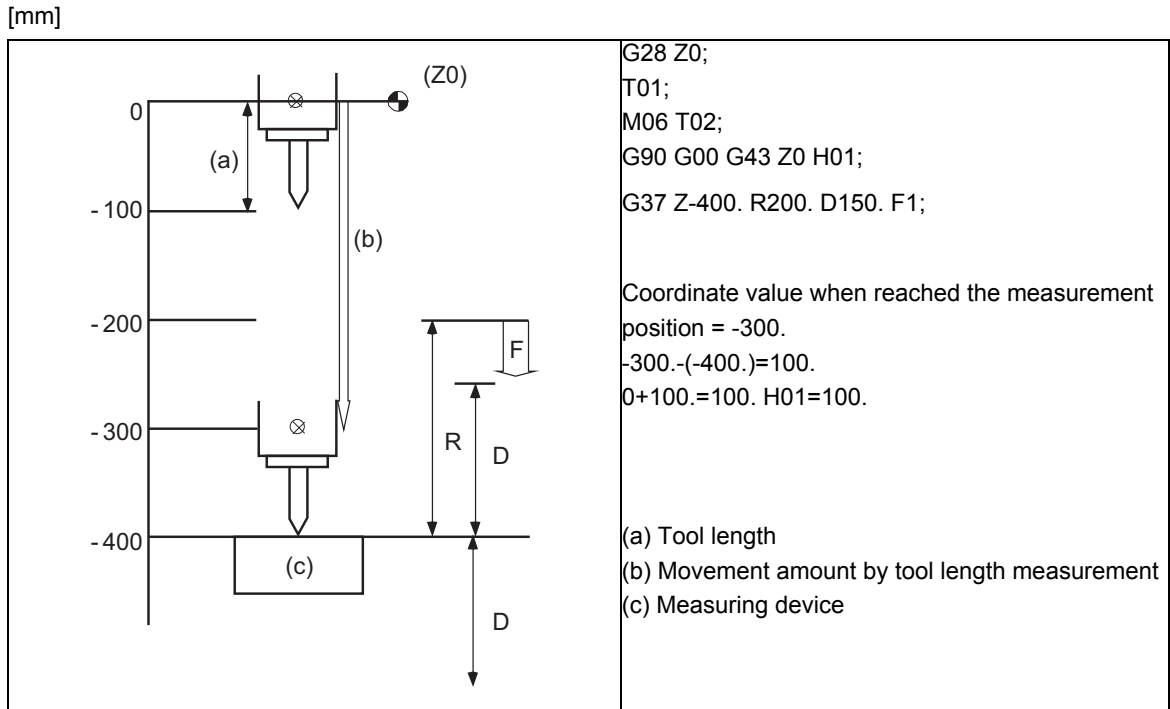
$$\text{Maximum overtravel [mm]} = \text{Measuring speed [mm/min]} * 1/60 * 1/\text{Position loop gain [1/s]}$$

The standard position loop gain is 33 (1/s).



Operation example

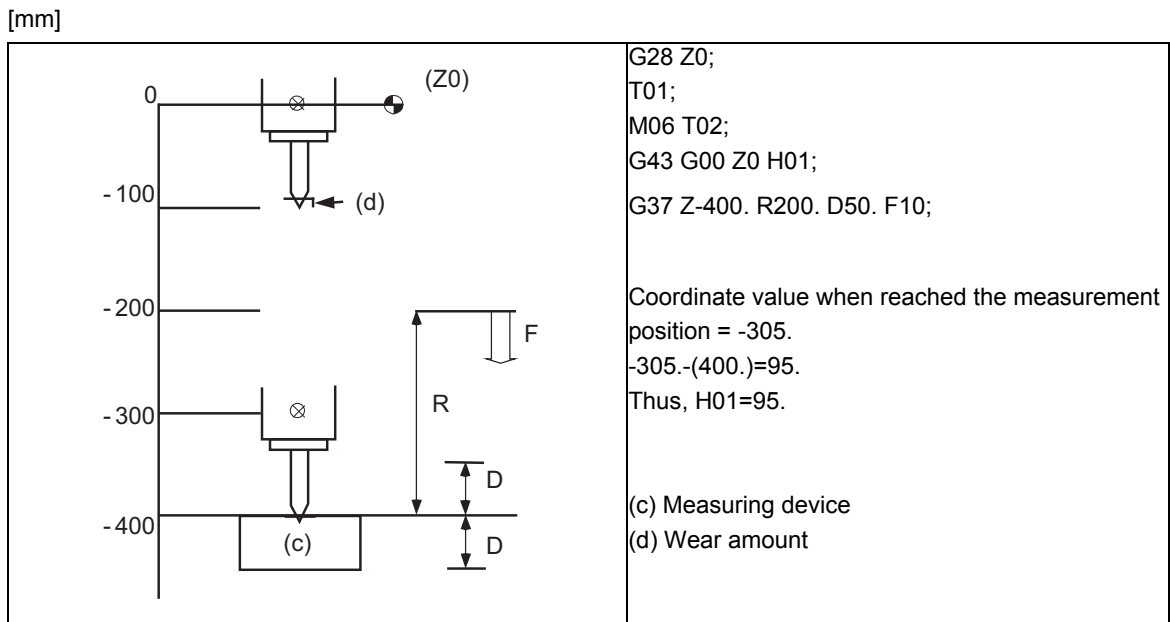
For new measurement



Note

(1) A new measurement is applied when the current tool length compensation amount is zero. Thus, length will be compensated whether or not length dimension by tool compensation memory type and length wear are differentiated.

When tool compensation is applied



Note

(1) A measurement for the wear amount is applied when the current tool length compensation amount is other than zero. Thus, length wear will be compensated if length dimension by tool compensation memory type and length wear are differentiated. If not differentiated, length dimension will be compensated.



Precautions

- (1) Program error (P600) occurs if G37 is commanded when the automatic tool length measurement function is not provided.
- (2) Program error (P604) will occur when no axis has been commanded in the G37 block or when two or more axes have been commanded.
- (3) Program error (P605) will occur when the H code is commanded in the G37 block.
- (4) Program error (P606) will occur when G43_H code is not commanded prior to the G37 block.
- (5) Program error (P607) will occur when the sensor signal is input outside the allowable measuring range or when the sensor signal is not detected even upon arrival at the end point.
- (6) When a manual interrupt is applied while the tool is moving at the measuring speed, a return must be made to the position prior to the interrupt and then operation must be resumed.
- (7) The data commanded in G37 or the parameter setting data must meet the following conditions:
| Measurement point start point | > R command or parameter r > D command or parameter d
- (8) When the D address and parameter d in (7) above are zero, the operation will be completed normally only when the commanded measurement point and sensor signal detection point coincide. Otherwise, program error (P607) will occur.
- (9) When the R and D addresses as well as parameters r and d in (7) above are all zero, program error (P607) will occur regardless of whether the sensor signal is present or not after the tool has been positioned at the commanded measurement point.
- (10) When the measurement allowable range is larger than the measurement command distance, it becomes the measurement allowable range for all axes.
- (11) When the measurement speed movement distance is larger than the measurement command distance, all axes move at the measurement speed.
- (12) When the measurement allowable range is larger than the measurement speed movement distance, the axis moves in the measurement allowable range at the measurement speed.
- (13) The automatic tool length measurement command (G37) must be commanded together with the G43H_ command that designates the offset No.
G43 H_
G37 Z_ R_ D_ F_;
- (14) If an axis other than Z is specified for the measuring axis in G37 while the parameter "#1080 DriI_Z" is set to "1", the program error (P606) occurs.

22.2 Skip Function; G31



Function and purpose

When the skip signal is input externally during linear interpolation based on the G31 command, the machine feed is stopped immediately, the coordinate value is read, the remaining distance is discarded and the command in the following block is executed.



Command format

G31 X__ Y__ Z__ α__ R__ F__ ;

X,Y,Z,α	Axis coordinate value; they are commanded as absolute or incremental values according to the G90/G91 modal when commanded. α is the additional axis.
R	Acceleration/deceleration command R0: Acceleration/deceleration time constant=0 (No automatic acceleration/deceleration after interpolation.) R1: Acceleration/deceleration time constant valid. Accelerate/decelerate with the time constant set with the parameters "#2102 skip_tL" and "#2103 skip_t1". R0 is applied when it is omitted.
F	Feedrate (mm/min)



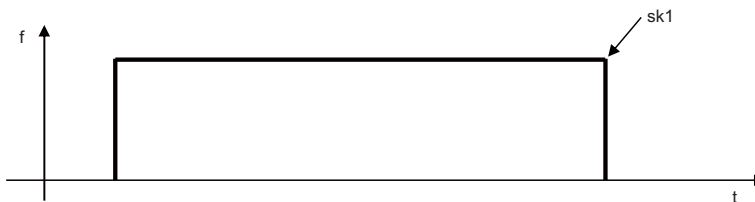
Detailed description

- (1) If an F command is programmed in the same block as G31, the commanded speed is set as the skip speed. If an F 1-digit feed command is issued to program the feedrate, F 1-digit feed is disabled. Note that, in the following cases, the skip speed and operations depend on the MTB specifications (parameter "#12022 skipF_spec/bit2").

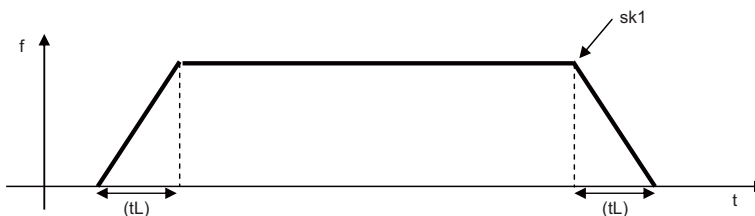
	#12022/bit2 = 0	#12022/bit2 = 1
Skip speed if an F command is not programmed in the G31 block	The value of parameter "#1174 skip_F" is used as the skip speed.	The skip speed is determined based on the modality of F when G31 is executed.
	A program error (P603) will also occur if the value of parameter "#1174 skip_F" is "0".	A program error (P62) will also occur if the value of F modality is "0".
Mode of commanded speed	Only feed per minute mode is available. Feed per minute mode is enabled even in feed per revolution mode.	Follows the mode (Feed per minute/Feed per revolution) that is active when G31 is executed.
Modality of F command	The F modal is not updated even if the G31 block contains an F command.	The F modal that is updated by an F command in the G31 block varies depending on the mode (Feed per minute/Feed per revolution) that is active when G31 is executed.

- (2) The maximum speed of G31 command is determined by the machine specification.
- (3) When R0 is commanded or the R command is omitted, the step acceleration/deceleration will be applied to G31 block after the interpolation without performing the automatic acceleration/deceleration. When R1 is commanded, the automatic acceleration/deceleration will be performed according to the cutting feed acceleration/deceleration mode set by the parameter "#2003 smgst" with the time constant set by the parameter "#2102 skip_tL" and "#2103 skip_t1". Even if G1 constant inclination acceleration/deceleration (the parameter "#1201 G1_acc" is set to "1") is valid, the time constant acceleration and deceleration will be performed.
- (4) When the R1 is commanded with the acceleration and deceleration command, the automatic acceleration and deceleration will be performed after the interpolation even if the skip single is input. Note that if the value of the parameter "#2102 skip_tL" and "#2103 skip_t1" are large, the movement will not stop immediately.

Acceleration/deceleration when R0 is commanded or R is omitted



Acceleration/deceleration when R1 is commanded



(sk1) Skip signal

(tL) Skip time constant

- (5) Command the acceleration/deceleration command (R0/R1) whenever G31 is commanded. If R0/R1 has not been commanded, or anything other than R0/R1 has been commanded, the acceleration/deceleration time constant is assumed to "0" (R0), and automatic acceleration/deceleration after interpolation will not be performed.

- (6) When G31 is commanded, the stop conditions (feed hold, interlock, override zero and stroke end) are valid. External deceleration is also valid.
For the validity of the following various functions, refer to the MTB specifications.
- Cutting feed override (parameter "#12022 skipF_spec/bit0")
 - Dry run (parameter "#12022 skipF_spec/bit1")
- (7) The G31 command is unmodal and it needs to be commanded each time.
- (8) If the skip command is input at the start of the G31 command, the G31 command will be completed immediately. When a skip signal has not been input until the completion of the G31 block, the G31 command will also be completed upon completion of the movement commands.
- (9) If the G31 command is issued during tool radius compensation or nose R compensation, program error (P608) will occur.
- (10) When there is no F command in the G31 command and the parameter speed is also zero, the program error (P603) will occur.
- (11) With machine lock or with the Z axis cancel switch ON when only the Z axis is commanded, the skip signal will be ignored and execution will continue as far as the end of the block.

Readout of skip coordinates

The coordinate positions for which the skip signal is input are stored in the system variables #5061 (1st axis) to #506n (n-th axis), so these can be used in the user macros.

```

:
G90 G00 X-100. ;
G31 X-200. F60 ; (Skip command)
#101=#5061 ; Skip signal input coordinate position (workpiece coordinate system) is readout to #101.
:

```

Note

- (1) Depending on the MTB specifications (parameter "#1366 skipExTyp"), the skip coordinate value may be "0" even if the G31 command is given in a 1-part system or in only a part of a multi-part system.

G31 coasting

The amount of coasting from when the skip signal is input during the G31 command until the machine stops differs according to the parameter "#1174 skip_F" or F command in G31.

The time to start deceleration to stop after responding to the skip signal is short, so the machine can be stopped precisely with a small coasting amount. The coasting amount can be calculated from the following formula.

$$\begin{aligned} \delta_0 &= \frac{F}{60} \times T_p + \frac{F}{60} \times (t_1 \pm t_2) \\ &= \underbrace{\frac{F}{60} \times (T_p + t_1)}_{\delta_1} \pm \underbrace{\frac{F}{60} \times t_2}_{\delta_2} \end{aligned}$$

δ_0 : Coasting amount (mm)

F : G31 skip speed (mm/min)

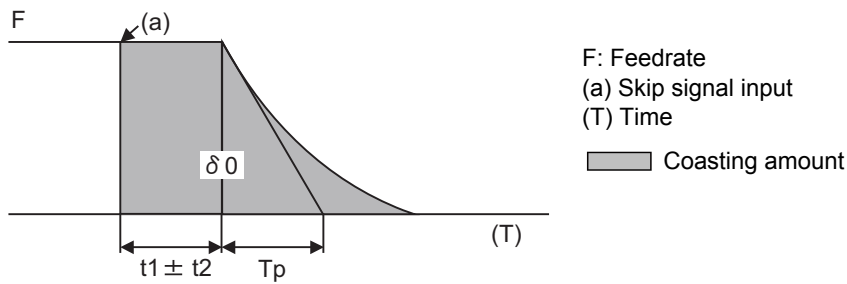
T_p : Position loop time constant (s) = (position loop gain)⁻¹

t_1 : Response delay time (s) = (time taken from the detection to the arrival of the skip signal at the controller via PC)

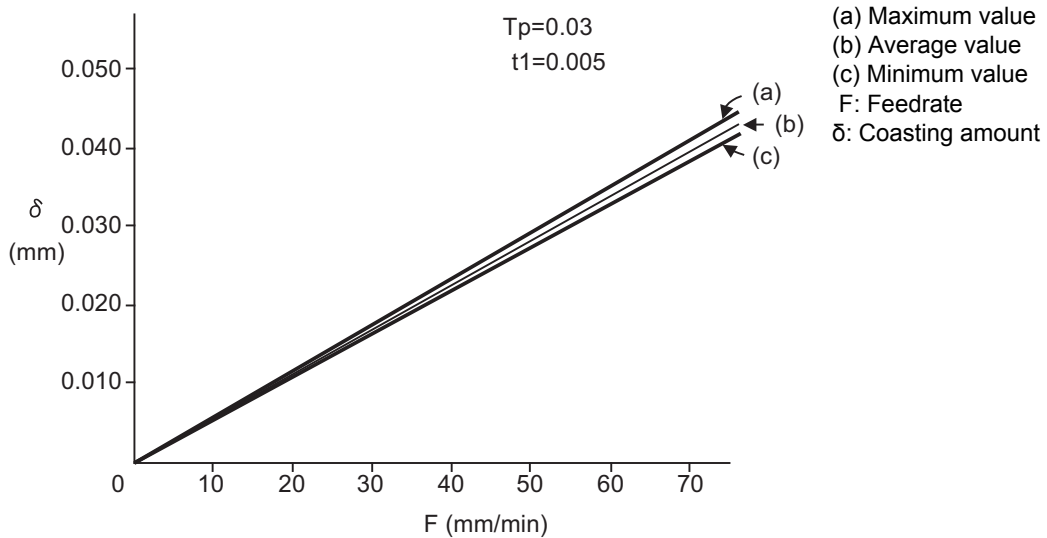
t_2 : Response error time 0.001 (s)

When G31 is used for calculation, the value calculated from the section indicated by δ_1 in the above equation can be compensated for, however, δ_2 results in calculation error.

Stop pattern with skip signal input is shown below.



The relationship between the coasting amount and speed when Tp is 30ms and t1 is 5ms is shown in the following figure.

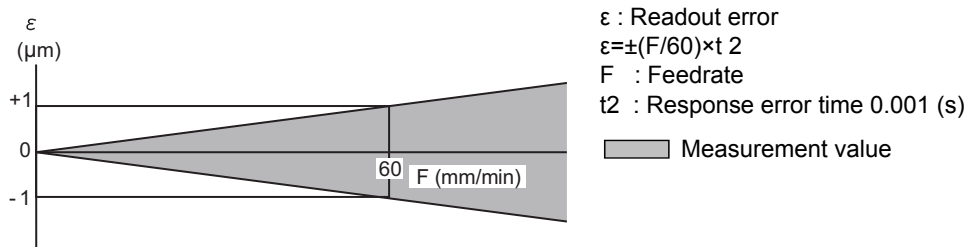


Readout error of skip coordinates mm

(1) Skip signal input coordinate readout

The coasting amount based on the position loop time constant Tp and cutting feed time constant Ts is not included in the skip signal input coordinate values.

Therefore, the workpiece coordinate values applying when the skip signal is input can be readout within the error range in the following formula as the skip signal input coordinate values. However, coasting based on response delay time t1 results in a measurement error and so compensation must be provided.



Readout error of skip signal input coordinates

Readout error with a 60mm/min feedrate is as shown below and the measurement value is within readout error range of $\pm 1\mu\text{m}$:

$$\epsilon = \pm (60/60) \times 0.001 = \pm 0.001 \text{ (mm)}$$

(2) Readout of other coordinates

The readout coordinate values include the coasting amount. Therefore, when coordinate values at the time of skip signal input is required, reference should be made to the section on the G31 coasting amount to compensate the coordinate value. As in the case of (1), the coasting amount based on the delay error time t2 cannot be calculated, and this generates a measuring error.

Examples of compensating for coasting

(1) Compensating for skip signal input coordinates

```

:
G31 X100.F100 ;      Skip command
G04 ;               Machine stop check
#101=#5061 ;        Skip signal input coordinate readout
#102=#110*#111/60 ; Coasting based on response delay time
#105=#101-#102 ;    Skip signal input coordinates
:
#110 = Skip feedrate;
#111 = Response delay time t1;

```

(2) Compensating for workpiece coordinates

```

:
G31 X100.F100 ;      Skip command
G04 ;               Machine stop check
#101=#5061 ;        Skip signal input coordinate readout
#102=#110*#111/60 ; Coasting based on response delay time
#103=#110*#112/60 ; Coasting based on position loop time constant
#105=#101-#102-#103 ; Skip signal input coordinates
:
#110 = Skip feedrate;
#111 = Response delay time t1;
#112 = Position loop time constant Tp;

```

Operation to be carried out when the skip command is executed on multiple part systems at the same time

The operation resulting from the G31 command executed simultaneously on multiple part systems depends on the MTB specifications (parameter "#1366 skipExTyp").

#1366	Operation
0	When any part system is executing the G31 command, the G31 command issued for other part systems is subjected to a block interlock state, and such G31 command will be executed after the current G31 command execution is completed. (No error is displayed.) In a single-block operation, for example, where the G31 block is started in multiple part systems at the same time, it is executed in the smallest part system first.
1	The G31 command is executed on multiple part systems at the same time. However, the skip coordinate position is not read and is set to "0" in all part systems. (*1)

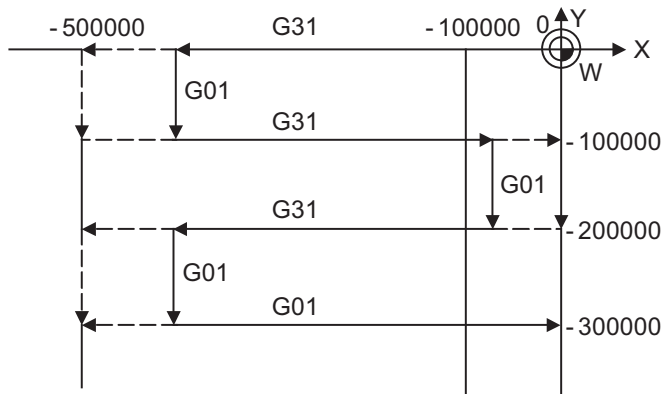
(*1) The skip coordinate position is also set to "0" when the G31 command is executed on a single part system. Furthermore, it is set to "0" when the G31 command is executed on one part system in a multiple part system configuration.

When the G31 command is used for measuring purposes, "#1366 skipExTyp" must be "0".



Operation example

```
G90 G00 X-100000 Y0;
G31 X-500000 F100;
G01 Y-100000;
G31 X-0 F100;
Y-200000;
G31 X-500000 F100;
Y-300000;
X0;
```



22.3 Multi-step Skip Function 1; G31.n, G04



Function and purpose

The setting of combinations of skip signals to be input enables skipping under various conditions. The actual skip operation is the same as G31.

The G commands which can specify skipping are G31.1, G31.2, G31.3, and G04, and the correspondence between the G commands and skip signals and settings for each parameter depend on the MTB specifications.



Command format

G31.1 X__Y__Z__α__R__F__ ;

X,Y,Z,α	Target coordinates
R	Acceleration/deceleration command R0: Acceleration/deceleration time constant=0 (No automatic acceleration/deceleration after interpolation.) R1: Acceleration/deceleration time constant valid. Accelerate/decelerate with the time constant set in the parameters "#2102 skip_tL" and "#2103 skip_t1". R0 is applied when it is omitted.
F	Feedrate (mm/min)

Same with G31.2 and G31.3; Ff is not required with G04.

As with the G31 command, this command executes linear interpolation and when the preset skip signal conditions have been met, the machine is stopped, the remaining commands are canceled, and the next block is executed.



Detailed description

- (1) The skip speed is specified by program command or parameter. Feedrate G31.1 set with the parameter corresponds to "#1176 skip1f", G31.2 corresponds to "#1178 skip2f", G31.3 corresponds to "#1180 skip3f", and G04 corresponds to "#1173 dwlskp". Note that the F modal is not updated in each case.
- (2) A command is skipped if it meets the specified skip signal condition.
- (3) The feedrates corresponding to the G31.1, G31.2, and G31.3 commands can be set by parameters.
- (4) The skip conditions (logical sum of skip signals that have been set) corresponding to the G31.1, G31.2, G31.3 and G04 commands can be set by parameters.

Parameter setting	Valid skip signal		
	1	2	3
1	○		
2		○	
3	○	○	
4			○
5	○		○
6		○	○
7	○	○	○

- (5) Details other than the above are the same as those on G31 (Skip function).



Operation example

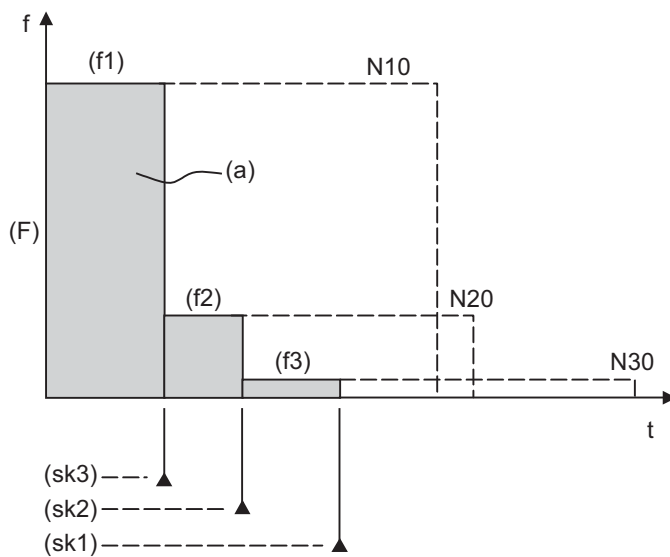
(1) The multi-step skip function enables the following control, thereby improving measurement accuracy and shortening the time required for measurement.

[Parameter settings]

Skip condition	Skip speed
G31.1 :7	20.0 mm/min(f1)
G31.2 :3	5.0 mm/min(f2)
G31.3 :1	1.0 mm/min(f3)

[Program example]

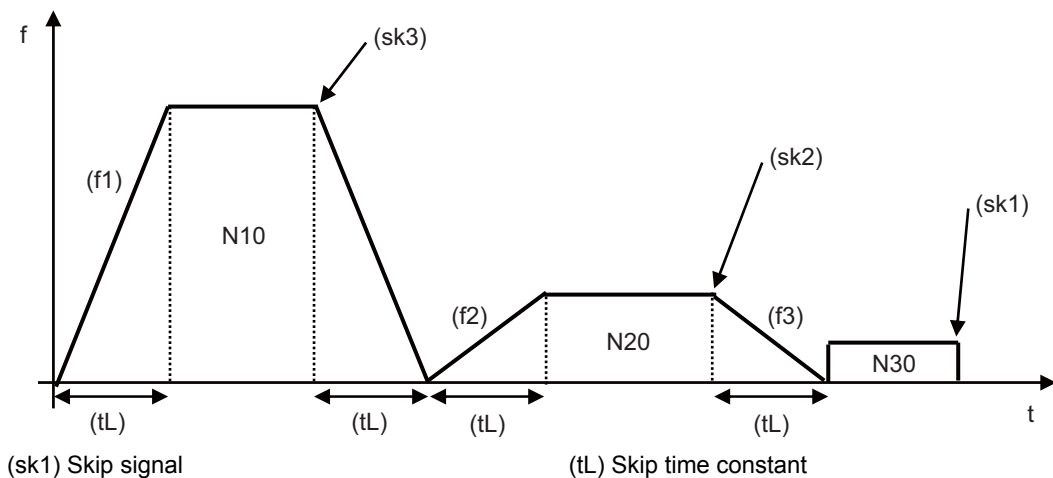
```
N10 G31.1 X200.0 ;
N20 G31.2 X40.0 ;
N30 G31.3 X1.0 ;
```



(a) Measurement distance
 (F) Skip speed
 (sk1) Input of skip signal 1
 (sk2) Input of skip signal 2
 (sk3) Input of skip signal 3

<Note>

•If skip signal 1 is input before skip signal 2 in the above operation, N20 is skipped at that point and N30 is also ignored.



(2) If a skip signal with the condition set during G04 (dwell) is input, the remaining dwell time is canceled and the following block is executed.

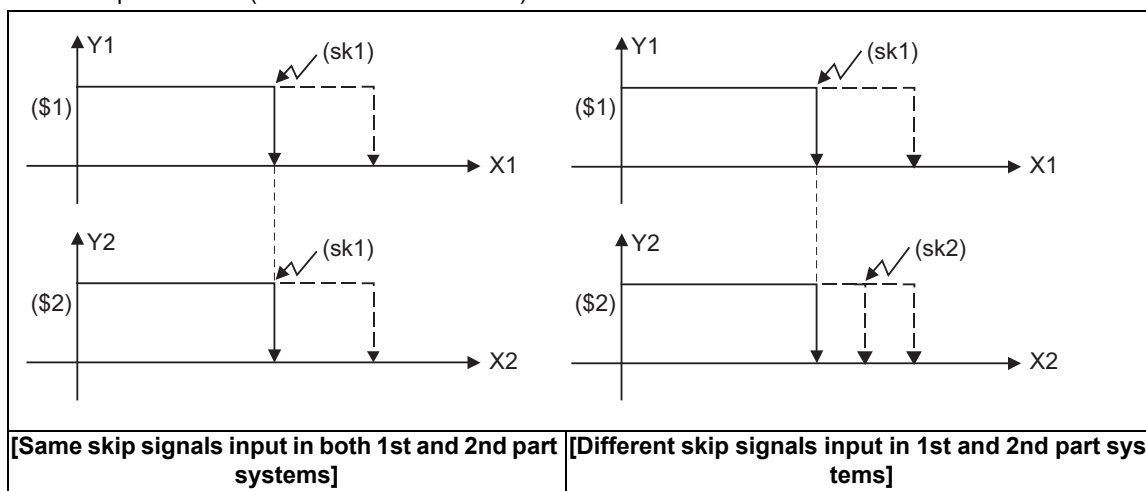
22.4 Multi-step Skip Function 2; G31 P



Function and purpose

During linear interpolation by the skip command (G31), operation can be skipped according to the conditions of the skip signal parameter Pp.

If multi-step skip commands are issued simultaneously in different part systems as shown in the left figure, both part systems perform skip operation simultaneously if the input skip signals are the same, or they perform skip operation separately if the input skip signals are different as shown in the right figure. The skip operation is the same as a normal skip command (G31 without P command).



(\$1) 1st part system

(\$2) 2nd part system

(sk1) Skip signal 1

(sk2) Skip signal 2

If the skip condition specified by the parameter "#1173 dwlskp" (indicating external skip signals 1 to 4) is met during execution of a dwell command (G04), the remaining dwell time is canceled and the following block is executed.



