MITSUBISHI ELECTRIC

Changes for the Better

Numerical Control (CNC)

Programming Manual (Machining Center System)

M800V/M80V Series

IB-1501621(ENG)-L

Introduction

This manual describes the information for programming on Mitsubishi Electric CNC.
Improper handling can cause unexpected malfunctions. To use this device correctly, be sure to read this manual before use.
Supported models of this manual are as follows:

Supported models	Details
M800VW Series	M850VW, M830VW
M800VS Series	M850VS, M830VS
M80VW Series	M80VW
M80V Series	M80V TypeA, M80V TypeB

Abbreviations for model names used in this manual are as follows:

Abbreviations	Supported models
M800V, M800V Series	M800VW Series/M800VS Series
M80V, M80V Series	M80VW Series/M80V Series
M800V/M80V, M800V/M80V Series	M800VW Series/M800VS Series/M80VW Series/M80V Series
M8V, M8V Series	M800VW Series/M800VS Series/M80VW Series/M80V Series

To safely use this CNC unit, thoroughly study the "Precautions for Safety" before use.
Be sure to keep this manual on hand so that users can refer to it at any time.
Also refer to the manuals on "Manual List" as necessary.

■Notes on reading this manual

(1) The description concerning "signals" in this manual refers to information transmission between a machine and PLC or between CNC and PLC. The method on controlling the signals (ON/OFF) differs depending on the machine. Refer to the manual issued by the machine tool builder (MTB).

(2) 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 this manual. Confirm the specifications for your machine with the manual issued by the MTB.

■Details described in this manual

(1) In this manual, the following abbreviations might be used.
L system: Lathe system
M system: Machining center system
MTB: Machine tool builder

(2) In this manual, some "auxiliary axis" indicates PLC axis where axis indexing is performed by index position command.

Manual List

Manuals related to M800V/M80V 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
M800V/M80V Series Instruction Manual (1/2)	IB-1501806	♦ Explanation for display unit and screen operation ♦ Settings before machining, setup, etc.
M800V/M80V Series Instruction Manual (2/2)	IB-1501807	♦ Explanation for settings at machining, operation status, etc. ♦ Explanation for other functions
M800V/M80V Series Programming Manual (Lathe System) (1/2)	IB-1501619	♦ G code programming for lathe system ♦ Basic functions, etc.
M800V/M80V Series Programming Manual (Lathe System) (2/2)	IB-1501620	♦ G code programming for lathe system ♦ Functions for multi-part system, high-accuracy function, etc.
M800V/M80V Series Programming Manual (Machining Center System) (1/2)	IB-1501621	♦ G code programming for machining center system ♦ Basic functions, etc.
M800V/M80V Series Programming Manual (Machining Center System) (2/2)	IB-1501622	♦ G code programming for machining center system ♦ Functions for multi-part system, high-accuracy function, etc.
M800V/M80V Series Alarm Manual	IB-1501808	♦ Alarms
M800V/M80V Series Parameter Manual	IB-1501809	♦ Parameters

Manuals for MTBs (NC)

Manual	IB No.	Purpose and Contents
M800V/M80V Series Specifications Manual (Function)	IB-1501610	• Model selection • Outline of various functions
M800V/M80V Series Specifications Manual (Hardware)	IB-1501611	• Model selection • Specifications of hardware unit
M800VW/M80VW Series Connection and Setup Manual	IB-1501612	• Detailed specifications of hardware unit • Installation, connection, wiring, setup (startup/adjustment)
M800VS/M80V Series Connection and Setup Manual	IB-1501613	• Detailed specifications of hardware unit • Installation, connection, wiring, setup (startup/adjustment)
M800V/M80V Series PLC Development Manual	IB-1501614	• Electrical design • I/O relation (assignment, setting, connection), field network • Development environment (PLC on-board, peripheral development environment), etc.
M800V/M80V Series PLC Programming Manual (1/2)	IB-1501667	• Electrical design • Sequence programming • Explanation for instructions, functions, and parameters
M800V/M80V Series PLC Programming Manual (2/2)	IB-1501668	• Electrical design • Sequence programming • Usage examples of instructions
M800V/M80V Series PLC Interface Manual (1/2)	IB-1501804	• Electrical design • List of interface signals between NC and PLC
M800V/M80V Series PLC Interface Manual (2/2)	IB-1501805	• Electrical design • Details of interface signals between NC and PLC
M800V/M80V Series Maintenance Manual	IB-1501617	• Cleaning and replacement for each unit • Other items related to maintenance
NC Designer2 Instruction Manual	IB-1501250	• Explanation for handling NC NC Designer2
High Speed Processing Unit User's Manual	IB-1501714	• Specifications for high speed processing unit (HPU)
NC Machining AI Diagnosis NC MachiningAID Instruction Manual	IB-1501762	• Explanation for handling NC MachiningAID

Manuals for MTBs (drive section)

Manual	IB No.	Contents
MDS-E/EH Series Specifications Manual	IB-1501226	• Specifications for power supply regeneration type
MDS-E/EH Series Instruction Manual	IB-1501229	• Instruction for power supply regeneration type
MDS-EJ/EJH Series Specifications Manual	IB-1501232	• Specifications for regenerative resistor type
MDS-EJ/EJH Series Instruction Manual	IB-1501235	• Instruction for regenerative resistor type
MDS-EM/EMH Series Specifications Manual	IB-1501238	• Specifications for multi-hybrid, power supply regeneration type
MDS-EM/EMH Series Instruction Manual	IB-1501241	• Instruction for multi-hybrid, power supply regeneration type
DATA BOOK	IB-1501252	• Specifications of servo drive unit, spindle drive unit, motor, etc.
MDS-EX-CVP Series Specifications and Instruction Manual	IB-1501587	• Specifications and instruction for the power supply unit with large capacity

Manuals for MTBs (Others)

Manual	No.	Purpose and Contents
GOT2000 Series User's Manual (Hardware)	SH-081194ENG	• Outline of hardware such as part names, external dimensions, installation, wiring, maintenance, etc. of GOTs
GOT2000 Series User's Manual (Utility)	SH-081195ENG	• Outline of utilities such as screen display setting, operation method, etc. of GOTs
GOT2000 Series User's Manual (Monitor)	SH-081196ENG	• Outline of each monitor function of GOTs
GOT2000 Series Connection Manual (Mitsubishi Electric Products)	SH-081197ENG	• Outline of connection types and connection method between GOT and Mitsubishi Electric connection devices
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
GT Designer3 (GOT2000) Screen Design Manual	SH-081220ENG	• Outline of screen design method using screen creation software GT Designer3
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-0800004	• Explanation for operations using data conversion tool GX Converter
GX Works2 Installation Instructions	BCN-P5999-0944	• Explanation for the operating environment and installation method of GX Works2
GX Works2 Version 1 Operating Manual (Common)	SH-080779ENG	• Explanation for the system configuration of GX Works2 and the functions common to Simple project and Structured project such as parameter setting, operation method for the online function
GX Works2 Version 1 Operating Manual (Simple Project)	SH-080780ENG	• Explanation for methods for such as creating and monitoring programs in Simple project of GX Works2
GX Works2 Version 1 Operating Manual (Simple Project, Function Block)	SH-080984ENG	• Explanation for methods for such as creating function blocks, pasting function blocks to sequence programs, and operating FB library in Simple project of GX Works2
GX Works2 Version 1 Operating Manual (Structured Project)	SH-080781ENG	• Explanation for methods for such as creating and monitoring programs in Structured project of GX Works2
GX Works3 Installation Instructions	BCN-P5999-0391	• Explanation for the operating environment and installation method of GX Works3
MELSEC-Q/L/F Structured Programming Manual (Fundamentals)	SH-080782ENG	• Explanation for programming methods, types of programming languages, etc. required to create structured programs
MELSEC-Q/L Structured Programming Manual (Application Functions)	SH-080784ENG	• Explanation for specifications and functions related to application functions which can be used in structured programs
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
MELSEC iQ-R Programming Manual (CPU Module Instructions, Standard Functions/Function Blocks)	SH-081266ENG	• Explanation for instructions, general-purpose functions, and general-purpose function blocks required for programming the sequencer MELSEC iQ-R series

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 indicated a thing that is pompously (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 Electric 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.

⚠ To protect the availability, integrity and confidentiality of the NC system against cyber-attacks including unauthorized access, denial-of-service (Dos) (*1) attack, and computer virus from external sources via a network, take security measures such as firewall, VPN, and anti-virus software.
(*1) Denial-of-service (Dos) refers to a type of cyber-attack that disrupts services by overloading the system or by exploiting a vulnerability of the system.

⚠ Mitsubishi Electric assumes no responsibility for any problems caused to the NC system by any type of cyber-attacks including DoS attack, unauthorized access and computer virus.

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.

⚠ CAUTION

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.

⚠ The program including a character of any language other than the display language is not correctly displayed. Do not edit such a program. Any part of the program other than the comment part may also be changed if edited.

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, CC-Link IE/field Basic, CC-Link IE TSN, 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®, Access® and Microsoft Edge 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 SD-3C, 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 is either a trademark or a registered trademark of PROFIBUS User Organization.
Oracle® and JavaScript are either trademarks or registered trademarks 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.
QR Code is a registered trademark of DENSO WAVE INCORPORATED.
TRELLIX is a registered trademark or a trademark of Musarubra US LLC or its subsidiaries in the United States and other countries.
Punchtap is licensed by EMUGE.
The term HDMI and the HDMI Logo are trademarks or registered trademarks of HDMI Licensing Administrator, Inc. in the United States and other countries.
Wi-Fi is either a trademark or a registered trademark of Wi-Fi Alliance.
BiSS is a registered trademark of iC-Haus GmbH.
SPRINT™ is a trademark of Renishaw plc.
Android and Google Chrome are either trademarks or registered trademarks of Google Inc.
iPadOS, macOS and Safari are trademarks of Apple Inc., registered in the U.S. and other countries and regions.

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 (A 급) 전자파적합기기로서 판매자 또는 사용자는 이 점을 주의하시기 바라며 가정외의 지역에 서 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 and Program Command 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 17
3.3.1 Optional Block Skip; / 17
3.3.2 Optional Block Skip Addition ; /n 19
3.4 G Code 21
3.4.1 Modal, Unmodal 21
3.4.2 G Code Lists 21
3.5 Precautions before Starting Machining 27

4 Pre-read Buffer 29
4.1 Pre-read Buffer 30

5 Position Commands 31
5.1 Position Command Methods ; G90,G91 32
5.2 Diameter Designation and Radius Designation 34
5.2.1 Diameter/Radius Designation Switch; G10.9 34
5.3 Inch/Metric Conversion; G20, G21 37
5.4 Decimal Point Input 40
5.4.1 Decimal Point Command Valid/Invalid Address Changeover 48

6 Interpolation Functions 49
6.1 Positioning (Rapid Traverse); G00 50
6.2 Linear Interpolation; G01 53
6.3 Circular Interpolation; G02, G03 55
6.4 R Specification Circular Interpolation; G02, G03 61
6.5 Plane Selection; G17, G18, G19 64
6.6 Thread Cutting 66
6.6.1 Constant Lead Thread Cutting; G33 66
6.6.2 Inch Thread Cutting; G33 70
6.7 Helical Interpolation; G02, G03 72
6.8 Unidirectional Positioning 77
6.8.1 Unidirectional Positioning; G60 77
6.8.2 Axis-based Unidirectional Positioning 79
6.9 Cylindrical Interpolation; G07.1 80
6.10 Circular Cutting; G12,G13 88
6.11 Polar Coordinate Interpolation; G12.1, G13.1/G112, G113 90
6.12 Exponential Interpolation; G02.3, G03.3 99
6.13 Polar Coordinate Command; G16 106
6.14 Spiral/Conical Interpolation; G02.1/G03.1 (Type 1), G02/G03 (Type 2) 113
6.15 3-dimensional Circular Interpolation; G02.4, G03.4 117
6.16 NURBS Interpolation; G06.2 123
6.17 Hypothetical Axis Interpolation; G07 129
6.18 Involute Interpolation; G02.2/G03.2 131

7 Feed Functions 145
7.1 Rapid Traverse Rate 146
7.1.1 Rapid Traverse Rate 146
7.1.2 G00 Feedrate Command (,F Command) 147

7.2 Cutting Feedrate 151
7.3 F1-digit Feed 152
7.4 Feed Per Minute/Feed Per Revolution (Asynchronous Feed/Synchronous Feed); G94, G95 154
7.5 Inverse Time Feed; G93 156
7.6 Feedrate Designation and Effects on Control Axes 161
7.7 Selection of Axis (Axes) for Feedrate Command; G130 166
7.8 Rapid Traverse Constant-gradient Acceleration/Deceleration 170
7.9 Rapid Traverse Constant-gradient Multi-step Acceleration/Deceleration 176
7.10 Cutting Feed Constant-gradient Acceleration/Deceleration 184
7.11 Exact Stop Check; G09 192
7.12 Exact Stop Check Mode; G61 196
7.13 Deceleration Check 197
7.13.1 Deceleration Check 197
7.13.2 Deceleration Check When Movement in the Opposite Direction Is Reversed 205
7.14 Rapid Traverse Block Overlap; G0.5 P1 208
7.14.1 Rapid Traverse Block Overlap for G00; G0.5 210
7.14.2 Rapid Traverse Block Overlap for G28 221
7.15 Automatic Corner Override 223
7.15.1 Automatic Corner Override ; G62 230
7.15.2 Inner Arc Override 231
7.16 Tapping Mode; G63 232
7.17 Cutting Mode; G64 233

8 Dwell 235
8.1 Dwell (Time-based Designation); G04 236
8.2 Dwell (Revolution-based Designation); G04 239

9 Miscellaneous Functions 243
9.1 Miscellaneous Functions (M8-digits) 244
9.2 Second Miscellaneous Functions (A8-digits, B8-digits or C8-digits) 246
9.3 Index Table Indexing 247

10 Spindle Functions 253
10.1 Spindle Functions 254
10.2 Constant Surface Speed Control; G96, G97 255
10.3 Spindle Clamp Speed Setting; G92 262
10.4 Spindle Position Control (Spindle/C Axis Control) 264
10.5 Spindle Speed Fluctuation Detection; G162/G163 273

11 Tool Functions (T command) 281
11.1 Tool Functions (T8-digit BCD) 282

12 Tool Compensation Functions 283
12.1 Tool Compensation 284
12.1.1 Tool Compensation 284
12.1.2 Number of Tool Offset Sets Allocation to Part Systems 288
12.2 Tool Length Offset/Cancel; G43, G44 / G49 290
12.3 Tool Radius Compensation; G38,G39/G40/G41,G42 308
12.3.1 Tool Radius Compensation Operation 309
12.3.2 Other Commands and Operations during Tool Radius Compensation 318
12.3.3 G41/G42 Commands and I, J, K Designation 328
12.3.4 Interrupts during Tool Radius Compensation 334
12.3.5 General Precautions for Tool Radius Compensation 336
12.3.6 Changing of Compensation No. during Compensation Mode 337
12.3.7 Start of Tool Radius Compensation and Z Axis Cut in Operation 339
12.3.8 Interference Check 341
12.3.9 Diameter Designation of Compensation Amount 351
12.3.10 Workpiece Coordinate Changing during Radius Compensation 353
12.3.11 Tool Offset Command Check 355
12.3.12 Arc Intersection Point Calculation Switch 355
12.4 Tool Nose Radius Compensation (for Machining Center System) 356
12.5 3-dimensional Tool Radius Compensation; G40/G41, G42 359
12.6 Tool Position Offset; G45 to G48 371
12.7 Tool Offset Number 379
12.7.1 H Stand-alone Command 379
12.7.2 Tool Offset Command Check 381

13 Fixed Cycle 383
13.1 Standard Fixed Cycle 384
13.1.1 Drilling, Spot Drilling; G81 389
13.1.2 Drilling, Counter Boring; G82 390
13.1.3 Deep Hole Drilling Cycle; G83 391
13.1.3.1 Deep Hole Drilling Cycle 391
13.1.3.2 Small Diameter Deep Hole Drilling Cycle 393
13.1.4 Tapping Cycle; G84 396
13.1.5 Boring; G85 412
13.1.6 Boring; G86 413
13.1.7 Back Boring; G87 414
13.1.8 Boring; G88 418
13.1.9 Boring; G89 419
13.1.10 Stepping Cycle; G73 420
13.1.11 Reverse Tapping Cycle; G74 422
13.1.12 Circular Cutting; G75 424
13.1.13 Fine Boring; G76 426
13.1.14 Thread Milling Cycle; G187 428
13.1.15 Punchtap Cycle 432
13.1.16 Precautions for Using a Fixed Cycle 445
13.1.17 Initial Point and R Point Level Return; G98, G99 447
13.1.18 Setting of Workpiece Coordinates in Fixed Cycle Mode 448
13.1.19 Drilling Cycle High-Speed Retract 449
13.1.20 Acceleration/Deceleration Mode Change in The Fixed Cycle for Drilling 453
13.1.21 Chip Removal 455
13.1.22 Cutting Reduction Amount Specification Method 460
13.2 Special Fixed Cycle 463
13.2.1 Bolt Hole Cycle; G34 464
13.2.2 Line at Angle; G35 465
13.2.3 Arc; G36 466
13.2.4 Grid; G37.1 467
13.3 Fixed Cycle for Turning Machining 469
13.3.1 Longitudinal Cutting Cycle; G174 470
13.3.2 Thread Cutting Cycle; G175 473
13.3.3 Face Cutting Cycle; G176 477

14 Macro Functions 481
14.1 Subprogram Control; M98, M99, M198 482
14.1.1 Subprogram Call; M98, M99 482
14.1.2 Subprogram Call; M198 488
14.1.3 Figure Rotation; M98 I_J_K_ 490
14.2 Variable Commands 493
14.3 User Macro 499
14.4 Macro Call Instructions 500
14.4.1 Simple Macro Calls; G65 501
14.4.2 Modal Call A (Movement Command Call) ; G66 505
14.4.3 Modal Call B (for Each Block); G66.1 507
14.4.4 G Code Macro Call 509
14.4.5 Miscellaneous Command Macro Call (for M, S, T, B Code Macro Call) 511
14.4.6 Detailed Description for Macro Call Instruction 513
14.4.7 Handling the Macro Call Command 515
14.4.8 ASCII Code Macro 519
14.5 Variables Used in User Macros 523
14.5.1 Common Variables 525
14.5.2 Extended Common Variable 526
14.5.2.1 Extended Common Variable I/II 526
14.5.2.2 Extended Common Variable III 528
14.5.2.3 Precautions for Extended Common Variable 530
14.5.3 Local Variables (#1 to #33) 531
14.5.4 System Variables 534
14.6 User Macro Commands 535
14.6.1 Operation Commands 535
14.6.2 Control Commands 540
14.6.3 External Output Commands; POPEN, PCLOS, DPRNT 546
14.6.4 Precautions 550
14.6.5 Actual Examples of Using User Macros 552

14.7 Macro Interruption; M96, M97 ... 556
14.8 Variable Value Display in Program Being Executed ... 567

15 Program Support Functions ... 571
15.1 Corner Chamfering I/Corner Rounding I ... 572
15.1.1 Corner Chamfering I ; G01 X_ Y_ ,C ... 572
15.1.2 Corner Rounding I ; G01 X_ Y_ ,R_ ... 574
15.1.3 Corner Chamfering Expansion/Corner Rounding Expansion ... 576
15.1.4 Interrupt during Corner Chamfering/Interrupt during Corner Rounding ... 578
15.2 Corner Chamfering II/Corner Rounding II ... 579
15.2.1 Corner Chamfering II ; G01/G02/G03 X_ Y_ ,C_ ... 579
15.2.2 Corner Rounding II ; G01/G02/G03 X_ Y_ ,R_ ... 581
15.2.3 Corner Chamfering Expansion/Corner Rounding Expansion ... 582
15.2.4 Interrupt during Corner Chamfering/Interrupt during Corner Rounding ... 582
15.3 Linear Angle Command; G01 X_/Y_ A_/,A_ ... 583
15.4 Geometric I; G01 A_ ... 584
15.5 Geometric IB ... 586
15.5.1 Geometric IB (Automatic Calculation of Contact Point of Two Circular Arcs); G02/G03 P_Q_ /R_ ... 587
15.5.2 Geometric IB (Automatic Calculation of Intersection Point between Line And Circular Arc) ; G01 A_ , G02/G03 P_Q_H_ ... 589
15.5.3 Geometric IB (Automatic Calculation of Contact Point between Line And Circular Arc) ; G01 A_ , G02/G03 R_H_ ... 592
15.6 Mirror Image by G code ; G50.1,G51.1 ... 594
15.7 Normal Line Control; G40.1/G41.1/G42.1 (G150/G151/G152) ... 598
15.8 Manual Arbitrary Reverse Run Prohibition ; G127 ... 618
15.9 Data Input by Program ... 624
15.9.1 Parameter Input by Program; G10 L70, G11 ... 624
15.9.2 Compensation Data Input by Program (Tool Compensation Amount) ; G10 L10/L11/L12/L13, G11 ... 626
15.9.3 Compensation Data Input by Program (Workpiece Offset Amount) ; G10 L2/L20, G11 ... 629
15.9.4 Compensation Data Input by Program (Turning Tool) ; G10 L12/L13, G11 ... 633
15.9.5 Tool Shape Input by Program; G10 L100, G11 ... 635
15.9.6 R-Navi Data Input by Program; G10 L110/L111, G11, G68.2, G69 ... 638
15.10 Tool Life Management ... 642
15.10.1 Inputting the Tool Life Management Data by G10 L3 Command; G10 L3, G11 ... 642
15.10.2 Inputting the Tool Life Management Data by G10 L30 Command; G10 L30, G11 ... 645
15.10.3 Precautions for Inputting the Tool Life Management Data ... 649
15.10.4 Tool Life Management Set Allocation to Part Systems ... 650
15.11 Interactive Cycle Insertion; G180 ... 652
15.11.1 Interactive Cycle Insertion ... 652
15.11.2 Interactive Macro ... 655
15.12 Axis Name Extension ... 656
15.13 Vibration Cutting Control (VCC); G08.5 ... 662
15.14 Two-dimensional Barcode Engraving Cycle; G136 ... 679
15.15 Text Engraving Cycle; G135 ... 694
15.16 Cutting Load Control; G162.1 ... 707
15.17 Orbit Machining; G08.6 ... 716

16 Multi-part System Control ... 731
16.1 Timing Synchronization Operation ... 732
16.1.1 Timing Synchronization Operation (! code) !n (!m ...) L ... 732
16.1.2 Timing Synchronization Operation with Start Point Designated (Type 1) ; G115 ... 736
16.1.3 Timing Synchronization Operation with Start Point Designated (Type 2) ; G116 ... 739
16.1.4 Timing Synchronization Operation Function Using M codes ; M*** ... 742
16.1.5 Timing Synchronization When Timing Synchronization Ignore Is Set ... 746
16.2 Mixed Control ... 749
16.2.1 Arbitrary Axis Exchange ; G140, G141, G142 ... 749
16.3 Control Axis Synchronization between Part Systems ; G125 ... 752
16.4 Sub Part System Control ... 762
16.4.1 Sub Part System Control I; G122 ... 762
16.5 Loader Control Part System ... 778

17 High-speed High-accuracy Control ... 781
17.1 High-speed Machining Mode ... 782
17.1.1 High-speed Machining Mode I, II; G05 P1, G05 P2 ... 782
17.2 High-accuracy Control ... 791
17.2.1 High-accuracy Control ; G61.1, G08 ... 791

17.2.2 SSS Control ... 811
17.2.3 Tolerance Control ... 815
17.2.4 Variable-acceleration Pre-interpolation Acceleration/Deceleration ... 819
17.2.5 Initial High-accuracy Control ... 822
17.2.6 High-speed Machining Mode and High-accuracy Control in Multi-part System ... 823
17.3 High-speed High-accuracy Control ... 825
17.3.1 High-speed High-accuracy Control I, II, III ; G05.1 Q1/Q0, G05 P10000/P0, G05 P20000/P0 ... 825
17.3.2 Fairing ... 843
17.3.3 Smooth Fairing ... 844
17.3.4 Cutting Speed Clamp with Acceleration Rate Judgment ... 853
17.3.5 High-speed Mode Corner Deceleration ... 854
17.3.6 Precautions on High-speed High-accuracy Control ... 855
17.4 Spline Interpolation ; G05.1 Q2/Q0 ... 858
17.5 Spline Interpolation 2; G61.4 ... 866
17.6 High-accuracy Spline Interpolation ; G61.2 ... 876
17.7 Machining Condition Selection I ; G120.1, G121 ... 878

18 Advanced Multi-Spindle Control Function ... 883
18.1 Spindle Synchronization ... 884
18.1.1 Spindle Synchronization I; G114.1 ... 885
18.1.2 Spindle Position Control (Spindle/C Axis Control) under Spindle Synchronization Control ... 896
18.2 Tool Spindle Synchronization I ... 901
18.2.1 Tool Spindle Synchronization IA (Spindle-Spindle, Polygon) ; G114.2, G113.1 ... 901
18.2.2 Tool Spindle Synchronization IB (Spindle-Spindle, Polygon) ; G51.2/G50.2 or G251/G250 ... 907
18.2.3 Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon) ; G51.2/G50.2 or G251/G250 ... 912
18.3 Tool Spindle Synchronization II ... 916
18.3.1 Tool Spindle Synchronization II (Hobbing) ; G114.3/G113 ... 916

19 Advanced Machining Control ... 933
19.1 Tool Position Compensation; G43.7/G49 ... 934
19.2 Tool Length Compensation Along the Tool Axis; G43.1/G49 ... 946
19.3 Tool Center Point Control; G43.4, G43.5/G49 ... 953
19.3.1 Circular Command in Tool Center Point Control (G43.4/G43.5) ... 986
19.4 Tool Cutting Point Control; G43.8/G43.9 ... 990
19.5 Inclined Surface Machining; G68.2, G68.3, G68.4/G69 ... 1010
19.5.1 How to Define Feature Coordinate System Using Euler Angles ... 1012
19.5.2 How to Define Feature Coordinate System Using Roll-Pitch-Yaw Angles ... 1014
19.5.3 How to Define Feature Coordinate System Using Three Points in a Plane ... 1016
19.5.4 How to Define Feature Coordinate System Using Two Vectors ... 1018
19.5.5 How to Define Feature Coordinate System Using Projection Angles ... 1020
19.5.6 Define by Selecting the Registered Machining Surface ... 1022
19.5.7 How to Define Feature Coordinate System Using Tool Axis Direction ... 1023
19.5.8 Tool Axis Direction Control; G53.1/G53.6 ... 1025
19.5.9 Details of Inclined Surface Machining Operation ... 1033
19.5.10 Rotary Axis Basic Position Selection ... 1038
19.5.11 Inclined Surface Machining Multi-command ... 1044
19.5.12 Relationship between Inclined Surface Machining and Other Functions ... 1047
19.5.13 Precautions for Inclined Surface Machining ... 1052
19.6 3-dimensional Tool Radius Compensation (Tool's Vertical-direction Compensation); G40/G41.2, G42.2 ... 1055
19.7 Workpiece Installation Error Compensation; G54.4 ... 1066

20 Coordinate System Setting Functions ... 1079
20.1 Coordinate Words and Control Axes ... 1080
20.2 Types of Coordinate Systems ... 1081
20.2.1 Basic Machine, Workpiece and Local Coordinate Systems ... 1081
20.2.2 Machine Zero Point and 2nd, 3rd, 4th Reference Position (Zero Point) ... 1082
20.2.3 Automatic Coordinate System Setting ... 1083
20.2.4 Coordinate System for Rotary Axis ... 1084
20.3 Basic Machine Coordinate System Selection; G53 ... 1087
20.4 Coordinate System Setting; G92 ... 1090
20.5 Local Coordinate System Setting; G52 ... 1092
20.6 Workpiece Coordinate System Selection and Extended Workpiece Coordinate System Selection; G54 to G59, G54.1 ... 1099
20.7 Workpiece Coordinate System Preset; G92.1 ... 1108
20.8 Workpiece Position Offset for Rotary Axis ; G54.2 ... 1113
20.9 3-dimensional Coordinate Conversion; G68/G69 ... 1125

20.10 Coordinate Rotation by Program; G68/G69 ... 1144
20.11 Coordinate Rotation Input by Parameter; G10 I_ J_/K_ ... 1155
20.12 Scaling; G50/G51 ... 1172
20.13 Automatic 1st Reference Position Return and Start Position Return; G28, G29 ... 1176
20.14 2nd, 3rd, 4th Reference Position Return; G30 ... 1181
20.15 Tool Change Position Return; G30.1 - G30.6 ... 1184
20.16 Reference Position Check; G27 ... 1187

21 Protection Function ... 1189
21.1 Stroke Check before Travel; G22/G23 ... 1190
21.1.1 Stroke Check before Travel in Stored Stroke Limit Area ... 1192
21.2 Enable Interfering Object Selection Data; G186 ... 1195

22 Measurement Support Functions ... 1199
22.1 Automatic Tool Length Measurement; G37 ... 1200
22.2 Skip Function; G31 ... 1204
22.3 Multi-step Skip Function 1; G31.n, G04 ... 1212
22.4 Multi-step Skip Function 2; G31 P ... 1214
22.5 Speed Change Skip; G31 Fn ... 1216
22.6 Torque Limitation Skip; G160 ... 1220
22.7 Programmable Current Limitation; G10 L14 ... 1224

23 System Variables ... 1225
23.1 System Variables List ... 1226
23.2 System Variables (G Command Modal) ... 1228
23.3 System Variables (Non-G Command Modal) ... 1229
23.4 System Variables (Modal Information at Macro Interruption) ... 1230
23.5 System Variables (Tool Information) ... 1232
23.6 System Variables (Tool Compensation) ... 1240
23.7 System Variables (Tool Life Management) ... 1241
23.8 System Variables (Workpiece Coordinate Offset) ... 1246
23.9 System Variables (Workpiece Position Offset Amount for Rotary Axis) ... 1247
23.10 System Variables (Extended Workpiece Coordinate Offset) ... 1248
23.11 System Variables (Position Information by Part System) ... 1250
23.12 System Variables (Position Information Common to Part System) ... 1255
23.13 System Variables (Alarm) ... 1257
23.14 System Variables (Message Display and Stop) ... 1258
23.15 System Variables (Cumulative Time) ... 1259
23.16 System Variables (Time Read Variables) ... 1260
23.17 System Variables (Machining Information) ... 1262
23.18 System Variables (Reverse Run Information) ... 1263
23.19 System Variables (Number of Workpiece Machining Times) ... 1263
23.20 System Variables (Mirror Image) ... 1263
23.21 System Variables (Acquiring the Program Execution Part System No.) ... 1264
23.22 System Variables (Display Axis Name Selection) ... 1265
23.23 System Variables (Coordinate Rotation Parameter) ... 1266
23.24 System Variables (Rotary Axis Configuration Parameter) ... 1267
23.25 System Variables (Normal Line Control Parameter) ... 1268
23.26 System Variables (Parameter Reading) ... 1269
23.27 System Variables (Workpiece Installation Error Compensation Amount) ... 1273
23.28 System Variables (Macro Interface Input (PLC -> NC)) ... 1274
23.29 System Variables (Macro Interface Output (NC -> PLC)) ... 1280
23.30 System Variables (R Device Access Variables) ... 1286
23.31 System Variables (PLC Data Reading) ... 1292
23.32 System Variables (Interfering Object Selection) ... 1295
23.33 System Variables (NC Data Reading/Writing with API Section and Sub-section Nos. Input/Output by Program) ... 1298
23.34 System Variables (Extended Common Variables) ... 1301
23.35 System Variables (Axis Exchange Information Acquisition) ... 1302
23.36 System Variables (Reference Axis Arrangement Information Acquisition) ... 1305

24 Appx.1: Fixed Cycles ... 1309

25 Appx.2: Command Value Range Lists ... 1315

1

Control Axes

1.1 Coordinate Words and Control Axes

Function and purpose

The NC axis, spindle, PLC axis and auxiliary axis are generically called the control axis.

NC axis	The axis is used to perform automatic operation by coordinate value command from machining program and to perform manual operation such as jog feed, handle feed etc. X, Y, Z, U, V, W, A, B and C can be used as the axis name. An axis where no motor is connected (hypothetical axis) can be set.
Spindle	The axis is used to rotate the part where the tool/workpiece is attached. The rotation is commanded by rotation speed and spindle forward run signal/spindle reverse run signal.
PLC axis	The axis is used to command the position and speed from user PLC. Two or more axes are not used together for interpolation; instead, each axis is used individually. Manual operation such as jog feed, handle feed, etc. can be performed. An axis where no motor is connected (hypothetical axis) can be set.

The following functions can be added depending on the parameter setting.

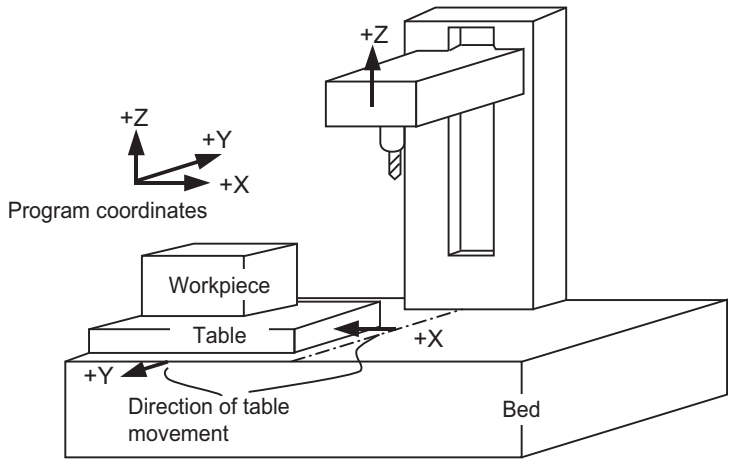
- Spindle position control (Spindle/C axis control)
  A spindle can be controlled as the rotary axis. A coordinate value (angle) can be commanded from machining program in the same way as NC axis.
- Spindle-mode rotary axis control
  NC axis can be controlled by the command of rotation speed and spindle forward run signal/spindle reverse run signal in the same way as spindle.

Note

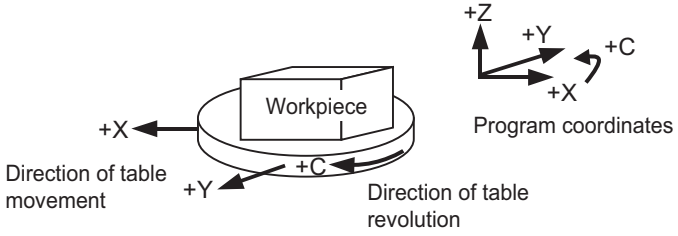
(1) The number of control axes that can be controlled by one CNC depends on the model.

To specify each machining direction, use alphabetical coordinate words that are predefined appropriately.

X-Y table



X axis, Y axis and rotating table



1.2 Coordinate Systems and Coordinate Zero Point Symbols



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

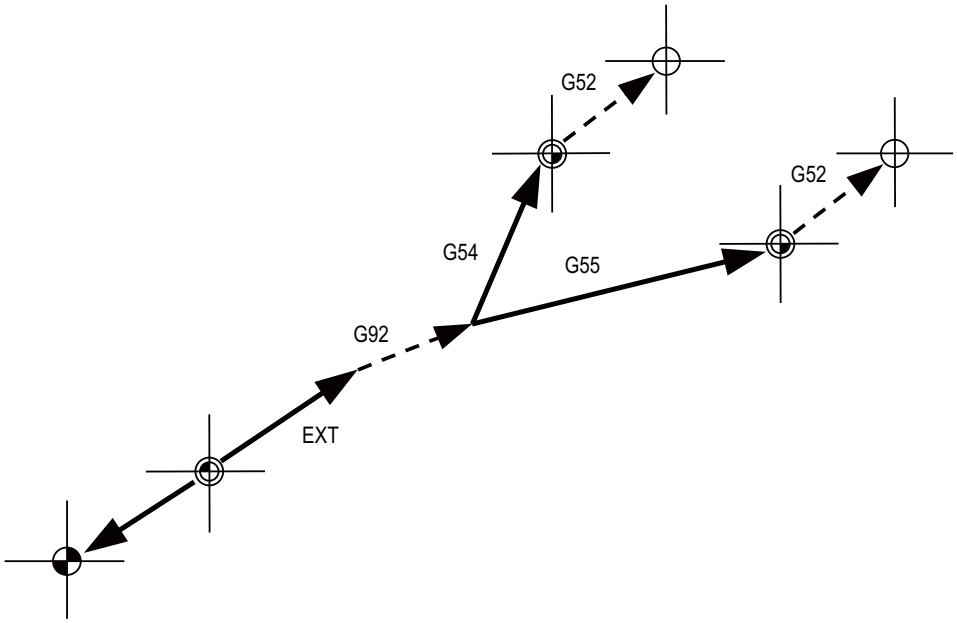


Basic machine coordinate zero point:
A position specific to machine



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 G54 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.

2

Minimum Command Unit

2.1 Input Setting Unit and Program Command 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 (Input 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 10μm input command increment with a CNC unit for which the command increment is set to 1μm 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 1μm input command increment with a CNC unit for which the command increment is set to 0.1μm and this function's parameter value is set to "10", this function enables the same machining as the original program.

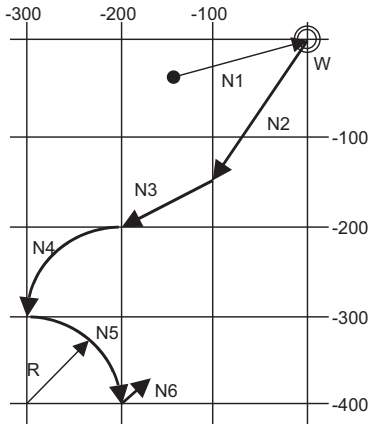
(3) This function cannot be used for the dwell function G04_X_(P_);.

(4) This function cannot be used for the compensation amount of the tool compensation input.

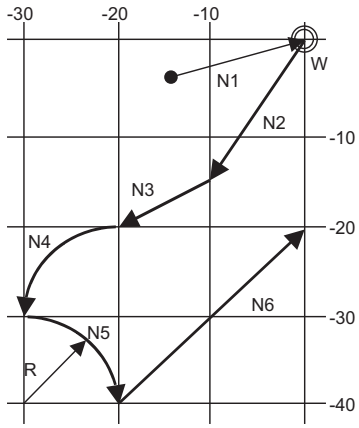
(5) This function can be used when decimal point type I is valid, but cannot be used when decimal point type II is valid.

(6) This function cannot be used for a tool shape setting command (in G10L100 format).

Program example (Machining program : programmed with 1=10μm) (CNC unit is 1=1μm 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.

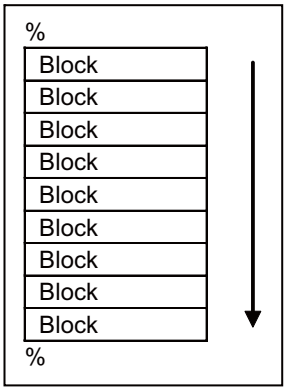
(3) When the indexing increment is set to 2 degrees, the rotary axis is set to the B axis, and the B axis is moved with JOG to the 1.234 position, an indexing error will occur if "G90B5." or "G91B2." is commanded.

3

Program Formats

3.1 Program Format

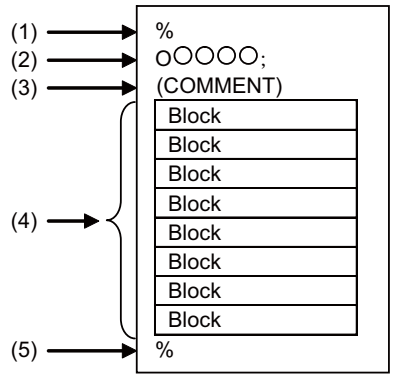
A collection of commands assigned to an NC to move a machine is called "program".
A program is a collection of units called "block" which specifies a sequence of machine tool operations.
Blocks are written in the order of the actual movement of a tool.
A block is a collection of units called "word" which 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

The program can be easier to read by describing the machining program title, note for the process, etc. as a comment.
The range from control out "(" to the next control in ")" is treated as a comment.
When control in ")" is omitted, the range till the end of the block is treated as a comment.

(4) Program section

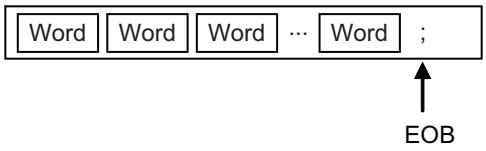
A program is a collection of several blocks.

(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.

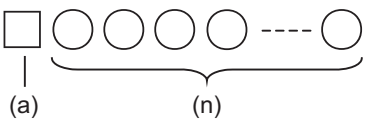
Block and word

[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]



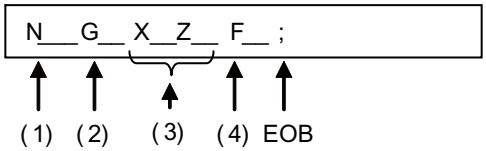
(a) Alphabet (address)
(n) Number

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 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.

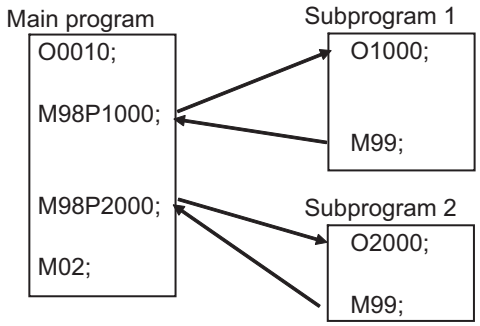
(3) Coordinate words

"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 commands" or "absolute 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	M800VW/M80VW	M800VS/M80V
NC memory	○	○
NC memory 2 (*1)	○	○
Serial	○	○
SD card in control unit	○	-
Front-side SD card	○	○
Ethernet	○	○
Display unit-side data server (HD/DS)	○	○
Front-side USB memory	○	○
Industrial PC (IPC)	-	○ (*2)

(*1) Only user programs can be stored in NC memory 2. MDI programs, fixed cycle programs, and machine tool builder macros cannot be stored.

(*2) This function is valid only for M80V Series. The availability of the function depends on the MTB specifications. (parameter "#1760 cfgPR10/bit2" (Enable HD mode on IPC))

Note

Note the following points to use NC memory 2.

(1) Since processing time is required for the command part which changes the program flow such as subprogram call, GOTO, and DO-END, the axis movement may slow down or stop.

(2) It requires time to search the sequence number and block number of a large-capacity program.

Program file format

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

(1) NC memory, NC memory 2 (Creates program on NC)

```
(COMMENT) ;
G28 XYZ ;
  :
  :
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.

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

[Single program]

```
CRLF
(COMMENT) CRLF
G28 XYZ CRLF
  :
  :
M02 CRLF
% ^Z
```

[Multiple programs]

```
CRLF
O100(COMMENT) CRLF
G28 XYZ CRLF
  :
  :
M02 CRLF
O101(COMMENT1) CRLF
  :
M02 CRLF
% ^Z
```

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 transfered 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)

```
% LF
O100(COMMENT) LF
G28 XYZ LF
 :
 :
M02 LF
%
```

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 transfered 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.

Character code of machining program

Chinese characters or the characters with umlaut symbol such as "ü" can be used in the messages of system variables #3000 and #3006. Since it cannot be input on the NC edit screen, create it with a general text editor.
Use the character code corresponding to the display language (parameter "#1043 lang").
Messages with character codes which do not correspond to the display language cause character corruption.

#1043 lang		Character code
0	English	Windows-1252
1	Japanese	Shift-JIS
11	German	Windows-1252
12	French	Windows-1252
13	Italian	Windows-1252
14	Spanish	Windows-1252
15	Chinese (traditional)	Big5
16	Korean (Hangeul)	KS C 5601-1987
17	Portugese	Windows-1252
18	Dutch	Windows-1252
19	Swedish	Windows-1252
20	Hungarian	Windows-1250
21	Polish	Windows-1250
22	Chinese (simplified)	GB2312
23	Russian	Windows-1251
24	Turkish	CP1254
25	Czech	Windows-1250
31	Indonesian	Windows-1252

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.

(6) When the line (block) with "/" code at its head is skipped by optional block skip function while any of the following functions is enabled, the block is considered as the block without movement.

- High-speed machining mode II
- High-speed high-accuracy control II
- High-speed high-accuracy control III

However, when the parameter "#1711 cfg11 /bit4" is set to "1", the block is skipped and is not considered as the block without movement. In the mode other than the above, the line (block) is skipped by optional block skip function and is not considered as the block without movement.

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

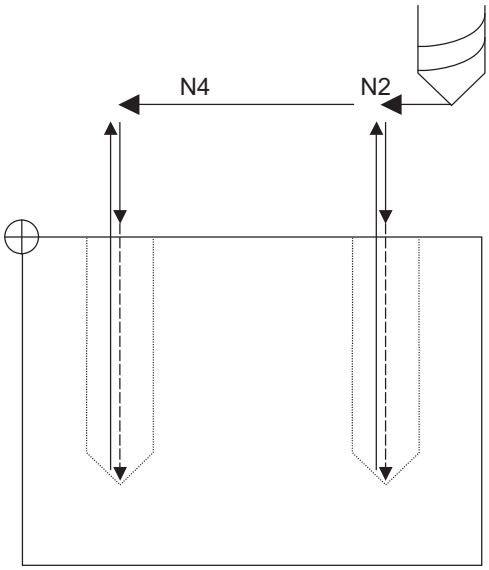
The block with "/n" (slash) code is skipped when the "/n" is programmed to the head of the block and the optional block skip n signal is turned ON. For a block with the "/n" code inside the block (not at the head of the block), the program is operated according to the value of the parameter "#1226 aux10/bit1"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

Program example

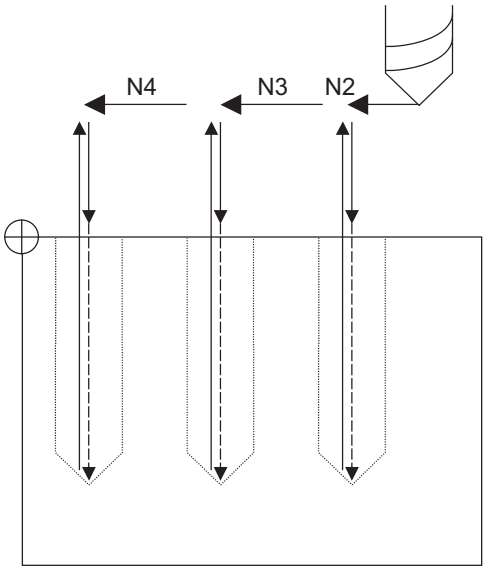
(1) When the 2 parts like the figure below are machined, the following program is used. When the optional block skip 5 signal is ON, the part 1 is created. When the optional block skip 5 signal is OFF, the part 2 is created.

```
     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 ;
/1/2    N02 G00 X50. ;
/1/2    N03 G01 Z-20. F100 ;
/1/2    N04 G00 Z3. ;
/1  /3  N05 G00 X30. ;
/1  /3  N06 G01 Z-20. F100 ;
/1  /3  N07 G00 Z3. ;
  /2/3  N08 G00 X10. ;
  /2/3  N09 G01 Z-20. F100 ;
  /2/3  N10 G00 Z3. ;
        N11 G28 X0 M05 ;
        N12 M02 ;
```

(a) Optional block skip 1 signal ON
(Optional block skip 2.3 signal OFF)
N01 -> N08 -> N09 -> N10 -> N11 -> N12

(b) Optional block skip 2 signal ON
(Optional block skip 1.3 signal OFF)
N01 -> N05 -> N06 -> N07 -> N11 -> N12

(c) Optional block skip 3 signal ON
(Optional block skip 1.2 signal OFF)
N01 -> N02 -> N03 -> N04 -> N11 -> N12

(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.;
N02 G01 F1000;
N03 X1. /1 Y1. /2 Z1.;
N04 M30;
```

N03 block will operate as follows.
(a) Optional block skip 1 signal ON
Optional block skip 2 signal OFF
"Y1. Z1." is ignored.
(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.

An unmodal command is a G code whose G code group is 00.
A modal command is a G code whose G code group is other than 00.

3.4.2 G Code Lists

G code	Group	Function	Section
Δ G00	01	Positioning	6.1
G0.5	28	Rapid traverse block overlap	7.14.1
Δ G01	01	Linear interpolation	6.2
G02	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
G03	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
G02.1	01	Spiral/Conical interpolation CW (type1)	6.14
G03.1	01	Spiral/Conical interpolation CCW (type1)	6.14
G02.2	01	Involute interpolation/Helical involute interpolation CW	6.18
G03.2	01	Involute interpolation/Helical involute interpolation CCW	6.18
G02.3	01	Exponential function interpolation positive rotation	6.12
G03.3	01	Exponential function interpolation negative rotation	6.12
G02.4	01	3-dimensional circular interpolation CW	6.15
G03.4	01	3-dimensional circular interpolation CCW	6.15
G04	00	Dwell	8.1 8.2
G05	00	High-speed machining mode	17.1
		High-speed high-accuracy control II/III	17.3
G05.1	00	High-speed high-accuracy control I	17.3
		Spline interpolation	17.4
G06.2	01	NURBS interpolation	6.16
G07	00	Hypothetical axis interpolation	6.17
G07.1 G107	21	Cylindrical interpolation	6.9
G08	00	High-accuracy control	17.2
G08.5	00	Vibration cutting control (VCC)	15.13
G08.6	00	Orbit machining	15.17
G09	00	Exact stop check	7.11

G code	Group	Function	Section
G10	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	20.11
G10.9	00	Diameter/Radius designation switch	5.2.1
G11	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
G12	00	Circular cutting CW	6.10
G13	00	Circular cutting CCW	6.10
G12.1 G112	21	Polar coordinate interpolation ON	6.11
* G13.1 G113	21	Polar coordinate interpolation cancel	6.11
G14			
* G15	18	Polar coordinate command OFF	6.13
G16	18	Polar coordinate command ON	6.13
Δ G17	02	X-Y plane selection	6.5
Δ G18	02	Z-X plane selection	6.5
Δ G19	02	Y-Z plane selection	6.5
Δ G20	06	Inch command	5.3
Δ G21	06	Metric command	5.3
G22	04	Stroke check before travel ON	21.1
G23	04	Stroke check before travel cancel	21.1
G24			
G25			
G27	00	Reference position check	20.16
G28	00	Reference position return	20.13
G29	00	Start position return	20.13
G30	00	2nd to 4th reference position return	20.14
G30.1	00	Tool change position return 1	20.15
G30.2	00	Tool change position return 2	20.15
G30.3	00	Tool change position return 3	20.15
G30.4	00	Tool change position return 4	20.15
G30.5	00	Tool change position return 5	20.15
G30.6	00	Tool change position return 6	20.15
G31	00	Skip/Speed change skip	22.2
		Multi-step skip 2	22.4
G31.1	00	Multi-step skip 1-1	22.3
G31.2	00	Multi-step skip 1-2	22.3
G31.3	00	Multi-step skip 1-3	22.3
G32			
G33	01	Thread cutting	6.6
G34	00	Special fixed cycle (bolt hole circle)	13.2.1
G35	00	Special fixed cycle (line at angle)	13.2.2
G36	00	Special fixed cycle (arc)	13.2.3
G37	00	Automatic tool length measurement	22.1

G code	Group	Function	Section
G37.1	00	Special fixed cycle (grid)	13.2.4
G38	00	Tool radius compensation vector designation	12.3
G39	00	Tool radius compensation corner arc	12.3
* G40	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	19.6
G41	07	Tool radius compensation (tool nose radius compensation) left	12.3
		3-dimensional tool radius compensation left	12.5
G42	07	Tool radius compensation (tool nose radius compensation) right	12.3
		3-dimensional tool radius compensation right	12.5
* G40.1 G150	15	Normal line control cancel	15.7
G41.1 G151	15	Normal line control left ON	15.7
G42.1 G152	15	Normal line control right ON	15.7
G41.2	07	3-dimensional tool radius compensation (Tool's vertical-direction compensation) (left)	19.6
G42.2	07	3-dimensional tool radius compensation (Tool's vertical-direction compensation) (right)	19.6
G43	08	Tool length compensation (+)	12.2
G44	08	Tool length compensation (-)	12.2
G43.1	08	Tool length compensation along the tool axis ON	19.2
G43.4	08	Tool center point control type1 ON	19.3
G43.5	08	Tool center point control type2 ON	19.3
G43.7	08	Tool position compensation start	19.1
G43.8	08	Tool cutting point control type 1	19.4
G43.9	08	Tool cutting point control type 2	19.4
G45	00	Tool position offset (extension)	12.6
G46	00	Tool position offset (reduction)	12.6
G47	00	Tool position offset (double elongation)	12.6
G48	00	Tool position offset (double contraction)	12.6
* G49	08	Tool length compensation cancel	12.2
		Tool length compensation along the tool axis cancel	19.2
		Tool center point control cancel	19.3
		Tool position compensation cancel	19.1
* G50	11	Scaling cancel	20.12
G51	11	Scaling ON	20.12
* G50.1	19	Mirror image by G code cancel	15.6
G51.1	19	Mirror image by G code ON	15.6
G50.2 G250	00	Tool spindle synchronization IB/IC cancel	18.2.2 18.2.3
G51.2 G251	00	Tool spindle synchronization IB/IC	18.2.2 18.2.3
G52	00	Local coordinate system setting	20.5
G53	00	Basic machine coordinate system selection	20.3
G53.1	00	Tool axis direction control (type 1)	19.5.8
G53.6	00	Tool axis direction control (type 2)	19.5.8

G code	Group	Function	Section
* G54	12	Workpiece coordinate system 1 selection	20.6
G55	12	Workpiece coordinate system 2 selection	20.6
G56	12	Workpiece coordinate system 3 selection	20.6
G57	12	Workpiece coordinate system 4 selection	20.6
G58	12	Workpiece coordinate system 5 selection	20.6
G59	12	Workpiece coordinate system 6 selection	20.6
G54.1	12	Extended workpiece coordinate system selection	20.6
G54.2	23	Workpiece position offset for rotary axis	20.8
G54.4	27	Workpiece installation error compensation	-
G60	00(01)	Unidirectional positioning	6.8.1
G61	13	Exact stop check mode	7.12
G61.1	13	High-accuracy control ON	17.2
G61.2	13	High-accuracy spline	17.6
G61.4	13	Spline interpolation 2	17.5
G62	13	Automatic corner override	7.15.1
G63	13	Tapping mode	7.16
* G64	13	Cutting mode	7.17
G65	00	User macro simple call	14.4.1
G66	14	User macro modal call A	14.4.2
G66.1	14	User macro modal call B	14.4.3
* G67	14	User macro modal call cancel	14.3
G68	16	Coordinate rotation by program ON	20.10
		3-dimensional coordinate conversion mode ON	20.9
G68.2	16	Inclined surface machining command	19.5
		R-Navi data input (Selecting the registered machining surface)	15.9.6
G68.3	16	Inclined surface machining command	19.5
G68.4	16	Inclined surface machining incremental multiple commands	19.5.11
* G69	16	Coordinate rotation by program cancel	20.10
		3-dimensional coordinate conversion mode OFF	20.9
		Inclined surface machining cancel	19.5
		R-Navi data input (Canceling the selected machining surface)	15.9.6
G70	09	User fixed cycle	
G71	09	User fixed cycle	
G72	09	User fixed cycle	
G73	09	Fixed cycle (step)	13.1.10
G74	09	Fixed cycle (reverse tap)	13.1.11
G74.5 G74.6 G74.8	09	Reverse Punchtap cycle	13.1.15
G75	09	Fixed cycle (circle cutting cycle)	13.1.12
G76	09	Fixed cycle (Fine boring)	13.1.13
G77	09	User fixed cycle	
G78	09	User fixed cycle	
G79	09	User fixed cycle	
* G80	09	Fixed cycle cancel	13.1
G81	09	Fixed cycle (drill/spot drill)	13.1.1
G82	09	Fixed cycle (drill/counter boring)	13.1.2
G83	09	Fixed cycle (deep-hole drilling/small-diameter deep-hole drilling)	13.1.3

G code	Group	Function	Section
G84	09	Fixed cycle (tapping)	13.1.4
G84.5 G84.6 G84.8	09	Punchtap cycle	13.1.15
G85	09	Fixed cycle (boring)	13.1.5
G86	09	Fixed cycle (boring)	13.1.6
G87	09	Fixed cycle (back boring)	13.1.7
G88	09	Fixed cycle (boring)	13.1.8
G89	09	Fixed cycle (boring)	13.1.9
Δ G90	03	Absolute command	5.1
Δ G91	03	Incremental command	5.1
G92	00	Coordinate system setting Spindle clamp speed setting	20.4 10.3
G92.1	00	Workpiece coordinate system preset	20.7
G93	05	Inverse time feed	7.5
Δ G94	05	Feed per minute (asynchronous feed)	7.4
Δ G95	05	Feed per revolution (synchronous feed)	7.4
Δ G96	17	Constant surface speed control ON	10.2
Δ G97	17	Constant surface speed control OFF	10.2
* G98	10	Fixed cycle Initial level return	13.1.17
G99	10	Fixed cycle (R point level return)	13.1.17
G100-G225	00	User macro (G code call) Max. 10	14.4.4
G113.1	00	Spindle synchronization I/Tool spindle synchronization (IA/IB/II) cancel	18.1.1
G114.1	00	Spindle synchronization I	18.1.1
G114.2	00	Tool spindle synchronization IA	18.2.1
G114.3	00	Tool spindle synchronization II	18.2.1
G120.1	00	Machining condition selection I	17.7
G121	00	Machining condition selection I cancel	17.7
G122	00	Activate sub part system I	16.4.1
G125	00	Control axis synchronization between part systems	16.3
G127	00	Prohibits the arbitrary reverse run in all part systems	15.8
G130	00	Selection of axis (axes) for feedrate command	7.7
G135	00	Text engraving cycle	15.15
G136	00	Two-dimensional barcode engraving cycle	15.14
G140	00	Arbitrary axis exchange command	16.2.1
G141	00	Arbitrary axis exchange return command	16.2.1
G142	00	Arbitrary axis exchange/reference axis arrange return command	16.2.1
G145	00	Cancel sub part systems	16.4
G160	00	Torque limitation skip	22.6
G162	00	Spindle speed fluctuation detection	10.5
G162.1	00	Cutting load control	15.16
G163	00	Spindle speed fluctuation detection cancel	10.5
G180	00	Interactive cycle insertion program	15.11
G186	00	Interference check III interfering object data enable command	21.2
G187	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.**

4

Pre-read Buffer

4.1 Pre-read Buffer



Function and purpose

During automatic processing, 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 program 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 control operation will not be effective for the program pre-read with the buffer register.

5

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

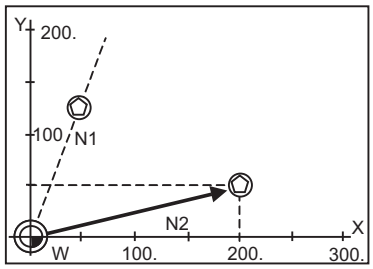
Detailed description

(1) Regardless of the current position, in the absolute 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 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 position.

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 mode or the incremental mode.

Tool

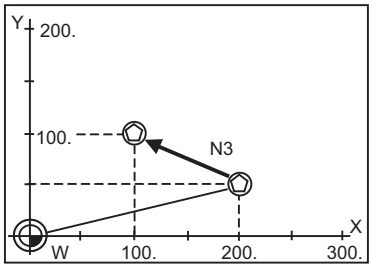
(2) For the next block, the last G90/G91 command that was given becomes the modal.

(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. ;

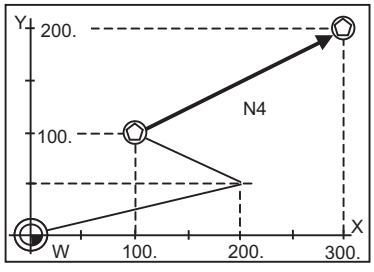
The X axis moves to -100.mm and the Y axis to +50.0mm as an incremental position, and as a result X moves to 100.mm and Y to 100.mm.



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

N4 G90 X300. G91 Y100. ;

The X axis is treated in the absolute 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 mode.



(4) When the power is turned ON, it is possible to select whether you want absolute commands or incremental 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 name 1__ Axis name 2__ ... Axis name n__ ;

Axis name n	Axis name for which diameter/radius designation is switched. Select radius or diameter designation with a value that follows the axis name. 0: Radius designation 1: Diameter designation If you do not command any axis name, 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 name, specify the axis name that is set in "#1013 axname" or "#1014 incax" (*1). If the specified axis name is not found, a program error (P33) will occur.

(*1) The axis name set in "#1014 incax" can be specified only when the setting value of "#1076 AbsInc" is "1". These settings depend on the MTB specifications.

(5) If a rotary axis is specified with "Axis name", a program error (P32) occurs.

(6) If a number with a decimal point is specified with "Axis name", 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 the Reset or Emergency stop, the diameter/radius designation before G10.9 is restored.
However, when "#1255 set27/bit3" is set to "1", the diameter/radius designation status is held even if resetting is performed. (This parameter setting depends on the MTB specifications.)

(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.

(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) Tool center point control
If tool center point control is given to an axis for which diameter/radius designation is being switched by G10.9, a program error (P941) occurs. If G10.9 is given to an axis that is in tool center point control, the program error (P705) occurs.

(12) Graphic check (2D/3D), Graphic trace
During graphic check or graphic trace, even if the diameter and radius are switched with G10.9, neither the drawing image nor coordinate value view is not switched.



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.

(8) Do not perform manual tool length measurement I for an axis for which the diameter/radius designation is switched. The correct tool length cannot be measured for an axis for which the diameter/radius designation is switched.

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

When G20 and G21 are switched, the command unit of the linear axis is switched.
The command unit of linear axis is inch system during G20 modal, and metric system during G21 modal.
The command unit of rotary axis is "degree" regardless of G20 or G21 modal.

(1) The setting and display unit depend on the setting of the parameter "#1226 aux10/bit6".

[#1226 aux10/bit6 = 0] (When the setting of "#1041 I_inch" is followed)

The display unit of the counter is determined by the unit system in the parameter "#1041 I_inch".

#1041 = 0: A value converted from inches to millimeters is displayed in G20 modal.

#1041 = 1: A value converted from millimeters to inches is displayed in G21 modal.

[#1226 aux10/bit6 = 1] (When the command unit (G20/G21 modal) is followed)

The counter and the parameter are determined by G20/G21 modal.

G21 command mode: Metric unit

G20 command mode: Inch unit

(2) The setting and display unit of user parameters follow the unit system of the parameter "#1041 I_inch".

(3) The setting and display unit of machine parameters follow the unit system of the parameter "#1040 M_inch".

(4) The command unit for when the power is turned ON and reset is performed is decided by combining the parameters "#1041 I_inch", "#1151 rstint", "#1152 I_G20" and "#1210 RstGmd/bit5".

Related parameters

The following parameters are related to inch/metric changeover, and depend on the MTB specifications.

(1) Initial inch/millimeter selection (#1041 I_inch)

The following unit systems are affected.

- Command unit for when the power is turned ON and reset is performed (inch/metric command mode)
  When NC is reset in reset modal retention (#1151 rstint = 0) or in reset modal retention (#1210 RstGmd/bit5 = 1) for G code of G code group 06, the unit follows the G20/G21 command modal.
- Unit system of position display (counter/user parameter/tool/workpiece offset)
- Input/output unit of user parameters
- User parameter unit related to length and speed
- Machine parameter "#1084 RadErr" (Arc error)

When the parameter is switched, the power needs to be turned ON again.

(2) Setting/display unit selection (#1226 aux10/bit6)

(a) The setting and display unit of counter and parameter (metric system/inch system) can be selected between the command unit determined by G20/G21 command and the unit determined by the parameter "#1041 I_inch".
When this parameter is set to "1" (command unit), the following data unit is switched to inch or millimeter depending on G20 or G21 command.

- Position display counter
- User parameter setting/display
- Feedrate display
- Workpiece/tool offset setting/display
- Handle feed
- Incremental feed

(b) When the parameter "#1226 aux10/bit6" is "1", the unit is converted and displayed at the moment when the inch and metric commands are switched.

(c) The variables of the machining program (system variables of coordinate system) are also switched at the same time.

(d) When the parameter "#1226 aux10/bit6" is "1" and multi-part system is used, the unit follows the command unit of each part system.

(e) This parameter is enabled immediately after it is set.

(f) This parameter is not related to PLC axis. Both display unit and command unit of PLC axis are determined by the setting of the parameter "#1042 pcinch".

(g) This parameter is enabled only when "#1041 I_inch" is "0" (Metric system). When "#1041 I_inch" is "1" (Inch system), units of all other than movement/speed command are always inch regardless of this parameter.

(3) Initial command unit (#1152 I_G20)

This specifies the inch/metric system mode (G20/G21) for when the power is turned ON and reset is performed.
This parameter is enabled by reset after it is set.

(4) Changing mode by G20/G21 not allowed (#1253 set25/bit1)

When the parameter "#1253 set25/bit1" is "1", if G21 (metric command) is issued during inch modal, the program error (P34) occurs. The same also applies to when G20 (inch command) is issued during metric modal.

This parameter is enabled by power ON after it is switched.
This parameter is enabled for MTB macro and user macro.

Output unit, command unit and setting unit

The unit is as follows depending on each parameter setting. (This depends on the MTB specifications.)

Item	#1041 I_inch = 0				#1041 I_inch = 1			
	#1226/bit6 = 0		#1226/bit6 = 1		#1226/bit6 = 0		#1226/bit6 = 1	
	G21	G20	G21	G20	G21	G20	G21	G20
Movement/speed command	Metric	Inch	Metric	Inch	Metric	Inch	Metric	Inch
Counter display	Metric		Metric	Inch	Inch		Inch	
Speed display	Metric		Metric	Inch	Inch		Inch	
User parameter setting/ display	Metric		Metric	Inch	Inch		Inch	
Workpiece/tool offset setting/display	Metric		Metric	Inch	Inch		Inch	
Handle feed command	Metric		Metric	Inch	Inch		Inch	

For rotary axis, the unit is "degree" regardless of the parameter setting.

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.
Th machine parameters are input/output with the unit set by the parameter "#1040 M_inch".
If "#1041 I_inch" and "#1040 M_inch" are not found in the input data, the unit will follow the unit currently set.

(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) The program error (P33) occurs 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)

(4) When the setting and display unit is inch, do not set addition of tool and workpiece offset data. The addition setting may cause an error.

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.
For decimal point valid addresses, commanded values without a decimal point can also be handled as a value with a decimal point. This depends on the MTB settings. (Parameter "#1078 Decpt2")



Detailed description

(1) The decimal point command is valid for the distances, angles, times, and speeds in machining programs.

(2) In decimal point input type I and decimal point input type II, the values of the data commands without the decimal points are shown in the table below.
(When the parameter "#1078 Decpt2" is set to "0", decimal point input type I is valid. When parameter "#1078 Decpt2" is set to "1", decimal point input type II is valid.)

Command	Parameter #1015 cunit (command increment)	Type I [μm, 10^{-4} inch, 10^{-3} deg]	Type II [mm, inch, deg]
X1 ;	10000	1000	1
	1000	100	1
	100	10	1
	10	1	1

(3) Refer to the table "Addresses used and validity of decimal point commands" for details on the addresses where the decimal point commands are enabled.

(4) In decimal point command, the valid range of command value is as shown below. (When "#1015 cunit" (program input command) is "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 10000000.000	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) Decimal point commands for decimal point invalid addresses are processed as an integer only data which everything after the decimal point is ignored. Decimal point invalid addresses include the following contents;

(D, H, L, M, N, O, S, T)

However, commands that use arithmetic formulas or variables are treated as data with decimal points.

(7) Command increment tenfold is applied when decimal point input type I is valid, however, it is not applied when decimal point input type II is valid.

Decimal point input type I and II, and decimal point command validity

Decimal point input type I and II will result as follows when decimal points are not used at an address where a decimal point command is valid.
Both decimal point input type I and II will produce the same result when a command uses a decimal point.

(1) Decimal point input type I

The unit of the commanded value varies depending on decimal point validity.
When a value without a decimal point is commanded at an address where a decimal point command is valid, the least significant digit is determined by the following parameters.

- Axis addresses: Parameter "#1015 cunit"
- Addresses related to distances and angles other than axis addresses: Parameter "#1003 iunit"

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

(2) Decimal point input type II

A value commanded without a decimal point is handled as a value with a decimal point.

(Example) When "X1" is commanded, the same result occurs as for an "X1." command.

The validity of the decimal point command for each address varies depending on the application as shown in the table below.

The validity of the decimal point command can be determined by an address regardless of the application.
For details, refer to "5.4.1 Decimal Point Command Valid/Invalid Address Changeover".

[Addresses used and 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	
	Invalid	Interactive cycle insertion: Cycle ID	
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)	
	Invalid	Tool shape input by program: Tool color	
	Invalid	R-Navi data input by program: Basic coordinate system	
	Invalid	R-Navi data input by program: Coordinate axis direction setting	

Address	Decimal point command	Usage	Remarks
D	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 synchronized 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
	Invalid	Tool spindle synchronization IA: Workpiece axis selection (Synchronized spindle)	
	Invalid	Tool spindle synchronization IB: Rotary tool axis selection (Synchronized spindle)	
	Invalid	Tool spindle synchronization II: Hob axis selection	
	Invalid	Designation of the spindle number for chip removal operation	
	Invalid	Skip signal command method	
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)	
	Invalid	Tool spindle synchronization IA: Rotary tool axis rotation ratio designation	
	Invalid	Tool spindle synchronization II: Rotation ratio designation (Hob axis)	
	Invalid	Designation of frequency in which chip removal is made	
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	Tool spindle synchronization IA: Rotary tool axis selection (Reference spindle)	
	Invalid	Tool spindle synchronization IB: Workpiece axis selection (Reference spindle)	
	Invalid	Tool spindle synchronization II: Hob axis 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.	

Address	Decimal point command	Usage	Remarks
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)	
	Valid	Cutting reduction amount (incremental position)	
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)	
	Valid	Minimum cutting amount (incremental position)	,K

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	
	Invalid	Tool spindle synchronization IA: Rotation ratio designation (Workpiece axis)	
	Invalid	Tool spindle synchronization II: Rotation ratio designation (Workpiece axis)	
	Invalid	Lead axis number	
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.	

Address	Decimal point command	Usage	Remarks
P	Invalid/Valid	Dwell time	Parameter
	Invalid	Subprogram program call: program No.	
	Invalid/Valid	Dwell at tap cycle hole base	Parameter
	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	Skip 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	
	Invalid	Tool center point control: G00 temporary cancel designation	
	Invalid	Tool spindle synchronization IB: Rotation ratio designation (Workpiece axis)	
	Invalid	Tool spindle synchronization IC: Rotation ratio designation (Spindle)	
	Valid	Tool spindle synchronization II: Gear torsion angle designation	
Q	Valid	Fixed cycle for drilling: Cut amount for each cutting pass or shift amount	
	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	Tool spindle synchronization IB/IC: Rotation ratio designation (Rotary tool axis)	
	Valid	Tool spindle synchronization II: Module or diametral pitch designation	
	Invalid	Orbit machining: Internal diameter machining/outer diameter machining/Q0 machining mode cancel	

Address	Decimal point command	Usage	Remarks
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 position)	
	Valid	Phase shift amount of synchronized spindle	
	Valid	Tool spindle synchronization II: Phase shift amount (Workpiece axis)	
	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	
	Valid	Orbit machining: Tool nose angle	
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
	Invalid	Synchronous tap: Designation of spindle rotation speed at the return	,S
	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	
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 input type I		Decimal point input type II
	1 = 1 μm	1 = 10 μm	1 = 1 mm
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 μm unit)	X123.450 mm	X12345.000 mm
#111 = 123 #112 = 5.55 X#111 Z#112	X123.000 mm Z5.550 mm	X123.000 mm Z5.550 mm	X123.000 mm Z5.550 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 123 mm command. It will not be 123 μm.

5.4.1 Decimal Point Command Valid/Invalid Address Changeover



Function and purpose

The validity of the decimal point command can be determined by an address regardless of the application.
The validity of this function depends on the MTB specifications (parameter "#1274 ext10/bit0").



Detailed description

When the parameter "#1274 ext10/bit0" is "1", whether the decimal point command is enabled or disabled is determined according to the address as shown in the following table.

Address where the decimal point command is enabled	X, Y, Z, U, V, W, A, B, C, E, F, G, I, J, K, Q, R
Address where the decimal point command is disabled	D, H, L, M, N, O, P, S, T

If a decimal point is designated for an address where the decimal point command is disabled, the program error (P530) occurs.
However, a command that uses arithmetic formulas or variables is treated as a command value without decimal point where the first decimal point is rounded off.

(Example) M[1+1.5]; (It is operated as "M3".)



Precautions

(1) When the address where the decimal point command is disabled is used in the axis address name, the decimal point command is disabled.
(2) The decimal point command is disabled for the address "P" of dwell command (G04) even when the parameter "#8112 DECIMAL PNT-P" is "1".
(3) The decimal point command is enabled for the argument of user macro call even when the address is the one where the decimal point command is disabled.
(4) In high-speed high-accuracy control II mode, whether the decimal point command is enabled or disabled depends on the function used even when the parameter "#1274 ext10/bit0" is "1".

6

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 "7.13 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").

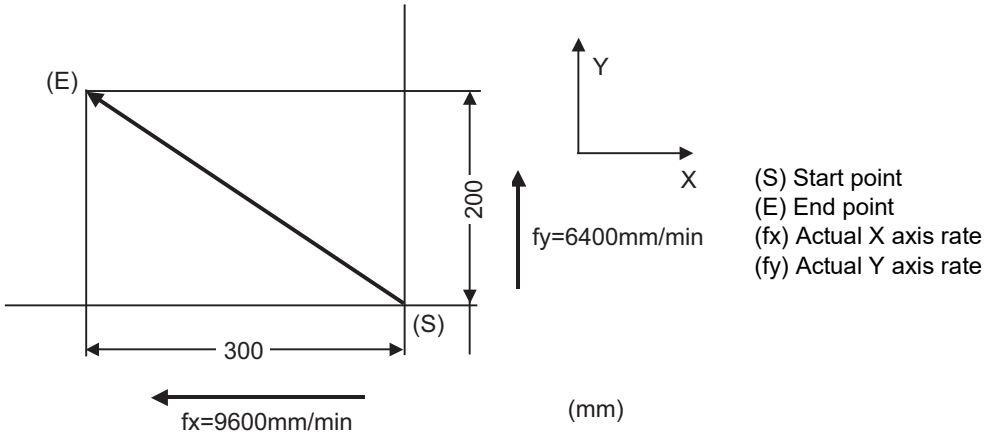
The positioning time does not change according to the path.

(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)

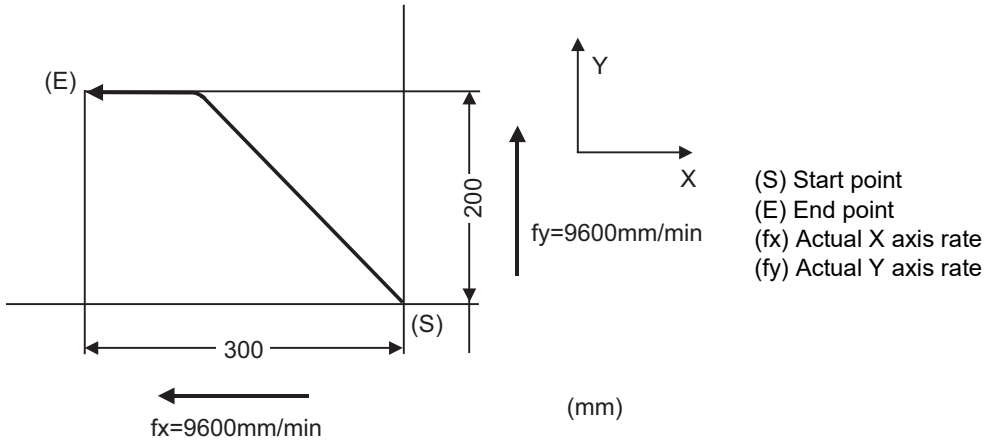


(2) Non-linear path (When parameter "#1086 G0Intp" is set to "1")

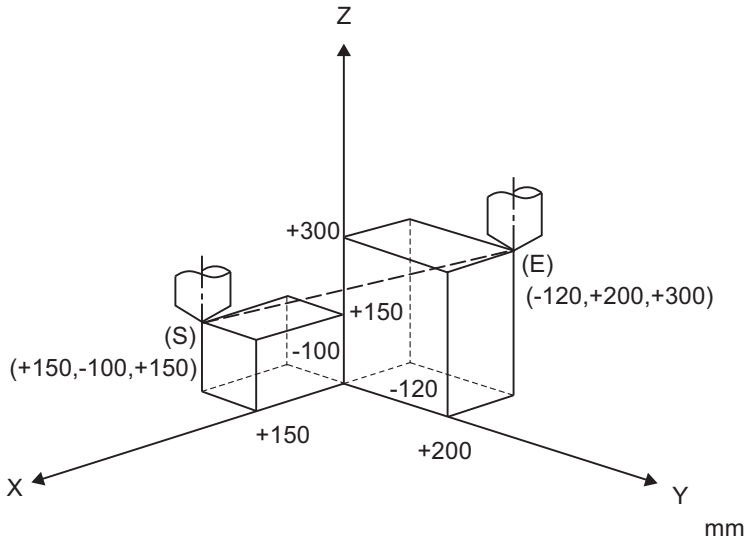
In positioning, the tool will move along the path from the start point to the end point at the rapid traverse rate of each axis.

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



(S) Start point
(E) End point

```
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 "7.13 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.

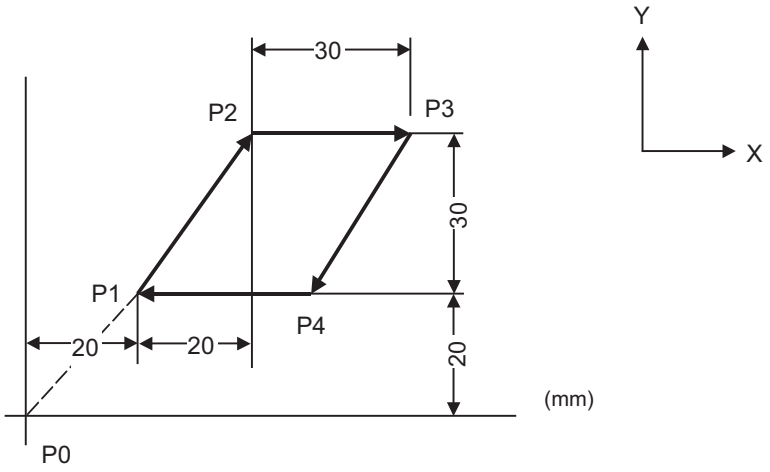
- When the error detection switch is ON.
- When G09 (exact stop check) is commanded in the same block.
- When G61 (exact stop check mode) is selected.

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

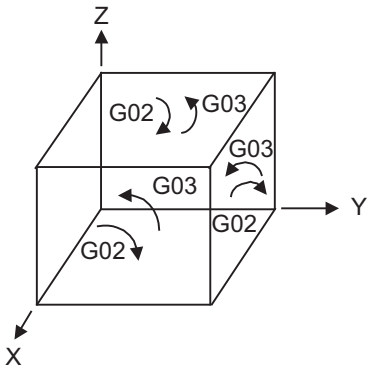
Detailed description

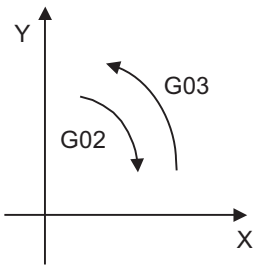
(1) For the arc command, the arc end point coordinates are assigned with addresses X, Y (or Z, or parallel axis X, Y, Z), and the arc center coordinate value is assigned with addresses I, J (or K).
Either an absolute position or incremental position can be used for the arc end point coordinate value command, but the arc center coordinate value must always be commanded with an incremental position 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 program 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.

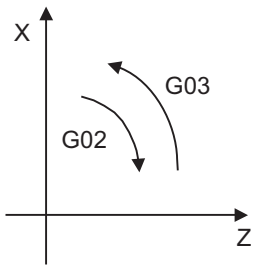
G02 CW (Clockwise)
G03 CCW (Counterclockwise)

(3) Select the XY plane, ZX plane or YZ plane to draw an arc on it, using the plane selection G code.

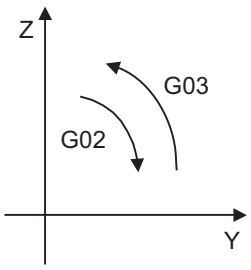




G17(X-Y) plane



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.

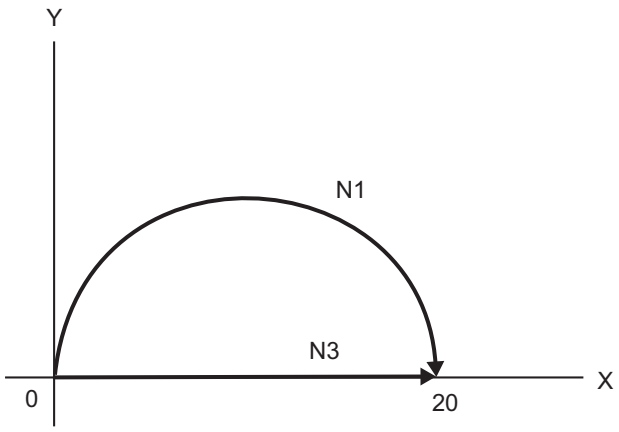
(a) Plane selection	Is there an arc parallel to one of the XY, ZX or YZ planes?
(b) Rotation direction	Clockwise (G02) or counterclockwise (G03)
(c) Arc end point coordinates	Set by addresses X, Y, Z.
(d) Arc center coordinates	Set by addresses I, J, K. (incremental commands)
(e) Feedrate	Set by address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

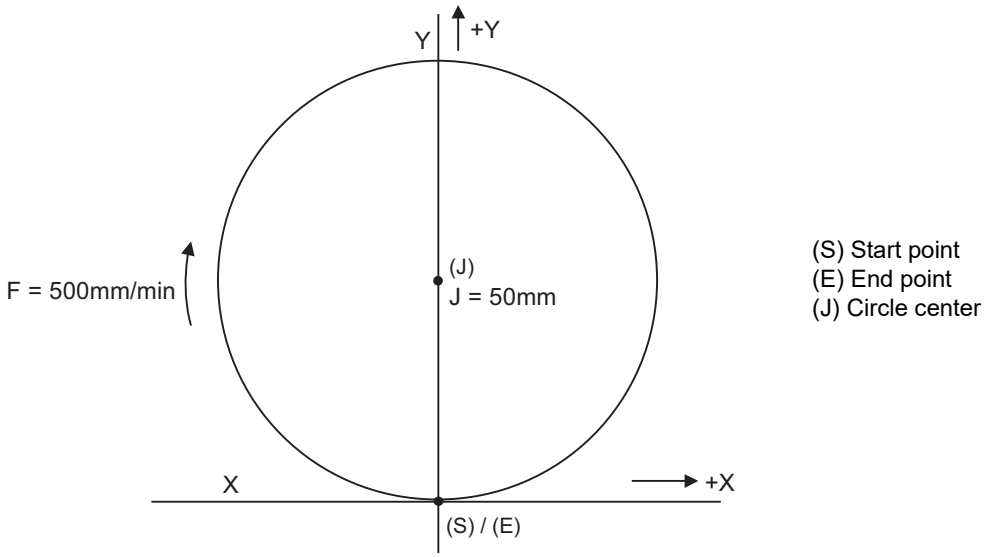



```
G90 X0 Y0 ;
N1 G02 X20. I10. F500 ;     ... (a)
N2 G00 X0 ;
N3 G02 X20. F500 ;          ... (b)
M02 ;
```

(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.

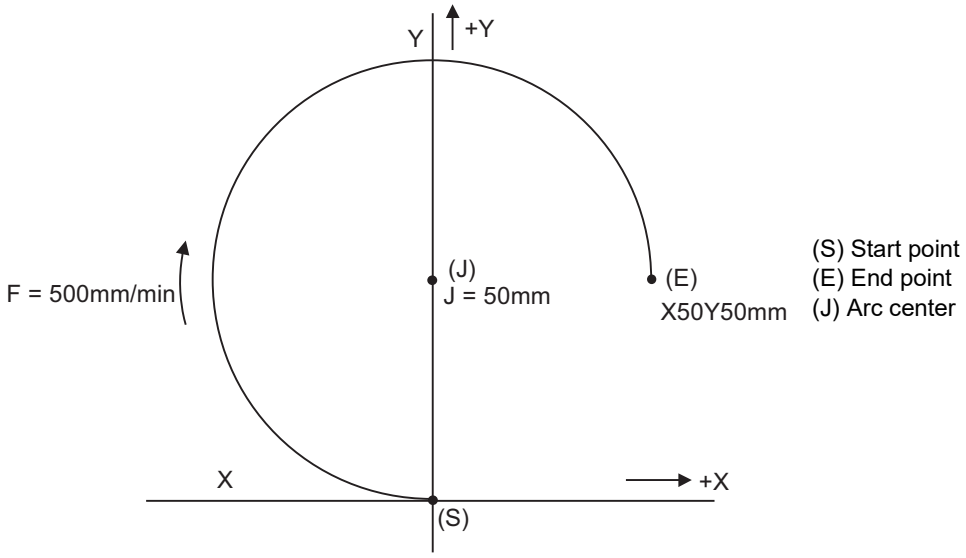
Program example

(Example 1)



G02 J50. F500;	Circle command

(Example 2)



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

Precautions

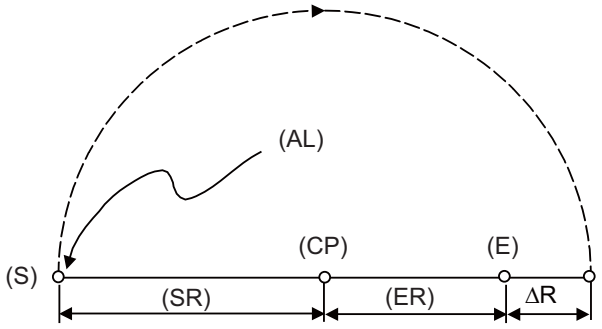
(1) The terms "clockwise" (G02) and "counterclockwise" (G03) used for circular operations are defined as a case where, in a right-hand coordinate system, the negative direction is viewed from the positive direction of the coordinate axis which is at right angles to the plane in question.

(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. ;

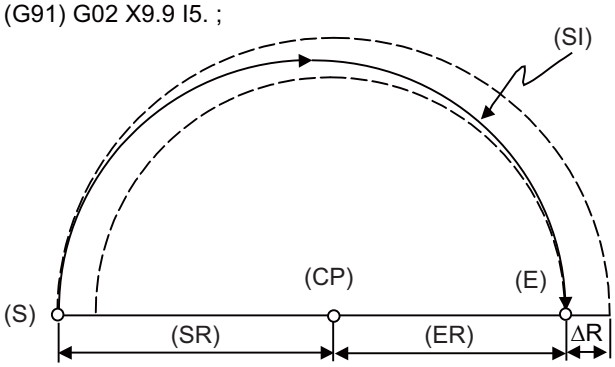


#1084 RadErr parameter value 0.100
Start point radius=5.000
End point radius=4.899
Error ΔR =0.101

(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. ;

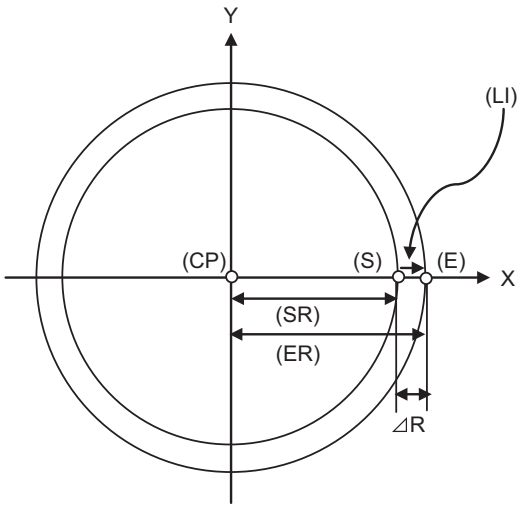
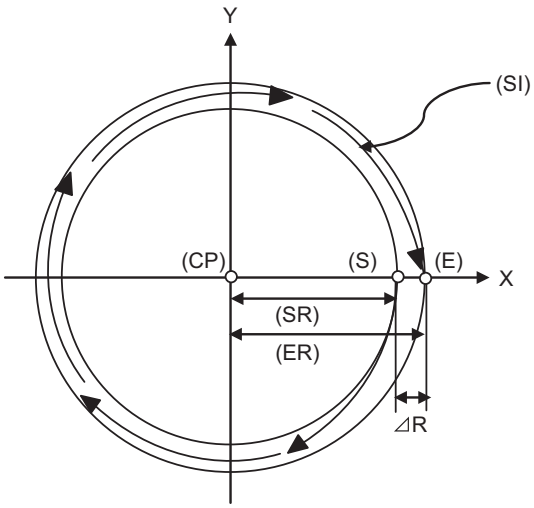


#1084 RadErr parameter value 0.100
Start point radius=5.000
End point radius=4.900
Error ΔR =0.100

(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.

(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/bit7 = 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)

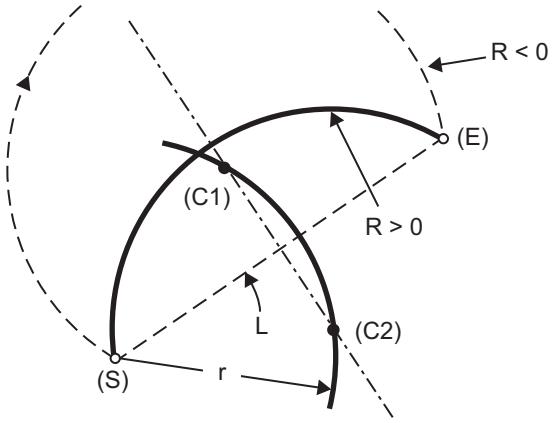
```
G03 X__ Y__ R__ F__ ;
```

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 program command unit 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.



(S) Start point
(E) End point
(C1): Arc center if R<0
(C2): Arc center if R>0

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

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

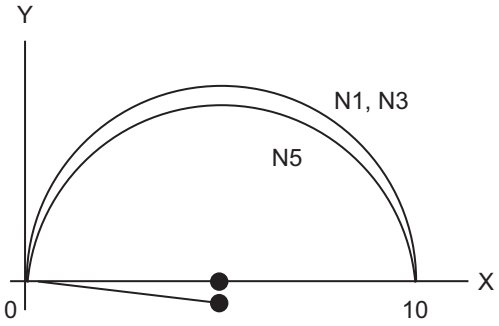
Where L is the line from the start point to the end point. If an R specification and I, J, (K) specification are given at the same time in the same block, the circular command with the R specification takes precedence. In the case of a full-circle command (where the start and end points coincide), an 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.

Command format

Normal lead thread cutting

G33 Z__(X__ Y__ α__) F__ Q__ L__ ;

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 (0.000 - 360.000°)
L	Lead axis No. (1 to maximum number of NC axes in a part system)

Precision lead thread cutting

G33 Z__(X__ Y__ α__) E__ Q__ L__ ;

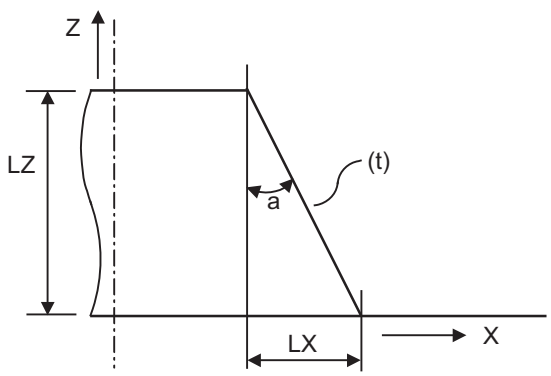
Z (X Y α)	End point of thread cutting
E	Lead of long axis (axis which moves most) direction
Q	Thread cutting start shift angle (0.000 - 360.000°)
L	Lead axis No. (1 to maximum number of NC axes in a part system)

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. (When the parameter "#1229 set01/ bit1" is set to "1", precision lead is designated.)

(2) The lead in the long axis direction is commanded for the taper thread lead. However, the lead axis can be commanded by commanding the address L. Set the address L to the axis No. of each part system.

(For example, set "L1" for the first axis, and "L2" for the second axis.)



(t) Tapered thread section

When a < 45°, lead is in LZ direction.

When a > 45°, lead is in LX direction.

When a = 45°, 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)	E (ridges/inch)
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 - 999.999	0.0001 - 999.9999	0.03 - 999.99	0.0001 - 999.9999	0.00001 - 999.99999	0.026 - 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)	E (ridges/inch)
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 - 999.99999	0.000001 - 999.999999	0.0255 - 224580.0000	0.000001 - 999.999999	0.0000001 - 999.9999999	0.02541 - 224719.00000

Thread cutting inch input

Input Setting unit	B (0.0001 inch)			C (0.00001 inch)		
Command address	F (inch/rev)	E (inch/rev)	E (ridges/inch)	F (inch/rev)	E (inch/rev)	E (ridges/inch)
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 - 39.3700	0.00001 - 39.37007	0.025 - 9999.999	0.00001 - 39.37007	0.000001 - 39.370078	0.0255 - 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)	E (ridges/inch)
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 - 39.370078	0.0000001 - 39.3700787	0.02541 - 9999.99999	0.0000001 - 39.3700787	0.00000001 - 39.37007873	0.025401 - 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 (δ1 and δ2) to the required thread length.

(9) The spindle rotation speed is subject to the following restriction:

1 <= R <= Maximum feedrate/Thread lead

Where R <= Tolerable speed of encoder (r/min)

R: Spindle rotation speed (r/min)

Thread lead = mm or inches

Maximum feedrate= mm/min or inch/mim (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) Whether dry run is valid or invalid for thread cutting depends on the MTB settings. (When the parameter "#1279 ext15/bit4" is "1", the dry run is invalid. However, when the "Spindle OFF mode" signal (YCD3) is ON, the dry run is valid regardless of the parameter setting, and depends on the "Dry run" signal (YC15).)
When the dry run is valid, the feedrate based on the 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. However, when the thread cutting override is valid, only the spindle override can be changed.

(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) In the thread cutting command, the part systems wait for the single rotation synchronization signal of the rotary encoder before starting movement.
When the thread cutting is commanded for one spindle from multiple part systems, operations are performed as follows.
Each part system where the thread cutting is commanded waits for the encoder single rotation synchronization signal before starting thread cutting. Therefore, the thread cutting is performed for one spindle from multiple part systems without performing the timing synchronization operation between part systems.

(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.

(22) When the lead axis designated with the L command does not move, the program error (P93) occurs.

(23) When the axis designated with the L command does not exist, the program error (P93) occurs.

(24) Lead axis designation with the L command is not modal. When there is no L command, the axis whose movement amount is the largest is the lead axis.

(25) Other function commands using the address L and the thread cutting command with the address L must not be issued in the same block but be issued in different blocks.

[When the command is issued in the same block as subprogram call (M98)]

The address L is handled as the number of repetitions of the subprogram call (M98).

[When the command is issued in the same block as timing synchronization command (!)]

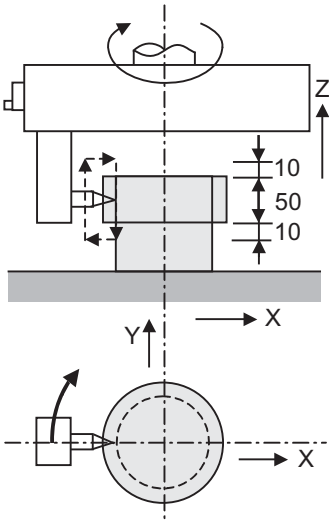
The address L is handled as the lead axis No. of thread cutting, and is not handled as timing synchronization number in the timing synchronization command (!).

(26) When the lead axis is designated with the L command, the movement amount of non-lead axis may be larger than that of lead axis. The cutting feed clamp speed is compared when thread cutting starts, and if either the lead axis or the non-lead axis exceeds the clamp speed, the operation error (M01 0107) occurs. However, in the second and subsequent blocks of continuous thread cutting, the lead is guaranteed and the operation error does not occur. Thread cutting continues with the speed exceeding the clamp speed.

Precautions

(1) For multiple spindle control II, do not change the "Encoder selection" signal (R2567) during the thread cutting. If it is changed, the thread ridges will lose their shape because the tool cannot be fed at the correct feedrate.

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__ L__ ;

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 (0.000 to 360.000°)
L	Lead axis No. (1 to maximum number of NC axes in a part system)

Detailed description

(1) The number of ridges in the long axis direction is assigned as the number of ridges per inch.

(2) The address E 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 "#1229 set01/bit1" to "0".)

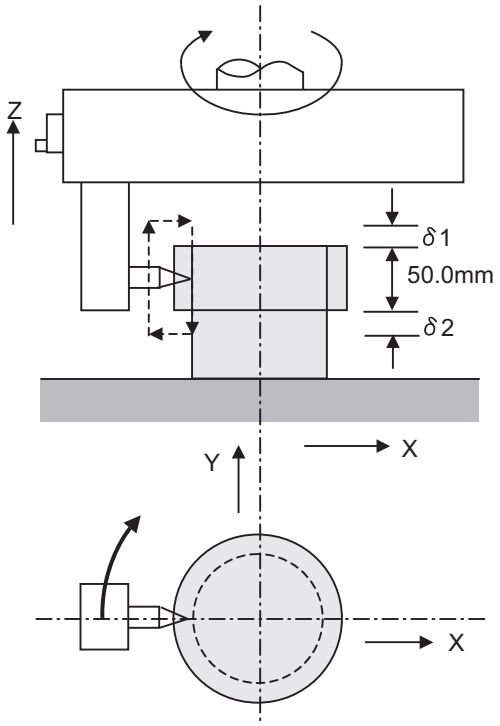
(3) The address E command value should be set within the lead value range when converted to lead.

(4) See Section "Constant lead thread cutting" for other details.

Program example

Thread lead: 3 threads/inch (= about 8.4666)

When programmed with δ1= 10 mm, δ2=10 mm using metric input

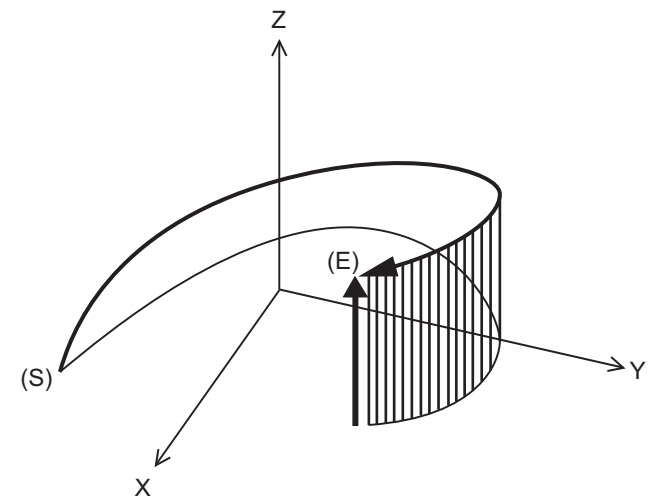


N210 G90 G00 X-200. Y-200. S50 M3 ;	
N211 Z110. ;	
N212 G91 G33 Z-70. E3.0 ;	(First thread cutting)
N213 M19 ;	
N214 G90 G00 X-210. ;	
N215 Z110. M0 ;	
N216 X-200. ; M3 ;	
N217 G04 X2.0 ;	
N218 G91 G33 Z-70. ;	(Second thread cutting)

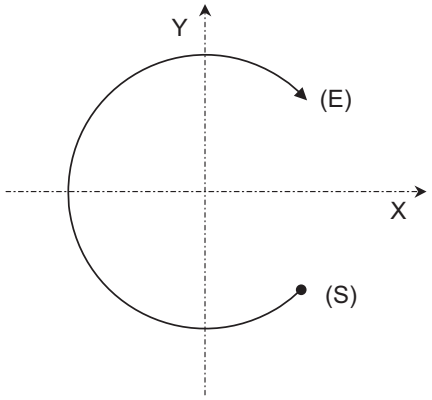
6.7 Helical Interpolation; G02, G03

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.
This command must be issued as the combination of the circular interpolation command with the height axis.



Program command path



XY plane projection path

(S) Start point
(E) End point

Program command path

Circular interpolation components

Linear interpolation components

Command format

Helical interpolation command (Specify arc center)

```
G17 G02/G03 X_ Y_ Z_ I_ J_ P_ F_ ;
G18 G02/G03 Z_ X_ Y_ K_ I_ P_ F_ ;
G19 G02/G03 Y_ Z_ X_ J_ K_ P_ F_ ;
```

Helical interpolation command (Specify radius (R))

```
G17 G02/G03 X_ Y_ Z_ R_ F_ ;
G18 G02/G03 Z_ X_ Y_ R_ F_ ;
G19 G02/G03 Y_ Z_ X_ 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, Z	Arc end point coordinates: (X, Y) is set in G17, (Z, X) in G18, and (Y, Z) in G19. Linear axis end point coordinates: Z is set in G17, Y in G18, and X in G19.
I, J, K	Arc center coordinates: (I, J) is set in G17, (K, I) in G18, and (J, K) in G19.
P	Number of pitches
R	Circular arc radius
F	Feedrate

Note

(1) In this manual, the following setting descriptions are used. I axis: X; J axis: Y; K axis: Z

(2) The arc center coordinates 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 program command unit differs. Command with a decimal point to avoid confusion.

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

(4) If a pitch command is issued with the ",P" address, a program error (P33) occurs.

(5) If the number of pitches is "0", address P can be omitted.

(6) If the radius (R) is designated, the number of pitches is ignored even when it is commanded.

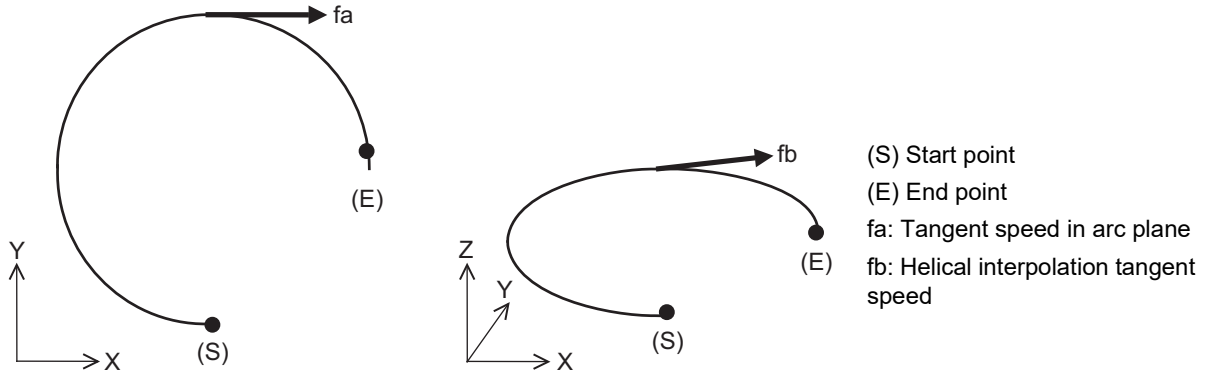
(7) Two or more axes can be designated for the linear interpolation axis.

Detailed description

Speed designation during the helical interpolation

Speed designation "F" during the helical interpolation has the following types. The available type depends on the MTB specifications.

Parameter #1235/bit0	Tangent speed (command value of address "F")
0	Speed designation for normal helical interpolation Commands the tangent speed (equivalent to "fb" in the lower-right figure) including interpolation component of the 3rd axis.
1	Speed designation by the arc plane component Commands the tangent speed (equivalent to "fa" in the lower-left figure) in the arc plane. At this time, the NC automatically calculates the helical interpolation tangent speed "fb" so that the tangent speed on the arc plane is "fa".



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

Speed designation by the arc plane component

If speed designation by the arc plane component 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.

For example, the program will be as follows.

G17	G91 G02 X10. Y10. Z-4. I10. F100 ;	Helical interpolation is performed with such speed that arc plane component is F100.
	G01 X20. ;	Linear interpolation is performed at the speed of F100.
	G02 X10. Y-10. Z4. J-10. ;	Helical interpolation is performed with such speed that arc plane component is F100.
	G01 Y-40. F120 ;	Linear interpolation is performed at the speed of F120.
	G02 X-10. Y-10. Z-4. I-10. ;	Helical interpolation is performed with such speed that arc plane component is F120.
	G01 X-20. ;	Linear interpolation is performed at the speed of F120.

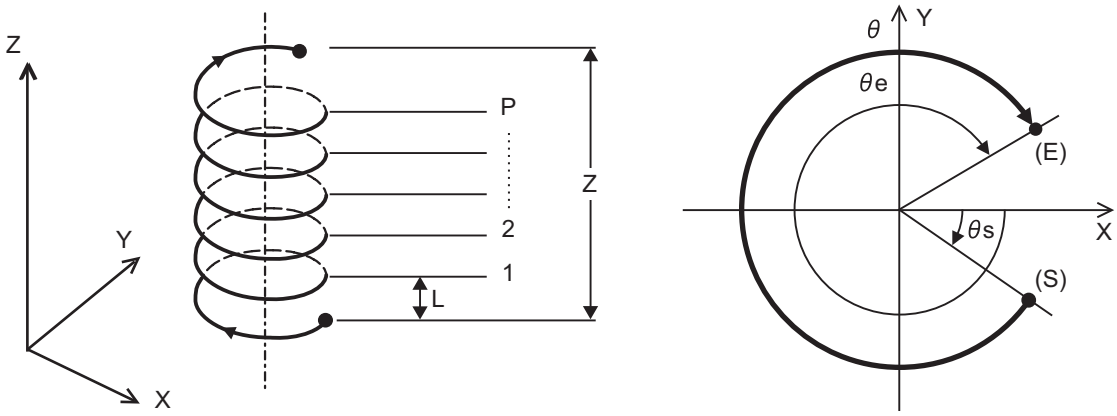
When the speed designation by the arc plane component is selected, only the helical interpolation speed command is converted to the speed commanded with the arc plane component and operates. The other linear and arc commands operate as normal speed commands.

(1) The actual feedrate display (Fc) indicates the tangent component of the helical interpolation.

(2) The modal value speed display (FA) indicates the command speed.

(3) 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 component speed will not be designated.

Number of pitches



(S) Start point (E) End point

(1) Pitch "L" is obtained with the following expression.

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

$$\theta = \theta e - \theta s = \tan^{-1}\frac{ye}{xe} - \tan^{-1}\frac{ys}{xs} \quad (0 \leq \theta < 2\pi)$$

xs, ys: Distance from the arc center to the start point (each of X and Y axes)

xe, ye: Distance from the arc center to the end point (each of X and Y axes)

(2) 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.

Plane selection

The helical interpolation arc plane selection is determined with the plane selection mode and axis address in the same manner as the circular interpolation. For the helical interpolation command, the plane where circular interpolation is executed is required to be commanded with the plane selection G code (G17, G18, G19), and two circular interpolation axes and three linear interpolation axes (axes which perpendicular to the arc plane) are required to be commanded.

X-Y plane circular, Z axis linear

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

Z-X plane circular, Y axis linear

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

Y-Z plane circular, X axis linear

Command the Y, Z and X 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.



Precautions and restrictions

(1) When executing helical interpolation, issue the circular interpolation command and another linear axis command (several axes can be commanded) that does not contain the arc axis.

(2) The number of axes that can be commanded simultaneously is less than or equal to the number of simultaneous contouring control axes.

(3) With helical interpolation, the axes that configure the plane are the circular interpolation axes, and the other axis is the linear interpolation axis.

(4) The movement of the linear interpolation axis is stopped and only the circular interpolation axes operate during the corner chamfering or corner rounding commands.

(5) Refer to description of "6.3 Circular Interpolation; G02, G03" for other precautions.

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

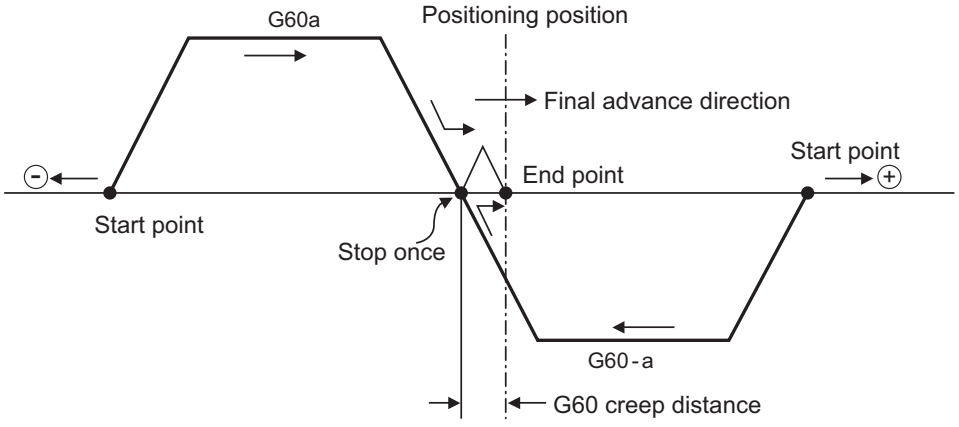
G60 X__ Y__ Z__ α__; ... Unidirectional positioning

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.

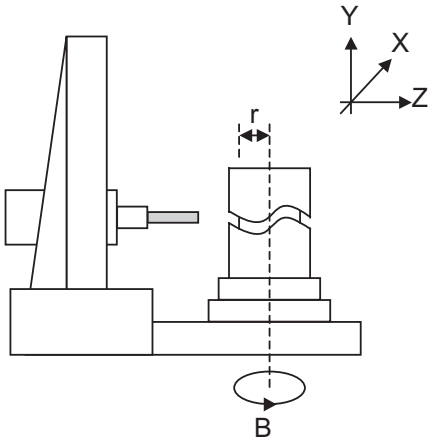
G00 X100.;	Normal positioning
G00 B100.;	Unidirectional positioning
G00X200.B200.;	Axis X: Normal positioning, Axis B: Unidirectional positioning

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

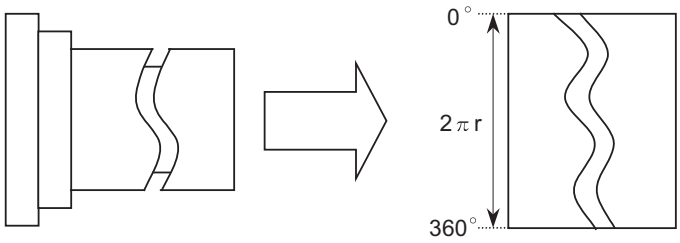
6.9 Cylindrical Interpolation; G07.1

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 alone in a 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.

(11) The unit of coordinate command during cylindrical interpolation is metric (mm) or inch (inch) for linear axis and degrees (°) for rotary axis.

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

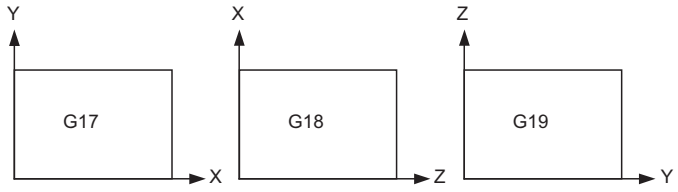
The axis used for cylindrical interpolation must be set with the plane selection command.

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.

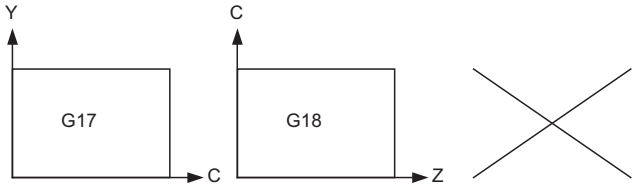
(Example)

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

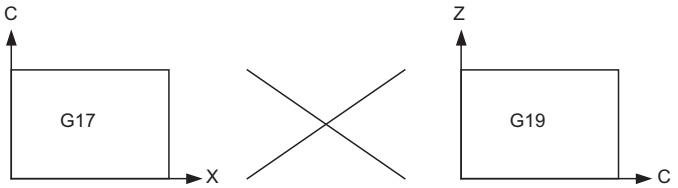
Basic coordinate system
X, Y, Z



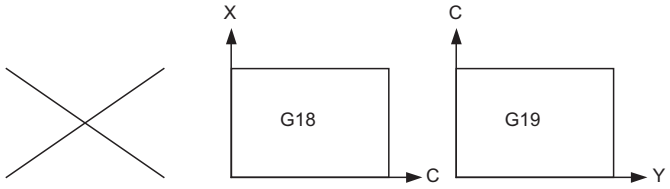
Cylindrical coordinate system
C, Y, Z
(Rotary axis is X axis' parallel axis)
#1029



Cylindrical coordinate system
X, C, Z
(Rotary axis is Y axis' parallel axis)
#1030



Cylindrical coordinate system
X, Y, C
(Rotary axis is Z axis' parallel axis)
#1031



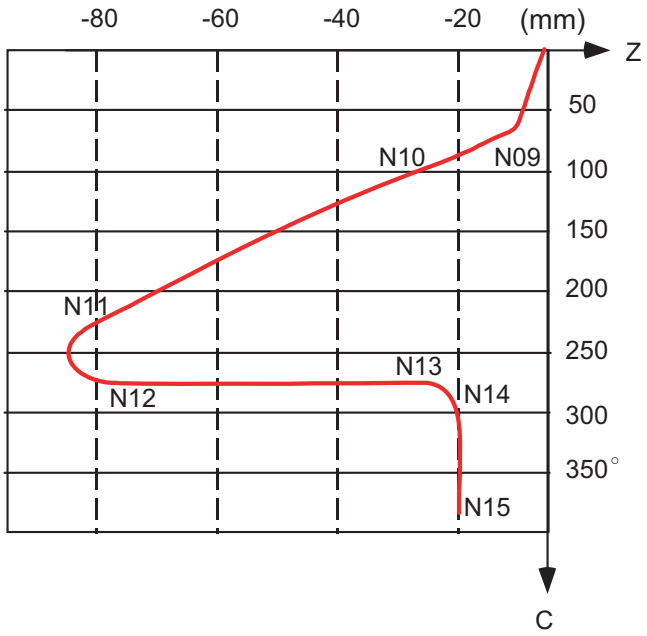
Program example

<Program>

```
N01 G28 XZC ;
N02 T0202 F500 ;
N03 G97 S100 M23 ;
N04 G00 X50. Z0. ;
N05 G94 G01 X40. F100. ;
N06 G19 C0 Z0 ;          Plane selection command for cylindrical interpolation and two axes command for interpolation
N07 G07.1 C20. ;         Cylindrical interpolation start
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 ;           Cylindrical interpolation cancel
N18 G01 X50. ;
N19 G00 X100. Z100. ;
N20 M25 ;
N21 M30 ;
```

<Parameters>

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



Relationship with other functions

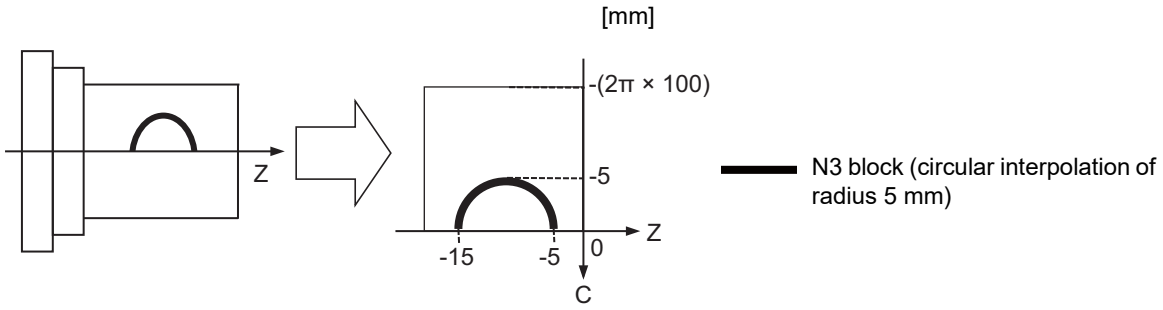
Circular interpolation

(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.)

(3) An arc is drawn on the developed surface of the cylinder in the circular interpolation, thus the calculation is performed with the C axis unit in millimeters (mm).

```
N1 G18 Z-5.C0.;     Perform positioning at Z-5 mm, C0 degrees. (The parameter "#1029 aux_I" is "C".)
N2 G07.1 C100.;     Start cylindrical interpolation mode of radius 100 mm.
N3 G02 Z-15. R5.;   Perform circular interpolation of radius 5 mm to Z-15 mm.
```



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 by commanding G40 alone, 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 - G42	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 - G89	Fixed cycle for drilling
G90/G91	Absolute/incremental command
G94	Asynchronous feed
G98	Hole drilling cycle initial return
G99	Hole drilling cycle R point return
G135	Text engraving cycle
G136	Two-dimensional barcode engraving cycle

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 axis-specific acceleration tolerance control (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.

(5) While the program format is switched, the H address is not handled as an axis address. When the cylindrical interpolation is commanded by handling H as the rotary axis name, the program error (P33) occurs.

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)

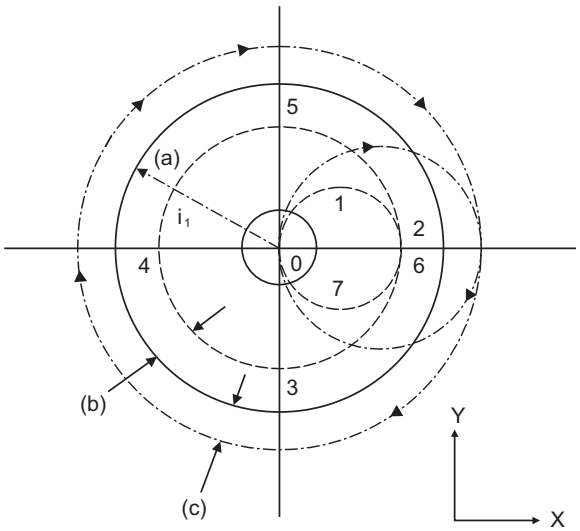
```
G13 I__ D__ F__ ;
```

I	Radius of circle (incremental position), 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 +

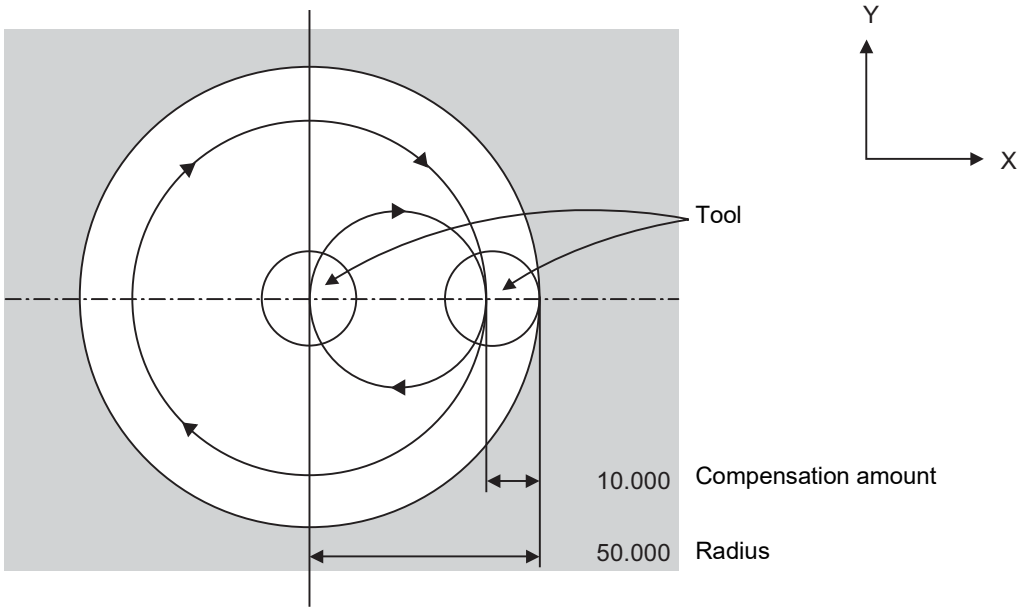
(c) d1 offset amount -

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

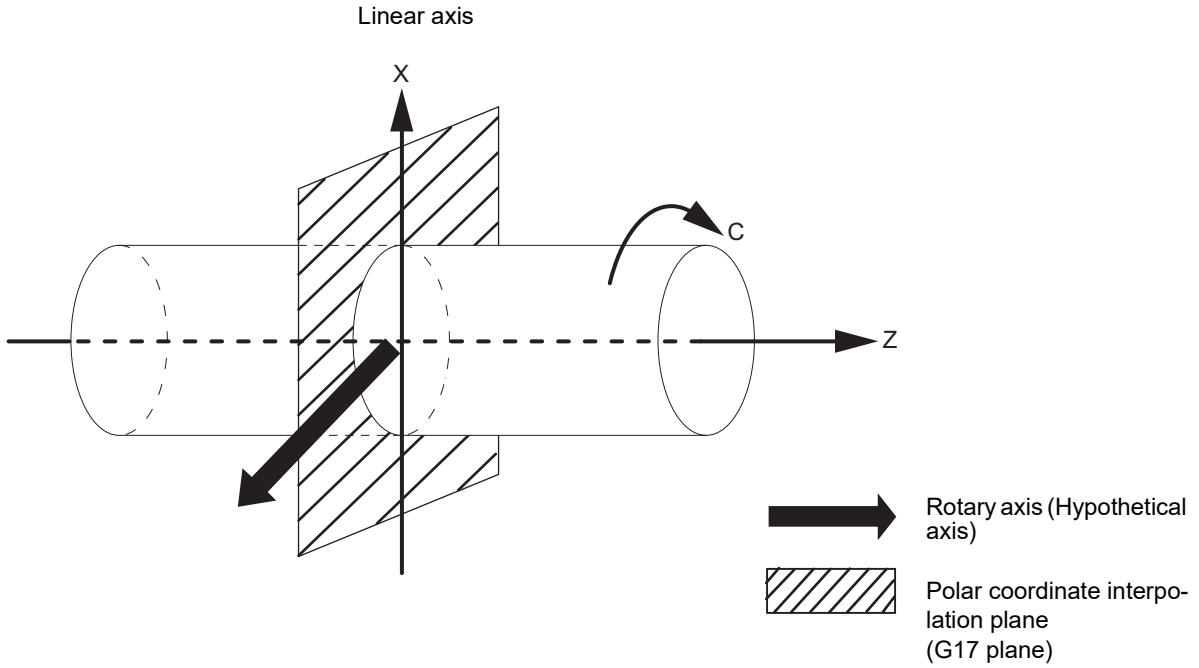
Function and purpose

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.

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

The plane that uses the linear axis as the plane's 1st 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 plane depends on the MTB specifications (parameter #1533 and #1761/bit0).)

The workpiece coordinate system zero point is used as the coordinate system zero point during polar coordinate interpolation.



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 is the polar coordinate interpolation.

```
G12.1 ;    Polar coordinate interpolation mode start
           (Polar coordinate interpolation starts.)
  :        (The coordinate commands in this interval is 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) G12.1 must be commanded alone in a block, which also applies to G13.1. If it is commanded in the same block with other G codes, the program error (P33) occurs.

(4) Linear interpolation or circular interpolation can be commanded during the polar coordinate interpolation mode.

(5) Either absolute command or incremental command can be specified for the coordinate command.

(6) Tool radius compensation can be applied on the program command. Polar coordinate interpolation is executed to the path after it has gone through a tool radius compensation.

(7) As for the feedrate, 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) If any of the axes commanded during polar coordinate interpolation has not completed the reference position return, the program error (P484) occurs.

(9) The deceleration check is made for the G12.1 and G13.1 command blocks.

(10) When the polar coordinate interpolation mode is canceled and switched to the turning mode, the plane selected before polar coordinate interpolation is restored.

(11) The polar coordinate interpolation mode is canceled when the power is turned ON or reset.

(12) Tool radius compensation must be canceled before canceling the polar coordinate interpolation mode.

(13) If the polar coordinate interpolation is commanded during the constant surface speed control mode (G96), the program error (P485) occurs.

Change of polar coordinate interpolation plane

The linear axis and rotary axis used for polar coordinate interpolation depend on the MTB specifications (parameter "#1533 millPax" (Polar coordinate linear axis name), "#1761 cfgPR11/bit0" (Plane selection method for Polar coordinate interpolation)).

Polar coordinate interpolation plane is determined with "#1533 millPax".

Whether the linear axis is used as the 1st plane axis or the 2nd plane axis of the polar coordinate interpolation plane is determined with "#1761 cfgPR11/bit0".

#1761 cfgPR11/bit0	#1533	Polar coordinate interpolation plane
0	X	G17 (XY plane)
	Y	G19 (YZ plane)
	Z	G18 (ZX plane)
	Blank (no setting)	G17 (XY plane)
1	X	G18 (ZX plane)
	Y	G17 (XY plane)
	Z	G19 (YZ plane)
	Blank (no setting)	G18 (ZX plane)

Note

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

(2) The program error (P485) occurs if the plane selection command (G17 to G19) is issued during the polar coordinate interpolation mode.

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 hypothetical axis command unit is not degree. Issue the command with the same unit (mm or inch) as the axis address for the linear axis.
The axis address for the hypothetical axis command depends on the MTB specifications. (Parameter "#1517 mill_C" (Milling interpolation hypothetical axis name))

#1761/ bit0	#1533	Polar coordinate interpolation plane	#1517 mill_C					
			0		1		2	
			Linear axis	Hypothetical axis	Linear axis	Hypothetical axis	Linear axis	Hypothetical axis
0	X	G17 (XY plane)	X	Y	X	C	X	Y
	Y	G19 (YZ plane)	Cannot be commanded		Y		Y	Z
	Z	G18 (ZX plane)	Z		Z		Z	X
	Blank (no setting)	G17 (XY plane)	X		X		X	Y
1	X	G18 (ZX plane)	X		X		X	Z
	Y	G17 (XY plane)	Cannot be commanded		Y		Y	X
	Z	G19 (YZ plane)	Z		Z		Z	Y
	Blank (no setting)	G18 (ZX plane)	X		X		X	Z

Note

♦The above table indicates the operation when the following settings are made.
- The parameters "#1026 base_I" to "#1028 base_K" (basic axes I, J, and K) are set to X, Y, and Z respectively.
- The parameter "#1516 mill_ax" is set to "C".

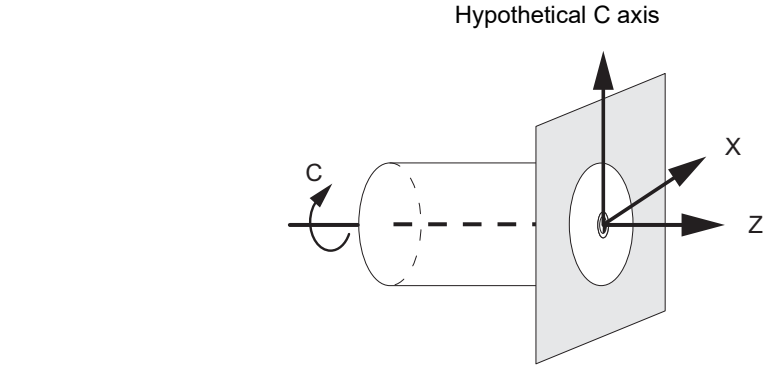
(2) The hypothetical axis coordinate value at the time G12.1 is commanded is "0". In other words, the position where G12.1 is commanded is interpreted as an angle of "0" degree, and the polar coordinate interpolation is started.

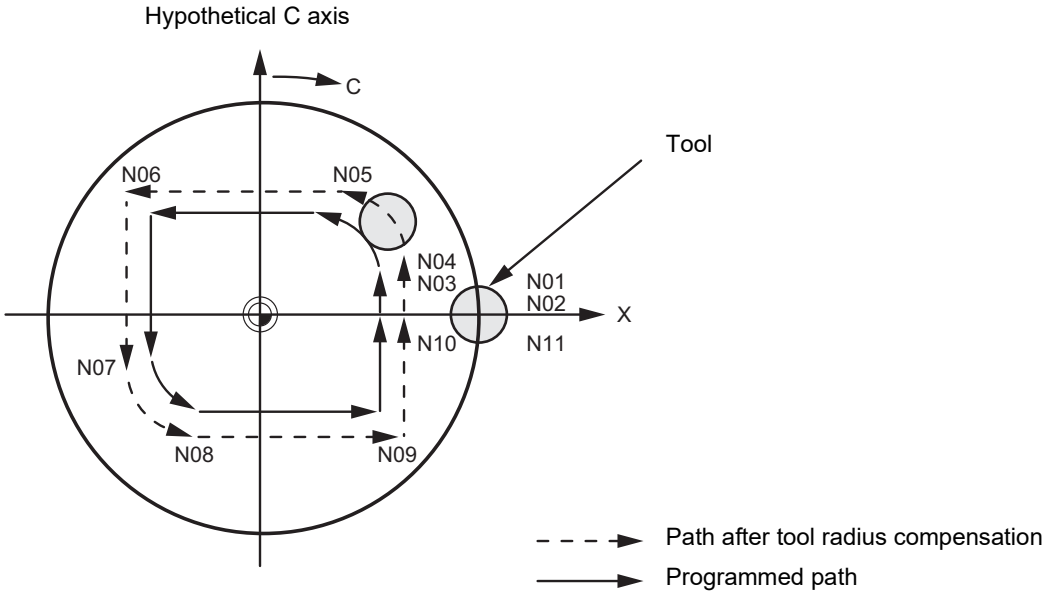
(3) The incremental command of when the parameter "#1076 AbsInc" is "1" is commanded with the axis address of the parameter "#1014 incax" (Increment command axis name) corresponding to "Linear axis" and "Hypothetical axis" in the above table. However, when "#1517 mill_C" is "0" (Y axis command), the incremental command is commanded with the axis address "V".

(4) The axis address that can be commanded depends on the settings of the parameters "#1517 mill_C", "#1013 axname" and "#1014 incax".

#1517 mill_C	Axis address that can be commanded
0	Y axis and V axis can be commanded even when they are not set in the parameters "#1013 axname" and "#1014 incax".
1 or 2	The axis address set in the parameter "#1013 axname" or "#1014 incax" can be commanded. If the axis which is not set to these parameters is specified, the program error (P610) occurs when G12.1 is commanded.

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;     (Depends on orthogonal coordinate positions on X-C hypothetical axis
                                    plane.)
N09 G01 X20.0;
N10 C0;
N11 G40 X40.0;
N12 G13.1;                          Polar coordinate interpolation mode: Cancel
 :
M30 ;
```

Relationship with other functions

Circular interpolation on polar coordinate interpolation plane

The arc radius address for carrying out circular interpolation during the polar coordinate interpolation mode is determined with the polar coordinate interpolation plane.

Polar coordinate interpolation plane	Center designation command
G17 (XY plane)	I, J
G19 (YZ plane)	J, K
G18 (ZX plane)	K, I

The arc radius can also be designated with the R command.
The plane must be selected before the G12.1 command.
If the plane selection (G16 to G19) is commanded during the G12.1 modal, the program error (P485) occurs.

Note

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

Tool radius compensation

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

(1) Command the plane selection in the same manner as polar coordinate interpolation.
To conduct tool radius compensation, it must be started up and canceled during the polar coordinate interpolation mode.

(2) The program error (P485) occurs if polar coordinate interpolation is executed during tool radius compensation.

(3) If the G12.1 or G13.1 command is issued with no movement command after the tool radius compensation is canceled, the position of the axis in the G12.1 or G13.1 command block is interpreted as the position after the tool radius compensation is canceled and the subsequent operations are performed.

Miscellaneous functions

(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 offset

(1) The program error (P481) occurs 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 -> 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 was issued, the following occurs:

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 orthogonal coordinate of polar coordinate interpolation is set for this shifted workpiece coordinate system.
The workpiece coordinate system shifted here is not reset even if the polar coordinate 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 polar coordinate interpolation mode

Operations vary depending on whether the synchronous feed during polar coordinate interpolation is valid or invalid. 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 polar coordinate interpolation mode.
If synchronous feed (G95) is commanded during polar coordinate interpolation, the program error (P481) occurs.
The operation when the polar coordinate interpolation mode is started or canceled varies depending on the feed mode that is active before the polar coordinate interpolation mode is started.

(a) If the feed mode before start is the asynchronous feed (G94)

The previous feed mode (asynchronous feed) and F command feedrate are used when the polar coordinate interpolation mode is started or canceled.

(b) If the feed mode before start is the synchronous feed (G95)

The asynchronous feed is forcibly set and the feedrate is canceled when the polar coordinate interpolation mode is started.

Set the feedrate with an F command after the polar coordinate interpolation mode is started. The program error (P62) occurs if the F command is not designated.

When the polar coordinate interpolation mode is canceled, the feed mode (synchronous feed) and feedrate before polar coordinate interpolation start are restored.

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

Both the synchronous and asynchronous feed are valid during the polar coordinate interpolation mode.
The feed mode and feedrate remain unchanged and take over the previous state when the polar coordinate interpolation mode is started or canceled.

Changes in the feed mode before and after polar coordinate interpolation mode

#1293 ext29/bit0	Changes in feed mode and feedrate		
	Before start	At start	At cancel
0	Asynchronous feed	Asynchronous feed Feedrate just before start	Asynchronous feed Feedrate just before cancel
	Synchronous feed	Asynchronous feed Feedrate is "0" (cancel)	Synchronous feed Feedrate before the polar coordinate interpolation is started
1	Asynchronous feed	Asynchronous feed Feedrate just before start	Feed mode just before cancel Feedrate just before cancel
	Synchronous feed	Synchronous feed Feedrate just before start	

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 polar coordinate interpolation plane. The hole drilling axis is an axis perpendicular to each polar coordinate interpolation plane.

Polar coordinate interpolation plane	Hole drilling axis
G17 (XY plane)	Z
G19 (YZ plane)	X
G18 (ZX plane)	Y
Blank (no setting)	Z

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

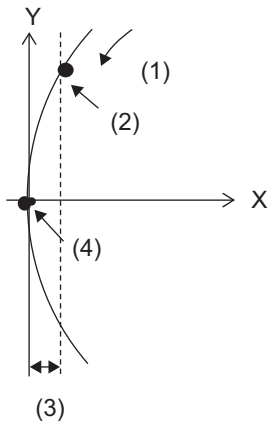
Shift amount address (I, J, K) in the G76 (fine boring) or G87 (back boring) command during the polar coordinate interpolation is determined with the polar coordinate interpolation plane.

Polar coordinate interpolation plane	Shift amount address
G17 (XY plane)	I, J
G19 (YZ plane)	J, K
G18 (ZX plane)	K, I
Blank (no setting)	I, J

Operation switching for circular interpolation in the center of a workpiece

In circular interpolation that passes through the center of the workpiece, a large movement occurs on the C axis when the axis passes through the center of the workpiece. You can switch whether to make the rotation direction of the C axis at that time a shortcut or to maintain the rotation direction up to that point, using the parameter "#19104 G12.1 no reversal". Set "0" to make a shortcut, and set "1" to maintain the rotation direction up to that point.
The parameter "#19104 G12.1 no reversal" can be switched only for the movement of the timing when the axis passes through the center of the workpiece. Specify the range to be judged as the center of the workpiece using the parameter "#19105 G12.1 zero range".
If one axis on the polar coordinate interpolation plane crosses the quadrant and the other axis is within the range of the parameter "#19105 G12.1 zero range", it is assumed that the axis passed through the center of the workpiece.
The rotation direction of the C axis used at the quadrant switching is the rotation direction of the C axis used immediately before the position of the axis on the polar coordinate interpolation plane enters the range specified with the parameter "#19105 G12.1 zero range".

[Timing to acquire the rotation direction of the C axis used at quadrant switching]



(1) Circular command direction
(2) Timing to acquire the movement direction of the C axis
(3) Range specified with the parameter #19105
(4) Timing at which the C axis rotates in the same direction as (2)
(When the parameter #19104 is set to "1")

If "1" is set to the parameter #19104, adjust the setting value of the parameter #19105.
The movement of the C axis becomes unstable near the center of the workpiece; therefore, set the parameter #19105 to a value larger than "0.0". Execute the machining program including the target circular interpolation, and check the rotation direction of the C axis at the center of the workpiece. If there is no problem, the setting is completed.
If the rotation direction of the C axis that passes through the center of the workpiece is not improved, change the setting value of the parameter #19105 to a larger value.

Note

(1) If the arc does not pass through the center of the workpiece, the direction does not change suddenly when the quadrant is switched. Set the parameter #19104 to "0".
(2) In the case of an arc of which the starting point is near the center of rotation (the starting point is within the range specified in the parameter #19105), the rotation direction of the C axis used at the time of quadrant switching is shortcut regardless of the setting of the parameter #19104. Therefore, set the starting point of the arc to a point away from the center of the workpiece (outside the range specified in the parameter #19105).

G code that can be used during the polar coordinate interpolation mode

G code	Details
G00	Positioning
G01	Linear interpolation
G02	Circular interpolation (CW)
G02.2	Involute interpolation (CW)
G03	Circular interpolation (CCW)
G03.2	Involute interpolation (CCW)
G04	Dwell
G09	Exact stop check
G40 - G42	Tool radius compensation
G61	Exact stop mode
G64	Cutting mode
G65	User macro (simple call)
G66	User macro (modal call)
G66.1	User macro (block call per macro)
G67	User macro cancel (modal call cancel)
G80 - G89	Fixed cycle for drilling
G90/G91	Absolute command/Incremental command
G94	Asynchronous feed
G95 (*1)	Synchronous feed
G98	Fixed cycle (Initial point return)
G99	Fixed cycle (R point return)

(*1) It is enabled only when the parameter "#1293 ext29/bit0" is "1".

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



Restrictions and precautions

(1) Program cannot be restarted (program restart) when the block is in the polar coordinate interpolation.

(2) The program error (P486) occurs if the polar coordinate interpolation command is issued during the mirror image.

(3) The program error (P481) occurs if the cylindrical interpolation or the polar coordinate interpolation is commanded during the polar coordinate interpolation mode.

(4) 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.)

(5) The feedrate during polar coordinate interpolation is the interpolation speed on the polar coordinate interpolation plane (orthogonal coordinate system).
(The relative speed with the tool varies 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 becomes very high.

(6) When the axis movement is commanded outside of the polar coordinate interpolation plane, axis movement is made independently of the polar coordinate interpolation.

(7) All the current position counters during polar coordinate interpolation indicate the actual coordinate value. However, the "remaining movement amount" indicates the movement amount on the polar coordinate plane.

(8) During polar coordinate interpolation, if X axis movable range is controlled in the plus side, X axis has to be moved to the plus area that includes "0" or more before the polar coordinate interpolation command is issued. If the X axis movable range is controlled on the minus side, the X axis has to be moved to the minus area that does not include "0" before the polar coordinate command is issued.

(9) Polar coordinate interpolation cannot be commanded when high-accuracy control is active. Polar coordinate interpolation cannot also be commanded when the parameter "#12053 EachAxAccCntrl" (Enable axis-specific acceleration tolerance control) is set to "0" (Optimal corner deceleration) or the parameter "#12060 VblAccPreInt" (Variable-acceleration pre-interpolation acceleration/deceleration ON) is set to "0" (Pre-interpolation acceleration/deceleration). In either case, if it is commanded, the program error (P126) occurs.

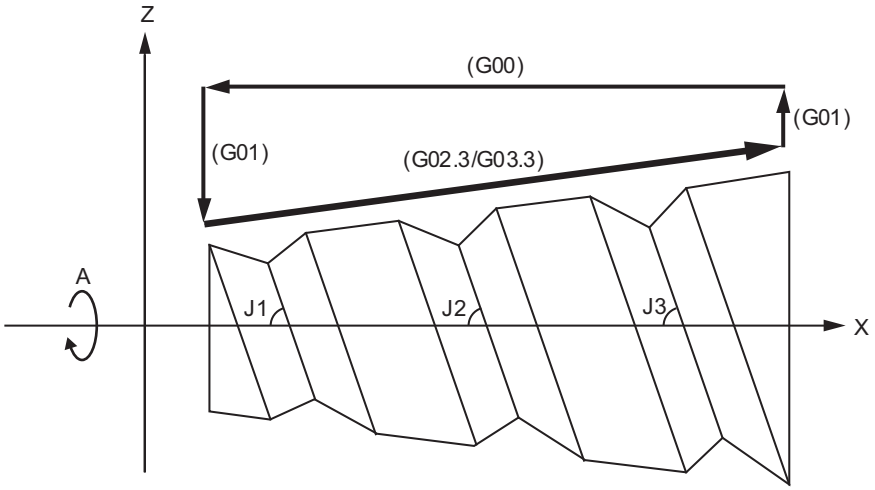
(10) While the program format is switched, the H address is not handled as an axis address. When the parameter "#1516 mill_ax" is set to "H", if polar coordinate interpolation is commanded during the program format switch in progress, the program error (P610) occurs.

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.

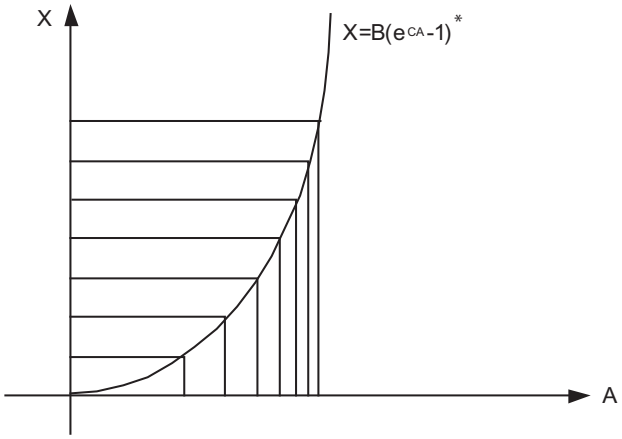
♦Uniform helix machining of taper shape



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 X__ Y__ Z__ I__ J__ R__ F__ Q__ K__ ;
```

Backward rotation interpolation (Modal)

```
G03.3 X__ Y__ Z__ I__ J__ R__ F__ Q__ K__ ;
```

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.

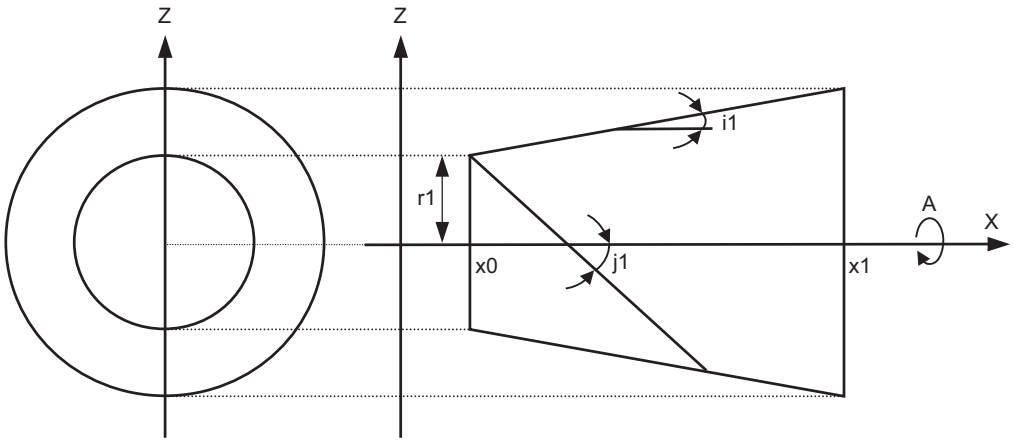
Setting unit	#1003 = B	#1003 = C	#1003 = D	#1003 = E	Unit
Millimeter system	0.001	0.0001	0.00001	0.000001	mm
Inch system	0.0001	0.00001	0.000001	0.0000001	inch

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 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]



X : X axis (linear axis) A : A axis (rotation axis) x0 : linear axis (X axis) start point

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.

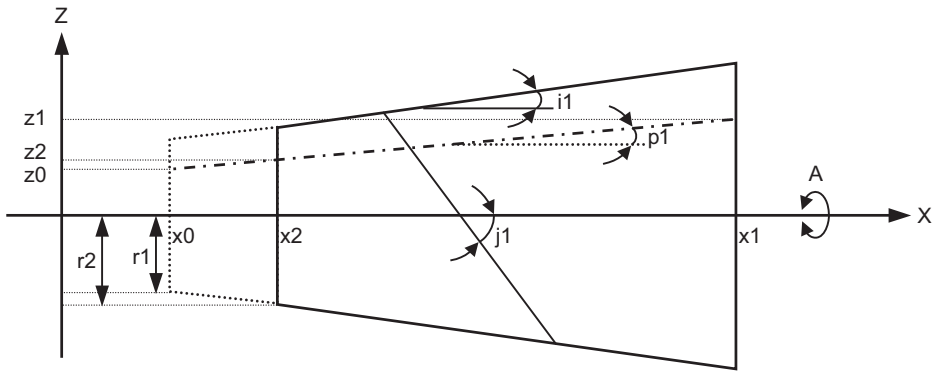
$X(\theta) = r1 * (e^{\theta/D} - 1) / \tan(i1)$ Linear axis (X) movement (1)

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

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

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

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

where, "D" is as follows.

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

Z(θ)	Absolute position from zero point of Z axis (axis that linearly interpolates with linear axis (X axis))
X(θ)	Absolute position from X axis (linear axis) start point
A(θ)	Absolute position 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.
$Z(\theta) = X(\theta) * \tan(p1) + z0$ ...(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).

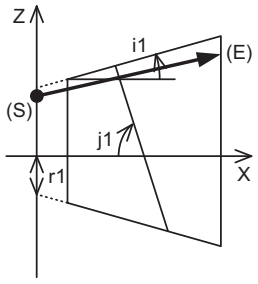
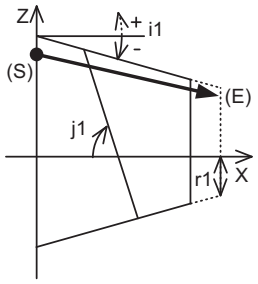
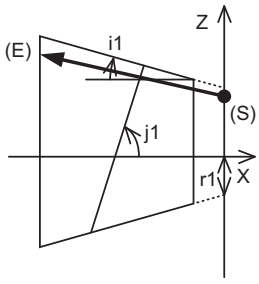
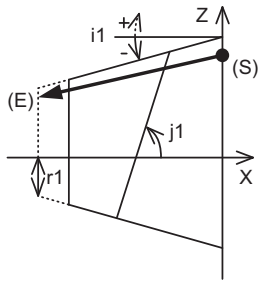
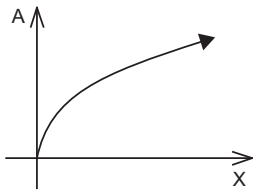
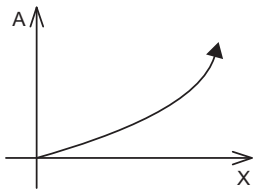
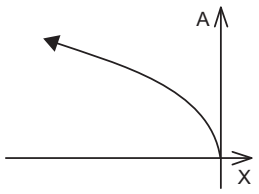
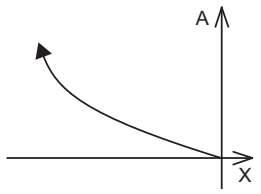
$r1 = r2 - \{(x2 - x0) * \tan(i1)\}$

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) G02.3 (equivalent to G03.3 if j1 < 0)

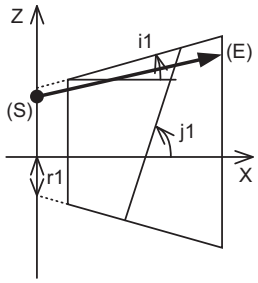
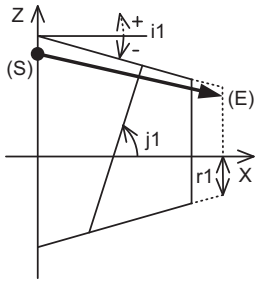
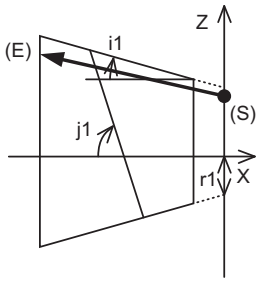
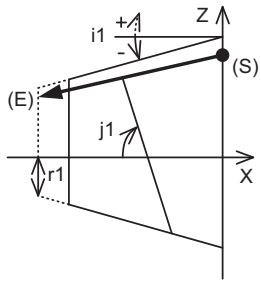
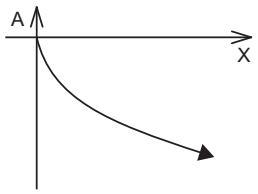
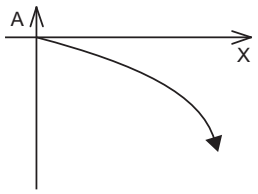
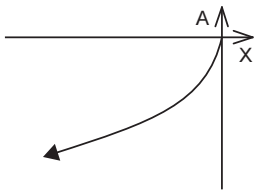
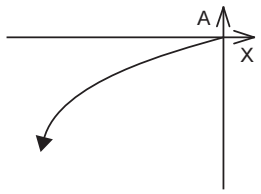
In the conditional figure below, the upper side shows a command, and the lower side shows an operation.

X movement direction > 0		X movement direction < 0	
i1 > 0	**i1 < 0**	**i1 > 0**	**i1 < 0**
			
			
N10 G28XYZC; N20 G91G0 X100. Z100.; N30 G02.3 X100. Z100. I50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X100. Z200.; N30 G02.3 X100. Z-100. I-50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X-100. Z100.; N30 G02.3 X-100. Z100. I50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X-100. Z200.; N30 G02.3 X-100. Z-100. I-50. J80. R105. F500.; N40 M30;

(S) Start point, (E) End point

(2) G03.3 (equivalent to G02.3 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 G03.3 X100. Z100. I50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X100. Z200.; N30 G03.3 X100. Z-100. I-50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X-100. Z100.; N30 G03.3 X-100. Z100. I50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X-100. Z200.; N30 G03.3 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 machining mode I, High-speed machining mode II
- High-accuracy control
- High-speed high-accuracy control I, High-speed high-accuracy control II, High-speed high-accuracy conrol III
- Scaling
- Tool length compensation along the tool axis
- Figure rotation
- Coordinate rotation by program
- Coordinate rotation by parameter
- 3-dimensional coordinate conversion

(4) The program error (P481) occurs if commands are issued during the polar coordinate interpolation, cylindrical interpolation or milling interpolation modes.

(5) The program error (P612) occurs if commands are issued during the scaling or mirror image.

(6) The program error (P34) occurs 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) While the program format is switched, the H address is not handled as an axis address. When the parameter "#1514 expLinax" or "#1515 expRotax" is set to "H", if G02.3/G03.3 is commanded while the program format is switched, the program error (P610) occurs.

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

Polar coordinate command mode ON

```
G16 ;
```

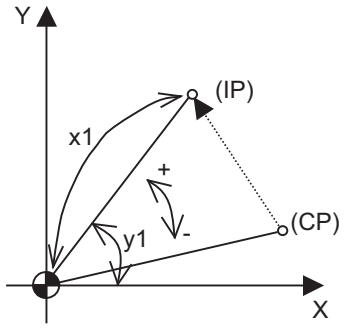
Polar coordinate command mode OFF

```
G15 ;
```

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



(CP) Current position
(IP) Commanded position

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 command (G90) and incremental command (G91).

(7) When the radius is commanded with the absolute position, 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 command, considering the end point of the previous block as the polar coordinate center, command the incremental position from that end point. The angle is commanded with the incremental position of the angle from the previous block.

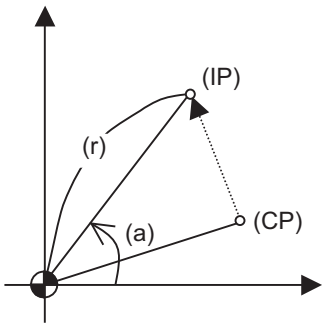
(9) When the radius is commanded with the negative value, the same operation as the command that the radius command value is changed to the absolute position and 180° is added to the angle command value.

Commanded position

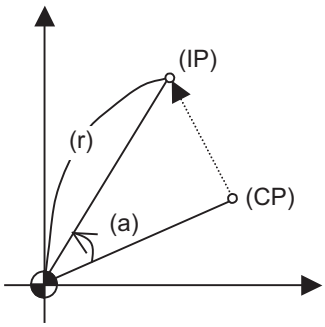
(1) When the zero point in the workpiece coordinate system is applied to the polar coordinate center

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

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.



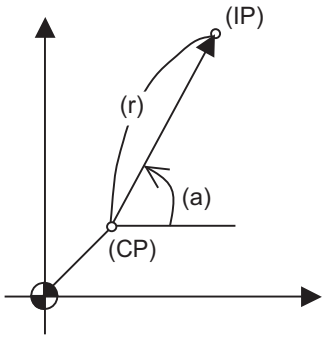
When the angle is the absolute command



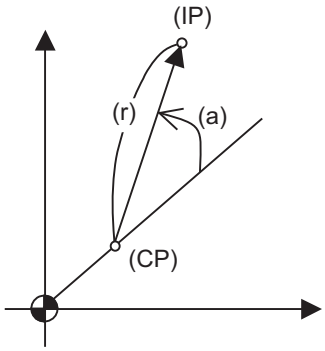
When the angle is the incremental command

(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 position.



When the angle is the absolute command

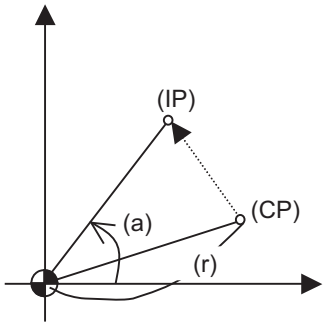


When the angle is the incremental command

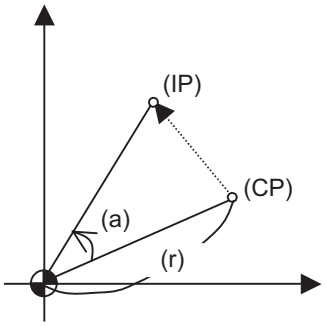
(CP) Current position (IP) Command position (a) Angle (r) Radius

(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 command



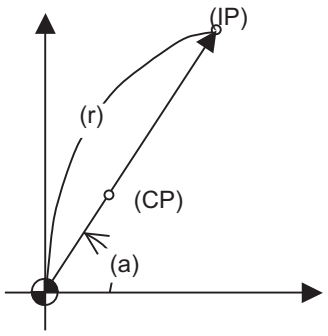
When the angle is the incremental command

(4) When the angle command is omitted

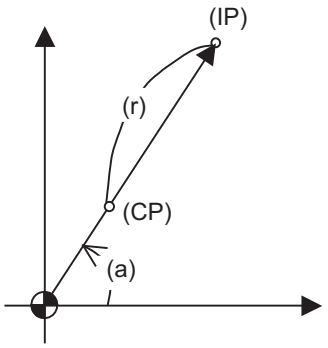
When the angle command is omitted, the angle of the present position in the workpiece coordinate system is applied to the angle command.

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



When the angle is the absolute command



When the angle is the incremental command

(CP) Current position (IP) Command position (a) Angle (r) Radius

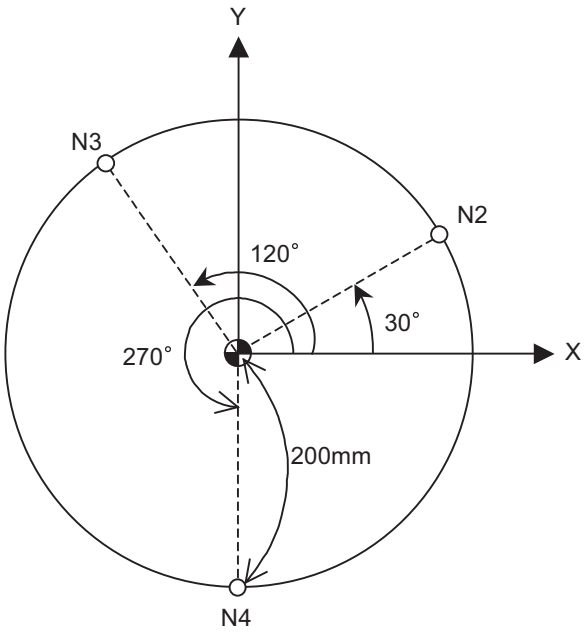
Axis command not interpreted as polar coordinate command

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 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 command and the angle is the incremental 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 position and machine position will deviate. The mirror center is set with an absolute position, 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 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. Conical interpolation is carried out by designating the end point in the height direction.

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

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__;
```

Description of each address

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 (Type 2)	Incremental/decremental amount of radius (Type 2)
L (Type 2)	Number of pitches (Number of spirals) (Type 2)
K (Type 2)	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.

(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 increment or decrement amount of radius 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 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) 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.

(3) The axis combination that can be simultaneously commanded depends on the specifications. The combination within that range is arbitrary.

(4) The feedrate is the constant tangent speed.

(5) 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.

(6) 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.

(7) 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.)	•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.)	•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.

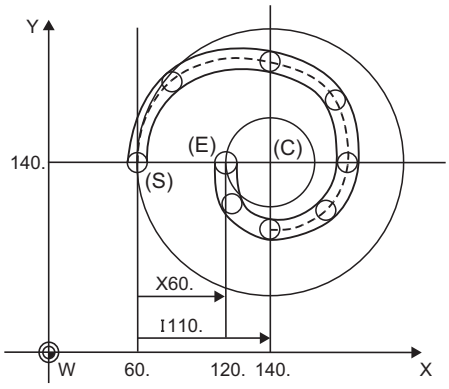
(b) Items for type 2 only

Setting Items	Command range (unit)	Error
Increment or decrement amount of radius	Range of coordinate command (mm/inch) (Decimal point command is possible.)	•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.)	•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 ;
```



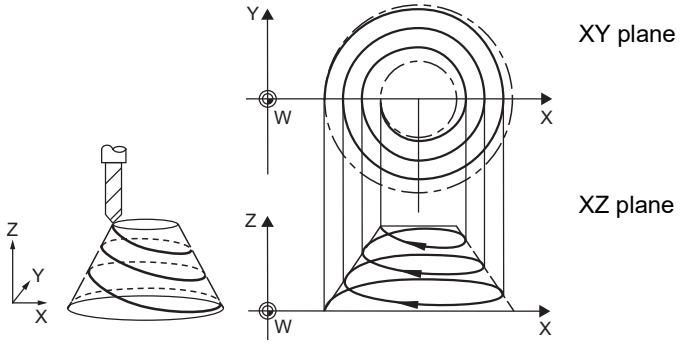
(S) Start point (E) End point (C) Center

(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

- Interpolation vector in the middle of a tool path is not assured.
- 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.
- It cannot be issued as an arc command for geometric IB.
- If the spiral/conical interpolation is commanded to the second geometric block, the program error (P33) occurs.
- A command cannot be issued to the geometric IB blocks even when the command format type 2 (G02/G03) is being designated.

(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.

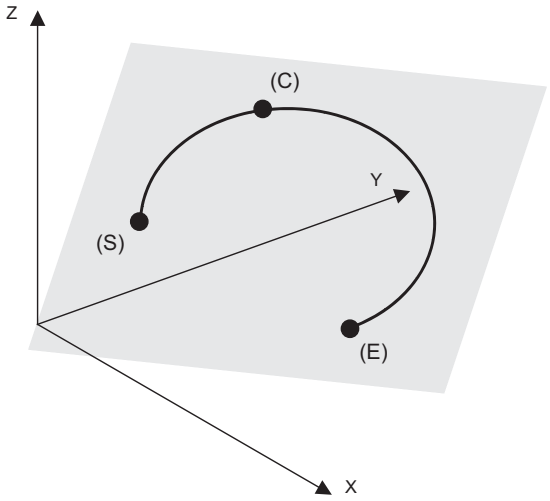
6.15 3-dimensional Circular Interpolation; G02.4, G03.4

Function and purpose

To issue a circular command on 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 which is uniquely 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, the program error (P39) occurs.

The speed command during 3-dimensional circular interpolation is the tangent speed on the arc.



(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 (The rotation direction cannot be specified.)
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 (Can be omitted.)

(1) The G02.4 and G03.4 operations are the same. (The rotation direction cannot be specified.)

(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 addresses in the block can 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 the 3-dimensional circular interpolation is commanded, 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) interpolates to the command point when the axis moves from the start point to intermediate point. The arbitrary axis designated in the end point command block (2nd block) interpolates to the command point when the axis moves 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 or equal to the number of simultaneous contour control axes.

(5) When the 3-dimensional circular interpolation is commanded while the incremental command is valid, designate the relative position of the intermediate point with respect to the start point in the intermediate point designation block. In the end point designation block, designate the relative position of the end point with respect to the intermediate point.

Detailed description

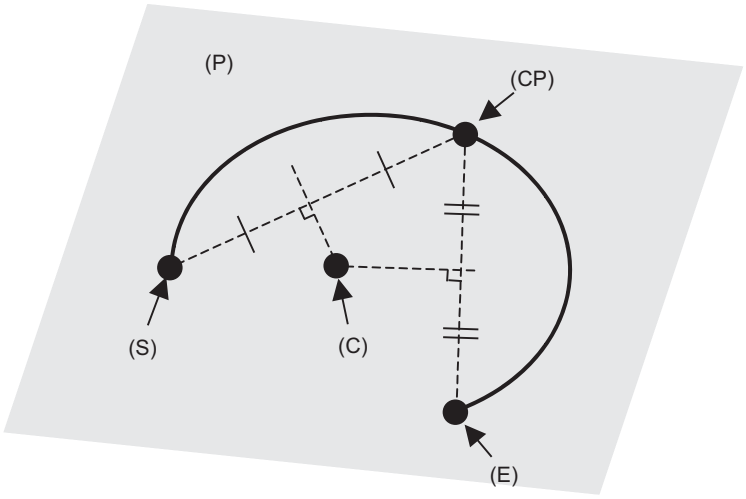
Designating intermediate point and end point

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

Linear interpolation is applied when the end point match the start point in the 3-dimensional circular interpolation command (refer to "When linear 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 the start point and the 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 linear interpolation is applied.
If the intermediate point is near the start point or the end point, an error may occur when the center coordinate of the arc is calculated.

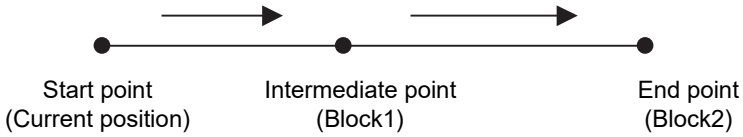
When linear interpolation is applied

In the following cases, linear interpolation is applied without executing 3-dimensional circular interpolation.

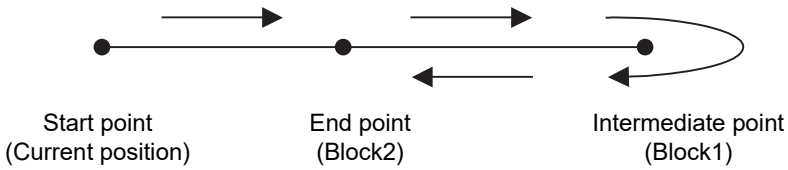
(1) When the start point, intermediate point, and end point are on the same line (refer to the following figure)
(If the end point exists between the start point and the intermediate point, axes move in the order of the start point, intermediate point, and end point.)

(2) When two of the start point, intermediate point and end point match
(Linear 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 the order of the start point, intermediate point, and end point.)

[When linear interpolation is applied]



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



Even if the end point exists between the start point and intermediate point, axes move in the order of the 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 the 01 group. Therefore, the command remains valid until another G command in the 01 group is issued. When the 3-dimensional circular interpolation commands are carried out continuously, the end point of the previous 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
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
G43.1	Tool length compensation along the tool axis	P75
G43.4/G43.5	Tool center point control	P941
G43.7	Tool position 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 cycles	P75

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

G code modal	Function name	Program error
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
G43.1	Tool length compensation along the tool axis	P75
G43.4/G43.5	Tool center point control	P942
G43.7	Tool position 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

Commands other than G code that causes a program error when commanded simultaneously with 3-dimensional circular interpolation

Command code	Function name	Program error
,C / ,R	Corner chamfering/Corner R	P75

Functions that cannot be used

If any of the following functions is used in 3-dimensional circular interpolation, an alarm occurs.

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

Graphic check

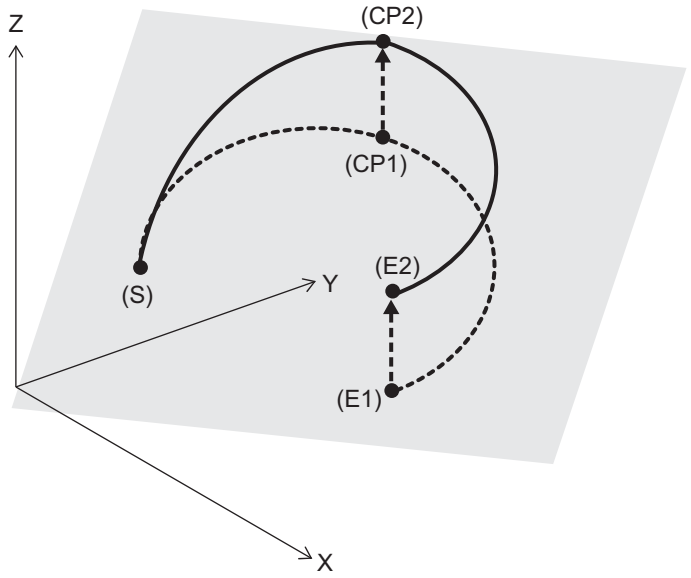
The path of 3-dimensional circular interpolation during the graphic check is drawn as linear in each range from the start point to the intermediate point and from the intermediate point to the end point.

Program restart

Restart search cannot be performed on blocks in which 3-dimensional circular interpolation is performed. Doing so may cause the program error (P49).

Tool length offset

To use 3-dimensional circular interpolation in combination with the tool length offset function, complete the tool compensation operation (movement of tool length and wear compensation amount) before starting 3-dimensional circular interpolation.
If 3-dimensional circular interpolation is commanded while the tool compensation operation is not completed, the arc path (refer to the following figure) is such that the start point coordinates of the arc are uncompensated and the intermediate point and end point coordinates are located at the compensated positions.



(S) Start point (uncompensated)
(E1) End point (before compensation)
(E2) End point (after compensation)
(CP1) Intermediate point (before compensation)
(CP2) Intermediate point (after compensation)

Arc path

Arc path without compensation

Tool compensation amount

To cancel the tool length offset during the 3-dimensional circular interpolation modal, use the G49 (cancel) command. If "G43 H0" is commanded, the program error (P75) occurs, and the tool length offset cannot be canceled.

High-speed machining mode II, High-accuracy control, High-speed high-accuracy control II/III

The high-speed machining mode II is temporarily canceled while 3-dimensional circular interpolation is being executed. Both of the high-speed high-accuracy control II and III function as the high-accuracy control mode.
The SSS control is temporarily canceled while 3-dimensional circular interpolation is being executed.

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



Precautions

(1) If this command is executed with a single block enabled, a block stop is carried out at the intermediate 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 micro 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

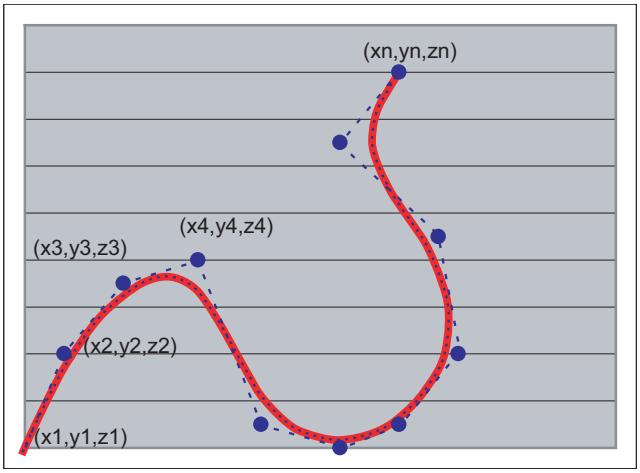
(1) Designate the stage P for the 1st block of NURBS interpolation.

(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.

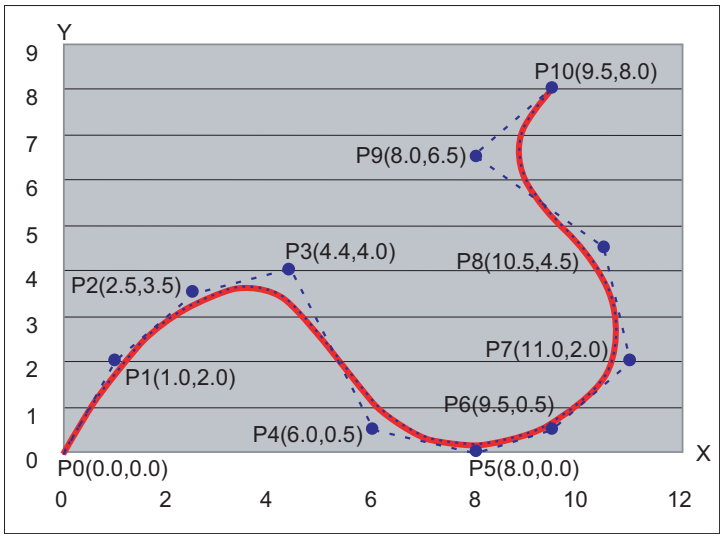
G06.2 Pp Km Xxn Yyn Zzn R1.0

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.

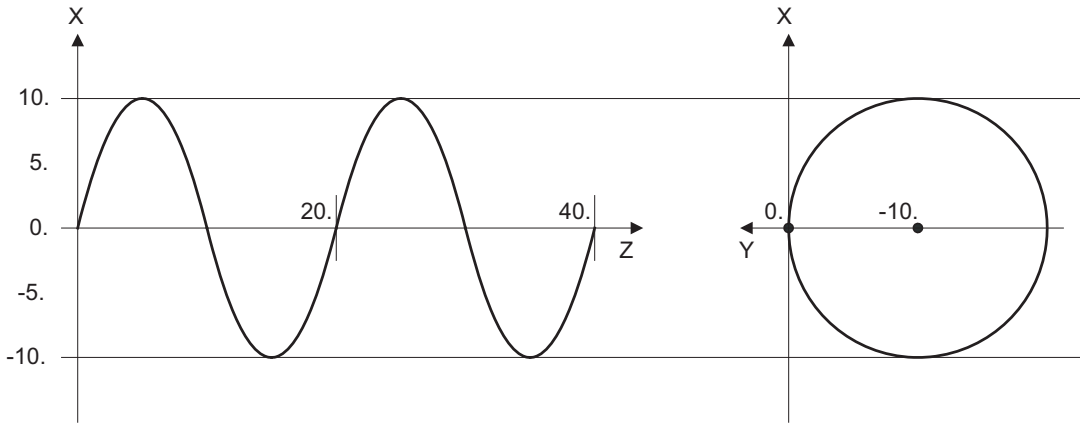
Type	Mode in which NURBS interpolation can be commanded
G group 0	High-speed high-accuracy control II (G05 P10000) High-speed high-accuracy control III (G05 P20000)
G group 5	Asynchronous feed (G94)
G group 7	Tool radius compensation cancel (G40)
G group 8	Tool length offset +/-(G43/G44) Tool length 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

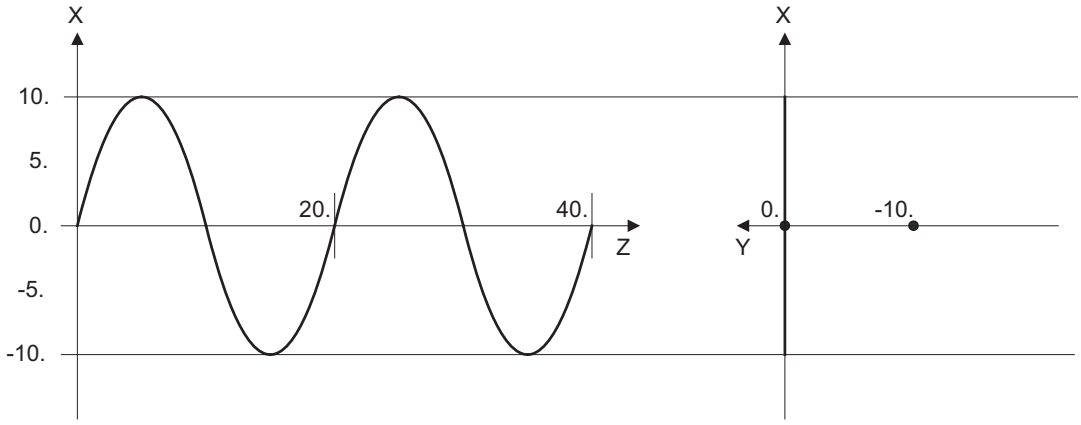
Function and purpose

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 α0 ;” to “G07 α1 ;”, α axis will be the hypothetical axis.

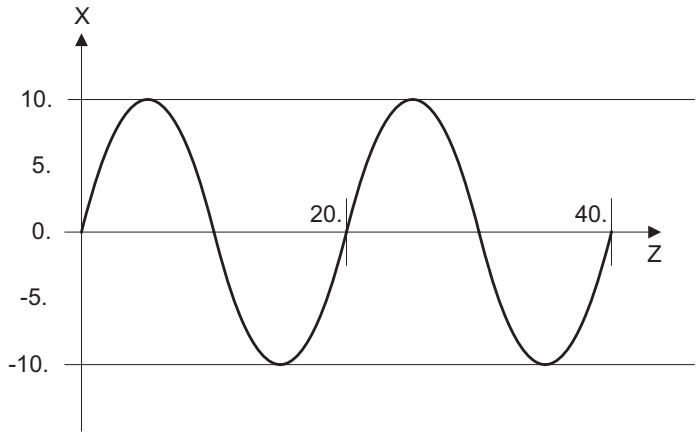
(2) Any axis among the NC axes can be designated as the hypothetical axis.

(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.

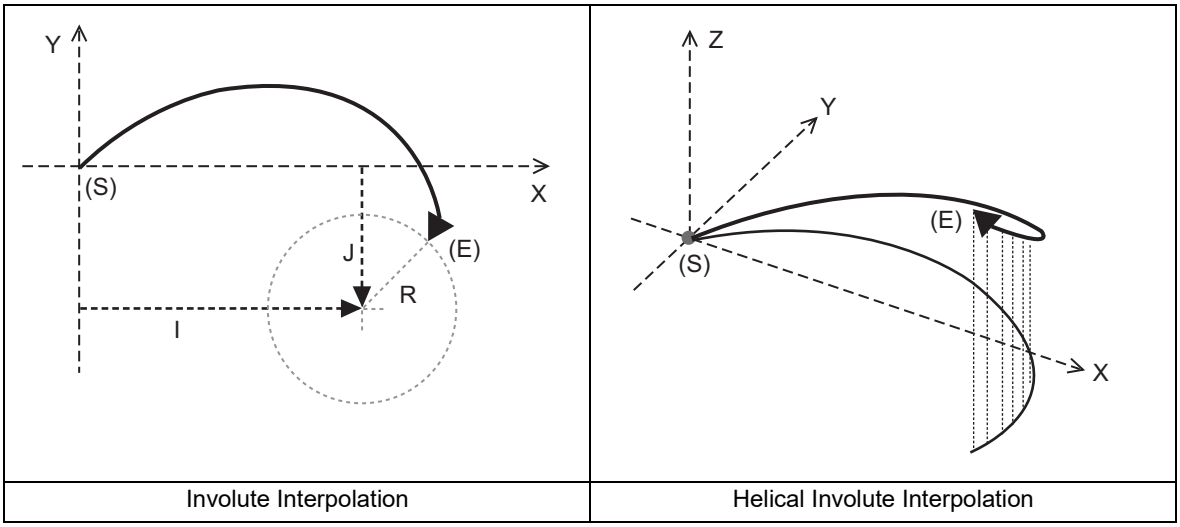
(7) When the hypothetical axis cancel is commanded to the actual axis, no error will occur and the axis remains as the actual axis.

(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. Also, a tool can helically travel while moving 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 micro 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 rate clamping during involute interpolation".



Involute Interpolation | Helical Involute Interpolation

(S) Start point
(E) End point

——— Program command path

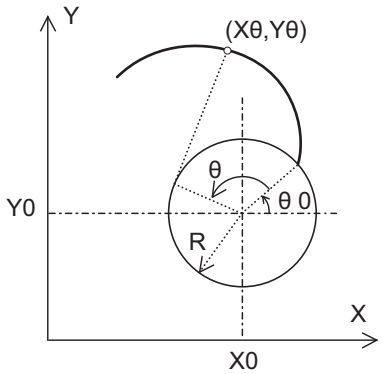
———— XY plane projection path in command program

Involute curve is obtained with the following expression:

$X(\theta) = R\{\cos(\theta+\theta0) + \theta*\sin(\theta+\theta0)\} + X0$

$Y(\theta) = R\{\sin(\theta+\theta0) - \theta*\cos(\theta+\theta0)\} + Y0$

Circle in the right figure is the base circle.



Unless otherwise stated, the operation on the designated plane in the helical involute interpolation is the same as the normal involute interpolation.

Command format

Involute Interpolation

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	Incremental position from start point to center of base circle (X axis)
J	Incremental position from start point to center of base circle (Y axis)
K	Incremental position from start point to 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.

Helical Involute Interpolation

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

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

G02.2 (G03.2) Y__ Z__ α__ 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)
α	Linear axis end point For α, command the linear axis name. If it is used for a rotary axis, the program error (P33) occurs. Multiple linear axes can be commanded.
I	Incremental position between start point and center of base circle (X axis)
J	Incremental position between start point and center of base circle (Y axis)
K	Incremental position 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

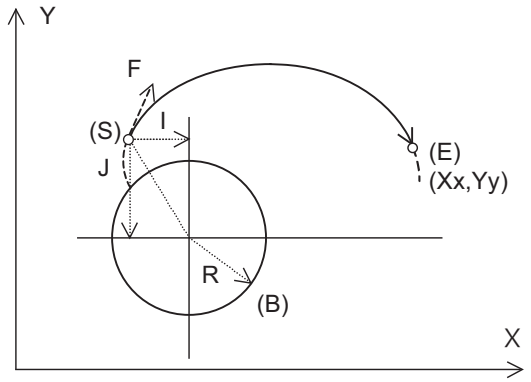
The following information is required to perform the involute interpolation:

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

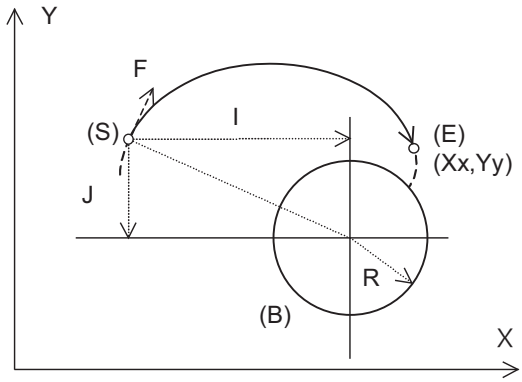
Details of operation (Involute interpolation)

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

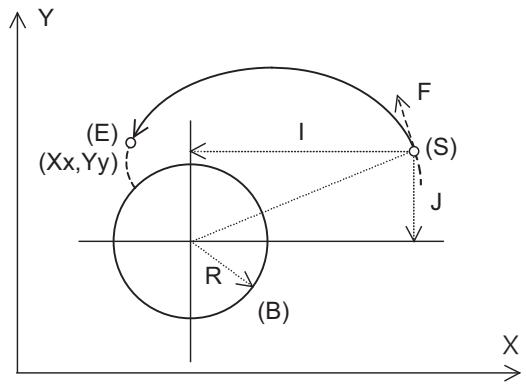
Operation example (G17 plane)



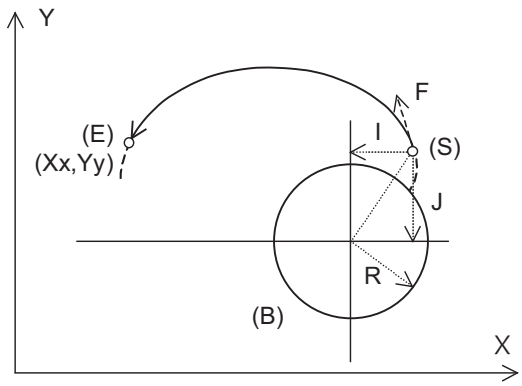
G02.2 (When moving away from the base circle)



G02.2 (When moving toward the base circle)



G03.2 (When moving toward the base circle)



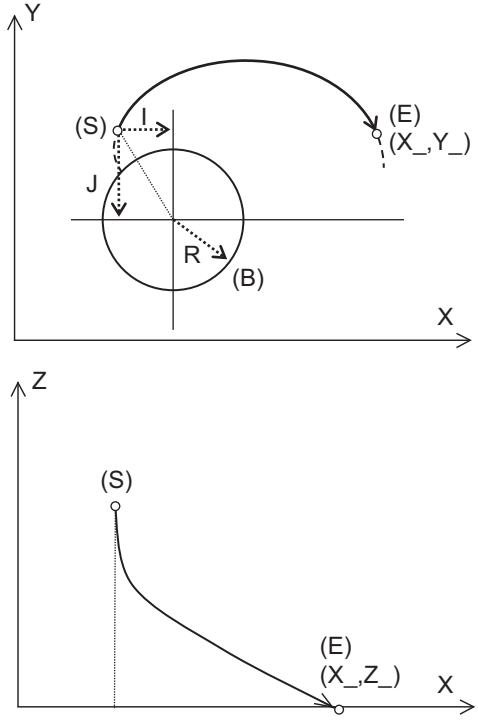
G03.2 (When moving away from the base circle)

(S) Start point
(E) End point
(B) Base circle

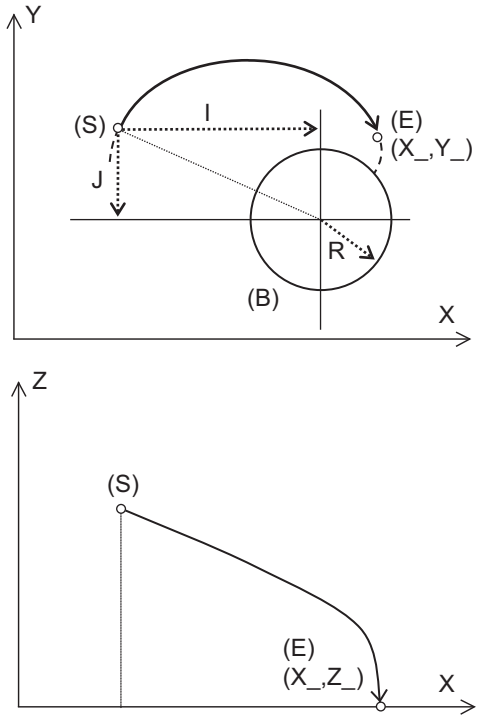
Details of operation (Helical involute interpolation)

The program helically interpolates the involute curve while performing involute interpolation on the selected plane. Plane selection is same as the circular command.

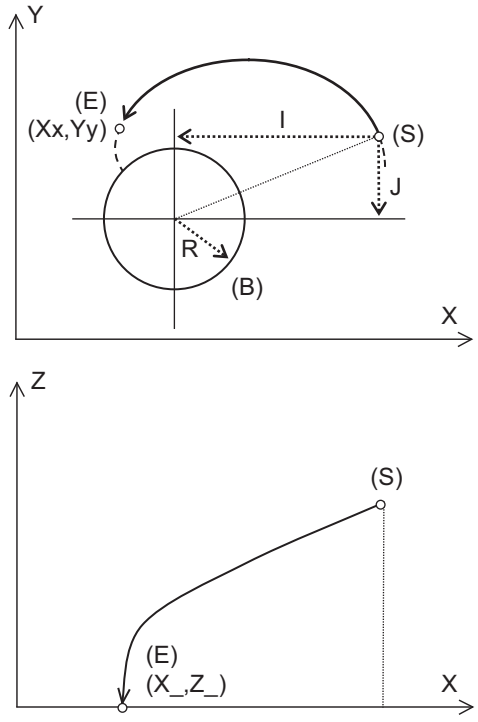
Operation example (When the Z axis is commanded in the negative direction on the G17 plane)



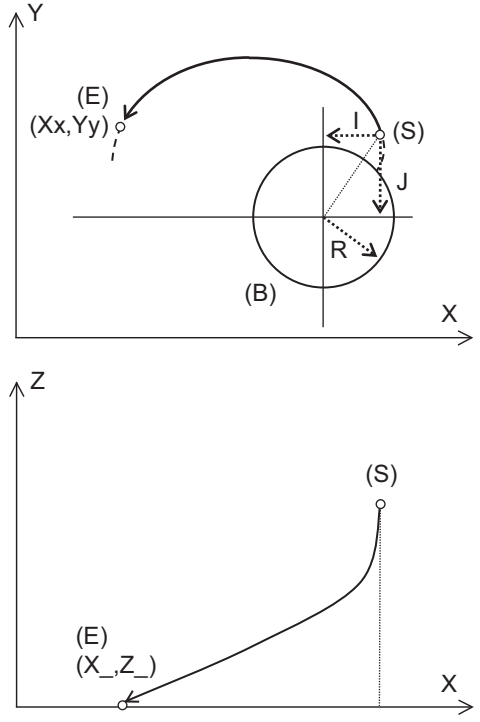
G02.2 (When moving away from the base circle)



G02.2 (When moving toward the base circle)



G03.2 (When moving toward the base circle)



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

(S) Start point
(E) End point
(B) Base circle

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

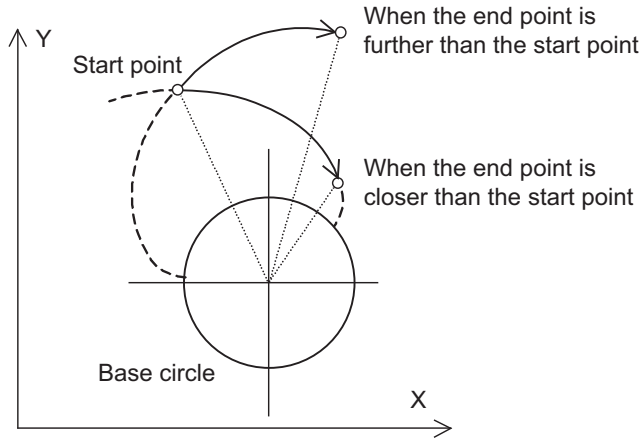
The start point of the involute interpolation is the current position.

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 position 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) are not commanded or both are "0" (equal to the start point), the program error (P33) occurs.

Feedrate

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

Tool radius compensation

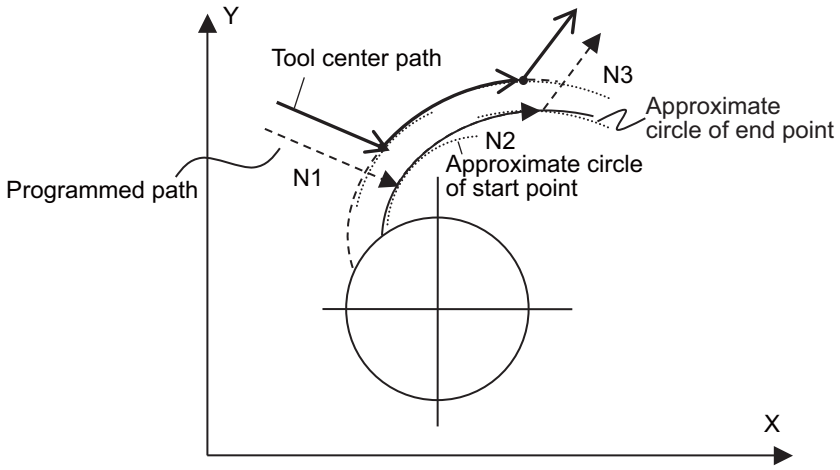
When using tool radius compensation, move to the starting position with the G41 or G42 command, and then issue the involute interpolation command (G02.2 or G03.2). Issue the G40 command after moving to the retract position with G01 or G00.

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". 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 or helical involute interpolation modal, the program error (P151) occurs.

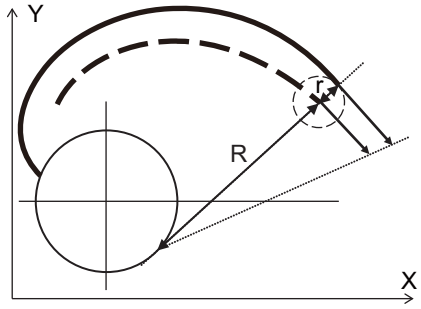
During the involute interpolation or helical involute interpolation, interference avoidance cannot be performed. Even when the parameter "#8102 COLL. ALM OFF" is "1", the program error (P153) occurs if an interference occurs.

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 speedat 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.



R: Curvature radius of tool center
r: Tool radius compensation amount
Programmed path
Tool center path

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. (This 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.

[Involute interpolation acceleration rate clamp (High-accuracy control mode only)]

In the involute interpolation, even when the tool moves at a constant speed in the tangential direction, the acceleration rate 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 rate set by the parameter or below according to the curvature radius in high-accuracy control mode.
The allowable acceleration rate is determined by the following expression:

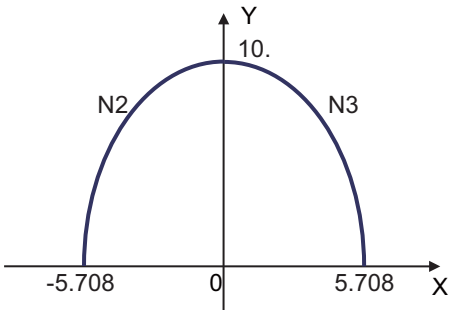
(Allowable acceleration rate) = (Maximum speed)/(Involute interpolation allowable acceleration rate)

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

(Clamp speed) = √ (Curvature radius) x (Allowable acceleration rate)

(Example) In the case of the following conditions:

Maximum speed "#1206 G1bF": 30000 (mm/min)
Involute interpolation allowable acceleration rate "#1559 IvAMax": 600 (ms)
Involute interpolation minimum feedrate "#1560 IvFMin": 1000 (mm/min)