Command format

G31 X__ Y__ Z__ α__ P__ R__ F__ ;

X Y Z α	Target coordinates
P	Skip signal command
R	Acceleration/deceleration command R0: Acceleration/deceleration time constant=0 (No automatic acceleration/deceleration after interpolation.) R1: Acceleration/deceleration time constant valid. Accelerate/decelerate with the time constant set in the parameters "#2102 skip_tL" and "#2103 skip_t1". R0 is applied when it is omitted.
F	Feedrate (mm/min)



Detailed description

- (1) The skip speed is specified by program command or parameter. The feedrate by the parameter is set by "#1174 skip_F". Note that the F modal is not updated in each case.
- (2) The skip signal is specified by skip signal command p. The command range of "p" is from 1 to 255. If outside the range is commanded, program error (P35) will occur.

Skip signal command P	Valid skip signal							
	8	7	6	5	4	3	2	1
1								○
2							○	
3							○	○
4						○		
5						○		○
6						○	○	
7						○	○	○
8					○			
:								
:								
:								
253	○	○	○	○	○	○		○
254	○	○	○	○	○	○	○	
255	○	○	○	○	○	○	○	○

- (3) The specified skip signal command is a logical sum of the skip signals.
(Example) G31 X100. P5 F100 ;
Operation is skipped if skip signal 1 or 3 is input.
- (4) If skip signal parameter Pp is not specified, it works as a skip function (G31), not as a multi-step skip function. If speed parameter Ff is not specified, the skip speed set by the parameter "#1174 skip_F" will apply.

[Relations between skip and multi-step skip]

Skip specifications	×		○	
	Condition	Speed	Condition	Speed
G31 X100 ;(Without P and F)	Program error (P601)		Skip 1	#1174 skip_F
G31 X100 P5 ;(Without F)	Program error (P602)		Command value	#1174 skip_F
G31 X100 F100 ;(Without P)	Program error (P601)		Skip 1	Command value
G31 X100; P5 F100;	Program error (P602)		Command value	Command value

- (5) If skip specification is effective and P is specified as an axis address, skip signal parameter P will be given a priority. The axis address "P" will be ignored.
(Example) G31 X100. P500 F100 ;
This is regarded as a skip signal. (The program error (P35) will occur.)
- (6) Other than above, the same detailed description as "Skip function; G31" applies.

22.5 Speed Change Skip; G31 Fn



Function and purpose

When the skip signal is detected during linear interpolation by the skip command (G31), the feedrate is changed.



Command format

G31 X_ (Y_) Z_ α_ R_ F_ F1 = _ ... Fn = _ ; ("n" is the skip signal 1 to 8) ... Skip command

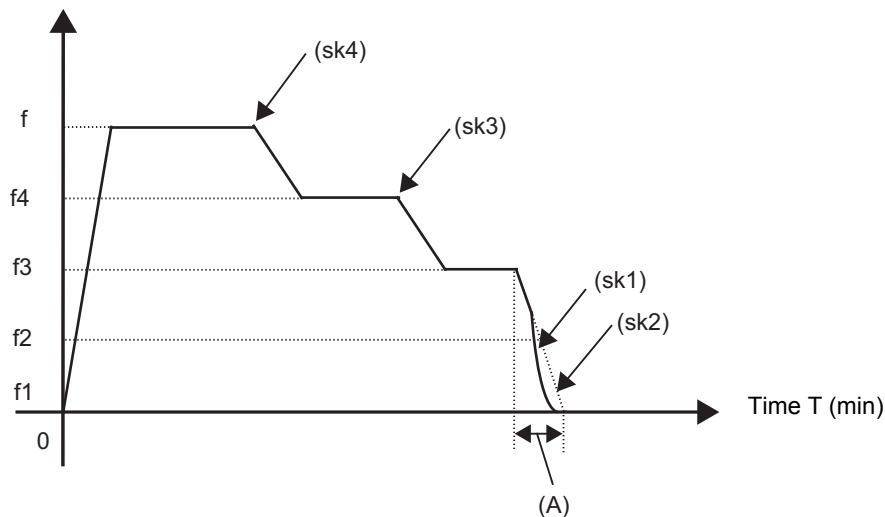
X, (Y,) Z, α	Target coordinates
R	Acceleration/deceleration command R0: Acceleration/deceleration time constant=0 When the movement is stopped by the skip signal detection, the step stop will occur. R1: Acceleration/deceleration time constant valid. When the movement is stopped by the skip signal detection, it will decelerate with the time constant set in the parameter "#2102 skip_tL" and "#2103 skip_t1". When omitted, R0 will be applied.
F	Feedrate when starting the cutting feed (mm/min)
Fn=	Feedrate after detecting the skip signal (mm/min) Fn = 0: Movement stop Fn ≠ 0 :Changing the feedrate to fn F1 = Feedrate after inputting the skip signal 1 : F8 = Feedrate after inputting the skip signal 8



Detailed description

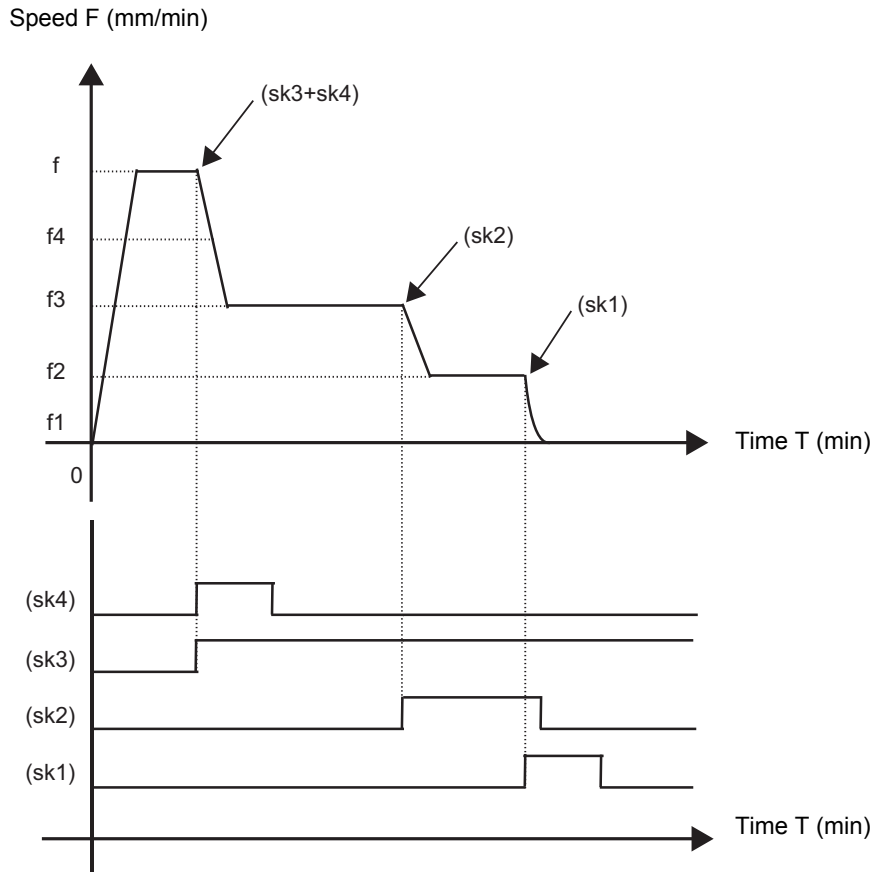
- (1) When the skip signal for which the feedrate $fn \neq 0$ is commanded, the speed is changed to the command speed corresponding to the skip signal.
- (2) When the skip signal for which the feedrate $fn = 0$ is commanded, the movement is stopped. If R0 is commanded or R command is omitted, the skip stop will occur when the movement is stopped by the skip signal detection without performing the automatic acceleration/deceleration by the skip time constant. When R1 is commanded, the automatic acceleration/deceleration will be performed with the skip time constant after the interpolation even if the movement is stopped by the skip signal detection. Note that if the value of the parameter "#2102 skip_tL" and "#2103 skip_t1" are large, it will not stop immediately. After the movement is stopped, the remaining movement commands are canceled and the following block will be executed.
- (3) When a skip signal has not been input until the completion of the G31 block, the G31 command will also be completed upon completion of the movement commands.
- (4) When the skip return is valid, the return operation by the skip signal detection is executed after the movement is stopped.
- (5) Even if G1 constant inclination acceleration/deceleration (#1201 G1_acc) is valid, the speed change skip will be the operation of the time constant acceleration and deceleration.
- (6) When the feedrate command ($Fn=fn$) is not specified after detecting the skip signal, the normal G31 skip operation will be applied.
- (7) If a skip signal (one of sk1 to sk4) are input during the deceleration (area (A) in the figure) after a move command has finished:
 - (a) A skip signal (sk2 in the figure) for changing speed is ignored.
 - (b) A skip signal (sk1 in the figure) for stopping the movement is executed and the speed is set to "0".

Speed F (mm/min)



- (8) The skip signal without commanding the feedrate in the program will be ignored.

- (9) The speed change or the movement stop is performed when detecting the rising edge of the skip signal. Note that if several rising edges are input at 3.5ms intervals or less, they may be judged as the simultaneous input. When they are judged as the simultaneous input, the smaller value will be valid. Shown below are changes in time (T) and speed (F) when skip signals, 1 (sk1) to 4 (sk4), are input.



- (10) If the G31 block is started with the skip signal input, that signal is considered to rise at the same time as the block starts.
- (11) If the skip signals for changing the speed and for stopping the movement are simultaneously input, the skip signal for stopping the movement will be valid regardless of the size of the number.
- (12) If the skip time constant "#2102 skip_tL" is illegal, an MCP alarm (Y51 15) will occur. If the "#2103 skip_t1" is illegal, an MCP alarm (Y51 16) will occur.
- (13) Other than above, the same detailed description as "Skip function; G31" applies.



Operation example

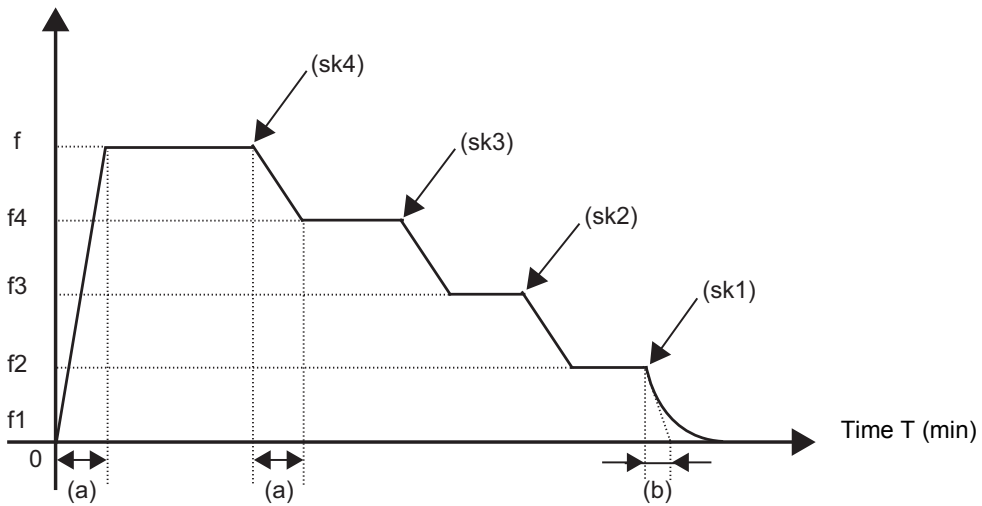
The following shows the operations when a skip time constant and skip signals, 1 (sk1) to 4 (sk4), are input.

(1) Example of when R is not commanded

Skip time constant ((a) in the figure) and position loop time constant ((b) in the figure)

G31 X100. Ff F1=0 F2=f2 F3=f3 F4=f4 ;

Speed F (mm/min)

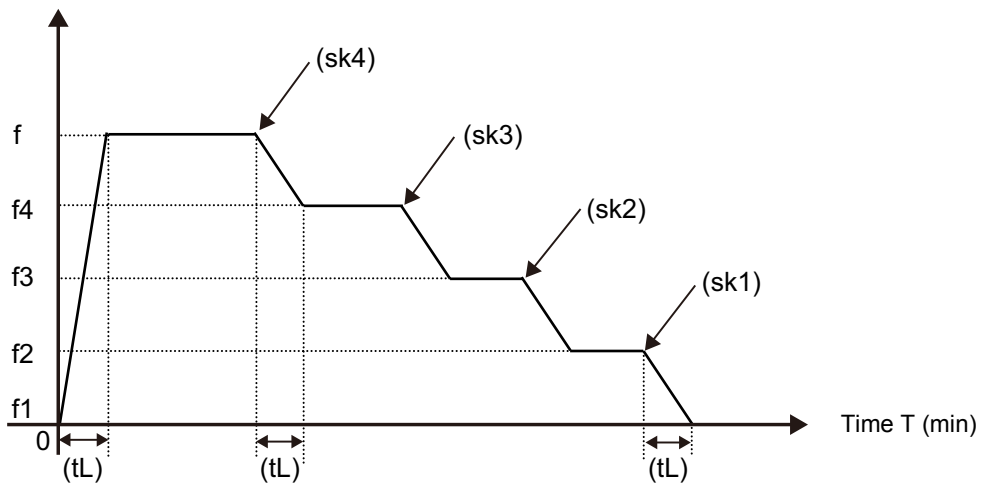


(2) Example of when R1 is commanded

Skip time constant ((tL) in the figure)

G31 X100. R1 Ff F1=0 F2=f2 F3=f3 F4=f4;

Speed F (mm/min)



22.6 Torque Limitation Skip; G160



Function and purpose

Axis movement is performed in the torque limited status, and the axis movement command is suspended to proceed to the next block when the current command value reaches the designated torque skip value and the torque skip turns ON. In addition to the torque, the droop value can be add to the condition of the skip ON (Droop skip). This function enables measurement without a sensor.



Command format

Torque limitation skip

```
G160 X/Y/α__ Q__ D__ F__ ;
```

The G160 command is unmodal (group 00). When executing the G160 command continuously, it must always be command for each block separately.

X/Y/α	Axis address and coordinate value command (mm/inch) (Decimal point command is possible)
Q	Torque skip value (0 to 500 (%))
D	Droop skip value (0 to 99999.999 mm, 0 to 9999.9999 inch)
F	Skip speed Set it in the range of feedrates. (mm/min, inch/min, mm/rev, inch/rev)

Note

- (1) Designate an axis that exists in the part system for the axis address. If an axis that does not exist in the part system, a program error (P32) will occur.
- (2) Only one axis can be commanded with the axis address. If no axis is specified or if two or more axes are specified in the same block, a program error (P595) will occur.
- (3) For spindle/C axis (C axis command), a Q command is specified with 121 to 500 %, the axis is clamped at 120%.
- (4) If a Q command is omitted, torque skip function is performed as specified by the MTB (parameter shown below).
NC axis (servo axis): SV014 ILMTsp (current limit value in special control)
Spindle/C axis (C axis command):
•For the normal spindle, SP065 TLM1 (torque limit 1)
•For spindle-mode servo, SV014 ILMTsp (current limit value in special control)
- (5) If D command is omitted, a skip operation is performed using the torque skip value only.
- (6) D command must be programmed within the excessive error width shown below.
NC axis (servo axis): SV023 OD1 (detected excessive error width when servo is on)
Spindle/C axis: SP023 OD1 (detected excessive error width (interpolation mode))
- (7) If an F command is omitted, the feedrate depends on the MTB specifications (parameter "#1174 skip_F").
- (8) A program error (P603) will occur if the skip speed in F command is 0.



Detailed description

Acceleration/deceleration when G160 is commanded

- ♦ Follow the acceleration/deceleration pattern for linear interpolation (G01).
- ♦ Even if G01 constant inclination acceleration/deceleration is valid, the time constant acceleration and deceleration will be performed.

Skip speed

If F command is programmed in the same block as G160, the commanded speed is set as the skip speed.

If an F 1-digit feed command is issued to program the feedrate, F 1-digit feed is disabled.

Note that, in the following cases, the skip speed and operations depend on the MTB specifications (parameter "#12022 skipF_spec/bit2").

	#12022/bit2 = 0	#12022/bit2 = 1
Skip speed if F command is not programmed in the G160 block	The value of parameter "#1174 skip_F" is used as the skip speed.	The skip speed is determined based on the modality of F when G160 is executed.
	A program error (P603) will also occur if the value of parameter "#1174 skip_F" is "0".	A program error (P62) will also occur if the value of F modality is "0".
Mode of commanded speed	Only feed per minute mode is available. Feed per minute mode is enabled even in feed per revolution mode.	Follows the mode (Feed per minute/Feed per revolution) that is active when G160 is executed.
Modality of F command	F modal is not updated even if the G160 block contains an F command.	The F modal that is updated by F command in the G160 block varies depending on the mode (Feed per minute/Feed per revolution) that is active when G160 is executed.

Control signals regarding speed control and stop

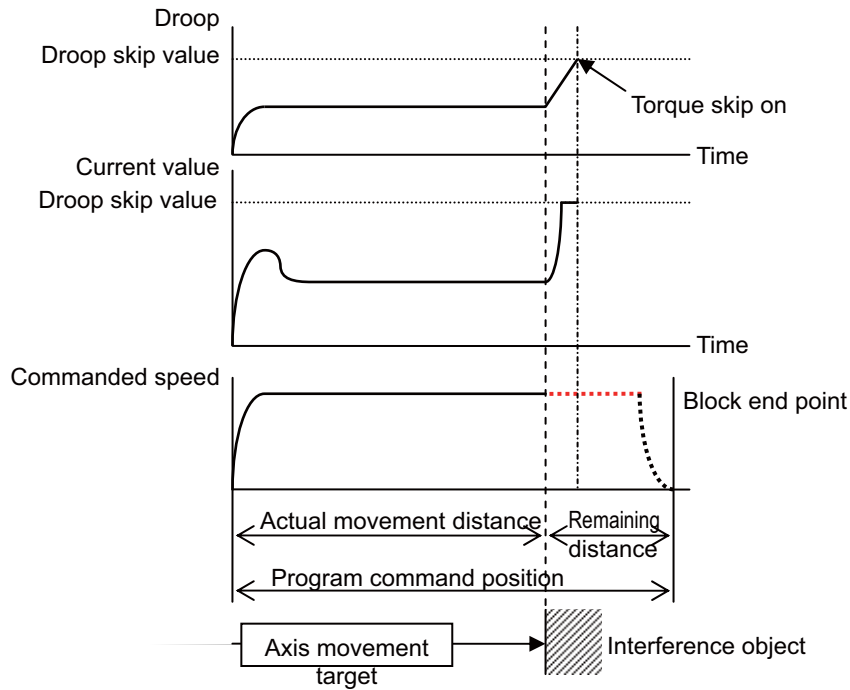
- (1) For the validity of the following various functions, refer to the MTB specifications.
 - ♦ Cutting feed override valid/invalid (parameter "#12022 skipF_spec/bit0")
 - ♦ Dry run valid/invalid (parameter "#12022 skipF_spec/bit1")
- (2) An operation error (M01 0102) occurs if 0% cutting feed override is performed when cutting feed override is invalid.
- (3) The stop conditions (feed hold, interlock, override zero and stroke end) and external deceleration are valid when torque limitation skip is used.
- (4) The machine lock signal is valid. (The counter is updated until the program reaches the end point of the block.)

Processing when the torque skip turns on

- (1) If the current value for the specified axis exceeds the torque skip value, the torque limit is reached and droop exceeds the droop skip value, the torque skip turns on. If there is no D command, the torque skip turns on when the torque limit is reached.
- (2) The current position when the torque skip turns on is regarded as the block end point and the remaining distance (command value - actual movement distance) is discarded.

Completion of skip command

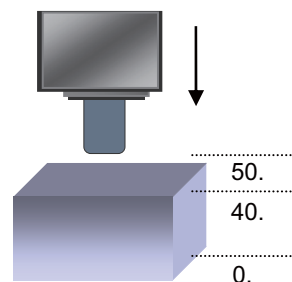
- (1) If the torque skip turns on during G160 command, the program completes the current block before moving on to the next block.
- (2) If the torque skip does not turn on until G160 command reaches the end point, the skip command completes at the end of the block and then the program moves on to the next block.
- (3) Set the skip coordinate values (workpiece coordinate values) to system variables (#5061 and onwards). When the tool has moved to the end point, set the end point position.



Program example

:	:
G28 Z200.	
T01;	Tool selection for measurement
G00 X50. Y50. Z100. ;	
G160 Z40. Q80 F20;	Torque skip command
#100=#5061;	Completion of skip command (Coordinate position (workpiece value) read)
:	

Workpiece radius measurement tool





Relationship with Other Functions

Manual arbitrary reverse run

The skip speed is controlled with the manual arbitrary reverse run speed. Torque skip command block cannot be executed in the reverse run.

Manual interruption

When a manual interrupt is applied during execution of torque skip, calculate the position shifted by the amount of the manual interruption as the skip position.

Skip variables

The torque skip position is common to skip variables (#5061 and onwards) for G31 skip function.

Geometric, Corner Rounding, Corner Chamfering

Geometric, Corner Rounding, and Corner Chamfering are not available for torque skip blocks. Program error (P595) will occur.

Torque limit

Torque skip command, if executed on the axis to which torque limits are applied, is based on the torque skip value in the G160 command.

Functions for which torque skip command is not available

Torque skip command (G160) cannot be commanded when any of the following functions is in use. (An error will occur.)

Function name	Error
Tool radius compensation (G40, G41, G42, G46)	Program error (P608)
Synchronous control (G114.1)	Program error (P595)
High-speed high-accuracy control (G05.1/G05)	Program error (P34)



Precautions

- (1) Decreasing the torque limit value may cause a torque limit to be applied during acceleration/deceleration.
- (2) When the reset button is pressed while torque skip is active, an axis moving with G160 stops. After the axis has stopped, the original torque is restored.
- (3) Writing parameters via a PLC or other host controller during execution of torque skip causes the torque limit value to be the setting value of servo parameter SV014, possibly causing it to be no longer correct torque skip value. (The PLC signal operations and setting values of the servo parameters are based on the MTB specifications.)
- (4) When using D command (droop skip value), command a value that does not exceed the excessive error width.
- (5) After a torque skip, the droop is canceled.
- (6) The droop is displayed in interpolation increments on the drive monitor. They are different from the command increments of D command.

22.7 Programmable Current Limitation; G10 L14



Function and purpose

This function allows the current limit value of the NC axis to be changed to a desired value in the program, and is used for the workpiece stopper, etc. "#2214 SVO14(current limit value in special control)" can be changed. The commanded current limit value is designated with a ratio of the limit current to the rated current.



Command format

G10 L14 Xn ;

L14	Current limit value setting (+ side/- side)
X	Axis address
n	Current limit value (%) Setting range: 1 to 999



Precautions

- (1) If the current limit value is reached when the current limit is valid, the current limit reached signal is output.
- (2) The following two modes can be used with external signals as the operation after the current limit is reached. The external signal determines which mode applies.
 - [Normal mode]
The movement command is executed in the current state.
During automatic operation, the movement command is executed until the end, and then move to the next block with the droops still accumulated.
 - [Interlock mode]
During the occurrence of the droops, it enters to the internal interlock state and the next movement will not be carried out.
During automatic operation, the operation stops at the corresponding block, and the next block is not moved to.
During manual operation, the following same direction commands are ignored.
- (3) The position droop generated by the current limit can be canceled when the current limit changeover signal of external signals is canceled. (Note that the axis must not be moving.)
- (4) The setting range of the current limit value is 1% to 999%. Commands that exceed this range will cause a program error (P35).
- (5) If a decimal point is designated with the G10 command, only the integer will be valid.
Example) G10 L14 X10.123 ; The current limit value will be set to 10%.
- (6) For the axis name "C", the current limit value cannot be set from the program (G10 command).
To set from the program, set the axis address with an incremental axis name, or set the axis name to one other than "C".

System Variables

23.1 System Variables List

The M800/M80 series provides the following system variables.

Note that the available types and numbers vary depending on the machine specifications and whether the machine is intended for use by a user or MTB.

○: Available

-: Unavailable

No.	Data type or use	Reading	Setting	Section
#1000 - #1035, #1200 - #1295	Signal input from PLC to NC	○ (*1)	-	23.25
#1100 - #1135 #1300 - #1395	Signal output from NC to PLC	○ (*1)	○ (*1)	23.26
#1900, #1901	Used for normal line control.	○	○	23.22
#2001 - #2000+n #2201 - #2200+n #2401 - #2400+n #2601 - #2600+n	Tool offset data Also refer to the variables #10001, #135001 and #230001 or larger.	○	○	23.6
#2501, #2601	External workpiece coordinate offset	○	○	23.10
#3000	Used to forcibly set to the alarm mode. Designate the number and message.	-	○	23.12
#3001, #3002	Cumulative time (integrating time)	○	-	23.14
#3001, #3002 #3011, #3012	Time read variables	○	○	23.15
#3003	♦Inhibition of single block stop ♦Inhibition of miscellaneous function finish signal waiting ♦Prohibition of program check reverse run	○	○	23.16
#3004	♦Automatic operation pause OFF ♦Cutting override OFF ♦G09 check OFF ♦Dry run invalid	○	○	
#3006	Used to display and stop a message.	-	○	23.13
#3007	Mirror image	○	-	23.19
#3901, #3902	Number of machining processes/Maximum number of machining processes	○	○	23.18
#4001 - #4021 #4201 - #4221	G command modal information	○	-	23.2
#4101 - #4120 #4301 - #4320	Non-G command modal information	○	-	23.3
#4401 - #4421 #4507 - #4520	Modal information at macro interruption	○	-	23.4
#5001 - #5160+n	Position information ♦End point coordinate position of the previous block ♦Machine coordinate position ♦Workpiece coordinate position ♦Skip coordinate position ♦Tool position compensation amount ♦Servo deviation amount ♦Macro interruption stop block coordinate position	○	-	23.11
#5201 - #5320+n	Workpiece coordinate offset data	○	○	23.8
#7001 - #8900+n	Extended workpiece coordinate offset data (48- or 96-set specification)	○	○	23.9
#10001 - #10000+n #11001 - #11000+n #16001 - #16000+n #17001 - #17000+n	Tool offset data Also refer to the variables #2001, #135001 and #230001 or larger.	○	○	23.6
#26000 - #26077	Workpiece installation error compensation amount	○	○	23.24

23 System Variables

No.	Data type or use	Reading	Setting	Section
#30060 - #30068	Coordinate Rotation Parameter	○	-	23.20
#31001 - #31023	Rotary axis configuration parameter	○	-	23.21
#31100, #31101	Number of reverse run enable blocks, reverse run enable counter	○	-	23.17
#40000 - #40097	Specification of the selected interfering object and interfering model coordinate system offset	○	○	23.29
#50000 - #50749 #51000 - #51749 #52000 - #52749	Data of user backup area for R device	○ (*1)	○ (*1)	23.27
#50000 - #51199	ZR device access variables (C80 series only)	○	○	23.30
#60000 - #64700	Tool life management	○	○ (*2)	23.7
#68000 - #68003	Tool management	○	○ (*2)	23.5
#68011 - #68023	Basic information	○	○ (*2)	
#68031 - #68040	Shape information	○	○	
#68051 - #68054	Cutting conditions	○	○	
#68061 - #68072	Additional information	○	○	
#68081 - #68088	Tool life	○	○	
#68101 - #68113	Compensation amount	○	○	
#100000	Parameter No. designation	-	○	23.23
#100001	Part system No. designation	-	○	
#100002	Axis No./spindle No. designation	-	○	
#100010	Parameter value read	○	-	
#100050 - #100054	NC data reading/writing with API section and sub-section designation	○	○	23.31
#100100	Device type designation	-	○	23.28
#100101	Device No. designation	-	○	
#100102	Number of read bytes designation	-	○	
#100103	Read bit designation	-	○	
#100110	Reading PLC data	○	-	
#101001 - #115950+n	Extended workpiece coordinate offset data (300-set specification)	○	○	23.9
#135001 - #135000+n #136001 - #136000+n #137001 - #137000+n #138001 - #138000+n	Tool offset data Also refer to the variables #2001, #10001 and #230001 or larger.	○	○	23.6
#230001 - #230000+n	Tool offset data Also refer to the variables #2001, #10001 and #135001 or larger.	○	○	23.6

(*1) Only for MTB. This cannot be designated by the user.

(*2) Some numbers are not available depending on the contents.

23.2 System Variables (G Command Modal)



Detailed description

Using variable Nos. #4001 to #4021, it is possible to read the modal commands which have been issued in previous blocks.

Similarly, it is possible to read the modals in the block being executed with variable Nos. #4201 to #4221.

Variable No.		Function	
Pre-read block	Execution block		
#4001	#4201	Interpolation mode	G00 : 0, G01 : 1, G02 : 2, G03 : 3, G33 : 33
#4002	#4202	Plane selection	G17 : 17, G18 : 18, G19 : 19
#4003	#4203	Absolute/incremental	G90 : 90, G91 : 91
#4004	#4204	No variable No.	
#4005	#4205	Feed designation	G94 : 94, G95 : 95
#4006	#4206	Inch/metric	G20 : 20, G21 : 21
#4007	#4207	Tool radius compensation	G40 : 40, G41 : 41, G42 : 42
#4008	#4208	Tool length compensation	G43:43, G44:44, G49:49
#4009	#4209	Fixed cycle	G80 : 80, G73-74 : 73-74, G76 : 76, G81-89 : 81-89
#4010	#4210	Return level	G98 : 98, G99 : 99
#4011	#4211		
#4012	#4212	Workpiece coordinate system	G54-G59 : 54-59, G54.1:54.1
#4013	#4213	Acceleration/deceleration	G61-G64 : 61-64, G61.1 : 61.1
#4014	#4214	Macro modal call	G66 : 66, G66.1 : 66.1, G67 : 67
#4015	#4215	Normal line control	G40.1 : 40.1, G41.1 : 41.1, G42.1 : 42.1
#4016	#4216		
#4017	#4217	Constant surface speed	G96 : 96, G97 : 97
#4018	#4218	No variable No.	
#4019	#4219	Mirror image	G50.1:50.1, G51.1:51.1
#4020	#4220		
#4021	#4221	No variable No.	