```
:
G00 X-5.708 Y0.;
N1 G61.1;
N2 G02.2 X0. Y10. I15.708 J10. R10. F3000;
N3 G02.2 X5.708 Y0. I-10. J0. R10.;
G64;
:
```

Allowable acceleration rate is as follows:

30000 (mm/min)/600 (ms) ≈ 833.333 (mm/s^2)

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

√2 (mm) × 833.333 (mm/s^2) ≈ 2449.489 (mm/min)

(1) When "0" is set in "#1560 IvFMin", the involute interpolation acceleration rate 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.

Helical involute interpolation speed designation

Whether the normal speed designation or the involute plane component speed designation is valid depends on the MTB specifications.

#1235 set07/bit0	Meaning
0	Resultant speed designation of all the commanded axes
1	Involute plane component speed designation

Involute interpolation override is valid regardless of the setting of this parameter.
Involute interpolation acceleration rate clamp is performed for the speed component on the involute plane.

Program example

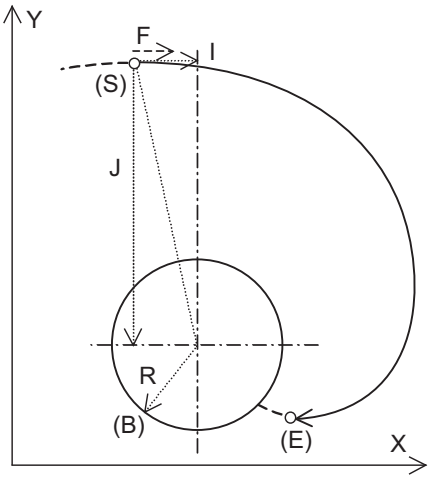
The program example of the absolute command on G17 plane is as below:

Involute Interpolation

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

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

(S) Start point
(E) End point
(B) Base circle

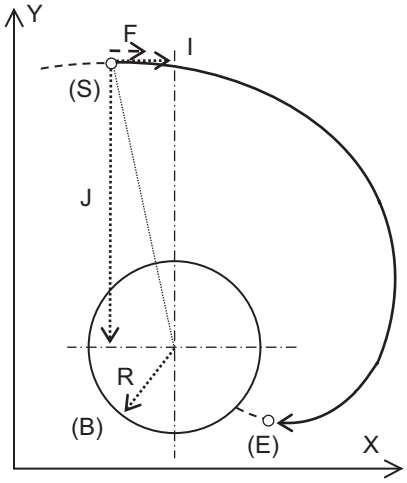


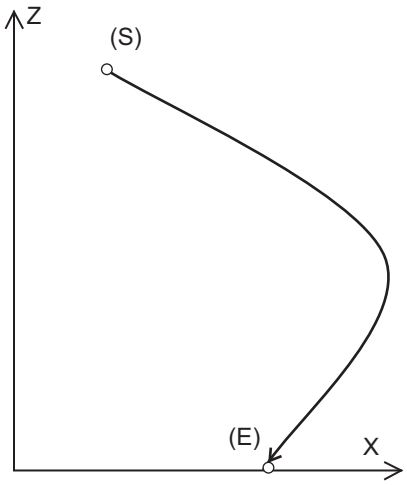
Helical Involute Interpolation

G02.2 X223. Y34. Z-500. I92. J-416. R100. F500;

Start point of involute interpolation	(50., 550.)
Linear axis (X axis) start point	0.
End point of involute interpolation (X, Y)	223., 34.
Linear axis end point (Z)	-500.
Incremental position between start point and center of base circle (I, J)	92., -416.
Base circle radius (R)	100.
Feedrate (F)	500

(S) Start point
(E) End point
(B) Base circle





Relationship with other functions

Commands that cannot be issued in involute interpolation modal or helical involute interpolation modal

When the following commands are issued in involute interpolation or helical 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/nose 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
G43.7	8	Tool position compensation
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
G13.1, G113 (*3)	21	Polar coordinate interpolation OFF

When the following functions are used in involute interpolation modal or helical 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 tool 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.

(*3) G13.1 or G113 in the polar coordinate interpolation mode causes the program error, but does not cause the error in the polar coordinate interpolation cancel mode.

Mode in which involute interpolation or helical involute interpolation cannot be commanded

When the involute interpolation or helical involute interpolation (G02.2, G03.2) 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, G42	7	Nose R compensation
G41.2, G42.2	7	3-dimensional tool radius compensation (Tool's vertical-direction compensation)
G43.4 G43.5	8	Tool center point control
G43.7	8	Tool position compensation
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	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

When the involute interpolation or helical involute interpolation (G02.2, G03.2) is commanded while the following are used, a program error occurs.

- Mirror image by parameter setting
- Mirror image by external input

Precautions and restrictions

(1) This function does not support SSS control. Even when "#8090 SSS ON" is "1" (ON), the involute block operates with SSS control OFF. It decelerates once before and after the involute block.

(2) Even when "#1572 cirorp" is "1" (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 micro segment processing capability is the same as the speed in the normal mode.

(4) This function does not support graphic check. The start point and end point are traced linearly in the involute block.

(5) G68 is a G code that is common to the coordinate rotation by program and the 3-dimensional coordinate conversion. The involute interpolation can be commanded during the execution of the coordinate rotation by program; however, when it is commanded in 3-dimensional coordinate conversion, a program error occurs.

(6) When this function is carried out by selecting a plane other than the one designated with "#8621 Coord rot plane (H)" and "#8622 Coord rot plane (V)" during the execution of the coordinate rotation by program, a program error occurs.

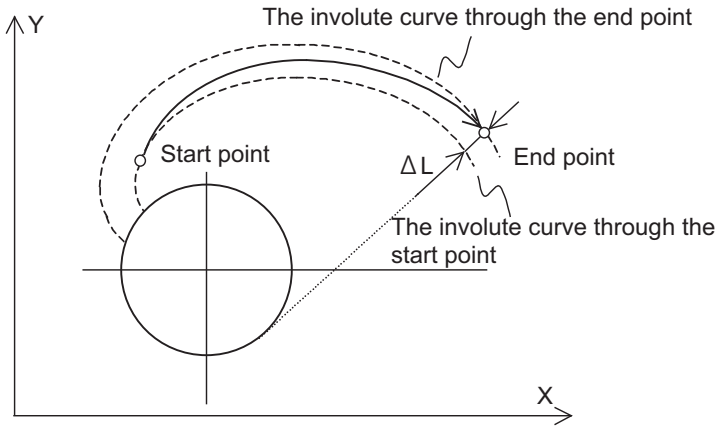
(7) When you carried out the coordinate rotation by program or the 3-dimensional coordinate conversion, command the involute interpolation after performing positioning or linear interpolation with the absolute position for the two axes within the planes to which coordinate rotation was performed. If the involute interpolation is commanded without performing positioning or linear interpolation, a program error may occur.

(8) The involute interpolation does not support reverse run. If the reverse run is performed to this function, a program error occurs.

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 Invlute 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 Invlute 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.

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 times of rotations is obtained by the angle indicated with θ in the calculation formula of the involute curve in the "Function and purpose" section. (θ ≤ 360° × 100)

$X(\theta) = R\{\cos(\theta+\theta0) + \theta*\sin(\theta+\theta0)\} + X0$

$Y(\theta) = R\{\sin(\theta+\theta0) - \theta*\cos(\theta+\theta0)\} + Y0$

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

Cutting override

When involute interpolation override is combined with cutting override, it is performed before cutting override.

7

Feed Functions

7.1 Rapid Traverse Rate

7.1.1 Rapid Traverse Rate



Function and purpose

The rapid traverse rate is set with the parameter "#2001 rapid" for each axis.
During high-accuracy control mode, the dedicated rapid traverse rate (parameter "2109 Rapid(H-precision)") is applied.
Override can be applied to the rapid traverse rate using the external signal supplied.

The rapid traverse rate is effective for the following commands.

G00, G27, G28, G29, G30, G60

The high-accuracy control mode rapid traverse rate is effective for the following commands.

- High-accuracy control
- High-speed high-accuracy control I
- High-speed high-accuracy control II
- High-speed high-accuracy control III
- SSS control
- High-accuracy spline control

Note

(1) For spindle position control (spindle/C axis control), the upper limit of rapid traverse rate is different. Refer to the "Parameter Manual" for the setting range.

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.

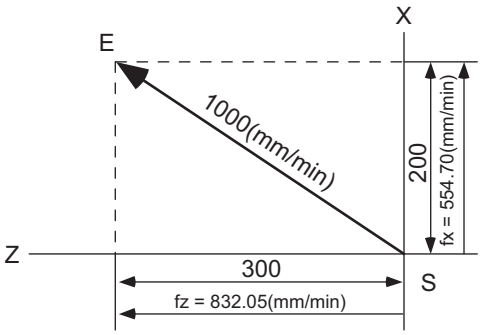
(3) ",F" command in the feed per revolution (G95) mode will also be considered a feed per minute feedrate.

(4) The motion of the ",F" command varies depending on the status of parameter "#1086 G0Intp".

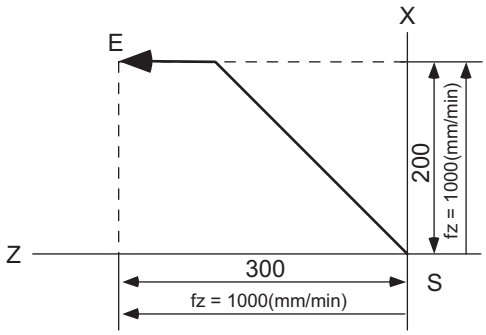
"#1086 G0Intp"	Handling of ",F"command
OFF (see figure shown at below left)	Handled as an interpolation speed.
ON (see figure shown at below right)	Handled as a commanded speed for each axis.

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.
:	

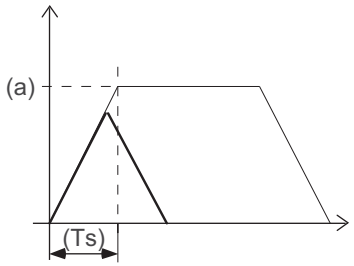
(2) Speed command for the movement to the initial point of the hole position for the drilling cycle (for the tapping cycle of machining center system)

:	
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 X 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-gradient acceleration/deceleration

When ",F" is specified, constant-gradient 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. When the ",F" command has not been issued, the parameter "#2001 rapid" (Rapid traverse rate) setting is applied to the feedrate. When the ",F" command has been issued, the ",F" command is applied to the feedrate.



——— When the travel distance is short
——— When the travel distance is long

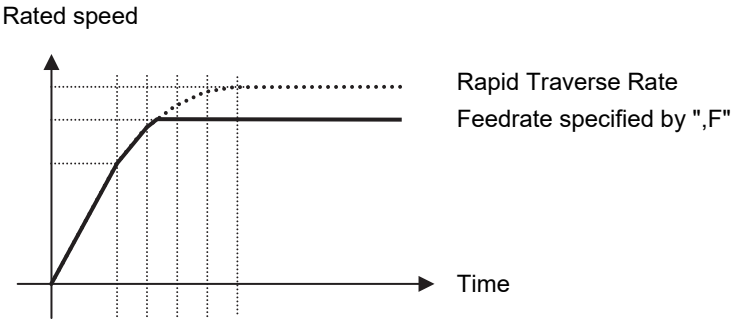
(a) Rapid traverse rate by #2001 or ",F" command speed
(Ts) Time constant

Rapid traverse constant-gradient multi-step acceleration/deceleration

The rapid traverse is performed at the feedrate specified with ",F" address.

The feedrate is accelerated or decelerated in accordance with the pattern of acceleration rate calculated from the following parameters:

- "#2001 rapid" (Rapid traverse rate)
- "#2151 rated_spd" (Rated speed)
- "#2153 G0t_rated" (G0 time constant up to rated speed)
- "#2152 acc_rate" (Acceleration rate in proportion to the maximum acceleration rate)



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

This function specifies the feedrate of the cutting commands, and a feed amount per spindle rotation or feed amount per minute is commanded. Once commanded, it is stored in the memory as a modal value. The feedrate modal is cleared to zero only when the power is turned ON. The maximum cutting feedrate is clamped at the value of cutting feedrate clamp parameter "#2002 clamp".
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.

During high-accuracy control, high-speed high-accuracy control I/II/III, high-accuracy spline control or SSS control, the cutting feed clamp speed for the high-accuracy control (parameter "#2110 Clamp (H-precision)") is applied.

- If the cutting feed clamp speed for the high-accuracy control mode is set to "0", the cutting feedrate will be clamped at the cutting feed clamp speed.
- The cutting feedrate is clamped at the cutting feed clamp speed for the high-accuracy control mode specified with the parameter.

(Example) Feed per minute (asynchronous feed)

	Feedrate	
G01 X100. Y100. F200 ;	200.0 mm/min	F200 or F200.000 gives the same rate.
G01 X100. Y100. F123.4 ;	123.4 mm/min	
G01 X100. Y100. F56.789 ;	56.789 mm/min	

Cutting feed clamp speed setting range (when input setting unit is 1 μm)

Command Mode	Feedrate command range	Remarks
mm/min	0.001 to 1000000	
inch/min	0.0001 to 100000	
°/min	0.001 to 1000000	

Note

(1) The program error (P62) occurs when there is no F command in the first cutting command (G01, G02, G03) after the power has been turned ON.

(2) For spindle position control (spindle/C axis control), the upper limit of cutting feed clamp speed is different. Refer to the "Parameter Manual" for the setting range.

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 f1digt" 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

Feed per minute (mm/min) (asynchronous feed)

G94;

Feed per revolution (mm/rev) (synchronous feed)

G95;



Detailed description

G94/G95 commands are modal commands.

(Example) After the G95 command is assigned, the G95 command is valid until the G94 command or G93 command (inverse time feed) is assigned next.

(1) The F code command range is as follows.

Metric input

Input Setting unit	**B (0.001 mm)**		**C (0.0001 mm)**	
Command Mode	Feed per minute	Feed per revolution	Feed per minute	Feed per revolution
Command Address	F (mm/min)	F (mm/rev)	F (mm/min)	F (mm/rev)
Minimum command unit	1 (=1.000) (1.=1.000)	1 (=0.001) (1.=1.000)	1 (=1.0000) (1.=1.0000)	1 (=0.0001) (1.=1.0000)
Command range	0.001 - 1000000.000	0.001 - 999.999	0.0001 - 1000000.0000	0.0001 - 999.9999

Input Setting unit	**D (0.00001mm)**		**E (0.000001mm)**	
Command Mode	Feed per minute	Feed per revolution	Feed per minute	Feed per revolution
Command Address	F (mm/min)	F (mm/rev)	F (mm/min)	F (mm/rev)
Minimum command unit	1 (=1.00000) (1.=1.00000)	1 (=0.00001) (1.=1.00000)	1 (=1.000000) (1.=1.000000)	1 (=0.000001) (1.=1.000000)
Command range	0.00001 - 1000000.00000	0.00001 - 999.99999	0.000001 - 1000000.000000	0.000001 - 999.999999

Inch input

Input Setting unit	B (0.0001inch)		C (0.00001inch)	
Command Mode	Feed per minute	Feed per revolution	Feed per minute	Feed per revolution
Command Address	F (inch/min)	F (inch/rev)	F (inch/min)	F (inch/rev)
Minimum command unit	1 (=1.0000) (1.=1.0000)	1 (=0.0001) (1.=1.0000)	1 (=1.00000) (1.=1.00000)	1 (=0.00001) (1.=1.00000)
Command range	0.0001 - 100000.0000	0.0001 - 999.9999	0.00001 - 100000.00000	0.00001 - 999.99999

Input Setting unit	D (0.000001inch)		E (0.0000001inch)	
Command Mode	Feed per minute	Feed per revolution	Feed per minute	Feed per revolution
Command Address	F (inch/min)	F (inch/rev)	F (inch/min)	F (inch/rev)
Minimum command unit	1 (=1.000000) (1.=1.000000)	1 (=0.000001) (1.=1.000000)	1 (=1.0000000) (1.=1.0000000)	1 (=0.0000001) (1.=1.0000000)
Command range	0.000001 - 100000.000000	0.000001 - 999.999999	0.0000001 - 100000.0000000	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 × N × OVR ..... (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 multiple 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, an operation error (M01 0105) occurs.

(4) Even in the machine lock status, the feedrate in feed per revolution follows the issued F command.

(5) Under dry run conditions, feed per minute applies and movement results at the manual feedrate (mm/min or inch/min).

(6) The feed mode in the following cases follows the modal (G94/G95).

- Tapping cycle (G84)
- Reverse tapping cycle (G74)

(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 depends on the MTB specifications (parameter "#1074 I_Sync").

7.5 Inverse Time Feed; G93

Function and purpose

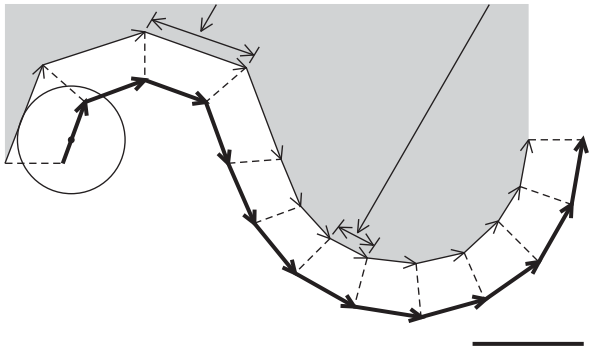
During inside cutting when machining curved shapes with radius compensation applied, the machining speed on the cutting surface becomes faster than the tool center feedrate. Therefore, problems such as reduced accuracy may occur.

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 micro 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

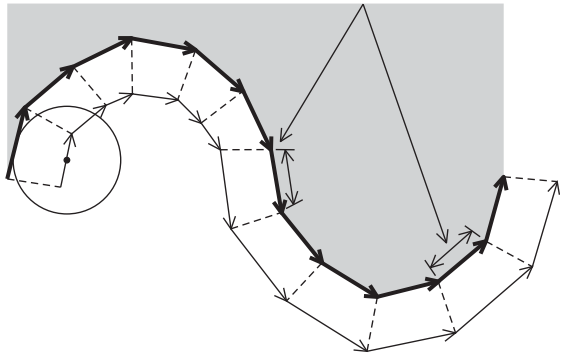


The speed of tool center is commanded, thus the actual speed at the cutting surface may become larger or smaller.

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) / Line segment length (mm)	Cutting point feedrate (inch/min) / Line segment length (inch)
In arc mode (G02, G03) (G02.1, G03.1)		Cutting point feedrate (mm/min) / Start point arc radius (mm)	Cutting point feedrate (inch/min) / 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).

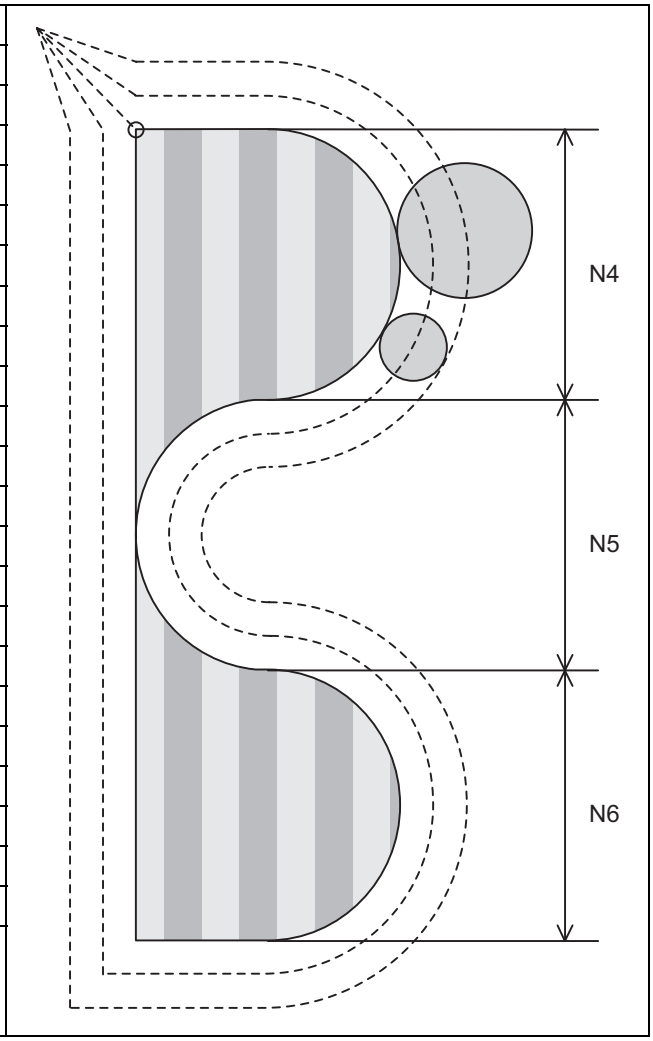
(4) The feedrate of the block inserted in tool radius compensation and corner R/C is the same speed as the feedrate of the block immediately before it.

(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

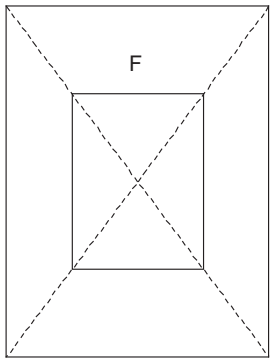
(1) Scaling (G51)

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.

$$F' = \frac{F}{2}$$

F = Feedrate (mm/min) / 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.

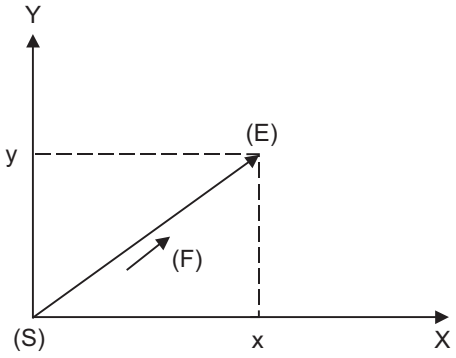
The assignment of the feedrate is described with the following related items.

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:



$$fx = f \times \frac{x}{\sqrt{x^2 + y^2}}$$ ... Feedrate for X axis

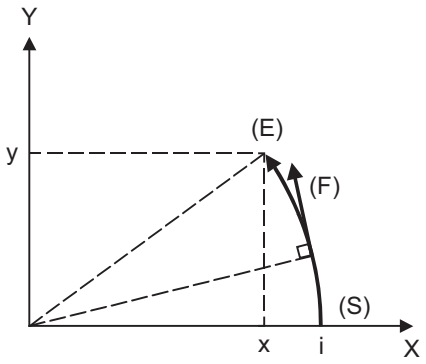
$$fy = f \times \frac{y}{\sqrt{x^2 + y^2}}$$ ... Feedrate for Y axis

(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.



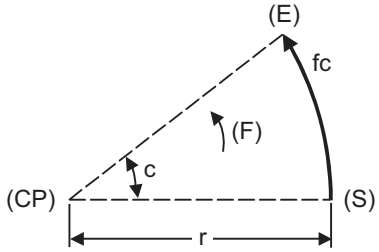
(S) Tool start point
(E) Tool end point
(F) Speed in this direction is "f".

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 rotary axis (C) 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:

$$fc = f \times \frac{\pi \cdot r}{180}$$

Therefore, the feedrate to be designated in the program must be as follows:

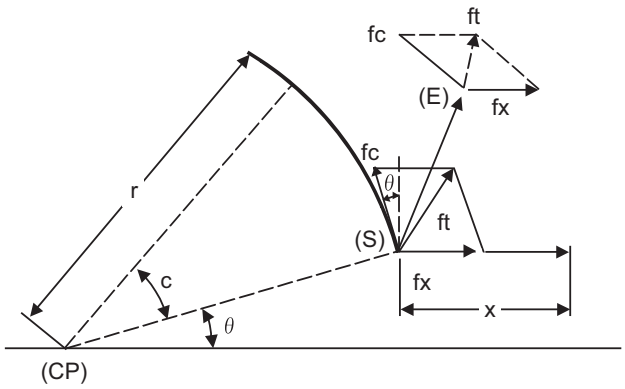
$$f = fc \times \frac{180}{\pi \cdot r}$$

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 the linear axis (X) and the rotary axis (C) are to be controlled simultaneously:

In the figure below, the X axis incremental command value is "x" and the C axis incremental command values is "c":



(S) Tool start point
(E) Tool end point
(CP) Center of rotation

Size and direction are fixed for "fx".
Size is fixed for "fc" but direction varies.
Size and direction vary for "ft".

X axis feedrate (linear speed) "fx" and C axis feedrate (angular speed) "ω" are expressed as:

$$fx = f \times \frac{x}{\sqrt{x^2 + c^2}} \quad \text{..... (1)}$$

$$\omega = f \times \frac{c}{\sqrt{x^2 + c^2}} \quad \text{..... (2)}$$

Linear speed "fc" based on C axis control is expressed as:

$$fc = \omega \times \frac{\pi \times r}{180} \quad \text{..... (3)}$$

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:

$$ftx = -r\sin\left(\frac{\pi}{180}\theta\right) \times \frac{\pi}{180}\omega + fx \quad \text{..... (4)}$$

$$fty = -r\cos\left(\frac{\pi}{180}\theta\right) \times \frac{\pi}{180}\omega \quad \text{..... (5)}$$

Where "r" is the distance (in millimeters) between center of rotation and 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:

$$ft = \sqrt{ftx^2 + fty^2}$$

$$= f \times \frac{\sqrt{x^2 - x \times c \times \mathrm{rsin}\left(\frac{\pi}{180}\theta\right)\frac{\pi}{90} + \left(\frac{\pi \times r \times c}{180}\right)^2}}{x^2 + c^2} \quad \text{..... (6)}$$

Consequently, feedrate "f" designated by the program must be as follows:

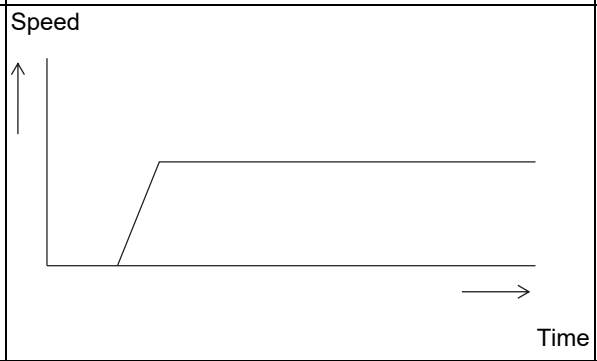
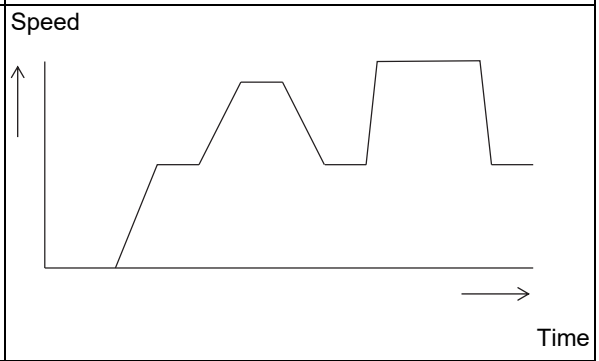
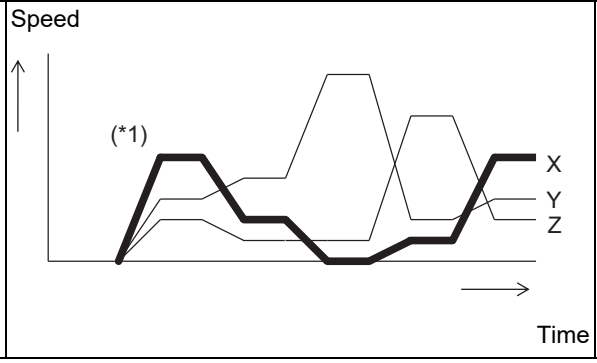
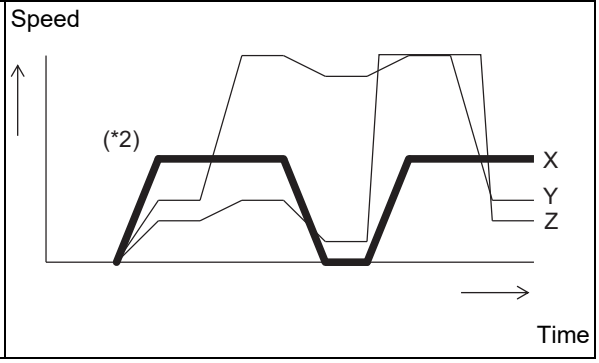
$$f = ft \times \frac{x^2 + c^2}{\sqrt{x^2 - x \times c \times \mathrm{rsin}\left(\frac{\pi}{180}\theta\right)\frac{\pi}{90} + \left(\frac{\pi \times r \times c}{180}\right)^2}} \quad \text{..... (7)}$$

"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 Selection of Axis (Axes) for Feedrate Command; G130

Function and purpose

This function designates the feedrate of a selected axis (using the F command) to perform machining.
When the feedrate of a specific axis fluctuates wildly, it may not be possible to achieve the desired surface finish.
Fluctuation can be suppressed using this function, which can result in improved surface quality.

<table>
<tr><th></th><th>When the function is invalid</th><th>When the function is valid
(with X axis being selected)</th></tr>
<tr><td>Resul-
tant
speed of
all axes</td><td>
</td><td>
</td></tr>
<tr><td>Feedrate
of each
axis</td><td>
</td><td>
</td></tr>
</table>

(*1) Speed differs in each block.

(*2) Speed is kept constant.

Command format

Selection of axis (axes) for feedrate command

G130 Axis name 1 Axis name 2 ... Axis name n ;	
Axis name n	Axis address for which axis (axes) for feedrate command is performed

Selection of axis (axes) for feedrate command cancel

G130;

(1) G130 is a non-modal command of group 0.

(2) G130 must be commanded alone in a block. If it is commanded in the same block with other G codes, a program error (P45) occurs.

(3) Designate the axis name "n" using an axis address set in the parameter "#1013 axname". If the designated axis does not exist, the program error (P32) occurs.

(4) When an axis address enabled with the axis name extension function is specified for the axis name "n", the program error (P32) occurs.

(5) The selection of axis (axes) for feedrate command is canceled with Reset 1, Reset 2, Reset & rewind, or Emergency stop.

Operation example

When using this function, designate the resultant speed for the target axis of the speed command using the F command.
For an axis that is not targeted for the speed command, adjust the speed so that no deviation occurs on the interpolation path.

[When not using the selection of axis (axes) for feedrate command]

:	:
G01 X100. Y150. Z200. F1000;	The F command is issued as the resultant speed of all axes. The travel speed is as follows: X axis: 371.391 (mm/min) Y axis: 557.086 (mm/min) Z axis: 742.781 (mm/min) All-axes resultant speed: 1000.000 (mm/min)
:	:

[When using the selection of axis (axes) for feedrate command]

(1) When selecting only a single axis for the speed command:

:	:
G130 X;	Select the X axis for the speed command.
G01 X100. Y150. Z200. F1000;	The F command is issued as the speed of the X axis. The travel speed is as follows: X axis: 1000.000 (mm/min) Y axis: 1500.000 (mm/min) Z axis: 2000.000 (mm/min) All-axes resultant speed: 2692.582 (mm/min)
:	:

(2) When selecting the linear axes for the speed command in the configuration of two linear axes and one rotary axis:

:	:
G130 XY;	Select a linear axis for the speed command.
G01 X100. Y150. C200. F1000;	The F command is issued as the resultant speed of the linear axes. The travel speed is as follows: X axis: 554.700 (mm/min) Y axis: 832.050 (mm/min) Linear-axis resultant speed: 1000.000 (mm/min) C axis: 1109.400 (mm/min) All-axes resultant speed: 1493.576 (mm/min)
:	:

(3) When switching the rotary axis, which is currently selected for the speed command, to another axis:

:	:
G130 C;	Select the C axis for the speed command.
G01 X100. Y150. C200. F1000;	The rotation speed is as follows: C axis: 1000.000 (mm/min)
G130 XY;	Select a linear axis for the speed command. (The previously selected C axis is canceled.)
G01 X200. Y300. C400. F800;	The travel speed is as follows: Linear-axis resultant speed: 800.000 (mm/min)
G130;	Cancel of axis (axes) for feedrate command
G01 X300. Y450. C600. F500;	The travel speed is as follows: All-axes resultant speed: 500.000 (mm/min)
:	:



Relationship with other functions

Relationship with G code group 1

The selection of axis (axes) for feedrate command is only available when G code group 1 is in the following modal.

G code	Function name	Operation in selection of axis (axes) for feedrate command
G00	Positioning	Operates at rapid traverse rate.
G01	Linear interpolation	The F command is issued as the resultant speed of the axis for the speed command.
G02/G03	Circular interpolation Helical interpolation	The F command is issued as the resultant speed of all axes.

If the selection of axis (axes) for feedrate command is applied during a modal other than the above, a program error (P581) occurs.

Functions that cannot be combined with the selection of axis (axes) for feedrate command

The selection of axis (axes) for feedrate command cannot be combined with the functions shown in the table below.

Column A: Operation when the combination function is commanded while the selection of axis (axes) for feedrate command is enabled

Column B: Operation when the selection of axis (axes) for feedrate command is commanded while the combination function is enabled

G code	Function name	A	B
G07	Hypothetical axis interpolation	-	P582
G12	Circular cutting		
G31	Skip		
G31.1-G31.3	Multi-step skip		
G34-G36,G37.1	Special fixed cycle		
G07.1	Cylindrical interpolation	P581	
G12.1/G112	Polar coordinate interpolation	P581	
G41.2/G41.3	3-dimensional tool radius compensation	P162	
G43.4/G43.5	Tool center point control	P942	
G54.4	Workpiece installation error compensation	P545	
G68	3-dimensional coordinate conversion Coordinate rotation by program	P581	
G68.2	Inclined surface machining command	P951	
G70-G89	Fixed cycle	P45	
G93	Inverse time feed	P581	
M98	Figure rotation	P581	
-	Coordinate rotation by parameter	P581	



Precautions

(1) When this function is used, the travel speed does not exceed the clamp speed of each axis.

(2) If all the movement axes are targeted for the speed command, the system performs the same operation as when this function is not used.

(3) When the movement axis is not targeted for the speed command, the F command is handled as the all-axes resultant speed.

(4) The selection of axis (axes) for feedrate command does not support the F 1-digit feed function. If the F 1-digit feed function is used, the speed commanded by F 1-digit command is handled as the resultant speed.

(5) The selection of axis (axes) for feedrate command does not support the dry run function. If the dry run function is used, the manual feedrate is handled as the resultant speed.

(6) The selection of axis (axes) for feedrate command does not support the variable-acceleration pre-interpolation acceleration/deceleration function. When the variable-acceleration pre-interpolation acceleration/deceleration is used, the F command is handled as the all-axes resultant speed.

7.8 Rapid Traverse Constant-gradient Acceleration/Deceleration

Function and purpose

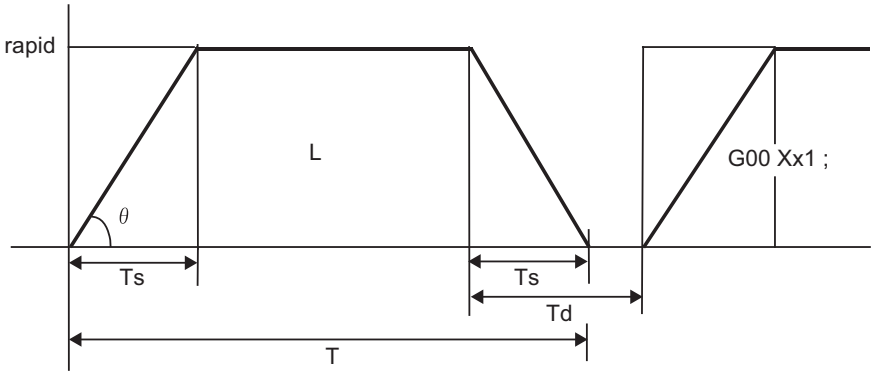
This function performs acceleration and deceleration at a constant gradient during linear acceleration/deceleration in the rapid traverse mode. The constant-gradient acceleration/deceleration method can be more beneficial in reducing cycle time in comparison to the acceleration/deceleration with fixed time constant method.

Detailed description

(1) Rapid traverse constant-gradient acceleration/deceleration are valid only for a rapid traverse (G00) command. Also, this function is effective only when the rapid traverse command acceleration/deceleration mode is linear acceleration/deceleration or soft acceleration/deceleration.

(2) The program format of G00 (rapid traverse command) when rapid traverse constant-gradient acceleration/deceleration is executed is the same as when this function is invalid (acceleration/deceleration with fixed time constant).

When the interpolation distance is long enough for the rapid traverse rate to be achieved



$$T = \frac{L}{rapid} + Ts$$

$$Td = Ts + n$$

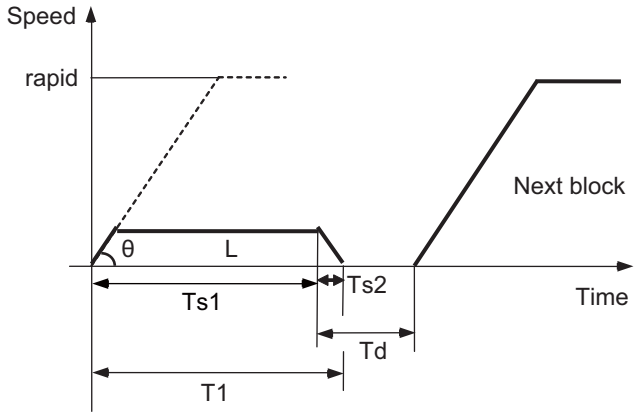
$$\theta = \tan^{-1}\left(\frac{rapid}{Ts}\right)$$

rapid : Rapid traverse rate
Ts : Acceleration/deceleration time constant
Td : Command deceleration check time
θ : Acceleration/deceleration gradient
T : Interpolation time
L : Interpolation distance
n : 0 to 14 ms

When the interpolation distance is so short that the rapid traverse rate is not achieved

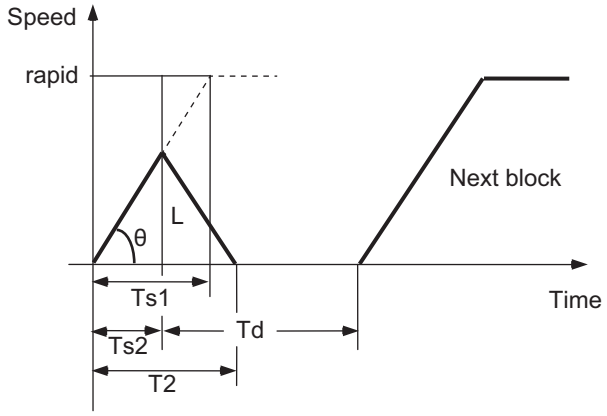
(1) Acceleration/deceleration with fixed time constant:

The gradient is determined by the rapid traverse rate; however, the cycle time is determined by the time constant, and the reaching speed is slower than constant-gradient acceleration/deceleration.



(2) Constant-gradient acceleration/deceleration:

The gradient is determined by the rapid traverse rate and the time constant. The reaching time is determined by interpolation distance; therefore, the cycle time is shorter than acceleration/deceleration with fixed time constant.



$T1 = Ts1 + Ts2$

$T2 = 2 \times \sqrt{Ts1 \times L / rapid}$

$Td = Ts2 + n$

$\theta = \tan^{-1}\left(\frac{rapid}{Ts1}\right)$

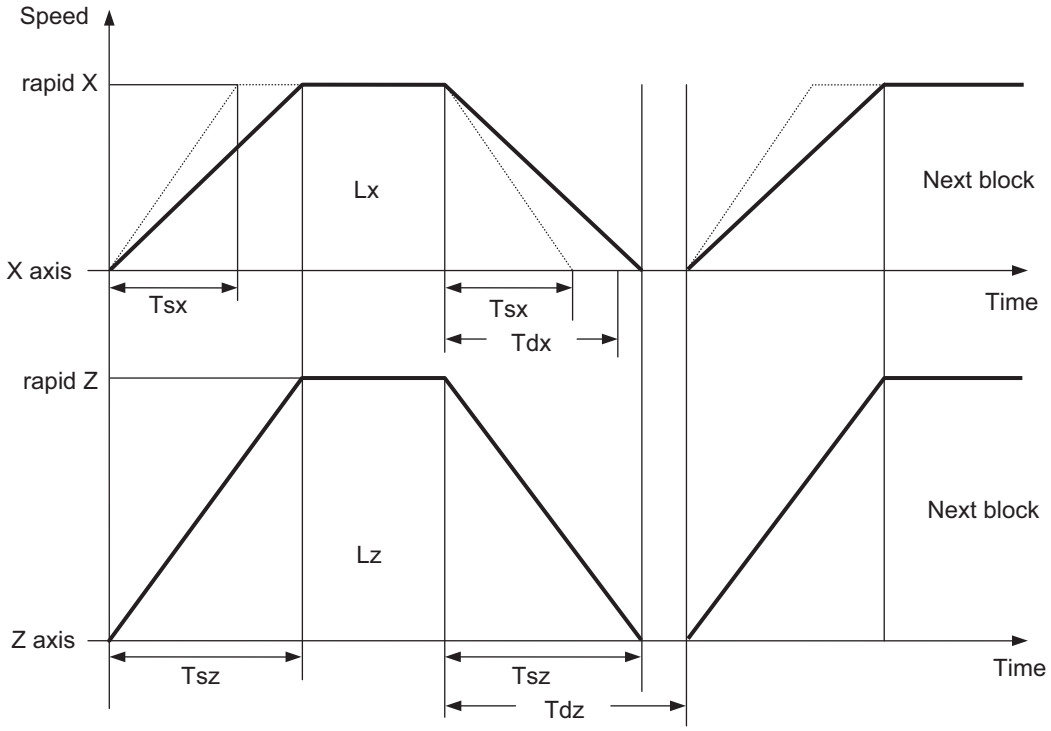
rapid : Rapid traverse rate (Parameter "#2001 rapid")
Ts1 : Acceleration/deceleration time (Parameter "#2004 G0tL")
Ts2 : Acceleration/deceleration time to reach the maximum speed
Td : Command deceleration check time
θ : Acceleration/deceleration gradient
T1 : Interpolation time (Acceleration/deceleration with fixed time constant)
T2 : Interpolation time (Constant-gradient acceleration/deceleration)
L : Interpolation distance
n : 0 to 14 ms

2-axis simultaneous interpolation (When linear interpolation is used, Tsx < Tsz, Lx ≠ Lz)

(1) Determination of deceleration check time for 2-axis interpolation
The time required for the command deceleration check during rapid traverse constant-gradient acceleration/deceleration is the longest one among the deceleration check times of the axes which are commanded simultaneously. The deceleration check times of each axis is determined by the following items; rapid traverse rate (set by the parameter "#2001 rapid"); rapid traverse acceleration/deceleration time constant (set by the parameter "#2004 G0tL"); and interpolation distance (L).

(2) Determination of rapid traverse constant-gradient acceleration/deceleration for 2-axis interpolation
When multi-axis simultaneous interpolation (linear interpolations) is performed during rapid traverse constant-gradient acceleration/deceleration, the longest one among the acceleration/deceleration times of all axes is applied to the other axes which were commanded simultaneously. The acceleration/deceleration time is determined for each axis by the following items; rapid traverse rate, rapid traverse acceleration/deceleration time constant, and the interpolation distance. Consequently, linear interpolation is performed even when the acceleration/deceleration time constant for each axis is different.

(3) Rapid traverse constant-gradient acceleration/deceleration priority parameter
The prioritized time constant is switched by setting the parameter "#1752 cfgPR02/bit4" (Prioritization of G0 constant-gradient acc/dec time constant).
When "#1752 cfgPR02/bit4" is "0", the axis of which the acceleration/deceleration time constant calculated in item (2) is greater than the rapid traverse acceleration/deceleration time constant (#2004) is accelerated/decelerated with the rapid traverse acceleration/deceleration time constant (#2004).
When "#1752 cfgPR02/bit4" is "1", the acceleration/deceleration is always performed according to the acceleration/deceleration time constant calculated in item (2).

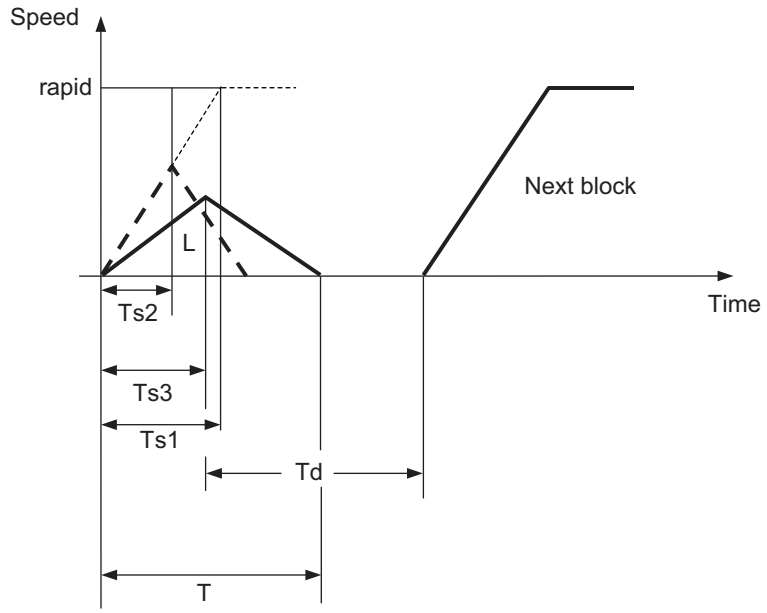


When the result is "Tsx < Tsz", the acceleration/deceleration time (Ts) of this block is set to Tsz (acceleration/deceleration time of the Z axis).
Since "Tdx < Tdz" is obtained as the result, the command deceleration check time (Td) of this block is set to Tdz (command deceleration check time of the Z axis).

Tsx : X axis acceleration/deceleration time
Tsz : Z axis acceleration/deceleration time
Tdx : X axis commanded deceleration check time
Tdz : Z axis commanded deceleration check time
Lx : X axis interpolation distance
Lz : Z axis interpolation distance

When the interpolation distance is so short that the acceleration/deceleration time is shorter than the minimum time constant for constant-gradient acceleration/deceleration

If a minimum time constant for constant-gradient acceleration/deceleration has been set in the parameter, acceleration/deceleration speed is adjusted to prevent the acceleration/deceleration time to reach the speed calculated by interpolation distance from going below the minimum time constant.



$T = 2 \times Ts3$

$Td = Ts3 + n$

rapid : Rapid traverse rate (Parameter "#2001 rapid")
Ts1 : Acceleration/deceleration time (Parameter "#2004 G0tL")
Ts2 : Acceleration/deceleration time to reach the maximum speed
Ts3 : Minimum time for constant-gradient acceleration/deceleration (Parameter "#2198 G0tMin")
Td : Command deceleration check time
T : Interpolation time
L : Interpolation distance
n : 0 to 14 ms

Rapid traverse time constant switching

Use the "Rapid traverse time constant: Switchover 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: Switchover request" signal is turned ON/OFF.

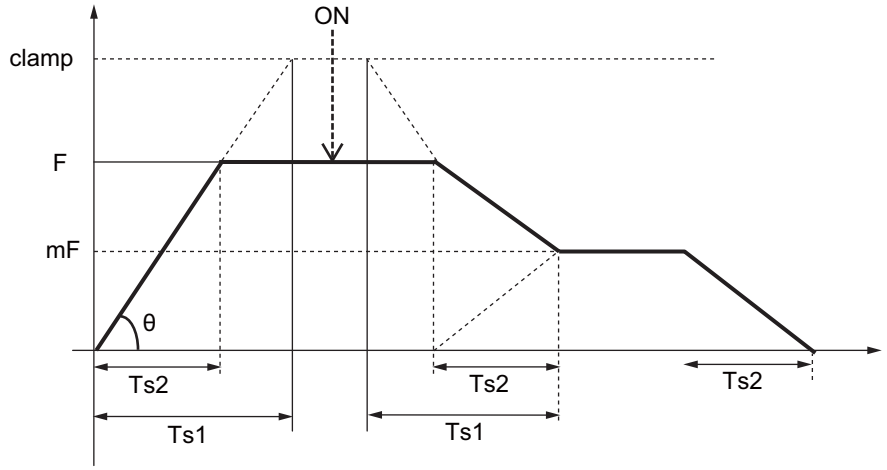
	Basic rapid traverse time constant (signal OFF)	Rapid traverse time constant for switching (signal ON)
Rapid traverse time constant	#2004 G0tL	#2598 G0tL_2
Rapid traverse time constant (primary delay) / 2nd step time constant of soft acceleration/deceleration	#2005 G0t1	#2599 G0t1_2

<Note>
- When #2598 is "0", use #2004. When #2599 is "0", use #2005.
- The time constant switched by the "Rapid traverse time constant: Switchover request" signal is only the rapid traverse time constant or the rapid traverse time constant (primary delay)/2nd step time constant of soft acceleration/deceleration.

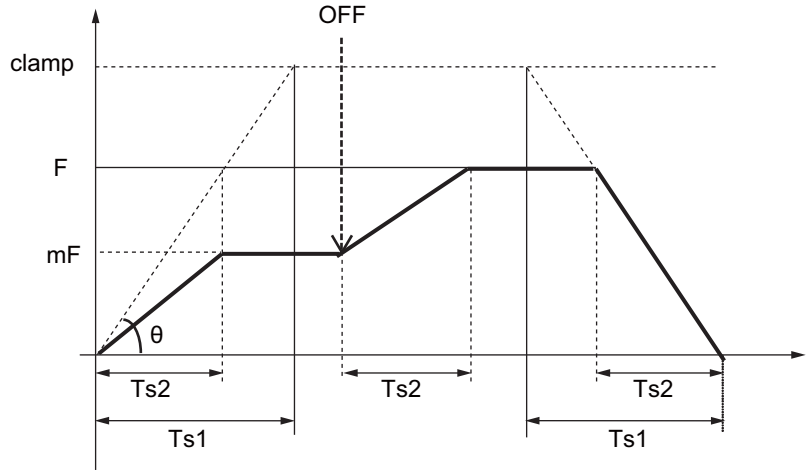
Relationship with other functions

(1) The constant-gradient acceleration/deceleration during dry run is disabled.
When the constant-gradient acceleration/deceleration is valid and the speed is changed by dry run during axis traveling, dry run is disabled in the currently executed block, and the constant-gradient acceleration/deceleration is performed at the same speed as before. Dry run is enabled from the next block, and the manual feedrate is applied. However, the acceleration/deceleration time remains unchanged.
When dry run is enabled and it canceled during axis traveling, dry run is immediately disabled, and the cutting feedrate is applied. After the speed was changed, the constant-gradient acceleration/deceleration is enabled.
When the high-speed machining mode is valid (under G5P2, under G5P10000, etc.), dry run is disabled. However, when the high-speed machining mode is canceled by the positioning command (G00 command) or single block stop command, dry run is enabled.
To enable dry run during the constant-gradient acceleration/deceleration, use the high-accuracy control function (G61.1). Do not use this function.

[When constant-gradient acceleration/deceleration is enabled and the "Dry run" signal is turned ON during axis traveling]



[When dry run is enabled and the "Dry run" signal is turned OFF during axis traveling]



clamp : Maximum cutting feedrate (Parameter "#2002 clamp")
F : Cutting feedrate
mF : Manual feedrate
Ts1 : Acceleration/deceleration time (Parameter "#2007 G1tL")
Ts2 : Acceleration/deceleration time to reach the cutting feedrate
θ : Acceleration/deceleration gradient
ON : "Dry run" signal ON
OFF : "Dry run" signal OFF

(2) The acceleration/deceleration method varies depending on the parameter setting (depending on the MTB specifications).

#1200	#1086	#1752 cfgPR02/bit2	Specified mode (*1)	Unspecified mode
1	1	0	Constant-time acceleration/deceleration method	Constant-gradient acceleration/deceleration method
		1	Constant-gradient acceleration/deceleration method	
1	0	-	Constant-gradient acceleration/deceleration method	
0	1	-	Constant-time acceleration/deceleration method	
0	0	-	Constant-time acceleration/deceleration method	

(*1) This is targeted for acceleration/deceleration in the following modes.

- ♦During inclined surface machining
- ♦During tool center point control
- ♦During workpiece installation error compensation
- ♦During tool center point control

(3) The constant-gradient acceleration/deceleration is valid for the rapid traverse command in the fixed cycle. To switch the rapid traverse time constant in the fixed cycle, issue the "Rapid traverse time constant: Switchover request" during G80 mode.
If the "Rapid traverse time constant: Switchover request" is issued in the fixed cycle, the time constant may not be switched.



Precautions

(1) Rapid traverse constant-gradient acceleration/deceleration is performed only when the motion path of the rapid traverse command is the interpolation type (parameter "#1086 G0Intp" is set to "0").

(2) 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 constant-gradient acceleration/deceleration. In this case, the acceleration time will be "G0tL+G0t1" or "G1tL+G1t1".

(3) When "#2003 smgst" 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.

(4) If a commanded travel distance in a block is small, acceleration/deceleration time becomes quite short when the constant-gradient 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 constant-gradient 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.

(5) When the parameter "#1253 set25/bit2" (Acceleration/deceleration mode change in hole drilling cycle) is set to "1" (constant-gradient, acceleration/deceleration after interpolation), the rapid traverse in the fixed cycle modal is set to the constant-gradient acceleration/deceleration regardless of the setting of the parameter "#1200 G0_acc" (Validate acceleration and deceleration with inclination constant G0).

(6) If any one of the following commands is issued in the next block in which the "Rapid traverse time constant: Switchover request" signal is turned ON, the time constant switching may be delayed.

- ♦G27 (Reference position check)
- ♦G28 to G30 (Reference position return)
- ♦G30.1 to G30.6 (Tool exchange position return)
- ♦G60 (Unidirectional positioning)

7.9 Rapid Traverse Constant-gradient 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-gradient multi-step acceleration/deceleration method makes for improved cycle time because the positioning time is shortened by using the motor ability to its maximum.

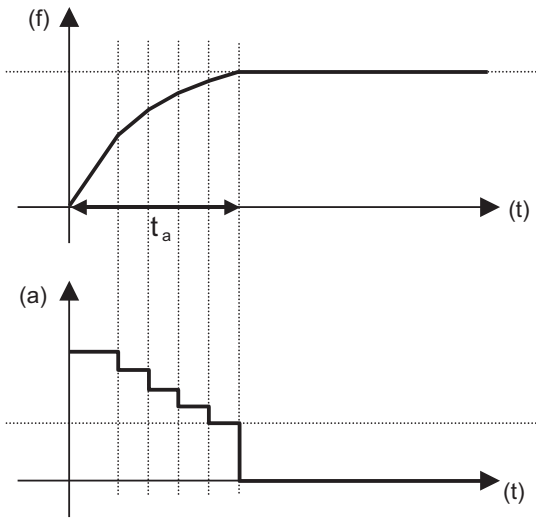
In general, the servomotor has the characteristic that the torque falls in the high-speed rotation range.

In the rapid traverse constant-gradient acceleration/deceleration method, the acceleration rate is treated as constant because this torque characteristic is not considered. So, It is necessary to use a minimum acceleration rate within the used speed range. Therefore, the margin of acceleration rate must be had in a low-speed range. Or if the acceleration rate 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-gradient multi-step acceleration/deceleration method.

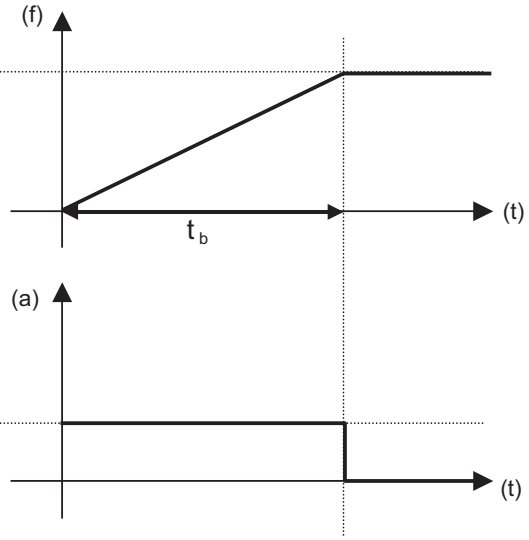
The acceleration/deceleration patterns in the case where rapid traverse constant-gradient multi-step acceleration/deceleration are performed are as follows.

[Rapid traverse constant-gradient multi-step acceleration/deceleration]



Number of steps is automatically adjusted by parameter setting.

[Rapid traverse constant-gradient acceleration/deceleration]



It was necessary to slow down the acceleration rate for high speed rotation.

(f) Speed (t) Time (a) Acceleration rate

Detailed description

Use conditions

(1) The validity of this function depends on the MTB specifications. (Parameter "#1205 G0bdcc")
However, note the following conditions.

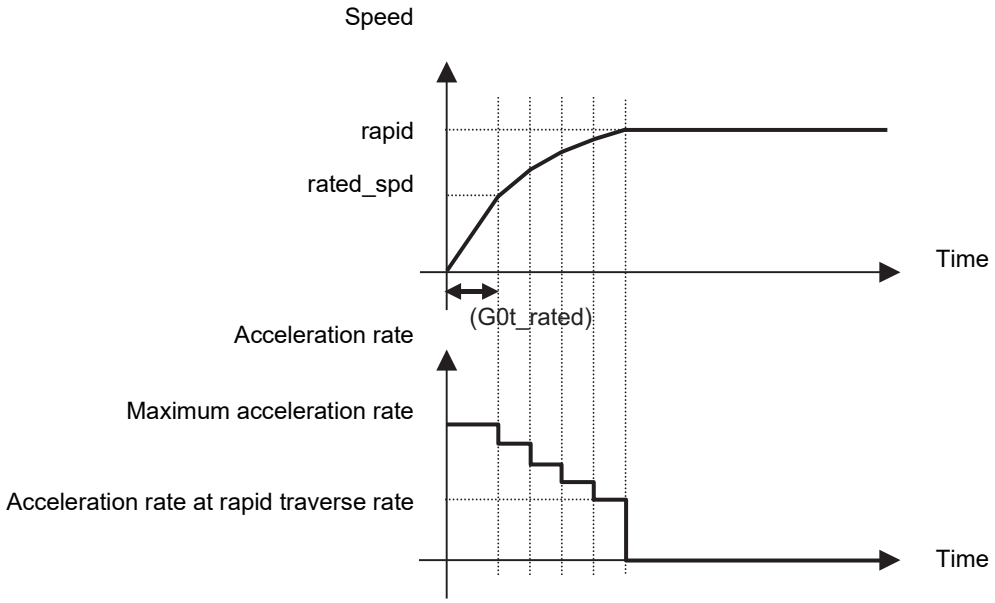
(a) "2" cannot be set to parameter "#1205 G0bdcc" except the 1st part system. When "2" is set to other than 1st part system, an MCP alarm (Y51 0017) will occur.

(b) When there is no specification for the rapid traverse constant-gradient 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 acceleration/deceleration with fixed time constant (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 rate at rapid traverse in ratio to the maximum acceleration rate [%]



Acceleration rate in proportion to the maximum acceleration rate = Acceleration rate at rapid traverse rate / maximum acceleration rate

(3) When either of the following conditions applies, this function is invalid and operates as "rapid traverse constant-gradient acceleration/deceleration". For the axis for which the rapid traverse constant-gradient 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 acceleration/deceleration with fixed time constant (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-gradient multi-step acceleration/deceleration	#1086 G00Intp	#1205 G0bdcc	Operation
G00 command	ON	0	0	Time constant acceleration/deceleration (interpolation type)
			1	Constant-gradient acceleration/deceleration (acceleration/deceleration before interpolation)
			2	Constant-gradient multi-step acceleration/deceleration
		1	Arbitrary	Acceleration/deceleration with fixed time constant (non-interpolation type)
	OFF	0	0	Acceleration/deceleration with fixed time constant (interpolation type)
			1	Constant-gradient acceleration/deceleration (acceleration/deceleration before interpolation)
			2	Acceleration/deceleration with fixed time constant (interpolation type)
		1	Arbitrary	Acceleration/deceleration with fixed time constant (non-interpolation type)
Manual rapid traverse	Arbitrary	Arbitrary	Arbitrary	Acceleration/deceleration with fixed time constant (non-interpolation type)

Decision method of steps

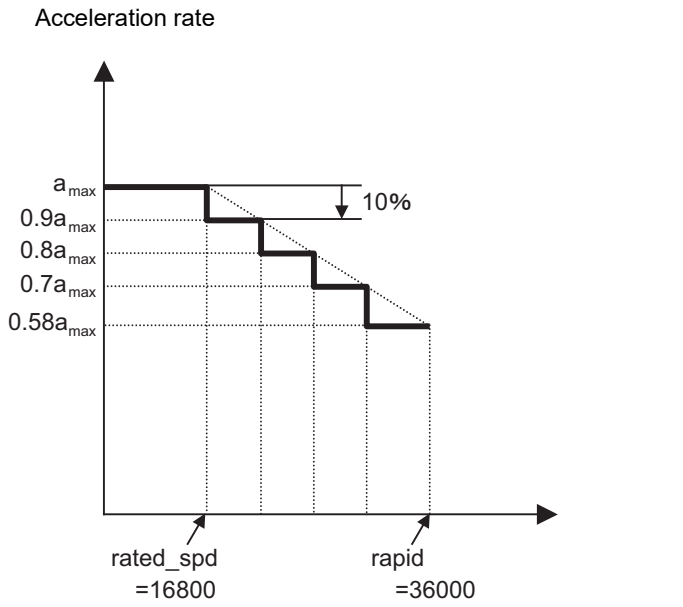
For rapid traverse constant-gradient multi-step acceleration/deceleration, the number of steps is automatically adjusted by set parameter.

The acceleration rate per step is assumed to be a decrease by 10% of the maximum acceleration rate per step. Therefore, the number of steps is decided as follows.

"Step" = (100 - "#2152 acc_rate") / 10 + 1 (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 [%]

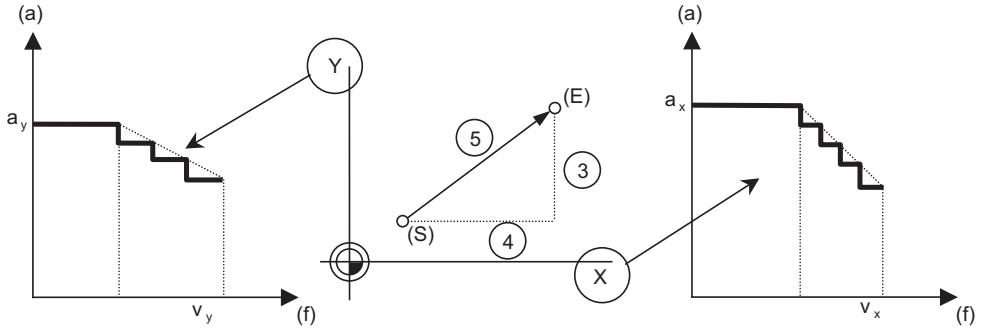


Pattern of acceleration rate at two or more axis interpolation

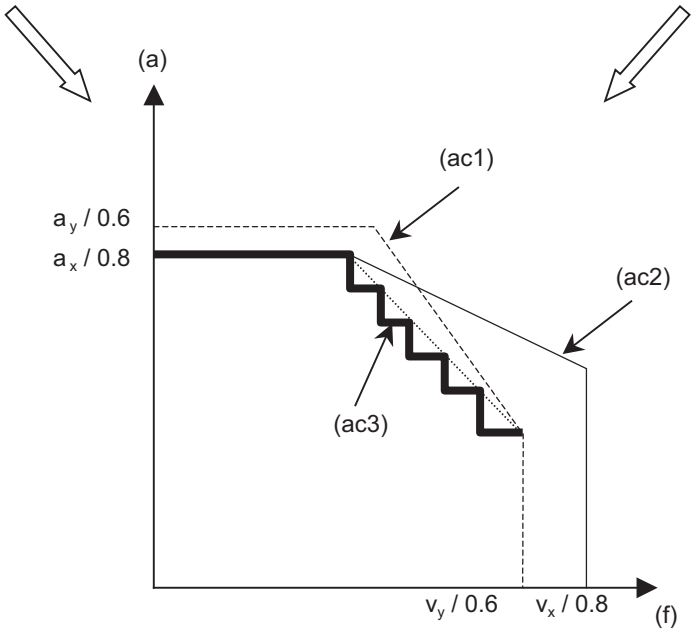
When there are two or more rapid traverse axes with a different pattern of the acceleration rate, there are the following two operation methods.

- 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-gradient multi-step acceleration/deceleration are valid only for an interpolation type. For the interpolation type, the pattern of the acceleration rate operates to the maximum acceleration rate within the range where tolerable acceleration rate of each axis is not exceeded.



[Pattern of acceleration rate of Y axis] **[Pattern of acceleration rate of X axis]**



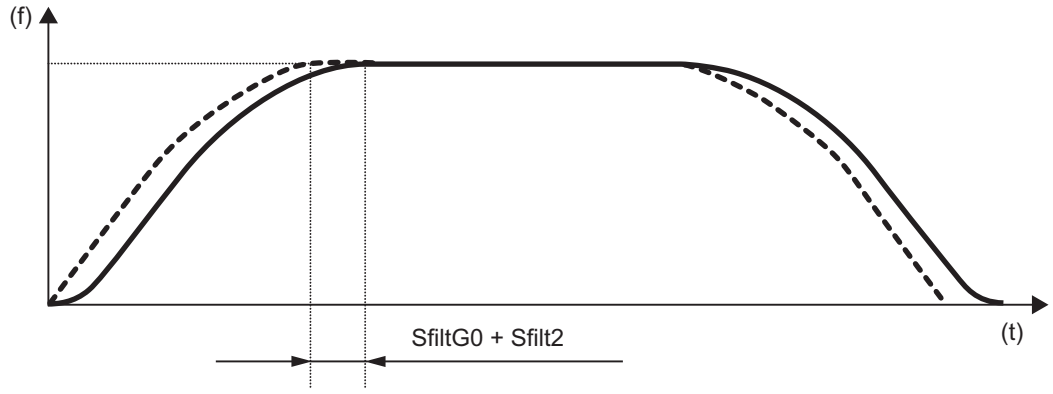
[Pattern of acceleration rate in the composite direction]

(a) Acceleration rate (f) Speed
(S) Start point (E) End point
(ac1) Pattern of acceleration rate when the axis moved to composite direction at Y axis rapid traverse rate
(ac2) Pattern of acceleration rate when the axis moved to composite direction at X axis rapid traverse rate
(ac3) Pattern of acceleration rate in the composite direction

S-pattern filter control

With S-pattern filter control, this enables the rapid traverse constant-gradient 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.



(f) Speed
(t) Time
No S-pattern filter control
S-pattern filter control

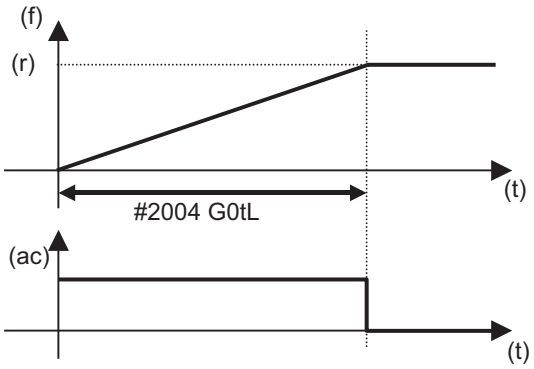
The high-accuracy control mode rapid traverse rate

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-gradient acceleration/deceleration".



(f) Speed
(t) Time
(r) Rapid traverse rate
(ac) Acceleration rate

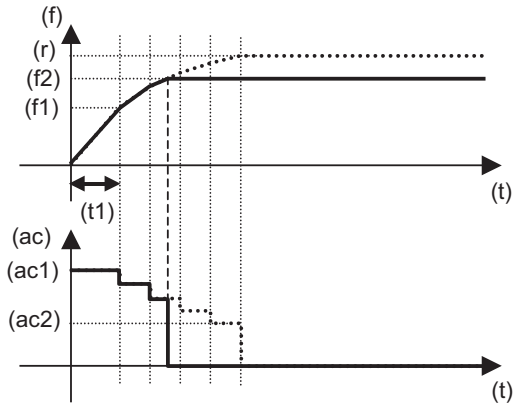
(2) When "The high-accuracy control mode rapid traverse rate" < "rapid traverse rate"

The rapid traverse is performed at the feedrate set in the parameter "#2109 Rapid (H-precision)".

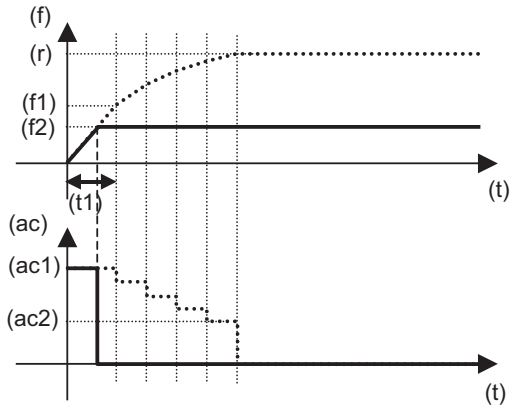
The feedrate is accelerated or decelerated in accordance with the pattern of acceleration rate calculated from the following parameters:

- "#2001 rapid" (Rapid traverse rate)
- "#2151 rated_spd" (Rated speed)
- "#2153 G0t_rated" (G0 time constant up to rated speed)
- "#2152 acc_rate" (Acceleration rate in proportion to the maximum acceleration rate)

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 rate
(ac1) Maximum acceleration rate
(ac2) Acceleration rate at rapid traverse rate
(r) Rapid traverse rate

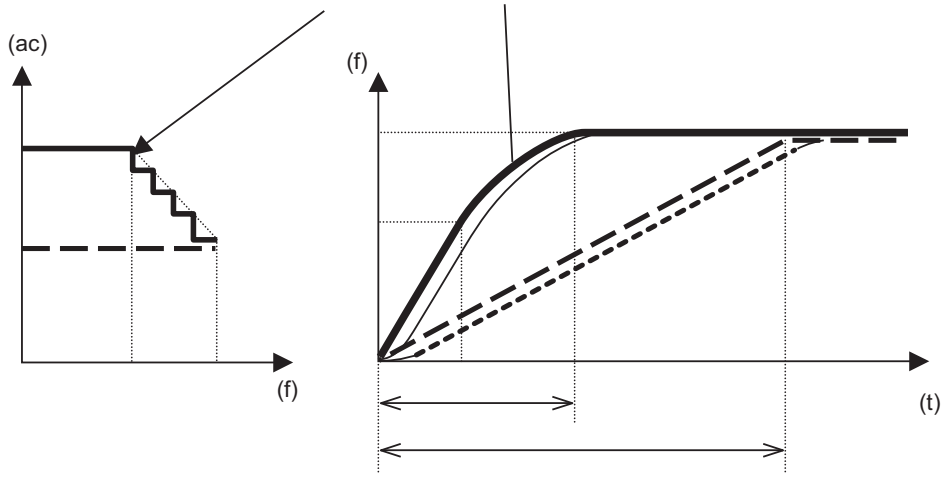
Precautions

(1) Rapid traverse constant-gradient multi-step acceleration/deceleration are valid only for a rapid traverse command. Note that when the manual rapid traverse, rapid traverse constant-gradient multi-step acceleration/deceleration cannot be used.
In this case, an acceleration/deceleration with fixed time constant (acceleration/deceleration after interpolation) is applied. Therefore, 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-gradient multi-step acceleration/deceleration and the manual rapid traverse as shown in figure.

Rapid traverse constant-gradient multi-step acceleration/deceleration



(f) Speed (t) Time (ac) Acceleration rate

Rapid traverse constant-gradient multi-step acceleration/deceleration
Manual rapid traverse (linear)
S-pattern filter control
Soft acceleration/deceleration

(2) Rapid traverse constant-gradient 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-gradient acceleration/deceleration, this function is invalid even if "2" is set to the parameter "#1205 G0bdcc". In this case, a normal acceleration/deceleration with fixed time constant (acceleration/deceleration after interpolation) is applied.

(4) When G00 non-interpolation type ("#1086 G0Intp" = "1"), rapid traverse constant-gradient multi-step acceleration/deceleration cannot be used. It is valid in interpolation mode only.

(5) When the rapid traverse constant-gradient multi-step acceleration/deceleration is applied, rapid traverse acceleration/deceleration types ("#2003 smgst" bit0 to bit3) are ignored.

(6) When the rapid traverse constant-gradient multi-step acceleration/deceleration is valid, G0 constant-gradient ("#1200 G0_acc") cannot be used. Even if G0 constant-gradient is valid ("#1200 G0_acc" = "1"), the setting is ignored.

(7) When the rapid traverse constant-gradient 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.

7.10 Cutting Feed Constant-gradient Acceleration/Deceleration



Function and purpose

This function performs linear acceleration/deceleration at a constant inclination in the cutting feed mode. The constant-gradient acceleration/deceleration method can be more beneficial in reducing cycle time in comparison to the acceleration/deceleration with fixed time constant method.



Detailed description

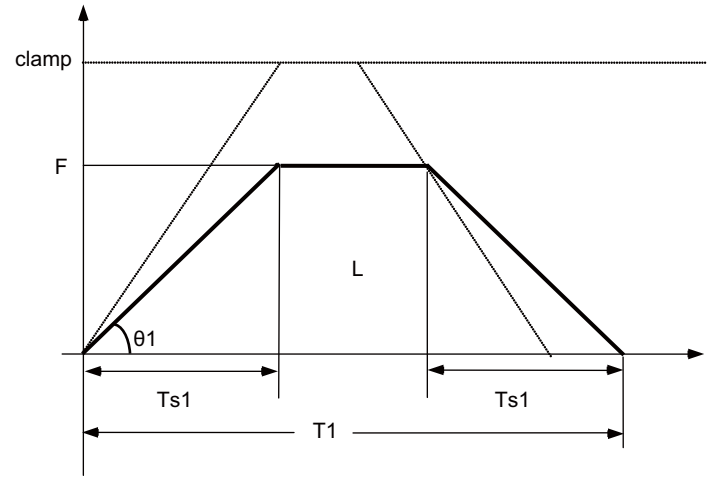
(1) Cutting feed constant-gradient acceleration/deceleration function is effective only when the commanded cutting feed acceleration/deceleration mode is linear method or soft method in linear interpolation (G01) command.

(2) The program format of linear interpolation when cutting feed constant-gradient acceleration/deceleration is executed is the same as when this function is invalid (acceleration/deceleration with fixed time constant).

(3) The acceleration/deceleration patterns in the case where cutting feed constant-gradient acceleration/deceleration is performed are as follows.

[When the interpolation distance is long enough for the cutting feedrate to be achieved]

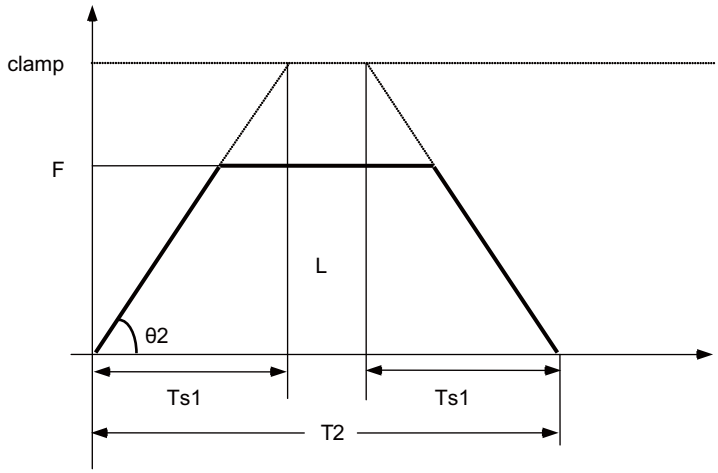
(a) Acceleration/deceleration with fixed time constant:



$$T1 = \frac{L}{F} + Ts1$$

$$\theta 1 = \tan^{-1}\left(\frac{F}{Ts1}\right)$$

(b) Constant-gradient acceleration/deceleration:



$$T2 = \frac{L}{F} + \frac{Ts1 \times F}{clamp}$$

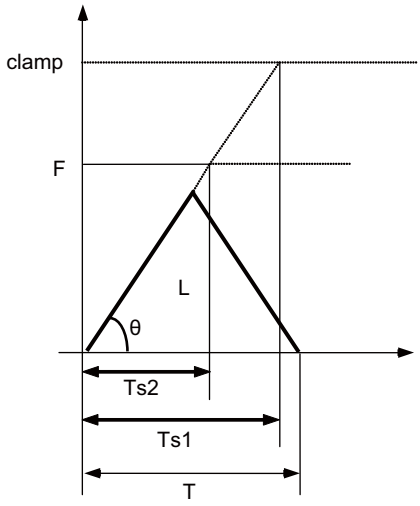
$$\theta 2 = \tan^{-1}\left(\frac{clamp}{Ts1}\right)$$

clamp : Maximum cutting feedrate (Parameter "#2002 clamp")
F : Cutting feedrate
Ts1 : Acceleration/deceleration time (Parameter "#2007 G1tL")
θ1 : Acceleration/deceleration gradient (Acceleration/deceleration with fixed time constant)
θ2 : Acceleration/deceleration gradient (Constant-gradient acceleration/deceleration)
T1 : Interpolation time (Acceleration/deceleration with fixed time constant)
T2 : Interpolation time (Constant-gradient acceleration/deceleration)
L : Interpolation distance

In the case of acceleration/deceleration with fixed time constant, the acceleration/deceleration gradient is determined by the cutting feedrate. In the case of constant-gradient acceleration/deceleration, it is determined by the maximum cutting feedrate; therefore, the cycle time will be shorter than in the former case.

[When the interpolation distance is so short that the cutting feedrate is not achieved]

The acceleration/deceleration gradient is determined by the maximum cutting feedrate.

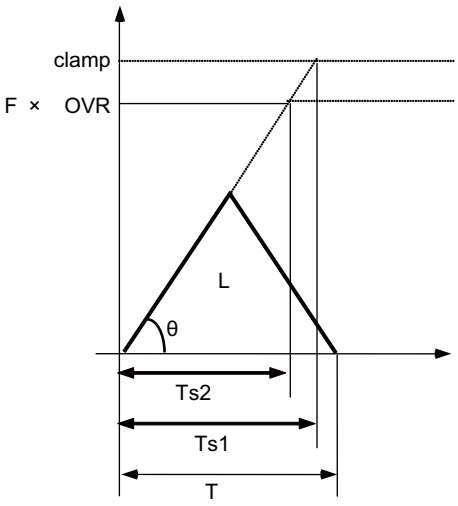


$$T = 2 \times \sqrt{Ts1 \times L / clamp}$$

$$\theta = \tan^{-1}\left(\frac{clamp}{Ts1}\right)$$

clamp : Maximum cutting feedrate (Parameter "#2002 clamp")
F : Cutting feedrate
Ts1 : Acceleration/deceleration time (Parameter "#2007 G1tL")
Ts2 : Acceleration/deceleration time to reach the cutting feedrate
θ : Acceleration/deceleration gradient
T : Interpolation time
L : Interpolation distance

[When the interpolation distance is so short that the maximum cutting feedrate is not achieved and the override for cutting feed constant-gradient acceleration/deceleration is activated]



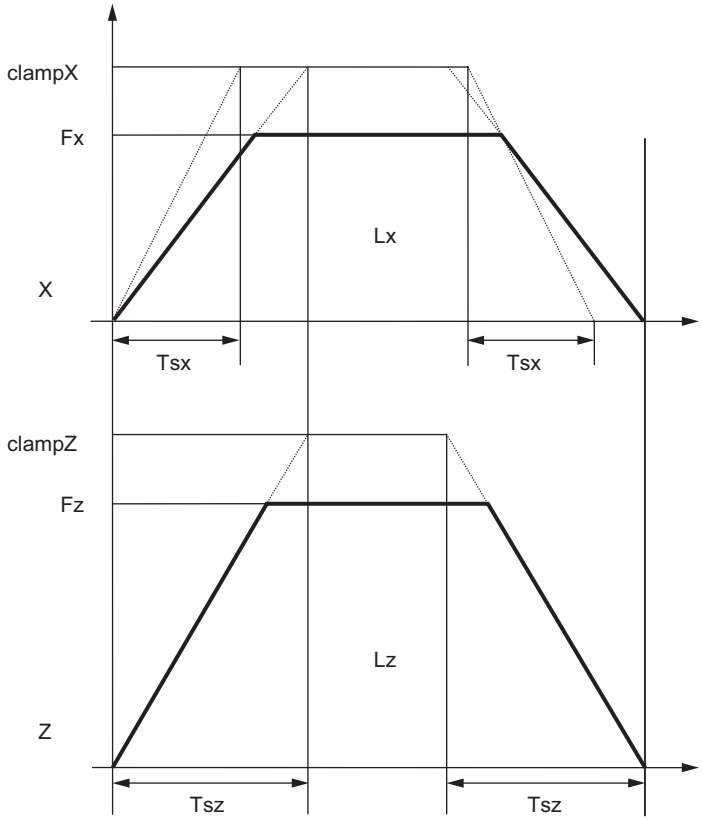
$$T = 2 \times \sqrt{Ts2 \times L / (F \times OVR)}$$

$$\theta = \tan^{-1}\left(\frac{clamp}{Ts1}\right)$$

clamp : Maximum cutting feedrate (Parameter "#2002 clamp")
F : Cutting feedrate
OVR : Maximum override value for cutting feed constant-gradient acceleration/deceleration (Parameter "#1367 G1AccOVRMax")
Ts1 : Acceleration/deceleration time (Parameter "#2007 G1tL")
Ts2 : Acceleration/deceleration time to reach the cutting feedrate
θ : Acceleration/deceleration gradient

[2-axis simultaneous interpolation (When Tsx < Tsz, Lx ≠ Lz)]

When multi-axis simultaneous interpolation is performed during linear interpolation constant-gradient acceleration/deceleration, the longest one among the acceleration/deceleration times of all axes is applied to the other axes which were commanded simultaneously.
The acceleration/deceleration time is determined for each axis by the following items; maximum cutting feedrates (parameter "#2002 clamp"); cutting feed acceleration/deceleration time constant (parameter "#2007 G1tL"); cutting feed rates (F); and the interpolation distance (L).
However, the axis of which the acceleration/deceleration time constant is greater than the cutting feed acceleration/deceleration time constant (parameter "#2007 G1tL") is accelerated/decelerated with the cutting feed acceleration/deceleration time constant.

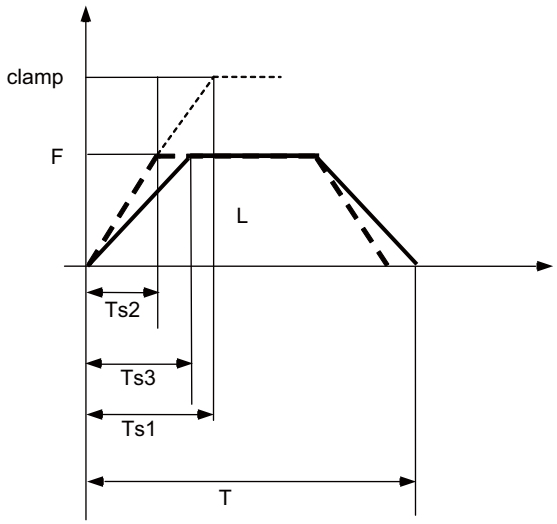


Tsx : X axis acceleration/deceleration time
Tsz : Z axis acceleration/deceleration time
Lx : X axis interpolation distance
Lz : Z axis interpolation distance
Fx : X axis feedrate
Fz : Z axis feedrate

When Tsx < Tsz, the acceleration/deceleration time (Ts) of this block is set to Tsz (acceleration/deceleration time of the Z axis).

[When the feedrate is so low that the acceleration/deceleration time is shorter than the minimum time constant for constant-gradient acceleration/deceleration]

Acceleration/deceleration speed is adjusted to prevent the acceleration/deceleration time calculated by the cutting feedrate from going below the minimum time constant.

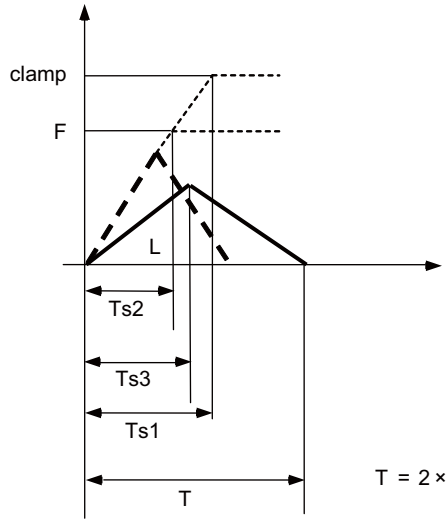


$$T = \frac{L}{F} + 2 \times Ts3$$

clamp : Maximum cutting feedrate (Parameter "#2002 clamp")
F : Cutting feedrate
Ts1 : Acceleration/deceleration time (Parameter "#2007 G1tL")
Ts2 : Acceleration/deceleration time to reach the cutting feedrate
Ts3 : Minimum time for constant-gradient acceleration/deceleration (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 constant-gradient acceleration/deceleration]

Acceleration/deceleration speed is adjusted to prevent the acceleration/deceleration time calculated by interpolation distance from going below the minimum time constant.



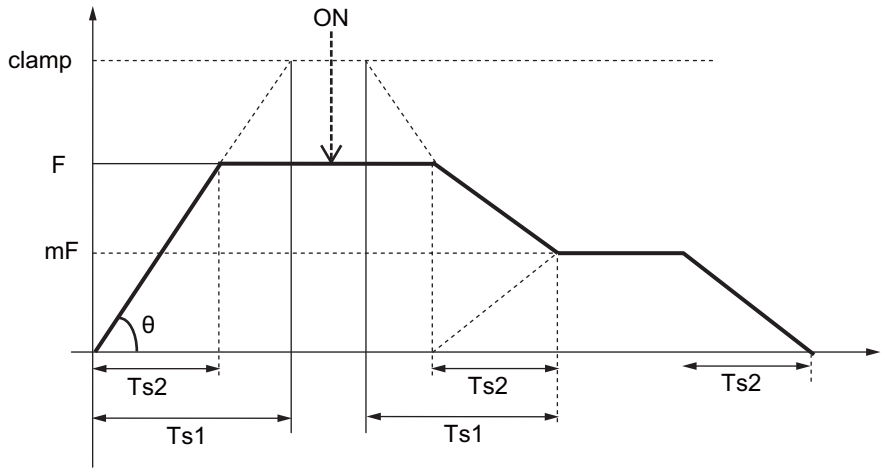
$$T = 2 \times Ts3$$

clamp : Maximum cutting feedrate (Parameter "#2002 clamp")
F : Cutting feedrate
Ts1 : Acceleration/deceleration time (Parameter "#2007 G1tL")
Ts2 : Acceleration/deceleration time to reach the cutting feedrate
Ts3 : Minimum time for constant-gradient acceleration/deceleration (Parameter "#2199 G1tMin")
T : Interpolation time
L : Interpolation distance

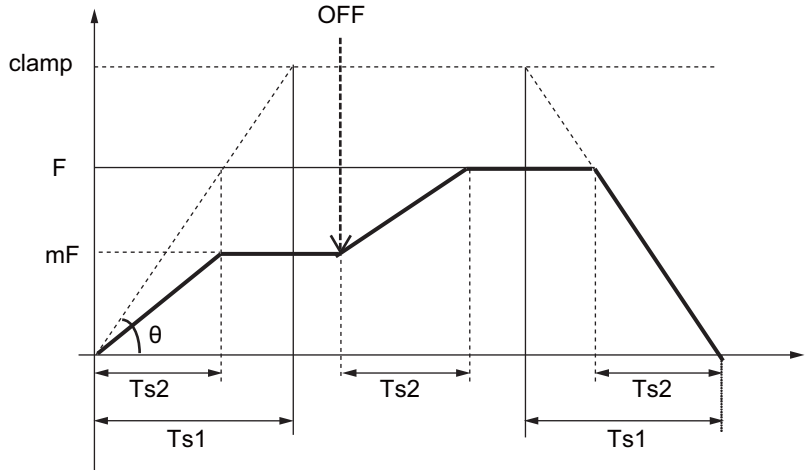
Relationship with other functions

(1) The constant-gradient acceleration/deceleration during dry run is disabled.
When the constant-gradient acceleration/deceleration is valid and the speed is changed by dry run during axis traveling, dry run is disabled in the currently executed block, and the constant-gradient acceleration/deceleration is performed at the same speed as before. Dry run is enabled from the next block, and the manual feedrate is applied. However, the acceleration/deceleration time remains unchanged.
When dry run is enabled and it canceled during axis traveling, dry run is immediately disabled, and the cutting feedrate is applied. After the speed was changed, the constant-gradient acceleration/deceleration is enabled.
When the high-speed machining mode is valid (under G5P2, under G5P10000, etc.), dry run is disabled. However, when the high-speed machining mode is canceled by the positioning command (G00 command) or single block stop command, dry run is enabled.
To enable dry run during the constant-gradient acceleration/deceleration, use the high-accuracy control function (G61.1). Do not use this function.

[When constant-gradient acceleration/deceleration is enabled and the "Dry run" signal is turned ON during axis traveling]



[When dry run is enabled and the "Dry run" signal is turned OFF during axis traveling]



clamp : Maximum cutting feedrate (Parameter "#2002 clamp")
F : Cutting feedrate
mF : Manual feedrate
Ts1 : Acceleration/deceleration time (Parameter "#2007 G1tL")
Ts2 : Acceleration/deceleration time to reach the cutting feedrate
θ : Acceleration/deceleration gradient
ON : "Dry run" signal ON
OFF : "Dry run" signal OFF

(2) The constant-gradient acceleration/deceleration is valid for the linear interpolation command in the fixed cycle. However, the constant-gradient acceleration/deceleration for linear interpolation is invalid for the linear interpolation command in the synchronous tapping cycle.
(The synchronous tapping acceleration/deceleration is enabled.)

Precautions

(1) If a value greater than 100 (%) is designated for cutting feed override under cutting feed constant-gradient acceleration/deceleration control, the acceleration/deceleration gradient 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%.

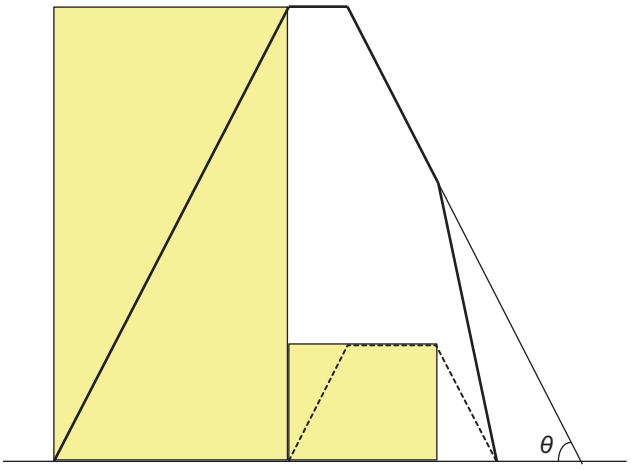
(2) If there are one or more NC control axes that are set to soft acceleration/deceleration for G1, the parameter "#1367 G1AccOVRMax" setting will be ignored and the cutting feed override value is handled as 100%.

(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 constant-gradient 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) feedrate is low, acceleration/deceleration time becomes quite short when the constant-gradient 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 constant-gradient 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.

(6) If the linear interpolation command is continuously executed in two blocks for the same axis during constant-gradient acceleration/deceleration for linear interpolation, the gradient at deceleration becomes greater than the reference value. Therefore, the load of the servo system may increase. Consider so that the speed command (F command value) does not change suddenly.

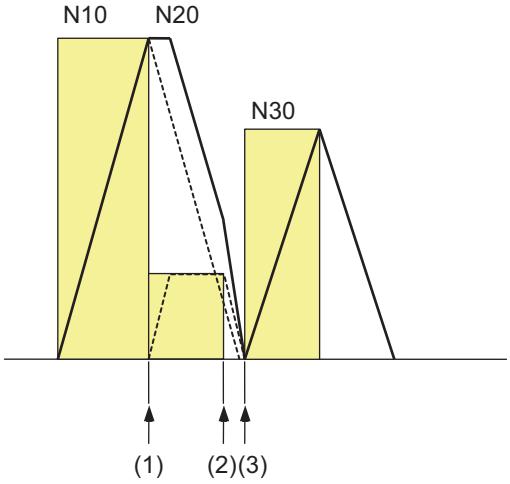


$$\theta = \tan^{-1}\left(\frac{\text{clamp}}{\text{Ts}}\right)$$

θ: Reference value of gradient

Restrictions

(1) When the linear interpolation command is continuously executed in two or more blocks for the same axis during constant-gradient acceleration/deceleration for linear interpolation, processing of the 3rd block is started after the deceleration processing of the 1st block was completed.