(Example)

```
G28 X0 Y0 Z0;
G90 G1 X100. F1000;
G91 G65 P300 X100. Y100.;
M02;
O300;
#1=#4003; -> Group 3G modal (pre-read) #1=91.0
#2=#4203; -> Group 3G modal (active) #2=90.0
G#1 X#24 Y#25;
M99;
%
```

23.3 System Variables (Non-G Command Modal)



Detailed description

Using variable Nos. #4101 to #4120, it is possible to read the modal commands which have been issued in previous blocks.

Similarly, it is possible to read the modals in the block being executed with variable Nos. #4301 to #4320.

Variable No.		Modal information	Variable No.		Modal information
Pre-read block	Execution block		Pre-read	Execution	
#4101	#4301		#4111	#4311	Tool length compensation No. H
#4102	#4302		#4112	#4312	
#4103	#4303		#4113	#4313	Miscellaneous function M
#4104	#4304		#4114	#4314	Sequence number N
#4105	#4305		#4115	#4315	Program number O (*1)
#4106	#4306		#4116	#4316	
#4107	#4307	Tool radius compensation No. D	#4117	#4317	
#4108	#4308		#4118	#4318	
#4109	#4309	Feedrate F	#4119	#4319	Spindle function S
#4110	#4310		#4120	#4320	Tool function T
			#4130	#4330	Extended workpiece coordinate system No. P

(*1) Programs are registered as files. When the program No. (file name) is read with #4115, #4315, the character string will be converted to a value.

(Example 1)

The file name "123" is the character string 0x31, 0x32, 0x33, so the value will be $(0x31-0x30)*100 + (0x32-0x30)*10 + (0x33-0x30) = 123.0$.

Note that if the file name contains characters other than numbers, it will be "blank".

(Example 2)

If the file name is "123ABC", it contains characters other than numbers, so the result will be "blank".

23.4 System Variables (Modal Information at Macro Interruption)



Detailed description

Modal information when control passes to the user macro interruption program can be known by reading system variables #4401 to #4520.

The unit specified with a command applies.

System variable	Modal information	
#4401	G code (group 01)	Some groups are not used.
⋮	⋮	
#4421	G code (group 21)	
#4507	D code	
#4509	F code	
#4511	H code	
#4513	M code	
#4514	Sequence number N	
#4515	Program number O (*1)	
#4519	S code	
#4520	T code	

The above system variables are available only in the user macro interrupt program.

If they are used in other programs, program error (P241) will occur.

(*1) Programs are registered as files. When the program No. (file name) is read with #4515, the character string will be converted to a value.

(Example 1)

The file name "123" is the character string 0x31, 0x32, 0x33, so the value will be $(0x31-0x30)*100 + (0x32-0x30)*10 + (0x33-0x30) = 123.0$.

Note that if the file name contains characters other than numbers, it will be "blank".

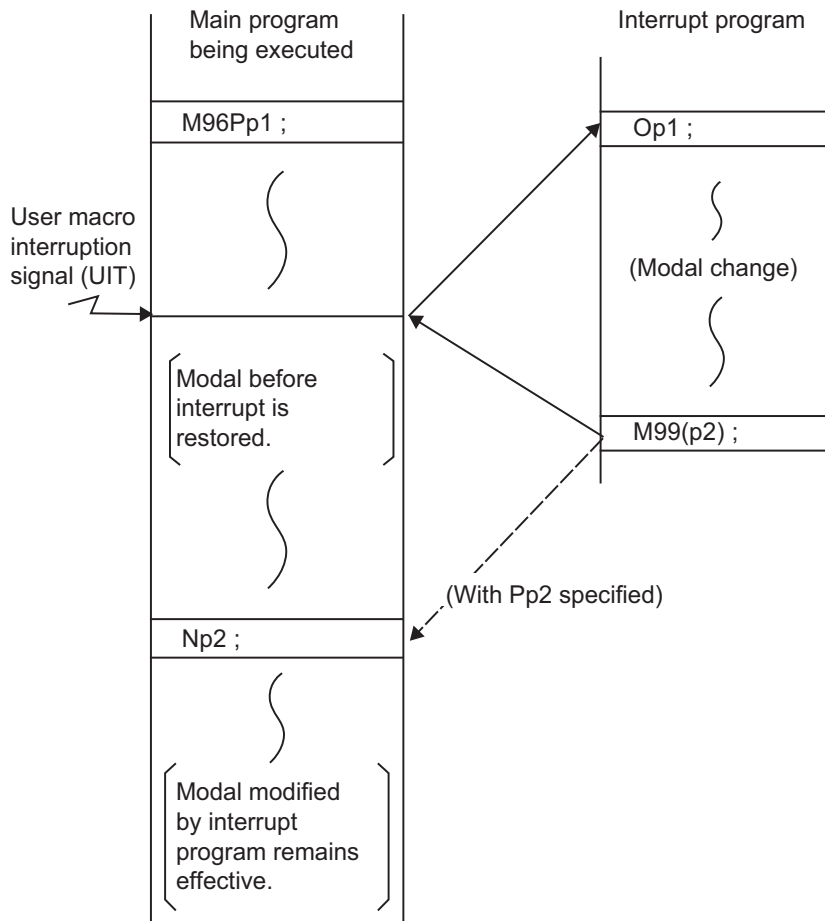
(Example 2)

If the file name is "123ABC", it contains characters other than numbers, so the result will be "blank".

Modal information affected by user macro interruption

If modal information is changed by the interrupt program, it is handled as follows after control returns from the interrupt program to the main program.

Returning with M99;	The change of modal information by the interrupt program is invalidated and the original modal information is restored. With interrupt type 1, however, if the interrupt program contains a move or miscellaneous function (MSTB) command, the original modal information is not restored.
Returning with M99P__ ;	The original modal information is updated by the change in the interrupt program even after returning to the main program. This is the same as in returning with M99P__ ; from a program called by M98, etc.



Modal information affected by user macro interruption

23.5 System Variables (Tool Information)

Tool management (#68000 - #68003)

Variable No.	Item / Description		Data range	Attribute												
#68000	Tool designation method	Method to designate the tool to be read or written 1: In-use tool designation 2: Tool number designation 3: Tool management screen registration number designation	1 to 3	-/W												
#68001	Tool selection No.	Designate the tool selection number that matches the setting of "#68000". <table border="1" data-bbox="603 640 1145 1025"> <thead> <tr> <th>#68000</th> <th>Details of "#68001"</th> <th>Data range</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>ATC magazine number (Used only when the ATC is added.)</td> <td>0 to 5</td> </tr> <tr> <td>2</td> <td>Tool No. (T No.) (Tool No. and compensation No. for lathe system)</td> <td>1 to 99999999</td> </tr> <tr> <td>3</td> <td>Tool management screen registration number</td> <td>1 to Number of managed tools</td> </tr> </tbody> </table>	#68000	Details of "#68001"	Data range	1	ATC magazine number (Used only when the ATC is added.)	0 to 5	2	Tool No. (T No.) (Tool No. and compensation No. for lathe system)	1 to 99999999	3	Tool management screen registration number	1 to Number of managed tools	Refer to the "Description" column.	-/W
#68000	Details of "#68001"	Data range														
1	ATC magazine number (Used only when the ATC is added.)	0 to 5														
2	Tool No. (T No.) (Tool No. and compensation No. for lathe system)	1 to 99999999														
3	Tool management screen registration number	1 to Number of managed tools														
#68003	Top vacant registration number on tool management screen	The tool number indicates a vacant line number. 0: No vacant registration number 1 to 999: Vacant registration number	0 to 999	R/-												

- If you command to read data to a write only variable or write to a read only variable, a program error (P241) will occur.
- If a value exceeding the allowable range is issued, a program error (P35) will occur.

(1) Tool designation method (#68000), Tool selection number (#68001)

Substitute a value to the parameters "#68000" and "#68001" to designate the tool to be read and written with the parameters "#68011" to "#68111".

The tool designation methods are classified into three types as shown below.

Tool designation method	Details	"#68000" setting value	"#68001" setting value
In-use tool designation	Reads or writes tool management data of the tool in use.	1	ATC magazine No.
Tool number designation	Reads or writes tool management data designated with the tool number.	2	Tool No. (T No.)
Tool management screen registration number designation	Reads or writes tool management data designated with the registration number.	3	Tool management screen registration number

(a) In-use tool designation (#68000=1)

For the in-use tool, when the R register is checked in the order from "1" to "3" shown below, if the value designated in the R register is other than "0", it is judged to be the in-use tool number.

- ♦ Tool life management spindle tool number in machining center system (R12200: 1st part system to R12270: 8th part system)
- ♦ ATC spindle tool number (R10620: magazine 1 to R10660: magazine 5)
- ♦ T code data (R536)

"#68001" designates the ATC magazine number.

If ATC is not used, this item does not need to be designated.

The "#68001" setting value has the meanings shown below.

"#68001" setting value	Meaning
"0" or no "#68001" command	Magazine 1
1 to 5	Magazine 1 to magazine 5

Note

- ♦ The in-use tool is determined when "#68000=1" or "#68001" is commanded.
To designate the tool which is exchanged after the in-use tool has been determined as an in-use tool, command "#68000=1" or "#68001" again.

(b) Tool number designation (#68000=2)

"#68001" designates the tool number.

In the lathe system, designate the T code (tool number and tool compensation number).

(c) Tool management screen registration number designation (#68000=3)

"#68001" designates the tool management screen registration number (line number).

Note

- ♦ If "#68000" is commanded multiple times, the last designation method will be valid.
- ♦ "#68000" and "#68001" are valid until they are reset. When the power is turned ON or when the system is reset, "0" is set.
- ♦ When #68000 is 2, and when there are multiple tools which have the same tool number and the same tool compensation number as the ones designated by "#68001", the tool that has been found first will be selected.
- ♦ A program error (P245) will occur when:
 - "#68000" is not designated;
 - "#68000=1 ;" is commanded while the in-use tool number is set to "0";
 - "#68000=1 ;" is commanded while the in-use tool number is not registered on the tool management screen;
 - "#68000=2 ;" is commanded while a read/write command is issued using "#68011" to "#68111" without commanding "#68001";
 - a tool not registered on the tool management screen with "#68011" is designated during the "#68000=2 ;" command;
 - a write command is issued with "#68011" during the "#68000=2 ;" command;
 - "#68000=3 ;" is commanded while a read/write command is issued using "#68011" to "#68111" without commanding "#68001";
 - "#68001=0 ;" is commanded.

(2) Top vacant registration number on tool management screen (#68003)

Designating this value reads the top vacant registration number on the tool management screen with "#68003".

Use example:

Follow the procedure below to measure the compensation amount with the measurement macro, etc. and search for and register a vacant registration number when registering a new tool.

Tool Manage Data						
No.	ToolNo.	ToolType	Use Dir.	Nomin	Supp	Stat
1	1	Lathing	L/F	0.0	0000	0000
2	10	BEM	CCW	0.0	0000	0000
3						
4	100	Groove	R/B	0.0	0000	0000
5	200	FEM	CCW	0.0	0000	0000
6						
7						

[Measurement macro program]

:

#68000 = 3 ;

:

Measurement

#68001=#68003 ;

Searches for a vacant registration number (No.3 in the example above), and designates registration number 3.

#68011=999 ;

Sets "999" to the tool management data "tool number" of tool management screen registration number 3.

Note

- If no vacant registration number is found because all numbers are registered, "0" is set when "#68003" is read out.
When "#68001=#68003";, "#68001" is set to "0", and a program error (P245) will occur.

Basic information ("#68011" to "#68023")

Variable No.	Item / Description		Data range	Attribute
#68011	Tool No.		0 to 99999999	R/W
#68012	Name		Eight one-byte alphanumeric characters	R/W
#68013	Type	0: No setting 1: Ball end mill 2: Flat end mill 3: Drill 4: Radius end mill 5: Chamfering 6: Tapping 7: Face mill 51: Turning 52: Slotting 53: Thread cutting 54: Turning drill 55: Turning tap	0 to 7, 51 to 55	R/W
#68014	Usage	0: No setting 1: External diameter 2: Internal diameter 3: Face	0 to 3	R/W
#68015	Direction: hand/rotation	<Mill tool, turning drill, turning tap> 0: CW 1: CCW 2: CW 3: CW <Turning, slotting, thread cutting> 0: Right hand / Front 1: Left hand / Front 2: Right hand / Rear 3: Left hand / Rear	0 to 3	R/W
#68016	Call		0.0 to 999.9 (mm) 0.00 to 99.99 (inch)	R/W
#68017	Number of blades		0 to 9	R/W
#68018	Tool ID		Eight one-byte alphanumeric characters	R/W
#68019	Supplementary information		0 to 65535	R/W
#68020	Conditions		0 to 65535	R/-
#68021	Mounting angle		0.0 to 359.999 (degree)	R/W
#68023	Gang tool offset Y		±9999.999 (mm) ±999.9999 (inch)	R/W

•If a value exceeding the allowable range is issued, a program error (P35) will occur.

23 System Variables

(1) Tool No. ("#68011")

The registered tool cannot be registered. If a tool is registered, the operation will be performed as shown below.

Type		Operation performed when a registered tool is designated
Machining center system	Life management I	Program error (P245)
	Life management II	Program error (P245)
	Life management III	Program error (P245)
	The life management specifications are invalid.	Program error (P245)
Lathe system	Life management I	Program error (P245)
	Life management II	Can be registered.
	The life management specifications are invalid.	Can be registered.

Example: When an attempt is made to change tool management data "tool number" of No.3 (3rd line) from "11" to "1" in life management II of the machining center system, the setting is as follows.

Tool Manage Data						
No.	ToolNo.	ToolType	Use Dir.	Nomin	Supp	Stat
1	1	FEM	CCW	0.0	0000	0000
2	2	BEM	CCW	0.0	0000	0000
3	11	L-drill	CW	0.0	0000	0000
4	12	Lathing O.D.	R/F	0.0	0000	0000

- #68000=3 Tool management screen registration number designation
- #68001=3 Designates No. 3 (3rd line)
- #68013=1 Tool No. 1 is already registered with No. 1 (1st line), causing a program error (P245).

(2) Tool name ("#68012"), Tool ID ("#68018"), Material ("#68053")

(a) Read

Reads data only with the variable No. designation of the DPRNT command.

Example 1: DPRNT [#68012] ; The tool name is read.

Example 2: #100=#68012 ; A program error (P243) will occur.

(b) Write

A string can be designated by enclosing it in parentheses ().

Example 1: #68012=(M-TOOL1) ; Data is written up to the number of valid characters, and the rest is ignored.

Example 2: #68012=#0 ; A string is cleared by writing "null" characters.

Example 3: #68012= M-TOOL1 ; If parentheses are omitted, a program error will occur.

(3) Type ("#68013") to tool nose point P ("#68111")

A program error will occur in the following case.

Operation	Operation result
Type ("#68013") to tool nose point P ("#68111") is read or written for the registration number with the tool number unspecified.	Program error (P245)

(4) Compensation amount ("#68103" to "#68111")

A program error will occur in the following case.

Operation	Operation result
The compensation amount ("#68103" to "#68111") is read or written for the tool with the compensation number unspecified.	Program error (P170)

(5) Tool life ("#68082" to "#68086")

A program error will occur in the following case.

Operation	Operation result
The tool life ("#68082" to "#68086") is read or written for the tool with the tool life group number unspecified in tool life management I and II of the machining center system, or in tool life management II of the lathe system.	Program error (P179)

Shape information ("#68031" to "#68040")

Variable No.	Item / Description	Data range	Attribute
#68031 to #68039	Tool shapes A to I	Length: 0 to 9999.999 (mm) 0 to 999.9999 (inch) Angle: 0 to 180.000 (degree)	R/W
#68040	Tool color 1: Gray 2: Red 3: Yellow 4: Blue 5: Green 6: Light blue 7: Purple 8: Pink	1 to 8	R/W

♦If a value exceeding the allowable range is issued, a program error (P35) will occur.

Cutting conditions ("#68051" to "#68054")

Variable No.	Item / Description	Data range	Attribute
#68051	Spindle rotation speed S	0 to 999999999	R/W
#68052	Feedrate F	0 to 1000000 (mm/min) 0 to 100000 (inch/min)	R/W
#68053	Material	Four one-byte alphanumeric characters	R/W
#68054	Coolant M code	0 to 999999999	R/W

♦If a value exceeding the allowable range is issued, a program error (P35) will occur.

Additional information ("#68061" to "#68072")

Variable No.	Item / Description	Data range	Attribute
#68061 to #68066	Customize 1 to 6	±999999999 (*1)	R/W
#68067 to #68072	Customize 7 to 12	±9999.999 (*1)	R/W

(*1) For customize data 1 to 12, the data range varies depending on the data format.

♦If a value exceeding the allowable range is issued, a program error (P35) will occur.

Tool life ("#68081" to "#68088")

Variable No.	Item / Description			Attribute
	Life management I	Life management II	Life management III	
#68081	Group No. (0 to 99999999)		(Not used)	R/W
#68082	Status (0 to 2)			R/W
#68083	Method (Ones digit: 0 to 2, 10,100s digit: 1 to 2)			R/W
#68084	Miscellaneous (0 to 65535)			R/W
#68085	Life time / Number of uses until life limit (0 to 4000 min. / 0 to 65000 sets)			R/W
#68086	Usage time / Number of uses (0 to 4000 min. / 0 to 65000 sets)			R/W
#68087	(Not used)	(Not used)	(Not used)	- / -
#68088	(Not used)	(Not used)	(Not used)	- / -

- If an unused variable is commanded, a program error (P241) will occur.
- If a value exceeding the allowable range is issued, a program error (P35) will occur.

Compensation amount ("#68101" to "#68113")

Variable No.	Item / Description			Attribute
	Compensation type I	Compensation type II	Compensation type III	
#68101	No. H (0 to number of tool offset sets)	No. H (0 to number of tool offset sets)	Tool length compensation No. (0 to number of tool offset sets)	R/W
#68102	(Not used)	No. D (0 to number of tool offset sets)	Wear compensation No. (0 to number of tool offset sets)	R/W
#68103	Tool length (±9999.999999 (mm) ±999.999999 (inch))	Length dimension (±9999.999999 (mm) ±999.999999 (inch))	Tool length X (±9999.999999 (mm) ±999.999999 (inch))	R/W
#68104	(Not used)	(Not used)	Tool length Z (±9999.999999 (mm) ±999.999999 (inch))	R/W
#68105	(Not used)	(Not used)	Additional axis tool length (±9999.999999 (mm) ±999.999999 (inch))	R/W
#68106	(Not used)	Length wear (±9999.999999 (mm) ±999.999999 (inch))	Wear X (±9999.999999 (mm) ±999.999999 (inch))	R/W
#68107	(Not used)	(Not used)	Wear Z (±9999.999999 (mm) ±999.999999 (inch))	R/W
#68108	(Not used)	(Not used)	Additional axis wear (±9999.999999 (mm) ±999.999999 (inch))	R/W
#68109	(Not used)	Radius dimension (±9999.999999 (mm) ±999.999999 (inch))	Tool nose radius (±9999.999999 (mm) ±999.999999 (inch))	R/W
#68110	(Not used)	Radius wear (±9999.999999 (mm) ±999.999999 (inch))	Radius wear (±9999.999999 (mm) ±999.999999 (inch))	R/W
#68111	(Not used)	(Not used)	Tool nose point P (0 to 9)	R/W
#68112	(Not used)	(Not used)	2nd additional axis tool length (±9999.999999 (mm) ±999.999999 (inch))	R/W
#68113	(Not used)	(Not used)	2nd additional axis wear (±9999.999999 (mm) ±999.999999 (inch))	R/W

♦If an unused variable is commanded, a program error (P241) will occur.

♦If a value exceeding the allowable range is issued, a program error (P35) will occur.

23.6 System Variables (Tool Compensation)



Detailed description

Tool compensation data can be read and set using the variable Nos.

Variable number range		Type 1	Type 2	Type 3
#10001 to #10000+n	#2001 to #2000+n	○	○ (*1)	○ Z axis shape (*1)
#11001 to #11000+n	#2201 to #2200+n	×	○ (*2)	○ Z axis wear (*2)
#16001 to #16000+n	#2401 to #2400+n	×	○ (*3)	○ Tool nose radius shape (*3)
#17001 to #17000+n	#2601 to #2600+n	×	○ (*4)	○ Tool nose radius wear (*4)
#135001 to #135000+n	-	×	×	○ X axis wear
#136001 to #136000+n	-	×	×	○ Y axis wear
#137001 to #137000+n	-	×	×	○ X axis shape
#138001 to #138000+n	-	×	×	○ Y axis shape
#230001 to #230000+n	-	×	×	○ Tool nose point

(*1) Length dimension

(*2) Length wear

(*3) Radius dimension

(*4) Radius wear

"n" in the table corresponds to the tool No. Maximum "n" value is the number of tool compensation sets.

The #10000s and #2000s are equivalent functions, however, the maximum value of "n" for #2000 order is "200".

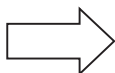
When the number of tool offset sets is larger than "200", use the variables of #10000s.

The tool compensation data is configured as data with a decimal point in the same way as other variables.

When "# 10001=100;" is programmed, "100.000" is set in tool compensation data.

Programming example

```
#101=100;
#10001=#101;
#102=#10001;
```



Common variable

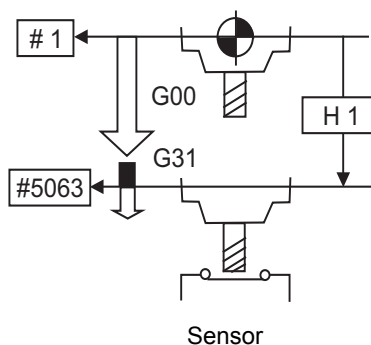
```
#101=100.0
#102=100.0
```

Tool compensation data

```
H1=100.000
```

(Example 1) Calculation and tool offset data setting

```
G28 Z0 T01. ; Reference position return
M06; Tool change (Spindle T01)
#1=#5003 ; Start point memory
G00 Z-500 ; Rapid traverse to safe position
G31 Z-100. F100.; Skip measurement
#10001=#5063-#1 ; Measurement distance calculation and tool compensation data setting
```



Note

(1) In (Example 1), no consideration is given to the delay in the skip sensor signal.

#5003 is the Z axis start point position and #5063 indicates the position at which the skip signal is input while G31 is being executed in the Z axis skip coordinates.

23.7 System Variables (Tool Life Management)



Detailed description

Definition of variable Nos.

- (1) Group number designation

#60000

Assign the value to this variable No. to designate the group number of the tool life management data to be read with parameters "#60002" to "#64700". If a group No. is not designated, the data of the group registered first is read. This is valid until reset. When the tool life management III are provided, the group No. other than 1 cannot be used.

- (2) Tool life management system variable No. (Read)

#60001 to #64700

| a | b | c | d | e |

| a | : "6" Fix (Tool life management)

| b | c | : Details of data classification

Data class	Details	Remarks
00	For control	Refer by data types
05	Group No.	Refer by registration No.
10	Tool No.	Refer by registration No.
15	Tool data flag	Refer by registration No.
20	Tool status	Refer by registration No.
25	Life data	Refer by registration No.
30	Usage data	Refer by registration No.
35	Tool length compensation data	Refer by registration No.
40	Tool radius compensation data	Refer by registration No.
45	Auxiliary data	Refer by registration No.

The group No. and life data are common for the group.

| d | e | : Registration No. or data type

Registration No.

1 to 200

Data type

Type	Details
1	Number of registered tools
2	Life current value
3	Tool selection No.
4	Number of remaining registered tools
5	Execution signal
6	Cutting time cumulative value (min)
7	Life end signal
8	Life prediction signal

List of variables

Variable No.	Item	Type	Details	Data range
60001	Number of registered tools	Common to system	Total number of tools registered in each group.	0 to 200
60002	Life current value	For each group (*1)	In-use tool usage time / number of uses Usage data of the spindle tool or in-use tool (#60003)	0 to 4000 min 0 to 9999 sets
60003	Tool selection No.		In-use tool registration number Registration number of the spindle tool (the first tool of ST:1 when the spindle tool is not data of the designated group, the first tool of ST:0 when ST:1 is not defined, or the last tool when all tools have reached the end of their lives)	0 to 200
60004	Number of remaining registered tools		No. of first registered tool that has not reached its life.	0 to 200
60005	Execution signal		"1" when this group is used in the program being executed. "1" when the group number of spindle tool data matches that of the designated group.	0/1
60006	Cutting time cumulative value (min)		Indicates the time that this group is used in the program being executed.	(Not used)
60007	Life end signal		"1" when lives of all tools in this group have expired. "1" when all registered tools in the designated group reach the end of their lives.	0/1
60008	Life prediction signal		"1" when a new tool is selected with the next command in this group. "1" when there are no tools in use (ST: 1) while there is an unused tool (ST: 0) in the designated group.	0/1

(*1) Designate group number "#60000".

23 System Variables

Variable No.	Item	Type	Details	Data range
60500 +***	Group No.	For each group/ registration number (*2)	This group's No.	1 to 99999999
61000 +***	Tool No.		Tool No. of the designated tool	1 to 99999999
61500 +***	Tool data flag		Parameters such as usage data count method, length compensation method, tool radius compensation method bit0,1: Tool length compensation data format bit2,3: Tool radius compensation data format 0: Compensation number method 1: Incremental value compensation amount method 2: Absolute value compensation amount method bit4,5: Tool life management method 0: Usage time 1: Number of mounts 2: Number of uses	0 to FF (H)
62000 +***	Tool status		Tool usage state 0: Unused tool 1: In-use tool 2: Normal life tool 3: Tool error 1 4: Tool error 2	0 to 4
62500 +***	Life data		Life time or No. of lives for each tool	0 to 4000 min 0 to 9999 sets
63000 +***	Usage data		Usage time or No. of uses for each tool	0 to 4000 min 0 to 9999 sets
63500 +***	Tool length compensation data		Length compensation data set as either compensation No. or length dimension method.	Compensation No.: "0" to the number of tool compensation sets Length dimension: ±99999.999
64000 +***	Tool radius compensation data		Radius compensation data set as either compensation No. or radius dimension method.	Compensation No.: "0" to the number of tool compensation sets Radius dimension: ±99999.999
64500 +***	Auxiliary data	Spare data	0 to 65535	

(*2) Designate group number "#60000" / registration number***.
However, group number is data common to groups.



Program example

(1) Normal commands

#101 = #60001 ;	Reads the number of registered tools.
#102 = #60002 ;	Reads the life current value.
#103 = #60003 ;	Reads the tool selection No.
#60000 = 10 ;	Designates the group No. of the life data to be read. Designated group No. is valid until reset.
#104 = #60004 ;	Reads the remaining number of registered tools in group 10.
#105 = #60005 ;	Reads the signal being executed in group 10.
#111 = #61001 ;	Reads the group 10, #1 tool No.
#112 = #62001 ;	Reads the group 10, #1 status.
#113 = #61002 ;	Reads the group 10, #2 tool No.
%	

(2) When the group number is not designated:

#104 = #60004 ;	Reads the remaining number of registered tools in the first registered group.
#111 = #61001 ;	Reads the #1 tool No. in the first registered group.
%	

(3) When an unregistered group number is designated (group 9999 does not exist):

#60000 = 9999 ;	Designates the group No.
#104 = #60004 ;	#104 = -1.

(4) When an unused registration number is designated (15 tools for group 10):

#60000 = 10 ;	Designates the group No.
#111 = #61016 ;	#111 = -1.

(5) When a registration number not defined in the specifications is designated:

#60000 = 10 ;	
#111 = #61017 ;	Program error (P241)

(6) When tool life management data is registered with G10 command after group No. is designated.

#60000 = 10 ;	Designates the group No.
G10 L3;	Starts the life management data registration. The group 10 life data is registered through the commands from G10 to G11.
P10 LLn NNn ;	10 is the group No., "Ln" is the life per tool, "Nn" is the method.
TTn ;	"Tn" is the tool No.
:	
G11 ;	Registers data in group 10 with the G10 command.
#111 = #61001 ;	Reads the group 10, #1 tool No.
G10 L3;	Starts the life management data registration. The life data other than group 10 is registered from G10 to G11.
P1 LLn NNn ;	1 is the group No., "Ln" is the life per tool, "Nn" is the method.
TTn ;	"Tn" is the tool No.
:	
G11 ;	Registers the life data with the G10 command. (The registered data is deleted.)
#111 = #61001 ;	Group 10 does not exist. #111 = -1.



Precautions

- (1) If the tool life management system variable is commanded without designating a group No., the data of the group registered at the head of the registered data will be read.
- (2) If a non-registered group No. is designated and the tool life management system variable is commanded, "-1" will be read as the data.
- (3) If an unused registration No. tool life management system variable is commanded, "-1" will be read as the data.
- (4) Once commanded, the group No. is valid until NC reset.
- (5) When the tool life management III are provided, the group No. other than 1 cannot be used.

23.8 System Variables (Workpiece Coordinate Offset)



Detailed description

By using variable Nos #5201 to #532n, it is possible to read out the workpiece coordinate system compensation data or to substitute values.

Note

- (1) The number of controllable axes varies depending on the specifications.
The last digit of the variable No. corresponds to the control axis No.