```
      :
      :
N10 G01 X100. ;
N20 G01 X20. ;
N30 G01 X70. ;
      :
      :
```

(1) Processing start of 2nd block

(2) Processing of the 3rd block is not started because deceleration processing of the 1st block is not completed.

(3) Processing start of 3rd block

7.11 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 "7.13 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

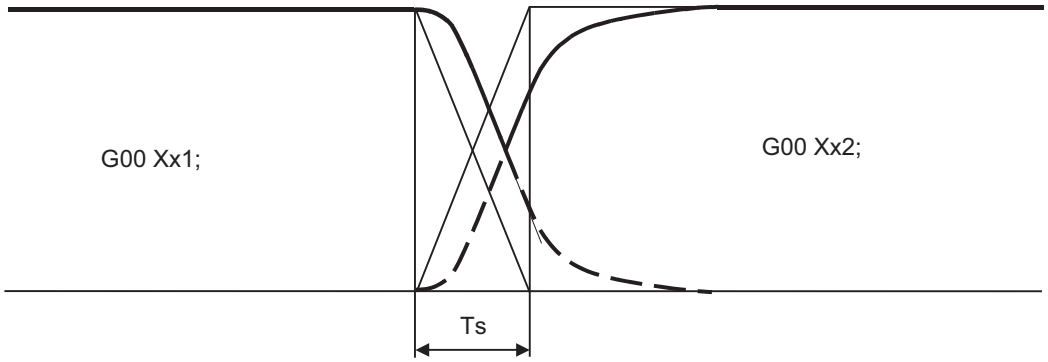
Exact stop check

G09 G01 (G02, G03);

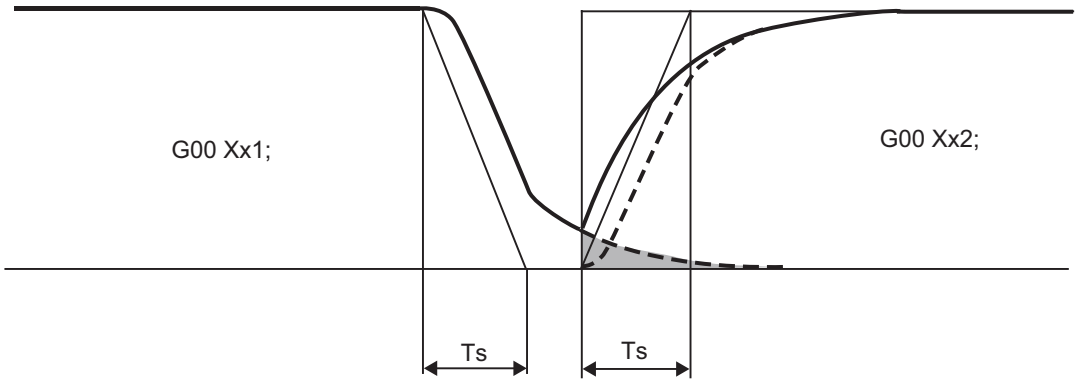
The exact stop check command G09 has an effect only with the cutting command (G01 - G03) in its particular block.

Detailed description

[With continuous cutting feed]



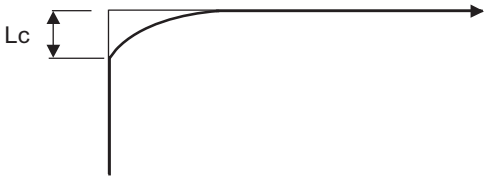
[With cutting feed in-position check]



Ts : Cutting feed acceleration/deceleration time constant

In-position width

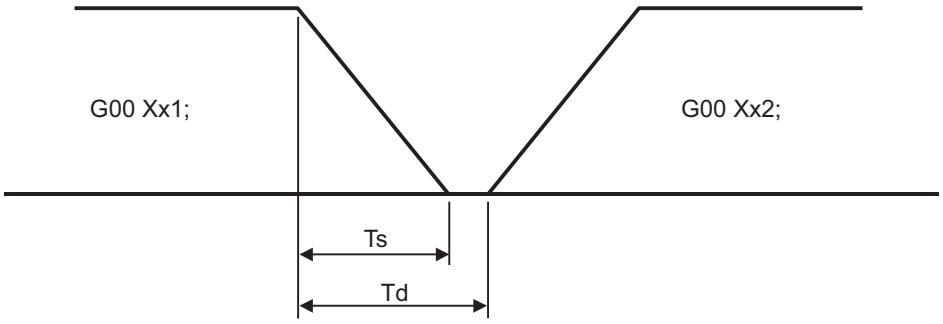
The in-position width, as shown in the figure above, is the remaining distance (shaded area in the above figure) of the previous block when the next block is started is set in the servo parameter "#2224 sv024". (This depends on the MTB specifications.)
The in-position width is designed to reduce the 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.)

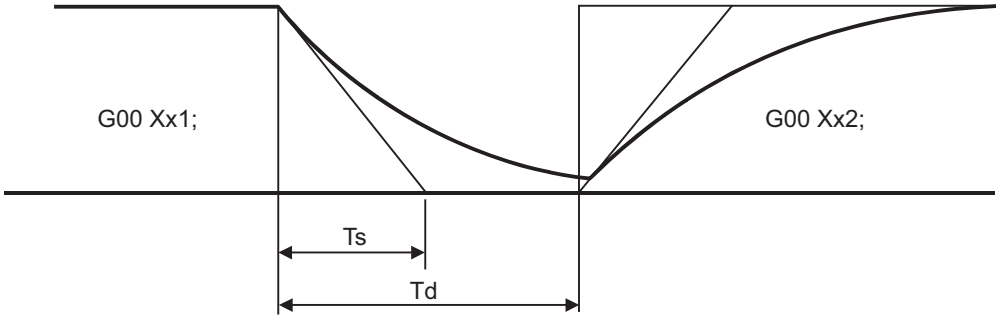
With deceleration check

(1) With linear acceleration/deceleration



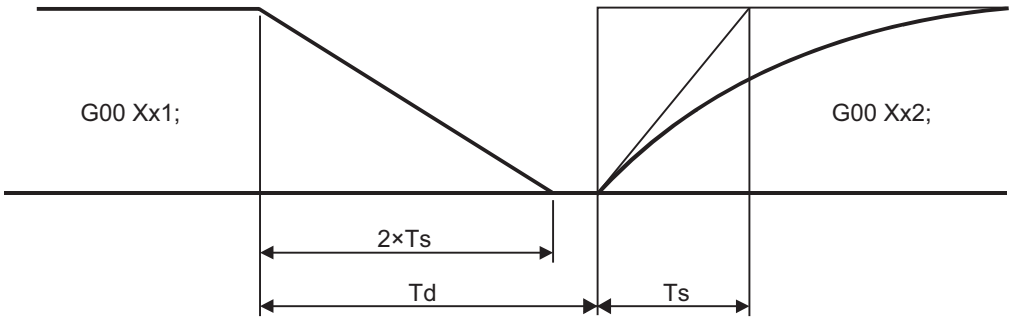
Ts: Acceleration/deceleration time Td: Deceleration check time Td = Ts + α (0 to 10ms) constant

(2) With exponential acceleration/deceleration



Ts: Acceleration/deceleration time Td: Deceleration check time Td = 2 x Ts + α (0 to 10ms) constant

(3) With exponential acceleration/linear deceleration



Ts: Acceleration/deceleration time Td: Deceleration check time Td = 2 x Ts + α (0 to 10ms) constant

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.

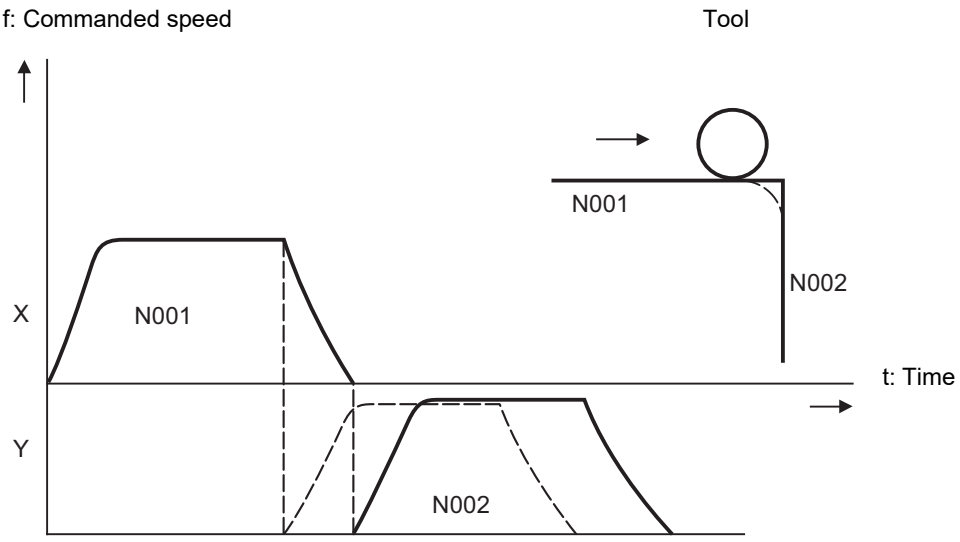
Note

- 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]



Solid line indicates speed pattern with G09 command
Broken line indicates speed pattern without G09 command

7.12 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.

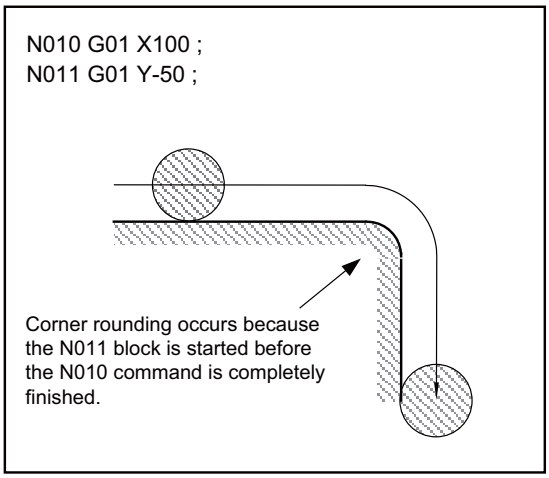
7.13 Deceleration Check

7.13.1 Deceleration Check

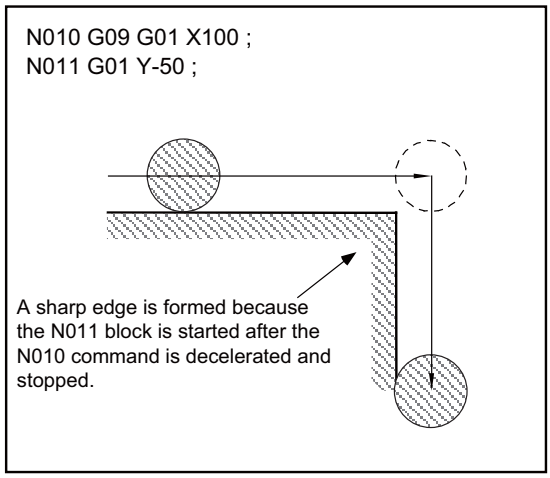
Function and purpose

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 G0Ipfg" is ON, or the movement reverses to the opposite direction.

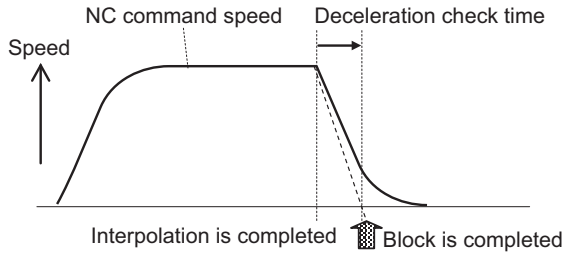
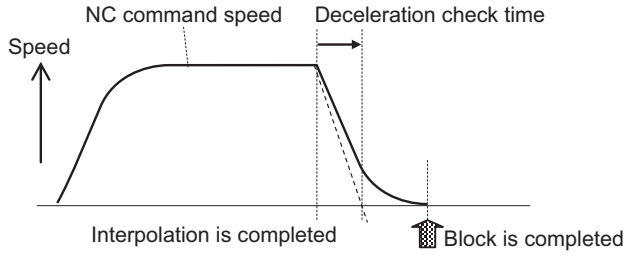
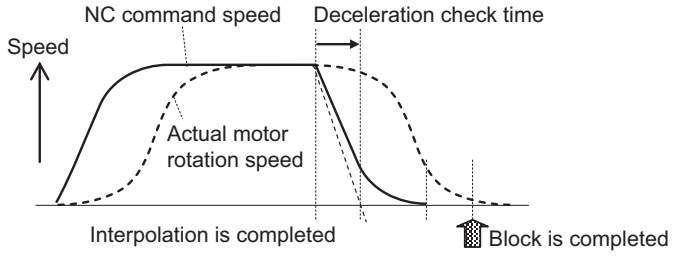
(3) A command deceleration check is performed when G01 => G01 block is specified, "#1503 G1Ipfg" is ON, or the movement reverses to the opposite direction.

For the deceleration check when movement in the opposite direction is reversed, refer to "7.13.2 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

(1) Command deceleration check method Deceleration is completed after the deceleration check time (*1) has passed after the interpolation.	
(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.	
(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.	

(*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).

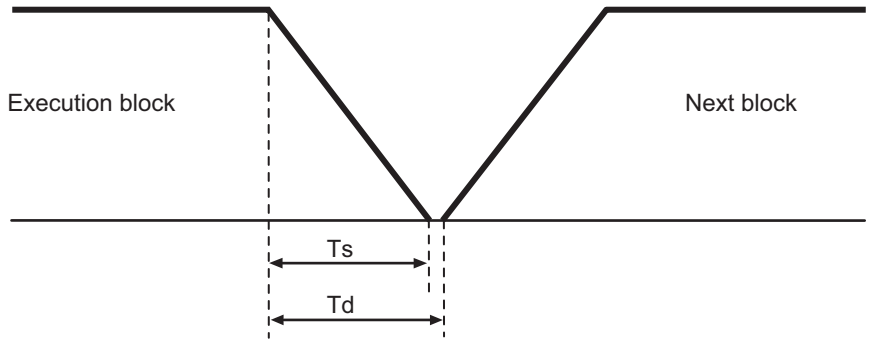
Parameters		Deceleration check method	Conditions of deceleration check
#1389 G1SmthChk	#1223 aux07/bit1		
0	0	Command deceleration check method	Deceleration check time has elapsed.
	1	In-position check method	Deceleration check time has elapsed, all axis smoothing has become zero, and all axes have become 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.

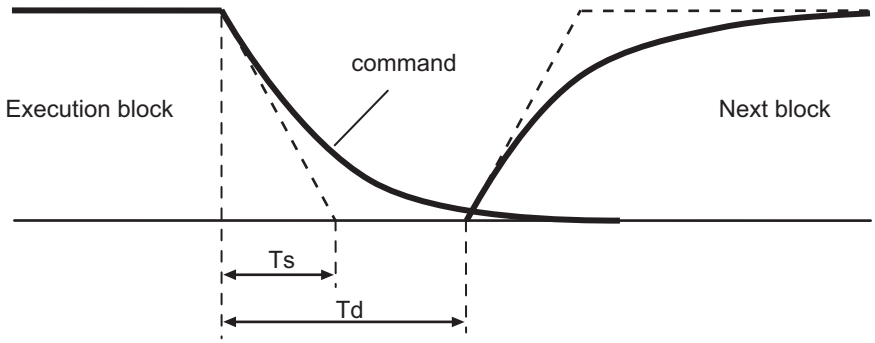
(a) For linear acceleration/deceleration



(Ts) Linear acceleration/deceleration time constant

(Td) Deceleration check time: Td = Ts + α (0 to 10 ms)

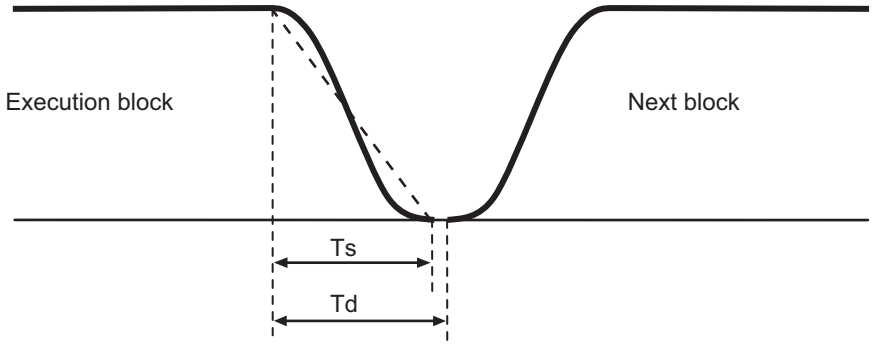
(b) For exponential acceleration/deceleration



(Ts) Exponential acceleration/deceleration time constant

(Td) Deceleration check time: Td = 2 x Ts + α (0 to 10 ms)

(c) For soft acceleration/deceleration



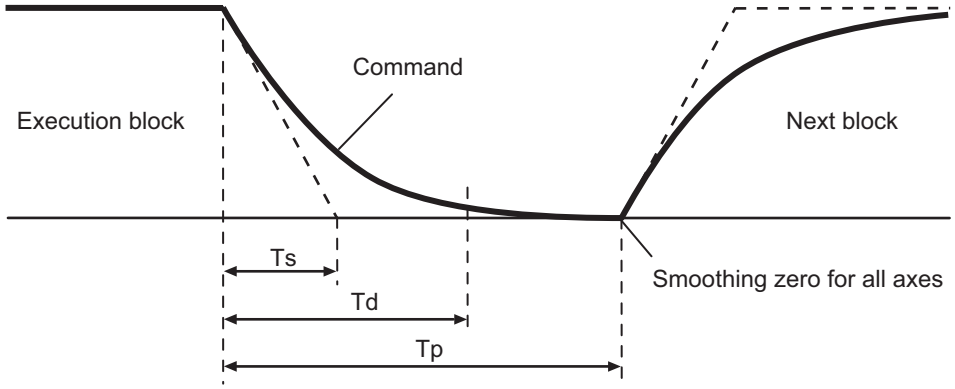
(Ts) Soft acceleration/deceleration time constant

(Td) Deceleration check time: Td = 2 x Ts + α (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



(Ts) Exponential acceleration/deceleration time constant

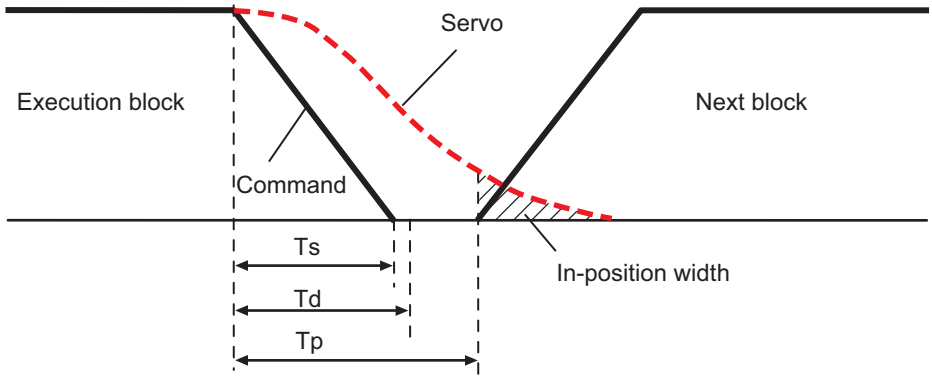
(Td) Deceleration check time

(Tp) 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



(Ts) Linear acceleration/deceleration time constant

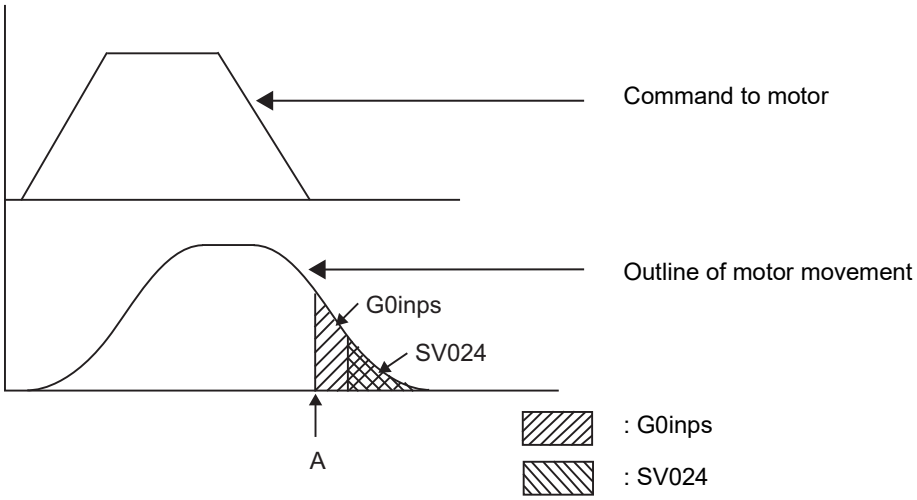
(Td) Deceleration check time

(Tp) Waiting time for a block to complete

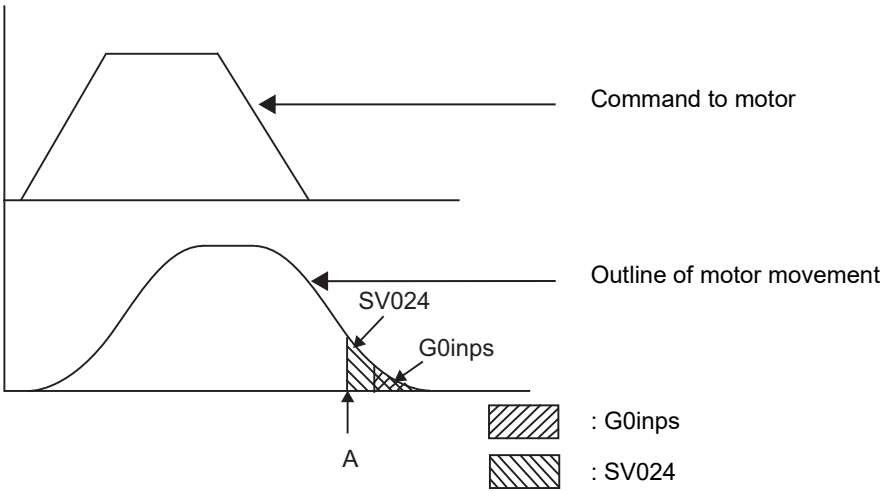
As shown in the figure above, the in-position width is the remaining distance from the previous block at the start of the next block. (Shaded area of the figure above).

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)

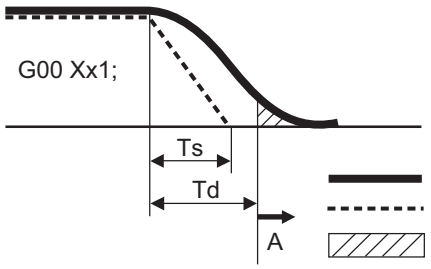
Execution of the next block starts after confirming that the position error amount in the block in which the deceleration check is to be performed is less than the in-position width.
When there are several movement axes, the system confirms that the position error amount of each movement axis in each 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.

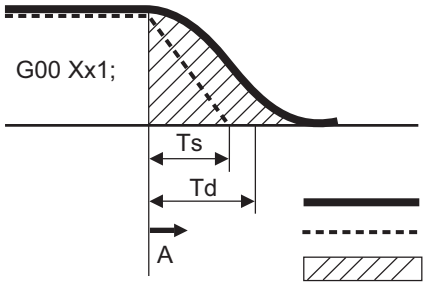


(Ts) Acceleration/deceleration time constant
(Td) Deceleration check time: Td = Ts + α (0 to 10 ms)

Servo machine position
Command
In-position width (Servo system position error amount)

(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.



(Ts) Acceleration/deceleration time constant
(Td) Deceleration check time: Td = Ts + α (0 to 10 ms)

Servo machine position
Command
In-position width (Servo system position error amount)

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 successive block from G01 to G01 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 G1Ipfg" is set to "1".
Note that the "#1502 G0Ipfg" setting will be followed if the axis direction reverses to the opposite direction in a successive block from G01 to G00.

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.

7.13.2 Deceleration Check When Movement in the Opposite Direction Is Reversed

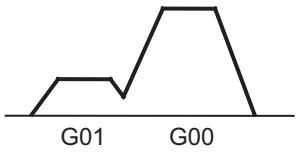
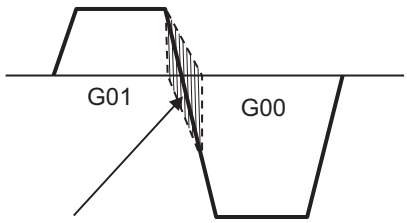
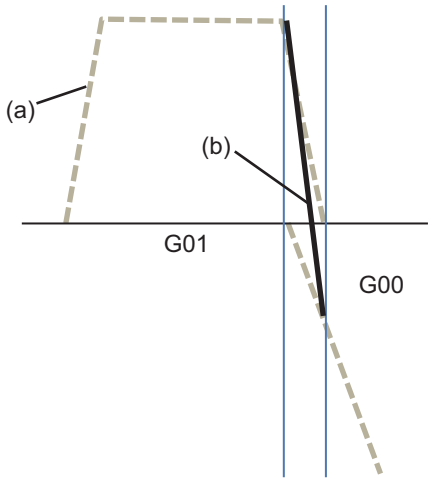
Function and purpose

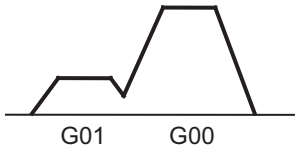
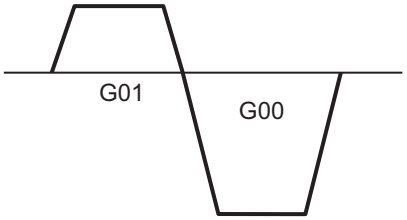
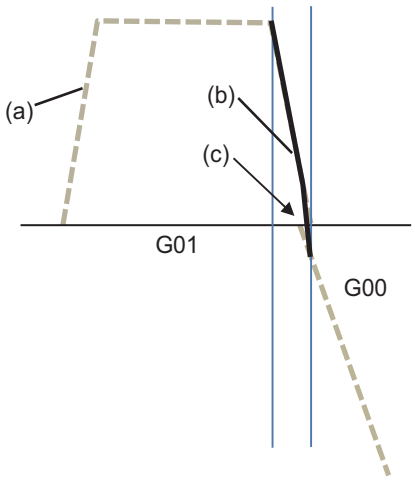
A deceleration check cannot be designated for G01 -> G00 or G01 -> G01, but it can be designated in the following manner only when the movement reverses to the opposite direction in successive blocks.
A deceleration check can also be executed if even one axis is moving in the opposite direction while 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 G0Ipfg").

	Same direction	Opposite direction
#1502 = 0		 The acceleration rate is excessive due to the G01 and G00 composite speed. Enlarged figure  (a) Commanded speed (b) Resultant speed

	Same direction	Opposite direction
#1502 = 1		 Command deceleration Enlarged figure  (a) Commanded speed (b) Resultant speed (c) Command deceleration is completed.

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)

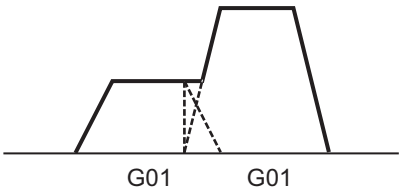
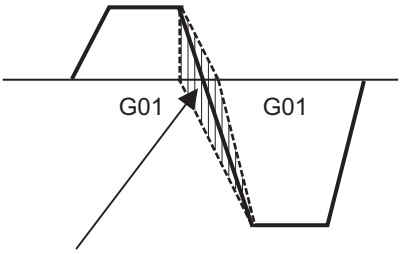
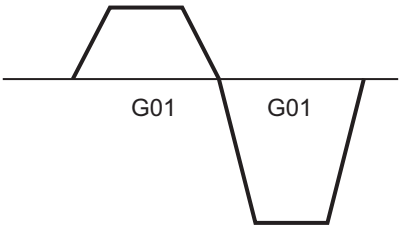
G91 G01 X100. Y100. F4000 ; G00 X100. Y100. ;	A deceleration check is not carried out, because both the X axis and the Y axis move in the same direction in the program.

(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 G1Ipfg").

	Same direction	Opposite direction
#1503 = 0		 The acceleration rate is excessive due to the G01 and G01 composite speed.
#1503 = 1		 Command deceleration

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)

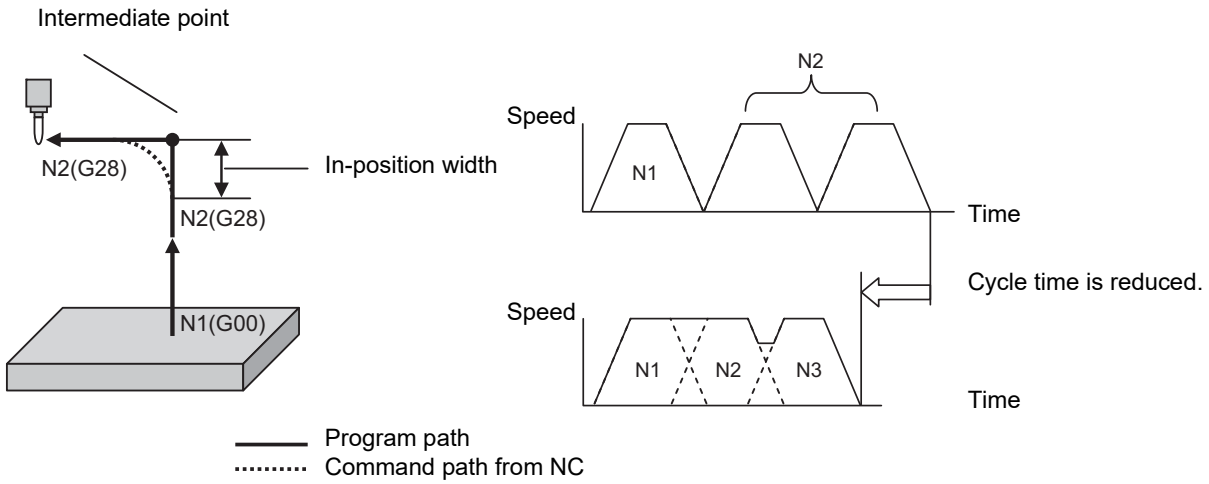
G91 G01 X100. Y80. F4000 ; G01 X80. ;	A deceleration check is not carried out, because the X axis moves in the same direction, and there is no Y axis movement command in the program.

7.14 Rapid Traverse Block Overlap; G0.5 P1

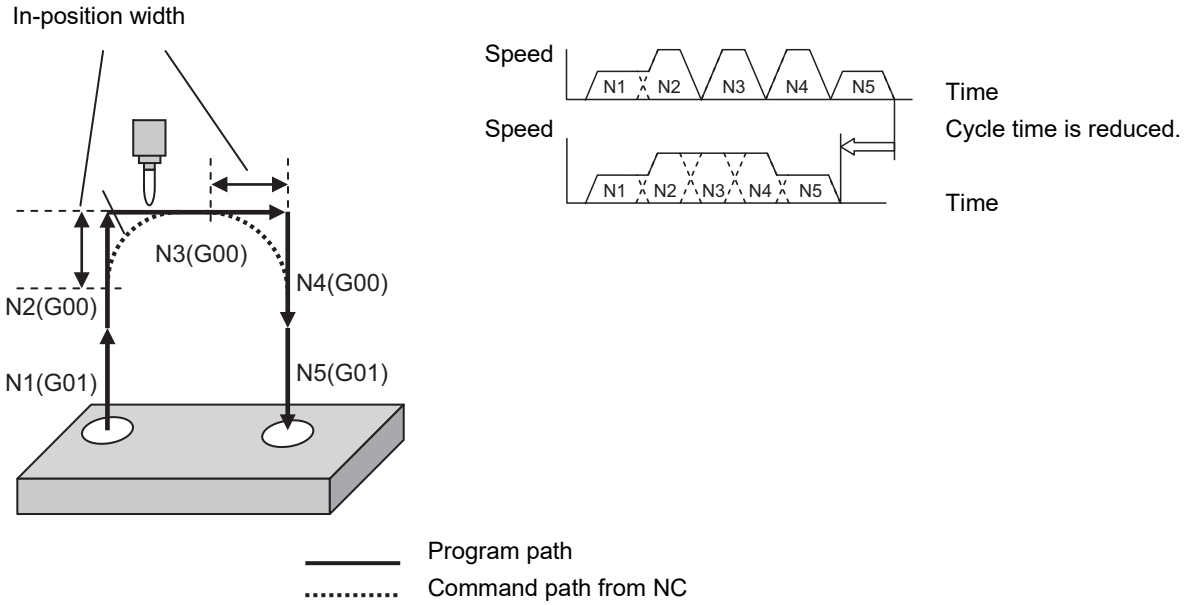
Function and purpose

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)



Example of behavior and velocity waveform 2 (example of application of rapid traverse block overlap in continuous drilling motion)



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 "7.13 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 "7.13 Deceleration Check".)	G00	3

For G00 overlap, refer to "7.14.1 Rapid Traverse Block Overlap for G00; G0.5". For G28/G30 overlap, refer to "7.14.2 Rapid Traverse Block Overlap for G28".

7.14.1 Rapid Traverse Block Overlap for G00; G0.5

Function and purpose

This function enables the next block to start (overlap) without waiting for the deceleration completion at the joint between blocks in the G00-G00 command, G00-G01 command and G01-G00 command (*1). This reduces the cycle time of machining including the G00-G00 command, G00-G01 command and G01-G00 command. (*2)
This function has two modes. The modes can be selected with the parameter "#1711 cfg11/bit2". (This depends on the MTB specifications. This is available in the M800/M80 Series S/W version F8 or later.)

#1711 cfg11/bit2	Details
0 (mode 1)	The cycle time can be reduced in normal acceleration/deceleration control (*3).
1 (mode 2)	It is also effective to reduce the cycle time in pre-interpolation acceleration/deceleration control. It is recommended to normally use the mode 2.

(*1) In the G01-G00 command, overlap is performed only in the mode 2 and high-accuracy mode.
(The high-accuracy mode means all of high-speed high-accuracy control I/II/III and high-accuracy control.)

(*2) Cycle time reduction may not be guaranteed in every case.

(*3) Pre-interpolation acceleration/deceleration is an acceleration/deceleration control method appropriate to high-accuracy machining.
In the following cases, G00 is operated in pre-interpolation acceleration/deceleration.
- The high-speed high-accuracy control mode is selected, and "#8090 SSS ON" is set to "1".
- "#1086 G00 interp OFF" is set to "0", and "#1205 G0bdcc" (Acceleration and deceleration before G0 interpolation) is not "0".

For the rapid traverse block overlap function, also refer to "7.14 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 "7.14.2 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	Linear 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) The program error (P33) occurs when this command is not issued alone in a block.
(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 is enabled.
Refer to the MTB specifications (parameter "#1442 G0ol").

(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: Invalid
:	

Motion subject to rapid traverse block overlap

(1) Mode 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")

Refer to the table below for details.

Control mode	Parameter		G code following positioning (G00)	
High-accuracy mode	#1086	#1205	G00	G01
OFF	0	0	○	○
		1	○	×
		2	○	×
	1	0/1/2	○	○
ON	0	0	○	×
		1	○	×
		2	○	×
	1	0/1/2	○	×

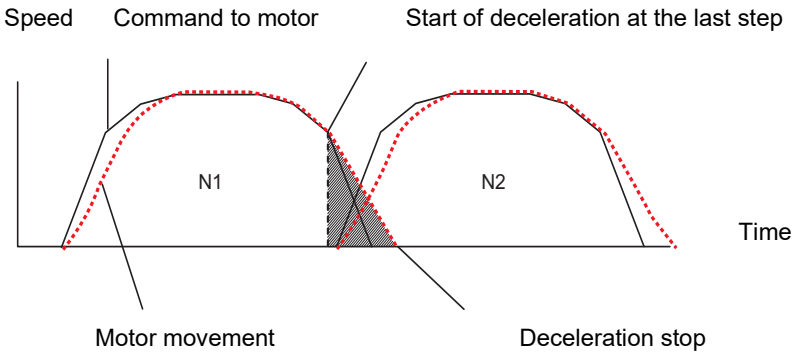
○: Subject to rapid traverse block overlap for G00

×: Not subject to rapid traverse block overlap for G00

<Note>

- 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.;



(2) Mode 2

Only in the mode 2 and high-accuracy mode, the positioning (G00) commanded following the linear interpolation (G01) is subject to rapid traverse block overlap. It may not be subject to rapid traverse block overlap depending on the control mode or parameter settings specified by the MTB. Refer to the table below for details.
The mode 2 can be used when the parameter "#1569 SfiltG0" is not set to "0".

Control mode	Parameter			G code following positioning (G00)		G code following linear interpolation (G01)	
High-accuracy mode	#8090 (SSS ON)	#1086 (G0intp)	#1205 (G0bdcc)	G00	G01	G00	G01
OFF	0/1	0	0	○	○	×	×
			1	○	×	×	×
			2	×	×	×	×
		1	0/1/2	○	○	×	×
ON	0	0	0	○	×	×	×
			1	○	○	○	×
			2	×	×	×	×
		1	0/1/2	○	×	×	×
	1	0	0	○	○	○	×
			1	○	○	○	×
			2	×	×	×	×
		1	0/1	○	○	○	×
			2	×	×	×	×

○: Subject to rapid traverse block overlap for G00

×: Not subject to rapid traverse block overlap for G00 (Operates with the conventional deceleration check function.)

(3) Common to mode 1 and mode 2

This function is valid even in a fixed cycle, subprogram or macro program.
(If the in-position width is specified in a fixed cycle command, that value is given priority.)

In addition, this function is valid if positioning (G00) is followed by the fixed cycle (G81, G83 only), subprogram call command or macro call command block when the rapid traverse block overlap for G00 is enabled.

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 G0olinps").

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.

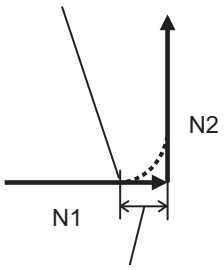
(a) For X axis

Program example

```
N1 G91 G00 X50.;
N2 Y50.;
```

Operation

Start position of N2 block



X axis in-position width = 0.5 mm

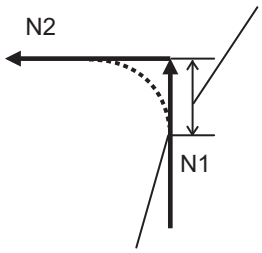
(b) For Y axis

Program example

```
N1 G91 G00 Y50.;
N2 X-50.;
```

Operation

Y axis in-position width = 1 mm



Start position of N2 block

(c) For X and Y axes

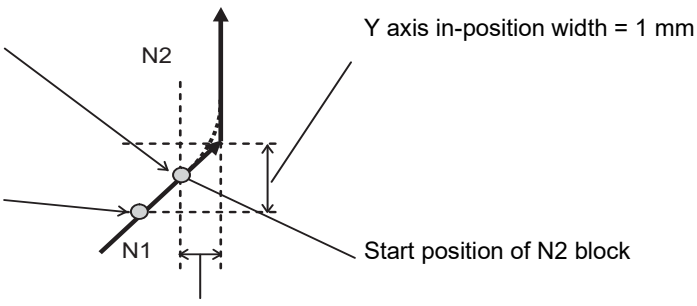
Program example

```
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 = 0.5 mm

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 block 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 G0olinps" and "#2632 G1olinps")

(a) Positioning (G00) - Positioning (G00): Parameter "#2631 G0olinps"

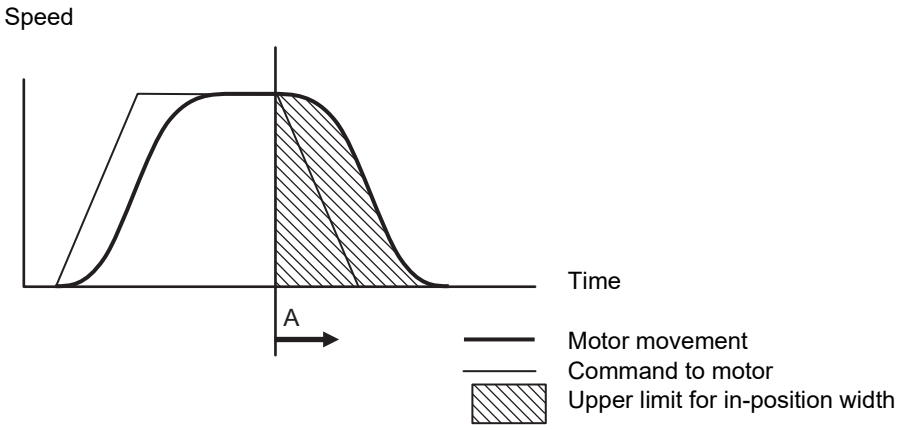
(b) Positioning (G00) - Cutting feed (G01) (high-accuracy mode is OFF): Parameter "#2632 G1olinps" (*1)

(*1) Positioning (G00) - Cutting feed (G01) may not be subject to rapid traverse block overlap depending on the current control mode or parameter settings specified by the MTB. Refer to "Motion subject to rapid traverse block overlap" for details.

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" and "#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 block 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

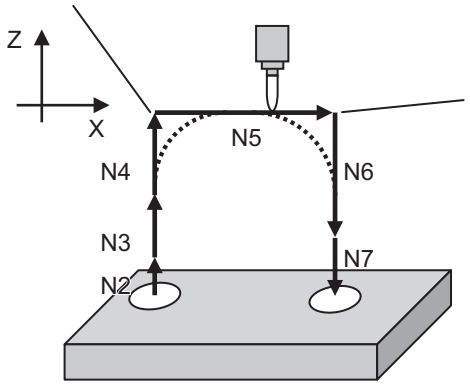
When the in-position width is specified with address J (G0.5P1 J_)

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 (G0olinps)	2 mm	1.5 mm
#2632 (G1olinps)	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.;
```

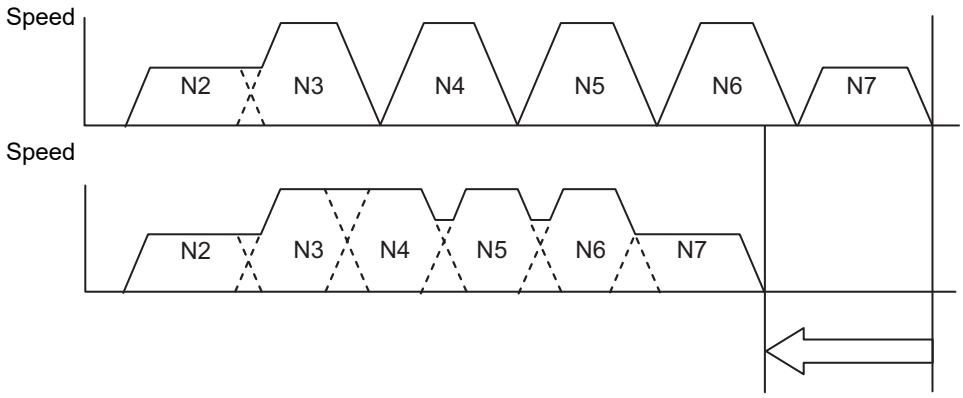
Z axis in-position width is set to 1 mm with the J address command.



X axis in-position width is set to 1 mm with the J address command.

Program path

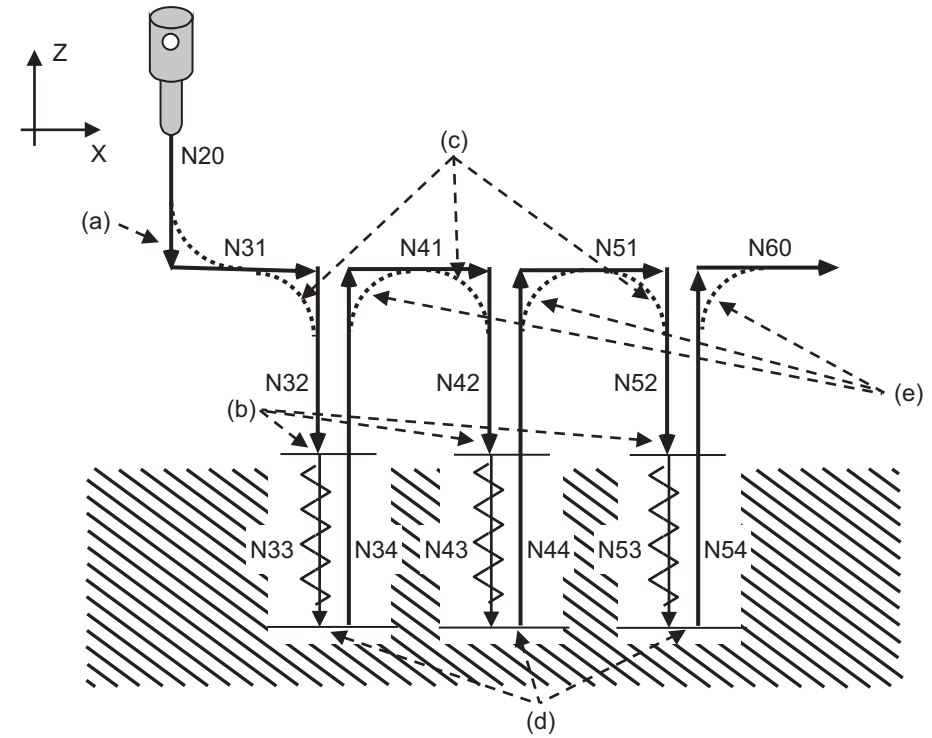
Command path from NC



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 Z-25.;	N32 G00 Z-25.;
N30 G81 X50. Y0. Z-25. R-25. F1000. L1.;	N33 G01 Z-25. F1000.;
N40 X50. ;	N34 G00 Z50.;
N50 X50. ;	
N60 G00 X50. ;	



Program path

Command path from NC

(a) Subject to rapid traverse block overlap (*1)
(Start G81 command before positioning is completed. (G81, G83 only))

(b) Subject to rapid traverse block overlap

(c) Subject to rapid traverse block overlap (*1)(*2)

(d) Not subject to rapid traverse block overlap (Not targeted)

(e) Subject to rapid traverse block overlap (*1)(*3)

(*1) When G00 is operated in pre-interpolation acceleration/deceleration, blocks subject to this function differs depending on the value of the parameter "#1253 set25/bit2".

[Relation between the blocks subject to rapid traverse block overlap during the drilling cycle and the parameter]

G00 acceleration/deceleration type	#1253 set25/bit2	Blocks ("previous block" - "following block")						
		N20-N31	N31-N32	N34-N41	N41-N42	N44-N51	N51-N52	N54-N60
Acceleration/deceleration after interpolation	0/1	○	○	○	○	○	○	○
Pre-interpolation acceleration/deceleration	0	○	○	×	○	×	○	×
	1	×	○	○	○	○	○	×

○: Subject to rapid traverse block overlap for G00

×: Not subject to rapid traverse block overalp for G00
(Motion by the conventional deceleration check)

(*2) When the ",I" address is commanded, the in-position width for fixed cycle is applied to the blocks from N31 to N32, from N41 to N42 and from N51 to N52, and the blocks are not subject to the rapid traverse block overlap.

(*3) When the ",J" address is commanded, the in-position width for fixed cycle is applied to the blocks from N34 to N41, from N44 to N51 and from N54 to N60, and the blocks are not subject to the rapid traverse block overlap.

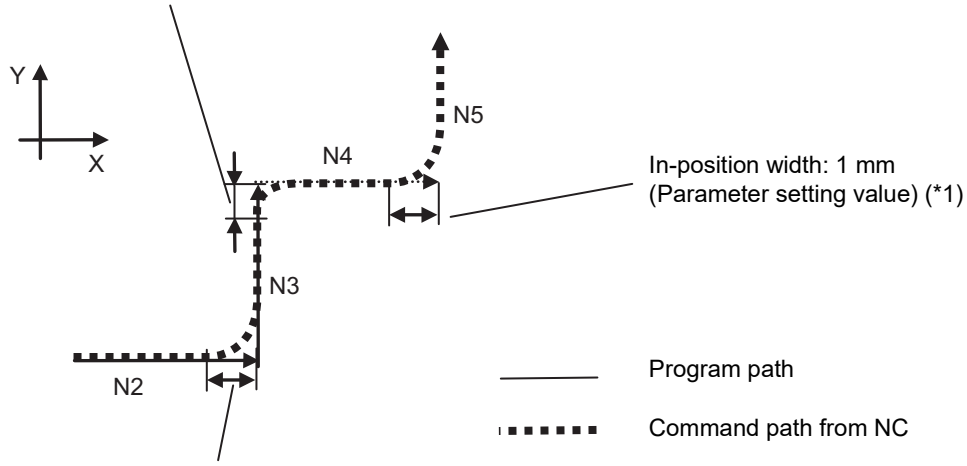
Relationship with Other Functions

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.;	Subject to rapid traverse block overlap for G00
N3 Y50. ,I1.5;	",I" address command is valid
N4 X50.;	Subject to rapid traverse block overlap for G00
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

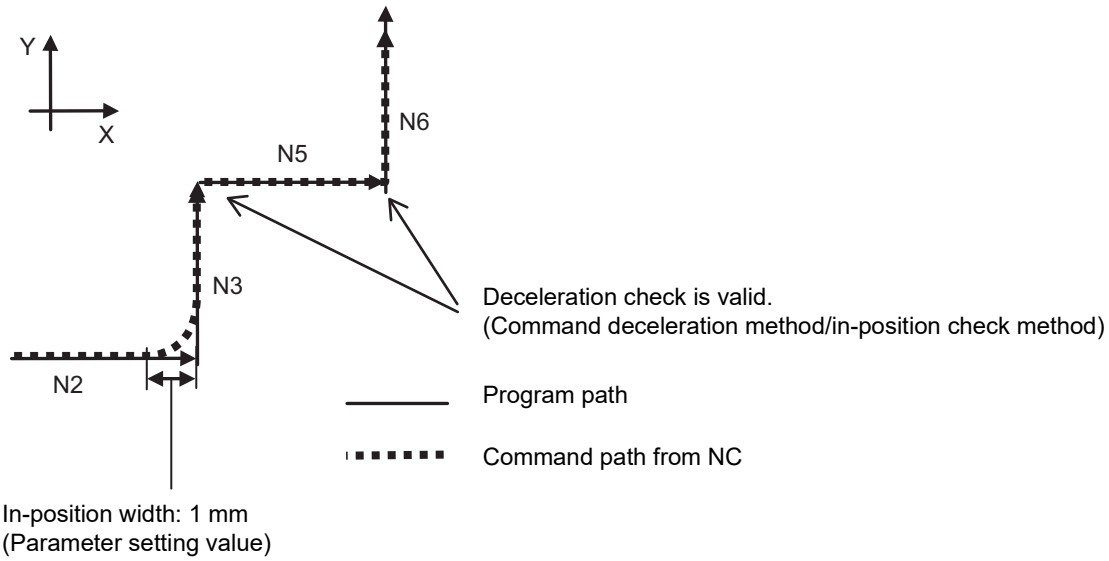
(1) The programmable in-position (",I" command) for G00 pre-interpolation acceleration/deceleration can only be used when the rapid traverse block overlap is enabled.

(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 block 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.;	Subject to rapid traverse block overlap for G00
N3 Y50.;	Deceleration check is valid.
N4 G0.5 P0;	
N5 X50.;	Deceleration check is valid.
N6 Y50.;	Deceleration check is valid.
:	



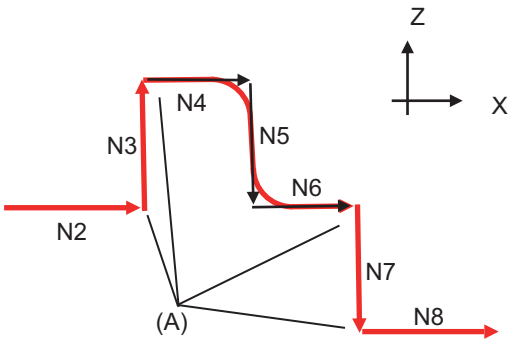
Tool compensation

In the block where the compensation amount of the tool compensation function below is changed, rapid traverse block overlap is not performed at either of the start point or the end point.

- Tool length offset
- Tool radius compensation
- 3-dimensional tool radius compensation
- Tool function (T command)
- Nose R compensation

The following example shows a case in which the Z axis length compensation amount of tool number 1 is 50 mm.

N1 G0.5 P1;	G00 rapid traverse block overlap valid command
N2 G91 G00 X50.;	
N3 G43 H1;	Tool length offset valid
N4 G00 X50.;	
N5 G00 Z-50.;	
N6 G00 X50.;	
N7 G49 ;	Tool length offset invalid
N8 G00 X50.;	
N9 G0.5 P0;	



(A) These are the start and end points of movement where the compensation amount is changed by G43/G49; therefore, the rapid traverse block overlap function is invalid.



Precautions

(1) When a block without a movement command is inserted between blocks (N1 and N2) that are subject to the rapid traverse block overlap, whether to perform the rapid traverse block overlap depends on the acceleration/deceleration mode of the N1 block.

[When N1 block is acceleration/deceleration after interpolation]

- The rapid traverse block overlap is performed.
- The parameter whose setting value is smaller between "#2631 G0olinps" and "#2632 G1olinps" is valid as in-position width.
- When the rapid traverse block overlap for G00 is disabled (#1442 G0ol = 0) and rapid traverse block overlap for G28 is enabled (#1443 G28ol = 1), block overlap is not performed even if there is a block without movement between G00 command and G28/30.

[When N1 block is acceleration/deceleration before interpolation]

- The rapid traverse block overlap is not performed.

(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) The rapid traverse block overlap is not performed during tool center point control or during inclined surface machining.

(6) The rapid traverse block overlap is not performed between a block which is operated in acceleration/deceleration after interpolation and a block which is operated in acceleration/deceleration before interpolation.

(7) When the variable-acceleration pre-interpolation acceleration/deceleration is enabled, the rapid traverse block overlap is not performed between blocks which include the linear interpolation (G01) before and after the blocks.

(8) At the joint between blocks from G00 to G28/G30 commands, if pre-read of the G28/G30 block cannot be performed in time when G00 is executed, the rapid traverse block overlap for G28 is not performed.

7.14.2 Rapid Traverse Block Overlap for G28

Function and purpose

This function enables the next block to start (overlap) without waiting for the deceleration completion at the joint between blocks in the G00-G28/G30 command or at the intermediate point of G28/G30. This reduces the cycle time of machining including the G00 or G28/G30 operation. (*1)

(*1) Cycle time reduction may not be guaranteed in every case.

#1711 cfg11/bit2	Details
0 (mode 1)	The cycle time can be reduced in normal acceleration/deceleration control.
1 (mode 2)	It is also effective to reduce the cycle time in pre-interpolation acceleration/deceleration control.

In the following cases, G00 is operated in pre-interpolation acceleration/deceleration.

- The high-speed high-accuracy control mode is selected, and "#8090 SSS ON" is set to "1".
- "#1086 G00 interp OFF" is set to "0", and "#1205 G0bdcc" (Acceleration and deceleration before G0 interpolation) is not "0".

For the rapid traverse block overlap function, also refer to "7.14 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 "7.14.1 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")

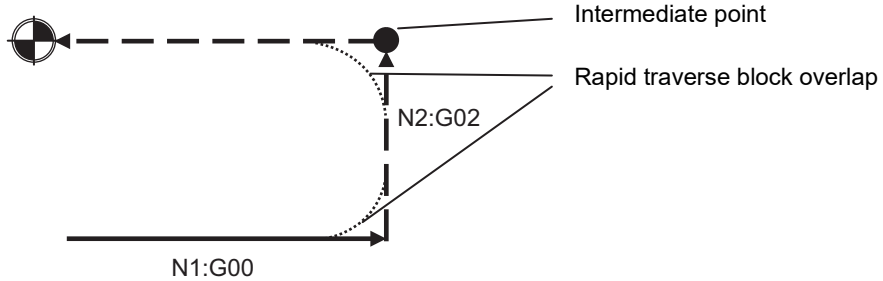
(2) High-speed reference position return is active. (Dog-type is not subject to this.)

(3) When the rapid traverse block overlap for G00 is enabled, a G00 command is followed by G28 or G30 positioning command.

Note

(1) 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").

(2) 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.

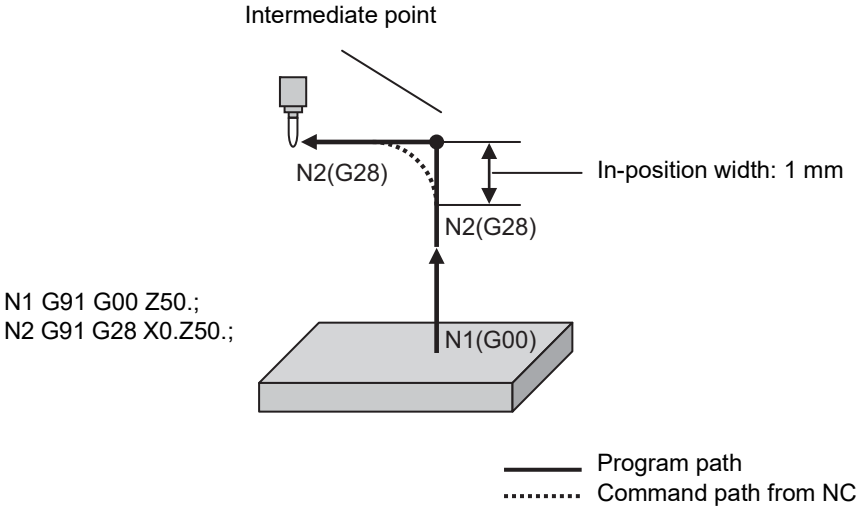
The in-position width depends on the MTB specifications (parameter "#2633 G28olinps").

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 G28olinps	0.5 mm	1 mm

Initial position of axes: X axis = -50 mm; Z axis = -100 mm



Relationship with Other Functions

Refer to "7.14 Rapid Traverse Block Overlap; G0.5 P1".

Precautions

Refer to "7.14 Rapid Traverse Block Overlap; G0.5 P1".

7.15 Automatic Corner Override

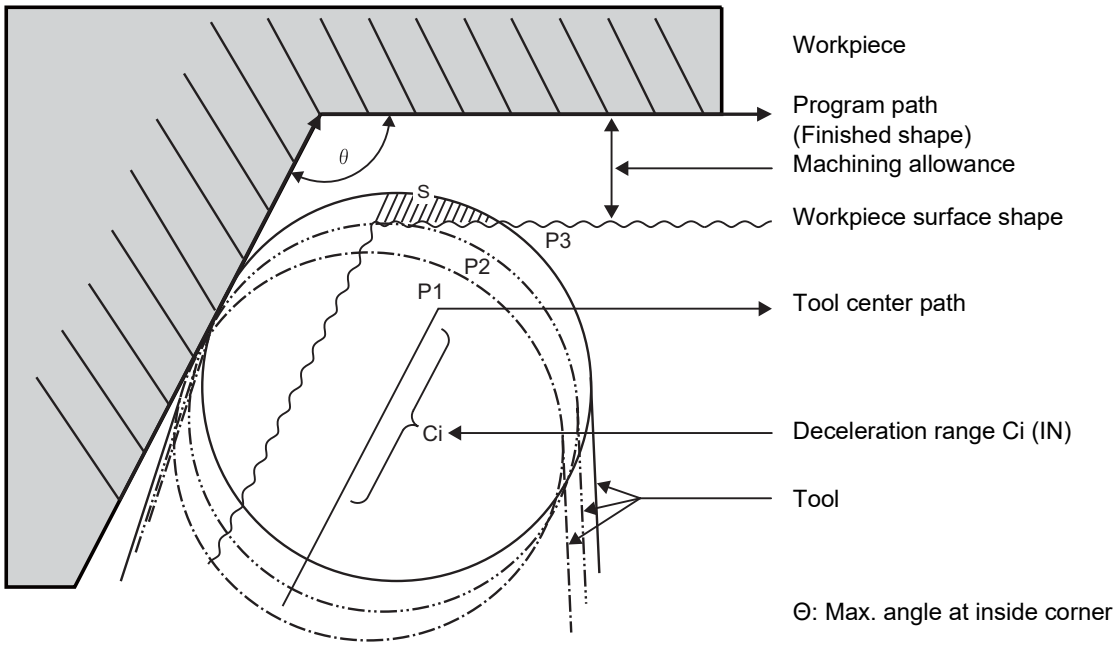
Function and purpose

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.

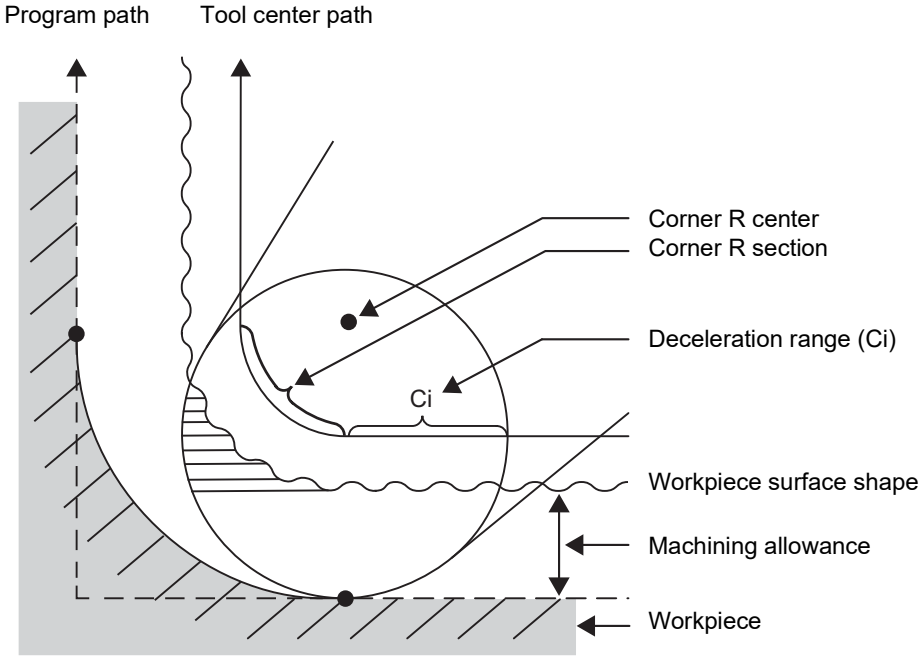
(2) If there is G62 command:
When the inside corner angle θ in the above figure is less than the angle set in the parameter, the override set into the parameter is automatically applied in the deceleration range Ci.

[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



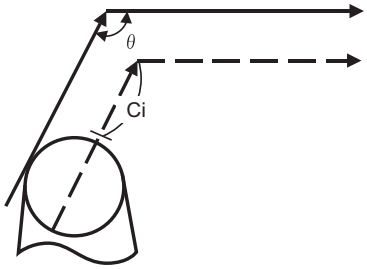
(1) The override set in the parameter is automatically applied at the deceleration range Ci and corner R section for inside offset with automatic corner R. (There is no angle check.)

Application example

The lines in the figure denote:

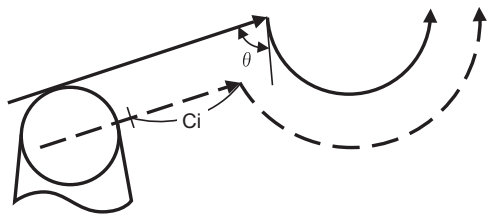
Programmed path
Tool center
Arc (inside offset)

(1) Linear - linear corner



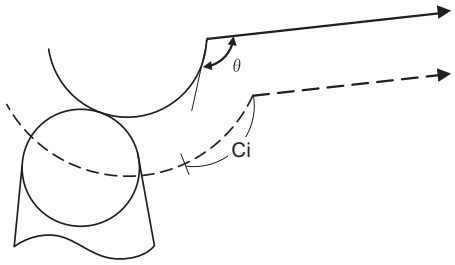
The override set in the parameter (#8007) is applied in the deceleration range Ci.

(2) Linear - arc (outside offset) corner



The override set in the parameter (#8007) is applied in the deceleration range Ci.

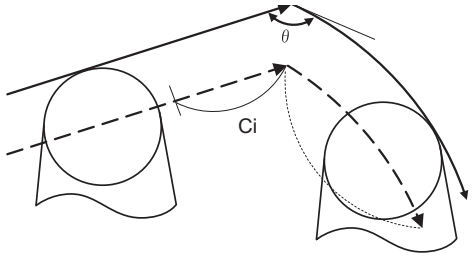
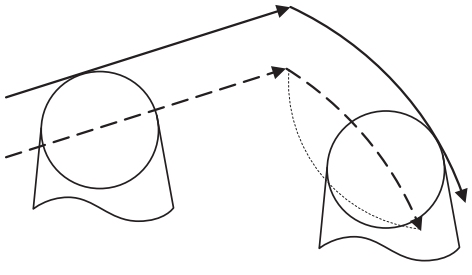
(3) Arc (outside offset) - linear corner



<Note>

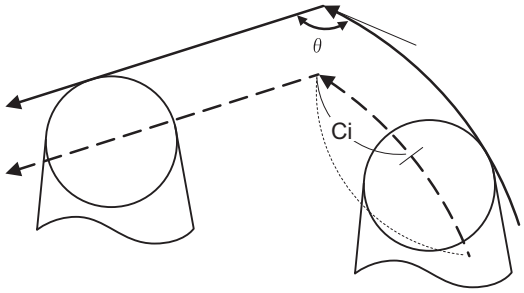
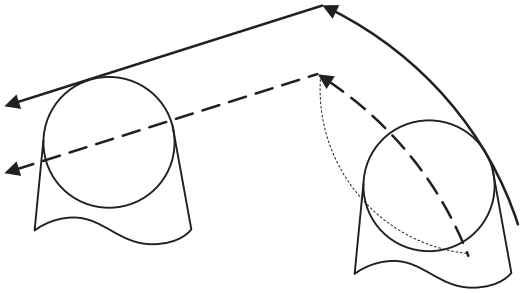
- The deceleration range Ci where the override is applied is the length of the arc with an arc command.

(4) Linear - arc (inside offset) corner

(a) If there is a G62 command:	(b) If there is no G62 command:
	
For straight lines, the override set in the parameter (#8007) is applied in the deceleration range Ci.	

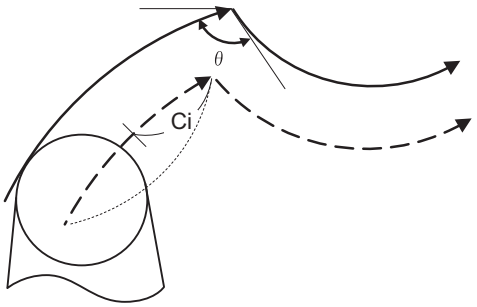
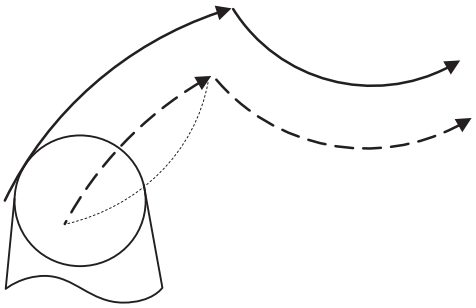
During cutting of arc (inside offset), an inner arc override is applied.

(5) Arc (inside offset) - linear corner

(a) If there is a G62 command:	(b) If there is no G62 command:
	
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)	

(*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:
	
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)	

(*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 "7.15.2 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)	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 IvOMin) 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.

7.15.1 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".

7.15.2 Inner Arc Override

Detailed description

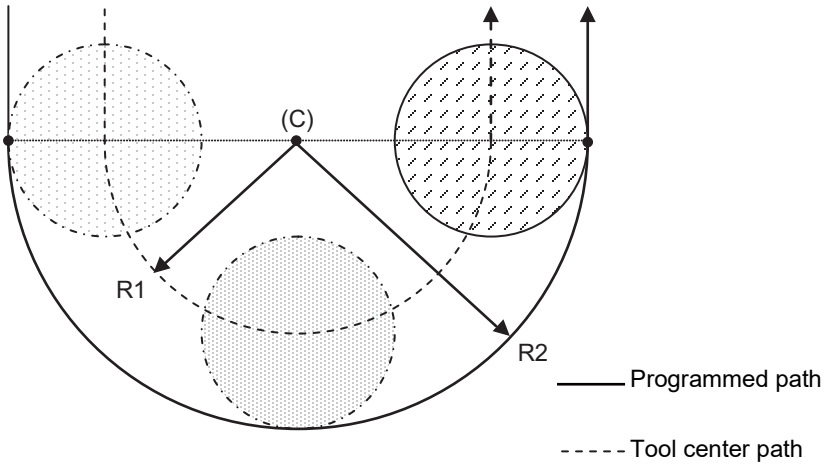
When cutting an arc (inside offset), applying the override given by the following expression to the commanded feedrate (F) causes that feedrate to become the F value for which the feedrate of the programmed path is commanded.

$$F \times \frac{R1}{R2}$$

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 x Parameter setting value / 100

(*1) R1/R2 ≤ #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 "7.15 Automatic Corner Override".

7.16 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

7.17 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

8

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.

The dwell functions are classified into two types: "dwell (time-based designation)" and "dwell (revolution-based designation)".
This chapter describes the dwell (time-based designation) function.
To perform the dwell (time-based designation), command "G04" in the asynchronous feed mode (G94). The machine waits for the specified amount of time before executing the next block.

Command format

Dwell (Time-based designation)

G94 G04 X__ ;

G94 G04 P__;

X/P	Dwell time

When the asynchronous feed mode (G94) is being executed, there is no need to specify G94.

The setting unit of the dwell time depends on the address type and the parameter setting value. For details, refer to "Detailed description".

Detailed description

(1) The decimal point command is enabled for the dwell time designation with X.

(2) When the parameter "#8112 DECIMAL PNT-P" is "1", the decimal point command is valid for the dwell time designated with the address P. When #8112 is "0", the decimal point command is invalid, and the number below the decimal point of the P command value is ignored.
When the decimal point command is valid and when it is invalid, the command range of the dwell time is respectively as follows.

The decimal point command is valid. (#8112 = 1)	**The decimal point command is invalid. (#8112 = 0)**
0 to 99999.999 (s)	0 to 99999999 (ms)

(3) When the parameter "#1078 Decpt2" is "1", the dwell time setting unit applied when there is no decimal point in the command value of the dwell time can be set to seconds. This is effective only for the addresses X and the address P for which the parameter "#8112 DECIMAL PNT-P" is "1".

(4) If any of the following conditions is met, the dwell time unit commanded by "P" is "ms"; however, the setting unit can be changed with the parameter "#19014 G04 P factor".
- "#8112 DECIMAL PNT-P" is set to "0".
- "#8112 DECIMAL PNT-P" is set to "1", "#1078 Decept2" is set to "0" and the value of address "P" has no decimal point.

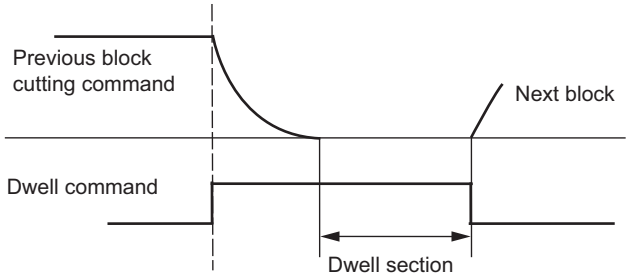
The actual duration of dwell (time) can be obtained from the following formula.

$P \times 10^{n}$

P: command value of address "P"
n: setting value of #19014 ("-3" to "3")

Setting value of #19014	Command increment (ms)	Maximum duration of dwell (ms)
-3	0.001	99999.999 (ms)
-2	0.01	999999.99 (ms)
-1	0.1	9999999.9 (ms)
0	1	99999999 (ms)
1	10	999999.99 (s)
2	100	9999999.9 (s)
3	1000	99999999 (s)

(5) When a cutting command is in the previous block, the dwell command starts counting the dwell time after the machine has decelerated and stopped. When the dwell command is issued in the same block as an M, S, T or B command, the count starts simultaneously.



(6) If a feed hold signal is input while dwelling is being performed, the count of the dwell time is interrupted. When the operation is restarted, the count of the dwell time is restarted.

(7) The dwell is valid during the interlock.

(8) The dwell is valid even for the machine lock.

(9) Depending on the MTB specifications, the dwell can be canceled by the skip signal (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

Here is a program example which satisfies the following conditions.

- #100 = 1000 ;
- The parameter "#19014 G04 P factor" is set to "0".

Command	Dwell time [s]			
	#1078 = 0		#1078 = 1	
	#8112 = 0	#8112 = 1	#8112 = 0	#8112 = 1
G04 X500 ;	0.5		500	
G04 X5000 ;	5		5000	
G04 X5 ;	5		5	
G04 X#100 ;	1000		1000	
G04 P5000 ;	5 (*1)		5 (*1)	5000
G04 P12.345 ;	0.012 (*1)	12.345	0.012 (*1)	12.345
G04 P#100 ;	1 (*1)	1000	1 (*1)	1000

(*1) When the parameter "#19014 G04 P factor" is set to a value other than "0", refer to the description for the parameter #19104 in "Detailed description".



Relationship with Other Functions

Fixed cycle command

The G04 operation in the fixed cycle subprogram follows the asynchronous feed (G94) or synchronous feed (G95) mode that is set when the fixed cycle is commanded. However, if the parameter "#8130 Dwell in rev." is "0" (invalid), the dwell (time-based designation) is carried out.

When the parameter "#19019 fixpro G04P factor" is "1", the factor of the dwell command value in the fixed cycle subprogram is changed according to the "#19014 G04 P factor" setting.
When the parameter #19019 is "0", the factor is "1" regardless of the "#19014 G04 P factor" value.

Synchronous tapping cycle command

The G04 operation in the synchronous tapping cycle subprogram is set to the dwell (time-based designation) regardless of whether the asynchronous (G94) or synchronous (G95) mode is set.

The larger value of the address P value and the setting value of "#1313 TapDwl" (Synchronous tap hole bottom wait time) is the wait time at the hole bottom in the synchronous tapping cycle.
When the parameter "#19019 fixpro G04P factor" is "1", the value obtained by multiplying the wait time value by the "#19014 G04 P factor" value is used as an actual wait time at the hole bottom.



Precautions and restrictions

(1) To use this function, the dwell time must be commanded after G04. When the dwell time is commanded before G04, the dwell time may differ from the estimated one.

8.2 Dwell (Revolution-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.

This chapter describes the dwell (revolution-based designation) function.
To perform the dwell (revolution-based designation), command "G04" in the synchronous feed mode (G95). The machine waits for the spindle to rotate the designated number of revolutions.

Command format

Dwell (Revolution-based designation)

```
G95 G04 X__ D__ ;
```

```
G95 G04 P__ D__ ;
```

X/P	Number of revolutions for dwell
D	Dwell target spindle (*1) For spindle number: 1 to n (n: Maximum number of available spindles) For spindle name: 1 to 9

(*1) There are two types of spindle designation methods: spindle number method and spindle name method, which vary depending on the MTB specifications. When the name (1 to 9) has been specified for all the spindles with the parameter "#3077 Sname" (Spindle command name), a command is given with the spindle name. For others, command with the spindle number.

G95 is not necessary when there is a "D" command in the same block as G04.
G95 is not necessary in the synchronous feed mode (G95).

The setting unit of the number of revolutions for dwell depends on the address type and the parameter setting value. For details, refer to "Detailed description".

Detailed description

(1) In multiple-spindle control II, the spindle designated with the address D is invalid. To command the dwell (revolution-based designation), G95 needs to be designated.

(2) The decimal point command is valid for the number of revolutions for dwell designated with the address X.

(3) When the parameter "#8112 DECIMAL PNT-P" is "1", the decimal point command is valid for the number of revolutions for dwell designated with the address P. When #8112 is "0", the decimal point command is invalid, and the number below the decimal point of the P command value is ignored.

(4) When the decimal point command is valid and when it is invalid, the command range of the number of revolutions for dwell is respectively as follows.

The decimal point command is valid (#8112 = 1)	The decimal point command is invalid. (#8112 = 0)
0 to 99999.999 (rev)	0 to 99999999 (0.001 rev)

(5) When the parameter "#1078 Decpt2" is "1", the setting unit applied when there is no decimal point in the command value of the number of revolutions for dwell can be set to revolutions (rev).

This is effective only for the addresses X and the address P for which the parameter "#8112 DECIMAL PNT-P" is "1".

(6) If any of the following conditions is met, the unit of the number of revolutions for dwell commanded by P is "0.001 (rev)", but the setting unit can be changed with the parameter "#19014 G04 P factor".
- "#8112 DECIMAL PNT-P" is set to "0".
- "#8112 DECIMAL PNT-P" is set to "1", "#1078 Decimal pnt type 2" is set to "0" and the value of address "P" has no decimal point.

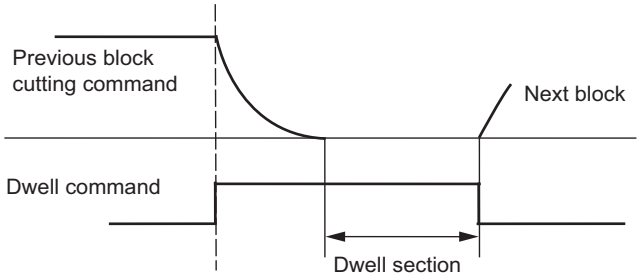
The actual duration of dwell (number of revolutions) can be obtained from the following formula.

$P \times 10^{n}$

P: command value of address "P"
n: setting value of #19014 ("-3" to "3")

Setting value of #19014	Command increment (rev)	Maximum duration of dwell (rev)
-3	0.000001	99.999999
-2	0.00001	999.99999
-1	0.0001	9999.9999
0	0.001	99999.999
1	0.01	999999.99
2	0.1	9999999.9
3	1	99999999

(7) When a cutting command is in the previous block, the dwell command starts counting the number of revolutions for dwell after the machine has decelerated and stopped. When the dwell command is issued in the same block as an M, S, T or B command, the count starts simultaneously.



When the number of revolutions for dwell is set to 100 (rev) and the spindle rotation speed is set to 1000 (r/min), the machine waits for 6 seconds during the dwell interval.

(8) Details for override

The dwell time is not affected by cutting feed override, rapid traverse override, or manual traverse override regardless of whether the dwell override/miscellaneous function time override function is valid or invalid ("#1436 mstsyn").

However, if the spindle rotation speed is changed by spindle override, the spindle rotation wait time is also changed.

(9) If a feed hold signal is input while dwelling is performed, the count of the number of revolutions for dwell is interrupted. When the operation is restarted, the count of the number of revolutions for dwell is restarted.

(10) The dwell is valid during the interlock.

(11) The dwell is valid even for the machine lock.

(12) Depending on the MTB specifications, the dwell can be canceled by the skip signal (parameter "#1173 dwlskp"). If the set skip signal is input during dwelling, dwelling is skipped by the number of remaining spindle rotation times, and the following block will be executed.

(13) Dwell is also stopped while the spindle is stopped. When the spindle starts to rotate again, dwell also restarts.

(14) When the parameter "#8130 Dwell in rev." is "1", dwell (revolution-based designation) is executed in the synchronous feed mode (G95).
When there is a "D" command in the same block as G04, dwell (revolution-based designation) is executed regardless of the #8130 setting value and irrespective of asynchronous mode or synchronous mode.

(15) The spindle that is targeted for dwelling varies depending on the following conditions.

When multiple-spindle control II is valid:

Dwell is applied to the spindle selected by the "encoder selection" signal from the PLC.

The spindle designated in "D" command is invalid.



Relationship with Other Functions

Spindle selection command

The spindle designation (D) for the dwell (revolution-based designation) is valid only in the G04 block.

Fixed cycle command

The G04 operation in the fixed cycle subprogram follows the asynchronous feed (G94) or synchronous feed (G95) mode that is set when the fixed cycle is commanded. However, if the parameter "#8130 Dwell in rev." is "0" (invalid), the dwell (time-based designation) is carried out.

When the parameter "#19019 fixpro G04P factor" is "1", the factor of the dwell command value in the fixed cycle subprogram is changed according to the "#19014 G04 P factor" setting. When the parameter #19019 is "0", the factor is "1" regardless of the "#19014 G04 P factor" value.

Synchronous tapping cycle command

The G04 operation in the synchronous tapping cycle subprogram is set to the dwell (time-based designation) regardless of whether the asynchronous (G94) or synchronous (G95) mode is set.



Precautions

(1) To use this function, the number of revolutions for dwell must be commanded after G04. When the number of revolutions for dwell is commanded before G04, the number of revolutions for dwell may differ from the estimated one.

9

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 multiple commands can be issued 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 the number of M commands in a single block is greater than the setting value of the parameter "#12005 Mfig", the commands issued later are valid.

Whether to BCD output or binary output the 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.

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) Individual 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.
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").
Sequence processing is unnecessary for the M commands (No M code signal nor strove signal is output).

Subprogram call/completion (M98, M99)

These commands are used as the return instructions from branch destination subprograms and branches to subprograms.
Sequence processing is unnecessary for the M commands (No M code signal nor strove signal is 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, 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 second miscellaneous function (A, B, or C) depends on the MTB specifications (parameter "#1170 M2name"). (Except the address that is used for the axis name and the increment command axis name.)

The second miscellaneous function can be issued for up to 4 sets in a block. The number of commands that can be issued within the same block depends on the MTB specifications (parameter "#12011 Bfig").
Whether to BCD output or binary output the second 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 secondary 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 secondary miscellaneous function.

		Additional axis name		
		A	**B**	**C**
2nd miscellaneous function	A	-	○	○
	B	○	-	○
	C	○	○	-



Precautions

(1) When A has been assigned as the secondary miscellaneous function address, the following commands cannot be used.
- Linear angle commands (",A" can be used.)
- Geometric command

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 ;

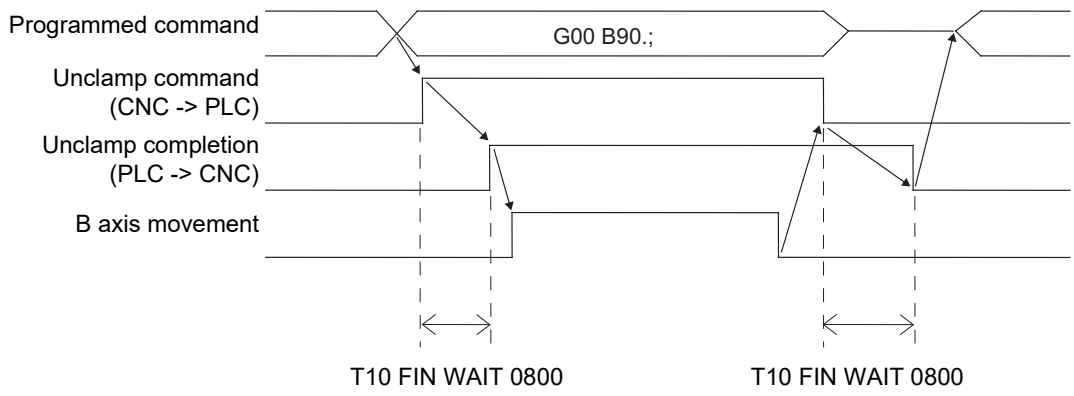
B	Index table indexing axis (designated with parameter "#2076 index_x")

Detailed description

Type A operations

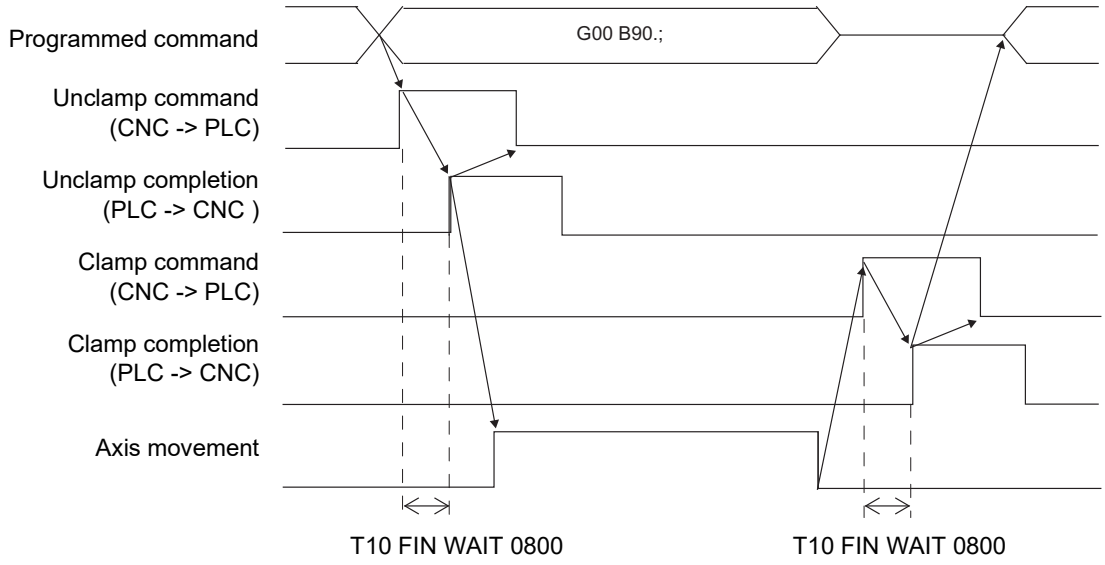
(1) The movement command (either absolute or incremental) for the selected axis is executed with the program command.
(2) The "Unclamp command" signal is now output prior to the axis movement.
(3) When the axes are unclamped, the "Unclamp completion" signal is turned ON by the PLC.
(Turn the signal ON after performing required process such as servo ON or the unclamp process.)
(4) After confirming 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 by the PLC.
(Turn the signal OFF after performing required process such as in-position check, servo OFF or the clamp process.)
(7) After confirming that the "Unclamp completion" signal is turned 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 confirming the "Unclamp completion" signal, turn the "Unclamp command" signal OFF, and the designated axis starts moving.
(5) Turn the "Unclamp completion" signal OFF by 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 by the PLC.
(Turn the signal OFF after performing required process such as in-position check, servo OFF or the clamp process.)
(8) After confirming 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 by 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" signal 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	⬧Set the parameter "#8209 G60 SHIFT" in indexing increment. If the setting value of #8209 is not in the indexing increment, the program error (P20) occurs. ⬧When an axis command that cannot be divided in the command unit is issued, the program error (P20) occurs. ⬧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.
Servo ON/OFF signal control	Perform the required process on the PLC.

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	(*2)
	G29 -> G00	(*2)
Tool change position return 1 to 6 (G30.1 to G30.6)	G00 -> G30.1	(*2)
Normal line control cancel (G40.1)	G40.1 -> G00	(*2)
Basic machine coordinate system selection (G53)	G00 -> G53	(*3)
	G53 -> G00	
Unidirectional positioning (G60)	G00 -> G60	(*2)
	G60 -> G00	(*1)
	G60 -> G60	
Program stop (M00)	M00	(*2)
Optional stop (M01)	M01	(*2)

(*1) The clamp and unclamp operations are NOT executed between blocks.

(*2) The clamp and unclamp operations are 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).

(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
Automatic 1st reference position return (G28)	G00 -> G28	(*1)	(*2)
	G28 -> G00	(*3)	(*4)
2nd, 3rd, and 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.

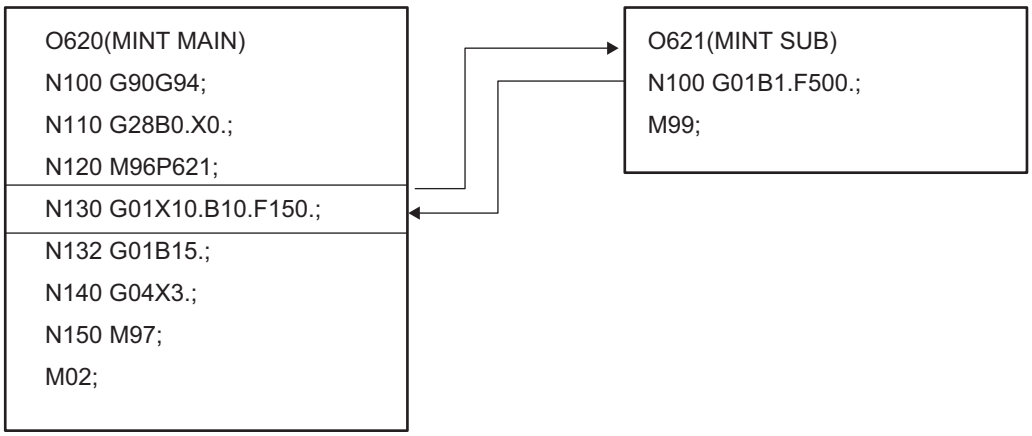
Parameter		Settings
#1112 S_TRG	0	Edge trigger mode
#1113 INT_2	0	Immediately start the interrupt program without waiting for the completion of currently executing block.
#8101 MACRO SINGLE	1	

(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] **[Interrupt program]**



(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 movement by "B1." command of N100 block in the interrupt program O621.

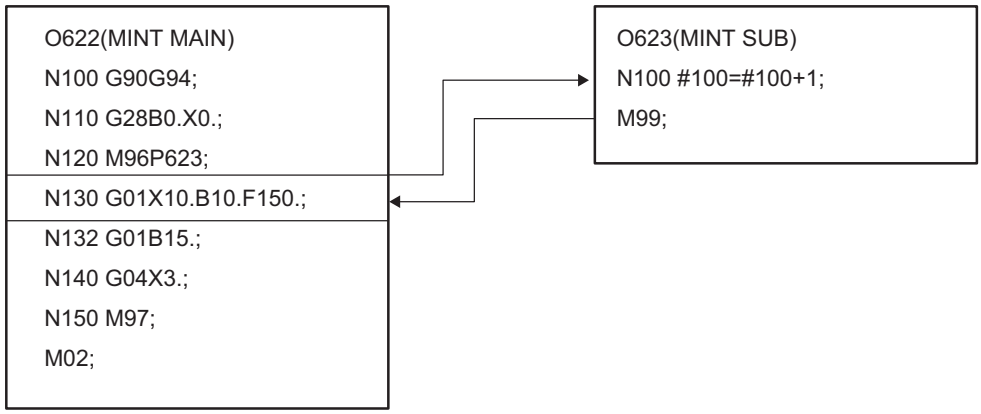
(d) Performs unclamp operation at the beginning of main program N132 block, and performs clamp operation after axis movement.

(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)

[Main program] **[Interrupt program]**



(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 the "Unclamp command" signal 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.

10

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.

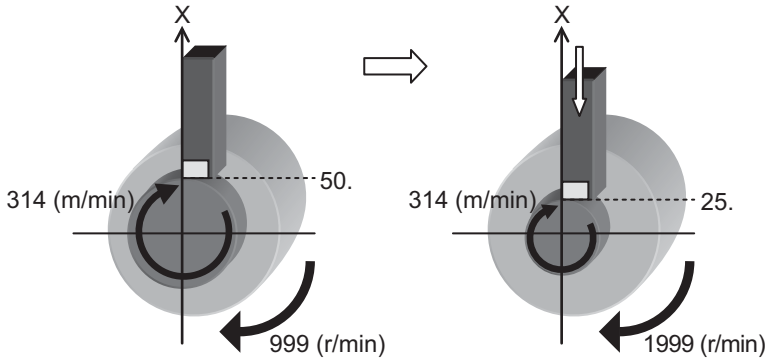
10.2 Constant Surface Speed Control; G96, G97

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: 25 mm (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 (50 mm to 25 mm).

$$\text{Spindle rotation speed (r/min)} = \frac{\text{Surface speed (m/min)}}{\text{G96 command value}} \Big/ \frac{\text{Workpiece surface (m/r)}}{\text{Automatically calculated from the workpiece zero point and tool nose position}}$$

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, the program error (P35) occurs.

(3) If the value of the P command exceeds the allowable range, the program error (P133) occurs.

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 the 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 command: When the spindle forward rotation signal is input from the user PLC by the M03 command, the spindle runs forward.
(The spindle speed is set to the previously commanded rotation speed.)

(b) G96 S100 X30. command: If an error occurs, axis movement is performed until the program is reset.

(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 clamp rotation speed initialization

(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.

(7) If a command is issued from other part systems to the spindle for constant surface speed control while the operation error (M01 1043) occurs, the rotation speed is set to the commanded value.

(8) If the program is interrupted by reset or emergency stop while the operation error (M01 1043) occurs, the constant surface speed control mode is canceled, causing the operation error to be canceled. Whether to initialize the modal by reset depends on the MTB specifications (parameter "#1210 RstGmd/bit10").

(9) Arbitrary axis exchange cannot be commanded for the axis included in the part system in which the operation error (M01 1043) occurs. Whether to wait until axis exchange becomes possible or cause the operation error (M01 1101) during waiting depends on the MTB specifications (parameter "#1432 Ax_Chg_Spec/bit0").

(10) If the maximum clamp rotation speed command is issued from other part systems to the spindle for constant surface speed control while the operation error (M01 1043) occurs, the operation depends on the MBT specifications (parameter "#1300 ext36/bit2" (Part system-based spindle clamp speed management)).

#1300 ext36/bit2 = 0: The operation error is canceled, and constant surface speed control starts.

#1300 ext36/bit2 = 1: The operation error is not canceled.

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 re-issued 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. The program error (P481) occurs. The cylindrical interpolation cannot be commanded during the constant surface speed control mode. The program error (P485) occurs.
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. The operation error (M01 1113) occurs.

Function name	Operation
Tapping cycle (G84/G88) Synchronous tapping cycle (G84/G88)	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. 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. 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. The operation error (M01 1113) occurs. The synchronous tapping cycle command cannot be executed in the constant surface speed control mode. The program error (P182) occurs. Also, the constant surface speed command cannot be executed during execution of the synchronous tapping cycle command. The program error (P186) occurs.
Arbitrary axis exchange (G140/G141)	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. The spindle rotation speed is maintained until the constant surface speed control axes are returned to the original arrangement or the constant surface speed control is canceled. The axis included in the part system in which the operation error (M01 1043) occurs cannot be exchanged.
Spindle synchronization control (G114.1) Tool spindle synchronization I (G114.2)	The constant surface speed control can be conducted for the reference spindle during spindle synchronization or tool spindle synchronization I. If the constant surface speed control command is issued to the synchronized spindle, the spindle synchronization command has priority, and the spindle rotates at the speed that is synchronized with the commanded speed of the reference spindle. The constant surface speed control for the synchronized spindle becomes valid after the spindle synchronization has been canceled. To conduct the constant surface speed control, issue the command to the reference spindle.
Tool spindle synchronization II (G114.3)	The constant surface speed control can be conducted for the reference spindle during tool spindle synchronization II.
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 command max./min. command value over" signal (X1883) turns 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_" or "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.

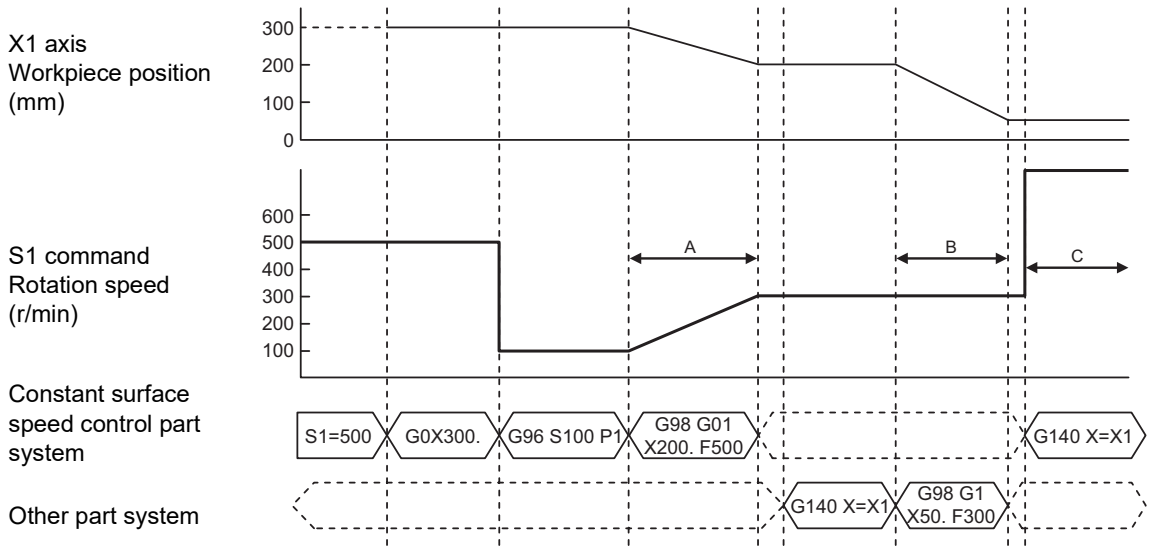
(4) If an axis number not registered in the command part system is commanded when the constant surface speed command is specified, it causes the program error (P133).

(5) When the arbitrary axis exchange command is specified in the constant surface speed control, the spindle speed clamp command is valid even if the spindle rotation speed is maintained.

(6) Note that the rotation speed of the constant surface speed spindle may vary significantly at restart of constant surface speed processing when constant surface speed control axes are repositioned until they are rearranged by the arbitrary axis exchange command, they are returned to the original arrangement after the spindle rotation speed has been kept, and constant surface speed processing restarts.

Machining program		Control axis sequence in constant surface speed command part system			Details of operation
Part system that the constant surface speed was commanded	Other part system	1st axis	2nd axis	3rd axis	
G140 X=X1 Z=Z1 C=C1 :	: :	X1 ↓	Z1 ↓	C1 ↓	
G96 S100 P1 : :	: : :	↓ ↓ ↓	↓ ↓ ↓	↓ ↓ ↓	Conducts constant surface speed control for the X1 axis.
: : : : : :	G140 X=X1 : : : : :	Z1 ↓ ↓ ↓ ↓ ↓	C1 ↓ ↓ ↓ ↓ ↓	- - - - - -	The first axis has been changed from the X1 axis to the Z1 axis; therefore, the spindle rotation speed is set to the constant rotation speed according to the final command value of the constant surface speed for the X1 axis.
G140 X=X1 :	: :	X1 ↓	- -	- -	The first axis returns to the X1 axis, and the constant surface speed calculation restarts.

<Operation diagram>



Interval A: The spindle rotation speed varies by constant surface speed control with changes of the X1 axis position.

Interval B: The X1 axis position is changed, but the spindle rotation speed does not change because the program is in the process of maintaining the spindle rotation.

Interval C: The surface speed calculation restarts, so the spindle rotation speed changes at once.

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

(1) 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_;".

(2) 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.

(3) 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 zero point 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. However, it can be switched 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. Switching to the C axis mode is performed with the zero point return type or deceleration stop type.
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/bit9, bitA").

(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 address or absolute command (G90). If the axis is designated with the incremental address or incremental command (G91), it causes a program error (P32).
- The zero point 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 zero point 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 zero point 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 zero point 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.

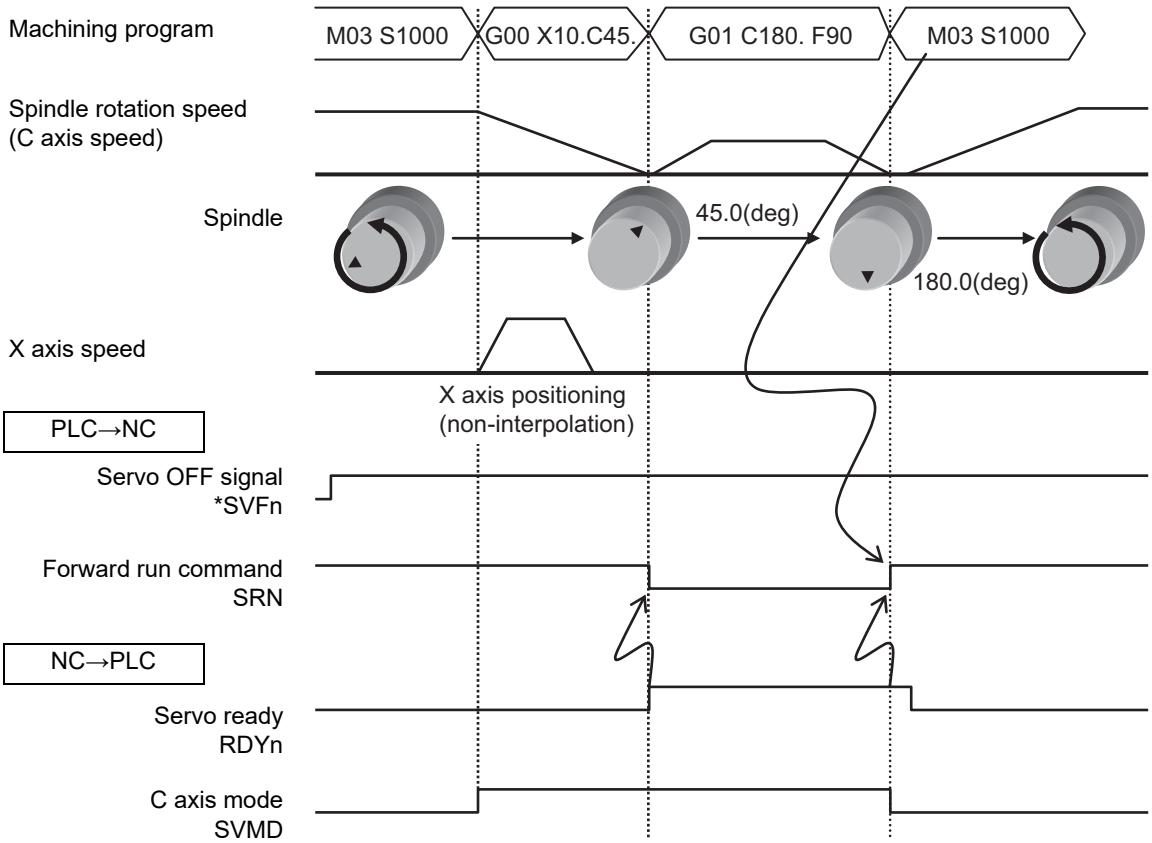
(3) Example in which the mode is not switched from the C axis mode to the spindle mode

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.;	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.
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

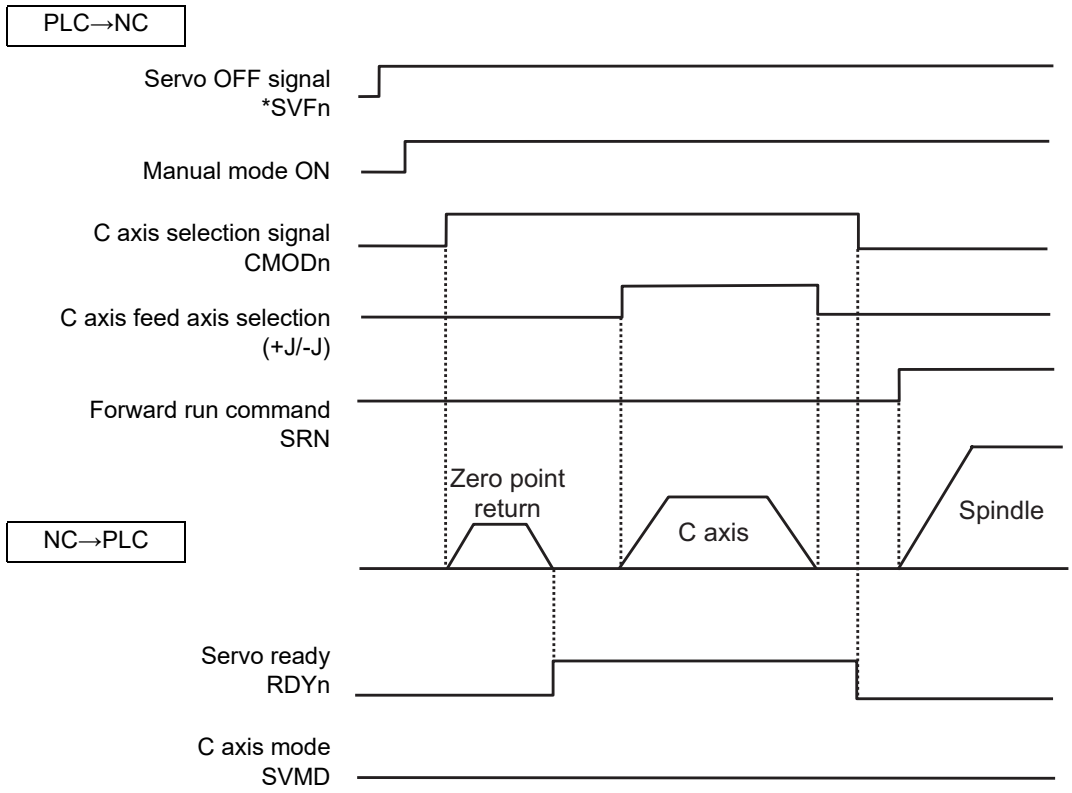
When the program command method is selected, switching operation is performed with the zero point return type.



Manual operation with the program command method selected

To rotate 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 while the "Servo OFF" signal (*SVFn) is ON. When switching to the spindle mode, change the C axis selection signal from ON to OFF. The switching operation is performed with the zero point return type or deceleration stop type. (Refer to "Designating the zero point return type or deceleration stop 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).

[C axis switching in manual operation mode (zero point return type)]



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)	

Designating the zero point return type or deceleration stop type

The operation differs depending on the method to switch from the spindle mode to the C axis mode. (The method depends on the MTB specifications.)

Method to switch	Operation contents
G00 command	The operation is performed with the zero point return type regardless of the setting of the parameter "#3106 zrn_typ/bit8" (Zero point return specifications).
"C axis selection" signal (CMOD)	The operation is performed as shown in the table below depending on combination of the parameters.
The power is turned ON when control mode at power ON is C axis mode ("#3129 cax_spec/bit2" is set to "1").	

#1711 cfg11 bit3	#3106 zrn_typ bit8	#3129 cax-_spec bit1	Operation contents
0	-	-	The zero point return to the Z phase position is performed.
1	0	-	The zero point return to the Z phase position is performed.
	1	0	The coordinate of the deceleration stop position is established by using the Z phase position as the coordinate zero point.
		1	The deceleration stop position is used as the coordinate zero point.

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)

The mode is switched to the C axis mode regardless of the state of the "Spindle orientation command" signal (ORC). However, in the C axis mode, the "Spindle orientation command" signal (ORC) is invalid.

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 zero point 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.

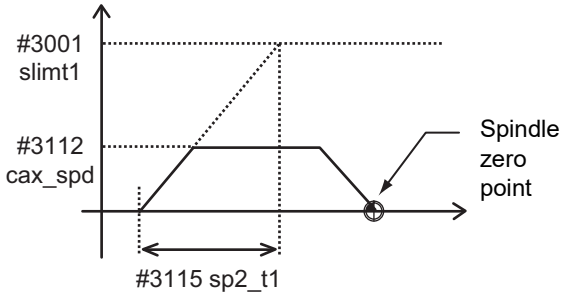
Spindle-mode servo motor control

The spindle position control is valid, excluding the following differences.

(1) Speed pattern of the zero point return operation

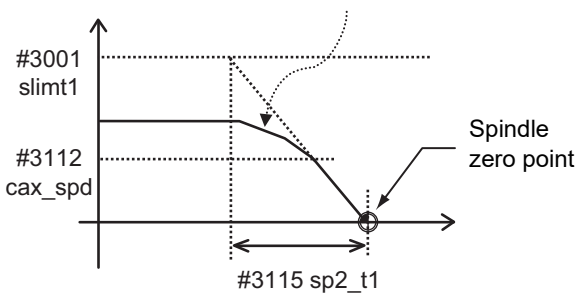
The zero point return from the stop mode is performed by the same operation as for the normal spindle. However, when the spindle returns from the rotation mode to the zero point, the rotation speed decelerates up to the C axis zero point return speed with the multi-step acceleration/deceleration pattern. 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 zero point return speed depend on the MTB specifications (parameters "#3054 sptc1" to "#3061 spdiv1" and "#3112 cax_spd").

Stop mode



Rotation mode

Multi-step acceleration/deceleration



Analog spindle, spindle control with pulse train output

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 address or absolute command (G90). If the incremental address or incremental 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").

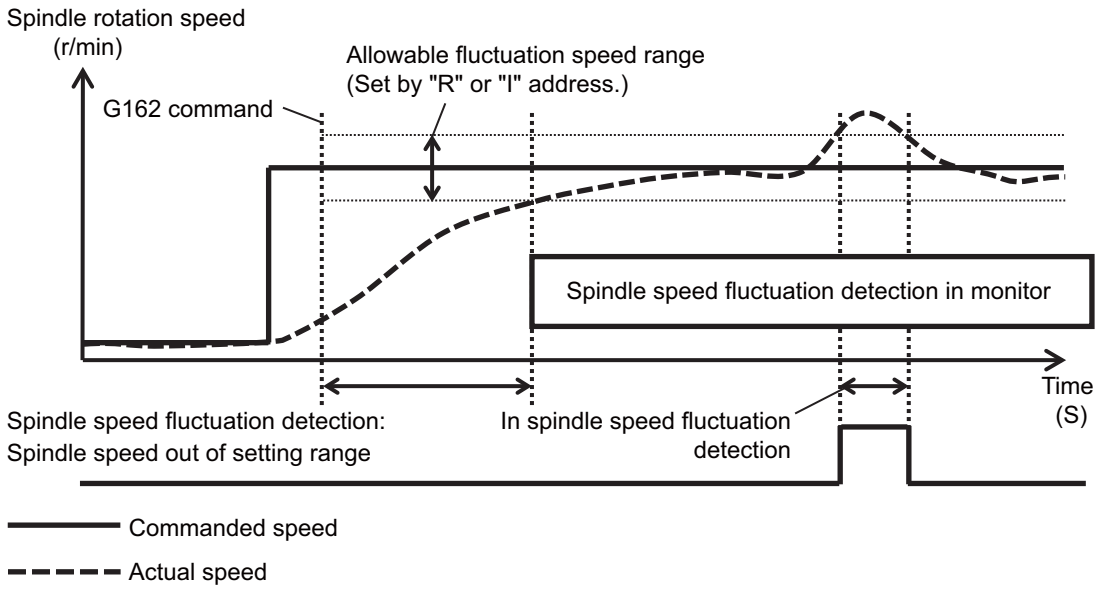
(6) When the Z phase detection speed limit is enabled (*1) and S command is made with the Z phase not detected, Z phase detection speed is limited so that it is less than or equal to the S command speed.

(*1) The validity depends on the MTB specifications (parameter "#3106 zrn_typ/bit3" is set to "1" and "#1762 cfgPR12/bit6" is set to "1").

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	♦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)	♦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 "#43071 sp_spd_flc_dtc_p" is used. ♦When the command value exceeds the command range, the program error (P35) occurs.
Q	1 to 100 (%)	♦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 sut" is used. ♦When the command value exceeds the command range, the program error (P35) occurs.

Address	Command range (unit)	Remarks
R	1 to 100 (%)	♦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 "#43072 sp_spd_flc_dtc_r" is used. ♦When the command value exceeds the command range, the program error (P35) occurs. ♦When the speed deviation for the command speed is smaller than 45 r/min, the allowable range of the speed deviation is 45 r/min.
I	0 to 999999 (r/min)	♦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 "#43073 sp_spd_flc_dtc_i" 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.

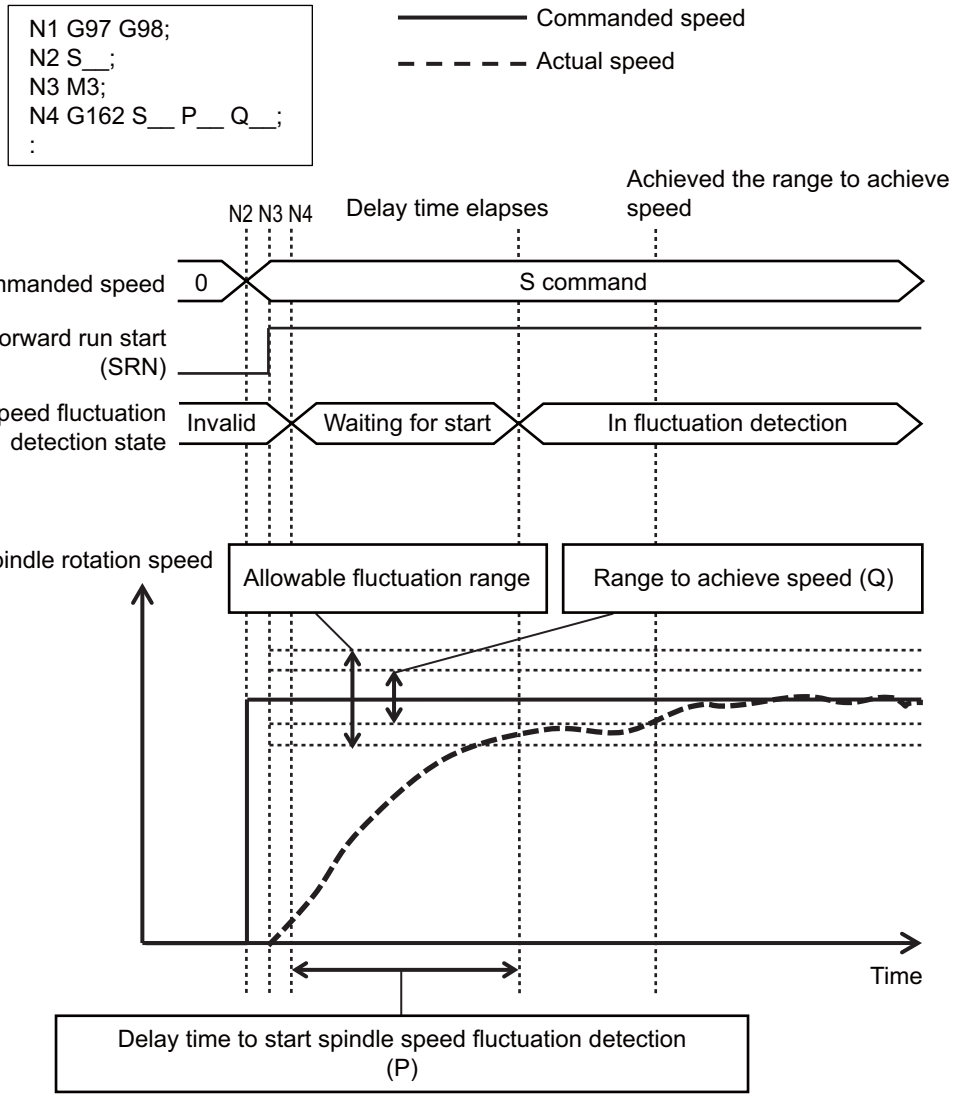
Operation example

Start timing of spindle speed fluctuation detection

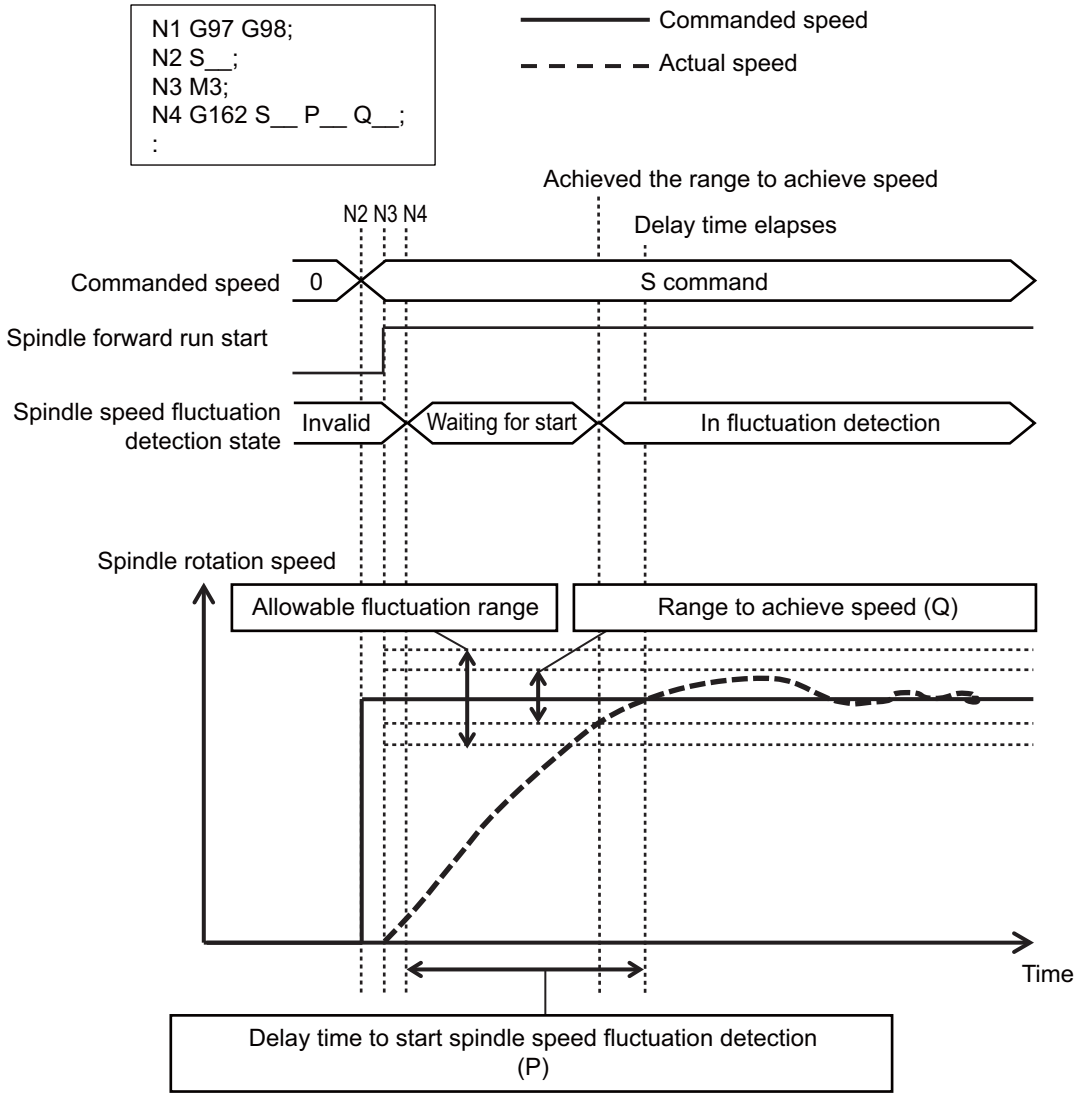
When one of the following conditions is satisfied after G162 command, the spindle speed fluctuation detection starts:

- 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

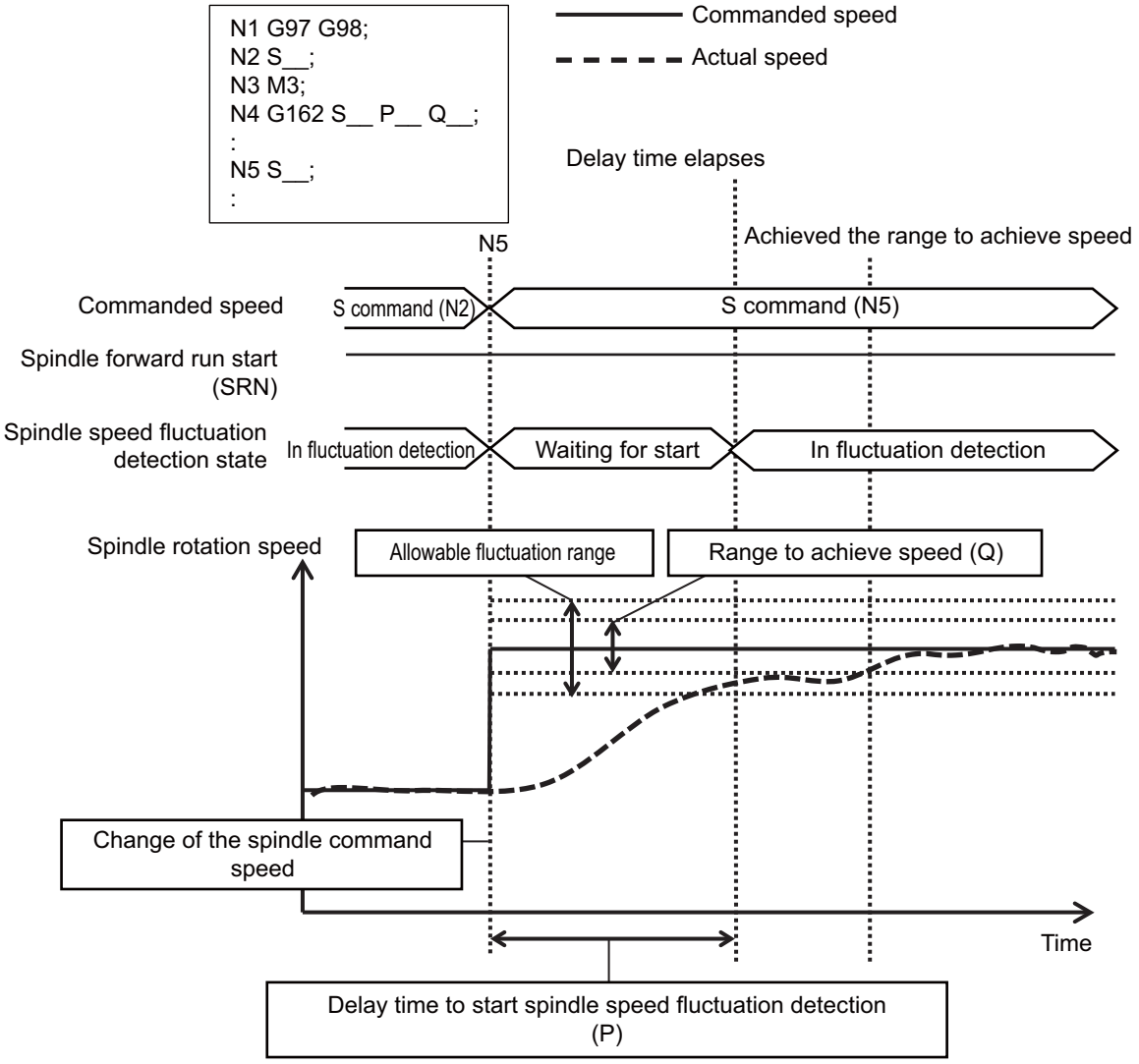


(2) Case in which the spindle actual speed is within the detection range to achieve spindle speed (set by "Q")



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.

11

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.

12

Tool Compensation Functions

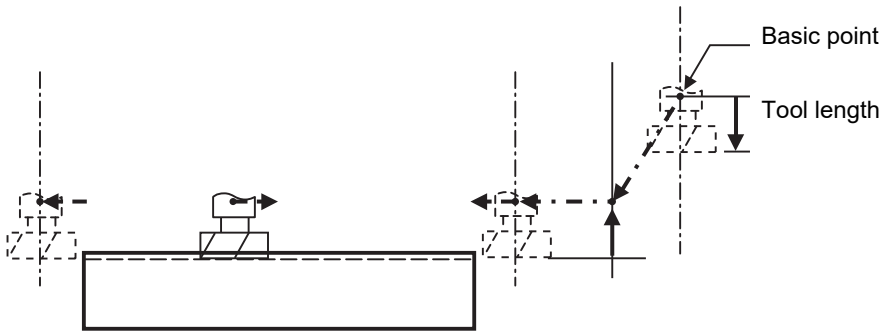
12.1 Tool Compensation

12.1.1 Tool Compensation

Function and purpose

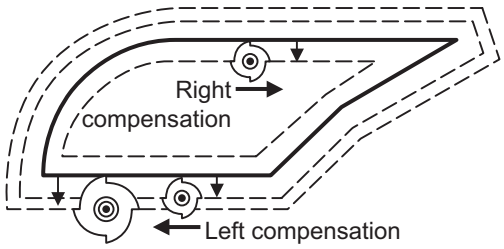
The basic tool compensation function includes the tool length compensation and tool radius compensation. Each compensation amount is designated with the tool compensation No. Each compensation amount is input from the setting and display unit or the program.

Tool length compensation



(Side view)

Tool radius compensation



(Plane view)

Tool compensation memory

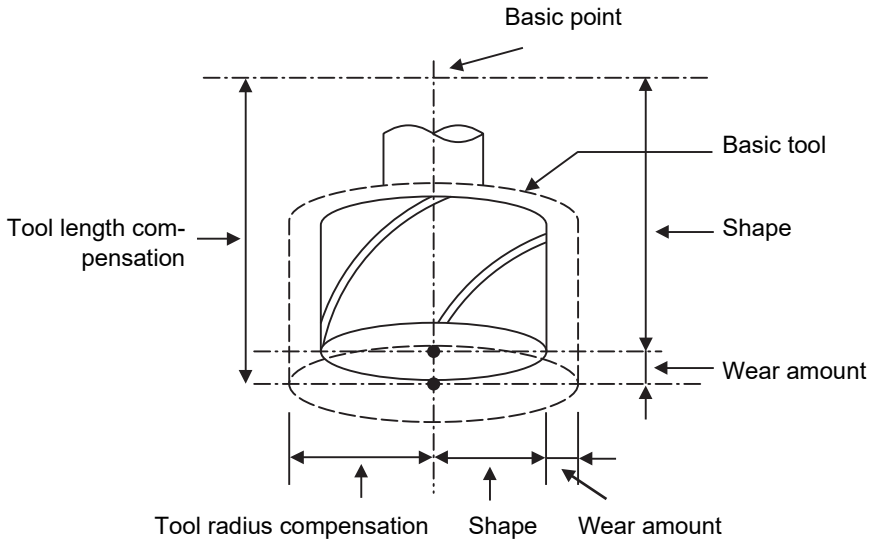
There are two types of tool compensation memories, types I and II, used to set and select the tool compensation amount. (The type used is determined by the MTB specifications.)
Each of types I and II can be changed to type III depending on the MTB settings (parameter "#1046 T-ofs disp type"). If the 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.



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.

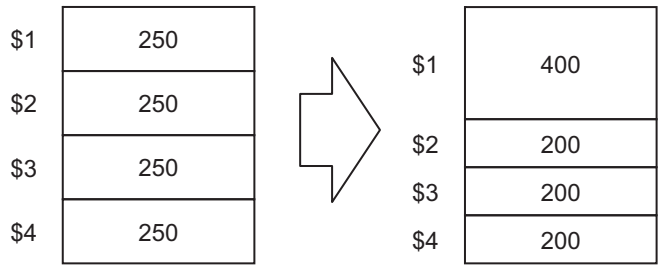
Fixed allocation: Automatically and evenly allocates to each part system.

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 1000. Number of tool offset sets in system is the total number of tool offset sets of all part systems.

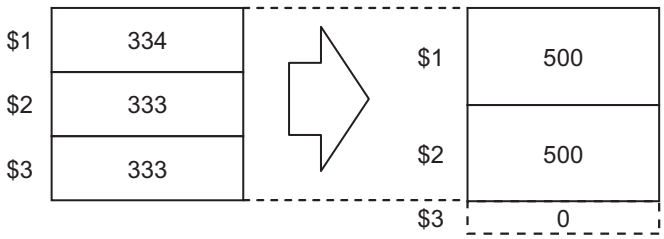
(1) Arbitrary allocation (with #1438 = 1)

The number of tool offset sets allocated to each part system depends on the MTB specifications (parameter "#12054 Tol-ofsnum").

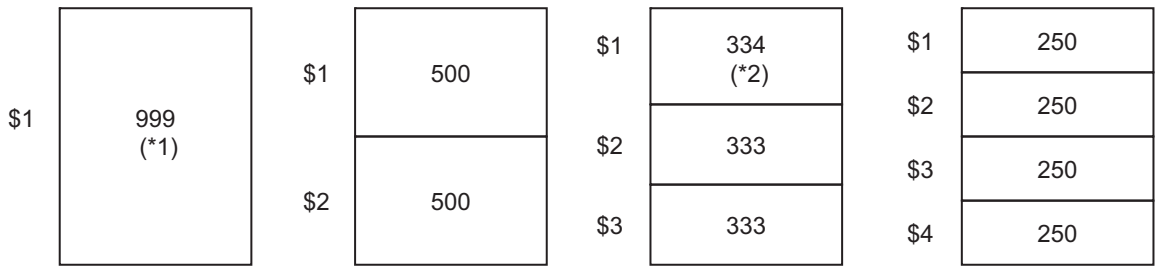
(a) When the number of tool offset sets is increased for the 1st part system of 4-part system



(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.

Note

- The number of the part systems depends on the model and MTB specifications.



Precautions

(1) For 1-part system, up to the number of tool offset sets in the system is available regardless of the parameter setting.

(2) 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.

(3) 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.

(4) Even if the specification allows arbitrary allocation, fixed allocation is applied if the parameter "#12054 Tol-Ofsnum" is set to "0".

(5) When entering offset data, if the number of offset data exceeds that of current tool offset sets, the excess offset data cannot be entered.

(6) When the number of tool offset sets is changed, erase all the tools (*1) on the maintenance screen, then register the tool compensation data again. To change the number of tool offset sets, contact the MTB representative.

(*1) When all the tools are erased, the tool compensation data is also erased. Output the tool offset data file (TOOL.OFS) before erasing all the tools to back up the tool compensation data.

(7) For fixed allocation, when the number of the tool offsets sets is "200" and the parameter "#1046 T-ofs disp type" (Tool compensation type switch) (M system) is set to "1", the number of tool offsets sets in the system is "99".

12.2 Tool Length Offset/Cancel; G43, G44 / G49

Function and purpose

The end position of the movement command for each axis can be compensated for by the preset offset 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 offset amount using this function.

Command format

Tool length offset start

```
G43 Zz Hh ; (+ direction)
```

```
G44 Zz Hh ; (- direction)
```

Tool length offset cancel

```
G49 Zz  ;
```

Detailed description

When the tool length offset start command (G43/G44) or the tool length offset cancel command (G49) is issued, there is a method to move the axis by the compensation amount, and a method to shift the coordinates by the compensation amount. This setting depends on the MTB specifications (parameter "#1268 ext04/bit6").

#1268 ext04/bit6	Tool length offset operation
0	Compensation operation by axis movement
1	Compensation operation by coordinate shift

When the power is turned on, the G49 modal is set. After the NC reset is performed, the operation follows the setting to retain the modal at resetting. This setting depends on the MTB specifications (parameters "#1210 RstGmd/bit18" and "#1210 RstGmd/bit7").

#1210 RstGmd/bit7	#1210 RstGmd/bit18	
	0	1
0	G49 modal (*1)	Retains the G43/ G44 modal.
1	Retains the G43/ G44 modal.	

(*1) If the NC is reset to G49 modal, no axis movement is performed.

Tool length offset by axis movement (#1268 ext04/bit6 = 0)

The movement amount is calculated with the following expressions when the G43 or G44 tool length offset command or G49 tool length offset cancel command is issued.

	Z axis movement amount	Operation
G43 Zz Hh1;	z + (lh1)	Offset in + direction by tool offset amount
G44 Zz Hh1;	z - (lh1)	Offset in - direction by tool offset amount
G49 Zz;	z - (+) (lh1)	Offset amount cancel

lh1: Offset amount for offset No. h1

Regardless of the absolute command or incremental command, the actual end point will be the position compensated for by the offset amount designated for the programmed movement command end point coordinate position.

The G49 (tool length offset cancel) mode is entered when the power is turned ON or when M02 has been executed.

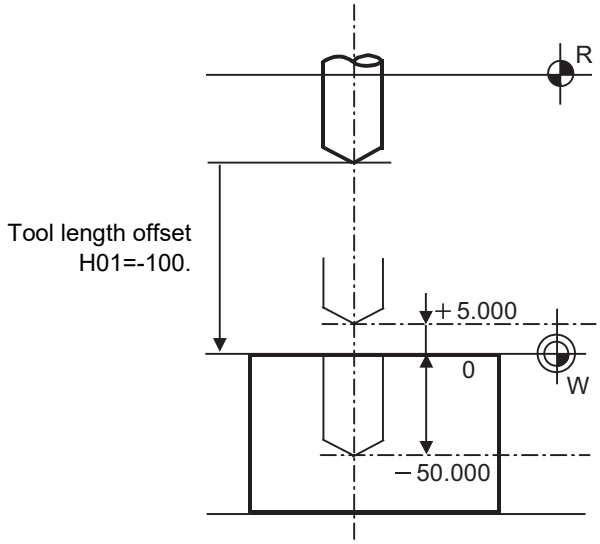
(Example 1) Absolute command (H01 = -100.)

```
N1 G91 G28 Z0 T01 M06 ;
N2 G90 G92 Z0 ;
N3 G43 Z5.000 H01 ;
N4 G01 Z-50.000 F500 ;
```

(Example 2) Incremental command (H01 = -100.)

```
N1 G91 G28 Z0 T01 M06 ;
N2 G91 G92 Z0 ;
N3 G43 Z5.000 H01 ;
N4 G01 Z-55.000 F500 ;
```

In block N3, the axis shifts only by the tool offset amount "-100.000" from the movement end point (position at +5.000), so the movement amount is set to "-95.000".



In the above (Example 1) and (Example 2), if no movement command is specified in the same block as G43 (when the movement command is divided into the next block), the axis moves by the tool length offset amount in block G43.

N3 G00 G43 H01 ;	Moves to the machining origin (movement by the tool offset amount).
G00 Z5.000 ;	Escape-moves by "+5.000".

Note

- If an axis command that contains the movement amount of "0" is specified in block G43, such as "G91 G43 Z0. H01;", the axis moves by the tool length offset amount regardless of the absolute command or the incremental command.

Tool length offset by coordinate shift (#1268 ext04/bit6 = 1)

(1) Coordinate shift amount by tool length offset

The coordinate shift is performed by the tool length offset amount when the G43/G44 command and the G49 command are specified. The axis movement amount is calculated in the coordinate system after the coordinate shift is complete.

As for the "G43 Zz1 Hh1;" command, the coordinate shift is first performed by "h1" and then the movement is performed by "z1" in the coordinate system.
For example, in the following program, the movement end points of blocks N10 and N11 are both "Z0.000", but the end point of block N11 is located in the coordinate system after the coordinate shift is complete, so the axis moves by the compensation amount.

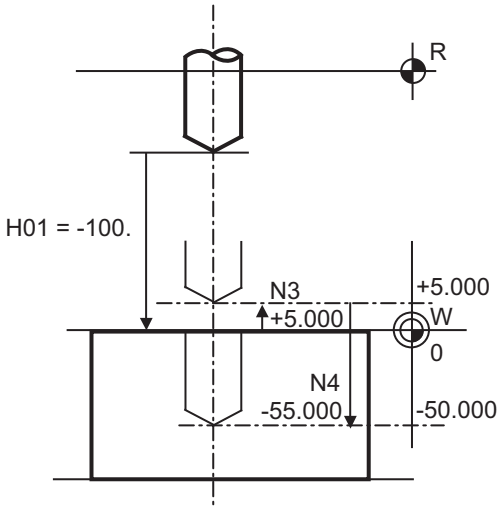
```
N10 G90 G00 Z0.000 ;
N11 G43 H01 Z0.000 ;
```

The end point of the movement command programmed with the absolute command is located at the same position as in the case of tool length offset by axis movement.
As for the incremental command, the increment is calculated in the coordinate system after the coordinate shift is completed, thereby, causing no movement by the tool length offset.
Therefore, the end point is different from the case of tool length offset by axis movement.

(Example 1) Absolute command (H01 = -100.000)

```
N1 G28 Z0 T01 M06 ;
N2 G90 G92 Z0 ;
N3 G00 G43 Z5.000 H01 ;
N4 G01 Z-50.000 F500 ;
```

In block N3, using the absolute command, the axis moves to the position "5.000" from the position "100.000" at which the coordinate shift is performed by the tool offset amount, so the movement amount is set to "-95.000".

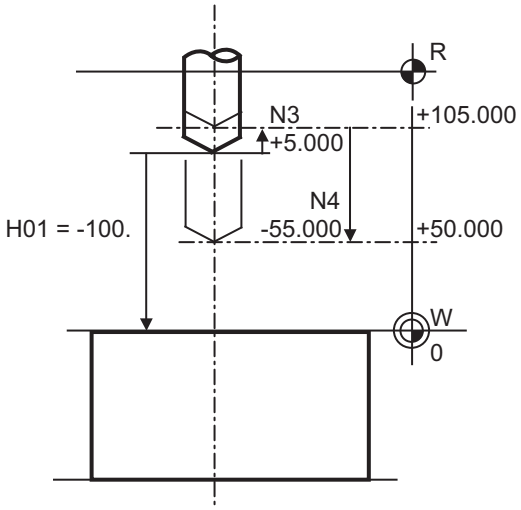


In the above (Example 1), the operation is the same even if no movement command is specified in the same block as G43 (*1).

(*1) For example, there is a case where the movement command is divided into the next block.

(Example 2) Incremental command (H01 = -100.000)

```
N1 G28 Z0 T01 M06 ;
N2 G91 G92 Z0 ;
N3 G00 G43 Z5.000 H01 ;
N4 G01 Z-55.000 F500 ;
```



In block N3, the axis moves from the coordinate system (100.000) after shifted by the tool offset amount, by the amount, "5.000", commanded by the incremental command, so the coordinate of the end point is set to "105.000". Therefore, if block N4 continues to be executed, the coordinate of the end point is set to "+50.000".

In this way, in the case of the coordinate shift type, it is not possible to perform positioning on the machining surface with only the incremental command, so be sure to issue the absolute command before the cutting block.

In the above (Example 2), the operation is the same even if no movement command is specified in the same block as G43 (*1).

(*1) For example, there is a case where the movement command is divided into the next block.

(2) Changing the tool length offset amount (Coordinate shift immediately after changing the compensation amount)

When the tool length offset amount is changed in the offset operation by coordinate shift (#1268/bit6 = 1) while G43 or G44 is being issued, "compensation operation by axis movement" and "compensation operation by coordinate shift" are provided as the compensation operation that is available when the incremental command is issued. The changed tool length offset amount is applied in the next compensation axis movement command block.

However, if the parameter "#1292 ext28/bit3" is "1", it is applied in the next block.

#1268 ext04/bit5	Tool length offset operation of changed amount
0	Compensation operation by axis movement (axis movement type)
1	Compensation operation by coordinate shift (coordinate shift type)

Machining program	#1268/bit5 = 0 Axis movement type		#1268/bit5 = 1 Coordinate shift type		Description
	Program position	Machine position	Program position	Machine position	
N10G10L10P1R-50. ;	0.000	0.000	0.000	0.000	Offset amount: "-50.000"
N20G91G0 ;	0.000	0.000	0.000	0.000	Incremental command
N30G43H1 ;	50.000	0.000	50.000	0.000	Tool length offset (coordinate shift)
N40Z-10. ;	40.000	-10.000	40.000	-10.000	Incremental command
N50G90G10L10P1R-200. ;	40.000	-10.000	40.000	-10.000	Change of offset amount: "-200.000"
N60G91G28Z0. ; (*1)	40.000	-160.000	190.000	-10.000	Coordinate shift or axis movement (Depends on "#1268/bit5".)
	0.000	0.000	0.000	0.000	Temporary cancel
N70G0Z-10. ;	190.000	-10.000	190.000	-10.000	Incremental command (coordinate shift)
N80G90Z0. ;	0.000	-200.000	0.000	-200.000	Absolute command
N90G49 ;	-200.000	-200.000	-200.000	-200.000	Tool length offset cancel

(*1) Indicates the intermediate point.

(3) Precautions for tool length offset by coordinate shift

(a) When "G43 Hh" or "G44 Hh" command is issued alone, no axis movement is performed, so immediately after being commanded, the tool nose position is not located in the position that takes the tool length offset into account.

- When issuing "G43 H" or "G44 H" command alone, be sure to issue the absolute command G00/G01 after that. If a command involving axis movement other than G00/G01 (*1) is issued before the absolute command, the path/end point may not be the one that was intended.

(b) When "G91 G43 Hh Zz" or "G91 G43 Hh1 Zz1" is commanded, the axis movement is performed, but the tool length offset amount is not applied because it is an incremental command.

- Even if an incremental command is executed at the same time, be sure to issue the absolute command G00/G01 after that. If a command involving axis movement other than G00/G01 (*1) is issued before the absolute command, the path/end point may not be the one that was intended.
- Do not command "G43 H" or "G44 H" at the same time as any movement command other than G00/G01. If they are commanded at the same time, an unintended path may result depending on the timing of the movement by the tool length offset amount.

(c) Even if the same program is used, the tool nose path is different between the tool length offset by axis movement and the tool length offset by coordinate shift.

- When changing the tool length offset method, be sure to check that there is no interference with the machine or workpiece.

(*1) This includes the fixed cycle program.

Offset number

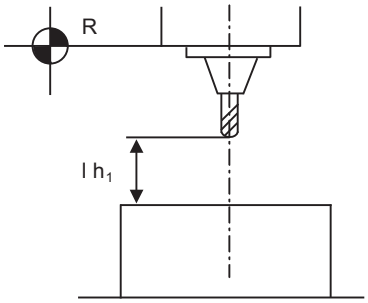
[Case to command address H in the same block as G43]

When address H is in the same block as G43 or G44, the offset number commanded with address H is set as the H modal.

(1) The offset amount differs according to the compensation type. The following example shows a case in which "G43 Hh1;" is commanded.

Type I

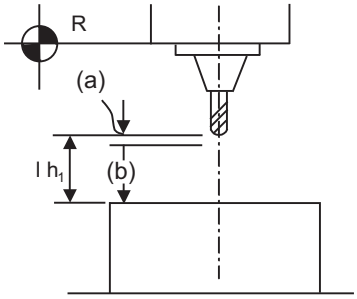
The compensation amount lh1 commanded with compensation No. h1 will be applied commonly regardless of the tool length offset amount, tool radius compensation amount, shape offset amount or wear compensation amount.



Type II

The compensation amount lh1 commanded with compensation No. h1 is as follows.

lh1: Shape offset (b) + wear compensation amount (a)



(2) The valid range of the offset No. will differ according to the specifications (No. of offset sets).

(3) If the commanded offset No. exceeds the specification range, the program error (P170) occurs.

(4) The offset No. commanded in the same block as G43 or G44 will be valid for the following modals.

(Example)

```
G43 Zz1 Hh1 ;        Tool length offset is executed with "h1".
  :
G49 Zz2 ;            Tool length offset is canceled.
  :
G43 Zz2 ;            Tool length offset is re-executed with "h1".
  :
```

(5) If G43 is commanded in the G43 modal, an offset of the difference between the offset No. data will be executed.

(Example)

```
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.

(6) If the H command is issued alone without a G43 command during the G43 modal, the offset does not change. However, only when "H0" is commanded, the H modal is cleared, so the tool length offset is canceled while the G43 modal is continued.

(7) When "H0" is specified, the tool length offset for all axes is canceled.

(8) The operation of the tool length offset depends on whether the axis movement or the coordinate shift is selected as the compensation operation.

[When the address H is commanded alone]

When the parameter "#1274 ext10/bit3" is "1", the H modal can be updated even if the address H of the tool compensation function is commanded alone.
Refer to "12.7.1 H Stand-alone Command" for details.

[Tool offset command check]

It is possible to make a program error occur when the tool offset No. is "0" or when the offset amount specified with the tool offset No. is "0", to prevent commands from being forgotten.
Refer to "12.7.2 Tool Offset Command Check" for details.

Axis valid for tool length offset

(1) When parameter "#1080 Dril_Z" is set to "1", the tool length offset is always applied to the Z axis.

(2) When the parameter "#1080 Dril_Z" is set to "0", the offset is applied to the axis that is commanded in the same block as G43.
When multiple axis commands are issued in the same block, the offset axis is determined in the order of the following parameters.