Coordinate name	1st axis	2nd axis	3rd axis	4th axis	nth axis	Remarks
External workpiece offset	#5201	#5202	#5203	#5204	#520n	External workpiece offset specifications are required.
G54	#5221	#5222	#5223	#5224	#522n	Workpiece coordinate system offset specifications are required.
G55	#5241	#5242	#5243	#5244	#524n	
G56	#5261	#5262	#5263	#5264	#526n	
G57	#5281	#5282	#5283	#5284	#528n	
G58	#5301	#5302	#5303	#5304	#530n	
G59	#5321	#5322	#5323	#5324	#532n	

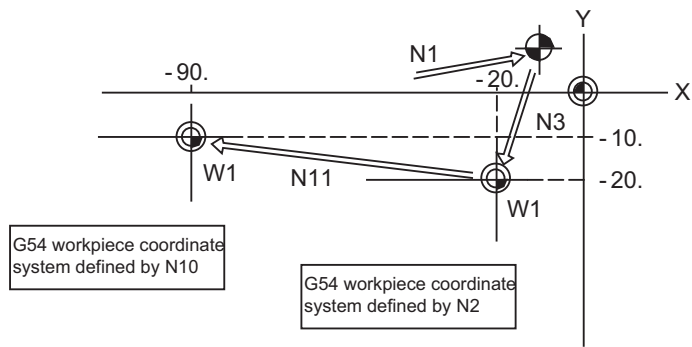
(Example 1)

```

N1 G28 X0 Y0 Z0 ;
N2 #5221=-20. #5222=-20. ;
N3 G90 G00 G54 X0 Y0 ;

N10 #5221=-90. #5222=-10. ;
N11 G90 G00 G54 X0 Y0 ;

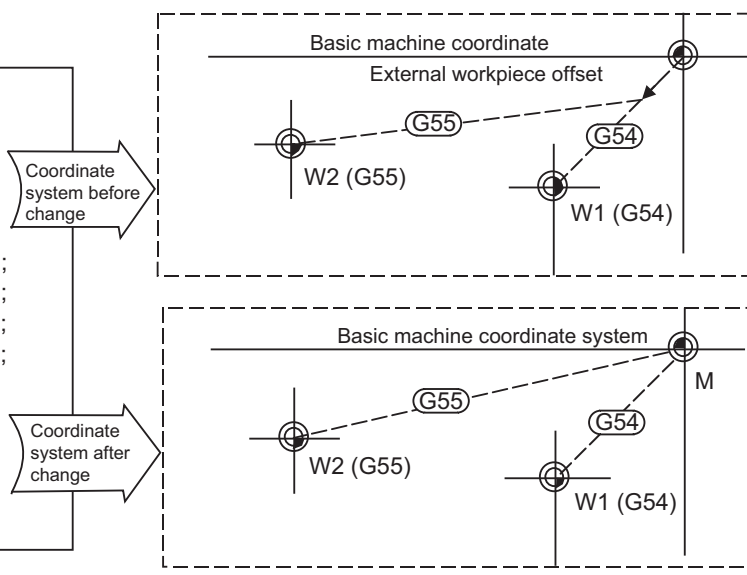
M02 ;
    
```



(Example 2)

```

N100 #5221=#5221+#5201 ;
      #5222=#5222+#5202 ;
      #5241=#5241+#5201 ;
      #5242=#5242+#5202 ;
      #5201=0 #5202=0 ;
    
```



This is an example where the external workpiece compensation values are added to the workpiece coordinate (G54, G55) system compensation values without changing the position of the workpiece coordinate systems.

23.9 System Variables (Extended Workpiece Coordinate Offset)



Detailed description

#7001 to #890n (48- or 96-set specification)

By using variable Nos #7001 to #890n, it is possible to read out the extended workpiece coordinate system compensation data or to substitute values.

Note

- (1) The system variables #7001 to #890n are available up to the valid number of sets. (You can use them for the 300-set specification also, but there are system variables corresponding to up to 96 sets only.) The last digit of the variable No. corresponds to the control axis No.

Table 1 of syst to em variables for extended workpiece coordinate system compensation (n=1 to 8)

	1st axis to nth axis		1st axis to nth axis		1st axis to nth axis		1st axis to nth axis
P1	#7001 to #700n	P25	#7481 to #748n	P49	#7961 to #796n	P73	#8441 to #844n
P2	#7021 to #702n	P26	#7501 to #750n	P50	#7981 to #798n	P74	#8461 to #846n
P3	#7041 to #704n	P27	#7521 to #752n	P51	#8001 to #800n	P75	#8481 to #848n
P4	#7061 to #706n	P28	#7541 to #754n	P52	#8021 to #802n	P76	#8501 to #850n
P5	#7081 to #708n	P29	#7561 to #756n	P53	#8041 to #804n	P77	#8521 to #852n
P6	#7101 to #710n	P30	#7581 to #758n	P54	#8061 to #806n	P78	#8541 to #854n
P7	#7121 to #712n	P31	#7601 to #760n	P55	#8081 to #808n	P79	#8561 to #856n
P8	#7141 to #714n	P32	#7621 to #762n	P56	#8101 to #810n	P80	#8581 to #858n
P9	#7161 to #716n	P33	#7641 to #764n	P57	#8121 to #812n	P81	#8601 to #860n
P10	#7181 to #718n	P34	#7661 to #766n	P58	#8141 to #814n	P82	#8621 to #862n
P11	#7201 to #720n	P35	#7681 to #768n	P59	#8161 to #816n	P83	#8641 to #864n
P12	#7221 to #722n	P36	#7701 to #770n	P60	#8181 to #818n	P84	#8661 to #866n
P13	#7241 to #724n	P37	#7721 to #772n	P61	#8201 to #820n	P85	#8681 to #868n
P14	#7261 to #726n	P38	#7741 to #774n	P62	#8221 to #822n	P86	#8701 to #870n
P15	#7281 to #728n	P39	#7761 to #776n	P63	#8241 to #824n	P87	#8721 to #872n
P16	#7301 to #730n	P40	#7781 to #778n	P64	#8261 to #826n	P88	#8741 to #874n
P17	#7321 to #732n	P41	#7801 to #780n	P65	#8281 to #828n	P89	#8761 to #876n
P18	#7341 to #734n	P42	#7821 to #782n	P66	#8301 to #830n	P90	#8781 to #878n
P19	#7361 to #736n	P43	#7841 to #784n	P67	#8321 to #832n	P91	#8801 to #880n
P20	#7381 to #738n	P44	#7861 to #786n	P68	#8341 to #834n	P92	#8821 to #882n
P21	#7401 to #740n	P45	#7881 to #788n	P69	#8361 to #836n	P93	#8841 to #884n
P22	#7421 to #742n	P46	#7901 to #790n	P70	#8381 to #838n	P94	#8861 to #886n
P23	#7441 to #744n	P47	#7921 to #792n	P71	#8401 to #840n	P95	#8881 to #888n
P24	#7461 to #746n	P48	#7941 to #794n	P72	#8421 to #842n	P96	#8901 to #890n

#101001 to #11595n (300-set specification)

By using variable Nos #101001 to #11595n, it is possible to read out the extended workpiece coordinate system compensation data or to substitute values.

Note

- (1) The system variables #101001 to #11595n are available when the 300-set specification is enabled. If you use the system variables #101001 to #11595n when the 300-set specification is disabled, the program error (P241) will occur.

The last digit of the variable No. corresponds to the control axis No.

Table 2 of syst to em variables for extended workpiece coordinate system compensation (n=1 to 8)

	1st axis to nth axis
P1	#101001 to #10100n
P2	#101051 to #10105n
P3	#101101 to #10110n
P4	#101151 to #10115n
P5	#101201 to #10120n
P6	#101251 to #10125n
P7	#101301 to #10130n
P8	#101351 to #10135n
:	:
:	:
P298	#115851 to #101585n
P299	#115901 to #101590n
P300	#118951 to #101595n

23.10 System Variables (External Workpiece Coordinate Offset)



Detailed description

The workpiece coordinate system compensation amount can be read using variables #2501 and #2601.

By substituting a value in these variable Nos., the workpiece coordinate system compensation amount can be changed.

System variable No.	External workpiece coordinate system offset amount
#2501	1st axis
#2601	2nd axis

23.11 System Variables (Position Information)



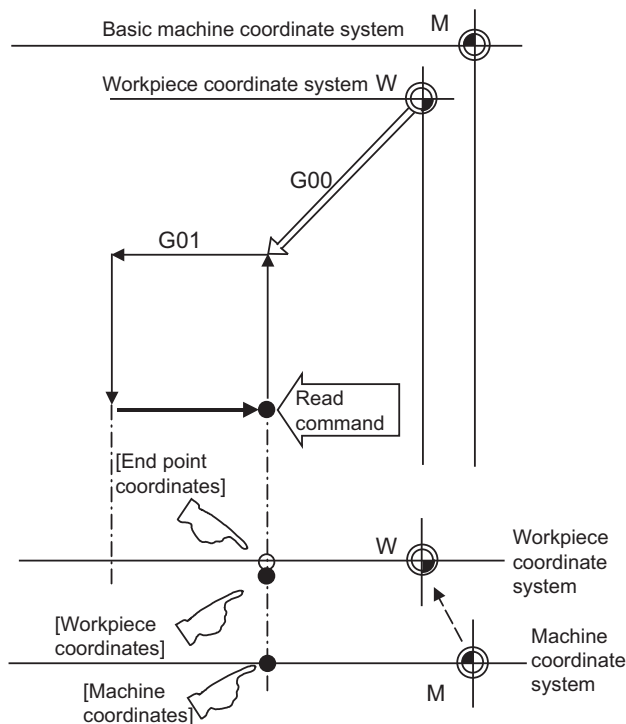
Detailed description

Using variable Nos. #5001 to #5160+n, it is possible to read the end point coordinates of the last block, machine coordinates, workpiece coordinates, skip coordinates, tool position compensation amount and servo deviation amounts.

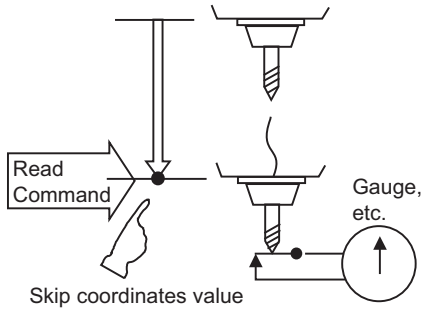
Position information			Axis No. (in part system)					Reading during movement	
			1	2	3	...	n		
End point coordinate of the last block			#5001	#5002	#5003	...	#5000+n	Enabled	
Machine coordinate			#5021	#5022	#5023	...	#5020+n	Disabled	
Workpiece coordinate			#5041	#5042	#5043	...	#5040+n	Disabled	
Skip coordinate	Parameters #8713	0	Workpiece coordinate system	#5061	#5062	#5063	...	#5060+n	Enabled
		1	Feature coordinate / Workpiece installation coordinate						
Feature coordinate/Workpiece installation coordinate			#5161	#5162	#5163	...	#5160+n		
Tool position compensation amount			#5081	#5082	#5083	...	#5080+n	Disabled	
Servo deviation amount			#5101	#5102	#5103	...	#5100+n	Enabled	
Start point coordinates of the block that was interrupted by macro interruption			#5121	#5122	#5123	...	#5120+n	Enabled	
End point coordinates of the block that was interrupted by macro interruption			#5141	#5142	#5143	...	#5140+n	Enabled	

Note

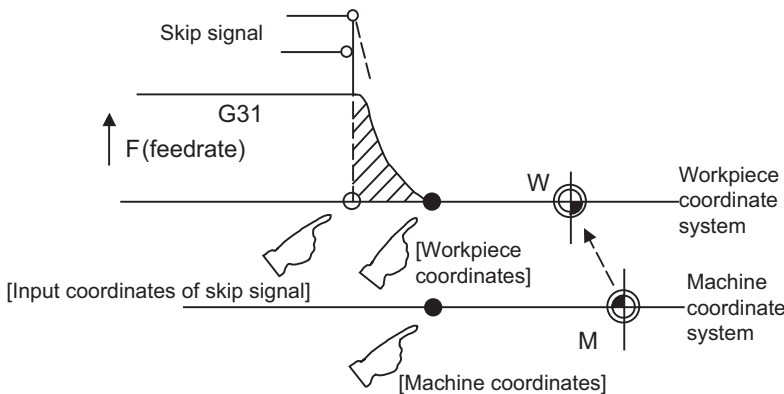
- The number of axes which can be controlled differs according to the specifications. The last digit of the variable No. corresponds to the control axis No.



- (1) The position of the end point coordinates is position on the workpiece coordinate system.
- (2) The end point coordinates, skip coordinates and servo deviation amounts can be read even during movement. However, it must first be checked that movement has stopped before reading the machine coordinates and the workpiece coordinates.
- (3) The skip coordinates indicates the position where the skip signal is turned ON in the G31 block. If the skip signal does not turn ON, they will be the end point position.
(For further details, refer to the section on Automatic Tool Length Measurement.)



- (4) The end point coordinates indicate the tool nose position regardless of the tool compensation and other such factors. On the other hand, the machine coordinates, workpiece coordinates and skip coordinates indicate the tool reference point position with consideration given to tool compensation.



For "●", check stop and then proceed to read.

For "○", reading is possible during movement.

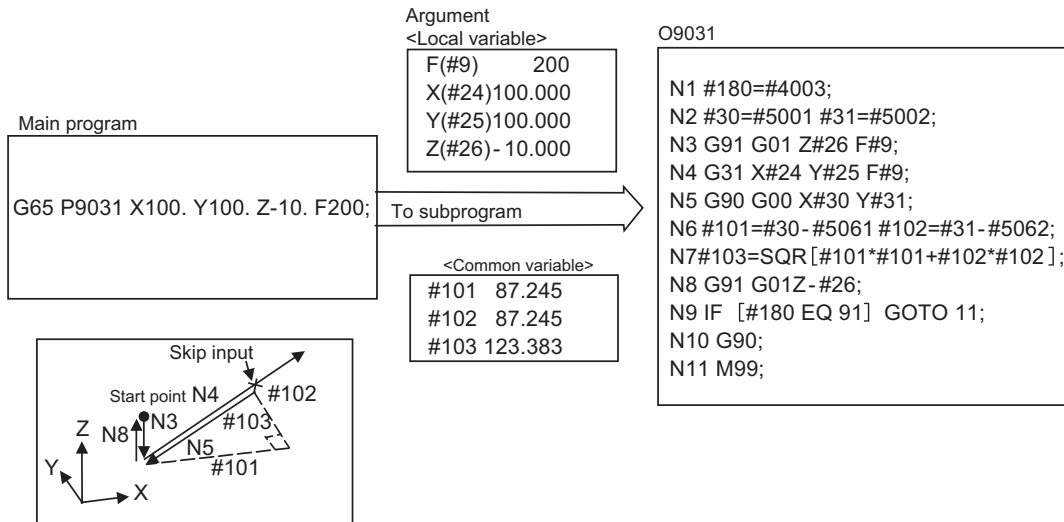
Note

- ◆ Multiple orthogonal axes move simultaneously while the inclined surface machining command is being executed.
The moved axis is updated at the workpiece coordinates of five orthogonal axes designated in the rotary axis configuration parameter. Therefore, if the skip signal is continuously input to the X, Y, and Z directions, the workpiece coordinate value of the skip coordinates is overwritten.
- ◆ Skip coordinate value is the position on the workpiece coordinate system, feature coordinate system, or workpiece installation coordinate system.
For #5061 to #5060+n, when the parameter "#8713 Skip coord. Switch" is set to "0", it is the position on the workpiece coordinate system, and when set to "1", it is the position on the feature coordinate system or workpiece installation coordinate system.
For #5161 to #5160+n, it is the position on the workpiece coordinate system while the inclined surface machining command or workpiece installation compensation is OFF.
For feature coordinate system, the skip coordinate value is on "the actual position where the tool length compensation is included" regardless of the setting of the parameter "#1287 ext23/bit1, bit2 (inclined surface coordinate display)".
- ◆ The values in the work installation coordinate system can be read for the orthogonal axes for the machining using 5 axes that have been set by the rotary axis configuration parameters. For the other axes, the values in the workpiece coordinate system are read.
- ◆ When the workpiece installation error compensation is OFF, the values in the skip coordinate system are read for all the axes.
- ◆ While the inclined surface machining command is disabled, the skip coordinate position is read in the workpiece coordinate system of all the axes.

- The coordinate value in variable Nos. #5061 to #5060+n or #5161 to #5160+n memorize the moments when the skip input signal during movement was input and so they can be read at any subsequent time. For details, refer to "22.2 Skip Function; G31".
- When the parameter "#1366 skipExTyp" (Multi-part system simultaneous skip command) is set to "1", the skip coordinate value will be "0", even if G31 command is given in one-part system or G31 command is given in only one of the multiple part systems.

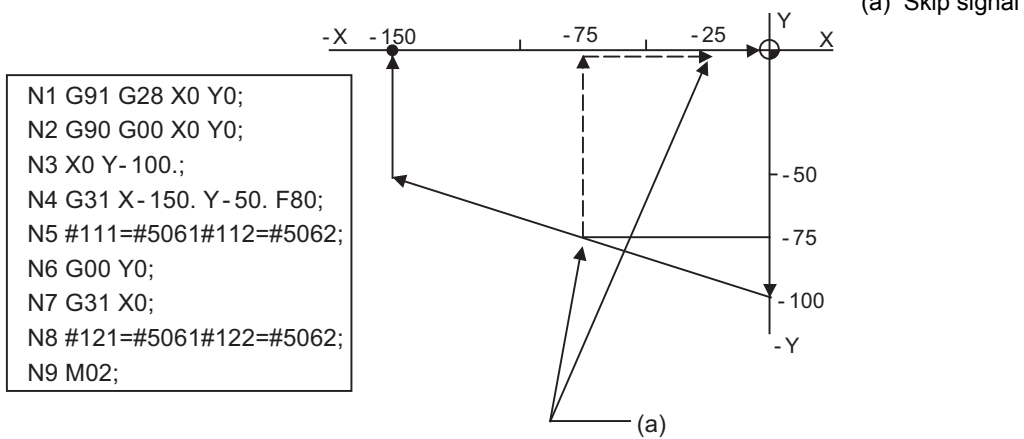
(Example 1) Example of workpiece position measurement

An example to measure the distance from the measured reference position to the workpiece edge is shown below.



#101	X axis measurement amount	N1	G90/G91 modal recording
#102	Y axis measurement amount	N2	X, Y start point recording
#103	Measurement linear segment amount	N3	Z axis entry amount
		N4	X, Y measurement (Stop at skip input)
#5001	X axis measurement start point	N5	Return to X, Y start point
#5002	Y axis measurement start point	N6	X, Y measurement incremental value calculation
		N7	Measurement linear segment calculation
#5061	X axis skip input point	N8	Z axis escape
#5062	Y axis skip input point	N9,N10	G90/G91 modal return
		N11	Main program return

(Example 2) Reading of skip input coordinates



```

N1 G91 G28 X0 Y0;
N2 G90 G00 X0 Y0;
N3 X0 Y-100.;
N4 G31 X-150. Y-50. F80;
N5 #111=#5061#112=#5062;
N6 G00 Y0;
N7 G31 X0;
N8 #121=#5061#122=#5062;
N9 M02;
    
```

#111=-75.+ε	#112=-75.+ε
#121=-25.+ε	#122=-75.+ε

"ε" is the error caused by response delay. Refer to "22.2 Skip Function; G31" for details.
 #122 is the N4 skip signal input coordinates as there is no Y command at N7.

23.12 System Variables (Alarm)



Detailed description

The NC unit can be forcibly set to the alarm state by using variable No. #3000.

#3000= 70 (CALL #PROGRAMMER #TEL #530) ;	
70	Alarm No.
CALL #PROGRAMMER #TEL #530	Alarm message

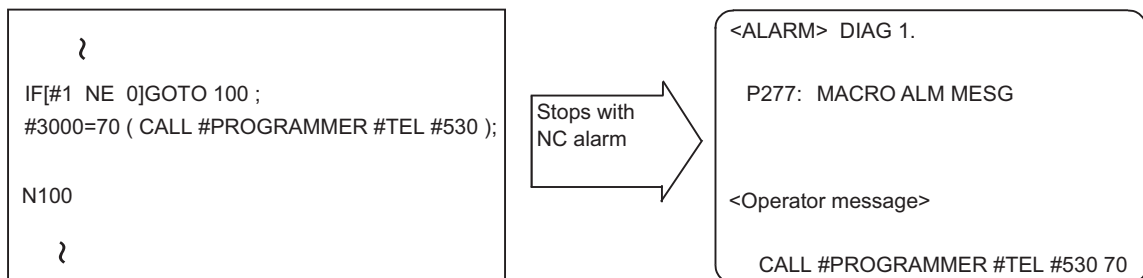
Any alarm number from 1 to 9999 can be specified.

The alarm message must be written in 31 or less characters.

NC alarm 3 signal (program error) is output.

The "P277: MACRO ALM MESG" appears in the <ALARM> column on "DIAG 1." screen and the alarm message " (CALL #PROGRAMMER #TEL #530)" and the alarm No. (70) will appear in the <Operator message>.

Example of program (alarm when #1 = 0)



Note

- (1) If zero or any number greater than 9999 is specified for the alarm No., the number will be invalid and it will not display. However, the operation will be in the alarm status, and the specified alarm message will appear.
- (2) Specify the alarm message by enclosing it in round parentheses after the alarm number. If there is any character string between the number and the alarm message enclosed in round parentheses, the alarm message will be invalid and it will not display. However, the operation will be in the alarm status, and the specified alarm No. will appear.
- (3) When 32 or more characters are specified for the alarm message, characters after the 32nd character will not display.
- (4) Spaces included in an alarm message character string are ignored, and will not display. To split the character string insert a character such as "." (period).

23.13 System Variables (Message Display and Stop)



Detailed description

By using variable No. #3006, the operation stops after the previous block is executed and, if message display data is commanded, the corresponding message and the stop No. will be indicated on the operator message area.

#3006 = 1(TAKE FIVE);	
1 to 9999	Stop No. (When Nos. other than 1 - 9999 are set, the command will be invalidated.)
TAKE FIVE	Message (Nothing will be displayed if no message is designated.)

The message should be written in 31 or less characters and should be enclosed by round parentheses.

23.14 System Variables (Cumulative Time)

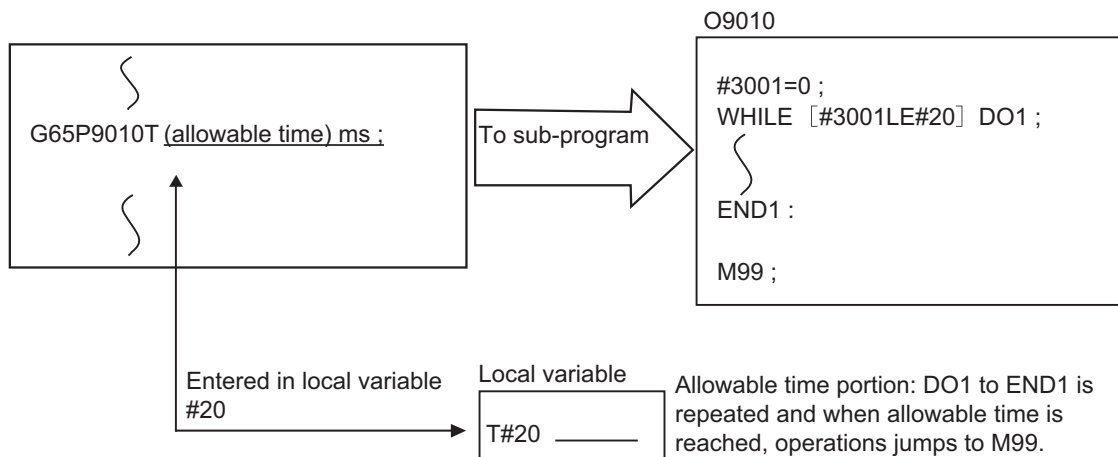


Detailed description

The integrating time during the power is turned ON or the automatic start is running, can be read or values can be substituted by using variable Nos. #3001 and #3002.

Type	Variable No.	Unit	Contents when power is switched on	Initialization of contents	Count condition
Power-on	3001	1ms	Same as when power is switched off	Substitute values to variables	At all times while power is ON
Automatic start	3002				In-automatic start

The cumulative time is reset to "0" at approximately 2.44×10^{11} ms (approximately 7.7 years).





Program example

Example of use (#3011, #3012)

(Example 1) To read the current date (February 14, 2001) in common variable #100

#100 = #3011 ; (20010214 is inserted in #100)

(Example 2) To write current time (18 hours, 13 minutes, 6 seconds) into system variable #3012

#3012 = 181306 ; (The command value cumulative time #2: time is set to 18:13:06.)

(Example 3) By setting the following program example, the machining start/end time (year/month/date/hour/minute/second) can be viewed.

```
#100=#3011 ; => Machining start year/month/date
#101=#3012 ; => Machining start hour/minute/second
G28 X0 Y0 Z0 ;

G92 ;
G0 X50. ;
:
:
:
#102=#3011 ; => Machining end year/month/date
#103=#3012 ; => Machining end hour/minute/second
M30 ;
```



Precautions

Limits and precautions for using time reading variable

- (1) #3011 reads the date as an eight-digit value, so the difference between the two dates read in will not be the difference of days.
- (2) #3012 reads the time as a six-digit value, so the difference between the two times read in will not be the difference of hours.

23.16 System Variables (Machining Information)



Detailed description

Contents of variable No. "#3003"

By substituting the values below in variable No. #3003, it is possible to suppress single block stop in the subsequent blocks or to advance to the next block without waiting for the miscellaneous function (M, S, T, B) finish (FIN) signal.

#3003/bit	Function	Set to "1"	Set to "0"
0	Inhibition of single block stop	Inhibits stop.	Does not inhibit stop.
1	Inhibition of miscellaneous function complete signal waiting	Does not wait for the signal.	Waits for the signal.
2	Prohibition of program check reverse run	Prohibits reverse run.	Allows reverse run.
3	(Not used)	-	-
4	(Not used)	-	-
5	(Not used)	-	-
6	(Not used)	-	-
7	(Not used)	-	-

Note

(1) Variable No. #3003 is set to zero by reset.

Contents of variable No. "#3004"

By substituting the values below in variable No. #3004, it is possible to make the feed hold, feedrate override and G09 functions either valid or invalid in the subsequent blocks.

#3004/bit	Function	Set to "1"	Set to "0"
0	Automatic operation pause OFF	Invalid	Valid
1	Cutting override OFF	Invalid	Valid
2	G09 check OFF	Invalid	Valid
3	(Not used)	-	-
4	Dry run invalid	Invalid	Valid
5	(Not used)	-	-
6	(Not used)	-	-
7	(Not used)	-	-

Note

(1) Variable No. #3004 is set to zero by reset.

(2) The functions are valid when the above bits are 0, and invalid when they are 1.

(3) When the feed hold is set to invalid with #3004, the following will occur when the feed hold switch is pressed.

- During thread cutting, block stop will be carried out at the end of the next block of the block where thread cutting is completed.
- During tapping with tap cycle, block stop will be carried out after the operation of R point return.
- In the case other than above, block stop will be carried out after the termination of the currently executing block.

23.17 System Variables (Reverse Run Information)



Detailed description

Variable No.	Usage	Details	Range
#31100	Number of available blocks for reverse run	+1 added number of the blocks that retained the reverse run information while the "Reverse run control mode" signal was ON	0 to 201
#31101	Counter of available blocks for reverse run	Number of available blocks for reverse run (value of #31100) when the "Reverse run" signal is turned ON to start. Turns "0" when the forward run has been executed for all the blocks. Displays "0" during normal operation.	0 to 201

23.18 System Variables (Number of Workpiece Machining Times)



Detailed description

The number of workpiece machining times can be read using variables #3901 and #3902.

By substituting a value in these variable Nos., the number of workpiece machining times can be changed.

Variable No.	Type	Data setting range
#3901	Number of workpiece machining times	0 to 999999
#3902	Maximum workpiece value	

Note

(1) The number of workpiece machining times must be a positive value.

23.19 System Variables (Mirror Image)



Detailed description

By reading variable No. #3007, it is possible to ascertain the status of mirror image of the each axis at the point.

The axis corresponds to each bit of "#3007" as shown below.

0: Mirror image invalid

1: Mirror image valid

The number of axes varies depending on your machine's specifications.

#3007

Bit	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
nth axis									8	7	6	5	4	3	2	1

23.20 System Variables (Coordinate Rotation Parameter)



Detailed description

The following variables can be read by the system variables of the variable command.

When an arbitrary axis is exchanged, this data is read in the axis arrangement that is set after the axis exchange has been completed.

Note that writing is not possible onto these variables.

Variable No.	Parameter No.	Description
#30060	#8621	Control axis No. on the coordinate rotation plane (horizontal axis)
#30061	#8622	Control axis No. on the coordinate rotation plane (vertical axis)
#30062	#8623	Coordinate rotation center (horizontal axis)
#30063	#8624	Coordinate rotation center (vertical axis)
#30064	#8627	Coordinate rotation angle
#30065	-	SIN data for the coordinate rotation angle [SIN(Coordinate rotation angle)]
#30066	-	COS data for the coordinate rotation angle [COS(Coordinate rotation angle)]
#30067	#8625	Coordinate rotation vector (horizontal axis)
#30068	#8626	Coordinate rotation vector (vertical axis)

23.21 System Variables (Rotary Axis Configuration Parameter)



Detailed description

The following rotary axis configuration parameters can be read by the system variables of the variable command. By substituting a value in these variables, the setting value of rotary axis configuration parameter can be changed.