#1031 aux_K > #1030 aux_J > #1029 aux_I > #1028 base_K > #1027 base_J > #1026 base_I

(Example)

Parameters

#1026 = X
#1027 = Y
#1028 = Z
#1029 = U

```
G43 Xx1 Yy1 Zz1 Uu1 Hh1 ;  Compensation to U axis
  :
G49 ;
G43 Xx1 Yy1 Zz1 Hh2 ;      Compensation to 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 offset is commanded for the rotary axis, set the rotary axis name for one of the parallel axes.

(3) When there is no axis command in the same block as G43, the compensation is applied to the Z axis.

(Example)

```
G43 Hh1 ;                  Offset to Z axis
  :
G49 ;                      Tool length offset cancel
  :
```

(Supplement) The above operations (1) to (3) are valid for the effective axis of the tool length offset, regardless of the setting of parameter "#1268 ext04/bit6" (Tool length compensation behavior).

Movement during other commands in tool length offset modal

(1) If the reference position return is performed by G28 or manually, the program changes to G49 mode (tool compensation cancel mode) when the reference position return is completed.

When G30 or G30.1 is commanded, the program also changes to the tool length offset cancel mode (G49 modal).

However, it is possible to temporarily cancel the tool length offset when commanding G28, G30, or G30.1 using parameters, or when manually operating the reference position return, and then apply the tool length offset when commanding the movement of the axis to be compensated. If no movement command is issued for the axis to be compensated in the next block, the tool length offset is not applied in that block. The tool length offset is applied when a movement command is issued.

However, if parameter "#1292 ext28/bit3" is set to "1", the tool length offset is applied in the next block. (This setting depends on the MTB specifications.)

When it is temporarily canceled, the modal is retained, thereby, causing G49 not to be commanded.

When the dog-type reference position return is performed, the tool length offset command is canceled.

[Operation at automatic reference position return]

	#1268 ext04/bit7	Automatic reference position return (G28)
When the axis reached the reference position:	0	Cancel
	1	Temporary cancel
When a movement command is issued for the axis to be compensated after the above	0	Cancel
	1	Applies the tool length offset before the axis reaches the reference position.

(Example 1) Case of dog-type and high-speed type (parameter #1268 ext04/bit7 = 0)

```
G43 Zz1 Hh1 ;
  :
G28 Zz2 ;          Canceled when reference position is reached. (Same as G49)
  :
G43 Zz2Hh2 ;
  :
G49 G28Zz2 ;       The Z axis is canceled, and then the reference position return operation is performed.
```

(Example 2) Case of high-speed type (parameter #1268 ext04/bit7 = 1)

```
G43 Zz1 Hh1 ;
  :
G28 Zz2 ;          When the axis reaches the reference position, the operation is temporarily canceled.
G01 Zz2 Ff2 ;      The end point is set to the coordinate position that is offset in the positive direction by the offset amount of offset number "h1" from the end point "z2".
```

[Operation at manual reference position return]

	#8122	Manual reference position return
When the axis reached the reference position:	0	Cancel
	1	Temporary cancel
When a movement command is issued for the axis to be compensated after the above	0	Cancel
	1	Applies the tool length offset before the axis reaches the reference position.

(Example 3) Case of dog-type and high-speed type (parameter #8122 = 0)

G43 Zz1 Hh1 ;

: If an interruption occurs in the manual dog-type reference position return, the operation is canceled when the axis reaches the reference position.

G01 Zz2 Ff2 ; Performs the same operation as that in G49 mode.

(Example 4) Case of high-speed type (parameter #8122 = 1)

G43 Zz1 Hh1 ;

: If an interruption occurs during the manual high-speed reference position return, the operation is temporarily canceled when the axis reaches the reference position.

G01 Zz2 Ff2 ; The end point is set to the coordinate position that is offset in the positive direction by the offset amount of offset number "h1" from the end point "z2".

(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.
However, if the parameter "#1292 ext28/bit3" is "1", the axis returns to the coordinates that are re-shifted by the tool length offset amount in the next block.

(3) When an incremental command is issued after a G53 movement command, or when an incremental command is issued after any of G28/G30/G30.1 manual reference position return operations was executed while the tool length offset is retained (#1268/bit7 = 1), the operation when an incremental command is issued is as shown in the table below depending on how the compensation amount is handled.

(a) When the parameter "#1268 ext04/bit6" is "0", the axis moves by the incremental command value + tool length offset amount.

[Example of G53 command (Tool length offset: -50.000)]

Machining program	Program position	Machine position
G90 G01 Z-50. F1000 ;	-50.000	-50.000
G43 H1 ;	-50.000	-100.000
G53 G90 G00 Z0 ;	0.000	0.000
G91 G01 Z-100. F1000 ;	-100.000	-150.000

[Example of G28 command (Tool length offset: -50.000)]

Machining program	Program position	Machine position
G90 G01 Z-50. F1000 ;	-50.000	-50.000
G43 H1 ;	-50.000	-100.000
G28 G91 G00 Z0 ;	0.000	0.000
G91 G01 Z-100. F1000 ;	-100.000	-150.000

(b) When the parameter "#1268 ext04/bit6" is "1", the axis moves only by the incremental command value.

[Example of G53 command (Tool length offset: -50.000)]

Machining program	Program position	Machine position
G90 G01 Z-50. F1000 ;	-50.000	-50.000
G43 H1 ;	0.000	-50.000
G53 G90 G00 Z0 ;	0.000	0.000
G91 G01 Z-100. F1000 ;	-50.000	-100.000

[Example of G28 command (Tool length offset: -50.000)]

Machining program	Program position	Machine position
G90 G01 Z-50. F1000 ;	-50.000	-50.000
G43 H1 ;	0.000	-50.000
G28 G91 G00 Z0 ;	0.000	0.000
G91 G01 Z-100. F1000 ;	-50.000	-100.000

Note

♦If the high-speed high-accuracy control II command (G05 P10000) or the cylindrical interpolation command (G07.1) is executed before a movement command, the program error (P29) occurs.

[When "#1268/bit7" is "1" and #8122 is "0"]
If the manual reference position return operation is performed after the axis returned to the reference position with the automatic reference position return command (G28), the tool length offset is retained.

[When "#1268/bit7" is "0" and #8122 is "1"]
If the automatic reference position return command (G28) is performed after the manual reference position return, the following operation is performed.

- When "#1091 Mpoint" is "1" (without movement at the intermediate point), the tool length offset is retained.
- When "#1091 Mpoint" is "0" (with movement at the intermediate point), the axis moves to the position where the tool length offset is applied up to the intermediate point, but the tool length offset is canceled after the axis reached the reference position.

Switching the tool length offset compensation start operation

If the tool length offset is temporarily canceled during tool length offset (G43/G44) with the following operations, the compensation restarts from the time the movement command for the axis to be compensated is issued.

♦When the parameter "#1268 ext04/bit7" is "1", the tool length offset has been temporarily canceled by commanding G28/G30/G30.n, etc.
♦The tool length offset has been temporarily canceled by commanding G53.

If the tool length offset amount is changed during tool length offset (G43/G44), in the same way, the changed tool length offset amount is applied from the time the movement command for the axis to be compensated is issued. However, if the parameter "#1292 ext28/bit3" is "1", compensation starts from the next block. (This setting depends on the MTB specifications.)

#1292 ext28/bit3	Start of offset
0	Starts offset from the time the movement command for the axis to be compensated is issued
1	Starts offset from the next block

This parameter is valid for both offset operations: "tool length offset by axis movement" (#1268 ext04/bit6 = 0) and "tool length offset by coordinate shift" (#1268 ext04/bit6 = 1).

The operations that start compensation from the next block are as follows.

In the following example, compensation starts from the block after G53. (#1292 ext28/bit3 = 1)

Machining program	#1268/bit6 = 0 Axis movement type		#1268/bit6 = 1 Coordinate shift type	
	Z axis			
	Program position	Machine position	Program position	Machine position
N10 G90G10L10P1R-50. ;	-8.000	-8.000	-8.000	-8.000
N20 G00G28Z0.X0.Y0. ;	0.000	0.000	0.000	0.000
N30 G91G01F1000 ;	0.000	0.000	0.000	0.000
N40 G43Z-170.H1 ;	-170.000	-220.000	-120.000	-170.000
N50 G90G53Z-10. ;	-10.000	-10.000	-10.000	-10.000
N60 ;	-10.000	-60.000	40.000	-10.000

(1) If the next block is a macro statement, when the parameter "#8101 MACRO SINGLE" is "0", compensation does not start in the macro statement, but when it is "1", compensation starts.

(2) Even in the case of the axis movement type, if the G code in the following table is used, compensation starts by the coordinate shift operation.

G code	Function	G code	Function
G04	Dwell	G116	Start point timing synchronization
G05.1	Spline interpolation / high-speed high-accuracy control I	G120.1	Machining condition selection I
G07	Hypothetical axis interpolation	G121	Machining condition selection I cancel
G08	High-accuracy control	G122	Activate sub part system I
G11	Programmable data input cancel	G130	Selection of axis (axes) for feedrate command
G12	Circular cutting CW	G140	Arbitrary axis exchange command
G13	Circular cutting CCW	G141	Arbitrary axis exchange return command
G34	Special fixed cycle (bolt hole circle)	G142	Arbitrary axis exchange/reference axis arrange return command
G35	Special fixed cycle (line at angle)	G145	Cancel sub part systems
G36	Special fixed cycle (arc)	G160	Torque limitation skip
G37.1	Special fixed cycle (grid)	G162	Spindle speed fluctuation detection
G52	Local coordinate system setting	G162.1	Cutting load control
G53	Basic machine coordinate system selection (*1)	G163	Spindle speed fluctuation detection cancel
G113.1	Spindle synchronization I cancel	G180	Interactive cycle insertion
G114.1	Spindle synchronization I	G186	Interference check III setting data valid
G115	Start point timing synchronization		

(*1) When the parameter "#1293 ext29/bit7" is "1", compensation starts. When "0" is set, compensation does not start.

(3) If the G code in the following table is commanded in the next block, compensation starts from the next block.

G code	Function	G code	Function
G05P0	High-speed high-accuracy control II/III OFF	G92.1	Workpiece coordinate system preset
G10	Compensation data input by program	G92 G53	Coordinate system setting (*1)

(*1) Available when G92 and G53 are commanded in the same block (G92 G53 X0 Y0 Z0;).

In the following example, compensation does not start in the G92.1 block, but from the next block. (#1292 ext28/bit3 = 1)

Machining program	#1268/bit6 = 0 Axis movement type		#1268/bit6 = 1 Coordinate shift type		Description
	Z axis				
	Program position	Machine position	Program position	Machine position	
N10 G90G10L10P1R-50. ;	-8.000	-8.000	-8.000	-8.000	
N20 G00G28Z0.X0.Y0. ;	0.000	0.000	0.000	0.000	
N30 G91G01F1000 ;	0.000	0.000	0.000	0.000	
N40 G43Z-170.H1 ;	-170.000	-220.000	-120.000	-170.000	
N50 G91G28Z0. ;	0.000	0.000	0.000	0.000	
N60 G90G92.1Z0. ;	0.000	0.000	0.000	0.000	Compensation does not start in the G92.1 block.
N70 ;	0.000	-50.000	50.000	0.000	Compensation starts in the next block.

(4) When the parameter "#1268 ext04/bit6" is set to "1" (coordinate shift type) and the coordinate system setting (G92) is commanded in the following cases, the program error (P294) occurs.

- The coordinate system setting (G92) has been commanded when the tool length offset amount was temporarily canceled by a command such as G53, G28, G30, or G30.1.
  However, if G92 and G53 are commanded in the same block (G92 G53 X0 Y0 Z0;), no error occurs.
- The tool length offset amount was changed during tool length offset compensation (G43/G44), and then coordinate system setting (G92) was commanded without the compensation amount being applied.

In the following example, the program error occurs in the G92 block. (#1292 ext28/bit3 = 1)

Machining program	#1268/bit6 = 0 Axis movement type		#1268/bit6 = 1 Coordinate shift type		Description
	Z axis				
	Program position	Machine position	Program position	Machine position	
N10 G90G10L10P1R-50. ;	-8.000	-8.000	-8.000	-8.000	
N20 G00G28Z0.X0.Y0. ;	0.000	0.000	0.000	0.000	
N30 G91G01F1000 ;	0.000	0.000	0.000	0.000	
N40 G43Z-170.H1 ;	-170.000	-220.000	-120.000	-170.000	
N50 G91G28Z0. ;	0.000	0.000	0.000	0.000	
N60 G90G92Z0. ;	0.000	0.000	-	-	For the coordinate shift type, the program error (P294) occurs.
N70 ;	0.000	-50.000	-	-	For the axis movement type, compensation starts.

(5) If the next block is a G53 block, whether or not compensation is started depends on the MTB specifications (parameter "#1293 ext29/bit7").

#1293 ext29/bit7	Operation
0	Compensation does not start.
1	Compensation starts.

When "#1293 ext29/bit7" is "1", the operation differs depending on whether an absolute command or an incremental command is issued in the G53 block.

(Example 1) Case where no axis command is specified for the tool length offset axis with a G53 absolute command (G90G53)

Machining program	#1268/bit6 = 0 Axis movement type		#1268/bit6 = 1 Coordinate shift type		Description
	Z axis				
	Program position	Machine position	Program position	Machine position	
N10 G90G10L10P1R-50. ;	-	-	-	-	
N20 G00G28Z0.X0.Y0. ;	0.000	0.000	0.000	0.000	
N30 G91G01F1000 ;	0.000	0.000	0.000	0.000	
N40 G43Z-170.H1 ;	-170.000	-220.000	-120.000	-170.000	
N50 G90G53Z-10. ;	-10.000	-10.000	-10.000	-10.000	
N60 G53 ;	40.000	-10.000	40.000	-10.000	The program position shifts by the tool length offset without movement of the axis.

(Example 2) Case where an axis command is specified for the tool length offset axis with a G53 absolute command (G90G53)

Machining program	#1268/bit6 = 0 Axis movement type		#1268/bit6 = 1 Coordinate shift type		Description
	Z axis				
	Program position	Machine position	Program position	Machine position	
N10 G90G10L10P1R-50. ;	-	-	-	-	
N20 G00G28Z0.X0.Y0. ;	0.000	0.000	0.000	0.000	
N30 G91G01F1000 ;	0.000	0.000	0.000	0.000	
N40 G43Z-170.H1 ;	-170.000	-220.000	-120.000	-170.000	
N50 G90G53Z-10. ;	-10.000	-10.000	-10.000	-10.000	
N60 G53Z-20. ;	-20.000	-20.000	-20.000	-20.000	The axis movement is performed while the tool length offset has been temporarily canceled.

(Example 3) Case where no axis command is specified for the tool length offset axis with a G53 incremental command (G91G53)

Machining program	#1268/bit6 = 0 Axis movement type		#1268/bit6 = 1 Coordinate shift type		Description
	Z axis				
	Program position	Machine position	Program position	Machine position	
N10 G90G10L10P1R-50. ;	-	-	-	-	
N20 G00G28Z0.X0.Y0. ;	0.000	0.000	0.000	0.000	
N30 G91G01F1000 ;	0.000	0.000	0.000	0.000	
N40 G43Z-170.H1 ;	-170.000	-220.000	-120.000	-170.000	
N50 G90G53Z-10. ;	-10.000	-10.000	-10.000	-10.000	
N60 G91G53 ;	40.000	-10.000	40.000	-10.000	The program position shifts by the tool length offset without movement of the axis.

(Example 4) Case where an axis command is specified for the tool length offset axis with a G53 incremental command (G91G53)

Machining program	#1268/bit6 = 0 Axis movement type		#1268/bit6 = 1 Coordinate shift type		Description
	Z axis				
	Program position	Machine position	Program position	Machine position	
N10 G90G10L10P1R-50. ;	-	-	-	-	Sets the tool length offset amount to "-50".
N20 G00G28Z0.X0.Y0. ;	0.000	0.000	0.000	0.000	
N30 G91G01F1000 ;	0.000	0.000	0.000	0.000	
N40 G43Z-170.H1 ;	-170.000	-220.000	-120.000	-170.000	
N50 G90G53Z-10. ;	-10.000	-10.000	-10.000	-10.000	
N60 G91G53Z-20. ;	20.000	-30.000	20.000	-30.000	The axis moves by the distance specified with the incremental command in the G53 block, and the program position shifts by the tool offset amount. (20.0 = -10. + 50. -20.)

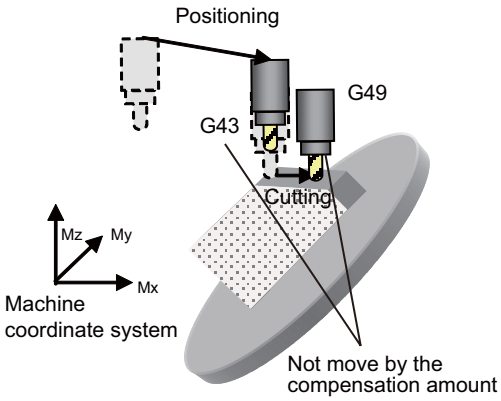
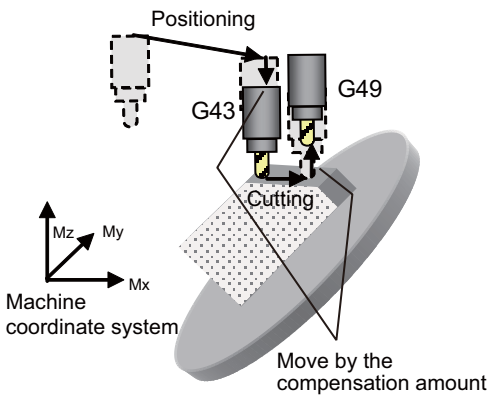
(6) If the following parameters are all set to "1", an MCP alarm (Y51 108) occurs, and the automatic operation cannot be started. (These settings depend on the MTB specifications.)

- "#1247 set19/bit0" (Movement by tool length compensation command)
- "#1292 ext28/bit3" (Operation at changing tool length compensation amount)

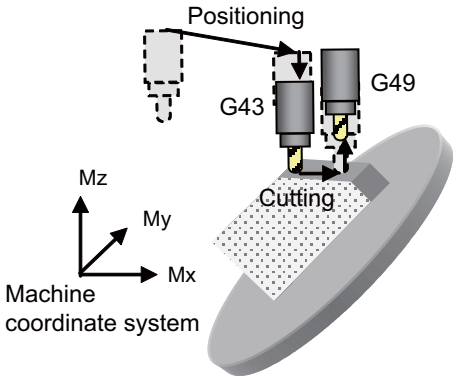
Movement by tool length compensation command

When there is no movement command in the same block where the tool length offset start (G43 or G44) or the tool length offset cancel (G49) is commanded (G43, G44 or G49 is commanded alone), the movement by the offset amount may not be carried out depending on the MTB specifications (parameter "#1247 set19/bit0" (movement by tool length compensation command)).

[Tool offset commanded alone]

Machining program	Not move when commanded alone (#1247/bit0 = 1)	Move when commanded alone (#1247/bit0 = 0)
: G54 A0. C0. ; G68.2 ; G53.1 ; G00XxYyZz ; G43H1 ; : G49 ;	If tool position offset is commanded alone in a block, the axis does not move, but the tool offset amount is applied to the program position counter. 	If the tool length offset is commanded alone in a block, the axis moves by the tool length offset amount. 

[Tool offset commanded with movement]

Machining program	
: G54 A0. C0. ; G68.2 ; G53.1 ; G00XxYyZz ; G43H1Z0 ; : G49Z10. ;	If the tool length offset command and axis movement command are issued in the same block, the axis moves to the end point that is obtained by adding the tool offset to the movement command. 

Note

(1) When the parameter "#1268 ext04/bit6" is "1" (compensation operation by coordinate shift), the compensation operation is performed by the coordinate shift regardless of the setting of the parameter "#1247 set19/bit0". When using the movement by tool length compensation command, set the parameter "#1268 ext04/bit6" to "0" (compensation operation by axis movement).

(2) If the parameter "#1247 set19/bit0" is "1" and any of the following parameters is set to "1", the MCP alarm (Y51 108) occurs, thereby, disabling the automatic operation from being started. (These settings depend on the MTB specifications.)

- "#1274 ext10/bit3" (Tool offset number H stand-alone command (for M system only))
- "#1292 ext28/bit3" (Operation at changing tool length compensation amount)

(3) For the G49 command, command it at a safe position where the tool does not interfere with the machine in consideration of the compensation canceling operation.
When the parameter "#1247 set19/bit0" is set to "0", the axis moves to the position where the compensation was canceled even though there is no axis command in the G49 command block.



Relationship with other functions

This section describes the operations when a G code other than the tool compensation is commanded during the tool compensation.
In particular, when the tool length compensation G code and another G code are commanded in the same block, the tool length compensation G code may not be executed or the program error may occur depending on the G code priority order.

(1) When the tool length compensation command (G43/G44/G49) and the G code in the table below are commanded in the same block, the program error (P45) occurs.

G code	Function
G04	Dwell
G10	Data input by program
G11	Data input by program cancel
G27	Reference position check
G28	Reference position return
G29	Start position return
G30	2nd to 4th reference position return
G37	Automatic tool length measurement
G52	Local coordinate system setting
G53	Machine coordinate system selection

(2) If a tool length compensation command (G43/G44/G49) and a circular command (G02/G03) are specified in the same block, the program error (P70) occurs.

(3) When compensation by the tool length compensation command (G43/G44) is applied to the height axis of the arc, the compensation movement is superimposed with circular movement if the axis moves by the specified tool length compensation amount in the circular command block.

(4) If the parameter "#1268 ext04/bit6" is "1" (compensation operation by coordinate shift), the program error (P29) occurs if the tool length compensation cancel command (G49) is issued in the same block as the fixed cycle for drilling or during the fixed modal for drilling.

(5) If a tool compensation amount rewriting command is issued by G10 compensation input during tool length compensation, the compensation amount is applied from the next compensation axis movement block.
However, if the parameter "#1292 ext28/bit3" is "1", it is applied from the next block.

(6) When commanding the workpiece coordinate system preset (G92.1), cancel the tool length compensation, nose R compensation, and tool diameter compensation. If such a command is executed without canceling, the workpiece coordinate is set to the value obtained by subtracting the workpiece coordinate offset amount from the machine position, so the compensation vector will be temporarily canceled.
If temporarily canceled, compensation restarts from the time the movement command is issued for the axis to be compensated, regardless of the setting of the parameter "#1292 ext28/bit3".

(Example) The tool offset amount is temporarily canceled in the G92.1 block, and compensation restarts when the movement command is issued for the axis to be compensated.

The following example shows a tool length offset for the axis movement type. (#1268 ext04/bit6 = 0)

Machining program	#1292/bit3 = 0 Compensation starts when a movement command is issued.		#1292/bit3 = 1 Compensation starts from the next block.		Remarks
	Z axis				
	Program position	Machine position	Program position	Machine position	
N10 G90G10L10P1R-50. ;	0.000	0.000	0.000	0.000	Offset amount: "-50.000"
N20 G00G28Z0.X0.Y0. ;	0.000	0.000	0.000	0.000	
N30 G91G01F1000 ;	0.000	0.000	0.000	0.000	
N40 G43Z-170.H1 ;	-170.000	-220.000	-170.000	-220.000	
N50 G90G00Z-50. ;	-50.000	-100.000	-50.000	-100.000	
N60 G92.1Z0. ;	-100.000	-100.000	-100.000	-100.000	Tool offset amount: Temporarily canceled
N70 ;	-100.000	-100.000	-100.000	-100.000	Compensation does not restart.
N80 G90G00Z0. ;	0.000	-50.000	0.000	-50.000	Compensation restarts with the Z axis movement command.

Multiple-axis synchronization control

Tool length offset for multiple-axis synchronization control is enabled depending on the MTB specifications. (Parameters "#1255 set27/bit5", "#2675 tcmp_top", "#1247 set19/bit0", "#1712 cfg12/bit2" and "#1712 cfg12/bit3")

(1) Tool length offset method

[#1255 set27/bit5 = 1]

Tool length offset for multiple-axis synchronization control is enabled and different amount of tool length offset 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 offset can be applied to each head and the tool length error between heads can be compensated.

[#1255 set27/bit5 = 0]

Tool length offset for multiple-axis synchronization control is disabled and the same tool length offset amount as the master axis is applied to the slave axis.

(2) Operation of tool length offset for multiple-axis synchronization control

With tool length offset for multiple-axis synchronization control, the tool compensation amount of different tool compensation No. can be applied to each axis. Tool compensation amount of the tool compensation No. that the value of the parameter "2675 tcmp_top" is added to commanded No. is applied.

Tool length offset for multiple-axis synchronization control is applicable only to axes which are actually in the synchronized state. Therefore, the axis where the bit of "Synchronization control operation method" signal (R2589) is turned OFF is not the target for tool length offset for multiple-axis synchronization control.

Tool length offset for multiple-axis synchronization control is started with G43, G44 command in the same way as the normal tool length offset. To cancel the tool length offset, command G49 in the same way as the normal tool length offset.

With tool length offset for multiple-axis synchronization control, the movement is made by the tool length offset amount in G43, G44 or G49 commanded block.
However, when the parameter "#1247 set19/bit0" (Movement by tool length compensation command) is "1", the axis moves by the compensation amount difference between the target axes immediately before the block where the target axis for compensation moves following G43, G44 or G49 commanded block. In this case, the master axis and the slave axis move according to each axis compensation amount, and the synchronization is temporarily disabled. Issue the movement command to the target axis for compensation in the positioning (G00) because machining is performed when the synchronization is enabled. When it is issued in other than G00, the program error (P46) occurs, causing the operation to be stopped.

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". For other axes, the setting value of the parameter #2675 is the tool compensation No., and the tool compensation amount is applied.

The movement command can be issued in the same block as the G43, G44 or G49 command. When the movement command is issued for the master axis in the same block as the G43, G44 or G49 command, compensation amount which is applied to the movement command of the master axis is applied to the target axes for multiple-axis synchronization control. The G43, G44 or G49 command block functions as G00 non-interpolation.

(3) Slave axis compensation method

The slave axis behavior is determined as follows depending on combination of the parameters "#1255 set27 bit5", "#1712 cfg12/bit2" and "#1712 cfg12/bit3".

[#1255 set27/bit5 = 1]

Compensation is performed with the tool length offset amount of the salve axis.

#1712 cfg12		Operation of tool length offset for multiple-axis synchronization control
bit2	bit3	
0	0	Compensation is performed from the machine position of the slave axis. The workpiece offset of the master axis is applied to the compensation.
1	0	For slave axis, compensation is performed from the machine position of the master axis.
0	1	The workpiece offset of the slave axis is applied to the compensation.
1	1	For slave axis, compensation is performed from the machine position of the master axis. The workpiece offset of the slave axis is applied to the compensation.

When the tool length offset is canceled, the compensation amount set according to the above table is also canceled.

[#1255 set27/bit5 = 0]

Compensation is performed with the tool length offset amount of the master axis.

(4) Function related to tool length offset

When "#1255 set27/bit5" is "0", each tool length offset function is available; however, the tool length offset for multiple-axis synchronization control is disabled. (The validity of each function depends on the MTB specifications.)

When "#1255 set27/bit5" is "1", the following tool length offset functions can be used during multiple-axis synchronization control.

Retain G43 at manual reference position return	Disabled (*1)
Movement by tool length compensation command	Enabled
Tool length offset (coordinate shift)	Disabled (*2)
Retain tool length offset at automatic reference position return	Disabled (*1)
H stand-alone command	Disabled (*3)
Tool length offset command check	Enabled
Switching the tool length offset compensation start operation	Disabled (*4)

(*1) For tool length offset for multiple-axis synchronization control, the settings of the parameters "#8122 Keep G43 MDL M-REF" and "#1268 ext04/bit7" are ignored. When the manual or automatic reference position return is performed during offset, the tool length offset is canceled. To restart the tool length offset, the tool length offset command (G43, G44) is required again.

(*2) If "#1268 ext04/bit6" is set to "1", the MCP alarm (Y51 108) occurs, thereby, disabling the automatic operation from being started.

(*3) If "#1274 ext10/bit3" is set to "1", the MCP alarm (Y51 108) occurs, thereby, disabling the automatic operation from being started.

(*4) If "#1292 ext28/bit3" is set to "1", the MCP alarm (Y51 108) occurs, thereby, disabling the automatic operation from being started.

(5) Precautions

(a) The maximum number of control axes includes the number of slave axes. The number of axes simultaneously commanded does not include the number of slave axes. However, in the G43 command block, the number of axes simultaneously commanded includes the number of slave axes. If the total number of axes commanded is equal to or larger than the number of axes for specifications, the program error (P10) occurs. In this case, correct the machining program, such as divide the axis command into two or more blocks.

(b) When the parameter "#1255 set27/bit5" is "0", align the tool ends of master axis and slave axis before starting the multiple-axis synchronization control.

(c) When the parameter "#1255 set27/bit5" is "1" and "#1712 cfg12/bit2" is "0", align the machine positions of master axis and slave axis before starting the multiple-axis synchronization control. Unless the machine positions are aligned, the tool end positions of master axis and slave axis may be deviated, causing a collision with the workpiece.

(d) When the tool life management II is enabled and the parameter "#1227 aux11/bit1" is "1", the tool compensation amount of the compensation No. 99 is applied by commanding "G43H99;" (or "G44H99;"). For tool length offset for multiple-axis synchronization control, the setting of the parameter "#1227 aux11/bit1" is disabled.

(e) For tool length offset for multiple-axis synchronization control, commands for non-target axes of multiple-axis synchronization control in the G43 (or G44) command block operates as G00 non-interpolation.

(f) For tool length offset for multiple-axis synchronization control, a movement command of the master axis is issued in the same block as G43, G44 or G49 command, it operates as G00 non-interpolation. If a G code movement command other than G00 is issued in the same block as G43, G44 or G49 command, the program error (P45) occurs. The program error (P45) always occurs regardless of the setting of the parameter "#1241 set13/bit0".

(g) Even when the restart search is performed for the block during tool length offset for multiple-axis synchronization control, the position cannot be moved to where the position after tool length offset for multiple-axis synchronization control is applied. To perform the restart search for a machining program including tool length offset for multiple-axis synchronization control, perform the restart search for a block where tool length offset for multiple-axis synchronization control is disabled.

(h) When the tool length offset is canceled by reset, the slave axis does not move by the canceled compensation amount. When a reset or emergency stop is performed, cancel the multiple-axis synchronization control, then command it again.

(i) A position error of each axis occurs after the emergency stop. Cancel the multiple-axis synchronization control, then command it again after position alignment.

(j) When a reset is performed during movement by the tool compensation amount of tool length offset for multiple-axis synchronization control, the positional relationship between the master axis and the slave axis is deviated. In this case, cancel the multiple-axis synchronization control, then command it again.

(k) When movement is commanded on the G53 machine coordinate system, the master axis moves to the machine position with the tool offset amount canceled; however, the slave axis moves without the tool offset amount canceled and with the synchronization state.

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)

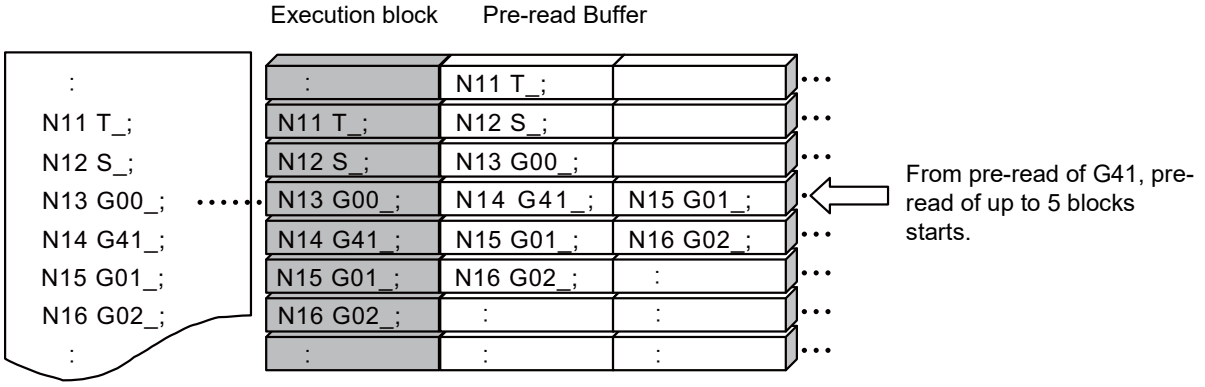
Tool radius compensation starts when all the following conditions are met in the compensation cancel mode.

(1) The movement command is issued after G41 or G42.
(2) The tool radius compensation offset No. is 0 < D ≤ max. offset No.
(3) The movement command of positioning (G00) or linear interpolation (G01) is issued.

At start of the compensation, regardless of continuous operation or single block operation, three blocks of movement commands are always pre-read consecutively before execution. If there are fewer than three blocks of movement commands, up to five blocks are pre-read.
Similarly, during the compensation mode, up to five blocks are read for compensation calculation.

The maximum number of blocks pre-read at start of the compensation and during the compensation mode depends on the MTB specifications (parameter "#1315 RcmpBlk").

[Control state diagram]

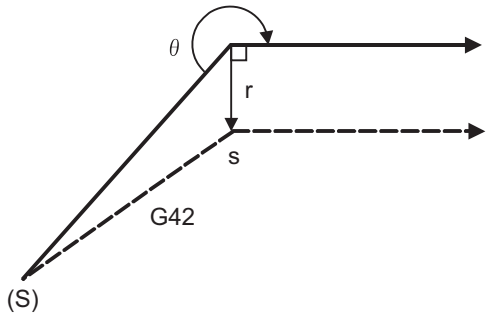
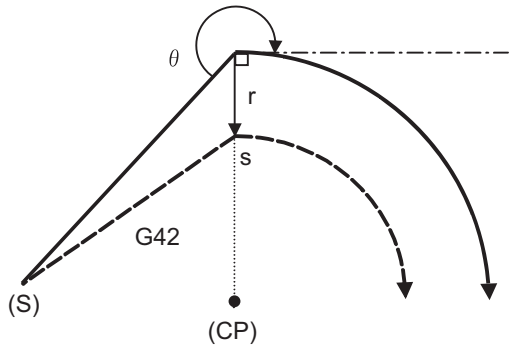


There are two ways of starting the compensation operation: type A and type B.
The type depends on the setting of the parameter "#1229 set01/bit2".
This type is used in common with the compensation cancel type.

Type	Description
Type A	When starting up/canceling a command block with nose R compensation and tool radius compensation, type A will not conduct intersection operation processing to the block and, instead, convert it to an offset vector which is vertical to the command vector.
Type B	When starting up/canceling a command block with nose R compensation and tool radius compensation, Type B will conduct intersection operation processing to the command block and the next block.

Start operation for tool radius compensation

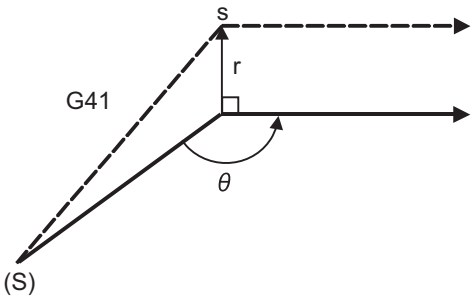
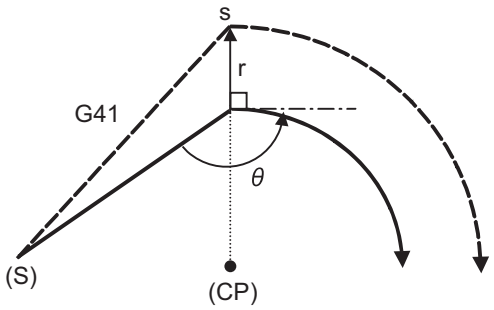
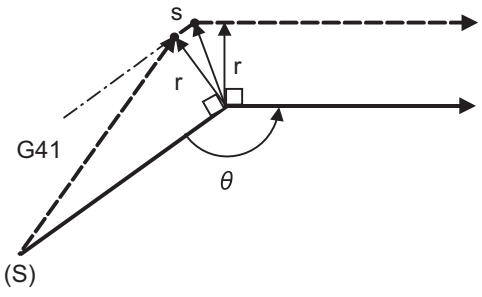
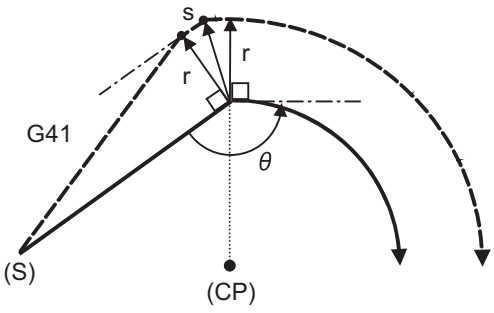
(1) Machining an inside corner

Linear -> Linear	Linear -> Circular
	

(S) Start point
r : Compensation amount
—— Program path

(CP) Center of arc
s: Stop point with single block
- - - - Tool center path

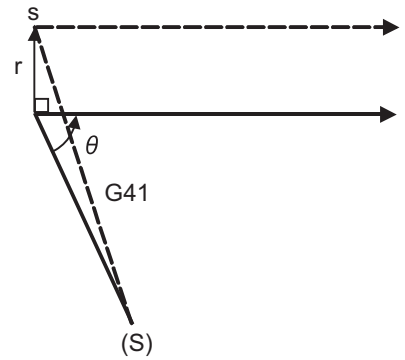
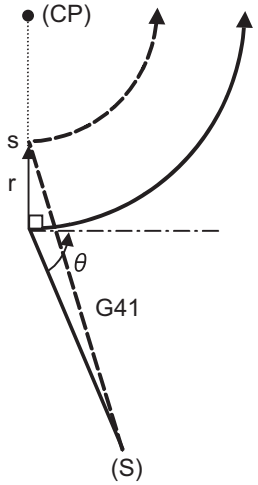
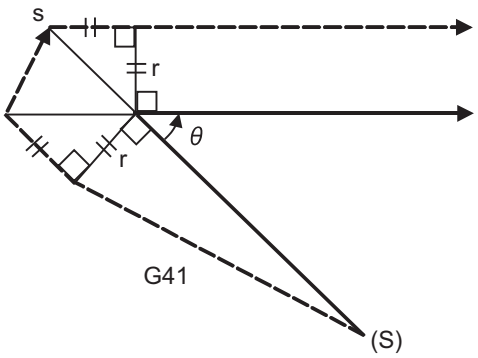
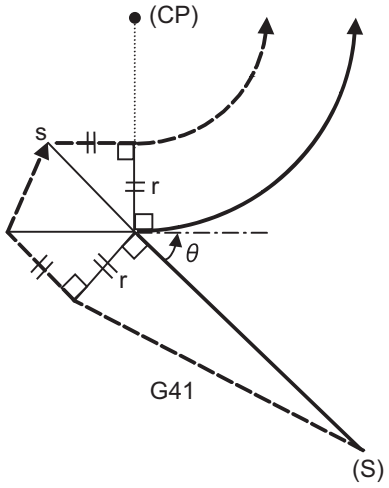
(2) Machining an outside corner (obtuse angle) [90° ≤ θ < 180°]

Linear -> Linear (Type A)	Linear -> Circular (Type A)
	
Linear -> Linear (Type B)	**Linear -> Circular (Type B)**
	

(S) Start point
r : Compensation amount
—— Program path

(CP) Center of arc
s: Stop point with single block
- - - - Tool center path

(3) Machining an outside corner (acute angle) [θ < 90°]

Linear -> Linear (Type A)	Linear -> Circular (Type A)
	
Linear -> Linear (Type B)	**Linear -> Circular (Type B)**
	

(S) Start point
(CP) Center of arc
r : Compensation amount
s: Stop point with single block
—— Program path
- - - - Tool center path

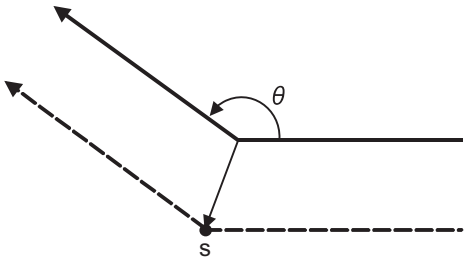
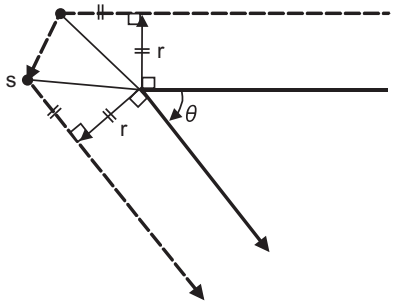
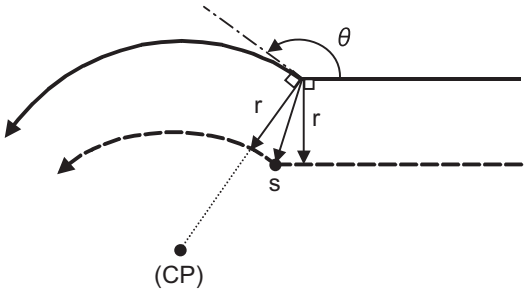
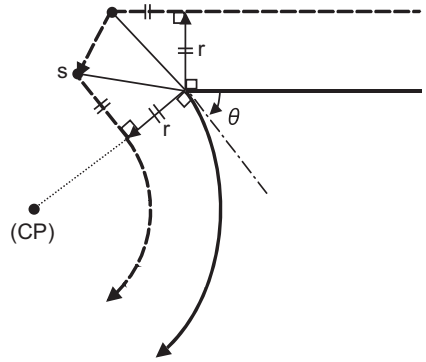
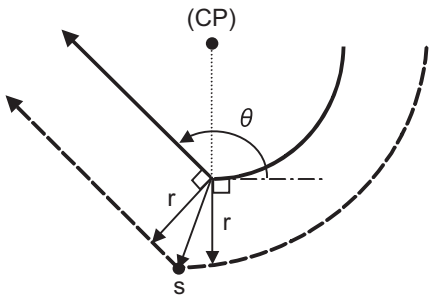
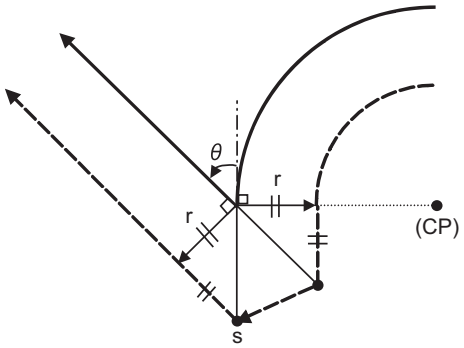
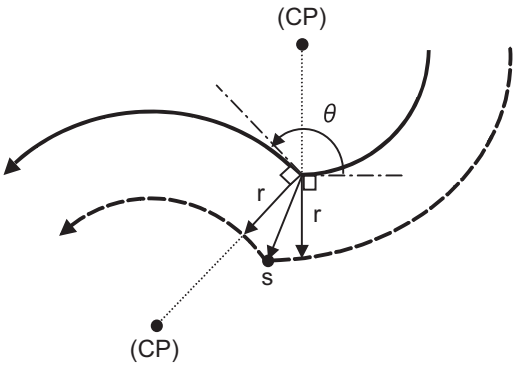
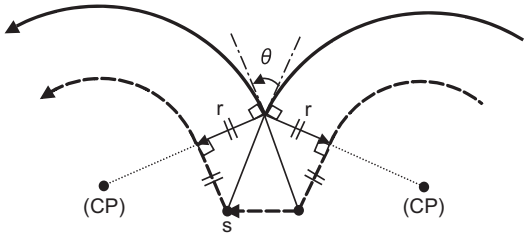
Note

•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.

(1) Machining an outside corner

Linear -> Linear (90° ≤ θ < 180°)	Linear -> Linear (0° < θ < 90°)
	
Linear -> Circular (90° ≤ θ < 180°)	**Linear -> Circular (0° < θ < 90°)**
	
Circular -> Linear (90° ≤ θ < 180°)	**Circular -> Linear (0° < θ < 90°)**
	
Circular -> Circular (90° ≤ θ < 180°)	**Circular -> Circular (0° < θ < 90°)**
	

(CP) Center of arc

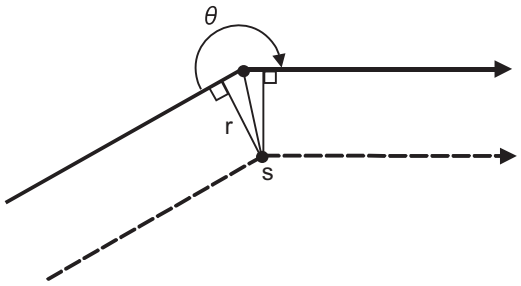
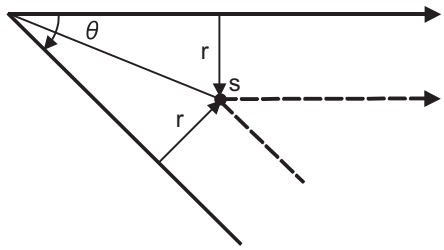
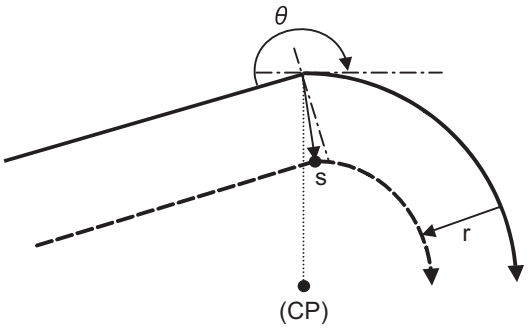
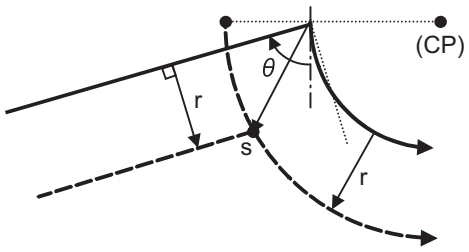
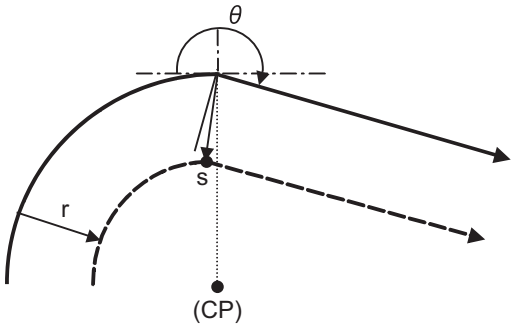
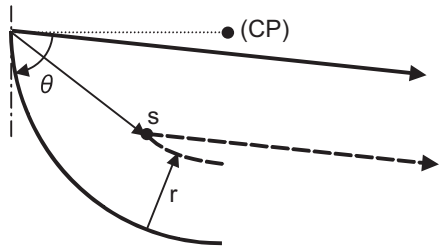
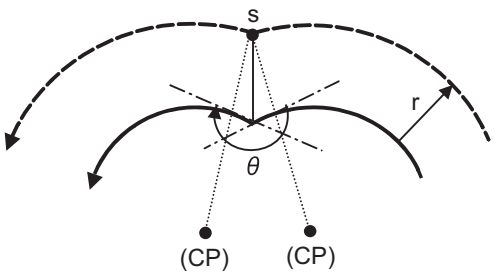
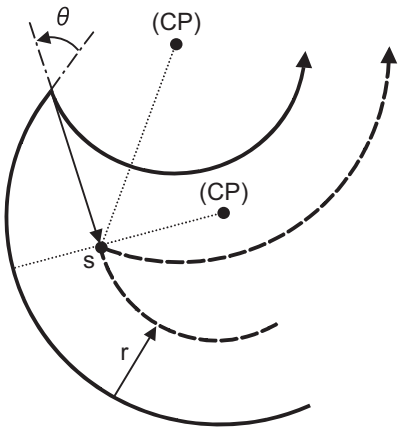
r : Compensation amount

s : Single block stop point

——— Program path

- - - - Tool center path

(2) Machining an inside corner

Linear -> Linear (Obtuse angle)	Linear -> Linear (Acute angle)
	
Linear -> Circular (Obtuse angle)	**Linear -> Circular (Acute angle)**
	
Circular -> Linear (Obtuse angle)	**Circular -> Linear (Acute angle)**
	
Circular -> Circular (Obtuse angle)	**Circular -> Circular (Acute angle)**
	

(CP) Center of arc

r : Compensation amount

s : Single block stop point

——— Program path

- - - - Tool center path

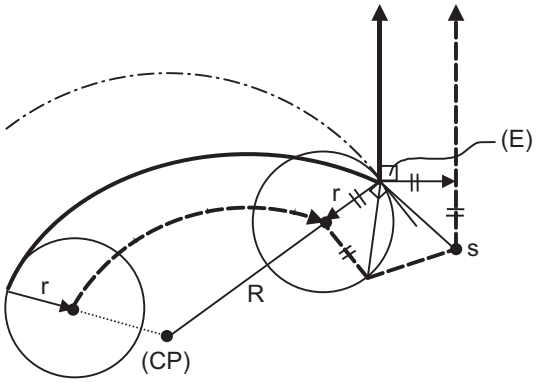
(3) When the arc end point is not on the arc

[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.



(E) End point of arc
(CP) Center of arc
r : Compensation amount

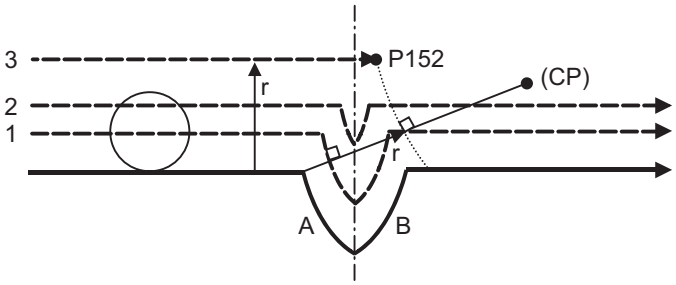
(4) When the inner intersection point does not exist

In the case shown in the figure below, the intersection point of arc A and arc 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 the pattern 3, compensation amount "r" is so large that an intersection does not exist and program error (P152) occurs.



(CP) Center of arc A
r : Compensation amount

Program path Tool center path

Tool radius compensation cancel

If any of the following conditions is met during the tool radius compensation mode, the tool radius compensation is canceled.

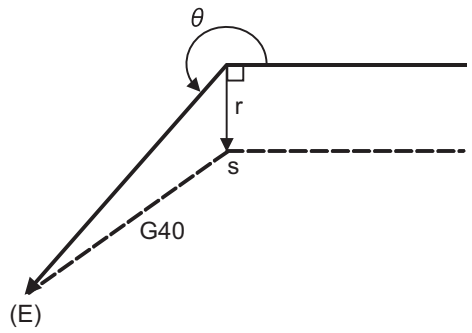
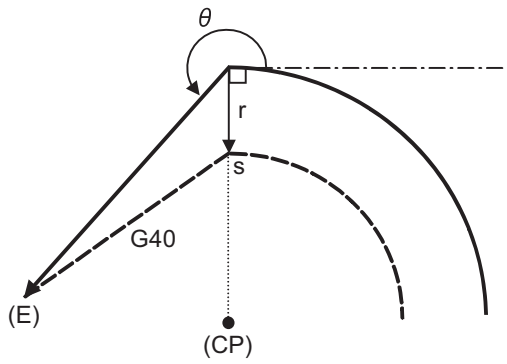
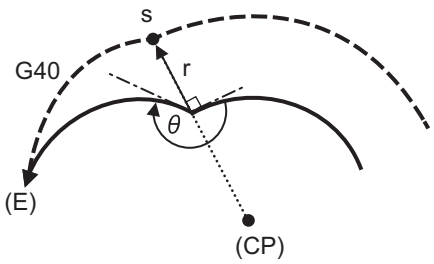
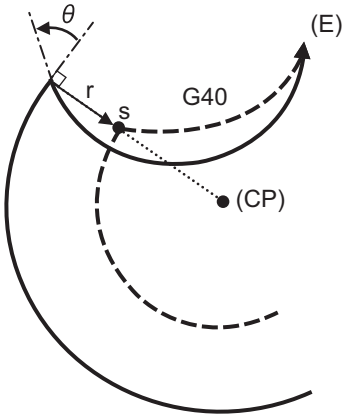
(1) The G40 command has been executed.
(2) Executed the compensation No.D00.

If the compensation is canceled during circular interpolation, spiral interpolation or involute interpolation mode, the program error (P151) occurs.
However, if the parameter "#1297 ext33/bit5" is set to "1", G40 can be commanded alone in a block only during circular interpolation or spiral interpolation mode. In this case, the cancellation operation is performed with the movement command following the G40 (commanded alone in a block).
The setting of the parameter "#1297 ext33/bit5" depends on the MTB specifications.

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

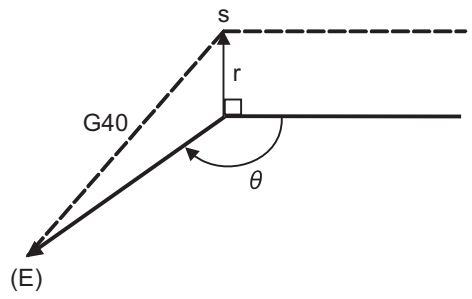
Linear -> Linear	Circular -> Linear
	
Circular -> Circular (Obtuse angle) (*1)	**Circular -> Circular (Acute angle) (*1)**
	

(*1) G40 command can be issued alone only when the parameter "#1297 ext33/bit5" is set to "1" and the circular interpolation or spiral interpolation mode is selected.

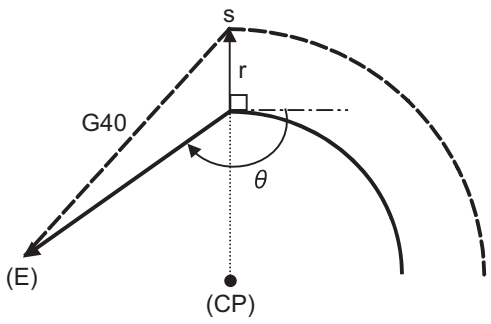
(E) End point
r : Compensation amount
——— Program path
(CP) Center of arc
s : Single block stop point
- - - - Tool center path

(2) Machining an outside corner (obtuse angle) [90° ≤ θ < 180°]

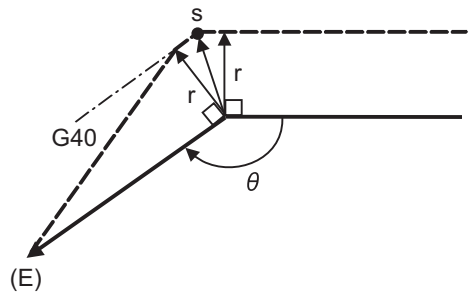
Linear -> Linear (Type A)



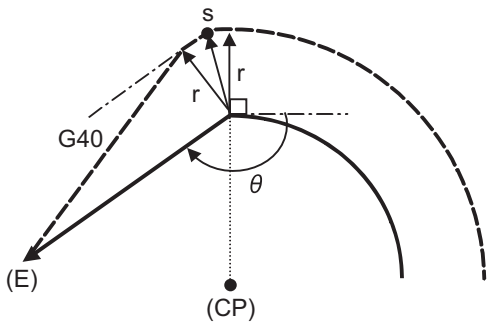
Circular -> Linear (Type A)



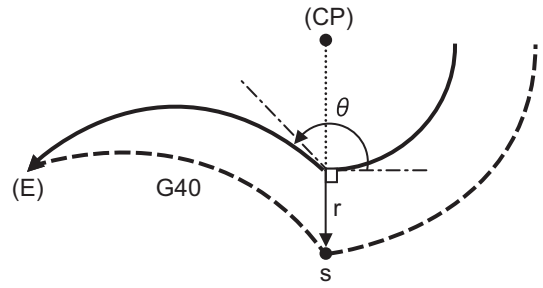
Linear -> Linear (Type B)



Circular -> Linear (Type B)



Circular -> Circular (90° ≤ θ < 180°) (*1)

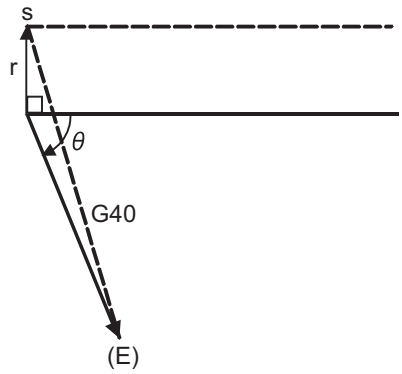


(*1) G40 command can be issued alone only when the parameter "#1297 ext33/bit5" is set to "1" and the circular interpolation or spiral interpolation mode is selected.

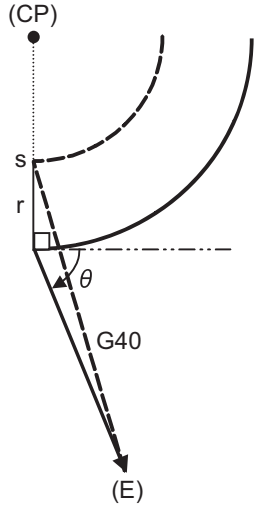
(E) End point
(CP) Center of arc
r : Compensation amount
s : Single block stop point
—— Program path
- - - - Tool center path

(3) Machining an outside corner (acute angle) [θ < 90°]

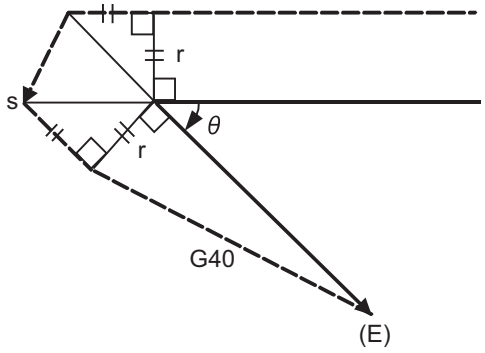
Linear -> Linear (Type A)



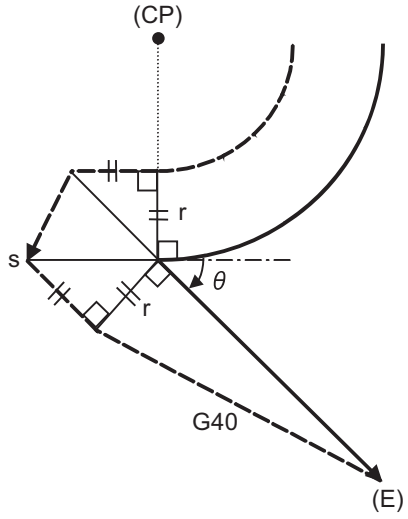
Circular -> Linear (Type A)



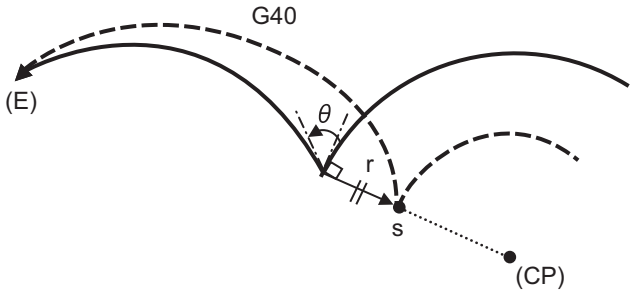
Linear -> Linear (Type B)



Circular -> Linear (Type B)



Circular -> Circular (0° < θ < 90°) (*1)



(*1) G40 command can be issued alone only when the parameter "#1297 ext33/bit5" is set to "1" and the circular interpolation or spiral interpolation mode is selected.

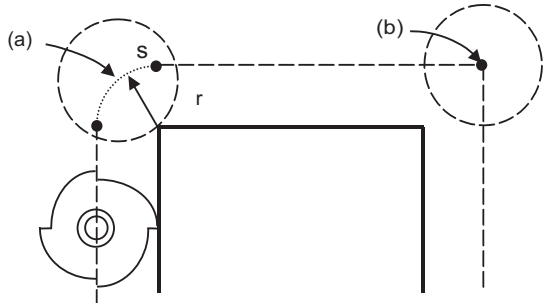
(E) End point
(CP) Center of arc
r : Compensation amount
s : Single block stop point
—— Program path
- - - - Tool center path

12.3.2 Other Commands and Operations during Tool Radius Compensation

Detailed description

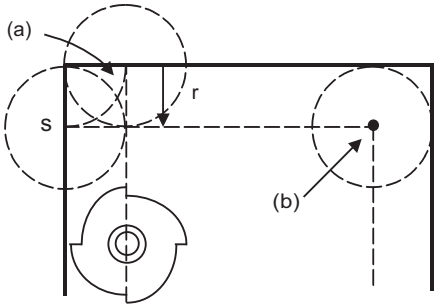
Insertion of corner arc

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.



(With G39 command) (No G39 command)

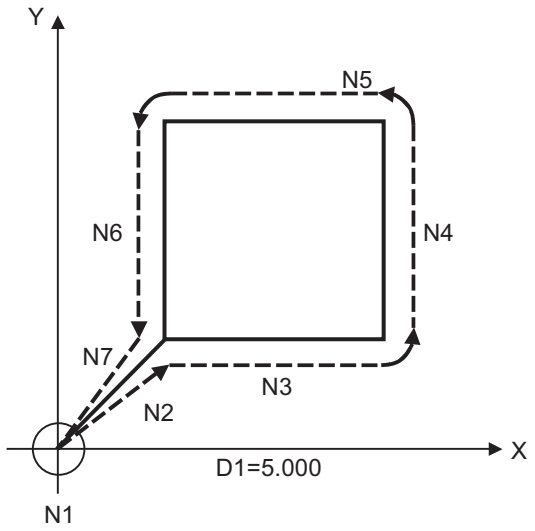
[For outer side compensation]



(With G39 command) (No G39 command)

[For inner side compensation]

(a) Inserted circular (b) Point of intersection r : Compensation amount s: Stop point with single block



Programmed path

Tool center path

```
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.

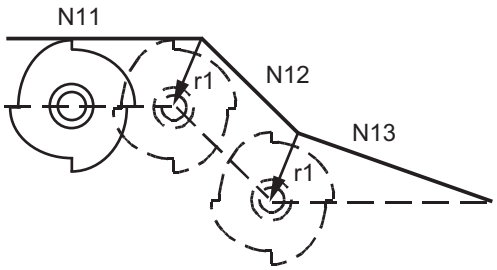
(1) **Holding of vector**

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.

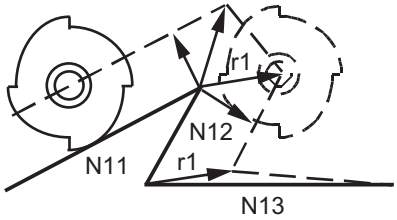
[Holding the inside compensation vector]



N11 G01 Xx11 ;
N12 G38 Xx12 Yy12 ;
N13 G40 Xx13 ;

r1: Vector at N11-N12 block intersection calculation

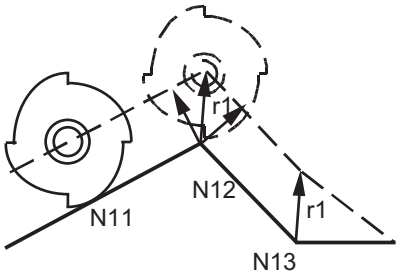
[Holding the outside compensation acute angle]



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

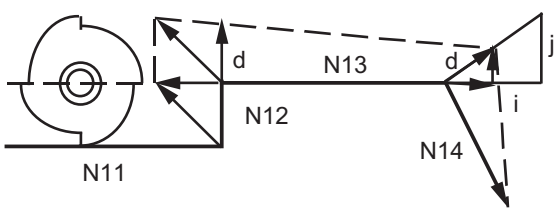
- - - - Tool center path

(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.

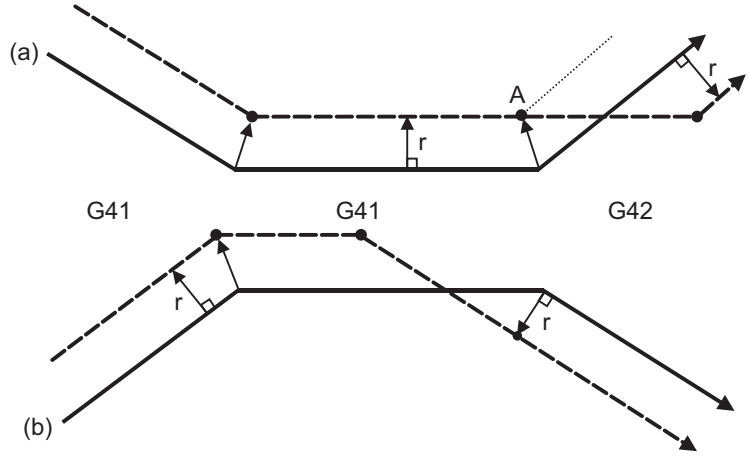
Changing the compensation direction during tool radius compensation

The compensation direction is determined by the tool radius compensation commands (G41, G42) and compensation amount sign.

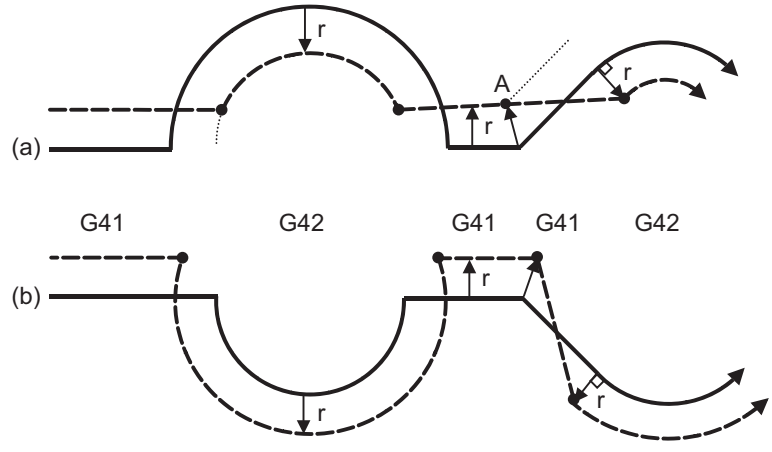
G Code	Compensation amount sign +	Compensation amount sign -
G41	Left-side compensation	Right-side compensation
G42	Right-side compensation	Left-side compensation

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.

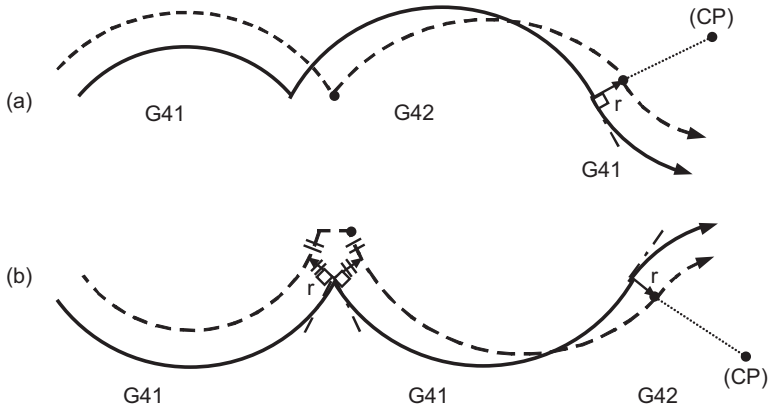


(3) Circular -> Circular

(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 arc



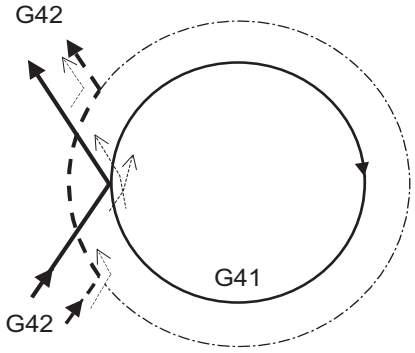
(4) Linear return



(5) In the following cases, it is possible that the arc may exceed 360°.

- The compensation direction is switched using G41/G42.
- I, J, K are commanded in G40.

In this case, the tool center passes a section where arcs are overlapped by the compensation. Uncut section will be left.




```
:
G42 G01 X_ Y_;
G41 G02 X_ Y_ I_ J_;
G42 G01 X_ Y_;
:
```

Program path

Tool center path

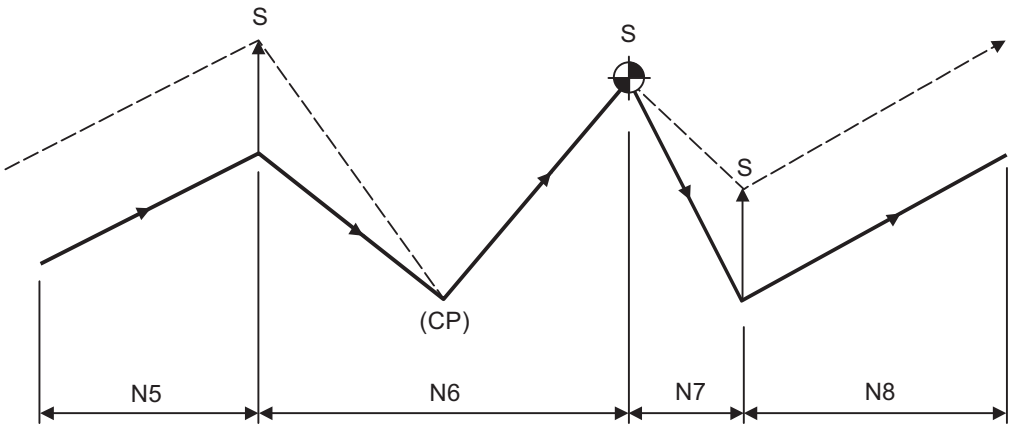
Section left uncut

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
N7     X30. Y-60. ;        point.
N8     X70. Y40. ;         (Reference position when there is no intermediate point)
     :
```

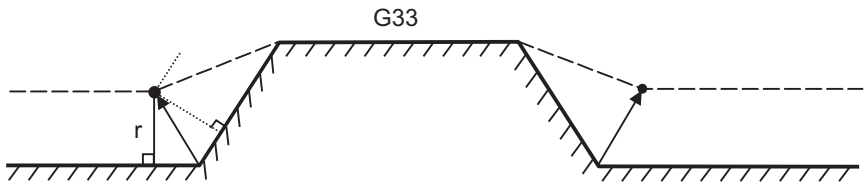
(2) The compensation vector will be eliminated temporarily with the G53 command (Basic machine coordinate system selection).

<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

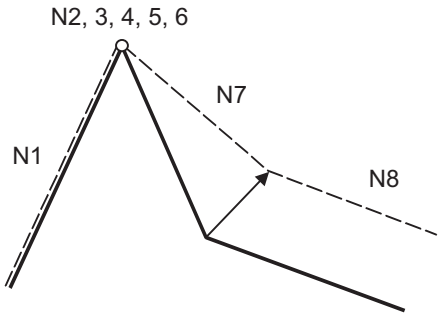
M00, M01, M02 and M30 are handled as pre-read inhibit M codes.

(1) When command is assigned at start of the compensation

Compensation is performed perpendicularly to the next movement block.
However, 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. ;
```

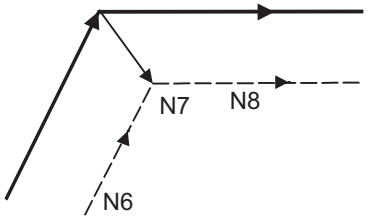
Block without movement



(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. ;
```

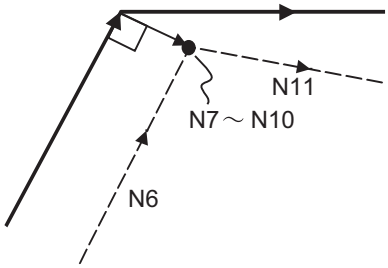


Block N7 is executed at N7 in the figure.

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. ;
```

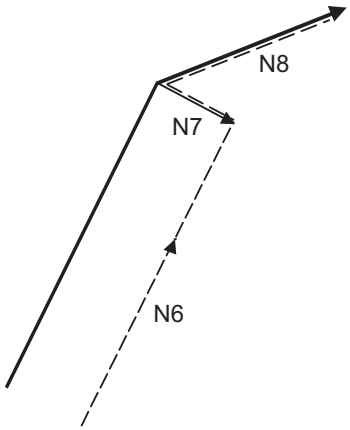
Block without movement



(3) When commanded together with compensation cancel

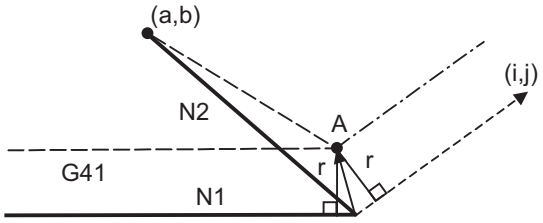
Only the compensation vectors are canceled when a block without movement is commanded together with the G40 command.

```
N6 X100. Y200. ;
N7 G40 M5 ;
N8 X100. Y50. ;
```



When I, J, K are commanded in G40

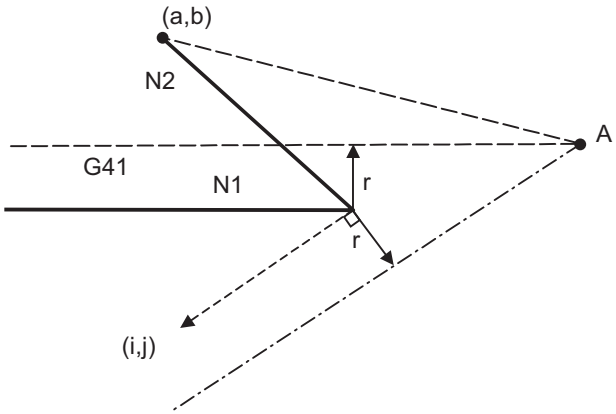
(1) If the final movement command block in the four blocks before the G40 block is the G41 or G42 mode, it will be assumed that the movement is commanded in the vector I, J or K direction from the end point of the final movement command. After interpolating between the hypothetical tool center path and point of intersection, it will be canceled. The compensation direction will not change.



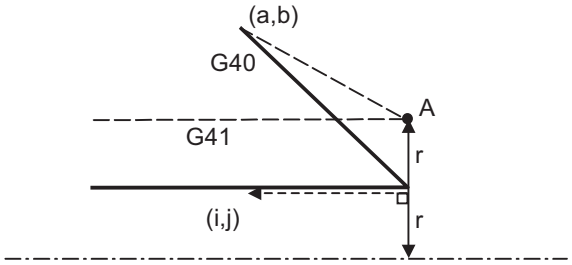
N1 (G41) G01 X_ ;
N2 G40 Xa Yb Ii Jj ;

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]

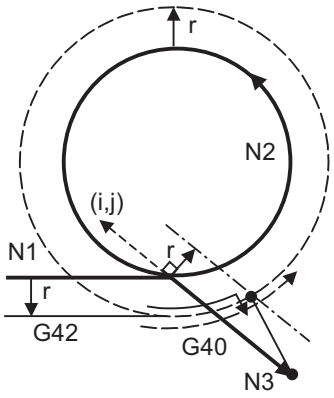


If the compensation vector obtained via a point of intersection calculation is extremely large, a perpendicular vector will be created in the block before G40.



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. ;

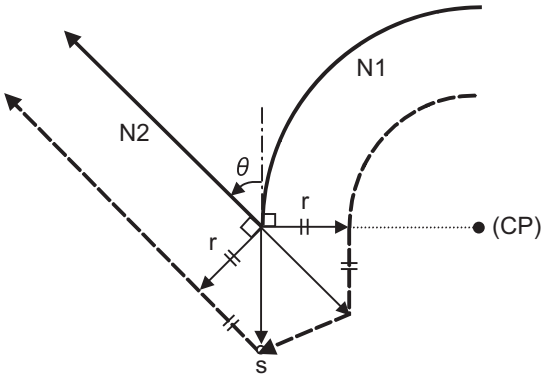
Programmed path

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 arc
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__ ;

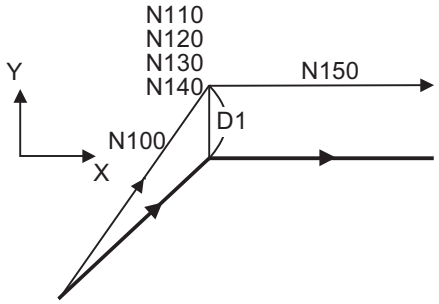
Assign a linear command (G00, G01) in a movement mode.

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

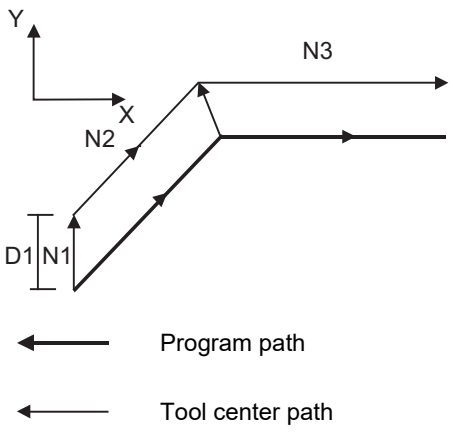



```
                    (G40)
                      ≀
N100  G91  G41  X100. Y100. I150. D1 ;
N110  G04  X1000 ;
N120  G01  F1000 ;
N130  S500 ;
N140  M03 ;
N150  X150. ;
                      ≀
```

Program path

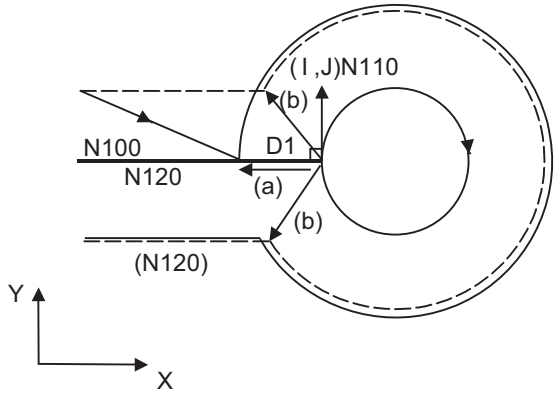
Tool center path

(2) When there are no movement commands at the compensation start.




```
(G40)
≀
N1  G41  I150. D1 ;
N2  G91  X100. Y100. ;
N3  X150. ;
≀
```

(3) When I, J has been commanded in the G41/42 mode (G17 plane)




```
≀
(G17  G41  G91)
N100  G41  G00 X150. J50. ;
N110  G02  I150. ;
N120  G00  X-150. ;
≀
```

(a) I, J type vector (b) Intersection point calculation type vector

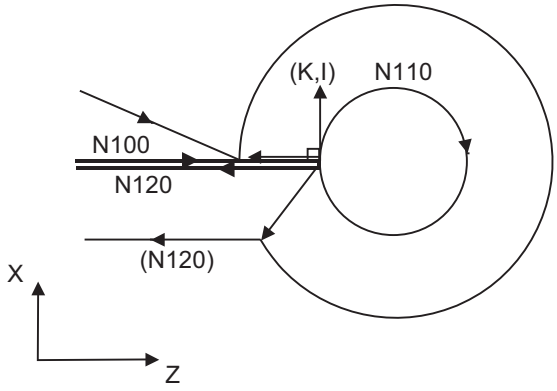
Program path

Tool center path

Path after intersection point calculation

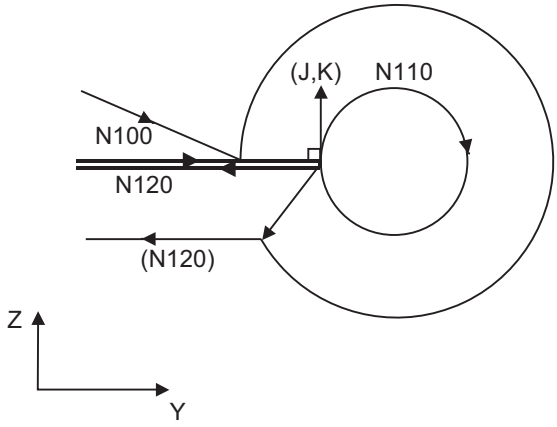
(Reference)

(a) G18 plane



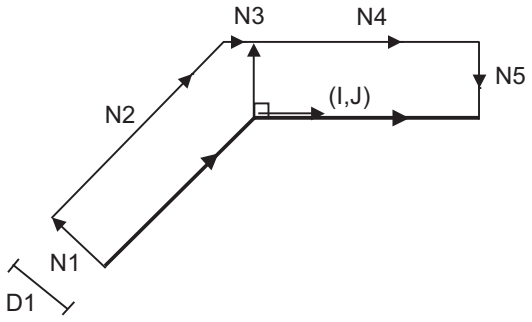

```
                ≀
          (G18  G41  G91)
N100  G41  G00  Z150. I50. ;
N110  G02  K50. ;
N120  G00  Z - 150. ;
                ≀
```

(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

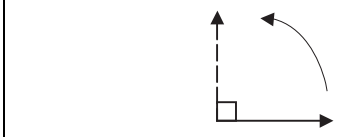
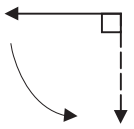



```
N1  G41  D1  G01  F1000 ;
N2  G91  X100. Y100. ;
N3  G41  I50. ;
N4  X150. ;
N5  G40 ;
```

Compensation vector direction

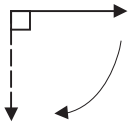
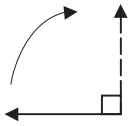
(1) In G41 mode

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

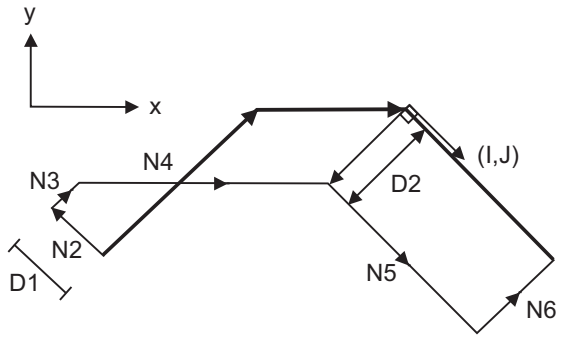
(2) In G42 mode

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.

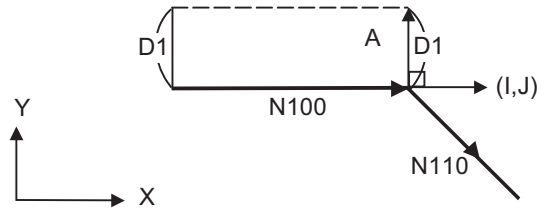



```
N1  G28  X0 Y0  ;
N2  G41  D1  F1000  ;
N3  G01  G91  X100.  Y100.  ;
N4  G42  X100.  I100.  J-100. D2 ;
N5  X100.  Y-100.  ;
N6  G40  ;
N7  M02  ;
%
```

Compensation amount for compensation vectors

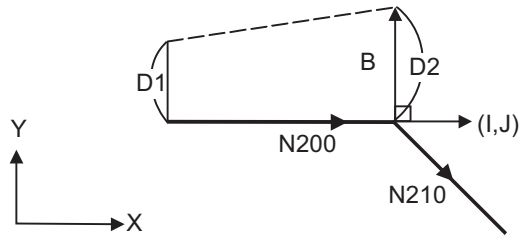
The compensation amount is determined by the offset No. (modal) in a block with the IJ designation.

<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.

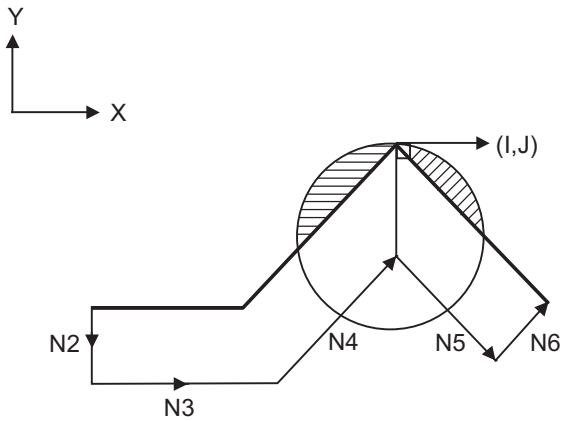



```
(G41 D1 G91)
      ≀
N200 G41 X150. I50. D2 ;
N210 X100. Y-100. ;
      ≀
```

Precautions

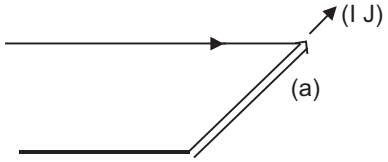
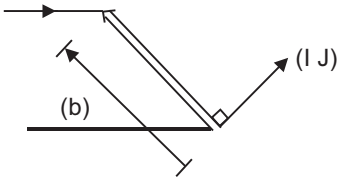
(1) Issue the I, J type vector in a linear mode (G00, G01). If it is in an arc mode at the start of compensation, program error (P151) will occur.
When it is in the compensation mode as well as in the arc mode, I, J will be designated at the center of the circular.

(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.



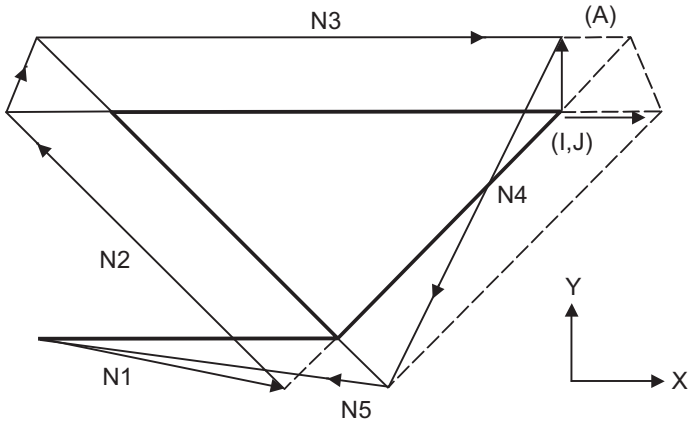

```
N1 G28 X0 Y0 ;
N2 G42 D1 F1000 ;
N3 G91 X100. ;
N4 G42 X100. Y100. I10. ;
N5 X100. Y-100. ;
N6 G40 ;
N7 M02 ;
```

(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

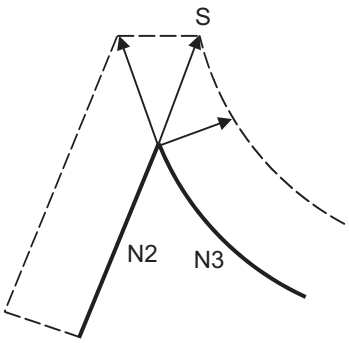
Detailed description

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	MDI interruption
N1 G41 D1;	
N2 X20. Y50. ;	
	<--- S1000 M3;
N3 G03 X40. Y-40. R70. ;	

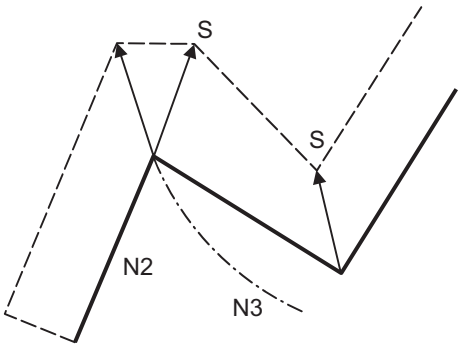


(2) Interrupt with movement

The compensation vectors are automatically re-calculated in the movement block after interrupt.

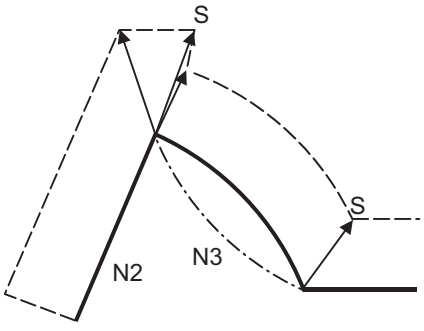
With linear interrupt

Automatic operation	MDI interruption
N1 G41 D1;	
N2 X20. Y50. ;	
	<--- X50. Y-30. ;
	X30. Y50. ;
N3 G03 X40.Y-40. R70. ;	



With circular interruption

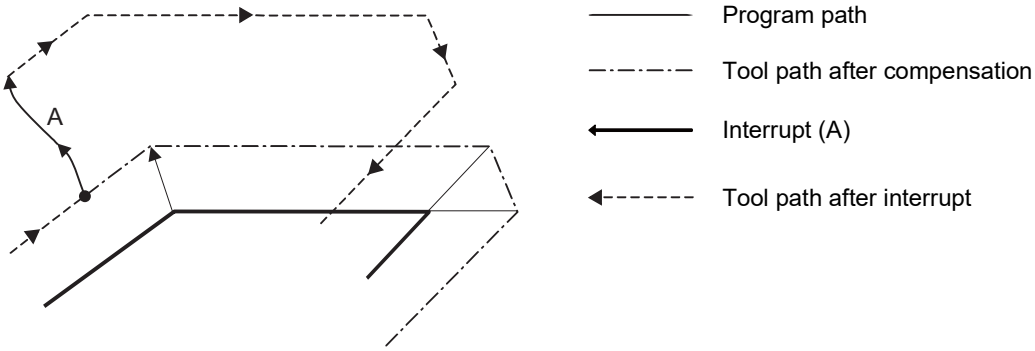
Automatic operation	MDI interruption
N1 G41 D1;	
N2 X20. Y50. ;	
	<--- G02 X40. Y-40. R70. ;
	G01 X40. ;
N3 G03 X40. Y-40. R70. ;	



Manual interruption

(1) Interrupt with manual absolute OFF.

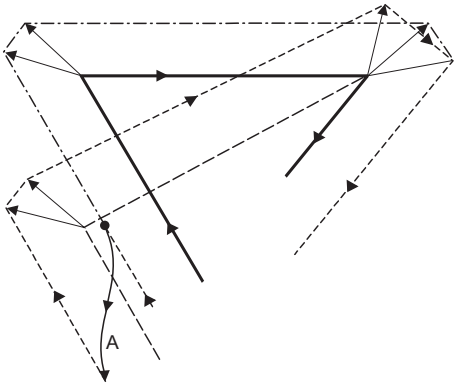
The tool path will deviate from the compensated path by the interrupt amount.



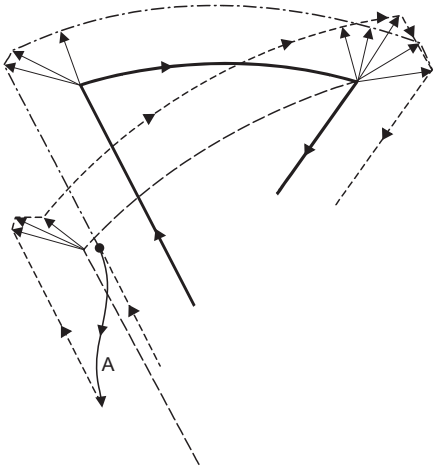
(2) Interrupt with manual absolute ON

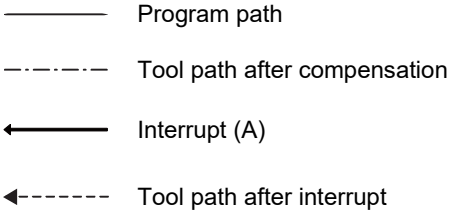
In the incremental mode, the same operation will be performed as the manual absolute OFF.
In the absolute mode, however, the tool returns to its original path at the end point of the block following the interrupted block, as shown in the figure.

[Line-Line-Line]



[Line-arc-Line]





12.3.5 General Precautions for Tool Radius Compensation

Precautions

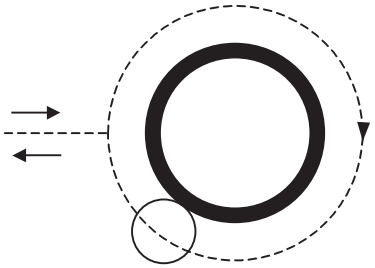
Assigning the compensation amounts

(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) 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.

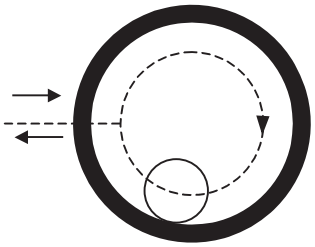
(2) Compensation amounts are normally changed when a different tool has been selected in the compensation cancel mode. However, when an amount is changed during the compensation mode, the vectors at the end point of the block are calculated using the compensation amount designated in that block.

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.)



G41 offset amount (+) or G42 offset amount (-)
(a)



G41 offset amount (-) or G42 offset amount (+)
(b)

---- Tool center path

12.3.6 Changing of Compensation No. during Compensation Mode

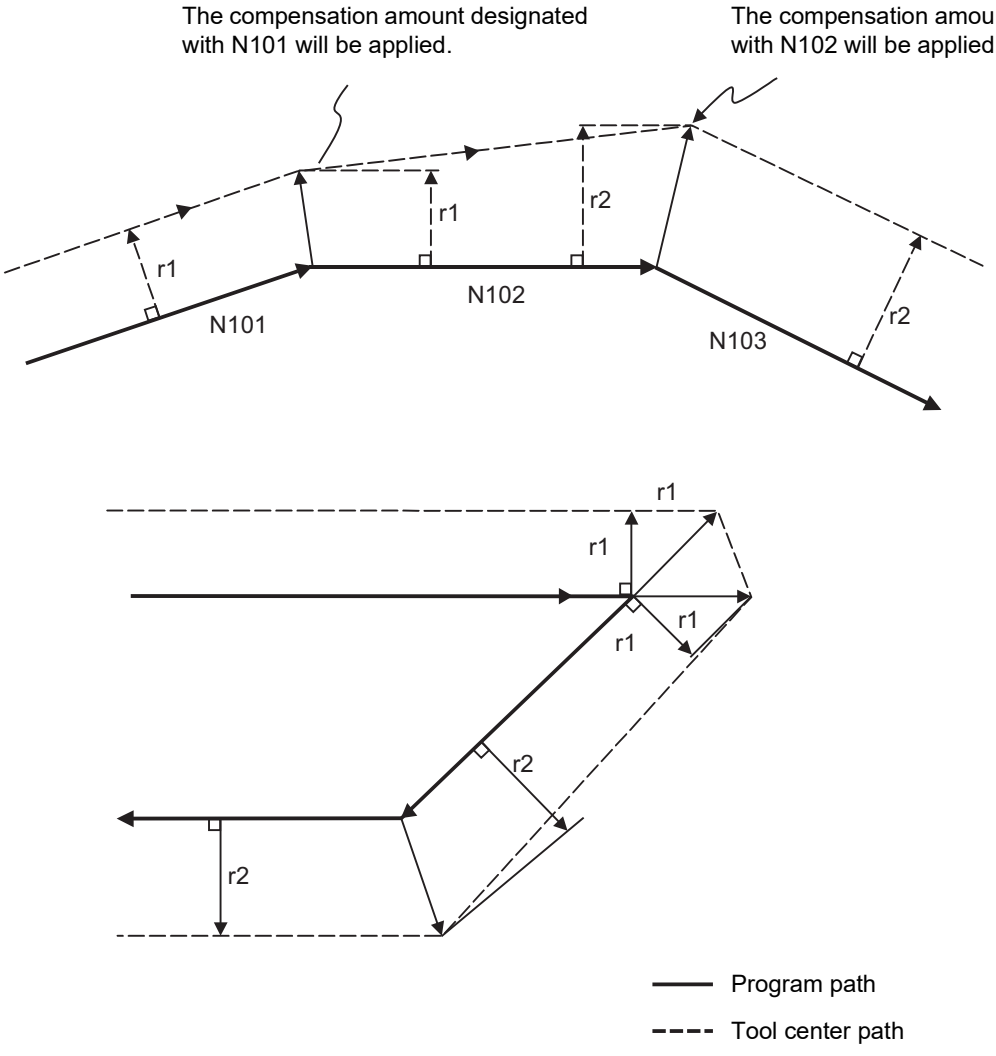
Function and purpose

As a principle, the compensation No. must not be changed during the compensation mode. If changed, the movement will be as shown below.

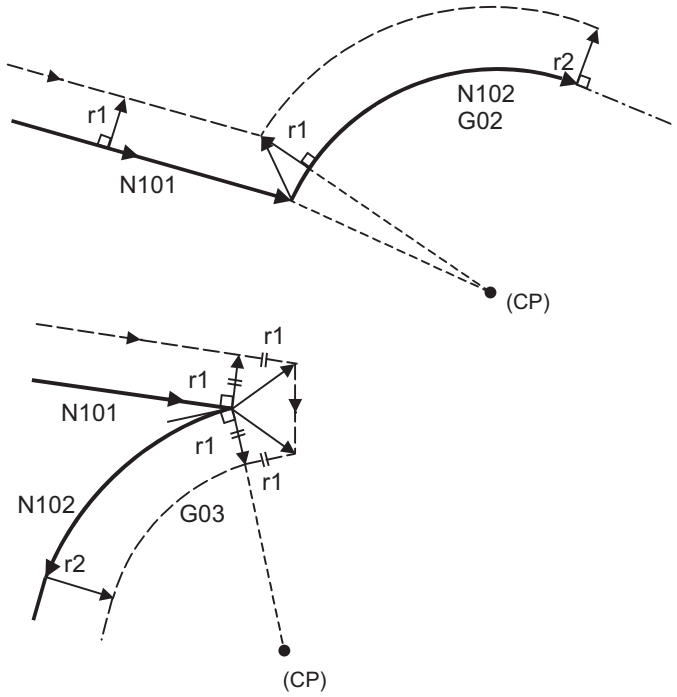
When compensation No. (compensation amount) is changed:

```
   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

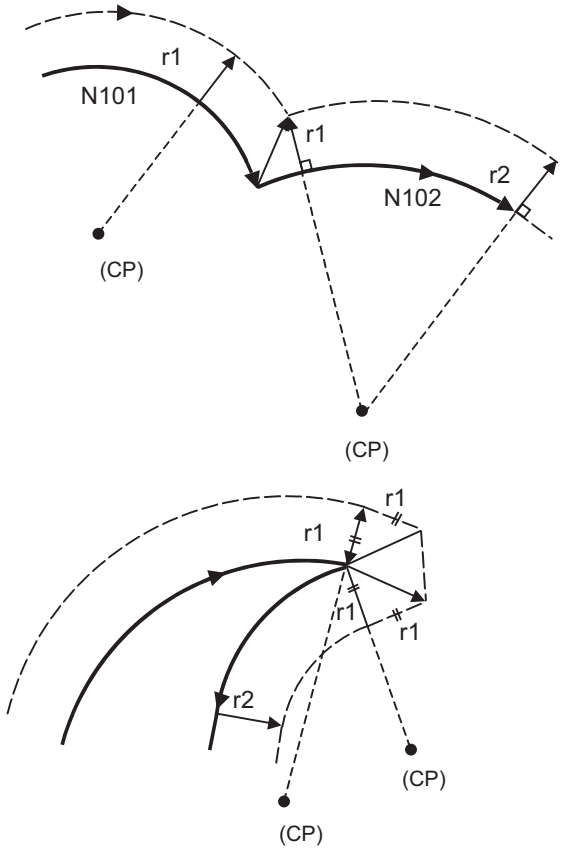


Linear ->circular



(CP) Arc center
Program path
Tool center path

Circular -> circular



(CP) Arc center
Program path
Tool center path

12.3.7 Start of Tool Radius Compensation and Z Axis Cut in Operation

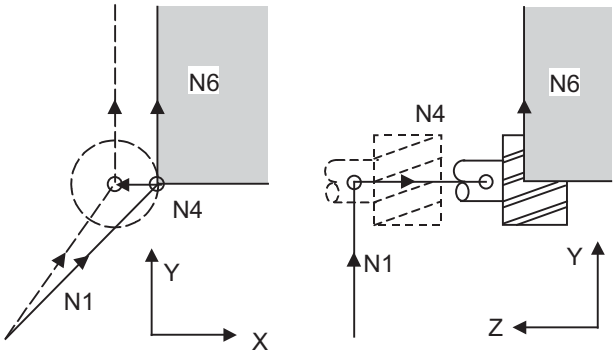
Function and purpose

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 ;
     :
     :
```



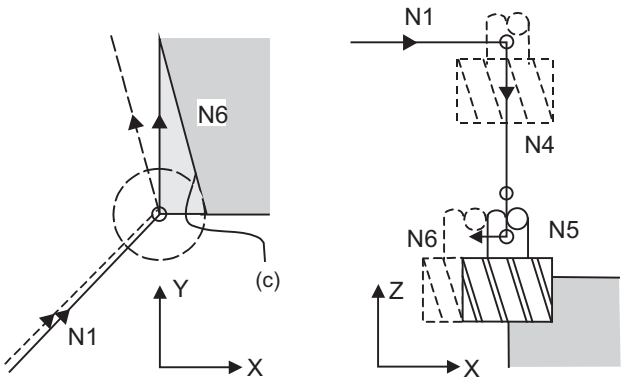
N4 Z axis lowers (1 block)

---- Tool center path

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.

When 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 ;
```

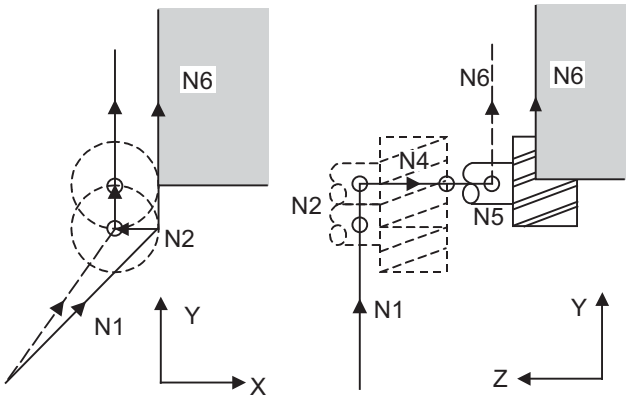


(c) Cut in

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

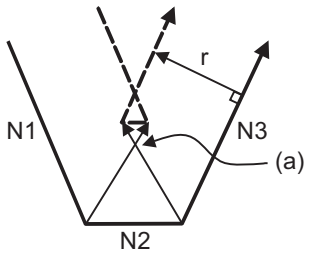
When tool radius is larger than the program path, a tool, compensated for by the tool radius compensation function, may sometimes cut into the workpiece. This is known as interference, and interference check is the function which prevents this from occurring.
The table below shows the three functions 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 micro segment program.

Detailed description

Conditions viewed as interference

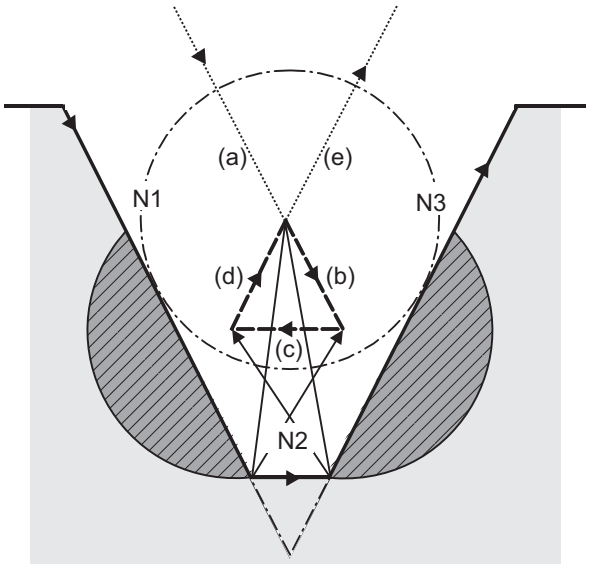
When there is a movement command in three of the five pre-read blocks, and if the compensation calculation vectors which are created at the contacts of movement commands intersect each other, it will be viewed as interference.



r : Compensation amount
(a) Vectors intersect
Program path
Tool center path

(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.;

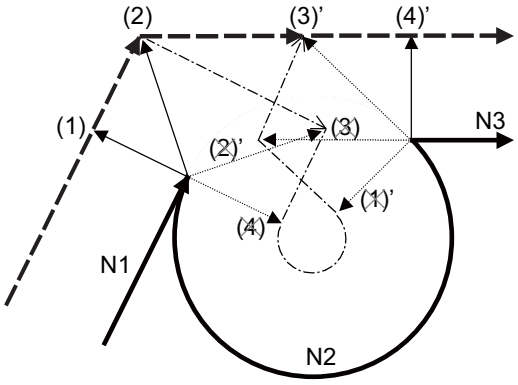
(2) With avoidance function

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

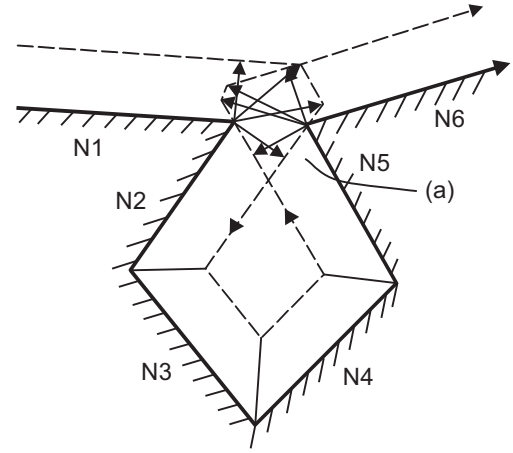
The tool center path will follow the path that connects (1), (2), (3), (4), (1)', (2)', (3)', (4)', as the interference avoidance path while cutting.

 (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



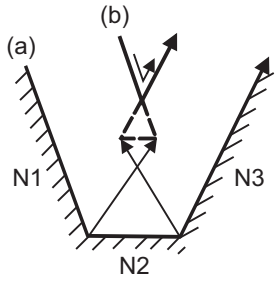
(a) Interference check is not possible

Program path

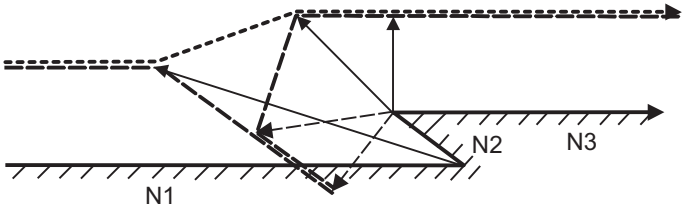
Tool center path

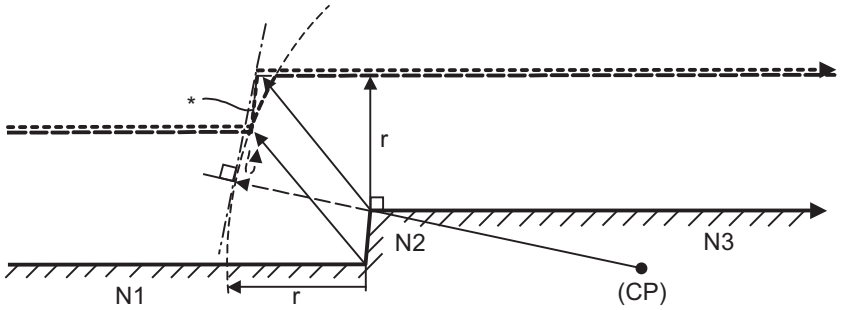
Operation when interference avoidance function is valid

The movement will be as shown below when the interference avoidance check is valid.



(a) Program path (b) Tool center path





Program path

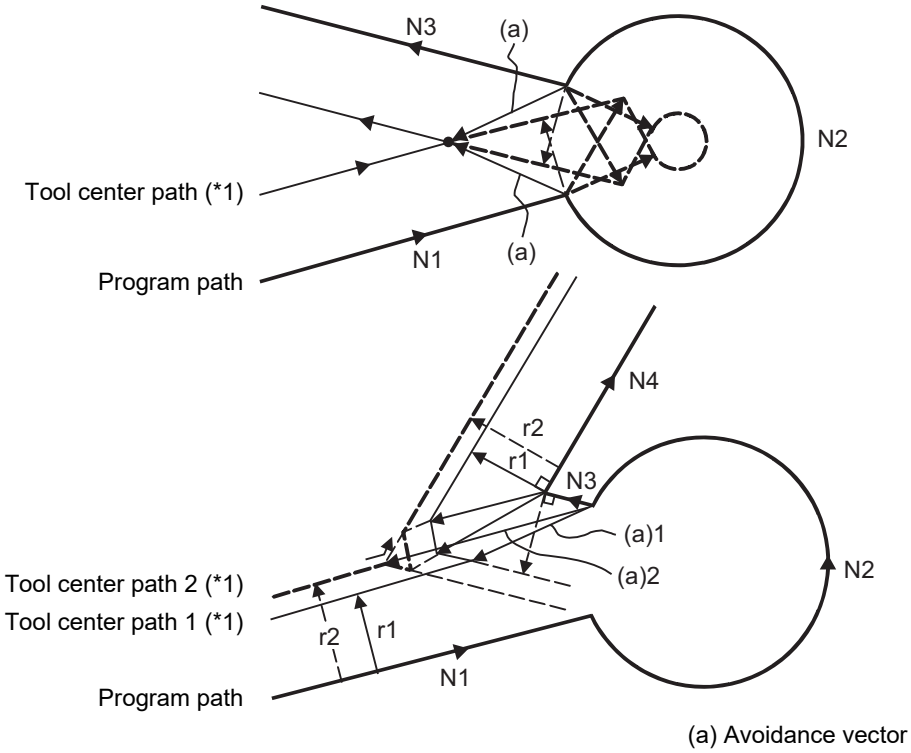
Tool center path without interference check

Tool center path when interference is avoided (*: Linear movement)

Valid vector

Invalid vector

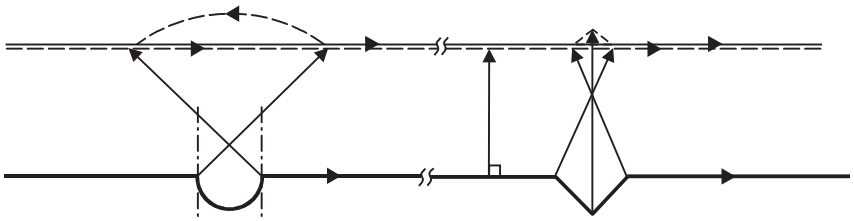
If all of the line vectors for the interference avoidance are deleted, create a new avoidance vector as shown in below to avoid the interference.



(a) Avoidance vector

(*1) Tool center path when interference is avoided

In the case of the figure below, the groove will be left uncut.



Program path

Tool center path without interference check

Tool center path when interference is avoided

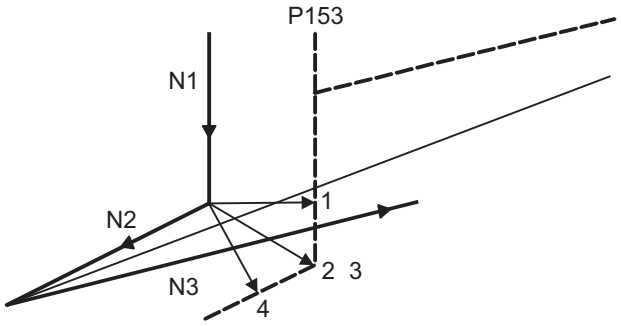
Interference check alarm operation

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

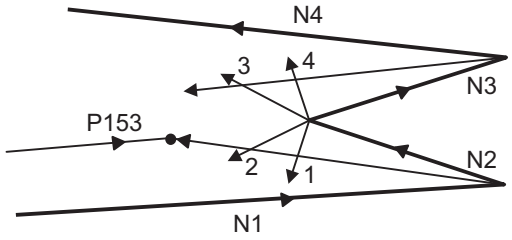
As shown in the figure below, when vectors 1 through 4 at the end point of the N1 block have all been deleted, program error (P153) will occur prior to N1 execution.



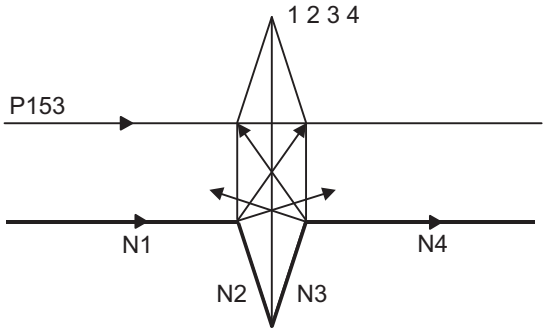
(2) When the interference check avoidance function has been selected

(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.

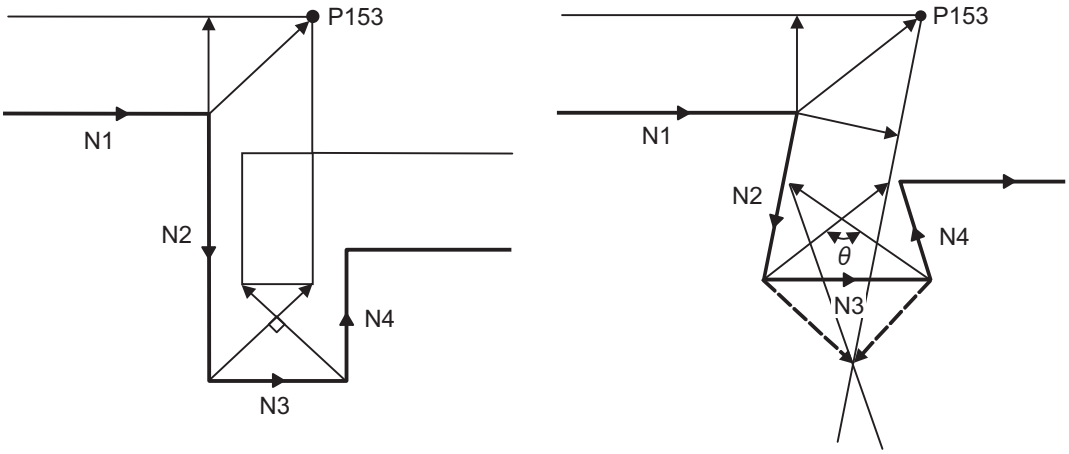


In the case shown in the figure below, the tool will move in the reverse direction at N2.
Program error (P153) now occurs before executing N1 and the operation stops.



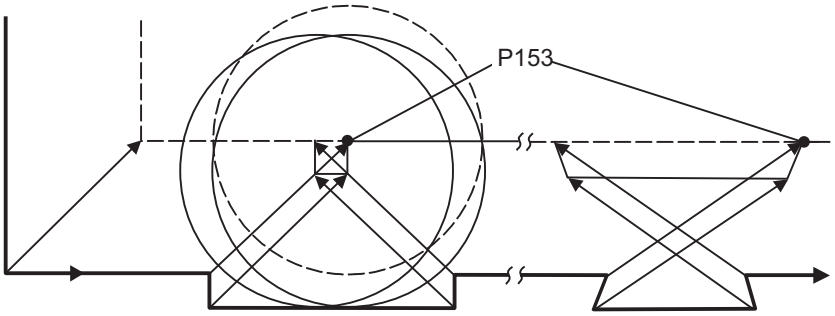
(Example 2) When avoidance vectors cannot be created

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

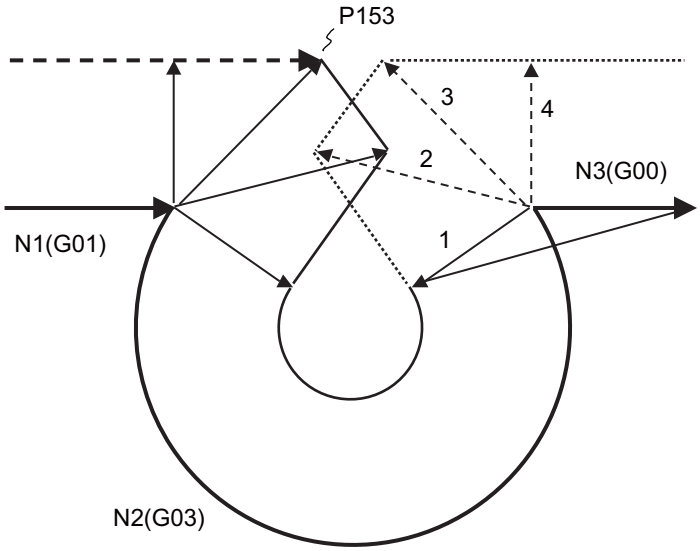
When grooves that are narrower than the tool diameter with parallel or widening bottom are programmed, it will still be regarded as interference even if there is actually no interference.



(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 G00 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

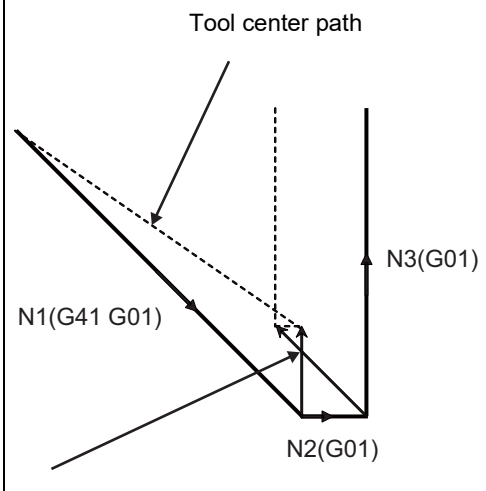
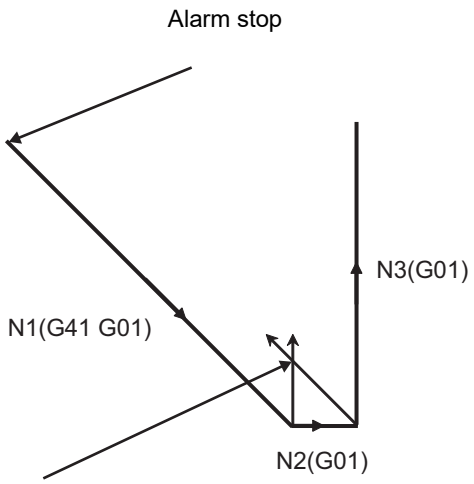
Tool center path when the interference check is invalid in N3 due to a command not to eliminate compensation vectors.

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.

(a) Interference check for start-up block invalid "#1241 set13/bit1" = "0" or "#8103 COLL. CHK OFF" = "1"	(b) Interference check for start-up block valid "#1241 set13/bit1" = "1" and "#8103 COLL. CHK OFF" = "0"
	
The interference with the compensation calculation vector of the contact between the N2 block and N3 block is not checked. Doing so will cause a cut in the N3 block.	The interference with the compensation calculation vector of the contact between the N2 block and N3 block is checked, and this is judged to be an alarm.

12.3.9 Diameter Designation of Compensation Amount

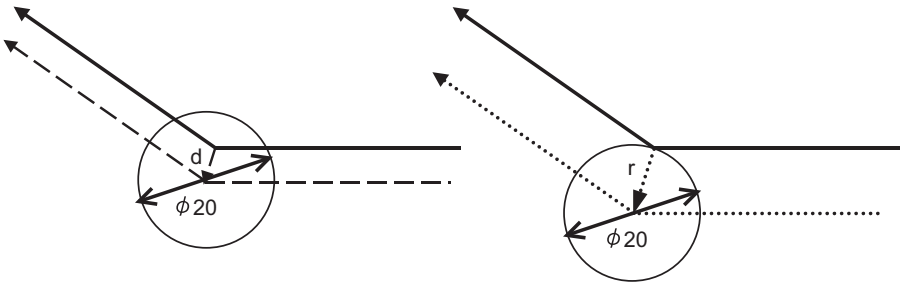
Function and purpose

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.

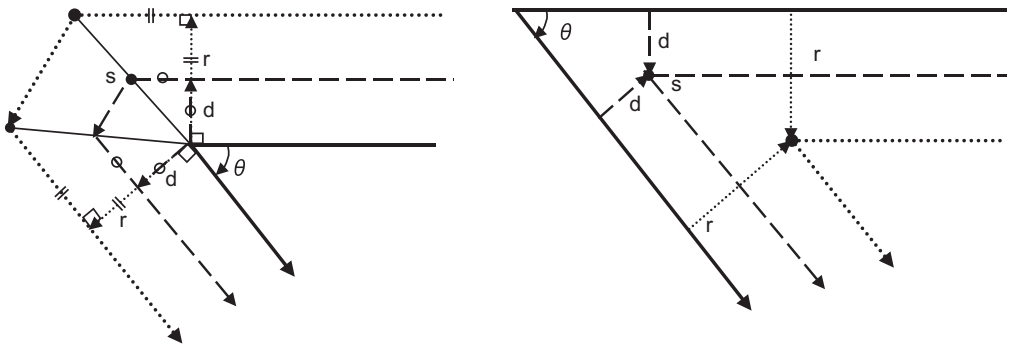
Operation example

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").)



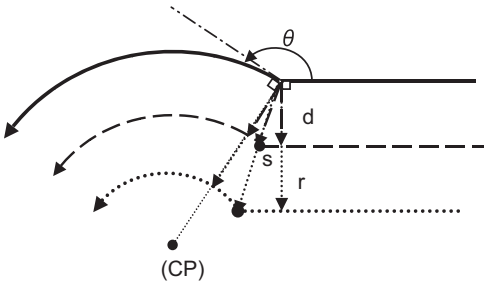
(1) Linear -> linear corner (acute angle)



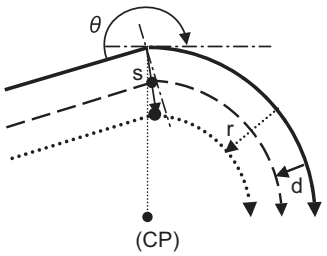
Outside of the corner　　Inside of the corner

Program path
Tool center path (When #8117 is ON)
Tool center path (When #8117 is OFF)

(2) Linear -> arc (obtuse angle)

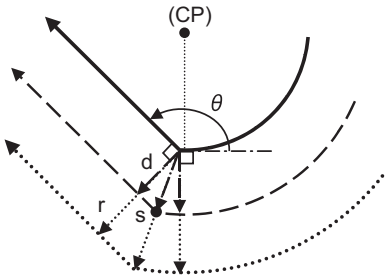


Outside of the corner

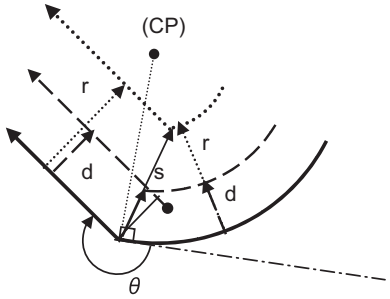


Inside of the corner

(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.

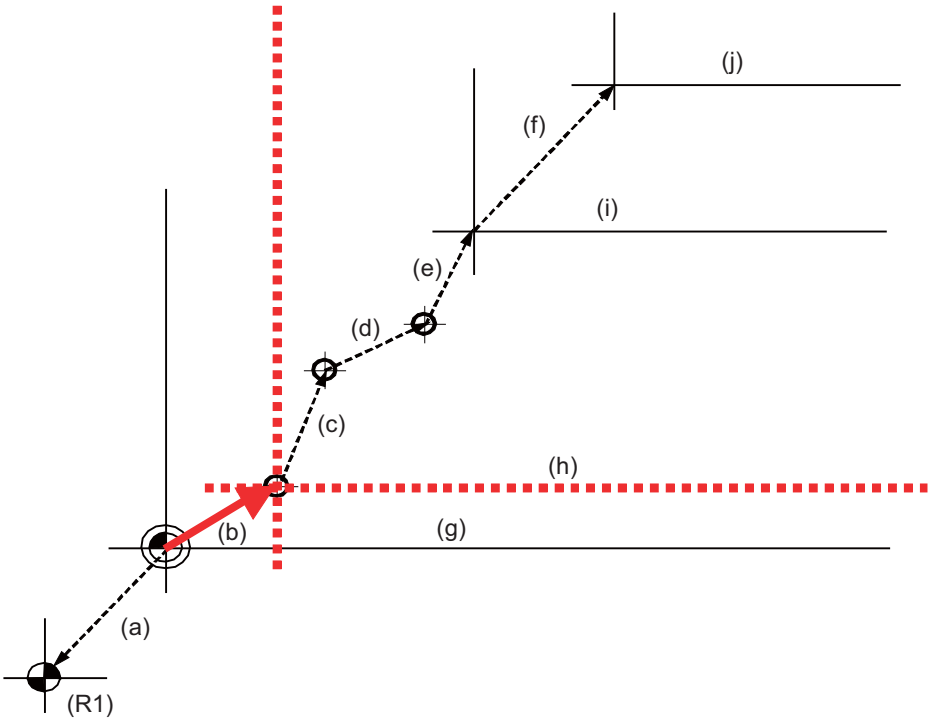
12.3.10 Workpiece Coordinate Changing during Radius Compensation

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.



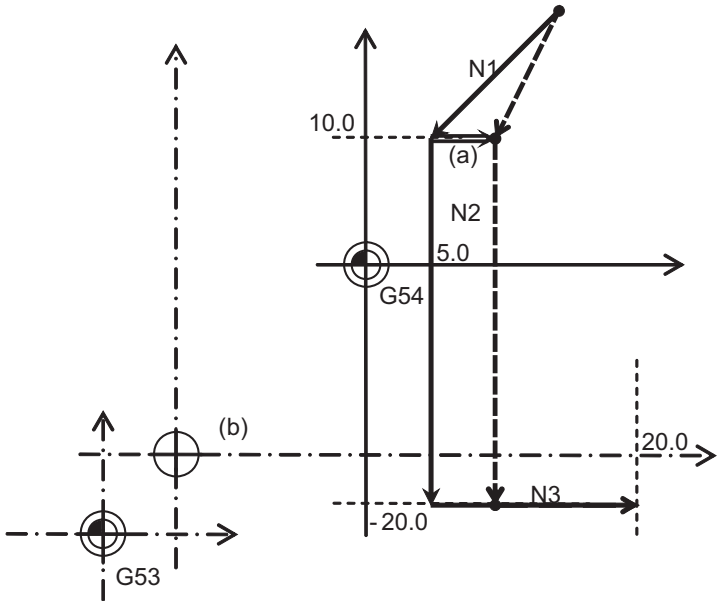
(R1) 1st reference position
(a) 1st reference position offset
(b) Interrupt amount offset
(c) Extended workpiece coordinate system offset
(d) G92 offset
(e) Workpiece coordinate system offset
(f) Local coordinate system offset
(g) G53 Basic machine coordinate system
(h) Program coordinate system
(i) G54 to G59/G54.1Pn Workpiece coordinate system/Extended workpiece coordinate system
(j) G52Local coordinate system

The coordinate system changed by parameter is as follows.

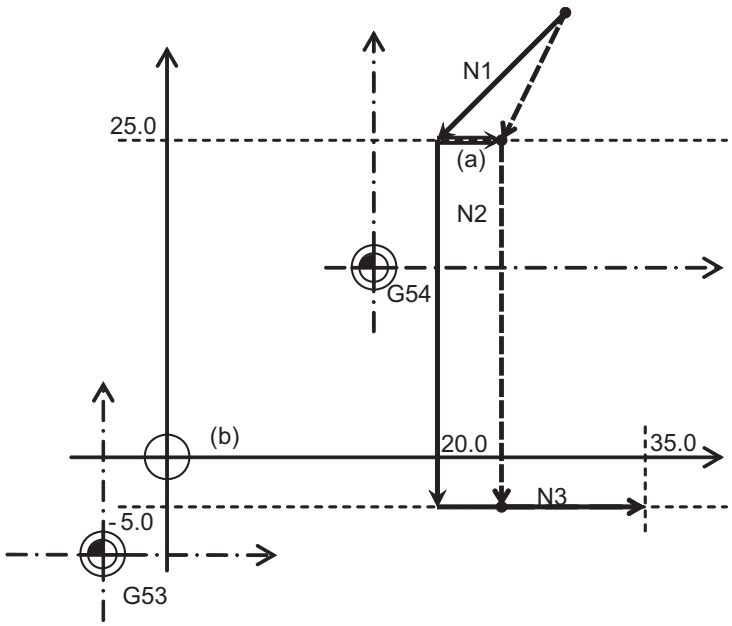
```
G90 G54 G00 X15. Y20.;
N1 G41 D3 X5. Y10. ;
N2 G01; Y-20 F1000;
N3 G40 X20. ;
M30 ;
```

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

12.3.11 Tool Offset Command Check

It is possible to make a program error occur when the tool offset No. is "0" or when the offset amount specified with the tool offset No. is "0", to prevent commands from being forgotten.

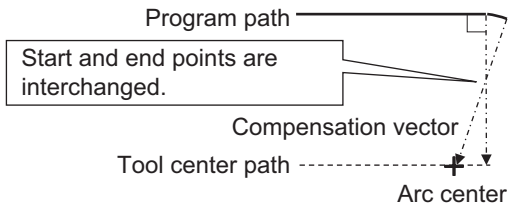
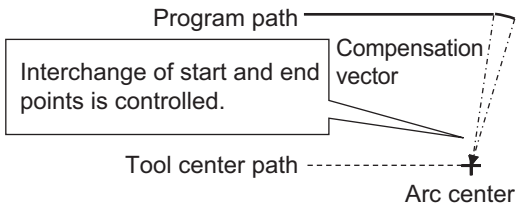
Note

(1) The D address of circle cutting (G12/G13) is not checked by the tool offset command check.

Refer to "12.7.2 Tool Offset Command Check" for details.

12.3.12 Arc Intersection Point Calculation Switch

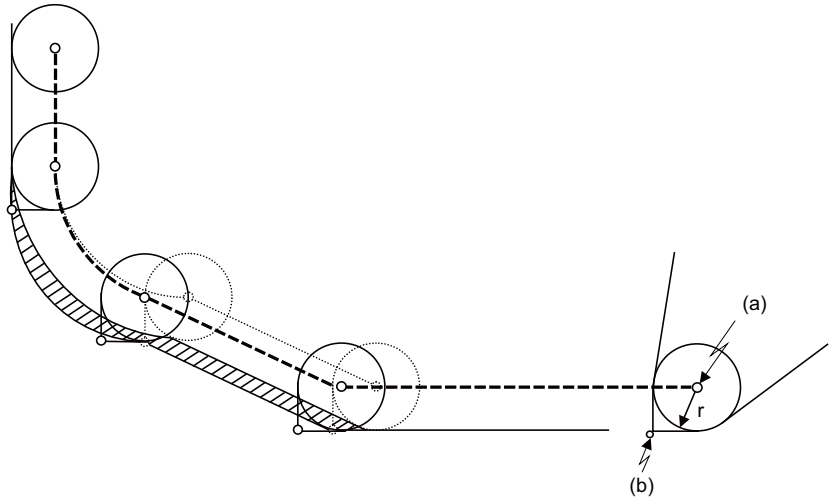
When the angle formed by the arc and the connecting movement block is near 180°, the start and end points after compensation are interchanged, and the program path direction is opposite to the tool center path direction. It may cause deceleration or unintended machining path.
In this case, interchange of the start and end points can be controlled by setting the control parameter "#8153 Arc calc method" to "1".

#8153 = 0	#8153 = 1
	

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**

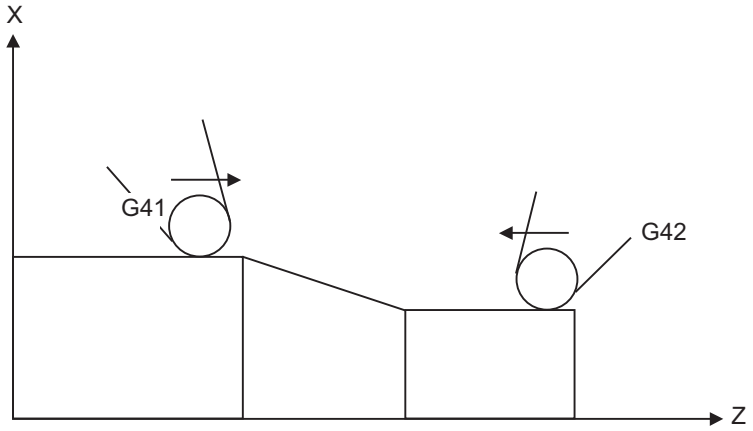
G41 (X__ Z__ D__);	**Tool nose radius compensation left**

G42 (X__ Z__ D__);	**Tool nose radius compensation right**

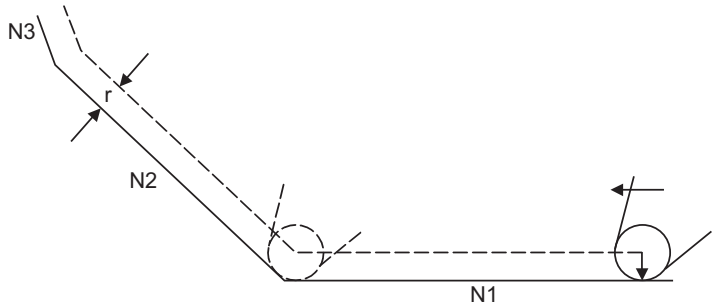
X	X axis end point coordinate (Absolute position of workpiece coordinate system)
Z	Z axis end point coordinate (Absolute position of workpiece coordinate system)
D	Compensation No. (The compensation No. setting range will differ according to the specifications (No. of compensation sets).)

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)

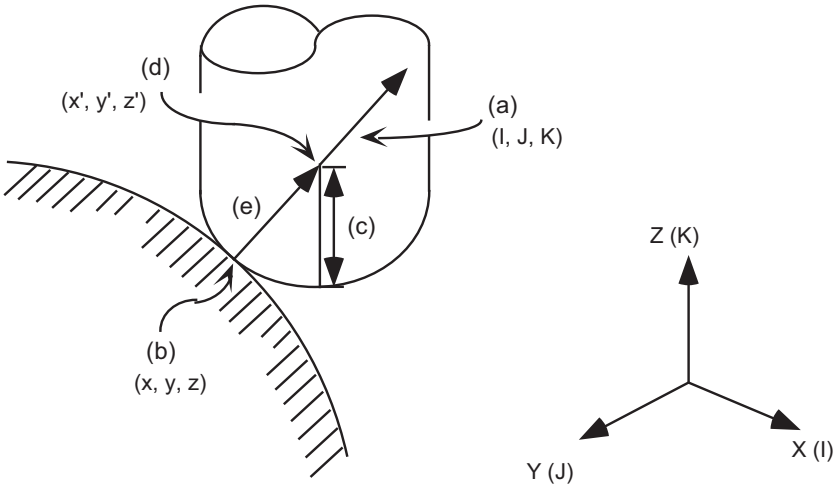
(6) A program error will occur when a tool nose radius compensation command is issued even though the tool nose radius compensation specification is not provided. (P150)

(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-dimentional 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 (I, J, K) (a).

Though two-dimensional tool radius compensation creates the vectors at a right angle to the (I, J, K) direction, three-dimensional tool radius compensation creates the vector in the (I, 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.

$$Hx = \frac{I}{\sqrt{(I^2 + J^2 + K^2)}} \times r$$

$$H_Y = \frac{J}{\sqrt{(I^2 + J^2 + K^2)}} \times r$$

$$H_Z = \frac{K}{\sqrt{(I^2 + J^2 + K^2)}} \times r$$

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 + Hx
y' = y + Hy
z' = z + Hz

Note

(1) Three-dimensional compensation vector (Hx, Hy, Hz) refers to the plane normal line vector whose direction is same as the plane normal line vector (I, 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", $\sqrt{(I^2 + J^2 + K^2)}$ 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-dimentional 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



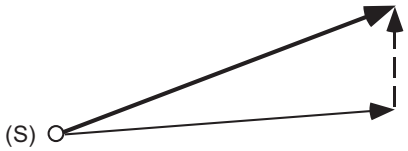
Detailed description

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 Ii Jj Kk ;	X Y Z space
	G17 ; G41 Yy Ii 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 Ii Jj Kk ;	X V Z space
	G17 W ; G41 Ww Ii Jj Kk ;	X Y W space
	G17 ; G41 Xx Yy Zz Ww Ii Jj Kk ;	X Y Z space
	G17 W ; G41 Xx Yy Zz Ww Ii Jj Kk ;	X Y W space
	G17 ; G41 Ii Jj Kk ;	X Y Z space
	G17 U ; G41 Ii Jj Kk ;	U Y Z space

Operation example

Compensation start: When there is a movement command

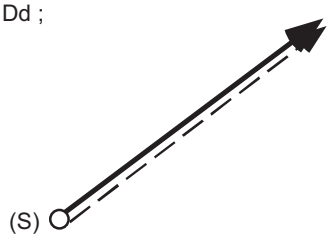
G41 Xx Yy Zz Ii Jj Kk Dd ;



(S) Start point
Tool center path
Programmed path
Three-dimensional compensation vector

Compensation start: When there is no movement command

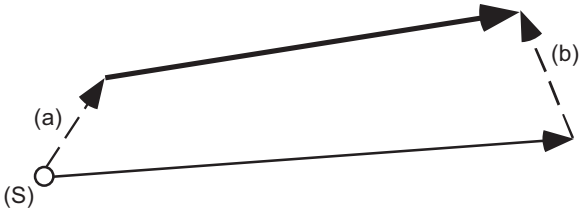
G41 Ii Jj Kk Dd ;



(S) Start point
Tool center path
Three-dimensional compensation vector

Movement during the compensation: 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: When there is no plane normal line vector command

Xx Yy Zz Ii Jj Kk ;



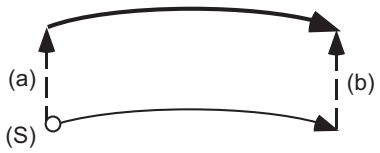
(S) Start point
(a) Old vector
(b) New vector
Tool center path
Programmed path

Movement during the compensation: 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

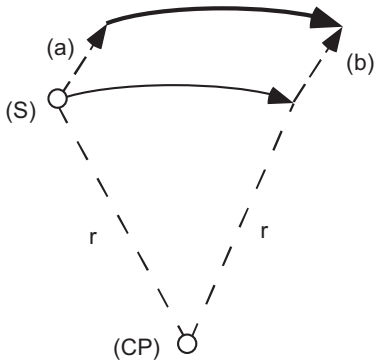


(S) Start point
(a) Old vector
(b) New vector
→ Tool center path
→ Programmed path

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.

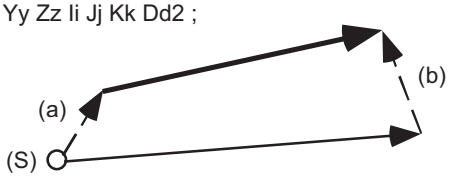
G02 Xx Yy (Zz) Ii Jj ; I, J(K) means the circular center
Or
G02 Xx Yy (Zz) Rr ; R-designated arc



(S) Start point
(a) Old vector
(b) New vector
(CP) Arc center
→ Tool center path
→ Programmed path

Movement during the tool radius compensation: When compensation amount is to be changed

G41 Xx Yy Zz Ii Jj Kk Dd1 ;
:
G41 Xx Yy Zz Ii Jj Kk Dd2 ;

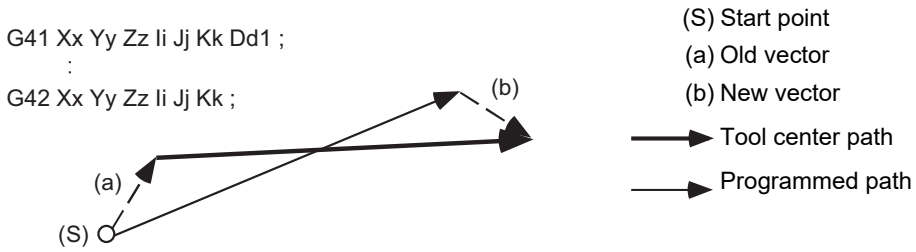


(S) Start point
(a) Old vector
(b) New vector
→ Tool center path
→ Programmed path

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: When compensation direction is to be changed

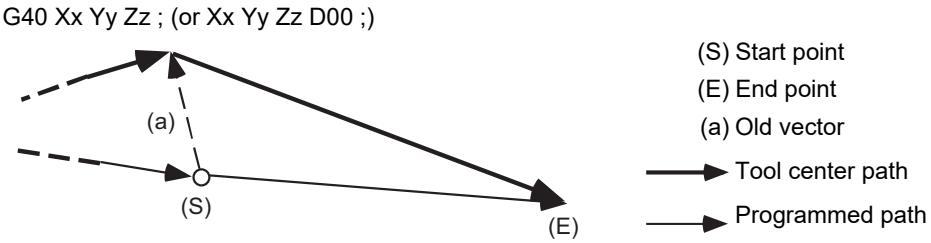


Note

(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



Tool radius compensation cancel: When there is no movement command



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. ;

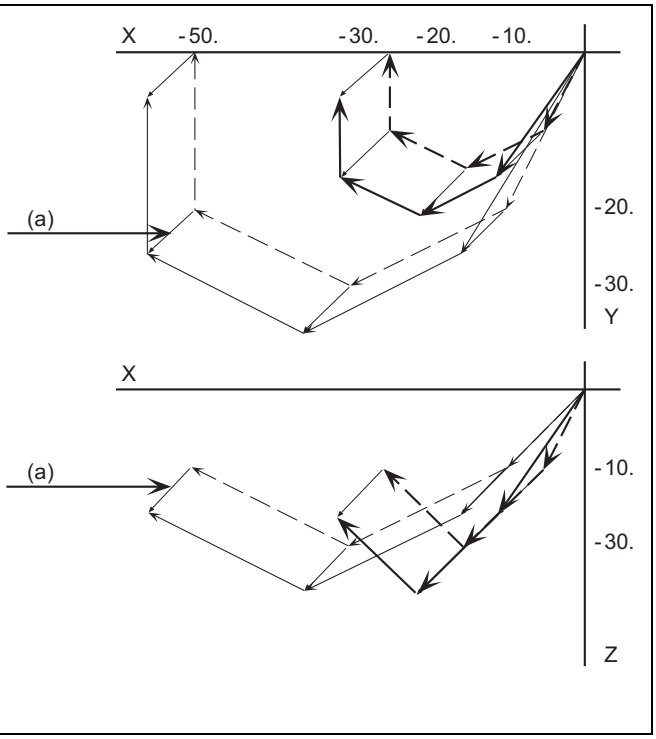

N1(  -5.000, -10.000, -10.000 )
N1( -10.773, -15.773, -15.773 )

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

Coordinate rotation by program 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 )

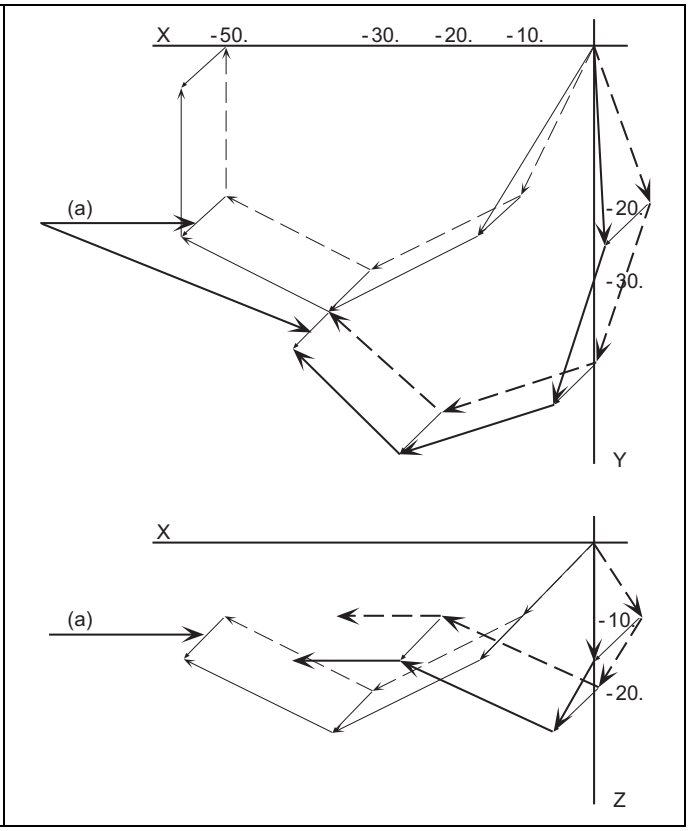
N2(   0.000, -42.426, -20.000 )
N2(   0.000, -50.591, -25.773 )

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.

D1=10.

G90 ;
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 G53 Y0 ;

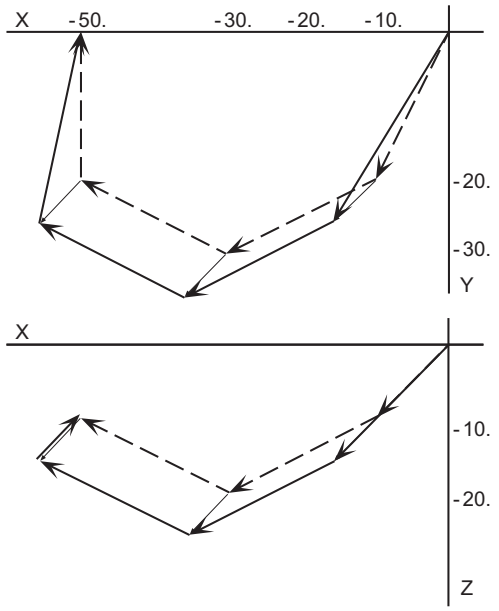
N1(-10.000, -20.000, -10.000)
N1(-15.773, -25.773, -15.773)

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(-50.000, 0.000, -10.000)
N4(-50.000, 0.000, -10.000)

* Upper: Program position
Lower: Program position after compensation



－－－－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.)

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 G90 G53 Y0.;

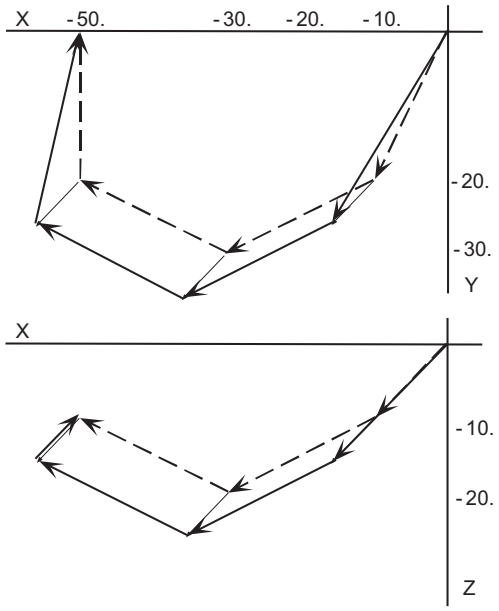
N1(-10.000, -20.000, -10.000)
N1(-15.773, -25.773, -15.773)

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(-50.000, 0.000, -10.000)
N4(-50.000, 0.000, -10.000)

* Upper: Program position
Lower: Program position after compensation



－－－－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).

```
D1=10.

G91 ;
N1 G92 G41 D1 X-10. Y-20. Z-10. I-5. J-5. K-5. ;
N2 X-20. Y-10. Z-10. ;
N3 X-30. Y-10. Z10. ;
N4 Y20. ;

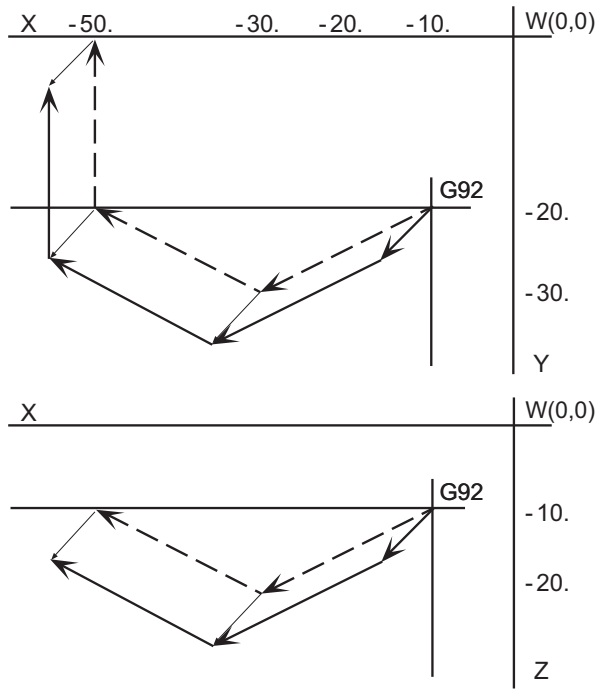
N1( -10.000, -20.000, -10.000 )
N1( -10.000, -20.000, -10.000 )