Variable No.	Parameter	
#31001	#7903 G92_CRD	Origin zero set coordinate selection
#31002	#7904 NO_TIP	Tool handle feed function selection
#31003	#7920 SLCT_T1	Rotary axis selection (Base-side rotary axis of tool rotation type)
#31004	#7923 DIR_T1	Rotation direction (Base-side rotary axis of tool rotation type)
#31005	#7924 COFST1H	Horizontal axis rotation center offset (Base-side rotary axis of tool rotation type)
#31006	#7925 COFST1V	Vertical axis rotation center offset (Base-side rotary axis of tool rotation type)
#31007	#7926 COFST1T	Height axis rotation center offset (Base-side rotary axis of tool rotation type)
#31008	#7930 SLCT_T2	Rotary axis selection (Tool-side rotary axis of tool rotation type)
#31009	#7933 DIR_T2	Rotation direction (Tool-side rotary axis of tool rotation type)
#31010	#7934 COFST2H	Horizontal axis rotation center offset (Tool-side rotary axis of tool rotation type)
#31011	#7935 COFST2V	Vertical axis rotation center offset (Tool-side rotary axis of tool rotation type)
#31012	#7936 COFST2T	Height axis rotation center offset (Tool-side rotary axis of tool rotation type)
#31013	#7940 SLCT_W1	Rotary axis selection (Base-side rotary axis of table rotation type)
#31014	#7943 DIR_W1	Rotation direction (Base-side rotary axis of table rotation type)
#31015	#7944 COFSW1H	Horizontal axis rotation center offset (Base-side rotary axis of table rotation type)
#31016	#7945 COFSW1V	Vertical axis rotation center offset (Base-side rotary axis of table rotation type)
#31017	#7946 COFSW1T	Height axis rotation center offset (Base-side rotary axis of table rotation type)
#31018	#7950 SLCT_W2	Rotary axis selection (Workpiece-side rotary axis of table rotation type)
#31019	#7953 DIR_W2	Rotation direction (Workpiece-side rotary axis of table rotation type)
#31020	#7954 COFSW2H	Horizontal axis rotation center offset (Workpiece-side rotary axis of table rotation type)
#31021	#7955 COFSW2V	Vertical axis rotation center offset (Workpiece-side rotary axis of table rotation type)
#31022	#7956 COFSW2T	Height axis rotation center offset (Workpiece-side rotary axis of table rotation type)
#31023	#7912 NO_MANUAL	Selection of manual feed for 3-dimensional

23.22 System Variables (Normal Line Control Parameter)



Detailed description

The normal line control parameter can be read or written using variable Nos. "#1900" and "#1901".

Variable No.	Details	
#1900	#8041	C-rot.R (Data with decimal point)
#1901	#8042	C-ins.R (Data with decimal point)

23.23 System Variables (Parameter Reading)



Function and purpose

Parameter data can be read in with the system variables.

Variable No.	Application
#100000	Parameter No. designation
#100001	Part system No. designation
#100002	Axis No./spindle No. designation
#100010	Parameter value read



Detailed description

The parameter values are read in with the following four blocks using these four system variables.

#100000 = 1001 ;	Designates the parameter No.
#100001 = 1 ;	Designates the part system No.
#100002 = 1 ;	Designates the axis No./spindle No.
#100 = #100010;	Reads the parameter value.

Parameter No. designation (#100000)

The parameter to be read in is designated by substituting the parameter No. in this system variable.

If the parameters are read without designating this No., the parameters will be read in the same manner as if the minimum parameter No. (#1) is designated. Once designated, the setting is held until the parameter No. is designated again or until it is reset.

A program error (P39) will occur if a nonexistent parameter No. is set.

Part system No. designation (#100001)

(1) System variable for part system No. designation

The part system No. of the parameter to be read in is designated by substituting an index value for this system variable. This designation will be ignored when reading in parameters that are not in a specific part system.

If the parameters are read without designating this No., the parameters will be read in the same manner as if the index value 0 (part system in running program) is designated. Once designated, the setting is held until the part system No. is designated again or until it is reset.

A program error (P39) will occur if a nonexistent part system No. is set.

(2) Index values

Index values	Parameters per part system
0	Running part system
1	1st part system
2	2nd part system
3	-
:	-
9	-
10	PLC axis

Axis No. /spindle No. designation (#100002)

(1) System variable for axis or spindle No. designation

The axis or spindle No. of the parameter to be read in is designated by substituting an index value for this system variable. This designation will be ignored when reading in parameters that are neither for a specific axis nor spindle.

The axis parameter index value is the value set in the part system designated with #100001.

Thus, when reading parameters that are not in the designated part system, the part system No. must be designated again.

The spindle parameter's index value is not affected by the part system designation.

If the parameters are read without designating this number, the parameters will be read in the same manner as when the index value 1 (1st axis/1st spindle in the designated part system) is designated. Once designated, the setting is held until the index value is designated again or until it is reset.

A program error (P39) will occur if a nonexistent axis/spindle No. is set.

(2) Index values

Index values	Axis parameter	Spindle parameter
1	1st axis	1st spindle
2	2nd axis	2nd spindle
3	3rd axis	3rd spindle
4	4th axis	4th spindle
5	5th axis	-
6	6th axis	-

Reading the parameters (#100010)

The designated parameter data is read with this system variable.

Data to be read as follows, depending on the parameter type.

Type	Read in data
Numeric value	The values displayed on the Parameter screen are output.
Text	ASCII codes are converted into decimal values.



Program example

- (1) To read the parameter "#1002 axisno (number of axes)" for each part system:

```
#100000 = 1002 ;    Designates [#1002].
#100001 = 1 ;      Designates [1st part system].
#101 = #100010;    Reads the number of axes in 1st part system.
#100000 = 1002 ;    Designates [#1002]. (can be omitted since parameter No. is same)
#100001 = 2 ;      Designates [2nd part system].
#102 = #100010;    Reads the number of axes in 2nd part system.
#100001 = 5 ;      Designates [5th part system]. (The program error (P39) will occur.)
#100001 = 10 ;     Designates [PLC axis].
#110 = #100010;    Reads the number of PLC axes.
```

- (2) To read the axis parameter "#2037 G53ofs (#1 reference position)":

[Conditions]	1 part system		2 part systems	
	<1st axis>	<2nd axis>	<1st axis>	<2nd axis>
#2037 G53ofs	100.000	200.000	300.000	400.000

[1st part system program]

```
#100002 = 1 ;      Designates [1st axis].
#100000 = 2037 ;   Designates [#2037].
#101 = #100010;    Reads the [#1 reference point] for the 1st axis.
                    (#101=100.000)
#100002 = 2 ;      Designates [2nd axis].
#102 = #100010;    Reads the [#2 reference point] for the 1st axis.
                    (#102=200.000)
#100001 = 2 ;      Designates [2nd part system].
#100002 = 1 ;      Designates [1st axis].
#201 = #100010;    Reads the [#2 reference position] for the 1st axis in the 1st part system.
                    (#201=300.000)
```

[2nd part system program]

```
#100002 = 1 ;      Designates [1st axis].
#100000 = 2037 ;   Designates [#2037].
#101 = #100010;    Reads the [#1 reference point] for the 1st axis.
                    (#101=300.000)
#100002 = 2 ;      Designates [2nd axis].
#102 = #100010;    Reads the [#2 reference point] for the 1st axis.
                    (#102=400.000)
#100001 = 1 ;      Designates [1st part system].
#100002 = 1 ;      Designates [1st axis].
#201 = #100010;    Reads the [#1 reference position] for the 1st axis in the 1st part system.
                    (#201=100.000)
```

(3) To read the parameter for each part system, axis, or spindle:

#100002 = 1 ; Designates [1st spindle].
 #100000 = 3001 ; Designates [#3001].
 #101 = #100010; Reads the [#3001 slimt1 (Number of limit rotation gears 00)] for 1st spindle.
 #100000 = 3002 ; Designates [#3002].
 #102 = #100010; Reads the [#3002 slimt2 (Number of limit rotation gears 01)] for 1st spindle.
 #100002 = 2 ; Designates [2nd spindle].
 #100000 = 3001 ; Designates [#3001].
 #201 = #100010; Reads the [#3001 slimt1 (Number of limit rotation gears 00)] for 2nd spindle.
 #100000 = 3002 ; Designates [#3002].
 #202 = #100010; Reads the [#3002 slimt2 (Number of limit rotation gears 01)] for 2nd spindle.

(4) To read the text type parameter "#1169 system name" (part system name):

[Conditions]	<1st part system>	<2nd part system>
#1169 system name	SYS1	SYS2

#100000 = 1169 ; Designates #1169.
 #100001 = 1 ; Designates 1st part system.
 #101 = #100010; This will be #101 = 1398362929 (0x53595331).



Precautions

- (1) The number of part systems, axes and spindles is set at the maximum number specified by the model.
- (2) The inch/metric changeover function for the setting and display is valid for the readout data.
- (3) The machining condition parameter group cannot set the parameters from the program using the G10 command, and cannot read the parameters using the system variables ("#100000" and later).

23.24 System Variables (Workpiece Installation Error Compensation Amount)



Detailed description

Using the system variables below, read/write of the workpiece installation error compensation amounts is enabled.

	Common	No.01	No.02	No.03	No.04	No.05	No.06	No.07
Workpiece installation error compensation amount Δx	#26000	#26010	#26020	#26030	#26040	#26050	#26060	#26070
Workpiece installation error compensation amount Δy	#26001	#26011	#26021	#26031	#26041	#26051	#26061	#26071
Workpiece installation error compensation amount Δz	#26002	#26012	#26022	#26032	#26042	#26052	#26062	#26072
Workpiece installation error compensation amount Δa	-	#26013	#26023	#26033	#26043	#26053	#26063	#26073
Workpiece installation error compensation amount Δb	-	#26014	#26024	#26034	#26044	#26054	#26064	#26074
Workpiece installation error compensation amount Δc	-	#26015	#26025	#26035	#26045	#26055	#26065	#26075
Primary rotary axis position	#26006	#26016	#26026	#26036	#26046	#26056	#26066	#26076
Secondary rotary axis position	#26007	#26017	#26027	#26037	#26047	#26057	#26067	#26077

(Note 1) The primary rotary axis position corresponds with the axis set by the parameter #7942, and the secondary rotary axis position corresponds with the axis set by the parameter #7952.

(Note 2) If the primary and secondary rotary axis positions are not of the table-side rotary axes, the set values are ignored.

(Note 3) The setting ranges are the same as those set in the workpiece installation error setting screen.

(Note 4) If the system variables #26000 to #26077 are written during workpiece installation error compensation, the program error (P545) will occur.

23.25 System Variables (Macro Interface Input (PLC -> NC))



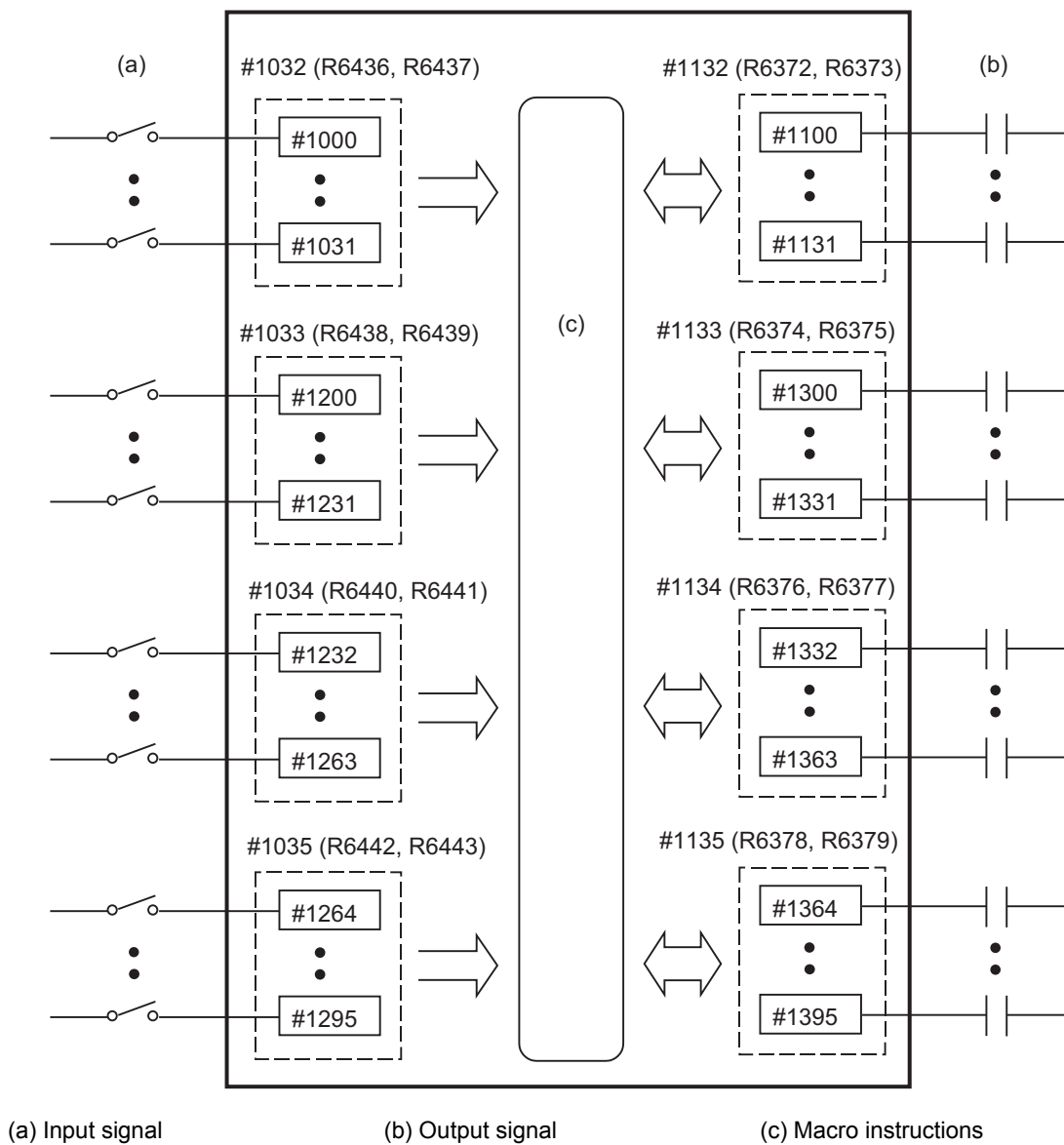
Function and purpose

The status of the interface input signals can be ascertained by reading out the values of variable Nos. #1000 to #1035, #1200 to #1295.

Note

- The interface output signals can be sent by substituting values in variable Nos. #1100 to #1135, #1300 to #1395. (For details of the system variables for the output signals, refer to "23.26 System Variables (Macro Interface Output (NC -> PLC))".)

Example of 1st part system





Detailed description

Variable Nos. #1000 to #1035, #1200 to #1295 are for readout only, and nothing can be placed in the left side member of their operation formula.

Input here refers to input to the NC.

Whether it is per part system or common between part systems depends on the MTB specifications (parameter "#1230 set02/bit07").

Data unit (32 bits)

All the input signals from #1000 to #1031 can be read at once by reading out the value of variable No. #1032.

The input signals from #1200 to #1231, #1232 to #1263, and #1264 to #1295 can be read by reading out the values of variable Nos. #1033 to #1035.

The data of the 1st part system (\$1) to the 8th part system (\$8) is as follows.

System variable	No. of points	Interface input signal							
		\$1	\$2	\$3	\$4	\$5	\$6	\$7	\$8
#1032	32	R6436, R6437	R6444, R6445	R6452, R6453	R6460, R6461	R6468, R6469	R6476, R6477	R6484, R6485	R6492, R6493
#1033	32	R6438, R6439	R6446, R6447	R6454, R6455	R6462, R6463	R6470, R6471	R6478, R6479	R6486, R6487	R6494, R6495
#1034	32	R6440, R6441	R6448, R6449	R6456, R6457	R6464, R6465	R6472, R6473	R6480, R6481	R6488, R6489	R6496, R6497
#1035	32	R6442, R6443	R6450, R6451	R6458, R6459	R6466, R6467	R6474, R6475	R6482, R6483	R6490, R6491	R6498, R6499

Bit unit

The input signal has only two values: "0" and "1".

Part system	\$1	\$2	\$3	\$4	\$5	\$6	\$7	\$8
R device	R6436- R6443	R6444- R6451	R6452- R6459	R6460- R6467	R6468- R6475	R6476- R6483	R6484- R6491	R6492- R6499

If the value is common between part systems, refer to the column of the 1st part system (\$1).

System variable	No. of points	Interface input signal register							
		\$1	\$2	\$3	\$4	\$5	\$6	\$7	\$8
#1000	1	R6436/ bit0	R6444/ bit0	R6452/ bit0	R6460/ bit0	R6468/ bit0	R6476/ bit0	R6484/ bit0	R6492/ bit0
#1001	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1002	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1003	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1004	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1005	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1006	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1007	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1008	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1009	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1010	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1011	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1012	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1013	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1014	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1015	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1016	1	R6437/ bit0	R6445/ bit0	R6453/ bit0	R6461/ bit0	R6469/ bit0	R6477/ bit0	R6485/ bit0	R6493/ bit0
#1017	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1018	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1019	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1020	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1021	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1022	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1023	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1024	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1025	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1026	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1027	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1028	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1029	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1030	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1031	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

System variable	No. of points	Interface input signal register							
		\$1	\$2	\$3	\$4	\$5	\$6	\$7	\$8
#1200	1	R6438/ bit0	R6446/ bit0	R6454/ bit0	R6462/ bit0	R6470/ bit0	R6478/ bit0	R6486/ bit0	R6494/ bit0
#1201	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1202	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1203	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1204	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1205	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1206	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1207	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1208	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1209	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1210	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1211	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1212	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1213	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1214	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1215	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1216	1	R6439/ bit0	R6447/ bit0	R6455/ bit0	R6463/ bit0	R6471/ bit0	R6479/ bit0	R6487/ bit0	R6495/ bit0
#1217	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1218	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1219	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1220	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1221	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1222	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1223	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1224	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1225	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1226	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1227	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1228	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1229	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1230	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1231	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

System variable	No. of points	Interface input signal register							
		\$1	\$2	\$3	\$4	\$5	\$6	\$7	\$8
#1232	1	R6440/ bit0	R6440/ bit0	R6448/ bit0	R6456/ bit0	R6472/ bit0	R6480/ bit0	R6488/ bit0	R6496/ bit0
#1233	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1234	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1235	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1236	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1237	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1238	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1239	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1240	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1241	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1242	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1243	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1244	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1245	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1246	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1247	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1248	1	R6441/ bit0	R6441/ bit0	R6449/ bit0	R6457/ bit0	R6473/ bit0	R6481/ bit0	R6489/ bit0	R6497/ bit0
#1249	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1250	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1251	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1252	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1253	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1254	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1255	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1256	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1257	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1258	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1259	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1260	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1261	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1262	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1263	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

System variable	No. of points	Interface input signal register							
		\$1	\$2	\$3	\$4	\$5	\$6	\$7	\$8
#1264	1	R6442/ bit0	R6450/ bit0	R6458/ bit0	R6466/ bit0	R6474/ bit0	R6482/ bit0	R6490/ bit0	R6498/ bit0
#1265	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1266	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1267	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1268	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1269	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1270	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1271	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1272	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1273	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1274	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1275	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1276	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1277	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1278	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1279	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1280	1	R6443/ bit0	R6451/ bit0	R6459/ bit0	R6467/ bit0	R6475/ bit0	R6483/ bit0	R6491/ bit0	R6499/ bit0
#1281	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1282	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1283	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1284	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1285	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1286	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1287	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1288	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1289	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1290	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1291	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1292	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1293	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1294	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1295	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

23.26 System Variables (Macro Interface Output (NC -> PLC))



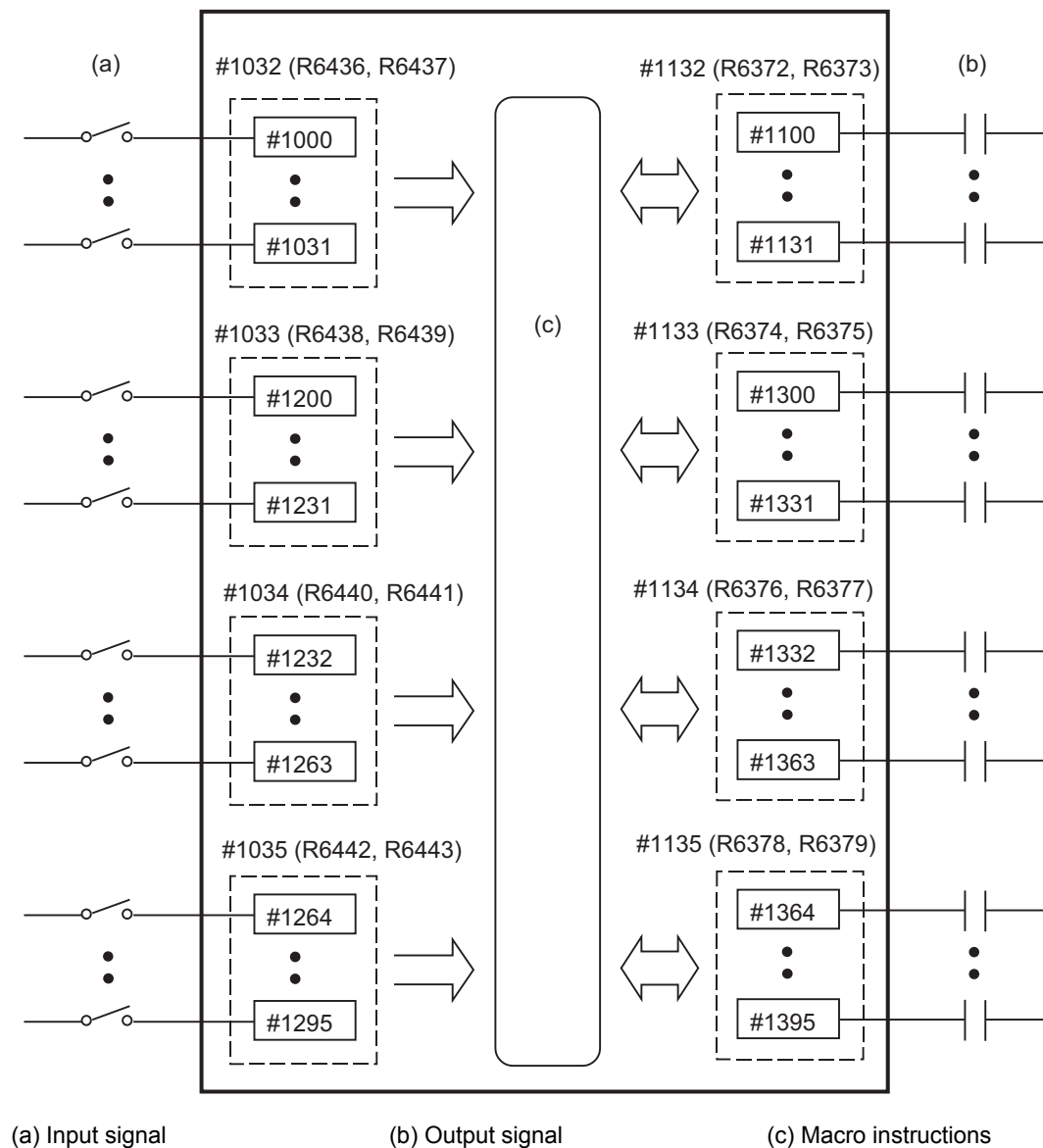
Function and purpose

The interface output signals can be sent by substituting values in variable Nos. #1100 to #1135, #1300 to #1395.

Note

- The status of the interface input signals can be ascertained by reading out the values of variable Nos. #1000 to #1035, #1200 to #1295. (For details of the system variables for the output signals, refer to "23.25 System Variables (Macro Interface Input (PLC -> NC))".)

Example of 1st part system





Detailed description

The status of the writing and output signals can be read in order to compensate the #1100 to #1135, #1300 to #1395 output signals.

Output here refers to the output from the NC side.

Whether it is per part system or common between part systems depends on the MTB specifications (parameter "#1230 set02/bit07").

Note

- (1) The last values of the system variables #1100 to #1135, #1300 to #1395 sent are retained as 1 or 0. (They are not cleared even by resetting.)
- (2) The following applies when any number except 1 or 0 is substituted into #1100 to #1131, #1300 to #1395.
<Blank> is treated as 0. All values other than <blank> or "0" are treated as 1.
Any value less than 0.00000001 is indefinite.

Data unit (32 bits)

All the output Nos. from #1100 to #1131 can be sent at once by substituting a value in variable No. #1132.

The output signals from #1300 to #1331, #1332 to #1363, and #1364 to #1395 can be sent by substituting a value in variable Nos. #1133 to #1135. (2^0 to 2^{31})

The data of the 1st part system (\$1) to the 8th part system (\$8) is as follows.

System variable	No. of points	Interface output signal							
		\$1	\$2	\$3	\$4	\$5	\$6	\$7	\$8
#1132	32	R6372, R6373	R6380, R6381	R6388, R6389	R6396, R6397	R6404, R6405	R6412, R6413	R6420, R6421	R6428, R6429
#1133	32	R6374, R6375	R6382, R6383	R6390, R6391	R6398, R6399	R6406, R6407	R6414, R6415	R6422, R6423	R6430, R6431
#1134	32	R6376, R6377	R6384, R6385	R6392, R6393	R6400, R6401	R6408, R6409	R6416, R6417	R6424, R6425	R6432, R6433
#1135	32	R6378, R6379	R6386, R6387	R6394, R6395	R6402, R6403	R6410, R6411	R6418, R6419	R6426, R6427	R6434, R6435

Bit unit

The output signal has only two values: "0" and "1".

Part system	\$1	\$2	\$3	\$4	\$5	\$6	\$7	\$8
R device	R6372- R6379	R6380- R6387	R6388- R6395	R6396- R6403	R6404- R6411	R6412- R6419	R6420- R6427	R6428- R6435

If the value is common between part systems, refer to the column of the 1st part system (\$1).

System variable	No. of points	Interface output signal register							
		\$1	\$2	\$3	\$4	\$5	\$6	\$7	\$8
#1100	1	R6372/ bit0	R6380/ bit0	R6388/ bit0	R6396/ bit0	R6404/ bit0	R6412/ bit0	R6420/ bit0	R6428/ bit0
#1101	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1102	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1103	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1104	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1105	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1106	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1107	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1108	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1109	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1110	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1111	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1112	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1113	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1114	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1115	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1116	1	R6373/ bit0	R6381/ bit0	R6389/ bit0	R6397/ bit0	R6405/ bit0	R6413/ bit0	R6421/ bit0	R6429/ bit0
#1117	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1118	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1119	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1120	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1121	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1122	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1123	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1124	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1125	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1126	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1127	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1128	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1129	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1130	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1131	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

System variable	No. of points	Interface output signal register							
		\$1	\$2	\$3	\$4	\$5	\$6	\$7	\$8
#1300	1	R6374/ bit0	R6382/ bit0	R6390/ bit0	R6398/ bit0	R6406/ bit0	R6414/ bit0	R6422/ bit0	R6430/ bit0
#1301	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1302	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1303	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1304	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1305	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1306	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1307	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1308	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1309	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1310	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1311	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1312	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1313	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1314	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1315	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1316	1	R6375/ bit0	R6383/ bit0	R6391/ bit0	R6399/ bit0	R6407/ bit0	R6415/ bit0	R6423/ bit0	R6431/ bit0
#1317	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1318	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1319	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1320	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1321	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1322	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1323	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1324	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1325	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1326	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1327	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1328	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1329	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1330	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1331	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

System variable	No. of points	Interface output signal register							
		\$1	\$2	\$3	\$4	\$5	\$6	\$7	\$8
#1332	1	R6376/ bit0	R6384/ bit0	R6392/ bit0	R6400/ bit0	R6408/ bit0	R6416/ bit0	R6424/ bit0	R6432/ bit0
#1333	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1334	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1335	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1336	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1337	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1338	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1339	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1340	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1341	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1342	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1343	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1344	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1345	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1346	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1347	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1348	1	R6377/ bit0	R6385/ bit0	R6393/ bit0	R6401/ bit0	R6409/ bit0	R6417/ bit0	R6425/ bit0	R6433/ bit0
#1349	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1350	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1351	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1352	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1353	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1354	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1355	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1356	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1357	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1358	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1359	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1360	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1361	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1362	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1363	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

System variable	No. of points	Interface output signal register							
		\$1	\$2	\$3	\$4	\$5	\$6	\$7	\$8
#1364	1	R6378/ bit0	R6386/ bit0	R6394/ bit0	R6402/ bit0	R6410/ bit0	R6418/ bit0	R6426/ bit0	R6434/ bit0
#1365	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1366	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1367	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1368	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1369	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1370	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1371	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1372	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1373	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1374	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1375	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1376	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1377	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1378	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1379	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1380	1	R6379/ bit0	R6387/ bit0	R6395/ bit0	R6403/ bit0	R6411/ bit0	R6419/ bit0	R6427/ bit0	R6435/ bit0
#1381	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1382	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1383	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1384	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1385	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1386	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1387	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1388	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1389	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1390	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1391	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1392	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1393	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1394	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1395	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

23.27 System Variables (R Device Access Variables)



Function and purpose

By using variable Nos. #50000 to #50749, #51000 to #51749, #52000 to #52749, it is possible to read data (R8300 to R9799, R18300 to R19799, R28300 to R29799) and substitute value in the R device user backup area.

Variable No.	R device	
#50000	R8300, R8301	User backup area (1500 points)
#50001	R8302, R8303	
:		
#50749	R9798, R9799	
Variable No.	R device	
#51000	R18300, R18301	User backup area (1500 points)
#51001	R18302, R18303	
:		
#51749	R19798, R19799	
Variable No.	R device	
#52000	R28300, R28301	User backup area (1500 points)
#52001	R28302, R28303	
:		
#52749	R29798, R29799	



Detailed description

These variables read and write the two words of R device.

Data range of these variables is -2147483648 to 2147483647.

Depending on the setting of the PLC bit selection parameter "#6455" (bit0 to bit2), these variables can be changed between decimal point valid or invalid for each user backup area.

The position of the decimal point when decimal point valid is selected, varies according to the parameters "#1003 iunit" (inupt setting unit) and "#1041 I_inch" (initial inch). (This depends on the MTB specifications.)

#1041 I_inch	#1003 iunit			
	B	C	D	E
0: Metric	Three digits after the decimal point	Four digits after the decimal point	Five digits after the decimal point	Six digits after the decimal point
1: Inch	Four digits after the decimal point	Five digits after the decimal point	Six digits after the decimal point	Seven digits after the decimal point

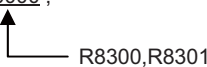
These variables are retained even when the power is off.

These are common among part systems.

Access from a machining program to R device

[Reading variables]

When the variable #50000 is used in a machining program as shown below, the data set in device R8300 and R8301 will be referred.

G0 X#50000 ; 	Device	Value	#50000
	R8301	0x0001	0x1e240 (Hex.) = 123456 (Decimal)
	R8300	0xe240	

(1) When decimal point invalid is selected:

Regardless of the setting of the parameter "#1003 iunit" (input setting unit) and "#1041 I_inch" (initial inch), the data set in the R device will be the command value.

In case of the above example, the command value will be "X123456."

(2) When decimal point valid is selected:

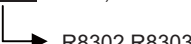
The data set in the R device will be read as a data with a decimal point.

The position of the decimal point will be as follows, according to the settings of the parameters "#1003 iunit" (input setting unit) and "#1041 I_inch" (initial inch).

#1041 I_inch	#1003 iunit			
	B	C	D	E
0: Metric	X123.456	X12.3456	X1.23456	X0.123456
1: Inch	X12.3456	X1.23456	X0.123456	X0.0123456

[Substitution into variables]

When substituting a value to the variable #50001 in a machining program as shown below, data will be set in the device R8302 and R8303.

#50001 = 123 ;


(1) When decimal point invalid is selected:

Regardless of the setting of the parameter "#1003 iunit" (input setting unit) and "#1041 I_inch" (initial inch), substituted value will be set in the R device.

#50001	Device	Value
123 (Decimal) = 0x7b (Hex.)	R8303	0x0000
	R8302	0x007b

When a value with a decimal point is substituted to a variable like "#50001 = 123.456 ;", the numbers after the decimal point will be truncated and "123" will be set.

(2) When decimal point valid is selected:

According to the settings of the parameter "#1003 iunit" (inupt setting unit) and "#1041 I_inch" (initial inch), values which are shifted for the number of decimals will be set in the R device, as shown below.

#1041 I_inch		0: Metric			
#1003 iunit		B	C	D	E
#50001		123000 (Decimal) = 0x1e078 (Hex.)	1230000 (Decimal) = 0x12c4b0 (Hex.)	12300000 (Decimal) = 0xbbaee0 (Hex.)	123000000 (Decimal) = 0x754d4c0 (Hex.)
Device	R8303	0x0001	0x0012	0x00bb	0x0754
	R8302	0xe078	0xc4b0	0xae0	0xd4c0
#1041 I_inch		1: Inch			
#1003 iunit		B	C	D	E
#50001		1230000 (Decimal) = 0x12c4b0 (Hex.)	12300000 (Decimal) = 0xbbaee0 (Hex.)	123000000 (Decimal) = 0x754d4c0 (Hex.)	1230000000 (Decimal) = 0x49504f80 (Hex.)
Device	R8303	0x0012	0x00bb	0x0754	0x4950
	R8302	0xc4b0	0xae0	0xd4c0	0x4f80

When a value with a decimal point is substituted to a variable like "#50001 = 123.456 ;", the value will directly be set.

#1041 I_inch		0: Metric			
#1003 iunit		B	C	D	E
#50001		123456 (Decimal) = 0x1e240 (Hex.)	1234560 (Decimal) = 0x12d680 (Hex.)	12345600 (Decimal) = 0xbc6100 (Hex.)	123456000 (Decimal) = 0x75bca00 (Hex.)
Device	R8303	0x0001	0x0012	0x00bc	0x075b
	R8302	0xe240	0xd680	0x6100	0xca00
#1041 I_inch		1: Inch			
#1003 iunit		B	C	D	E
#50001		1234560 (Decimal) = 0x12d680 (Hex.)	12345600 (Decimal) = 0xbc6100 (Hex.)	123456000 (Decimal) = 0x75bca00 (Hex.)	1234560000 (Decimal) = 0x4995e400 (Hex.)
Device	R8303	0x0012	0x00bc	0x075b	0x4998
	R8302	0xd680	0x6100	0xca00	0xe400

If the number of decimals of the substituted data exceeds the number of significant figures, the value will be rounded off to the number of significant figures and will be set.

When "#50001 = 123.4567899 ;",

#1041 I_inch		0: Metric			
#1003 iunit		B	C	D	E
#50001		123457 (Decimal) = 0x1e241 (Hex.)	1234568 (Decimal) = 0x12d688 (Hex.)	12345679 (Decimal) = 0xbc614f (Hex.)	123456790 (Decimal) = 0x75bcd16 (Hex.)
Device	R8303	0x0001	0x0012	0x00bc	0x075b
	R8302	0xe241	0xd688	0x614f	0xcd16
#1041 I_inch		1: Inch			
#1003 iunit		B	C	D	E
#50001		1234568 (Decimal) = 0x12d688 (Hex.)	12345679 (Decimal) = 0xbc614f (Hex.)	123456790 (Decimal) = 0x75bcd16 (Hex.)	1234567899 (Decimal) = 0x499602db (Hex.)
Device	R8303	0x0012	0x00bc	0x075b	0x4996
	R8302	0xd688	0x614f	0xcd16	0x02db

Use of R device access variables in control command

These variables can be used in control command.

However, note that the variable value and the condition of true/false differ between decimal point valid variables and invalid variables.

```
IF [#50003 EQ 1] GOTO 30 ;
G00 X100 ;
N30
```

(1) When decimal point invalid is selected:

Regardless of the setting of the parameter "#1003 iunit" (input setting unit) and "#1041 I_inch" (initial inch), R device value of #50003 whose condition is true, will be "1".

#50003	Device	Value
1 (Decimal) = 0x01 (Hex.)	R8307	0x0000
	R8306	0x0001

(2) When decimal point valid is selected:

The condition is true when #50003 is "1". So the R device value of #50003 will be as follows depending on the setting of the parameter "#1003 iunit" (input setting unit) and "#1041 I_inch" (initial inch).

#1041 I_inch	0: Metric				
#1003 iunit	B	C	D	E	
#50003	1000 (Decimal) = 0x3e8 (Hex.)	10000 (Decimal) = 0x2710 (Hex.)	100000 (Decimal) = 0x186a0 (Hex.)	1000000 (Decimal) = 0xf4240 (Hex.)	
Device	R8307	0x0000	0x0000	0x0001	0x000f
	R8306	0x03e8	0x2710	0x86a0	0x4240
#1041 I_inch	1: Inch				
#1003 iunit	B	C	D	E	
#50003	10000 (Decimal) = 0x2710 (Hex.)	100000 (Decimal) = 0x186a0 (Hex.)	1000000 (Decimal) = 0xf4240 (Hex.)	10000000 (Decimal) = 0x989680 (Hex.)	
Device	R8307	0x0000	0x0001	0x000f	0x0098
	R8306	0x2710	0x86a0	0x4240	0x9680

Substitution between R device access variable and other variables.**[Substitution into R device access variables]**

Common variables and coordinates variables can be substituted to the R device access variables.

(Example 1) Common variables

```
#101 = -123.456 ;
#50004 = #101 ;
```

(Example 2) #5063 : Skip coordinates #5063

```
#50004 = #5063 ;
```

(1) When decimal point invalid is selected:

Regardless of the settings of the parameter "#1003 iunit" (input setting unit) and "#1041 I_inch" (initial inch), the value which is rounded off will be set.

When the common variable and coordinate variable in the above example are "-123.456":

#50004	Device	Value
-123 (Decimal) = 0xfffff85 (Hex.)	R8309	0xffff
	R8308	0x0085

(2) When decimal point valid is selected:

Substitution will be as follows according to the settings of the parameter "#1003 iunit" (input setting unit) and "#1041 I_inch" (initial inch).

#1041 I_inch	0: Metric				
#1003 iunit	B	C	D	E	
#50004	-123456 (Decimal) = 0xfffe1dc0 (Hex.)	-1234560 (Decimal) = 0xffed2980 (Hex.)	-12345600 (Decimal) = 0xff439f00 (Hex.)	-123456000 (Decimal) = 0xf8a43600 (Hex.)	
Device	R8309	0xfffe	0xffed	0xff43	0xf8a4
	R8308	0x1dc0	0x2980	0x9f00	0x3600
#1041 I_inch	1: Inch				
#1003 iunit	B	C	D	E	
#50004	-1234560 (Decimal) = 0xffed2980 (Hex.)	-12345600 (Decimal) = 0xff439f00 (Hex.)	-123456000 (Decimal) = 0xf8a43600 (Hex.)	-1234560000 (Decimal) = 0xb66a1c00 (Hex.)	
Device	R8309	0xffed	0xff43	0xf8a4	0xb66a
	R8308	0x2980	0x9f00	0x3600	0x1c00

[Substitution of R device access variables]

```
#50005 = 123.456789 ;
#102 = #50005 ;
```

- (1) When decimal point invalid is selected:

Regardless of the settings of the parameter "#1003 iunit" (input setting unit) and "#1041 I_inch" (initial inch), #102 will be "123".

- (2) When decimal point valid is selected:

Substitution will be as follows according to the settings of the parameter "#1003 iunit" (input setting unit) and "#1041 I_inch" (initial inch).

#1041 I_inch	0: Metric			
#1003 iunit	B	C	D	E
#102	123.4570	123.4568	123.4568	123.4568
#1041 I_inch	1: Inch			
#1003 iunit	B	C	D	E
#102	123.4568	123.4568	123.4568	123.4568

**Precautions**

- (1) The position of a decimal point changes depending on the settings of the parameter "#1003 iunit" (input setting unit) and "#1041 I_inch" (initial inch). Fix the decimal point position while considering these parameter settings when setting a number to an R device.
- (2) These variables do not handle <Blank>. If #0<Blank> is substituted, it will be converted into "0".
Therefore, when comparing this variable after substituting #0<Blank> and #0<Blank> with a conditional expression (EQ), it will not be formed.
- (3) If a value exceeding the allowable range is substituted into this variable, a program error (P35) will occur.
- (4) When these values are used as decimal point invalid, the settings of "#1078 Decpt2" (Decimal point type 2) and "#8044 UNIT*10" will not be applied.
- (5) When a graphic is being checked, writing into R device will not be executed even if a value is substituted into these variables.
For reading of these variables (reference to the R device value) during a graphic check, "0" is always read.

23.28 System Variables (PLC Data Reading)



Function and purpose

PLC data can be read in with the system variables.

Variable No.	Application
#100100	Device type designation
#100101	Device No. designation
#100102	Number of read bytes designation
#100103	Read bit designation
#100110	Reading PLC data

Note

- (1) These can be used only with some models.
- (2) The readable devices are limited.



Detailed description

The PLC data is read in with the following five blocks using these five system variables.

- #100100 = 1; Designates the device type.
- #100101 = 0; Designates the device No.
- #100102 = 1; Designates the number of bytes.
- #100103 = 2; Designates the bit. (Valid only when reading word device bits.)
- #100=#100110; Reads in the PLC data.

Device designation (#100100)

- (1) System variable for device designation

The type of device to be read in can be designated by substituting the device designation value in this system variable.

If the data is read without designating this variable, the data will be read in the same manner as when the minimum value (0: M device) of the device designation value is designated. Once designated, the setting is held until the device is designated again or until it is reset.

A program error (P39) will occur if a nonexistent device is set.

- (2) Device designation value

[M800/M80 series]

Device designation value	Device	Unit	Device No.	Device designation value	Device	Unit	Device No.
0	M	Bit	M0 to M61439	10	F	Bit	F0 to F2047
1	D	Word	D0 to D4095	13	L	Bit	L0 to L1023
2	C	Bit	C0 to C511	18	V	Bit	V0 to V511
4	X (*1)	Bit	X0 to X1FFF	19	ST	Bit	ST0 to ST127
5	Y (*1)	Bit	Y0 to Y1FFF	20	SD	Word	SD0 to SD2047
6	R	Word	R0 to R32767	21	SB (*1)	Bit	SB0 to SB3FF
7	T	Bit	T0 to T2047	22	SW (*1)	Word	SW0 to SW3FF
9	SM	Bit	SM0 to SM2047	23	B (*1)	Bit	B0 to BDFFF
				24	W (*1)	Word	W0 to W2FFF

[C80 series]

Device designation value	Device	Unit	Device No.	Device designation value	Device	Unit	Device No.
0	M	Bit	M0 to M61439	10	F	Bit	F0 to F2047
1	D	Word	D0 to D8191	13	L	Bit	L0 to L1023
2	C	Bit	C0 to C511	18	V	Bit	V0 to V511
4	X (*1) (*2)	Bit	X0 to X1FFF	19	ST	Bit	ST0 to STI27
5	Y (*1) (*2)	Bit	Y0 to Y1FFF	20	SD	Word	SD0 to SD4095
6	R (*2)	Word	R0 to R32767	21	SB (*1)	Bit	SB0 to SB3FF
7	T	Bit	T0 to T2047	22	SW (*1)	Word	SW0 to SW1023
9	SM (*2)	Bit	SM0 to SM4095	23	B (*1)	Bit	B0 to BDFFF
				24	W (*1)	Word	W0 to W2FFF

The unit indicates the amount of data per device No. "Word" is 16 bits, and "Bit" is one bit.

(*1) Device of which the device number is indicated in hexadecimal notation.

(*2) The device marked by an asterisk (*) in the Device column has the determined use; therefore, do not use the undefined device number even for a vacant device.

Device No. designation (#100101)

The device to be read in is designated by substituting the device No. in this system variable.

Convert a device expressed as a hexadecimal into a decimal when designating.

If the data is read without designating this number, the data will be read in the same manner as when the minimum device No. (0) is designated. Once designated, the setting is held until the device No. is designated again or until it is reset.

A program error (P39) will occur if a nonexistent device No. is set.

Number of bytes designation (#100102)

(1) System variable for number of bytes designation

The reading size is designated by substituting the number of bytes designation value in this system variable.

If the data is read without designating this number, the data will be read in the same manner as when the minimum device designation value (0: M device) is designated. Once designated, the setting is held until the number of bytes is designated again or until it is reset.

A program error (P39) will occur if a number of bytes that does not exist in the specifications is set.

(2) Number of bytes designation value

Number of bytes designation value	Read in data			Operation	
	Size	Sign	Range	Word device	Bit device
0	1 bit	-	0 to 1	The number of bits designated is read in.	The bits for the designated device No. are read in.
1	1 bytes	No	0 to 255	The low-order byte is read in.	8 bits are read in from the designated device No.
101		Yes	-128 to 127		
2	2 bytes	No	0 to 65535	Two bytes are read in.	16 bits are read in from the designated device No.
102		Yes	-32768 to 32767		
4	4 bytes	No	0 to 4294967295	The designated device (L) and next device (H) are read in.	32 bits are read in from the designated device No.
104		Yes	-2147483648 to 2147483647		

0 to 4 are designated without a sign, and 101 to 104 are designated with a sign.

Bit designation (#100103)

(1) System variable for bit designation

The bit to be read in is designated by substituting the bit designation value in this system variable.

This designation is valid only when reading the bits for a 16-bit device, and is invalid for the others.

If the data is read without designating this number, the data will be read in the same manner as if the minimum bit designation value (0: bit 0) is designated. Once designated, the setting is held until the bit is designated again or until it is reset.

A program error (P39) will occur if a nonexistent bit is set.

(2) Bit designation value

Bit designation value	Read in bit
0	Bit 0
1	Bit 1
:	:
15	Bit 15

Reading PLC data (#100110)

The data for the designated device is read in with this system variable.

Refer to the table for number of bytes designation for details on the range of data read in.



Program example

(1) To read a bit device

```
#100100 = 0;      Designates [M device].
#100101 = 0;      Designates [Device No. 0].
#100102 = 0;      Designates [Bit].
#100 = #100110;   Reads M0 (one bit).
#100102 = 1;      Designates [1 byte].
#101 = #100110;   Reads M0 to M7 (8 bits).
                  (If M7 to M0 is 0001 0010, this will be #102 = 18 (0x12).)
#100102 = 102;    Designates [Signed two bytes].
#102 = #100110;   Reads M0 to M15 (16 bits).
                  (If M15 to M0 is 1111 1110 1101 1100, this will be #102 = -292 (0xFEDC).)
#100102 = 4;      Designates [4 byte].
#104 = #100110;   Reads M0 to M31 (32 bits).
                  (If M31 to M0 is 0001 0010 0011 0100 0101 0110 0111 1000, #104 = 305419896
                  (0x12345678).)
```

(2) To read a word device

```
#100100 = 1;      Designates [D device].
#100101 = 0;      Designates [Device No. 0].
#100102 = 0;      Designates [Bit].
#100103 = 1;      Designates [Bit 1].
#100 = #100110;   Reads the D0 bit 1.
                  (If D0 = 0x0102, #101 =1.)
#100102 = 1;      Designates [1 byte].
#101 = #100110;   Reads the low-order byte of D0.
                  (If D0 = 0x0102, #101 =2.)
#100102 = 2;      Designates [2 byte].
#102 = #100110;   Reads D0. (If D0 = 0x0102, #102 =258.)
#100102 = 104;    Designates [Signed four bytes].
#104 = #100110;   Reads D0 and D1.
                  (If D0 = 0xFFFFE and D1 = 0xFFFF, #104 =-2.)
```



Precautions

- (1) As the PLC data is read asynchronously from the ladder execution, the data is not necessarily the one which was gained when the program was executed. Be careful when reading devices which are changing.
- (2) If reading of a nonexistent device is attempted by designating the device No. and number of bytes, the 0 value will be read in only for the nonexistent section.
- (3) When "1" is set to the parameter "#1316 CrossCom", #100100 to #100110 cannot be used as system variables to read PLC data.

23.29 System Variables (Interfering Object Selection)



Detailed description

Select 16 interfering objects to use in the interference check III with system variables or R register.

Refer to the "PLC Interface Manual" for the R register.

When selecting an interfering object, specify the specification of the selected interfering object and interfering model coordinate system offset 1.

The write command to the system variables (#40000 to #40097) is possible only in the machine tool builder macro programs (O100010000 to O199999998).

System variable	R register	Item	Details	Setting range (unit)	
				Upper: System variable	
				Lower: R register	
#40000	R20304	Interfering object enable/disable designation	Set enable/disable for each interfering object. Bit designation (0: enable 1: disable) bit0: Disable 1st interfering object : bitF: Disable 16th interfering object	0 to 65535 (decimal)	
				0x0000 to 0xFFFF (hexadecimal)	
#40001	R20305	preliminary		0	0
#40002	R20306	1st interfering object selection	Select interfering object definition No. to use.	0 to 128 (0: not selected)	
				0 to 128 (0: not selected)	
#40003	R20307	1st interfering object specification	In the configured solid specification of the interfering object definition, specify alarm area/warning area/solid setting invalid of the solid in which switching method is selected. 0, 1: Alarm area 2: Warning area 3: Solid setting invalid	0 to 3	
				0 to 3	
#40004	R20308 (L)	1st interfering model coordinate system I axis offset 1	Set the interfering model coordinate system offset with a radius value. (I axis direction) (*1)	-99999.999 to 99999.999 (mm) (radius value)	
	R20309 (H)				
#40005	R20310 (L)	1st interfering model coordinate system J axis offset 1	Set the interfering model coordinate system offset with a radius value. (J axis direction) (*1)	-99999999 to 99999999 (μm) (radius value)	
	R20311 (H)				
#40006	R20312 (L)	1st interfering model coordinate system K axis offset 1	Set the interfering model coordinate system offset with a radius value. (K axis direction) (*1)		
	R20313 (H)				
:	:				
#40077	R20426	16th interfering object selection	Same as above	Same as above	
#40078	R20427	16th interfering object specification selection	Same as above	Same as above	
#40079	R20428 (L)	16th interfering model coordinate system I axis offset 1	Same as above	Same as above	
	R20429 (H)				
#40080	R20430 (L)	16th interfering model coordinate system J axis offset 1	Same as above	Same as above	
	R20431 (H)				

System variable	R register	Item	Details	Setting range (unit)
				Upper: System variable
				Lower: R register
#40081	R20432 (L) R20433 (H)	16th interfering model coordinate system K axis offset 1	Same as above	Same as above
#40082	R20434	1st interfering object Interference check III: Specifying disabled interfering object	Select an interfering object that you do not check the interference with the 1st interfering object. bit0: Disable 1st interfering object (inaction data) bit1: Disable 2nd interfering object : bitF: Disable 16th interfering object	0 to 65535 (decimal) 0x0000 to 0xFFFF (hexadecimal)
#40083	R204325	2nd interfering object Interference check III: Specifying disabled interfering object	Select an interfering object that you do not check the interference with the 2nd interfering object. bit0: Disable 1st interfering object bit1: Disable 2nd interfering object (inaction data) : bitF: Disable 16th interfering object	0 to 65535 (decimal) 0x0000 to 0xFFFF (hexadecimal)
:	:			
#40097	R20449	16th interfering object Interference check III: Specifying disabled interfering object	Select an interfering object that you do not check the interference with the 16th interfering object. bit0: Disable 1st interfering object bit1: Disable 2nd interfering object : bitF: Disable 16th interfering object (inaction data)	0 to 65535 (decimal) 0x0000 to 0xFFFF (hexadecimal)

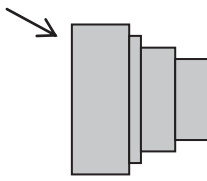
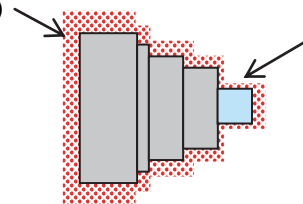
(*1) The interfering model coordinate system offset is the sum of the interfering model coordinate system offsets 1 and 2.

Interference check III: Specifying disabled interfering object

(Example) In the case that you do not check the interference between the 1st interfering object and the 2nd interfering object

"R20434 (#40082): 0x0002 (disable 2nd interfering object)" or "R20435 (#40083): 0x0001 (disable 1st interfering object)"

Since each interfering object is designated to perform the interference check, the setting of the interference check III specifying disabled interference object is repeated, but if either one is on disabled setting, the interference check is not performed.

Back side of spindle part (without a workpiece)	Back side of spindle part (with a workpiece)
1st interfering object (back side of spindle part) 	1st interfering object (back side of spindle part)  2nd interfering object (workpiece part)
	<p>By specifying the interference III disabled between the back side of spindle part (1st interfering object) and workpiece part (2nd interfering object), these 2 parts are treated as one interfering object.</p>



Precautions

- (1) When the interfering object selection is input with the system variables, the system variable in which the command range integer is set in R register with the value after the decimal point being ignored.
 - (a) When any value out of the setting range is input in #40000 to #40097, the low-order 16 bits of the input value are set in R register.
 - (b) When "#0" <empty> is input in #40000 to #40097, "0" is set in R register.
- (2) If you have made a write command to the system variable (#40000 to #40097) in a program except for the machine tool builder macro program, the program error (P241) occurs.

23.30 System Variables (ZR Device Access Variables) [C80]

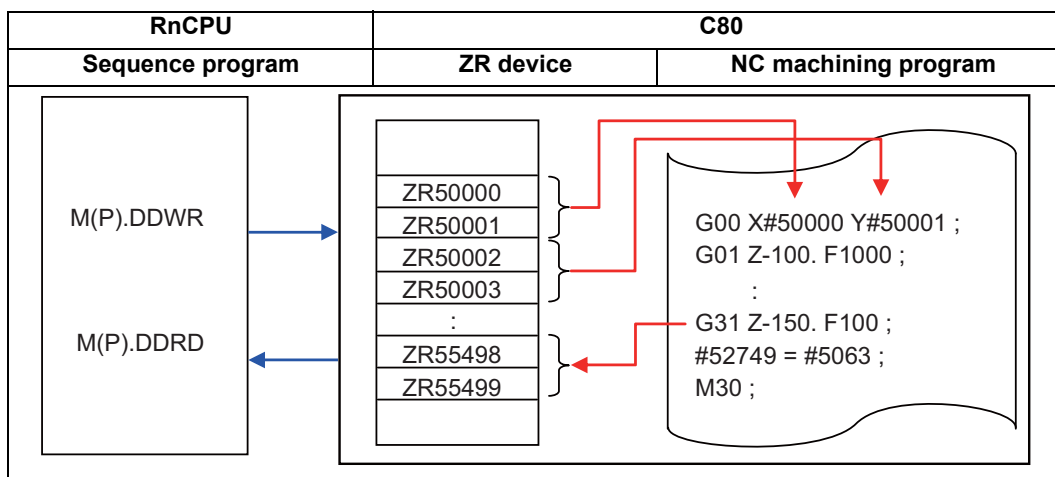


Detailed description

System variables that can read and write data from and to the ZR device are provided by 2,250 sets (#50000 to #52749).

Data can be read and written between the NC machining program and RnCPU sequence program by using the ZR device as shown below.

How to handle the ZR device in the RnCPU sequence program depends on the MTB specifications. Refer to the "PLC Interface Manual" for the DDWR/DDRD command.



Number of variable sets

The table below shows a list of variables specific to C80.

A ZR device access variable is based on long-type data, and a ZR device on word-type data.

Therefore, when this variable is read or a value is substituted to this variable, it reads and writes two words of the ZR device. The correspondence between the ZR device access variable numbers and ZR device numbers is shown below.

Variable No. (2,250 sets)		Corresponding ZR device (4,500 units)
#50000 - #50749	#50000	ZR50000, ZR50001
	#50001	ZR50002, ZR50003
	#50002	ZR50004, ZR50005
	:	:
	#50000+n	ZR50000+2n, ZR50000+2n+1
	#50749	ZR51498, ZR51499
#51000 - #51749	#51000	ZR52000, ZR52001
	:	:
	#51749	ZR53498, ZR53499
#52000 - #52749	#52000	ZR54000, ZR54001
	:	:
	#52749	ZR55498, ZR55499

- (1) The data range of these variables is -2147483648 to 2147483647.
- (2) The ZR device is backed up even when the power is turned OFF; therefore, the value is maintained after the power has been turned ON again.
- (3) Whether this variable is used with the decimal point invalid or valid can be selected for each user backup area according to the MTB specifications (parameter "#6455 bit0 -bit2").

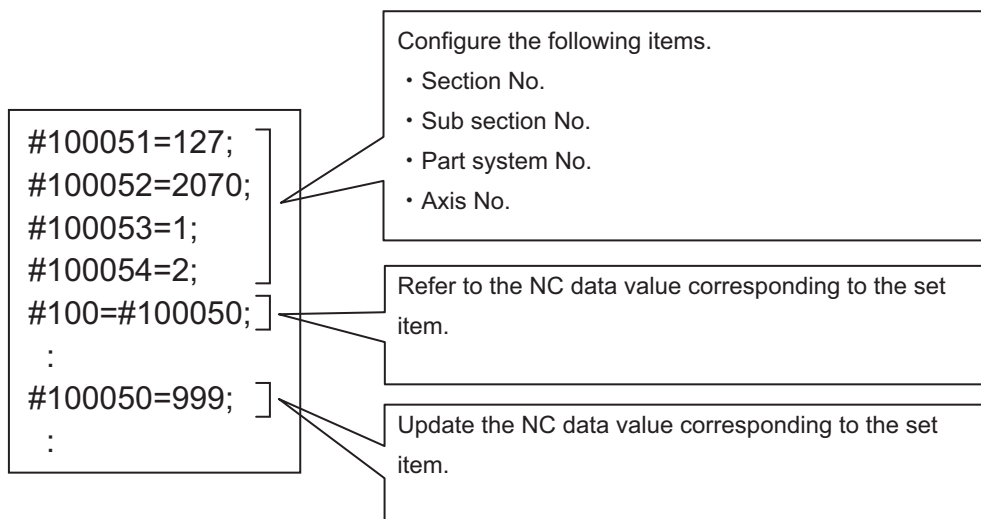
- (4) When "decimal point valid" is selected, the position of the decimal point depends on the MTB specifications (parameters "#1003 iunit" (input setting unit) and "#1041 I_inch" (initial inch)). Therefore, to set a numeric value to a ZR device, consider the position of the decimal point according to these parameters.

The table below shows the number of digits that is valid after the decimal point.

#1041 I_inch	#1003 iunit			
	B	C	D	E
Metric	3 digits	4 digits	5 digits	6 digits
Inch	4 digits	5 digits	6 digits	7 digits

23.31 System Variables (NC Data Reading/Writing with API Section and Sub-section Nos. Input/Output by Program)

Use system variables to designate the section No., sub-section No., part system No., and axis No.; you can read and write data in the NC.



Variable No.	Item	Value	Description
#100050	NC data	Signed 4-byte data value	At reading: Reads the data values corresponding to various numbers shown below. At writing: Sets the designated value as the NC data value.
#100051	Section No.	Integer value	Designate the data type.
#100052	Sub-section No.	Integer value	Designate data.
#100053	Part system No.	0 to 8	Designate the part system No. If the part system No. is omitted, the system runs as the self part system. No part system needs to be designated for data if one is not required.
#100054	Axis No.	0 to 32	Designate the axis No. If the axis No. is omitted, the system runs using a single axis. No axis needs to be designated for data if one is not required.

Note

(1) A program error (P35) occurs in the following cases:

- The section No. is invalid.
- The sub-section No. is invalid.
- The part system No. is invalid.
- The axis No. is invalid.

(2) A program error (P243) occurs in the following cases:

- The read command is issued to the write-only variable.
- The write command is issued to the read-only variable.
- The write command is issued when data writing is not possible.

Use examples

(1) When referencing NC data

Setting	Left-side value	Right-side value
#100051=127 ;	Section No. designation	Axis parameter
#100052=2070 ;	Sub-section No. designation	Number of rotary axis division
#100053=1 ;	Part system No. designation	1st part system
#100054=2 ;	Axis No. designation	2nd axis
#100=#100050 ;	NC data reference	
#100053=2 ;	Part system No. designation	2nd part system
#101=#100050 ;	NC data reference	
#100053=3 ;	Part system No. designation	3rd part system
#102=#100050 ;	NC data reference	
#100053=4 ;	Part system No. designation	4th part system
#103=#100050 ;	NC data reference	

Note

- ♦The section No., sub-section No., part system No., and axis No. can be designated in random order. When these values have already been designated in the previous operation, they can be omitted. However, designate them before reading NC data.

(2) When updating NC data

Setting	Left-side value	Right-side value
#100051=127 ;	Section No. designation	Axis parameter
#100052=2070 ;	Sub-section No. designation	Number of rotary axis division
#100053=1 ;	Part system No. designation	1st part system
#100054=1 ;	Axis No. designation	1st axis
#100050=999 ;	NC data update	Designate the maximum value in the range of 0 to 999.
#100054=2 ;	Axis No. designation	2nd axis
#100050=500 ;	NC data update	Designate the maximum value in the range of 0 to 999.
#100054=3 ;	Axis No. designation	3rd axis
#100050=10 ;	NC data update	Designate the maximum value in the range of 0 to 999.

Note

- ♦The section No., sub-section No., part system No., and axis No. can be designated in random order. When these values have already been designated in the previous operation, they can be omitted. However, designate them before writing NC data.

(3) When assigning the part system No. and axis No. to variables

Setting	Left-side value	Right-side value
#100051=127 ;	Section No. designation	Axis parameter
#100052=2070 ;	Sub-section No. designation	Number of rotary axis division
#100=8 ;	Maximum value of part system No.	
#101=5 ;	Maximum value of axis No.	
#102=#101 ;	Retains the maximum value of the axis No.	
WHILE[#100GT0]DO1 ;	Repeats processing by a number of times equivalent to the part system No. (repeats processing of 1 to 8 part systems).	
WHILE[#101GT0]DO2 ;	Repeats processing by a number of times equivalent to the axis No. (repeats processing of 1 to 5 axes).	
#100053=#100 ;	Part system No. designation	
#100054=#101 ;	Axis No. designation	
#103=#100050 ;	NC data reference	
#103=#103-2 ;	Changes the NC data value to a value that is 2 lower than the current value.	
#100050=#103 ;	NC data update	
#101=#101-1 ;	Subtracts the axis No.	
END2	Ends repeat processing by a number of times equivalent to the axis No.	
#101=#102 ;	Returns the axis No. to the initial setting value.	
#100=#100-1 ;	Subtracts the axis No.	
END1	Ends repeat processing by a number of times equivalent to the part system No.	



Precautions

- (1) NC data is read and written by accessing "#100050" for reading/writing. Therefore, designate the section No., sub-section No., part system No., and axis No. before accessing "#100050".
- (2) "#100051" to "#100054" (section No., sub-section No., part system No., and axis No.) are cleared by NC reset 1 or Reset & rewind. After NC reset 1 or Reset & rewind was commanded, designate those numbers again.
- (3) It is prohibited to pre-read "#100050" to "#100054".
- (4) When the system runs with a graphic check (background or foreground), writing is not performed.
- (5) If NC data reading or writing fails, buffer correction cannot be performed.

Appx.1: Fixed Cycles

[G81(O10000810) Drill, spot drill]

```
G.1 ;
IF[#30] GOTO1 ;
Z#2 G#6 H#7 ;
#2=##5#3003=#8 OR1 ;
G1 Z#3 ;
IF[#4 EQ#0] GOTO2 ;
G4 P#4 ;
N2 ;
#3003=#8 ;
G0 Z-#3-#2, I#23 ;
N1 M99 ;
```

[G82(O10000820) Drill, counter boring]

```
G.1 ;
IF[#30] GOTO1 ;
Z#2 G#6 H#7 ;
#2=##5#3003=#8 OR1 ;
G1 Z#3 ;
G4 P#4 ;
#3003=#8 ;
G0 Z-#3-#2, I#23 ;
N1 M99 ;
```

[G83(O10000830) Deep hole drill cycle]

```
G.1 ;
IF[#30] GOTO2 ;
#29=#11#28=0 ;
Z#2 G#6 H#7 ;
#2=##5#3003=#8 OR1 ;
DO1 ;
#28=#28-#11#26=-#28-#29 ;
Z#26 ;
IF[ ABS[#28] GE[ ABS[#3]]] GOTO1 ;
G1 Z#29 ;
G0 Z#28 ;
#29=#11+#14 ;
END1 ;
N1 G1 Z#3-#26 ;
IF[#4 EQ#0] GOTO3 ;
G4 P#4 ;
N3 ;
#3003=#8 ;
G0 Z-#3-#2, I#23 ;
N2 M99 ;
```

[G84(O10000840) Tap cycle]

```
G.1 ;
IF[#30] GOTO9 ;
Z#2 G#6 H#7 ;
#2=##5#3003=#8 OR1#3004=#9 OR3 ;
IF[#11] GOTO1 ;
GOTO2 ;
N1 ;
IF[#14] GOTO5 ;
N2 G1 Z#3 ;
GOTO7 ;
N5 ;
#29=0#28=#11 ;
DO1 ;
#29=#29+#11 ;
IF[ ABS[#29] GE[ ABS[#3]]] GOTO6 ;
G1 Z#28 ;
M#53 ;
G1 Z-#14 ;
M#54 ;
#28=#11+#14 ;
END1 ;
N6 G1 Z#3-#29+#28 ;
N7 G4 P#4 ;
M#53 ;
#3900=1 ;
G1 Z-#3 ;
#3004=#9 ;
G4 P#56 ;
M#54 ;
#3003=#8 ;
G0 Z-#2, I#23 ;
N9 M99 ;
```

[G85(O10000850) Boring 1]

```

G.1 ;
IF[#30] GOTO1 ;
Z#2 G#6 H#7 ;
#2=##5#3003=#8 OR1 ;
G1 Z#3 ;
IF[#4 EQ#0] GOTO2 ;
G4 P#4 ;
N2 ;
#3003=#8 ;
Z-#3 ;
G0 Z-#2, I#23 ;
N1 M99 ;

```

[G88(O10000880) Boring 3]

```

G.1 ;
IF[#30] GOTO1 ;
Z#2 G#6 H#7 ;
#2=##5#3003=#8 OR1 ;
G1 Z#3 ;
G4 P#4 ;
#3003=#8 ;
M5 ;
#3003=#8 OR1 ;
G0 Z-#3-#2 ;
#3003=#8 ;
M3 ;
N1 M99 ;

```

[G86(O10000860) Boring 2]

```

G.1 ;
IF[#30] GOTO1 ;
Z#2 G#6 H#7 ;
#2=##5#3003=#8 OR1 ;
G1 Z#3 ;
G4 P#4 ;
M5 ;
G0 Z-#3-#2 ;
#3003=#8 ;
M3 ;
N1 M99 ;

```

[G89(O10000890) Boring 4]

```

G.1 ;
IF[#30] GOTO1 ;
Z#2 G#6 H#7 ;
#2=##5#3003=#8 OR1 ;
G1 Z#3 ;
G4 P#4 ;
#3003=#8 ;
Z-#3 ;
G0 Z-#2, I#23 ;
N1 M99 ;

```

[G87(O10000870) Back boring] (*1)

```

G.1 ;
IF[#30] GOTO1 ;
#3003=#8 OR1 ;
M19 ;
X#12 Y#13 ;
#3003=#8 ;
Z#2 G#6 H#7 ;
#3003=#8 OR1 ;
#3906=1 ;
G#26 X-#12 Y-#13 ;
#3003=#8 ;
M3 ;
#3003=#8 OR1 ;
G1 Z#3 ;
M19 ;
G0 X#12 Y#13 ;
G1 Z#3 ;
#3003=#8 ;
X-#12 Y-#13 ;
M3 ;
N1 M99 ;

```

[G73(O10000831) Step cycle]

```

G.1 ;
IF[#30] GOTO2 ;
#29=0#28=#11 ;
Z#2 G#6 H#7 ;
#2=##5#3003=#8 OR1 ;
DO1 ;
#29=#29+#11 ;
IF[ ABS[#29] GE[ ABS[#3]]] GOTO1 ;
G1 Z#28 ;
G4 P#4 ;
G0 Z-#14 ;
#28=#11+#14 ;
END1 ;
N1 G1 Z#3-#29+#28 ;
G4 P#4 ;
#3003=#8 ;
G0 Z-#3-#2, I#23 ;
N2 M99 ;

```


[G74(O100000841) Reverse tap cycle]

```

G.1 ;
IF[#30] GOTO9 ;
Z#2 G#6 H#7 ;
#2=##5#3003=#8 OR1#3004=#9 OR3 ;
IF[#11] GOTO1 ;
GOTO2 ;
N1 ;
IF[#14] GOTO5 ;
N2 G1 Z#3 ;
GOTO7 ;
N5 ;
#29=0#28=#11 ;
DO1 ;
#29=#29+#11 ;
IF[ ABS[#29] GE[ ABS[#3]]] GOTO6 ;
G1 Z#28 ;
M#53 ;
G1 Z-#14 ;
M#54 ;
#28=#11+#14 ;
END1 ;
N6 G1 Z#3-#29+#28 ;
N7 G4 P#4 ;
M#53 ;
#3900=1 ;
G1 Z-#3 ;
#3004=#9 ;
G4 P#56 ;
M#54 ;
#3003=#8 ;
G0 Z-#2, I#23 ;
N9 M99 ;

```

[G75(O100000851) Circle cutting cycle]

```

G.1 ;
IF[#30] GOTO1 ;
#28=#18 ;
IF[#28 GE0] GOTO2 ;
#27=3#28=-#28 ;
GOTO3 ;
N2#27=2 ;
N3#26=#4 ;
IF[#26 GE#28] GOTO1 ;
Z#2 G#6 H#7 ;
#2=##5#3003=#8 OR1 ;
G1 Z#3 ;
#28=#28-#26#29=#28/2 ;
G#27 X-#28 I-#29 ;
I#28 P1 ;
X#28 I#29 ;
#3003=#8 ;
G0 Z-#3-#2, I#23 ;
N1 M99 ;

```

[G76(O100000861) Fine boring] (*1)

```

G.1 ;
IF[#30] GOTO1 ;
Z#2 G#6 H#7 ;
#2=##5#3003=#8 OR1 ;
G1 Z#3 ;
M19 ;
#3906=1 ;
G#26 X#12 Y#13 ;
G0 Z-#3-#2 ;
#3003=#8 ;
X-#12 Y-#13 ;
M3 ;
N1 M99 ;

```

(*1)The machining programs described in this manual can be used when the specifications are enabled, which the tool moves with rapid traverse by shift amount.

Since the system must be upgraded to the S/W version C4 of M800 Series/M80 Series or later and then the fixed cycles must be replaced, contact the MTB to use this function.

Appx.2: Command Value Range Lists

<Lists of command values and setting value ranges>

(1) Linear axis: Input unit [mm]

(M system)

Input setting unit	0.001	0.0001	0.00001	0.000001
Max. stroke (Value in machine coordinate system)	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Max. command value	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Rapid traverse rate (including the rate at dry running)	1 to 1000000 mm/min	1 to 1000000 mm/min	1 to 1000000 mm/min	1 to 1000000 mm/min
Cutting feedrate (including the rate at dry running) Asynchronous feed (feed per minute)	0.001 to 1000000.000 mm/min	0.0001 to 1000000.0000 mm/min	0.00001 to 1000000.00000 mm/min	0.000001 to 1000000.000000 mm/min
Synchronous feed (feed per revolution)	0.001 to 999.999 mm/rev	0.0001 to 999.9999 mm/rev	0.00001 to 999.99999 mm/rev	0.000001 to 999.999999 mm/rev
2nd to 4th reference position offset (Value in machine coordinate system)	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Tool compensation amount (Shape)	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Tool compensation amount (Wear)	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Incremental feed amount	0.001 mm/pulse	0.0001 mm/pulse	0.00001 mm/pulse	0.000001 mm/pulse
Handle feed amount	0.001 mm/pulse	0.0001 mm/pulse	0.00001 mm/pulse	0.000001 mm/pulse
Soft limit range (Value in machine coordinate system)	±99999.999 mm	±99999.9999 mm	±99999.99999 mm	±99999.999999 mm
Dwell time	0 to 99999.999 s	0 to 99999.9999 s	0 to 99999.99999 s	0 to 99999.999999 s
Backlash compensation amount	±9999999 pulse	±9999999 pulse	±9999999 pulse	±9999999 pulse
Pitch error compensation amount	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse
Thread lead (F)	0.001 to 999.999 mm/rev	0.0001 to 999.9999 mm/rev	0.00001 to 999.99999 mm/rev	0.000001 to 999.999999 mm/rev
Thread lead (Precision E)	0.0001 to 999.9999 mm/rev	0.00001 to 999.99999 mm/rev	0.000001 to 999.999999 mm/rev	0.0000001 to 999.9999999 mm/rev
Thread lead (Threads/inch)	0.03 to 999.99	0.026 to 222807.017	0.0255 to 224580.0000	0.02541 to 224719.00000

(2) Linear axis: Input unit [inch]

(M system)

Input setting unit	0.0001	0.00001	0.000001	0.0000001
Max. stroke (Value in machine coordinate system)	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Max. command value	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Rapid traverse rate (including the rate at dry running)	1 to 100000 inch/min	1 to 100000 inch/min	1 to 100000 inch/min	1 to 100000 inch/min
Cutting feedrate (including the rate at dry running) Asynchronous feed (feed per minute)	0.0001 to 100000.0000 inch/min	0.00001 to 100000.00000 inch/min	0.000001 to 100000.000000 inch/min	0.0000001 to 100000.0000000 inch/min
Synchronous feed (feed per revolution)	0.0001 to 999.9999 inch/rev	0.00001 to 999.99999 inch/rev	0.000001 to 999.999999 inch/rev	0.0000001 to 999.9999999 inch/rev
2nd to 4th reference position offset (Value in machine coordinate system)	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Tool compensation amount (Shape)	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Tool compensation amount (Wear)	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Incremental feed amount	0.0001 inch/pulse	0.00001 inch/pulse	0.000001 inch/pulse	0.0000001 inch/pulse
Handle feed amount	0.0001 inch/pulse	0.00001 inch/pulse	0.000001 inch/pulse	0.0000001 inch/pulse
Soft limit range (Value in machine coordinate system)	±9999.9999 inch	±9999.99999 inch	±9999.999999 inch	±9999.9999999 inch
Dwell time	0 to 99999.999 s	0 to 99999.9999 s	0 to 99999.99999 s	0 to 99999.999999 s
Backlash compensation amount	±9999999 pulse	±9999999 pulse	±9999999 pulse	±9999999 pulse
Pitch error compensation amount	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse
Thread lead (F)	0.0001 to 39.3700 inch/rev	0.00001 to 39.37007 inch/rev	0.000001 to 39.370078 inch/rev	0.0000001 to 39.3700787 inch/rev
Thread lead (Precision E)	0.00001 to 39.37007 inch/rev	0.000001 to 39.370078 inch/rev	0.0000001 to 39.3700787 inch/rev	0.00000001 to 39.37007873 inch/rev
Thread lead (Threads/inch)	0.025 to 9999.999	0.0255 to 9999.9999	0.02541 to 9999.99999	0.025401 to 9999.999999

(3) Rotary axis: Degree [°]

(M system)

Input setting unit	0.001	0.0001	0.00001	0.000001
Max. stroke (Value in machine coordinate system)	±99999.999 °	±99999.9999 °	±99999.99999 °	±99999.999999 °
Max. command value	±99999.999 °	±99999.9999 °	±99999.99999 °	±99999.999999 °
Rapid traverse rate (including the rate at dry running)	1 to 1000000 °/min	1 to 1000000 °/min	1 to 1000000 °/min	1 to 1000000 °/min
Cutting feedrate (including the rate at dry running) Asynchronous feed (feed per minute)	0.001 to 1000000.000 °/min	0.0001 to 1000000.0000 °/min	0.00001 to 1000000.00000 °/min	0.000001 to 1000000.000000 °/min
Synchronous feed (feed per revolution)	0.001 to 999.999 °/rev	0.0001 to 999.9999 °/rev	0.00001 to 999.99999 °/rev	0.000001 to 999.999999 °/rev
2nd to 4th reference position offset (Value in machine coordinate system)	±99999.999 °	±99999.9999 °	±99999.99999 °	±99999.999999 °
Incremental feed amount	0.001 °/pulse	0.0001 °/pulse	0.00001 °/pulse	0.000001 °/pulse
Handle feed amount	0.001 °/pulse	0.0001 °/pulse	0.00001 °/pulse	0.000001 °/pulse
Soft limit range (Value in machine coordinate system)	±99999.999 °	±99999.9999 °	±99999.99999 °	±99999.999999 °
Backlash compensation amount	±9999999 pulse	±9999999 pulse	±9999999 pulse	±9999999 pulse
Pitch error compensation amount	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse	-32768 to 32767 pulse

Index

Refer to Programming Manual (Machining Center System) (1/2) for Chapter 14 and previous chapters (page 480 and before).

Refer to Programming Manual (Machining Center System) (2/2) for Chapter 15 and succeeding chapters (page 481 and later).

Symbols

In (Im ...) L	588
/	16
/n	18

Numerics

2nd, 3rd, and 4th Reference Position (Zero Point) Return ...	937
3-dimensional Circular Interpolation	109
3-dimensional Coordinate Conversion	887
3-dimensional Tool Radius Compensation	317
3-dimensional Tool Radius Compensation (Tool's Vertical-direction Compensation)	830

A

Acceleration Clamp Speed	693
Acceleration/Deceleration Mode Change in Hole Drilling Cycle ...	389
Actual Examples of Using User Macros	467
Allocation of the Number of Tool Life Management Sets to Part Systems ...	559
Arbitrary Axis Exchange	604
Arc	394
ASCII Code Macro	439
Automatic Coordinate System Setting	859
Automatic Corner Override	198, 205
Automatic Tool Length Measurement	954
Axis Name Extension	565
Axis-based Unidirectional Positioning	74

B

Back Boring	365
Basic Machine Coordinate System Selection	863
Basic Machine, Workpiece and Local Coordinate Systems ...	857
Bolt Hole Cycle	392
Boring	363, 364, 368, 369

C

Changing of Compensation No. during Compensation Mode ...	295
Circular Cutting	82, 374
Circular Interpolation	49
Command Value Range Lists	1054
Common Variables	445
Compensation Data Input by Program	537
Compensation Data Input by Program (Turning Tool) ...	543
Constant Lead Thread Cutting	60
Constant Surface Speed Control	225
Control Commands	455
Coordinate Rotation by Program	905
Coordinate Rotation Input by Parameter	912
Coordinate System for Rotary Axis	860
Coordinate System Setting	866
Coordinate Systems and Coordinate Zero Point Symbols ...	3
Coordinate Words and Control Axes	2, 856
Corner Chamfering Expansion/Corner Rounding Expansion ...	486, 492
Corner Chamfering I	482
Corner Chamfering I / Corner Rounding I	482
Corner Chamfering II	489
Corner Chamfering II / Corner Rounding II	489
Corner Rounding I	484
Corner Rounding II	491
Cutting Feed Constant Inclination Acceleration/Deceleration ...	165
Cutting Feedrate	137
Cutting Mode	208

Cylindrical Interpolation	75
---------------------------------	----

D

Deceleration Check	176
Deceleration Check when Movement in The Opposite Direction Is Reversed	184
Decimal Point Input	37
Deep Hole Drilling Cycle	344
Define by Selecting the Registered Machining Surface ...	800
Detailed Description for Macro Call Instruction	437
Details of Inclined Surface Machining Operation	811
Diameter Designation of Compensation Amount	310
Diameter/Radius Designation Switch	32
Drilling Cycle High-Speed Retract	385
Drilling, Counter Boring	343
Drilling, Spot Drilling	342
Dwell (Time-based designation)	210

E

Enable Interfering Object Selection Data	950
Exact Stop Check	171
Exact Stop Check Mode	175
Exponential Interpolation	91
External Output Commands	461

F

F1-Digit Feed	138
Face Cutting Cycle	405
Fairing	683
Feed Per Minute/Feed Per Revolution (Asynchronous Feed/Synchronous Feed)	140
Feedrate Designation and Effects on Control Axes	148
Figure Rotation	417
File Format	14
Fine Boring	376
Fixed Cycle for Turning Machining	397
Fixed Cycles	338, 1050

G

G Code Lists	20
G Code Macro Call	434
G Command Mirror Image	504
G0.5	188
G0.5 P1	186
G00	44
G00 Feedrate Command (,F command)	133
G01	47
G01 A_	494
G01 A_ , G02/G03 P_Q_H_	499
G01 A_ , G02/G03 R_H_	502
G01 X_Y_ ,C	482
G01 X_Y_ ,R_	484
G01 X_/Y_ A_/A_	493
G01/G02/G03 X_Y_ ,C_	489
G01/G02/G03 X_Y_ ,R_	491
G02, G03	49, 55
G02.1/G03.1 (Type 1), G02/G03 (Type 2)	105
G02.2/G03.2	122
G02.3,G03.3	91
G02.4, G03.4	109
G02/G03 P_Q_ /R_	497
G04	210

G05 P1, G05 P2	624	G53	863
G05.1 Q1/Q0, G05 P10000/P0, G05 P20000/P0	667	G53.1/G53.6	803
G05.1 Q2/Q0	698	G54 to G59 (G54.1)	872
G06.2	114	G54.4	841
G07	120	G60	72
G07.1	75	G61	175
G09	171	G61.1, G08	633
G10 I_ J_ /K_	912	G61.2	715
G10 L100, G11	545	G61.4	706
G10 L110/L111, G11, G68.2, G69	548	G62	205
G10 L12/L13, G11	543	G63	207
G10 L14	976	G64	208
G10 L2/L10/L11/L12/L13/L20, G11	537	G65	426
G10 L3, G11	552	G66	430
G10 L30, G11	555	G66.1	432
G10 L70/L100 ,G11	534	G68.2, G68.3/G69	789
G10.9	32	G68/G69	887, 905
G114.1	723	G73	370
G115	591	G74	372
G116	594	G75	374
G12,G13	82	G76	376
G12.1,G13.1/G112,G113	84	G81	342
G120.1, G121	717	G82	343
G122	607	G83	344
G127	528	G84	349
G140, G141, G142	604	G85	363
G16	98	G86	364
G160	972	G87	365
G162/G163	242	G88	368
G17, G18, G19	58	G89	369
G17, G18, G19 and G02, G03	66	G90,G91	30
G174	398	G92	232, 866
G175	401	G92.1	882
G176	405	G93	143
G180	561	G94,G95	140
G186	950	G96, G97	225
G187	377	G98, G99	383
G20, G21	35	General Precautions for Tool Radius Compensation	294
G22/G23	946	Geometric	494
G26	572	Geometric IB	496
G27	943	Geometric IB (Automatic Calculation of Linear - Arc Intersection) ...	499, 502
G28,G29	933	Geometric IB (Automatic Calculation of Two-arc Contact) ...	497
G30	937	Grid	395
G30.1 - G30.6	940		
G31	958	H	
G31 Fn	968	Helical Interpolation	66
G31 P	966	High-accuracy Control	633
G31.n, G04	964	High-accuracy Spline Interpolation	715
G33	60, 64	High-speed High-accuracy Control	667
G34	392	High-speed High-accuracy Control I, II, III	667
G35	393	High-speed Machining Mode I, II	624
G36	394	High-speed Mode Corner Deceleration	694
G37	954	How to Define Feature Coordinate System Using Euler Angles ...	790
G37.1	395	How to Define Feature Coordinate System Using Projection Angles	798
G38, G39/G40/G41, G42	266	How to Define Feature Coordinate System Using Roll-Pitch-Yaw Angles ...	792
G40.1/G41.1/G42.1(G150/G151/G152)	508	How to Define Feature Coordinate System Using Three Points in a Plane ...	794
G40/G41, G42	317	How to Define Feature Coordinate System Using Tool Axis Direction ...	801
G40/G41.2, G42.2	830	How to Define Feature Coordinate System Using Two Vectors ...	796
G41/G42 Commands and I, J, K Designation	286	Hypothetical Axis Interpolation	120
G43, G44 / G49	258		
G43.1/G49	751	I	
G43.4, G43.5/G49	758	Inch Thread Cutting	64
G43.7/G49	744	Inch/Metric Conversion	35
G45 to G48	329	Inclined Surface Machining	789
G50.1, G51.1	504	Index Table Indexing	217
G50/G51	929	Indexing Increment	8
G52	868	Initial High-accuracy Control	664

Initial Point and R Point Level Return	383
Inner Arc Override	206
Input Command Increment Tenfold	7
Input Setting Unit	6
Inputting the Tool Life Management Data by G10 L3 Command ...	552
Inputting the Tool Life Management Data by G10 L30 Command ...	555
Interactive Cycle Insertion	561
Interactive Macro	564
Interference Check	300
Interrupt during Corner Chamfering/Interrupt during Corner Rounding	488, 492
Interrupts during Tool Radius Compensation	292
Inverse Time Feed	143
Involute Interpolation	122
L	
Line at Angle	393
Linear Angle Command	493
Linear Interpolation	47
Local Coordinate System Setting	868
Local Variables (#1 to #33)	446
Longitudinal Cutting Cycle	398
M	
M***	597
M198	416
M96, M97	471
M98 I_J_K_	417
M98, M99	410
Machine Zero Point and 2nd, 3rd, 4th Reference Position (Zero Point) ...	858
Machining Condition Selection I	717
Machining Interruption [C80]	572
Macro Call Instructions	426
Macro Interruption	471
Manual Arbitrary Reverse Run Prohibition	528
Miscellaneous Command Macro Call (for M, S, T, B Code Macro Call) ...	435
Miscellaneous Functions (M8-digits)	214
Modal Call A (Movement Command Call)	430
Modal Call B (for Each Block)	432
Modal, Unmodal	20
Multi-part System Simultaneous High-accuracy	665
Multiple Spindle Synchronization Set Control [C80]	739
Multi-step Skip Function 1	964
Multi-step Skip Function 2	966
N	
Normal Line Control	508
Number of Tool Offset Sets Allocation to Part Systems ...	256
NURBS Interpolation	114
O	
Operation Commands	450
Optional Block Skip	16
Optional Block Skip Addition	18
Other Commands and Operations during Tool Radius Compensation ...	276
P	
Parameter Input by Program	534
Plane Selection	58
Polar Coordinate Command	98
Polar Coordinate Interpolation	84
POPEN, PCLOS, DPRNT	461
Position Command Methods	30
Positioning (Rapid Traverse)	44
Precautions	465
Precautions Before Starting Machining	25
Precautions for Inclined Surface Machining	826
Precautions for Inputting the Tool Life Management Data ...	558
Precautions for Using a Fixed Cycle	381
Precautions on High-speed High-accuracy Control	695
Pre-read Buffer	28
Program Format	10
Programmable Current Limitation	976
R	
R Specification Circular Interpolation	55
Rapid Traverse Block Overlap	186
Rapid Traverse Block Overlap for G00	188
Rapid Traverse Block Overlap for G28	196
Rapid Traverse Constant Inclination Acceleration/Deceleration ...	152
Rapid Traverse Constant Inclination Multi-step Acceleration/Deceleration ...	157
Rapid Traverse Rate	132
Reference Position (Zero Point) Return	933
Reference Position Check	943
Relationship between Inclined Surface Machining and Other Functions ...	822
Reverse Tapping Cycle	372
R-Navi Data Input by Program	548
Rotary Axis Basic Position Selection	816
S	
Scaling	929
Second Miscellaneous Functions (A8-digits, B8-digits or C8-digits) ...	216
Setting of Workpiece Coordinates in Fixed Cycle Mode ...	384
Simple Macro Calls	426
Skip Function	958
Small Diameter Deep Hole Drilling Cycle	346
Smooth Fairing	684
Special Fixed Cycle	391
Speed Change Skip	968
Spindle Clamp Speed Setting	232
Spindle Functions	224
Spindle Position Control (Spindle/C Axis Control)	234
Spindle Position Control (Spindle/C Axis Control) under Spindle Synchronization Control	734
Spindle Speed Fluctuation Detection	242
Spindle Synchronization	722
Spindle Synchronization I	723
Spiral/Conical Interpolation	105
Spline Interpolation	698
Spline Interpolation 2	706
SSS Control	653
Start of Tool Radius Compensation and Z Axis Cut in Operation ...	298
Stepping Cycle	370
Stroke Check before Travel	946
Stroke Check before Travel in Stored Stroke Limit Area ...	948
Sub Part System Control I	607
Subprogram Call	410, 416
System Variables	449
System Variables (Alarm)	1005
System Variables (Coordinate Rotation Parameter) ...	1011
System Variables (Cumulative Time)	1006
System Variables (Extended Workpiece Coordinate Offset) ...	999
System Variables (External Workpiece Coordinate Offset) ...	1000
System Variables (G Command Modal)	980
System Variables (Interfering Object Selection)	1041
System Variables (Machining Information)	1009
System Variables (Macro Interface Input (PLC -> NC)) ...	1019
System Variables (Macro Interface Output (NC -> PLC)) ...	1025
System Variables (Message Display and Stop)	1006
System Variables (Mirror Image)	1010
System Variables (Modal Information at Macro Interruption) ...	982
System Variables (NC Data Reading/Writing with API Section and Sub-section Nos. Input/Output by Program)	1046
System Variables (Non-G Command Modal)	981
System Variables (Normal Line Control Parameter) ...	1013
System Variables (Number of Workpiece Machining Times) ...	1010

System Variables (Parameter Reading)	1014
System Variables (PLC Data Reading)	1037
System Variables (Position Information)	1001
System Variables (R Device Access Variables)	1031
System Variables (Reverse Run Information)	1010
System Variables (Rotary Axis Configuration Parameter) ...	1012
System Variables (Time Read Variables)	1007
System Variables (Tool Compensation)	992
System Variables (Tool Information)	984
System Variables (Tool Life Management)	993
System Variables (Workpiece Coordinate Offset)	998
System Variables (Workpiece Installation Error Compensation Amount) ...	1018
System Variables (ZR Device Access Variables) [C80] ...	1044
System Variables List	978

T

Tapping Cycle	349
Tapping Mode	207
Thread Cutting	60
Thread Cutting Cycle	401
Thread Milling Cycle	377
Timing Synchronization Operation	588
Timing Synchronization Operation (! code)	588
Timing Synchronization Operation Function Using M codes ...	597
Timing Synchronization Operation with Start Point Designated (Type 1) ...	591
Timing Synchronization Operation with Start Point Designated (Type 2) ...	594
Timing Synchronization When Timing Synchronization Ignore Is Set ...	601
Tolerance Control	657
Tool Axis Direction Control	803
Tool Center Point Control	758
Tool Change Position Return	940
Tool Compensation	252
Tool Functions (T8-digit BCD)	250
Tool Length Compensation Along the Tool Axis	751
Tool Length Compensation/Cancel	258
Tool Nose Radius Compensation (for Machining Center System) ...	314
Tool Position Compensation	744
Tool Position Offset	329
Tool Radius Compensation	266
Tool Radius Compensation Operation	267
Tool Shape Input by Program	545
Torque Limitation Skip	972

U

Unidirectional Positioning	72
User Macro	425
User Macro Commands	450

V

Variable Commands	420
Variable-acceleration Pre-interpolation Acceleration/Deceleration ...	661
Variables Used in User Macros	443

W

Workpiece Coordinate Changing during Radius Compensation ...	312
Workpiece Coordinate System Preset	882
Workpiece Coordinate System Setting and Offset	872
Workpiece installation error compensation	841

Revision History

Date of revision	Manual No.	Revision details
Apr. 2015	IB(NA)1501277-A IB(NA)1501278-A	First edition created.
Sep. 2015	IB(NA)1501277-B IB(NA)1501278-B	<p>The descriptions of M800 Series/M80 Series were revised in response to S/W version A4.</p> <p>The following chapters were added.</p> <ul style="list-style-type: none"> ♦7.14.2 Inner Arc Override ♦15.9.3 Tool Shape Input by Program; G10 L100, G11 ♦15.9.4 R-Navi Data Input by Program; G10 L110, G11 ♦17.2.3 Tolerance Control ♦17.5 Spline Interpolation 2; G61.4 ♦18.1.6 Define by Selecting the Registered Machining Surface <p>The following chapters were revised.</p> <ul style="list-style-type: none"> ♦1.1 Coordinate Words and Control Axes ♦3.4.2 G Code Lists ♦5.3 Decimal Point Input ♦7.3 F1-digit Feed ♦7.12 Deceleration Check ♦7.14 Automatic Corner Override ♦10.2 Constant Surface Speed Control; G96, G97 ♦11.1 Tool Functions (T8-digit BCD) ♦12.3 Tool Length Compensation in the Tool Axis Direction; G43.1/G49 ♦13.1.4 Tapping Cycle; G84 ♦14.4 Macro Call Instructions ♦14.5.2 Local Variables (#1 to #33) ♦15.8 Manual Arbitrary Reverse Run Prohibition; G127 ♦16.2 Sub Part System Control ♦17.1 High-speed Machining Mode ♦17.2 High-accuracy Control ♦17.3 High-speed High-accuracy Control ♦17.7 Machining Condition Selection I; G120.1, G121 ♦18.1 Inclined Surface Machining; G68.2, G68.3 ♦19.3 Basic Machine Coordinate System Selection; G53 ♦19.6 Workpiece Coordinate System Setting and Offset; G54 to G59 (G54.1) ♦19.8 3-dimensional Coordinate Conversion; G68/G69 <p>The following chapters were moved.</p> <ul style="list-style-type: none"> ♦Parameter Input by Program; G10 L70/L100, G11 (15.6 -> 15.9.1) ♦Compensation Data Input by Program; G10 L2/L10/L11, G11 (12.7 -> 15.9.2) ♦Tool Life Management Data Input; G10,G11 (12.7 -> 15.10) <p>Other contents were added/revised/deleted according to specification.</p>
Apr. 2016	IB(NA)1501277-C IB(NA)1501278-C	<p>The descriptions of M800 Series/M80 Series were revised in response to S/W version B2.</p> <p>The following chapters were added.</p> <ul style="list-style-type: none"> ♦12.5 Tool Nose Radius Compensation (for Machining Center System) ♦12.8 Tool Position Compensation; G43.7 ♦15.9.3 Compensation Data Input by Program (Turning Tool); G10 L12/L13, G11 ♦16.2 Mixed Control ♦16.2.1 Arbitrary Axis Exchange; G140, G141, G142 ♦22 System Variables

(Continue to the next page)

Date of revision	Manual No.	Revision details
		<p style="text-align: right;">(Continued from the previous page)</p> <p>The following chapters were revised.</p> <ul style="list-style-type: none"> ♦Introduction ♦3.4 G Codes ♦5.4 Decimal Point Input ♦6.3 Circular Interpolation; G02/G03 ♦6.4 R Specification Circular Interpolation; G02, G03 ♦6.7 Helical Interpolation; G17, G18, G19, and G02, G03 ♦7.1 Rapid Traverse Rate ♦7.3 F1-digit Feed ♦7.7 Rapid Traverse Constant Inclination Acceleration/Deceleration ♦7.13 Rapid Traverse Block Overlap; G0.5 P1 ♦9.3 Index Table Indexing ♦10.2 Constant Surface Speed Control; G96, G97 ♦10.4 Spindle Position Control (Spindle/C Axis Control) ♦12.1 Tool Compensation ♦13.1.4 Tapping Cycle; G84 ♦14.1 Subprogram Control; M98, M99, M198 ♦14.2 Variable Commands ♦14.4 Macro Call Instructions ♦14.6 User Macro Commands ♦15.7 Normal Line Control; G40.1/G41.1/G42.1 (G150/G151/G152) ♦15.8 Manual Arbitrary Reverse Run Prohibition; G127 ♦15.9 Data Input by Program ♦16.3.1 Sub Part System Control I; G122 ♦17.1 High-speed Machining Mode ♦17.2 High-accuracy Control ♦17.3 High-speed High-accuracy Control ♦18.1 Inclined Surface Machining; G68.2, G68.3 ♦19.6 Workpiece Coordinate System Setting and Offset; G54 to G59 (G54.1) ♦19.10 Coordinate Rotation Input by Parameter ; G10 I_ J_/K_ ♦21.2 Skip Function; G31 <p>Other mistakes were corrected.</p>
Sep. 2016	IB(NA)1501277-D IB(NA)1501278-D	<p>The descriptions were revised in response to S/W version C1 of M800 Series/M80 Series.</p> <p>The descriptions were revised in response to S/W version A1 of C80 Series.</p> <p>The following chapters were added.</p> <ul style="list-style-type: none"> ♦18.3 Tool Center Point Control; G43.4, G43.5/G49 ♦18.5 3-dimensional Tool Radius Compensation (Tool's vertical-direction compensation); G40/G41.2,G42.2 ♦20.2 Enable Interfering Object Selection Data; G186 ♦22.29 System Variables (Interfering Object Selection) ♦22.30 System Variables (ZR Device Access Variables) [C80] <p>The following chapters were revised.</p> <ul style="list-style-type: none"> ♦Introduction ♦Precautions for Safety ♦3.2 File Format ♦3.4 G Codes ♦6.9 Cylindrical Interpolation; G07.1 ♦7.4 Feed Per Minute/Feed Per Revolution (Asynchronous Feed/Synchronous Feed); G94, G95 ♦12.3 Tool Radius Compensation ; G38,G39/G40/G41,G42 ♦14.1 Subprogram Control; M98, M99, M198 ♦14.6 User Macro Commands ♦15.9.1 Parameter Input by Program; G10 L70/L100, G11 ♦16.3 Sub Part System Control ♦17.1 High-speed Machining Mode ♦17.3 High-speed High-accuracy Control ♦17.5 Spline Interpolation 2; G61.4 <p style="text-align: right;">(Continue to the next page)</p>

Date of revision	Manual No.	Revision details
		<p style="text-align: right;">(Continued from the previous page)</p> <ul style="list-style-type: none"> ♦18.1 Tool Position Compensation; G43.7 ♦18.4 Inclined Surface Machining; G68.2, G68.3 ♦18.5 3-dimensional Tool Radius Compensation (Tool's vertical-direction compensation); G40/G41.2, G42.2 ♦19.7 Workpiece Coordinate System Preset; G92.1 ♦19.10 Coordinate Rotation Input by Parameter ; G10 I_ J_/K_ ♦21.6 Torque Limitation Skip; G160 ♦22.1 System Variable List ♦22.5 System Variables (Tool Information) ♦22.15 System Variables (Time Read Variables) ♦22.21 System Variables (Rotary Axis Configuration Parameter) ♦22.27 System Variables (R Device Access Variables) ♦22.28 System Variables (PLC Data Reading) <p>The following chapters were moved.</p> <ul style="list-style-type: none"> ♦Tool Length Compensation in the Tool Axis Direction ; G43.1/G49 (12.3 -> 18.2) ♦Tool Position Compensation; G43.7/G49 (12.8 -> 18.1) <p>Other mistakes were corrected.</p>
Mar. 2017	IB(NA)1501277-E IB(NA)1501278-E	<p>The descriptions of M800 Series/M80 Series were revised in response to S/W version C3.</p> <p>The following chapters were added.</p> <ul style="list-style-type: none"> ♦6.18 Involute Interpolation; G02.2/G03.2 ♦10.5 Spindle Speed Fluctuation Detection; G162/G163 ♦13.1.14 Thread Milling Cycle; G187 ♦15.11 Axis Name Extension ♦18.1 Spindle Synchronization ♦21.1.1 Stroke Check Before Travel in Stored Stroke Limit Area ♦25 Appendix 2: Command Value Range Lists <p>The following chapters were revised.</p> <ul style="list-style-type: none"> ♦Introduction ♦3.1 Program Format ♦3.4 G Codes ♦5.3 Decimal Point Input ♦6.9 Cylindrical Interpolation ; G07.1 ♦6.16 NURBS Interpolation ; G06.2 ♦7.14 Automatic Corner Override ♦10.1 Spindle Functions ♦13.1 Fixed Cycles ♦14.1 Subprogram Control; M98, M99, M198 ♦14.6 User Macro Commands ♦14.7 Macro Interruption; M96, M97 ♦15.10 Inputting The Tool Life Management Data ; G10,G11 ♦17.1 High-speed Machining Mode ♦17.3 High-speed High-accuracy Control ♦17.4 Spline Interpolation ; G05.1 Q2/Q0 ♦17.5 Spline Interpolation 2; G61.4 ♦19.1 Tool Position Compensation; G43.7/G49 ♦19.3 Tool Center Point Control; G43.4, G43.5/G49 ♦19.4 Inclined Surface Machining ; G68.2, G68.3/G69 ♦21.1 Stroke Check before Travel ; G22/G23 <p>The following chapters were removed.</p> <ul style="list-style-type: none"> ♦15.10 Tool Life Management II ; G10 L3, G11 <p>Other mistakes were corrected.</p>

Date of revision	Manual No.	Revision details
Aug. 2017	IB(NA)1501277-F IB(NA)1501278-F	<p>The descriptions were revised in response to S/W version A2 of C80 Series.</p> <p>The following chapters were revised.</p> <ul style="list-style-type: none"> ♦10.4 Spindle Position Control (Spindle/C Axis Control) ♦14.1 Subprogram Control; M98, M99, M198 ♦17.2.6 Multi-part System Simultaneous High-accuracy ♦18.1.2 Spindle Position Control under Spindle Synchronization Control ♦20.3 Basic Machine Coordinate System Selection ; G53 ♦23.30 System Variables (ZR Device Access Variables) [C80] <p>Other mistakes were corrected.</p>
Dec. 2017	IB(NA)1501277-G IB(NA)1501278-G	<p>The descriptions of M800 Series/M80 Series were revised in response to S/W version C7.</p> <p>The following chapters were added.</p> <ul style="list-style-type: none"> ♦5.2 Diameter Designation and Radius Designation ♦13.3 Fixed Cycle for Turning Machining ♦15.11 Interactive Cycle Insertion; G180 ♦19.3.1 Arc Command during Tool Center Point Control(G43.4/G43.5) ♦23.31 System Variables (NC Data Reading/Writing with API Section and Sub-section Designation) <p>The following chapters were revised.</p> <ul style="list-style-type: none"> ♦3.4.2 G Code Lists ♦5.4 Decimal Point Input ♦6.10 Circular Cutting ; G12,G13 ♦6.15 3-dimensional Circular Interpolation ; G02.4,G03.4 ♦10.2 Constant Surface Speed Control ; G96,G97 ♦10.4 Spindle Position Control (Spindle/C Axis Control) ♦13.1.7 Back Boring ; G87 ♦13.1.13 Fine Boring ; G76 ♦15.8 Manual Arbitrary Reverse Run Prohibition ; G127 ♦15.9 Data Input by Program ♦17.1 High-speed Machining Mode ♦17.2 High-accuracy Control ♦17.3 High-speed High-accuracy Control ♦17.7 Machining Condition Selection I ; G120.1,G121 ♦19.3 Tool Center Point Control; G43.4, G43.5/G49 ♦19.4 Inclined Surface Machining ; G68.2, G68.3/G69 ♦20.6 Workpiece Coordinate System Setting and Offset ; G54 to G59 (G54.1) ♦20.7 Workpiece Coordinate System Preset ; G92.1 ♦20.8 3-dimensional Coordinate Conversion ; G68/G69 ♦23.1 System Variables List ♦23.11 System Variables (Position Information) ♦24 Appx.1: Fixed Cycles <p>Other mistakes were corrected.</p>

Date of revision	Manual No.	Revision details
Apr. 2018	IB(NA)1501277-H IB(NA)1501278-H	<p>The descriptions were revised in response to MITSUBISHI CNC E80 Series.</p> <p>The descriptions were revised in response to S/W version D1 of MITSUBISHI CNC M800 Series/M80 Series.</p> <p>The following chapter was added.</p> <ul style="list-style-type: none"> ♦19.6 Workpiece Installation Error Compensation; G54.4 <p>The following chapters were revised.</p> <ul style="list-style-type: none"> ♦Introduction ♦10.4 Spindle Position Control (Spindle/C Axis Control) ♦12.2 Tool Length Compensation/Cancel; G43, G44 / G49 ♦13.1.4 Tapping Cycle; G84 ♦15.10.2 Inputting the Tool Life Management Data by G10 L30 Command; G10 L30,G11 ♦15.10.3 Precautions for Inputting the Tool Life Management Data ♦19.3 Tool Center Point Control; G43.4, G43.5/G49 ♦19.4.12 Precautions for Inclined Surface Machining ♦23.7 System Variables (Tool Life Management) <p>Other mistakes were corrected.</p>
Sep. 2018	IB(NA)1501277-J IB(NA)1501278-J	<p>The descriptions were revised in response to S/W version B0 of C80 Series.</p> <p>The following chapters were added.</p> <ul style="list-style-type: none"> ♦15.14 Machining Interruption [C80]; G26 ♦18.2 Multiple Spindle Synchronization Set Control [C80] <p>The following chapters were revised.</p> <ul style="list-style-type: none"> ♦3.2 File Format ♦3.4.2 G Code Lists ♦5.4 Decimal Point Input ♦7.13.1 Rapid Traverse Block Overlap for G00; G0.5 ♦10.4 Spindle Position Control (Spindle/C Axis Control) ♦14.1 Subprogram Control; M98, M99, M198 ♦14.2 Variable Commands ♦16.2.1 Arbitrary Axis Exchange ; G140, G141, G142 ♦16.3.1 Sub Part System Control I; G122 ♦17.2 High-accuracy Control ♦18.1 Spindle Synchronization ♦19.3 Tool Center Point Control; G43.4, G43.5/G49 ♦19.6 19.6 Workpiece Installation Error Compensation; G54.4 ♦23.7 System Variables (Tool Life Management) <p>Other mistakes were corrected.</p>

Global Service Network

AMERICA

MITSUBISHI ELECTRIC AUTOMATION INC. (AMERICA FA CENTER)

Central Region Service Center (Chicago)
500 CORPORATE WOODS PARKWAY, VERNON HILLS, ILLINOIS 60061, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650
Minneapolis, MN Service Satellite
Detroit, MI Service Satellite
Grand Rapids, MI Service Satellite
Lima, OH Service Satellite
Cleveland, OH Service Satellite
Indianapolis, IN Service Satellite
St. Louis, MO Service Satellite

South/East Region Service Center (Georgia)

1845 SATELLITE BOULEVARD STE. 450, DULUTH, GEORGIA 30097, U.S.A.
TEL: +1-678-258-4529 / FAX: +1-678-258-4519
Charleston, SC Service Satellite
Charlotte, NC Service Satellite
Raleigh, NC Service Satellite
Dallas, TX Service Satellite
Houston, TX Service Satellite
Hartford, CT Service Satellite
Knoxville, TN Service Satellite
Nashville, TN Service Satellite
Baltimore, MD Service Satellite
Pittsburg, PA Service Satellite
Newark, NJ Service Satellite
Syracuse, NY Service Satellite
Ft. Lauderdale, FL Service Satellite
Lafayette, LA Service Satellite

Western Region Service Center (California)

5900-B KATELLA AVE. - 5900-A KATELLA AVE. CYPRESS, CALIFORNIA 90630, U.S.A.
TEL: +1-714-699-2625 / FAX: +1-847-478-2650
San Francisco, CA Service Satellite
Seattle, WA Service Satellite
Denver, CO Service Satellite

Canada Region Service Center (Toronto)

4299 14TH AVENUE MARKHAM, ONTARIO L3R 0J2, CANADA
TEL: +1-905-754-3805 / FAX: +1-905-475-7935
Edmonton, AB Service Satellite
Montreal, QC Service Satellite

Mexico Region Service Center (Querétaro)

Parque Tecnológico Innovación Querétaro, Lateral Carretera Estatal 431, Km 2+200, Lote 91 Modulos 1 y 2
Hacienda la Machorra, CP 76246, El Marqués, Querétaro, México
TEL: +52-442-153 4250
Monterrey, NL Service Satellite
Mexico City, DF Service Satellite
Aguascalientes, AGS, Service Satellite

BRAZIL

Mitsubishi Electric do Brasil Comércio e Serviços Ltda.

Votorantim Office
AV. GISELE CONSTANTINO,1578, PARQUE BELA VISTA, VOTORANTIM-SP, BRAZIL CEP:18.110-650
TEL: +55-15-3023-9000
JOVIMAQ – Joinville, SC Service Satellite
MAQSERVICE – Canoas, RS Service Satellite

EUROPE

MITSUBISHI ELECTRIC EUROPE B.V.

European Service Headquarters (Dusseldorf, GERMANY)
Mitsubishi-Electric-Platz 1 40882 RATINGEN, GERMANY
TEL: +49-2102-486-1850 / FAX: +49-2102-486-5910

South Germany Service Center (Stuttgart)

KURZE STRASSE, 40, 70794 FILDERSTADT-BONLANDEN, GERMANY
TEL: +49-711-770598-121 / FAX: +49-711-770598-141

France Service Center (Paris)

25, BOULEVARD DES BOUVETS, 92741 NANTERRE CEDEX FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

France Service Satellite (Lyon)

120, ALLEE JACQUES MONOD 69800 SAINT PRIEST FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

Italy Service Center (Milan)

PALAZZO SIRIO INGRESSO 1, VIA COLLEONI, 7, 20864 AGRATE BRIANZA(MI), ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

Italy Service Satellite (Padova)

VIA G. SAVELLI, 24 - 35129 PADOVA, ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

U.K. Service Center

TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +49-2102-486-1850 / FAX: +49-2102-486-5910

Spain Service Center

CTRA. RUBI, 76-80 8174 SAINT CUGAT DEL VALLES, BARCELONA SPAIN
TEL: +34-935-65-2236 / FAX: +34-935-89-1579

Poland Service Center

UL.KRAKOWSKA 50, 32-083 BALICE, POLAND
TEL: +48-12-347-6500 / FAX: +48-12-630-4701

Hungary Service Center

MADARASZ IRODAPARK, MADARASZ VIKTOR UT 47-49, 1138 BUDAPEST, HUNGARY
TEL: +48-12-347-6500 / FAX: +48-12-630-4701

Turkey Service Center

MITSUBISHI ELECTRIC TURKEY A.S
SERIFALI MAHALLESİ NUTUK SOKAK. NO.5 34775
UMRANIYE, ISTANBUL, TURKEY
TEL: +90-216-526-3990 / FAX: +90-216-526-3995

Czech Republic Service Center

AutoCont Control Systems s.r.o (Service Partner)
KAFKOVA 1853/3, 702 00 OSTRAVA 2, CZECH REPUBLIC
TEL: +420-59-5691-185 / FAX: +420-59-5691-199

Russia Service Center

MITSUBISHI ELECTRIC RUSSIA LLC
LETNIKOVSKAYA STREET 2, BLD.1, 5TH 115114 MOSCOW, RUSSIA
TEL: +7-495-721-2070 / FAX: +7-495-721-2071

Sweden Service Center

HAMMARBACKEN 14, P.O.BOX 750 SE-19127, SOLLENTUNA, SWEDEN
TEL: +46-8-6251000 / FAX: +46-8-966877

Bulgaria Service Center

AKHNATON Ltd. (Service Partner)
4 ANDREJ LJAPCHEV BLVD. POB 21, BG-1756 SOFIA, BULGARIA
TEL: +359-2-8176009 / FAX: +359-2-9744061

Ukraine Service Center (Kharkov)

CSC Automation Ltd. (Service Partner)
4 B. YEVHENA SVERSTYUKA STR., 02002 KIEV, UKRAINE
TEL: +380-44-494-3344 / FAX: +380-44-494-3366

Belarus Service Center

TECHNIKON Ltd. (Service Partner)
NEZAVISIMOSTI PR.177, 220125 MINSK, BELARUS
TEL: +375-17-393-1177 / FAX: +375-17-393-0081

South Africa Service Center

MOTIONTRONIX (Service Partner)
P.O. BOX 9234, EDLEEN, KEMPTON PARK GAUTENG, 1625, SOUTH AFRICA
TEL: +27-11-394-8512 / FAX: +27-11-394-8513

ASEAN

MITSUBISHI ELECTRIC ASIA PTE. LTD. (ASEAN FA CENTER)
Singapore Service Center
307 ALEXANDRA ROAD #05-01/02 MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 / FAX: +65-6476-7439

Philippines Service Center
Flexible Automation system corporation (Service Partner)
UNIT NO.411, ALABAMG CORPORATE CENTER KM 25. WEST SERVICE ROAD
SOUTH SUPERHIGHWAY, ALABAMG MUNTINLUPA METRO MANILA, PHILIPPINES 1771
TEL: +63-2-807-2416 / FAX: +63-2-807-2417

VIETNAM

MITSUBISHI ELECTRIC VIETNAM CO.,LTD
Vietnam Ho Chi Minh Service Center
UNIT 01-04, 10TH FLOOR, VINCOM CENTER 72 LE THANH TON STREET, DISTRICT1,
HO CHI MINH CITY, VIETNAM
TEL: +84-28-3910 5945 / FAX: +84-28-3910 5946

Vietnam Hanoi Service Center
24TH FLOOR, HANDICO TOWER, PHAM HUNG ROAD, ME TRI HA WARD,
NAM TU LIEM DISTRICT, HA NOI CITY, VIETNAM
TEL: +84-24-3937-8075 / FAX: +84-24-3937-8076

INDONESIA

PT. MITSUBISHI ELECTRIC INDONESIA
Indonesia Service Center (Cikarang)
JL. KENARI RAYA BLOK G2-07A, DELTA SILICON 5, LIPPO CIKARANG - BEKASI 17550, INDONESIA
TEL: +62-21-2961-7797 / FAX: +62-21-2961-7794

MALAYSIA

MITSUBISHI ELECTRIC SALES MALAYSIA SDN. BHD.
Malaysia Service Center (Kuala Lumpur Service Center)
LOT 11, JALAN 219, P.O BOX 1036, 46860 PETALING JAYA, SELANGOR DARUL EHSAN. MALAYSIA
TEL: +60-3-7960-2628 / FAX: +60-3-7960-2629
Johor Bahru Service Satellite

THAILAND

MITSUBISHI ELECTRIC FACTORY AUTOMATION (THAILAND) CO.,LTD.
Thailand Service Center (Bangkok)
12TH FLOOR, SV.CITY BUILDING, OFFICE TOWER 1, NO. 896/19 AND 20 RAMA 3 ROAD,
KWAENG BANGPONGPANG, KHET YANNAWA, BANGKOK 10120,THAILAND
TEL: +66-2-682-6522 / FAX: +66-2-682-6020
BOWIN Service Center (Chonburi)

INDIA

MITSUBISHI ELECTRIC INDIA PVT., LTD.
CNC Technical Center (Bangalore)
PLOT NO. 56, 4TH MAIN ROAD, PEENYA PHASE 3,
PEENYA INDUSTRIAL AREA, BANGALORE 560058, KARNATAKA, INDIA
TEL : +91-80-4655-2121 FAX : +91-80-4655-2147
Chennai Service Satellite
Coimbatore Service Satellite
Hyderabad Service Satellite

North India Service Center (Gurgaon)
2ND FLOOR, TOWER A&B, DLF CYBER GREENS, DLF CYBER CITY,
DLF PHASE-III, GURGAON- 122 002, HARYANA, INDIA
TEL: +91-124-4630 300 FAX: +91-124-4630 399
Ludhiana Service Satellite
Pant Nagar Service Satellite
Delhi Service Satellite
Jamshedpur Service Satellite
Manesar Service Satellite

West India Service Center (Pune)
EMERALD HOUSE, EL-3, J BLOCK, M.I.D.C., BHOSARI, PUNE - 411026, MAHARASHTRA, INDIA
TEL : +91-20-2710 2000 FAX : +91-20-2710 2100
Kolhapur Service Satellite
Aurangabad Service Satellite
Mumbai Service Satellite

West India Service Center (Ahmedabad)
204-209, 2ND FLOOR, 31FIVE, CORPORATE ROAD PRAHLADNAGAR,
AHMEDABAD -380015, GUJARAT, INDIA
TEL : + 91-079-6777 7888
Rajkot Service Satellite

CHINA

MITSUBISHI ELECTRIC AUTOMATION (CHINA) LTD. (CHINA FA CENTER)
China Shanghai Service Center
1-3,5-10,18-23/F, NO.1386 HONG QIAO ROAD, CHANG NING QU,
SHANGHAI 200336, CHINA
TEL: +86-21-2322-3030 / FAX: +86-21-2322-3000*8422
China QINGDAO Service Center
China SUZHOU Service Center
China Ningbo Service Partner
China Jinan Service Partner
China Hangzhou Service Partner
China Suzhou Service Partner

China Beijing Service Center
5/F, ONE INDIGO, 20 JIUXIANQIAO ROAD CHAOYANG DISTRICT,
BEIJING 100005, CHINA
TEL: +86-10-6518-8830 / FAX: +86-10-6518-2938
China Beijing Service Partner

China Tianjin Service Center
UNIT 2003, TIANJIN CITY TOWER, NO 35 YOUYI ROAD, HEXI DISTRICT,
TIANJIN 300061, CHINA
TEL: +86-22-2813-1015 / FAX: +86-22-2813-1017

China XIAN Service Center
China Changchun Service Partner

China Chengdu Service Center
1501-1503,15F.GUANG-HUA CENTRE BUILDING-C,NO.98 NORTH GUANG HUA 3th RD.,
CHENGDU,610000,CHINA
TEL: +86-28-8446-8030 / FAX: +86-28-8446-8630

China Shenzhen Service Center
LEVEL8, GALAXY WORLD TOWER B, 1 YABAO ROAD, LONGGANG DISTRICT,
SHENZHEN 518129, CHINA
TEL: +86-755-2399-8272 / FAX: +86-755-8229-3686

China Dongguang Service Center
China Xiamen Service Partner
China DongGuang Service Partner

China Dalian Service Center
DONGBEI 3-S, DALIAN ECONOMIC & TECHNICAL DEVELOPMENTZONE, LIAONING PROVINCE,
116600, CHINA
TEL: +86-411-8765-5951 / FAX: +86-411-8765-5952

KOREA

MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD. (KOREA FA CENTER)
Korea Service Center
8F GANGSEO HANGANG XI-TOWER A, 401 YANGCHEON-RO, GANGSEO-GU,
SEOUL 07528 KOREA
TEL: +82-2-3660-9609 / FAX: +82-2-3664-8668
Korea Daegu Service Satellite

TAIWAN

MITSUBISHI ELECTRIC TAIWAN CO., LTD. (TAIWAN FA CENTER)
Taiwan Taichung Service Center
NO.8-1, INDUSTRIAL 16TH RD., TAICHUNG INDUSTRIAL PARK, SITUN DIST.,
TAICHUNG CITY 40768, TAIWAN
TEL: +886-4-2359-0688 / FAX: +886-4-2359-0689

Taiwan Taipei Service Center
10F, NO.88, SEC.6, CHUNG-SHAN N. RD., SHI LIN DIST., TAIPEI CITY 11155, TAIWAN
TEL: +886-2-2833-5430 / FAX: +886-2-2833-5433

Taiwan Tainan Service Center
11F-1, NO.30, ZHONGZHENG S. ROAD, YONGKANG DISTRICT, TAINAN CITY 71067, TAIWAN,
TEL: +886-6-252-5030 / FAX: +886-6-252-5031

OCEANIA

MITSUBISHI ELECTRIC AUSTRALIA PTY. LTD.
Oceania Service Center
348 VICTORIA ROAD, RYDALMERE, N.S.W. 2116 AUSTRALIA
TEL: +61-2-9684-7269 / FAX: +61-2-9684-7245

Notice

Every effort has been made to keep up with software and hardware revisions in the contents described in this manual. However, please understand that in some unavoidable cases simultaneous revision is not possible.

Please contact your Mitsubishi Electric dealer with any questions or comments regarding the use of this product.

Duplication Prohibited

This manual may not be reproduced in any form, in part or in whole, without written permission from Mitsubishi Electric Corporation.

COPYRIGHT 2015-2018 MITSUBISHI ELECTRIC CORPORATION
ALL RIGHTS RESERVED

MITSUBISHI CNC

MITSUBISHI ELECTRIC CORPORATION

HEAD OFFICE : TOKYO BLDG.,2-7-3 MARUNOUCHI,CHIYODA-KU,TOKYO 100-8310,JAPAN

MODEL	M800/M80/E80/C80 Series
MODEL CODE	
Manual No.	IB-1501277