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( -50.000,   0.000, -10.000 )
N4( -55.773,  -5.773, -15.773 )
```

* Upper: Program position
Lower: Program position after compensation



－－－－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 )

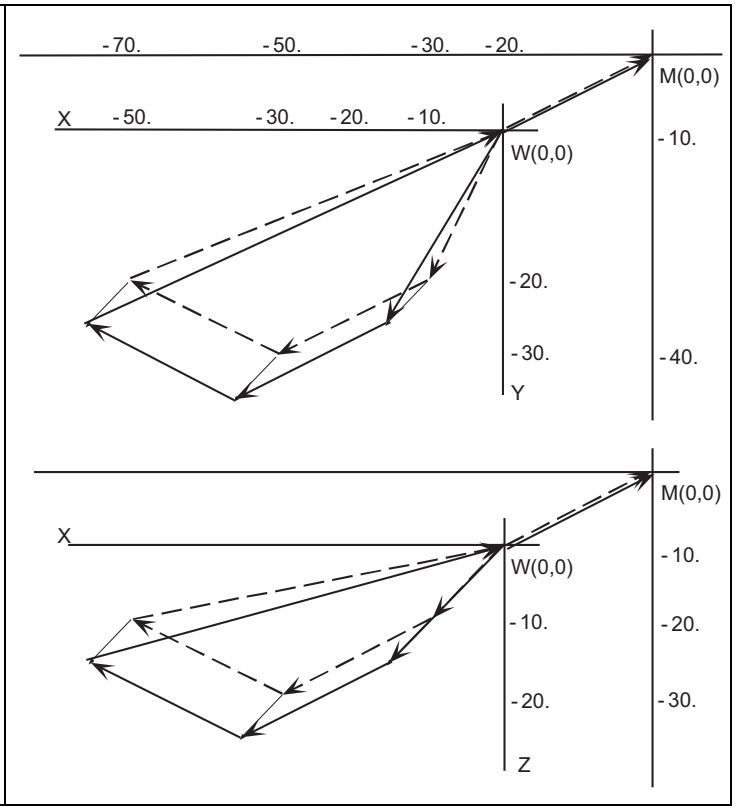
N3( -50.000, -20.000, -10.000 )
N3( -55.773, -25.773, -15.773 )

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 compensation



－－－－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.

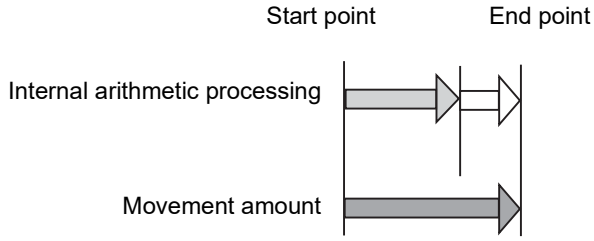
The number of sets for the compensation differ according to machine specification.

D01 to Dn

(The numbers given are the total number of sets for tool length compensation, tool position compensation and tool radius compensation.)

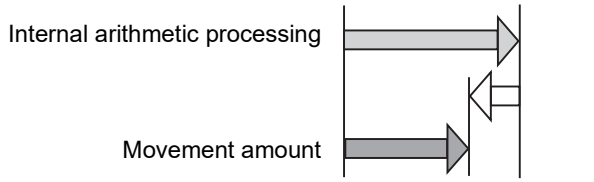
G45 command

Expansion by compensation amount only



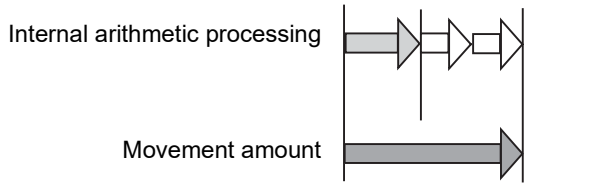
G46 command

Reduction by compensation amount only



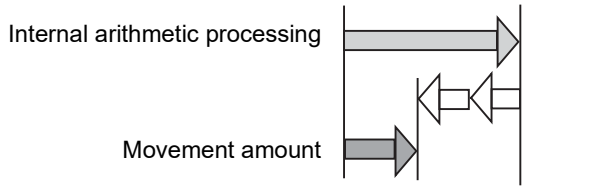
G47 command

2 expansion by compensation amount



G48 command

2 reduction by compensation amount





(Program command value) (compensation amount) (Movement amount after 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 positions are given below.

Command	Movement amount of equivalent command (assigned compensation amount = I)	Example (when X = 1000)
G45 Xx Dd	X(x+I)	I= 10 X= 1010 I= -10 X= 990
G45 X-x Dd	X-(x+I)	I= 10 X= -1010 I= -10 X= -990
G46 Xx Dd	X(x-I)	I= 10 X= 990 I= -10 X= 1010
G46 X-x Dd	X-(x-I)	I= 10 X= -990 I= -10 X= -1010
G47 Xx Dd	X(x+2*I)	I= 10 X= 1020 I= -10 X= 980
G47 X-x Dd	X-(x+2*I)	I= 10 X= -1020 I= -10 X= -980
G48 Xx Dd	X(x-2*I)	I= 10 X= 980 I= -10 X= 1020
G48 X-x Dd	X-(x-2*I)	I= 10 X= -980 I= -10 X= -1020

Tool offset command check

It is possible to make a program error occur when the tool offset No. is "0" or when the offset amount specified with the tool offset No. is "0", to prevent commands from being forgotten.

Note

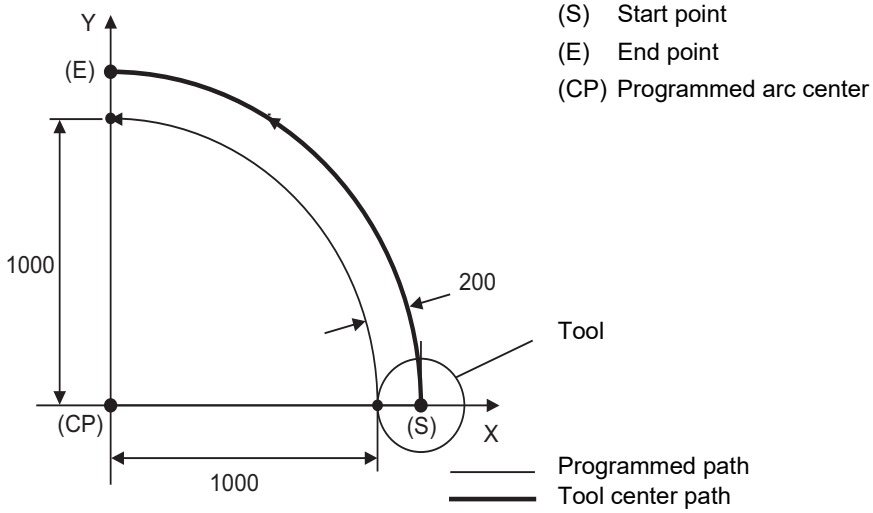
(1) The D address of circle cutting (G12/G13) is not checked by the tool offset command check.

Refer to "12.7.2 Tool Offset Command Check" for details.

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) occurs.

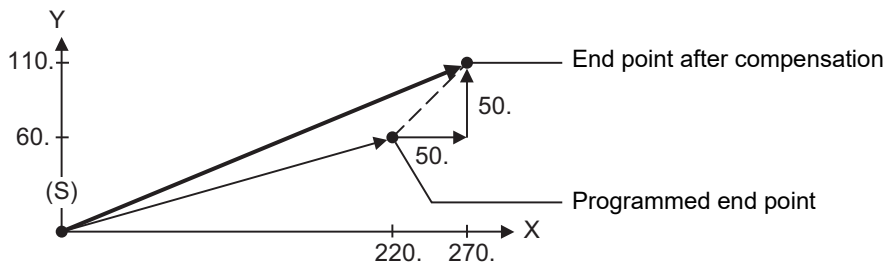
These G codes are unmodal and are effective only in the command block.

Even with an absolute 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 command, compensation can be applied to movement amounts (incremental position) 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. (However, this is within the number of simultaneous contouring control axes.)

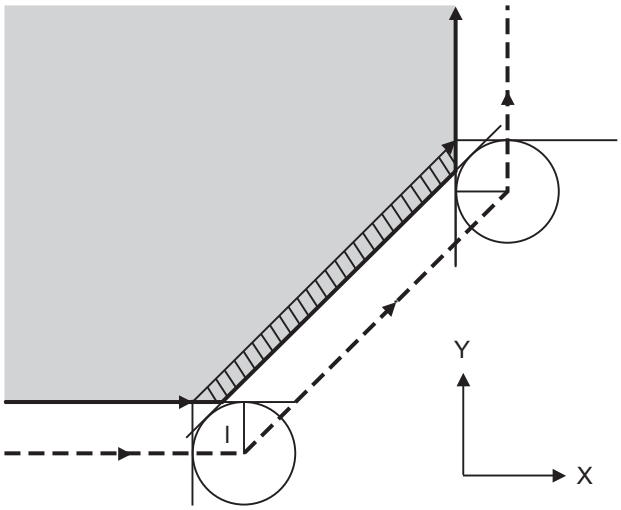
G01 G45 X220. Y60. D20 ;
(D20) = +50. 000



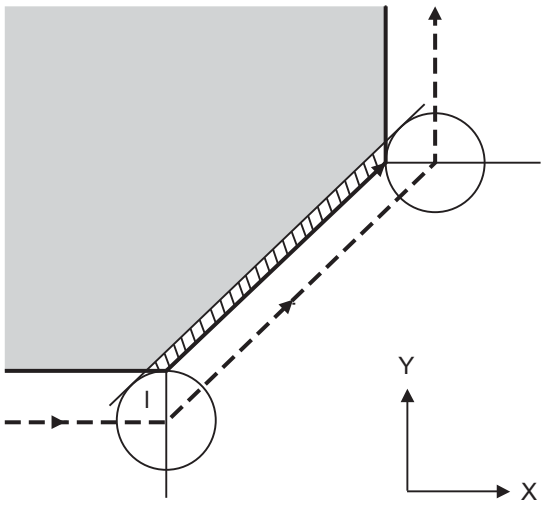
(S) Start point

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 ;

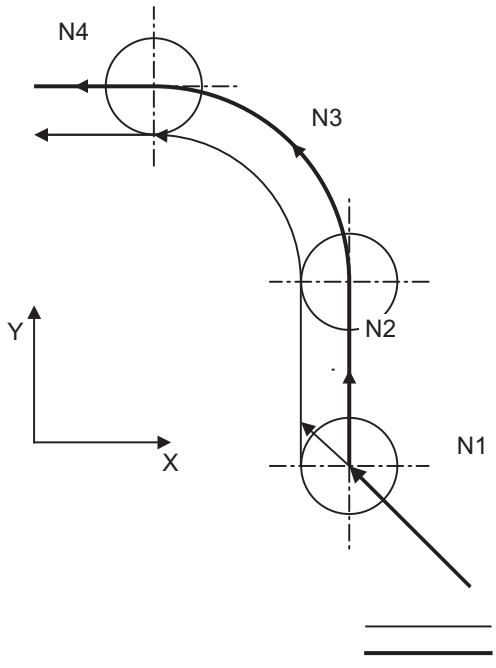
Programmed path Tool center path

Workpiece

Undercutting (figure above) or overcutting (figure below)

I = Compensation amount setting

(Example 2)

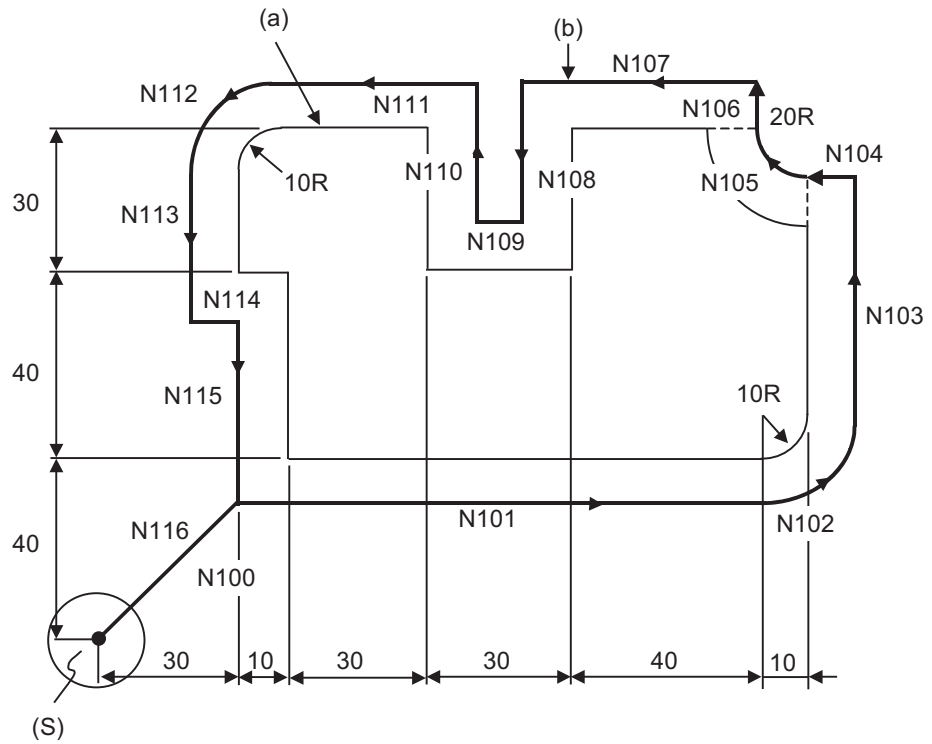


N1 G46 G00 Xx1 Yy1 Dd1 ;
N2 G45 G01 Yy2 Ff2 ;
N3 G45 G03 Xx3 Yy3 Ii3 ;
N4 G01 Xx4 ;

Program path
Tool center path

(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

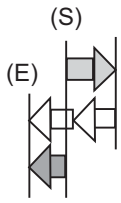
Compensation amount D01 = 10.000mm (Compensation amount of tool radius)

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.



Program command G48 X20.000 D01 ;

Compensation Compensation amount = +15.000

Tool movement Actual movement = X - 10.000

(S) Start point

(E) End point

(3) When a zero movement amount has been specified in the incremental 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 command, the operation is completed immediately and the tool does not move for the compensation amount.

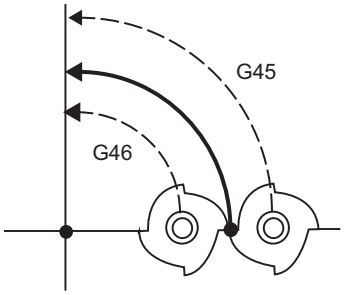
(4) In the case of circular interpolation, tool radius compensation is possible by the G45 to G48 commands only for one quadrant, two quadrants (semi sphere) or three quadrants when the start and end points are on the axis.

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.

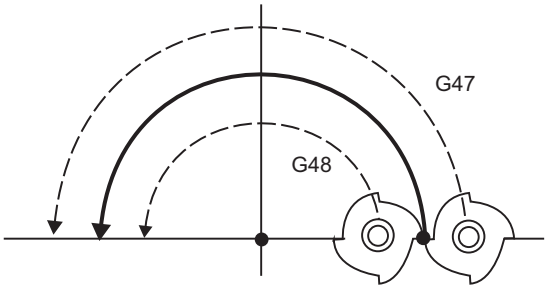
[1/4 circular arc]



G45 for compensation outside the circle
G46 for compensation inside the circle

Program path
Compensated path

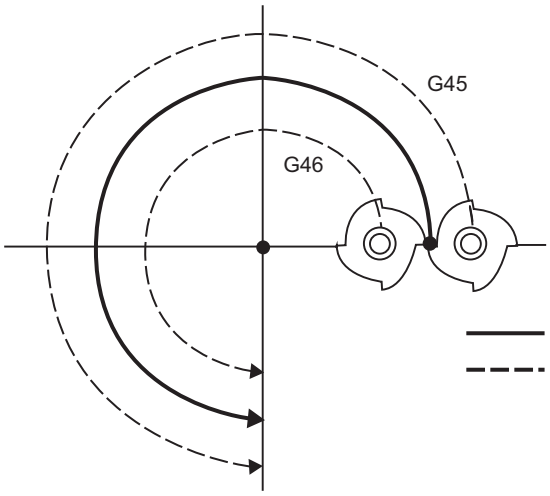
[1/2 circular arc]



G47 for compensation outside the circle
G48 for compensation inside the circle

Program path
Compensated path

[3/4 circular arc]



G45 for compensation outside the circle
G46 for compensation inside the circle

Program path
Compensated path

12.7 Tool Offset Number

12.7.1 H Stand-alone Command

When the parameter "#1274 ext10/bit3" is "1", the H modal can be updated even if the address H of the tool compensation function is commanded alone. (This setting depends on the MTB specifications.)

#1274 ext10/bit3	H stand-alone command valid/invalid
0	Invalid
1	Valid

The target tool length offset commands are as shown in the table below.

G code	Function
G43	Tool length offset (+)
G44	Tool length offset (-)
G43.1	Tool length compensation along the tool axis
G43.4	Tool center point control type 1
G43.5	Tool center point control type 2
G43.8	Tool cutting point control type 1
G43.9	Tool cutting point control type 2

(1) The offset No. commanded with the H address will be valid for the subsequent modals.
Below are examples of the tool length offset (G43). For other tool compensations, read the G code and compensation number differently.

(Example 1)

G49 ; :	Tool length offset is canceled.
Hh1 ; :	Offset number "h1" is set to H modal. (Tool length offset is not performed.)
G43 Zz1 ; :	Tool length offset is executed with "h1".
Zz2 Hh2 ; :	The axis moves by "z2 + (lh2 - lh1)". (The H stand-alone command in G43 is valid.)
G49 ; :	Tool length offset is canceled. (H modal is maintained.)
G43 Zz3 ; :	Tool length offset is re-executed with "h2".
H0 ; :	The tool length offset amount is set to "0" to conduct offset.
Hh1 ;	Tool length offset is executed with "h1".

(Example 2) If "#1080 Dril_Z" is "0", offset is applied to the axis commanded in the same block as G43.

G49 ; :	Tool length offset is canceled.
Hh2 ; :	Offset number "h1" is set to H modal. (Tool length offset is not performed.)
G43 Xx1 ; :	Tool length offset is performed on the X axis with "h2".
G43 Yy1 ; :	Tool length offset is performed on the Y axis with "h2".
G43 Zz1 ;	Tool length offset is performed on the Z axis with "h2".

(2) For the tool length offset, the H stand-alone command in G43 modal applies to the axis last compensated with G43.

(3) For the tool length offset, the cancellation with the "H0" command applies to the axis last compensated with G43, not all axes.

(4) If the cancellation is performed with the "H0" command, the modal of G code group 8 is maintained.

(5) H stand-alone command is disabled in the following cases.

- Program parameter input (G10, G11) using address H is in progress.
- There is a command (M96, M98) that uses address H as the sequence number.
- This is a G code block that uses address H. (For the target G codes, refer to the table below.)

G code	Function
G02/G03	Circular interpolation (Geometric IB)
G114.1	Spindle synchronization I
G114.2	Tool spindle synchronization IA
G51.2	Tool spindle synchronization IB
G114.3	Tool spindle synchronization II
G122	Calling sub part system

- A G code (G04, G52, G07) that invalidates address H and address D is commanded.
- A special fixed cycle (G34, G35, G36, G37.1) that invalidates address H is in progress.
- This is a circular cutting (G12, G13) or automatic tool length measurement (G37) block that specifies address H as the program error (P32).
- The program format is being switched from the lathe system to the machining center system by the lathe-system program format switch function (G188).

(6) The operation of the tool length offset depends on whether the axis movement or the coordinate shift is selected as the compensation operation.

<Note>

If the following parameters are all set to "1", the MCP alarm (Y51 108) occurs, thereby, disabling the automatic operation from being started. These settings depend on the MTB specifications.

- "#1247 set19/bit0" (Movement by tool length compensation command)
- "#1274 ext10/bit3" (Tool offset number H stand-alone command (for M system only))

12.7.2 Tool Offset Command Check

It is possible to make a program error occur when the tool offset No. is "0" or when the offset amount specified with the tool offset No. is "0", to prevent commands from being forgotten.

Target function	G code
Tool length offset	G43, G44, G43.1, G43.4, G43.5, G43.8, G43.9
Tool radius compensation	G41, G42, G41.2, G42.2
Tool position offset	G45, G46, G47, G48
Tool position compensation	G43.7

Which cases are considered program errors depends on the MTB specifications (parameter "#1381 TolOfsCmd-Check_M").

Below are examples of the tool length offset (G43). For other tool compensations, read the G code and compensation number differently.

[Check when the offset number is "0"] (#1381/bit0 = 1)

If the offset number is "0" when a tool compensation command is issued, the program error (P170) occurs.

(Example 1)

H0 ;	Offset number "0" The tool length offset is being canceled, so the program error (P170) does not occur.
G43 ;	When a tool length offset command is issued, the program error (P170) occurs.

(Example 2)

G43 H0 ; :	If a tool length offset command and "H0" are commanded in the same block, the program error (P170) occurs.

(Example 3)

G43 Hh1 ;	Tool length offset is performed with offset number "h1".
H0 ; :	The program error (P170) occurs.

[Check when the offset amount is "0"] (#1381/bit1 = 1)

If the offset amount is "0" when a tool compensation command is issued, the program error (P169) occurs.
For tool position compensation, an error occurs if the tool offset amounts of the three basic axes are all "0".

(Example 1)

Hh1 ;	The tool offset amount (lh1) corresponding to the offset number (h1) is "0", however, the tool length offset is being canceled, the program error (P169) does not occur here.
G43 ; :	When a tool length offset command is issued, the program error (P169) occurs.

(Example 2)

G43 ;	Tool length offset command
Hh1 ; :	The tool offset amount (lh1) corresponding to the offset number (h1) is "0", so the program error (P169) occurs.

(Example 3)

G43 Hh1 ; :	The tool offset amount (lh1) corresponding to the offset number (h1) is "0", so the program error (P169) occurs.

13

Fixed Cycle

This chapter explains fixed cycles.

⚠ CAUTION

⃠ Do not change the fixed cycle program without prior approval from the MTB.

[Precautions for MTB]

- The G.1 command is used to realize the fixed cycle operation. Therefore, do not edit the blocks before the G.1 command such as deleting or inserting another command before the G.1 command. If you edit the program in such a manner, the fixed cycle program is not performed properly.
- The fixed cycle program uses modal information of group 01 dedicated for the fixed cycle program so that the modal of the main program that called the fixed cycle program is not affected. Therefore, there is no need to edit the fixed cycle program based on the modal when returning to the main program. In the fixed cycle program, the modal of group 01 is set to G00 after the G.1 command is executed.

13.1 Standard Fixed Cycle



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	Chip removal	Cutting reduction amount specification	Application
		Dwell	Spindle					
G80	-	-	-	-	-	-	-	Cancel
G81	Cutting feed	-	-	Rapid traverse	Possible	Possible	-	Drill, spot drilling cycle
G82	Cutting feed	Yes	-	Rapid traverse	-	Possible	-	Drill, counter boring cycle
G83	Intermittent feed	-	-	Rapid traverse	Possible	Possible	Possible	Deep hole drilling cycle
G84	Cutting feed	Yes	Reverse rotation	Cutting feed	-	-	-	Tapping cycle
G84	Cutting feed	Yes	Reverse rotation	Cutting feed	-	-	Possible	Deep-hole tapping cycle Pecking 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	Possible	Possible	Stepping cycle
G74	Cutting feed	Yes	Forward rotation	Cutting feed	-	-	-	Reverse tapping cycle

G code	Hole drilling start (-Z direction)	Operation at hole bottom		Return operation (+Z direction)	Retract at high speed	Chip removal	Cutting reduction amount specification	Application
		Dwell	Spindle					
G74	Cutting feed	Yes	Forward rotation	Cutting feed	-	-	Possible	Deep-hole reverse tapping cycle Pecking 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
G84.5 G84.6 G84.8	Cutting feed	Yes	Reverse rotation	Cutting feed	-	-	-	Punchtap cycle
G74.5 G74.6 G74.8	Cutting feed	Yes	Forward rotation	Cutting feed	-	-	-	Reverse Punchtap cycle

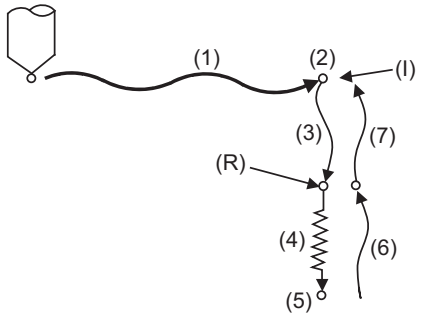
A fixed cycle mode (G group 09) can be canceled by G80 command and other hole machining modes or G command in the 01 group. At the same time, various other data are also set to "0".

Only for fixed cycle where the cutting amount per time is specified and the workpiece is cut multiple times, the cutting reduction amount specification method is available. It is possible to perform more efficient machining than when the cutting amount is constant. For details of the operation and enabling conditions, refer to "13.1.22 Cutting Reduction Amount Specification Method".

Detailed description

Basic operations of fixed cycle for drilling

There are 7 actual operations which are each described below.



(I) Initial point
(R) R point return

(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.

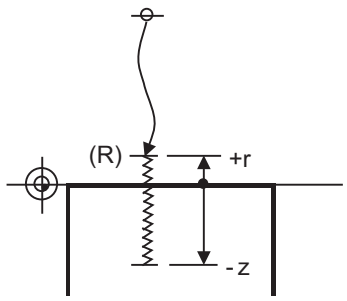
(7) The tool is returned to the initial point at rapid traverse rate.

Note

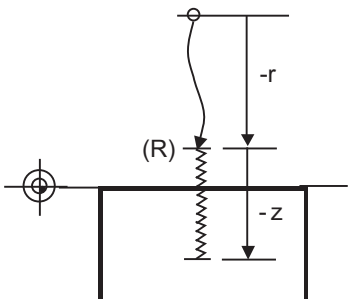
(1) Whether the fixed cycle is to be completed at operation 6 or 7 can be selected by G98/G99 commands. (Refer to "Initial point and R point level return; G98, G99")

Difference between absolute command and incremental command

For absolute command



For incremental command



(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 positions, 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;

:

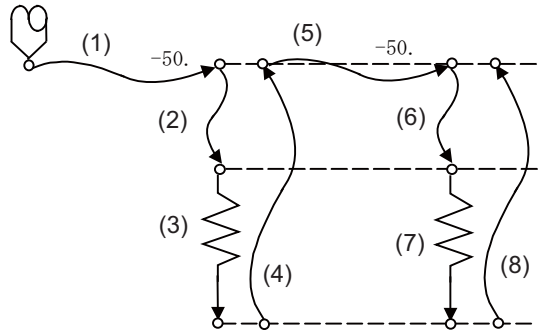


Fig. 1 Operation when number of repetitions L is designated

Operation pattern	,I	,J
(1)	Valid	-
(2)	-	Invalid
(3)	-	Invalid
(4)	-	Valid
(5)	Valid	-
(6)	-	Invalid
(7)	-	Invalid
(8)	-	Valid

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)

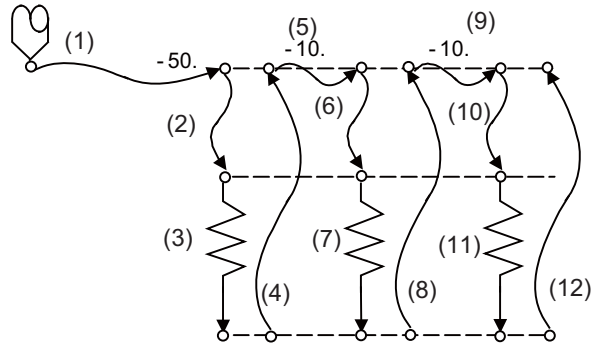


Fig. 2 Operation in fixed cycle modal

Operation pattern	,I	,J
(1)	Valid	-
(2)	-	Invalid
(3)	-	Invalid
(4)	-	Valid
(5)	Invalid	-
(6)	-	Invalid
(7)	-	Invalid
(8)	-	Valid
(9)	Valid	-
(10)	-	Invalid
(11)	-	Invalid
(12)	-	Valid

13.1.1 Drilling, Spot Drilling; G81

Command format

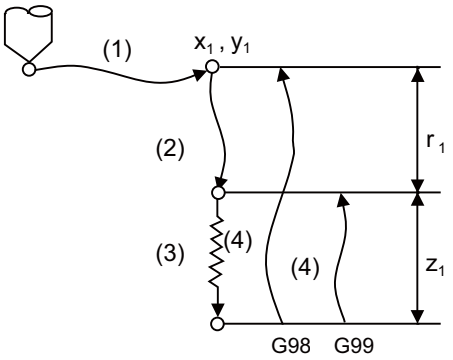
G81 Xx1 Yy1 Zz1 Rr1 Ff1 Ll1,Ii1,Jj1 Dd1 Ee1;

Xx1	Designation of hole drilling position (absolute/incremental position)
Yy1	Designation of hole drilling position (absolute/incremental position)
Zz1	Designation of hole bottom position (absolute/incremental position) (modal)
Rr1	Designation of R point position (absolute/incremental position) (modal)
Ff1	Designation of feedrate for cutting feed (modal)
Ll1	Designation of number of repetitions. (0 to 9999) When ″0″ is set, no execution.
,Ii1	Positioning axis in-position width
,Jj1	Drilling axis in-position width
Dd1	Designation of spindle number for which chip removal is made (*1)(*2)
Ee1	Designation of frequency in which chip removal is made (*2)

(*1) There are two types of spindle designation methods: spindle number method and spindle name method, which vary depending on the MTB specifications. When the name (1 to 9) has been specified for all the spindles with the parameter "#3077 Sname" (Spindle command name), a command is given with the spindle name. For others, command with the spindle number.

(*2) For details, refer to "13.1.21 Chip Removal".

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

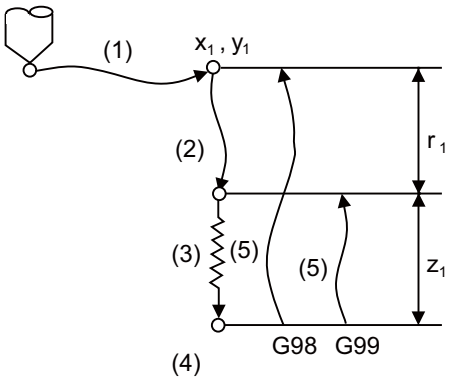
G82 Xx1 Yy1 Zz1 Rr1 Ff1 Pp1 Ll1 ,Ii1 ,Jj1 Dd1 Ee1;

Xx1	Designation of hole drilling position (absolute/incremental position)
Yy1	Designation of hole drilling position (absolute/incremental position)
Zz1	Designation of hole bottom position (absolute/incremental position) (modal)
Rr1	Designation of R point position (absolute/incremental position) (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.
,Ii1	Positioning axis in-position width
,Jj1	Drilling axis in-position width
Dd1	Designation of spindle number for which chip removal is made (*1)(*2)
Ee1	Designation of frequency in which chip removal is made (*2)

(*1) There are two types of spindle designation methods: spindle number method and spindle name method, which vary depending on the MTB specifications. When the name (1 to 9) has been specified for all the spindles with the parameter "#3077 Sname" (Spindle command name), a command is given with the spindle name. For others, command with the spindle number.

(*2) For details, refer to "13.1.21 Chip Removal".

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 Ll1 ,Ii1 ,Jj1 Dd1 Ee1 Jj2 ,Kk1;

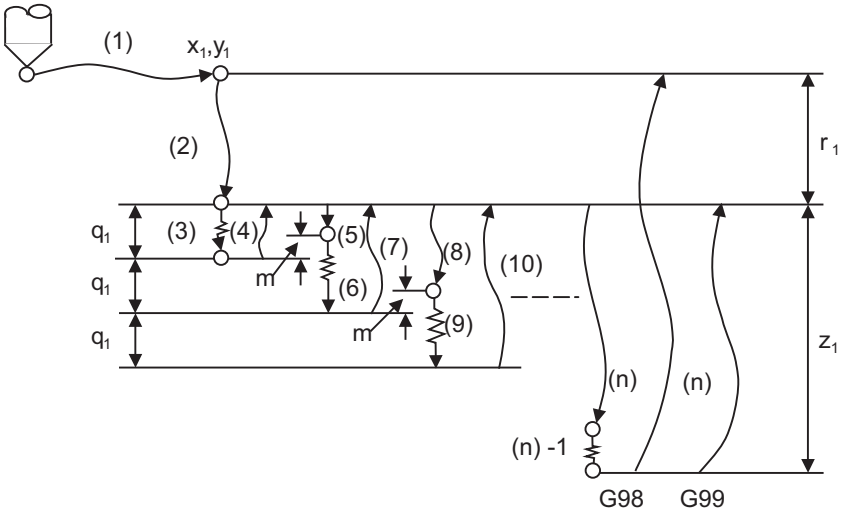
Xx1	Designation of hole drilling position (absolute/incremental position)
Yy1	Designation of hole drilling position (absolute/incremental position)
Zz1	Designation of hole bottom position (absolute/incremental position) (modal)
Rr1	Designation of R point position (absolute/incremental position) (modal)
Qq1	Cut amount for each cutting pass (incremental position) (modal)
Ff1	Designation of feedrate for cutting feed (modal)
Ll1	Designation of number of repetitions. (0 to 9999) When ″0″ is set, no execution.
,Ii1	Positioning axis in-position width
,Jj1	Drilling axis in-position width
Dd1	Designation of spindle number for which chip removal is made (*1)(*2)
Ee1	Designation of frequency in which chip removal is made (*2)
Jj2	Cutting reduction amount (incremental position) (*3) When omitted, the mode will follow the setting of parameter (#19444).
,Kk1	Minimum cutting amount (incremental position) (*3) When omitted, the mode will follow the setting of parameter (#19445).

(*1) There are two types of spindle designation methods: spindle number method and spindle name method, which vary depending on the MTB specifications. When the name (1 to 9) has been specified for all the spindles with the parameter "#3077 Sname" (Spindle command name), a command is given with the spindle name. For others, command with the spindle number.

(*2) For details, refer to "13.1.21 Chip Removal".

(*3) For details, refer to "13.1.22 Cutting Reduction Amount Specification Method".

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.

When the cutting reduction amount "j2" or the minimum cutting amount "k1" is specified, the cutting reduction amount specification method is used. Drilling is performed while the cut amount for each cutting pass "q1" is gradually reduced. For details of the operation, refer to "13.1.22 Cutting Reduction Amount Specification Method".

13.1.3.2 Small Diameter Deep Hole Drilling Cycle



Command format

G83 Xx1 Yy1 Zz1 Rr1 Qq1 Ff1 Ii1 Pp1 Dd1 Ee1 Jj2 ,Kk1;

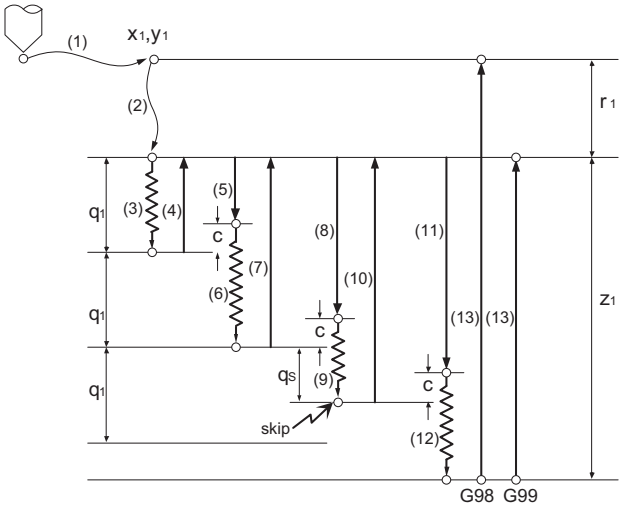
Xx1	Designation of hole drilling position (absolute/incremental position)
Yy1	Designation of hole drilling position (absolute/incremental position)
Zz1	Designation of hole bottom position (absolute/incremental position) (modal)
Rr1	Designation of R point position (absolute/incremental position) (modal)
Qq1	Cut amount for each cutting pass (incremental position) (modal)
Ff1	Designation of feedrate for cutting feed (modal)
Ii1	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
Dd1	Designation of spindle number for which chip removal is made (*1)(*2)
Ee1	Designation of frequency in which chip removal is made (*2)
Jj2	Cutting reduction amount (incremental position) (*3) When omitted, the mode will follow the setting of parameter (#19444).
,Kk1	Minimum cutting amount (incremental position) (*3) When omitted, the mode will follow the setting of parameter (#19445).

(*1) There are two types of spindle designation methods: spindle number method and spindle name method, which vary depending on the MTB specifications. When the name (1 to 9) has been specified for all the spindles with the parameter "#3077 Sname" (Spindle command name), a command is given with the spindle name. For others, command with the spindle number.

(*2) For details, refer to "13.1.21 Chip Removal".

(*3) For details, refer to "13.1.22 Cutting Reduction Amount Specification Method".

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,Ii1;	
(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 ,Ii1 ,Jj1 Ll1 (Kk1) Jj2 ,Kk2;

Xx1	Designation of hole drilling position (absolute/incremental position)
Yy1	Designation of hole drilling position (absolute/incremental position)
Zz1	Designation of hole bottom position (absolute/incremental position) (modal)
Rr1	Designation of R point position (absolute/incremental position) (modal)
Qq1	Cut amount for each cutting pass (incremental position) (modal)
Ff1	During synchronous tapping: Designation of drilling axis feed amount (tapping pitch) per spindle revolution (modal) Also refer to "Speed command of synchronous tapping". 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
,Ii1	Positioning axis in-position width
,Jj1	Drilling axis in-position width
Ll1	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")
Jj2	Cutting reduction amount (incremental position) (*1) When omitted, the mode will follow the setting of parameter (#19444).
,Kk2	Minimum cutting amount (incremental position) (*1) When omitted, the mode will follow the setting of parameter (#19445).

(*1) For details, refer to "13.1.22 Cutting Reduction Amount Specification Method".

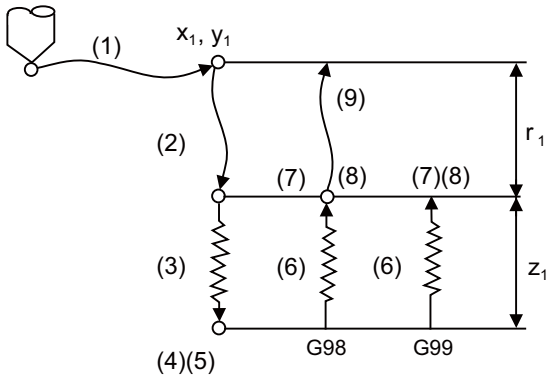
Note

(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.
When ",S" command is omitted or when ",S0" is commanded, the value of "spindle rotation speed at return" is obtained by the following formula.

((S command value) × (setting value of the parameter "#1172 tapovr")) / 100

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 reverse 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.

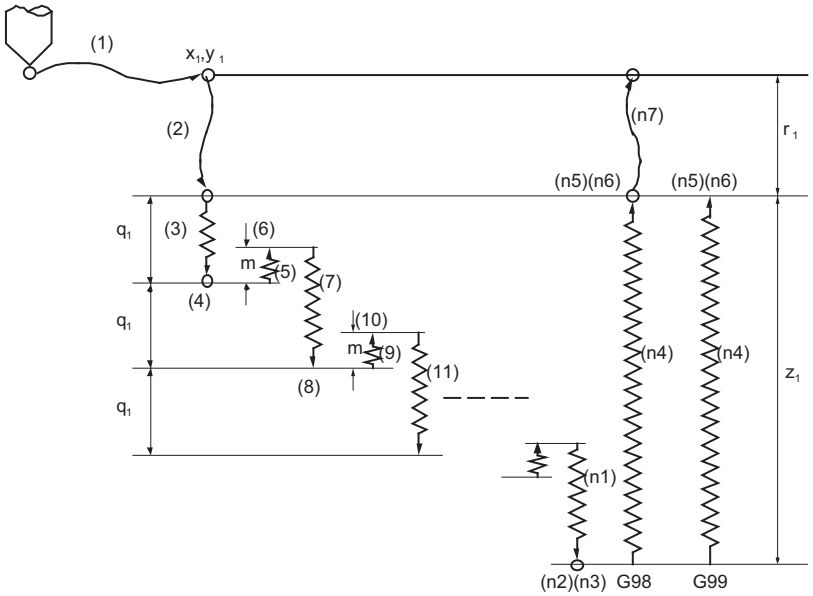
The operation stops at after the (1), (2) and (9) commands during single block operation.

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 ,Ii1;	
(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

(a) The load applied to the tool can be reduced by designating the depth of cut per pass and cutting the work-piece to the hole bottom for a multiple number of passes.

(b) Whether the pecking tapping cycle or the deep-hole tapping cycle is valid depends on the MTB specifications (parameter "#1272 ext08/bit4").
When the depth of cut per pass (address "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 "0".

(c) When the cutting reduction amount "j2" or the minimum cutting amount "k1" is specified, the cutting reduction amount specification method is used. Drilling is performed while the cut amount for each cutting pass "q1" is gradually reduced. For details of the operation, refer to "13.1.22 Cutting Reduction Amount Specification Method".

(d) When G84 is executed, the override will be canceled and the override will automatically be set to 100% in the cutting operation. During a pulling operation from the hole bottom to the R point, the override set in the parameter "#1172 tapovr" will be enabled. Note, however, that it will be disabled during a return operation by the return amount "m".

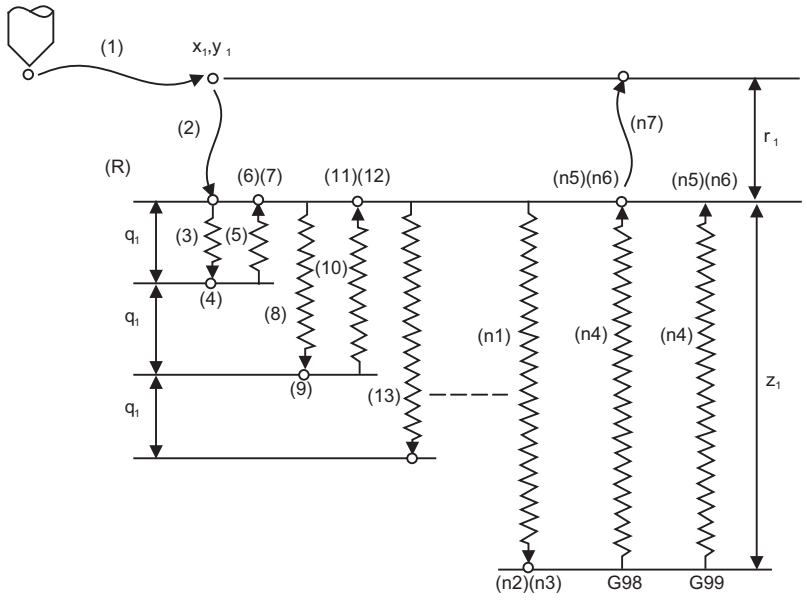
(e) The amount retracted from the hole bottom is set to the parameter "#8018 G84/G74 n".

(f) Dry run is valid for a positioning command when the parameter "#1085 G00 DRY RUN" is "1". 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.

(g) During single block operation, the tool does not stop during cutting or returning, but stops after completing an R point/initial point return.

(h) During the G84 mode, the NC signal "Tapping" will be output.

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

Operation pattern	Program	
(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;	
(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 the cutting reduction amount "j2" or the minimum cutting amount "k1" is specified, the cutting reduction amount specification method is used. Drilling is performed while the cut amount for each cutting pass "q1" is gradually reduced. For details of the operation, refer to "13.1.22 Cutting Reduction Amount Specification Method".

(d) 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)

(e) 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.

(f) During single block operation, the tool does not stop during cutting or returning, but stops after completing an R point/initial point return.

(g) During the G84 mode, the NC signal "Tapping" will be output.

(h) During the G84 synchronous tapping modal, the M3, M4, M5 and S code will not be output.

(i) If the command value of F becomes extremely small such as around "F < 0.01 mm/rev" during synchronous tapping, the spindle does not rotate smoothly. So make sure to command a value larger than "0.01 mm/rev". The unit of F can be selected between mm/rev and mm/min.

(j) 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.

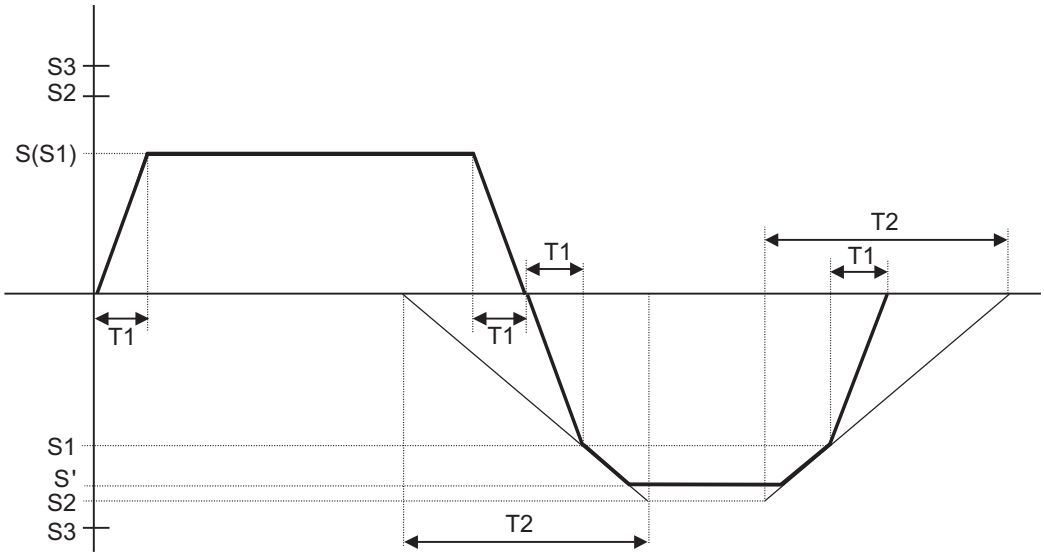
(k) If the operation is interrupted by a cause such as an emergency stop or reset during the deep-hole tapping cycle, a tap retract is executed when the tap retract signal is input.

(l) 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

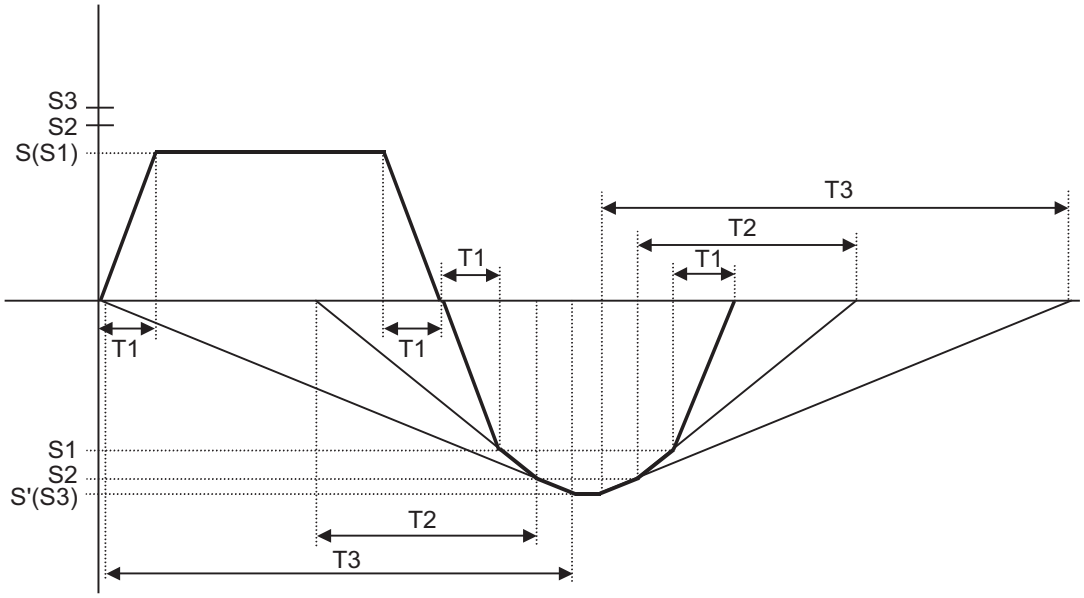
This function enables to make spindle acceleration/deceleration pattern closer to that of the speed loop by dividing the spindle and drilling axis acceleration/deceleration pattern into up to three stages during synchronous tapping.
The acceleration/deceleration pattern can be set up to three stages 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 No designation	OFF	
	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	
		F0 to F8 (no decimal point)	F1-digit Feed
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 taping 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".

Speed command of synchronous tapping

(1) Feed per minute command of the synchronous tapping

Whether the F command at synchronous tapping is feed per minute or feed per revolution depends on the setting values of the parameters "#1292 ext28/bit1" and "#1268 ext04/bit2" and G code group 5 modal (G94/G95). (The number of significant digits differs when there is no decimal point.) These parameters setting depend on the MTB specifications.

#1292/bit1	#1268/bit2	During G94 modal (feed per minute)	During G95 modal (feed per revolution)
0	0	Feed per revolution (*1)	Feed per revolution (*1)
	1	Feed per minute (*2)	Feed per revolution (*1)
1 (*3)	- (*4)	Feed per minute (*2)	Feed per revolution (*1)

(*1) The F command is set to feed per revolution (mm/rev, inch/rev).

(*2) The F command is set to feed per minute (mm/min, inch/min).

Pitch = F command value / S command value

(*3) When the parameter "#1292 ext28/bit1" is set to "1", the F command of synchronous tapping is regarded as an F modal value of cutting feed.
The F modal value updated by the F command of synchronous tapping differs depending on the mode (feed per minute/feed per revolution) when G84 is executed.

(*4) The setting value of the parameter "#1268 ext04/bit2" does not affect feed per minute/feed per revolution of synchronous tapping.

Note

- The G94 command, which is a modal command, is valid until the G95 (feed per revolution) command is issued next.
- If the E address (number of screw threads per inch) is issued while feed per minute is valid, the program error (P32) occurs.
- If an F 1-digit feed command is issued to program the feedrate, F 1-digit feed is disabled.
- When the F command is not in the G84 command, the F command modal value is as shown below.

#1292/bit1	Synchronous tapping speed
0	The F command modal value of synchronous tapping is enabled. When there is no modal value, the program error (P183) occurs.
1	The F command modal value of cutting feed is enabled. When there is no modal value, the program error (P62) occurs.

- When the parameter "#1268 ext04/bit2" is set to "1", if the G code group 5 modal (G94/G95) is switched during synchronous tapping, the program error (P184) occurs.

(2) Change of F command unit at feed per revolution

The F command unit is as follows depending on the MTB specifications. (Parameter "#1271 ext07/bit2")

#1271/bit2	G code group 5 modal	Unit when "F1" is commanded	
		Metric input	Inch input
0	G94	F1 (= 1.00 [mm/rev]) (F1. = 1.00 [mm/rev])	F1 (= 1.000 [inch/rev]) (F1. = 1.000 [inch/rev])
	G95	F1 (= 1.00 [mm/rev]) (F1. = 1.00 [mm/rev])	F1 (= 1.000 [inch/rev]) (F1. = 1.000 [inch/rev])
1	G94	F1 (= 1.00 [mm/min]) (F1. = 1.00 [mm/min])	F1 (= 0.010 [inch/min]) (F1. = 1.000 [inch/min])
	G95	F1 (= 0.0001 [mm/rev]) (F1. = 1.0000 [mm/rev])	F1 (= 0.000001 [inch/rev]) (F1. = 1.00000 [inch/rev])

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)

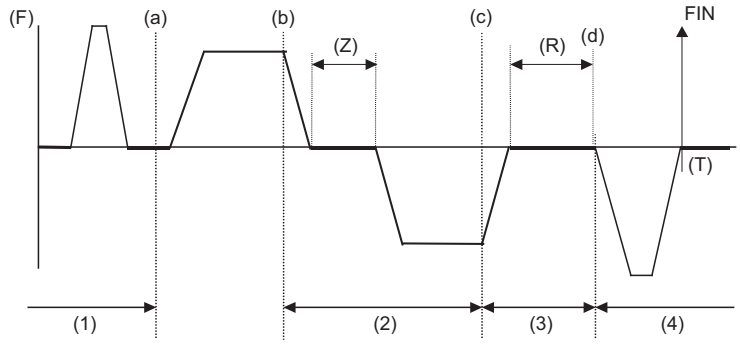
#1223 aux07				"P" designation of G84/ G74 command	In-position check during synchronous tapping		
bit3	bit4	bit5	bit2				
Synchronous tap in-position check improvement	Hole bottom	R point	I point -> R point		Hole bottom	R point	I point -> R point
0	-	-	-	-	yes	yes	yes
1	-	-	-	No "P" designation Example: 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				
Synchronous tap in-position check improvement	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.

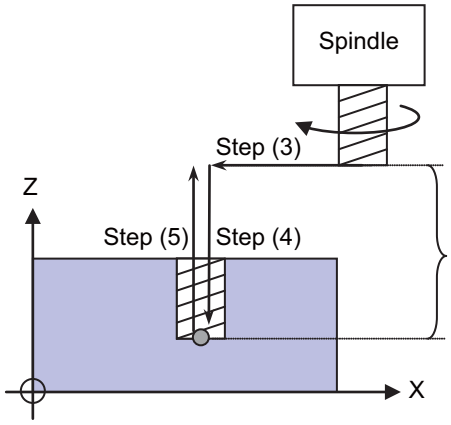
[Operation procedure]

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.



Depends on the parameter (#11030) setting.
OFF: Synchronizes with the spindle (tapping).
ON: Does not synchronize with the spindle (positioning).

[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.)

Synchronous tapping with analog I/F spindle (analog spindle synchronous tapping)

This function performs tapping using the analog-connected spindle. By using this function, various tapping cycle functions 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 combined 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 the execution of the synchronous tapping with analog I/F spindle, the tap retract can be used as well as the normal synchronous tapping. However, when the emergency stop is canceled before commanding the tapping retract, the analog I/F spindle needs to be ready to rotate by a voltage command from the NC.

[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/bit6" (Analog spindle synchronous tapping ON) is set to "0", 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.

Synchronous tapping by spindle control with pulse train output

This function performs tapping using the spindle controlled with pulse train output. By using this function, various tapping cycle functions can be used using the spindle connected to the inverter, etc.

When this function is used, the inverter, etc. which supports the function to control the spindle operation with the pulse train input needs to be connected. Also, the parameter "#1295 ext31/bit6" (Analog spindle synchronous tapping ON) needs to be set to "1".

[Relationship with other functions]

(1) The following functions cannot be combined with the synchronous tapping using the pulse-train output spindle. If the synchronous tapping using the pulse-train output spindle is commanded while the following function is used, the program error (P182) occurs.
♦Coordinate rotation by program

(2) When the reset or emergency stop is performed during the execution of the synchronous tapping with pulse train output, the tap retract can be used as well as the normal synchronous tapping. However, when the emergency stop is canceled before commanding the tapping retract, the connected inverter spindle needs to be ready to rotate by the pulse train output from the NC.

(3) Spindle selection can be performed for the pulse-train output spindle by multiple-spindle control. However, to use encoder input, set the pulse-train output spindle to the 1st spindle.

[Precautions]

(1) The pecking tapping cycle or deep-hole tapping cycle cannot be commanded for the synchronous tapping using the pulse-train output spindle. If commanded, the program error (P182) occurs.

(2) The synchronous tap with multi-step acceleration deceleration cannot be used for the synchronous tapping using the pulse-train output spindle. Regardless of the value of the parameter "#1223 aux07/bit7" (Synchronous tap method), the acceleration/deceleration operation for synchronous tapping is performed in the conventional method.

(3) The high-speed synchronous tapping cannot be used for the synchronous tapping using the pulse-train output spindle. 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 pulse-train output spindle in multiple-spindle control mode when the parameter "#1295 ext31/bit6" (synchronous tapping with analog I/F spindle ON) is set to "0", 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 synchronous tapping error display function is disabled for the synchronous tapping using the pulse-train output spindle, and the synchronous tapping error display always shows "0".

(6) If the reset or emergency stop is performed during the synchronous tapping using the pulse-train output spindle, the controller stops outputting pulse signals to the spindle.

(7) For the spindle control with pulse train output, a different acceleration/deceleration control method is used at spindle rotation command and synchronous tapping, compared to the normal spindle control. Therefore, to operate a program that transitions from the spindle rotation command to the synchronous tapping command, wait until the spindle has sufficiently slowed down, and issue the synchronous tapping command.

(8) If reset is performed during forward rotation or reverse rotation of pulse-train output spindle, the pulse train will not stop being output, and the spindle remains rotated. However, if an emergency stop is performed during forward rotation or reverse rotation, the controller stops outputting pulse signals to the spindle.

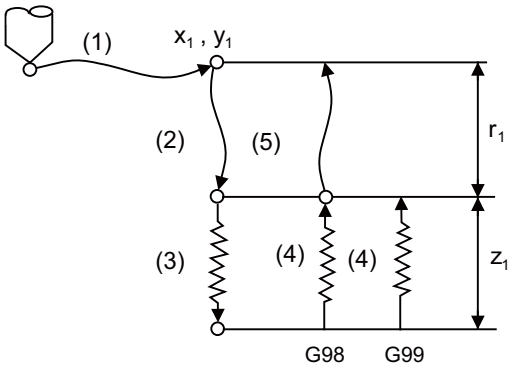
13.1.5 Boring; G85

Command format

G85 Xx1 Yy1 Zz1 Rr1 Ff1 Ll1 ,Ii1 ,Jj1;

Xx1	Designation of hole drilling position (absolute/incremental position)
Yy1	Designation of hole drilling position (absolute/incremental position)
Zz1	Designation of hole bottom position (absolute/incremental position) (modal)
Rr1	Designation of R point position (absolute/incremental position) (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.
,Ii1	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.

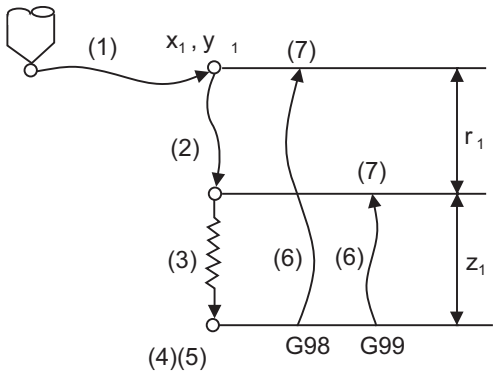
13.1.6 Boring; G86

Command format

G86 Xx1 Yy1 Zz1 Rr1 Ff1 Pp1 Ll1 ;

Xx1	Designation of hole drilling position (absolute/incremental position)
Yy1	Designation of hole drilling position (absolute/incremental position)
Zz1	Designation of hole bottom position (absolute/incremental position) (modal)
Rr1	Designation of R point position (absolute/incremental position) (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 Ll1;

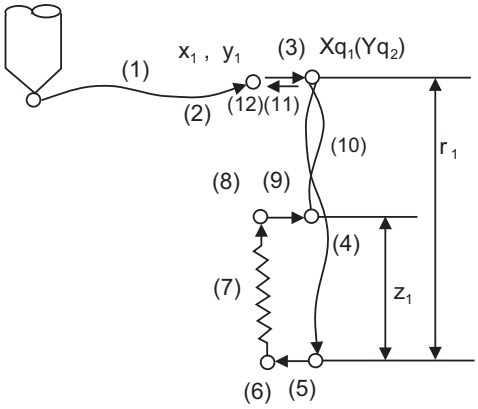
Xx1	Designation of hole drilling position (absolute/incremental position)
Yy1	Designation of hole drilling position (absolute/incremental position)
Zz1	Designation of hole bottom position (absolute/incremental position) (modal)
Rr1	Designation of R point position (absolute/incremental position) (modal)
Iq1 Jq2 Kq3 (*1)	Designation of shift amount (incremental position) (modal) 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)".
Ff1	Designation of feedrate for cutting feed (modal)
Ll1 (*1)	Designation of number of repetitions (0 to 9999) Not executed when "0" is set.

(*1) Depending on the parameter setting, the number of repetitions can be designated by K address. ("#1268 ext04/bit4", "#1271 ext04/bit1")

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

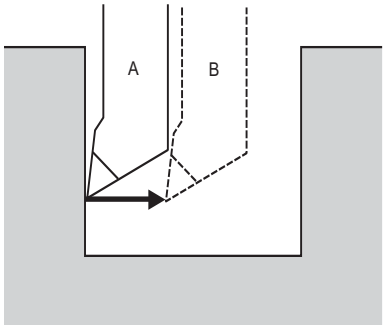


Operation pattern	Program	
(1)	G00 Xx1 Yy1;	
(2)	M19 ;	Spindle orientation
(3)	G00 Xq1 (Yq2) ;	Shift
(4)	G00 Zr1;	
(5)	G00 X-q1(Y-q2)/G01 X-q1(Y-q2)Ff1 ;	Shift
(6)	M3;	Spindle forward rotation
(7)	G01 Zz1 Ff1;	
(8)	M19 ;	Spindle orientation
(9)	G00 Xq1 (Yq2);	Shift
(10)	G00 Z-(r1+z1);	G98 mode
	G00 Z-(r1+z1);	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 by the designated shift amount (*1) with 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 rapid traverse, it is necessary to upgrade the M8 series software to version C4 or later and to replace the fixed cycle program. For details, consult the MTB.

(2) Command I, J, and K with incremental positions 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 addresses I and 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.

(4) When both parameters "#1268 ext04/bit4" and "#1271 ext04/bit1" are set to "1", the address K in the G76 and G87 commands specifies the number of repetitions instead of the shift amount. These parameters depend on the MTB specifications.

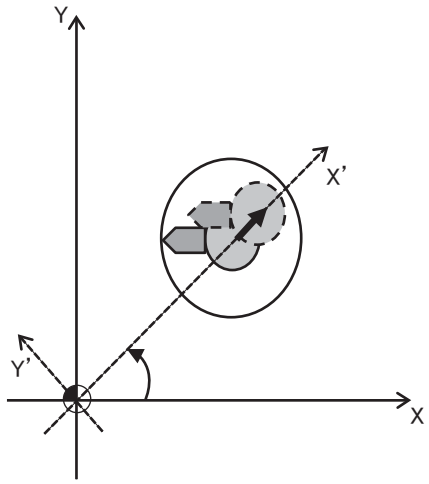
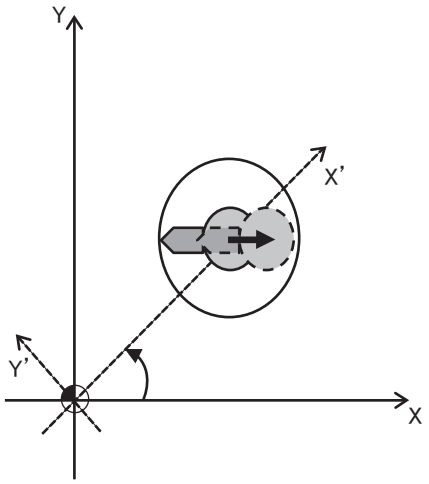
#1268 ext04/bit4: Enable address K to be used for specifying the repetition count in G76/G87 command

#1271 ext04/bit1: Address specifying fixed cycle repetition count

Coordinate system at tool node shift

Whether the tool node shift is performed in the commanded coordinate system or machine coordinate system depends on the MTB specifications (parameter "#1751 cfgPR01/bit1"). When the plane perpendicular to the tool axis is assigned to the rotation coordinate system by the G68 command (coordinate rotation by program), etc., the shift direction is set as shown below depending on the setting value.

The figure below shows the shift direction when the X-Y plane perpendicular to the Z axis (tool axis) is rotated.

Shifted in the commanded coordinate system. (#1751 cfgPR01/bit1 = 0)	Shifted in the machine coordinate system (in the same direction as the tool nose). (#1751 cfgPR01/bit1 = 1)
	

Note

(1) In order to switch the coordinate system to shift the tool nose, the system must be upgraded to the M8 series S/W version E1 or later, and then the fixed cycles need to be replaced. Contact the MTB to use this function.

(2) When the plane including the tool axis is rotated, shift the tool nose in the commanded coordinate system regardless of the setting of the parameter "#1751 cfgPR01/bit1".

(3) The tool nose shift in the mirror image ON state is performed in the mirror image OFF state; therefore, the tool nose shifts to the tool nose direction regardless of the setting of the parameter "#1751 cfgPR01/bit1". However, the shift direction is designated based on the parameter setting when:

- The coordinate rotation is commanded in the mirror image ON state.
- Mirror image is turned ON during the coordinate rotation command.

(4) When the machine coordinate system is set (#1751 cfgPR01/bit1 = 1), do not use the figure rotation simultaneously.

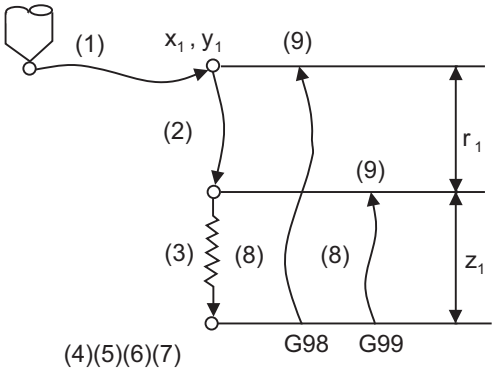
13.1.8 Boring; G88

Command format

G88 Xx1 Yy1 Zz1 Rr1 Ff1 Pp1 Ll1;

Xx1	Designation of hole drilling position (absolute/incremental position)
Yy1	Designation of hole drilling position (absolute/incremental position)
Zz1	Designation of hole bottom position (absolute/incremental position) (modal)
Rr1	Designation of R point position (absolute/incremental position) (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)	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.

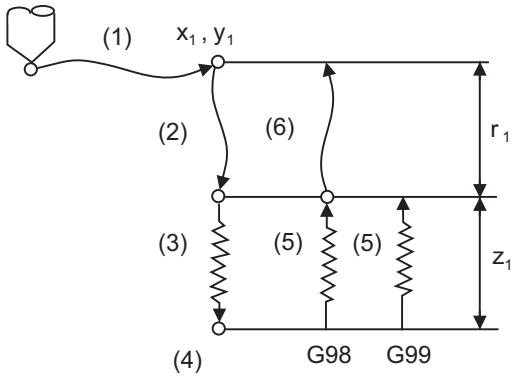
13.1.9 Boring; G89

Command format

G89 Xx1 Yy1 Zz1 Rr1 Ff1 Pp1 Ll1 ,Ii1,Jj1;

Xx1	Designation of hole drilling position (absolute/incremental position)
Yy1	Designation of hole drilling position (absolute/incremental position)
Zz1	Designation of hole bottom position (absolute/incremental position) (modal)
Rr1	Designation of R point position (absolute/incremental position) (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.
,Ii1	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 ,Ii1 ,Jj1 Dd1 Ee1 Jj2 ,Kk1;

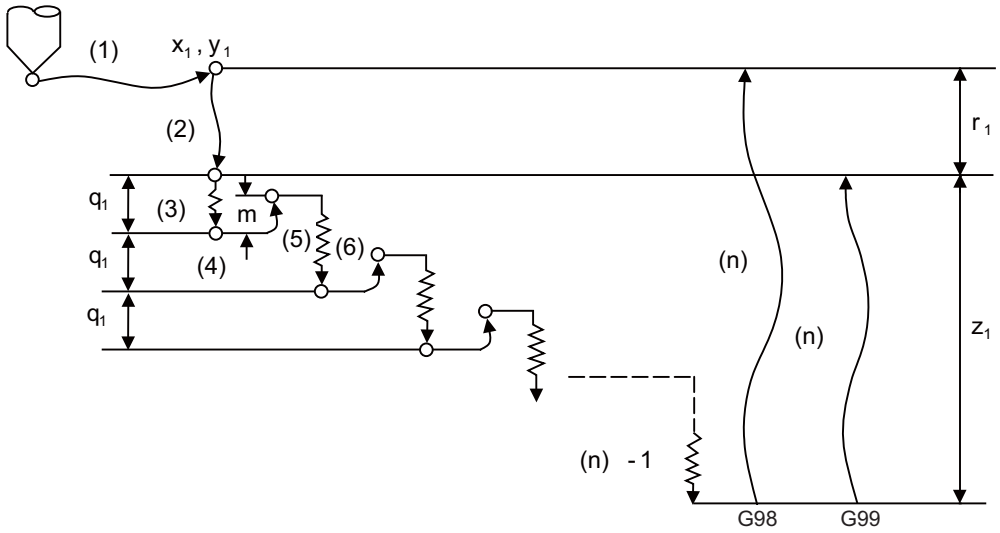
Xx1	Designation of hole drilling position (absolute/incremental position)
Yy1	Designation of hole drilling position (absolute/incremental position)
Zz1	Designation of hole bottom position (absolute/incremental position) (modal)
Qq1	Cut amount for each cutting pass (incremental position) (modal)
Rr1	Designation of R point position (absolute/incremental position) (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.
,Ii1	Positioning axis in-position width
,Jj1	Drilling axis in-position width
Dd1	Designation of spindle number for which chip removal is made (*1)(*2)
Ee1	Designation of frequency in which chip removal is made (*2)
Jj2	Cutting reduction amount (incremental position) (*3) When omitted, the mode will follow the setting of parameter (#19444).
,Kk1	Minimum cutting amount (incremental position) (*3) When omitted, the mode will follow the setting of parameter (#19445).

(*1) There are two types of spindle designation methods: spindle number method and spindle name method, which vary depending on the MTB specifications. When the name (1 to 9) has been specified for all the spindles with the parameter "#3077 Sname" (Spindle command name), a command is given with the spindle name. For others, command with the spindle number.

(*2) For details, refer to "13.1.21 Chip Removal".

(*3) For details, refer to "13.1.22 Cutting Reduction Amount Specification Method".

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.

When the cutting reduction amount "j2" or the minimum cutting amount "k1" is specified, the cutting reduction amount specification method is used. Drilling is performed while the cut amount for each cutting pass "q1" is gradually reduced. For details of the operation, refer to "13.1.22 Cutting Reduction Amount Specification Method".

13.1.11 Reverse Tapping Cycle; G74

Command format

G74 Xx1 Yy1 Zz1 Qq1 Rr1 Ff1 Pp1 ,Rr2 Ss1 ,Ss2 Ll1 ,Ii1,Jj1 Jj2 ,Kk1;

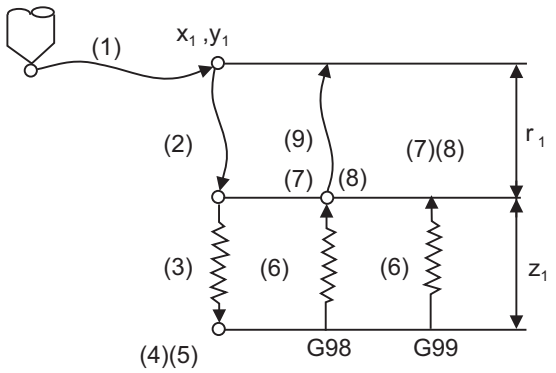
Xx1	Designation of hole drilling position (absolute/incremental position)
Yy1	Designation of hole drilling position (absolute/incremental position)
Zz1	Designation of hole bottom position (absolute/incremental position) (modal)
Qq1	Cut amount for each cutting pass (incremental position) (modal)
Rr1	Designation of R point position (absolute/incremental position) (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
Ll1	Designation of number of repetitions (0 to 9999) When ″0″ is set, processing is not executed.
,Ii1	Positioning axis in-position width
,Jj1	Drilling axis in-position width
Jj2	Cutting reduction amount (incremental position) (*1) When omitted, the mode will follow the setting of parameter (#19444).
,Kk1	Minimum cutting amount (incremental position) (*1) When omitted, the mode will follow the setting of parameter (#19445).

(*1) For details, refer to "13.1.22 Cutting Reduction Amount Specification Method".

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

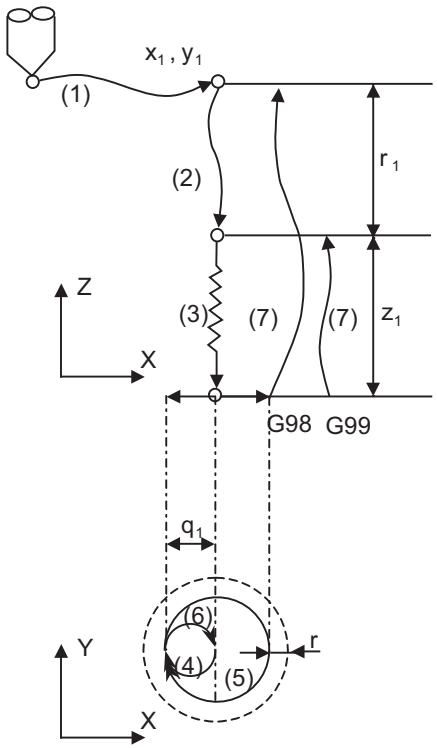
Circular cutting starts with the X and Y axes positioned at the center of the circle, and the Z axis cuts into the commanded position. Then, the tool cuts the inner circumference of the circle drawing a true circle and returns to the center position.

Command format

G75 Xx1 Yy1 Zz1 Rr1 Qq1 Pp1 Ff1 Ll1 ;

Xx1	Designation of hole drilling position (absolute/incremental position)
Yy1	Designation of hole drilling position (absolute/incremental position)
Zz1	Designation of hole bottom position (absolute/incremental position) (modal)
Rr1	Designation of R point position (absolute/incremental position) (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		
(1)	G00 Xx1 Yy1;		n:q1 ≥0 → G02 q1 <0 → G03 r: Tool radius compensation amount of the No. commanded with p1.
(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);	G98 mode	
	G00 Z-z1;	G99 mode	

The operation stops at after the (1), (2) and (6) commands during single block operation.

13.1.13 Fine Boring; G76

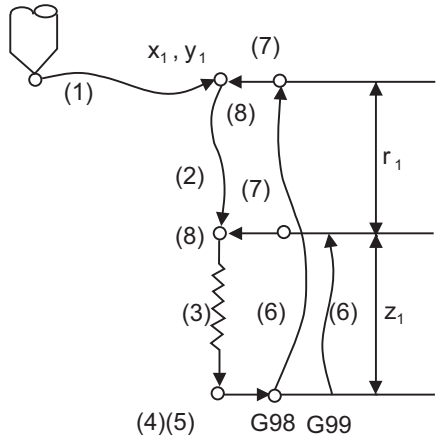


Command format

G76 Xx1 Yy1 Zz1 Rr1 Iq1 Jq2 Kq3 Ff1 Ll1;	
Xx1	Designation of hole drilling position (absolute/incremental position)
Yy1	Designation of hole drilling position (absolute/incremental position)
Zz1	Designation of hole bottom position (absolute/incremental position) (modal)
Rr1	Designation of R point position (absolute/incremental position) (modal)
Iq1 Jq2 Kq3 (*1)	Designation of shift amount (incremental position) (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)
Ll1 (*1)	Designation of number of repetitions (0 to 9999) When ″0″ is set, processing is not executed.

(*1) Depending on the parameter setting, the number of repetitions can be designated by K address. ("#1268 ext04/bit4", "#1271 ext04/bit1")

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".

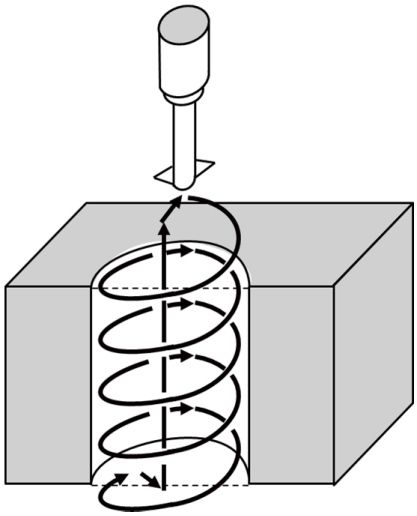
Coordinate system at tool nose shift

Refer to "Coordinate system at tool nose shift" 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. •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 • 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. •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). •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. •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 •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. •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.

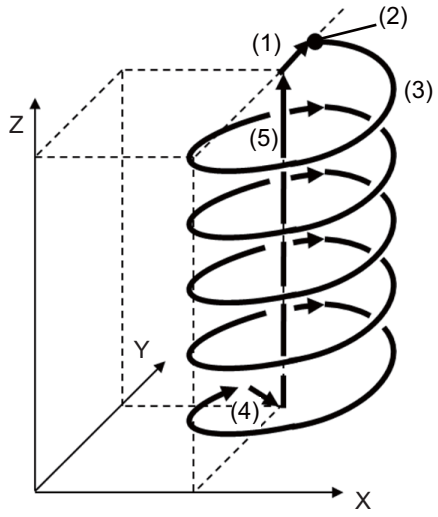
(1) The axis moves (approaches) with G01 from the center of the hole to the radius direction.

(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.

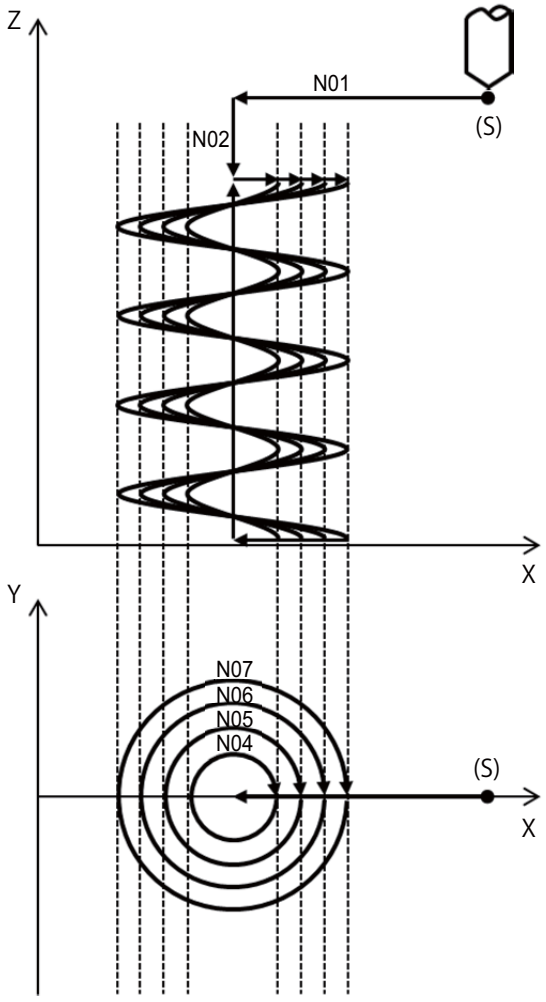
Plane selection	Hole drilling axis
G17 (X-Y)	Zp
G18 (Z-X)	Yp
G19 (Y-Z)	Xp

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 "13.1.16 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 bock 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.)

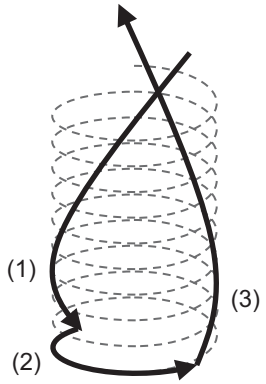
13.1.15 Punchtap Cycle

Function and purpose

This function performs cutting and tapping by synchronously controlling the spindle and servo axis using the dedicated tool for Punchtap. In general, the machining time can be shortened compared to the tapping cycle.
The fixed cycle subprogram dedicated to the Punchtap cycle must be registered.

There are three patterns of operations in the Punchtap cycle, which are selected by the G code.

Pattern	Features
PT1.0	Fastest process out of three patterns
PT1.5	Process that optimizes the machining load, including the "deburring process"
PT2.0	Process that minimizes the machining load, including the "deburring process" and "screw cleaning process"



Dotted line: Path of the tool nose in conventional tapping
Solid line: Path of the tool nose in Punchtap

(1) Helical grooving
(2) Tapping
(3) Retract

Command format

When the drilling axis advances in the negative direction, the spindle rotates forward in the positive Punchtap cycle. The spindle rotation reverses in the reverse Punchtap cycle.

Positive Punchtap cycle

G84.5 X__ Y__ Z__ R__ F__ I__ S__ ,S__ P__ ,I__ ,J__ ,D__ L__ ;	**Performance PT1.0**
G84.6 X__ Y__ Z__ R__ F__ I__ S__ ,S__ P__ ,I__ ,J__ ,D__ L__ ;	**Medium PT1.5**
G84.8 X__ Y__ Z__ R__ F__ I__ S__ ,S__ P__ ,I__ ,J__ ,D__ L__ ;	**Soft PT2.0**

Reverse Punchtap cycle

G74.5 X__ Y__ Z__ R__ F__ I__ S__ ,S__ P__ ,I__ ,J__ ,D__ L__ ;	**Performance PT1.0**
G74.6 X__ Y__ Z__ R__ F__ I__ S__ ,S__ P__ ,I__ ,J__ ,D__ L__ ;	**Medium PT1.5**
G74.8 X__ Y__ Z__ R__ F__ I__ S__ ,S__ P__ ,I__ ,J__ ,D__ L__ ;	**Soft PT2.0**

X	Hole position data
Y	Hole position data
Z	Hole bottom position (Z)
R	R point position
F	Tapping speed (PD)
I	Helical grooving speed (Ph)
S	Spindle rotation speed command
,S	Spindle rotation speed command at retract of Punchtap cycle
P	Dwell time required to pull up the tool from the hole bottom or R point
,I	Positioning axis in-position width
,J	Drilling axis in-position width
,D	Untwisting retract amount (BF)
L	Number of repetitions

Explanation of address

Address	Command range (unit)	Remarks
X	-99999.999 to 99999.999 (mm)	⬧Designate the hole drilling position on the X axis. ⬧The command can be omitted.
Y	-99999.999 to 99999.999 (mm)	⬧Designate the hole drilling position on the Y axis. ⬧The command can be omitted.
Z	-99999.999 to 99999.999 (mm)	⬧Designate the hole bottom position.
R	-99999.999 to 99999.999 (mm)	⬧Designate the R point position. ⬧The command can be omitted.
F	0.001 to 1000000.000 (mm/min) or 0.001 to 999.999 (mm/rev)	⬧Designate the tapping feedrate for cutting feed. The unit "mm/min" is used for G94 modal and "mm/rev" is used for G95 modal.
I	0.001 to 999999999.999 (mm/min) or 0.001 to 999.999 (mm/rev)	⬧Designate the helical grooving feedrate for cutting feed. The unit "mm/min" is used for G94 modal and "mm/rev" is used for G95 modal.
S	-99999999 to 99999999 (r/min)	⬧S commands of the "Sn = *****" type are ignored in the Punchtap cycle. (n: spindle number, *****: rotation speed) ⬧If an S command is issued during the Punchtap cycle modal, the program error (P186) occurs. ⬧If no S command is included in the same block, the program error (P181) occurs.
,S	0 to 99999 (r/min)	⬧When the value is smaller than the spindle rotation speed (S command), the value of the spindle speed is valid even at the retract. ⬧When the spindle rotation speed at retract is not "0", the parameter "#1172 tapovr" (tap retract override) is invalid. ⬧The command can be omitted.
P	0 to 99999 (s)	⬧The command can be omitted. ⬧When the command is omitted, dwelling is not performed. ⬧Compared to the setting value of the parameter "#1313 TapDwl" (synchronous tapping hole bottom waiting time), the larger value is set as the dwell time.
,I	0.001 to 999.999 (mm)	⬧The command can be omitted.
,J	0.001 to 999.999 (mm)	⬧The command can be omitted.
,D	-99999.999 to 99999.999 (mm)	⬧The sign is ignored. ⬧The command can be omitted.
L	0 to 9999	⬧The command can be omitted.

The positioning plane is determined by the G17, G18 and G19 plane selection commands, and the hole drilling axis is the axis perpendicular to the plane.

Detailed description

High-speed tapping is performed by the Punchtap cycle that performs the helical grooving up to the hole bottom with the cutting edge for the helical groove on the tip of the tool and the tapping with the cutting edge, and then pulls up the tool along the helical groove.
The cutting edges for tapping are arranged helically on the tool, and by cutting the helical groove according to this pitch, the tool makes cuttings up to the hole bottom.

Operations (2), (3) and (5) make cuttings at the helical grooving speed (Ph) (mm/rev), and operation (4) makes cuttings at the tapping speed (PD) (mm/rev).
Refer to the figure below for the stop position during single block operation. The tool does not stop at the in-position check position (does not stop during cutting).

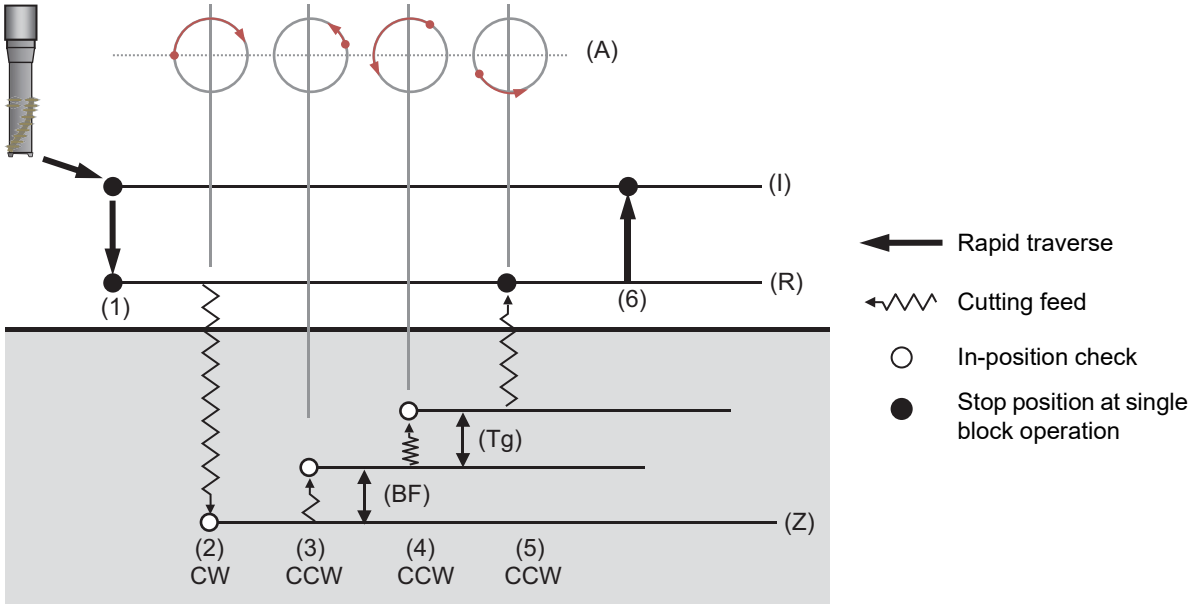
For tapping, the spindle rotation amount (εg) and drilling axis movement amount (Tg) is obtained as shown below. For 180° in the calculation formula, the setting value of the parameter "#11773 ptapag" (rotation angle of tapping) is actually used.

Spindle rotation amount (angle)

$$\varepsilon g = 180° \times (1 + \frac{PD}{Ph - PD})$$

Drilling axis movement amount (mm)

$$Tg = PD \times \frac{\varepsilon g}{360°}$$



(A) Spindle rotation direction and angle (bottom view)
(I) Initial point height
(R) R point height
(Z) Hole bottom height
(Tg) Movement amount at tapping
(BF) Retract amount

Operation	Details of operation	Spindle rotation amount	Spindle rotation direction
(1)	Moves the tool to the R point with rapid traverse.	-	Stop
(2)	Performs helical grooving at the helical grooving speed (Ph). If a dwell command is issued, dwelling is performed.	$360° \times \frac{Z}{Ph}$	CW
(3)	Retracts the tool by the untwisting retract amount (BF) at the helical grooving speed (Ph).	$360° \times \frac{BF}{Ph}$	CCW
(4)	Performs tapping at the tapping speed (PD).	εg	CCW
(5)	Pulls up the tool to the R point at the helical grooving speed (Ph). If a dwell command is issued, dwelling is performed.	$360° \times (\frac{Z - BF - Tg}{Ph})$	CCW
(6)	Pulls up the tool to the initial point.	-	Stop

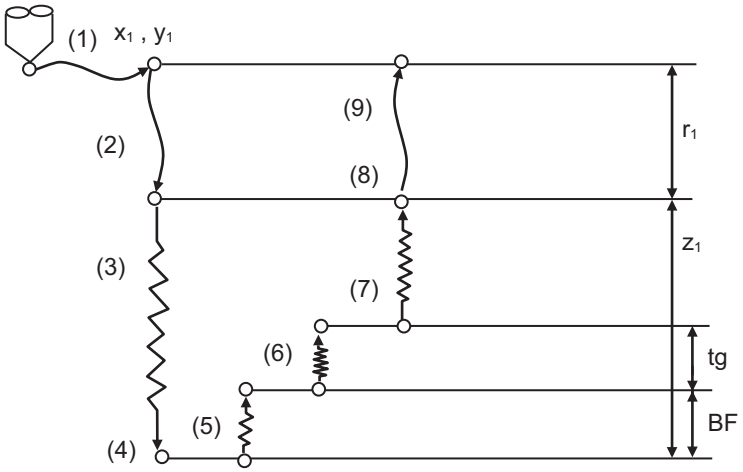
Details of operation

This section describes how each Punchtap cycle operates by each G code.

The "ΔD" of G84.6 (PT1.5) and G84.8 (PT2.0) is "#11772 ptapd" (deburring adjustment amount), which is set when the finish is substandard.

G84.5 (PT1.0)

G84.5 Xx1 Yy1 Zz1 Rr1 Ff1 Ii1 Ss1 ,Ss2 Pp1 ,Ii1 ,Jj1 ,Dd1 Ll1;

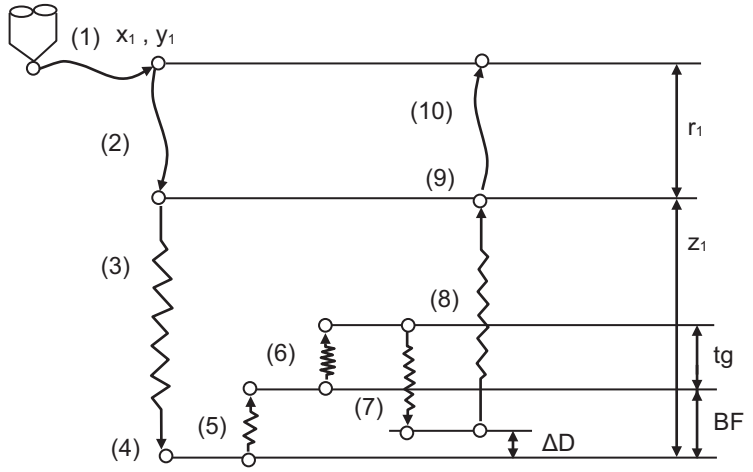


BF: Retract amount (,D)
tg: Tapping movement amount

Operation pattern	Program	
(1)	G00 Xx1 Yy1 ;	
(2)	G00 Zr1 ;	
(3)	G01 Zz1 Ii1 ;	Helical grooving (Forward rotation, Helical grooving speed)
(4)	G04 Pp1 ;	
(5)	G01 Z-BF Ii1 ;	Untwisting retract (Reverse rotation, Helical grooving speed)
(6)	G01 Z-tg Ff1 ;	Tapping (Reverse rotation, Tapping speed)
(7)	G01 Z-(z1-BF-tg) Ii1 ;	Retract (Reverse rotation, Helical grooving speed)
(8)	G04 Pp1 ;	
(9)	G98 mode G00 Z-r1 ;	
	G99 mode No movement	

G84.6 (PT1.5)

G84.6 Xx1 Yy1 Zz1 Rr1 Ff1 Ii1 Ss1 ,Ss2 Pp1 ,Ii1 ,Jj1 ,Dd1 Ll1;

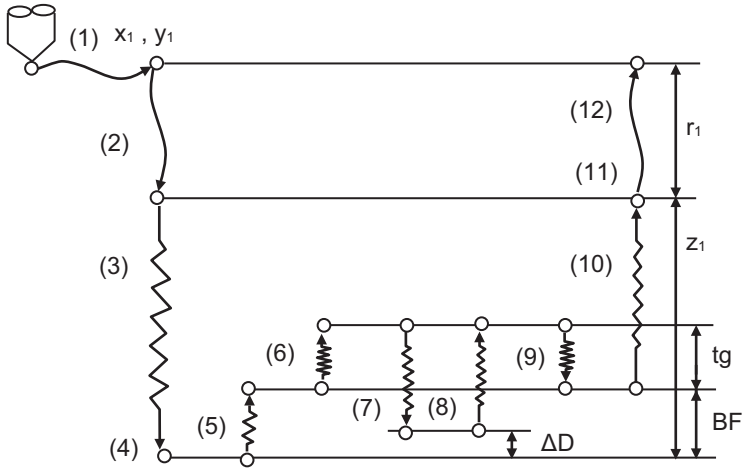


BF: Retract amount (,D)
tg: Tapping movement amount
ΔD: "#11772 ptapd"
(Deburring adjustment amount)

Operation pattern	Program	
(1)	G00 Xx1 Yy1 ;	
(2)	G00 Zr1 ;	
(3)	G01 Zz1 Ii1 ;	Helical grooving (Forward rotation, Helical grooving speed)
(4)	G04 Pp1 ;	
(5)	G01 Z-BF Ii1 ;	Untwisting retract (Reverse rotation, Helical grooving speed)
(6)	G01 Z-tg Ff1 ;	Tapping (Reverse rotation, Tapping speed)
(7)	G01 Z(BF+tg-ΔD) Ii1 ;	Deburring (Reverse rotation, Helical grooving speed)
(8)	G01 Z-(z1-ΔD) Ii1 ;	Retract (Reverse rotation, Helical grooving speed)
(9)	G04 Pp1 ;	
(10)	G98 mode G00 Z-r1 ;	
	G99 mode No movement	

G84.8 (PT2.0)

G84.8 Xx1 Yy1 Zz1 Rr1 Ff1 Ii1 Ss1 ,Ss2 Pp1 ,Ii1 ,Jj1 ,Dd1 Ll1;



BF: Retract amount (,D)
tg: Tapping movement amount
ΔD: "#11772 ptapd"
(Deburring adjustment amount)

Operation pattern	Program	
(1)	G00 Xx1 Yy1 ;	
(2)	G00 Zr1 ;	
(3)	G01 Zz1 Ii1 ;	Helical grooving (Forward rotation, Helical grooving speed)
(4)	G04 Pp1 ;	
(5)	G01 Z-BF Ii1 ;	Untwisting retract (Reverse rotation, Helical grooving speed)
(6)	G01 Z-tg Ff1 ;	Tapping (Reverse rotation, Tapping speed)
(7)	G01 Z(BF+tg-ΔD) Ii1 ;	Deburring (Reverse rotation, Helical grooving speed)
(8)	G01 Z-(BF+tg-ΔD) Ii1 ;	Deburring retract (Reverse rotation, Helical grooving speed)
(9)	G01 Z-tg Ff1 ;	Screw cleaning (Forward rotation, Tapping speed)
(10)	G01 Z-(z1-BF) Ii1 ;	Retract (Reverse rotation, Helical grooving speed)
(11)	G04 Pp1 ;	
(12)	G98 mode G00 Z-r1 ;	
	G99 mode No movement	

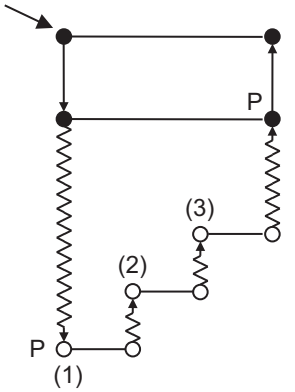
In-position check

The in-position check in each process other than the R point retract of the Punchtap cycle can be finely adjusted by the parameter setting. The parameter setting depends on the MTB specifications.

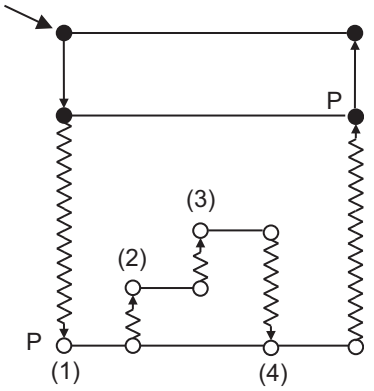
Parameter	Setting value	Deceleration check operation
#1702 cfg02/bit0 to bit5	0	Command deceleration
	1	In-position check with "#11771 PTapInp"

In the Punchtap cycle, it is possible to set whether or not to perform the in-position check at each of points (1) to (6) in the figure below.
Each of points (1) to (6) corresponds to each bit of the parameter "# 1702 cfg02", which can be set to "valid" or "invalid" individually.
Dwelling is also performed at the P point.
No switching parameter is provided to pull up the tool to the R point, and the in-position check is always performed.
When the "Error detection" signal is ON, the in-position check is performed with the parameter "#11771 PTapInp" regardless of the setting value of the parameter "#1702 cfg02".

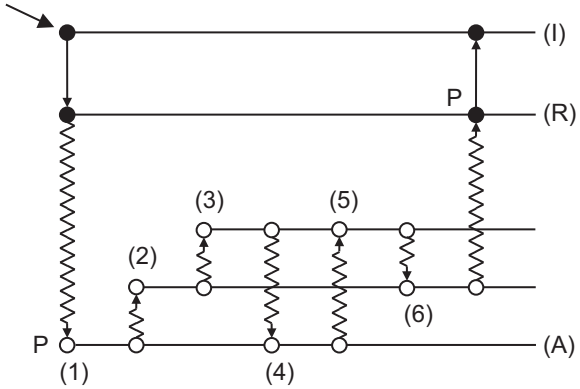
For G84.5 (PT1.0)



For G84.6 (PT1.5)



For G84.8 (PT2.0)



(I) Initial point
(R) R point (r)
(A) Hole bottom

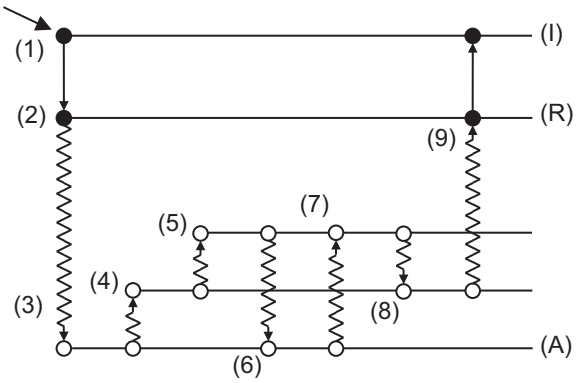
#1702	bit5	bit4	bit3	bit2	bit1	bit0
G84.5 (PT1.0)	-	-	-	(3)	(2)	(1)
G84.6 (PT1.5)	-	-	(4)	(3)	(2)	(1)
G84.8 (PT2.0)	(6)	(5)	(4)	(3)	(2)	(1)

[Setting example of parameter to perform in-position check at points (1), (3), (4), and (6) above]

Set bit0, bit2, bit3, and bit5 of the parameter #1702 to "1".

The following parameters are used to judge the in-position check at each point.

For G84.8 (PT2.0)



(I) Initial point
(R) R point (r)
(A) Hole bottom

	Parameters used for in-position check
(1)	"#2077 G0inps" or "#2224 SV024 INP" (*1)
(2)	"#2077 G0inps" or "#2224 SV024 INP" (*1)
(3)	"#11771 PTapInp" for each (*2)
(4)	
(5)	
(6)	
(7)	
(8)	
(9)	"#2078 G1inps" or "#2224 SV024 INP" (*1)

(*1) The setting value of the parameters "#2077 G0inps" and "#2078 G1inps", is compared with parameter "#2224 SV024 INP", and the larger value is used to perform in-position check.

(*2) When the parameter "#11771 PTapInp" is set to "0", in-position check is performed with the value of the parameter "#2224 SV024 INP".

Spindle zero point return operation in Punchtap

The spindle zero point return operation in Punchtap is the same as the synchronous tapping.
In addition, the parameter in the zero point return operation can be changed with the G10L70 command.

Feedrate

The tapping speed and helical grooving speed specified with the Punchtap cycle command can be switched to the feed per minute using the parameter.

The combination of the parameter "#1268 ext04/bit2" (Enable synchronous tapping per minute) and G code determines the meaning of the F command (tapping speed) and I command (helical grooving speed).

#1268/bit2	During G94 modal (feed per minute)	During G95 modal (feed per revolution)
1	Feed per minute	Feed per revolution
0	Feed per revolution	Feed per revolution

To update the F modal, set the parameter "#1292 ext28/bit1" (Address F given in sync tap cycle) to "1", not the parameter "#1268 ext04/bit2" (Enable synchronous tapping per minute).

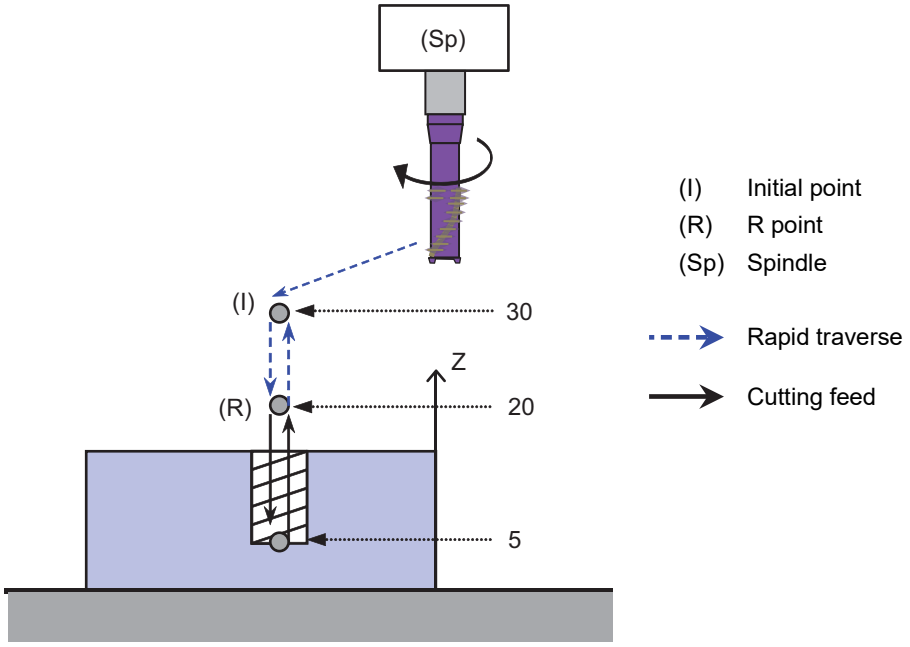
Feed per minute
Ff1, Ii1 : Feedrate (mm/min, inch/min)
(Pitch calculation formula) Pitch = F command value / S command value

Feed per revolution
Ff1, Ii1 : Feedrate (mm/rev, inch/rev)

The I command is retained in each of G94 and G95, and it is retained even after the fixed cycle is canceled with G80.
If the I command is omitted, Punchtap is performed using the previous I command as the helical grooving speed.
If the I command is omitted without issuing the I command even once, a program error (P184) will occur.

Program example

```
G90 G98 ;
G00 X30. Y0. Z30. ;
G95 ;
G84.5 X30. Z5. R20. F1. S500 I31.5. ;
G80 ;
G94 ;
```



Relationship with other functions

(1) High-speed synchronous tapping

Even in the Punchtap cycle, the high-speed synchronous tapping that corrects the tracking delay in the communication between drive units is valid.
The reduction rate in Punchtap depends on the MTB specifications (parameter "#3136 ptaptr").
Other parameters are also used as the synchronous tapping.
The restrictions and enabling conditions related to the high-speed synchronous tapping are the same as for the synchronous tapping.

(2) Tapping retract

The "Tap retract" signal (YC5C) is invalid in the Punchtap cycle.
The "Tap retract possible" signal is not output even if the Punchtap cycle is interrupted.

Tapping retract is not available when the Punchtap cycle is used.
When the Punchtap cycle machining is interrupted due to power OFF, etc., pull out the tool by one of the following methods.

- Rotate the spindle in the same direction as when the Punchtap cycle is commanded, crush the screw thread, and pull out the tool.
- While rotating the spindle in the opposite direction to when the Punchtap cycle is commanded, manually operate the Z axis to pull out the tool.

(3) Reference position retract

The "Reference position retract" (YC2D) signal is invalid in cutting feed of the Punchtap cycle, and the reference position return is also not performed without a reset stop.
This signal is valid in rapid traverse, and the reference position return is performed after a reset stop.

(4) Coordinate system operation, inclined surface machining command

Do not command the Punchtap cycle while the hole drilling axis (Z axis) is rotated by coordinate rotation by program or 3-dimensional coordinate conversion.
Also, do not command the Punchtap cycle during the inclined surface machining command.
If any of these commands are issued, tapping may not be available.

(5) Chopping

The Punchtap cycle can even be executed during chopping, however, if the chopping axis is included in the positioning axis or drilling axis, the operation error (M01 0151) occurs, and the machine stops in the interlock state.

(6) Stroke limit

The stroke limit is valid for both the positioning axis and drilling axis.
When the drilling axis is stopped at the stroke limit, the spindle also stops synchronously.

(7) Coil switch

In the Punchtap cycle, the spindle motor operates with L coil regardless of the spindle rotation speed.
However, when the parameter "#1239 set11/bit0" (Coil switching method) is set to "1" and the parameter "#1223 aux07/bit7" (Synchronous tap method) is set to "0", the type of coil is switched to H coil according to the change of the spindle rotation speed. Since the torque may get insufficient in such a case, command the spindle rotation speed of the L coil so that the coil is not switched. (The settings of these parameters depend on the MTB specifications.)
When the coil switch is performed during the Punchtap cycle, the coil is switched after the cycle ends.

(8) Graphic check

The helical grooving speed is not reflected on the cutting speed.

(9) Synchronous tapping with analog I/F spindle

The Punchtap cycle is not available for the analog spindle.
The program error (P182) occurs if commanded.

(10) Manual arbitrary reverse run

The manual arbitrary reverse run is only valid in the actual cutting mode. If the Punchtap cycle is commanded while the dry run operation mode is enabled, the program error (P182) occurs. Also, the Punchtap cycle is handled as a reverse run prohibited block.

(11) Finish shape view programming

The path to be traced by using Punchtap cycle is not reflected.
The G code of the Punchtap cycle is ignored, and the subsequent codes are reflected on the finishing shape.

(12) Synchronous control

When the reference axis of synchronous control is a tapping axis, in-position check in the Punchtap cycle is disabled.



Precautions

(1) The multi-step acceleration/deceleration is invalid in the Punchtap cycle.

(2) Dry run is valid only for rapid traverse sections.

(3) If the "External deceleration" signal is turned ON during Punchtap, the feedrate does not change even when deceleration conditions are satisfied.

(4) The cutting feed override is disabled during the Punchtap cycle, and the feedrate is 100%. Rapid traverse override is valid.

(5) The spindle override is invalid during the Punchtap cycle.

(6) The automatic machine lock is valid. If the Punchtap cycle is executed while the drilling axis is machine-locked, the spindle does not rotate either.

(7) The parameters ("#3106", "#3110", and "#3111") of the tap starting angle are also used with the synchronous tapping.

(8) The G94 command is a modal command and valid until the G95 command is issued next.

(9) For the "F" command and "I" command, the feed per minute and the feed per revolution are switched in the same conditions.

(10) When the automatic operation is started in the miscellaneous function lock ("MST lock" is ON), the spindle is not synchronized even after block stop or feed hold stop processing has been ended.

(11) The "NC output" signal under tapping is output during the Punchtap cycle.

(12) The Punchtap cycle is always set in the synchronous mode. The ",R0" command is ignored. The setting of the parameter "#1229 set01/bit4" (synchronous tapping) is also irrelevant.

(13) The "Spindle OFF mode" signal is valid. If an attempt is made to execute the Punchtap cycle in the Spindle OFF mode, the Punchtap cycle is executed without rotating the spindle. This is used for program check. Dry run is enabled in the Spindle OFF mode.

(14) When the feed hold button is pressed during cutting feed of the punch tapping cycle, the machining does not stop immediately but stops when the block to pull-up tool on the helical groove is completed. When the feed hold button is pressed during rapid traverse, the machining stops immediately.

(15) The "Synchronous tapping command polarity reversal" signal is valid.

(16) The tapping error displayed by selecting [Drv mon] - [Spindle unit] is not displayed correctly during the Punchtap cycle.
To check the tapping error, sample the commanded position and FB position of the tapping axis, and the command position and FB position of the spindle, and then calculate the error using the following formula.
Switch the pitch depending on the FB position of the tapping axis and the spindle.

$$\text{Tapping error [deg]} = \frac{\left|\begin{array}{c}\text{Tapping axis command position} \\ -\ \text{Tapping axis FB position}\end{array}\right|}{\text{Pitch of each process}} \times 360 - \left|\text{Spindle command position} - \text{Spindle FB position}\right|$$

13.1.16 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

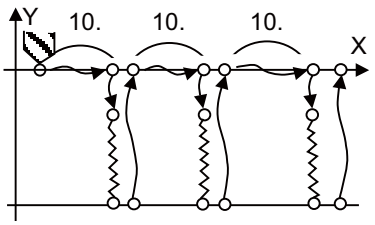
(9) When the fixed cycle is executed, only the modal command issued in the fixed cycle program will be valid in the fixed cycle subprogram. The modal of the program which called the fixed cycle will not be affected.

(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 mode, the positioning will also be incremented each time.

(Example) G91 G81 X10. Z-50. R-20. F100. L3;



(13) If the spindle rotation speed value during return is smaller than the spindle speed value, the spindle rotation speed value is valid even during return.

(14) If gradients of the 2nd and 3rd acceleration/deceleration stages according to the spindle rotation speed and time constants set in the parameters are each steeper than the previous stage's gradients, the previous stage's gradient will be valid.

(15) If the values set in the spindle specification parameter "tap rotation speed" and "the synchronous tap change-over spindle rotation speed 2" exceed the maximum rotation speed, the spindle rotation speed will be clamped at the maximum rotation speed.

(16) If the spindle rotation speed is not 0 during return, the taping retract override value will be invalid.

(17) As shown below, in a block where the movement direction of either axis reverses, the servo system load will greatly increase, so do not command the in-position width in the machining program.

G01 X100. ,I10.0;

X-200.;

(18) If the in-position width commanded by the programmable in-position width command is increased, the positioning time and linear interpolation time can be reduced. However, the position error amount of the previous block will also increase before the next block starts, and the actual machining could be obstructed.

(19) The in-position width and the position error amount are constantly compared, so the position error amount at the point to be judged as in-position will be smaller than the commanded in-position width.

(20) If the in-position width commanded with the programmable in-position command is small, the commanded deceleration check or in-position check by the parameters may be carried out first.

(21) Synchronous or asynchronous tapping can be selected with the M function.

Base specification parameters

#	Item		Details	Setting range
1272 (PR)	ext08	bit1	M-function synchronous tapping cycle valid.	0: Invalid 1: Valid

Synchronous tapping cannot be selected with the M function when this parameter is OFF.

Base specification parameters

#	Item	Details	Setting range
1513	stapM	M code for synchronous tapping selection	0 to 99999999

The synchronous tapping mode is selected with the miscellaneous function code set with this parameter.

The M function can be commanded just before or in the same block as the tapping command.

To use this parameter, validate "#1272 ext08/bit1" (M function synchronous tapping cycle).

The selection of synchronous or asynchronous tappinf will follow the combination shown below.

	Combination											
Program command (,R0/1)	0	0	0	0	1	1	1	1	No command			
#1229 set01/bit4 (Synchronous tap)	0	0	1	1	0	0	1	1	0	0	1	1
M function code (M**)	×	○	×	○	×	○	×	○	×	○	×	○
Synchronous/asynchronous selection	A	A	A	A	B	B	B	B	A	B	B	B

× Not commanded
○ Commanded
A Asynchronous tapping
B Synchronous tapping

<Note>

- Do not use M00, M01, M02, M30, M98 and M99.
- Depending on the model, selection by the M function may not be available.
- These parameters depend on the MTB specifications.

(22) Even when the parameter "#1151 rstin" is set to "0", the fixed cycle will be canceled if NC reset 1 is carried out while executing the fixed cycle.

(23) If a tapping axis is under machine lock, normal synchronous tapping is applied even though high-speed synchronous tapping function is enabled.

(24) When the parameter "#1253 set25/bit2" (Acceleration/Deceleration mode change in hole drilling cycle) is set to "1" and the override value is set to less than 100%, the cycle time may become longer than that when the parameter "#1253 set25/bit2" is set to "0".

13.1.17 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.

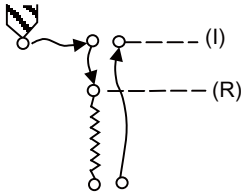
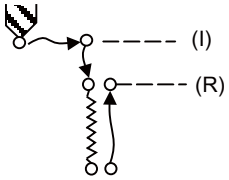
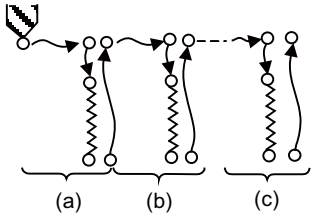
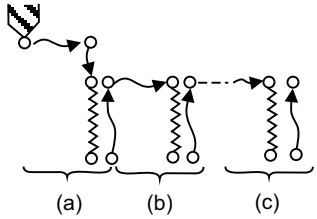
Command format

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 ;	 Initial level return is executed.	 R point level return is executed.
Two or more executions	G81 X100. Y100. Z-50. R25. L5 F1000 ;	 Initial level return is executed for all times.	

(a) First time (b) Second time (c) Last time

13.1.18 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.

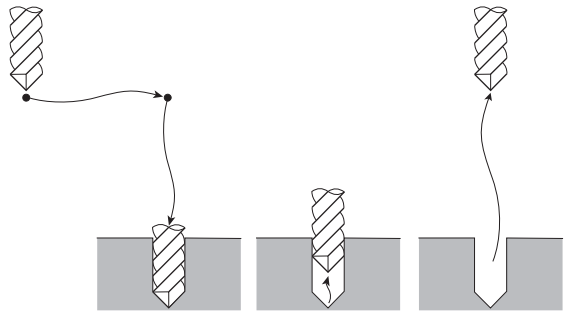
<Example>

```
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 ;
```

13.1.19 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) in the figure) and returns to the initial point or R point in rapid traverse ((2) in the figure).

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. These parameters depend on the MTB specifications.

- #2170 Lmc1QR (Lost motion compensation gain 1 for high-speed retract)
  (corresponds to "#2216 SV016 (LMC1)" (Lost motion compensation 1))
- #2171 Lmc2QR (Lost motion compensation gain 2 for high-speed retract)
  (corresponds 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 hole 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.

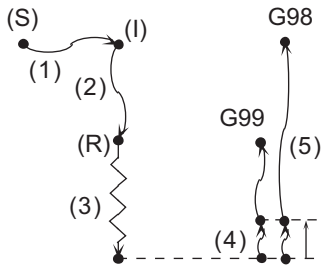
(4) This function is invalid during the following command modal:
In this case, the drill moves in the ordinary rapid traverse even if "#8123" is enabled.

- G43.1 (Tool length compensation in the tool axis direction)
- G43.4, G43.5 (Tool center point control)
- G68 (3-dimensional coordinate conversion)

(5) While the multiple-axis synchronization control is being executed, if the drilling cycle high-speed retract in which the master axis is set as the drilling axis is commanded, the slave axis moves at the same speed as the master axis.

Details of operation

Operation at G81 command

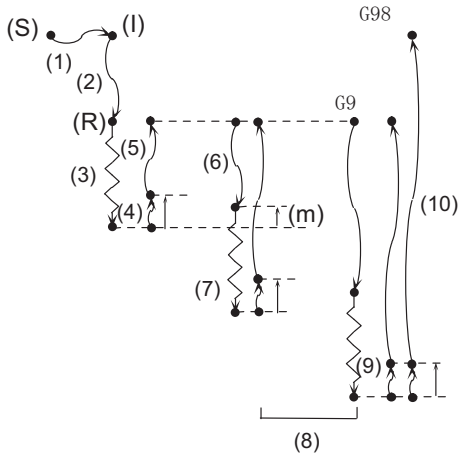


(1) Moves from start point to initial point
(2) Moves from initial point to R point
(3) Cutting feed
(4) Retracted at high-speed
(5) Returns to R point or initial point

(I) Initial point
(S) Start point
(R) R point

During single block operation, the axis stops after (1), (2) and (5) only.

Operation at G83 command

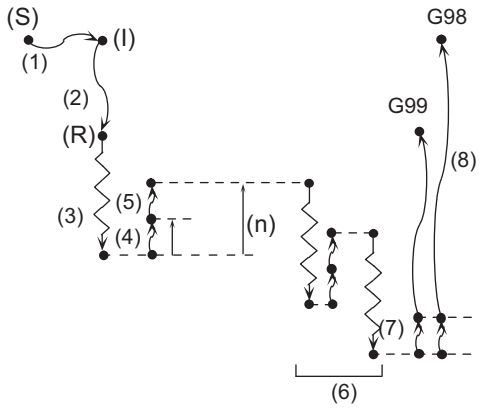


(1) Moves from start point to initial point
(2) Moves from initial point to R point
(3) Cutting feed
(4) Retracted at high-speed
(5) Returns to R point
(6) Moves to the position where the "G83 Return amount" is added to the previous cutting feed position.
(7) Cutting feed
(8) Repeats (4) to (7)
(9) Retracted at high-speed
(10) Returns to R point or initial point

(I) Initial point
(S) Start point
(R) R 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.

13.1.20 Acceleration/Deceleration Mode Change in The Fixed Cycle for Drilling

Function and purpose

This function switches the acceleration/deceleration mode for fixed cycle for drilling between the constant-gradient 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 the fixed cycle for drilling) enabled, operation will be as follows.

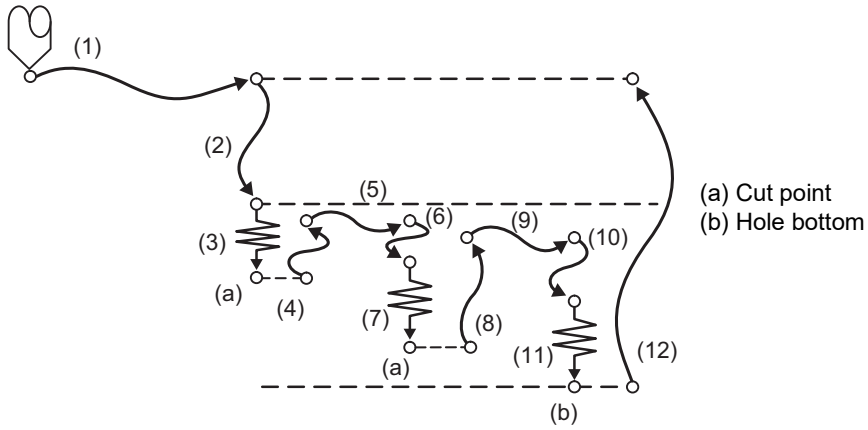
(1) Acceleration/deceleration mode will be either linear or soft method. (Unless soft acceleration/deceleration is applied, the linear method will always be applied.)

(2) Operation is performed based on the parameter setting that enables the constant-gradient acceleration/deceleration after interpolation.
Acceleration/deceleration gradient for G00 (rapid traverse) is determined with "#2001 rapid" (rapid traverse rate) and "#2004 G0tL" (G0 time constant (linear)), and acceleration/deceleration gradient for G01 (cutting feed) is determined with "#2002 clamp" (cutting feedrate for clamp) and "#2007 G1tL" (G1 time constant (linear)).
Refer to "7.9 Rapid Traverse Constant-gradient Multi-step Acceleration/Deceleration" or "7.10 Cutting Feed Constant-gradient Acceleration/Deceleration" for details on the constant-gradient 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).			

13.1.21 Chip Removal

Function and purpose

The chip removal function removes chips that have adhered to the tool during fixed cycle hole drilling by reversing the spindle.
Using addresses D and E, specify the target spindle and the frequency of the reverse operation.

The commanded reversal spindle number and reversal frequency are valid only for the fixed cycle of G81 (drill, spot drill), G82 (drill, counter boring), G83 (deep-hole drilling cycle), and G73 (stepping cycle).

Command format

The command format is the same as that for fixed cycles (G81, G82, G83, and G73).

Detailed address setting

Address	Meaning of address	Details	Command range	Remarks
D	Reversal spindle number	Commands the spindle number for the chip removal operation. (*1)	For spindle number: 1 to n (n: Maximum number of available spindles) For spindle name: 1 to 9	♦If a value outside the command range is specified, the program error (P35) occurs. ♦The data is held as modal information. (*2)(*3)
E	Reversal frequency	Commands the frequency to perform the chip removal operation when D is commanded. When "0" is commanded, the chip removal operation is not performed. <Example> If "1" is commanded, the chip removal operation is performed every time of drilling. If "2" is commanded, the chip removal operation is performed every two times of drilling.	0 to 999	

(*1) There are two types of spindle designation methods: spindle number method and spindle name method, which vary depending on the MTB specifications. When the name (1 to 9) has been specified for all the spindles with the parameter "#3077 Sname" (Spindle command name), a command is given with the spindle name. For others, command with the spindle number.

(*2) Command address D and address E when commanding the G code for chip removal for the first time (at the start of modal).
When starting modal, command both address D and address E. If only one of them is commanded, the program error (P33) occurs.
Also, if address D and address E are commanded during modal without commanding them at the start of modal, the program error (P33) occurs.

(*3) For details on the modal specifications, refer to "Modal specifications for address D and address E" below.

Modal specifications for address D and address E

(1) If address D is not commanded, the reversal spindle commanded previously with address D is inherited.
If the same address D as the previous one is commanded, the same operation is performed.

G91 G83 X-50. Z-50. R-50. Q-10. P3000 F2000 L3 D1 E2;	
X-50. L3;	The previous block (D1) is inherited.
G83 X-50. L3 D1 E2;	The previous block (D1) is inherited.

(2) If a reversal spindle different from the previous one is commanded with address D, the program error (P33) occurs.

G91 G83 X-50. Z-50. R-50. Q-10. P3000 F2000 L2 D1 E1;	
X-50. L2 D2;	The program error (P33) occurs.

(3) If address E is not commanded, the reversal frequency commanded previously with address E is inherited.
Counting the cumulative number of drilling operations to determine whether it is time to remove chips continues from the previous address E command block.
If the same address E as the previous one is commanded, the same operation is performed.

G91 G83 X-50. Z-50. R-50. Q-10. P3000 F2000 L3 D1 E2;	
X-50. L3;	The previous block (E2) is inherited. Counting of the cumulative number of drilling operations is continued.
G83 X-50. L3 E2;	The previous block (E2) is inherited. Counting of the cumulative number of drilling operations is continued.

(4) If a reversal frequency different from the previous one is commanded with address E, the reversal frequency is changed.
The cumulative number of drilling operations to determine the time of chip removal is cleared.
Unlike the final drilling, the chip removal is not performed for the last drilling of the block for which the reversal frequency is not yet changed.

G91 G83 X-50. Z-50. R-50. Q-10. P3000 F2000 L3 D1 E2;	
X-50. L3 E3;	The reversal frequency (E3) is changed. The cumulative number of drilling operations is cleared.
G83 X-50. L3 E2;	The reversal frequency (E2) is changed. The cumulative number of drilling operations is cleared.

(5) If the fixed cycle mode is canceled with the G80 command or the G command of group 1, the address D and E commands are cleared.

G91 G83 X-50. Z-50. R-50. Q-10. P3000 F2000 L3 D1 E2;	
G80;	D1 and E2 are cleared.
G83 X-50. Z-50. R-50. Q-10. P3000 F2000 L3;	The chip removal is not performed.

Detailed description

Operation conditions

The target fixed cycles are G81, G82, G83, and G73.

To perform chip removal, it is necessary to set the command address D (reversal spindle number), command address E (reversal frequency), and parameter "#8100 Chip removal speed".

The chip removal operation up to the final drilling

(1) The spindle reverse operation starts at the initial point after drilling or at the R point.
After waiting for top speed of the spindle reverse, movement to the next drilling position starts. Set the spindle reverse rotation speed with the parameter "#8100 Chip removal speed". The reverse rotation speed is regarded to reach the commanded speed when the speed is 95% or more of the speed set in the parameter "#8100 Chip removal speed".

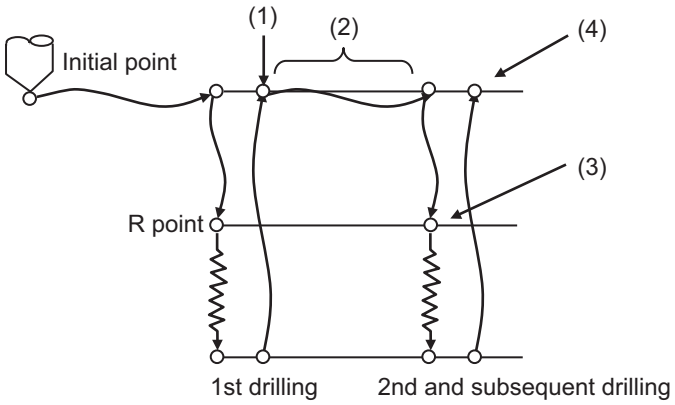
(2) The spindle rotates forward as soon as the movement to the next drilling position starts.

(3) The drilling starts after the spindle forward rotation speed reaches the commanded speed. If the spindle forward rotation speed does not reach the commanded speed yet before the arrival at the R point of the next drilling position, the drilling waits until the speed reaches the commanded value. The forward rotation speed is regarded to reach the commanded speed when the speed is 95% or more of the speed commanded for drilling.

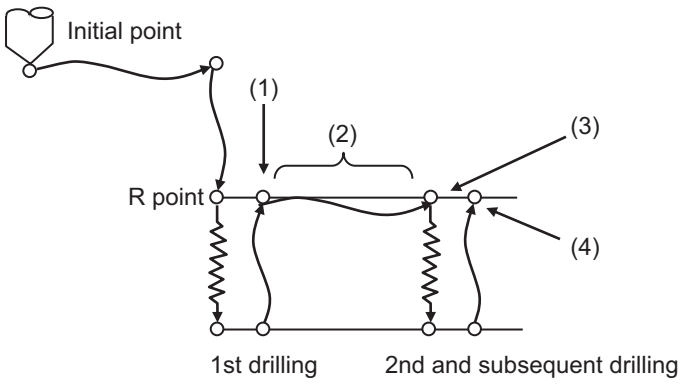
(4) The spindle reverse operation is performed every number of drill operations as specified by address E.
The spindle reverse operation is also performed after the final drilling is completed.

(Example) When "E2" is commanded, the spindle reverse operation is performed every two times of drilling.

[Initial point return]



[R point return]



If the NC is reset when the spindle is reversed, the rotation direction returns to the one commanded by the user PLC.

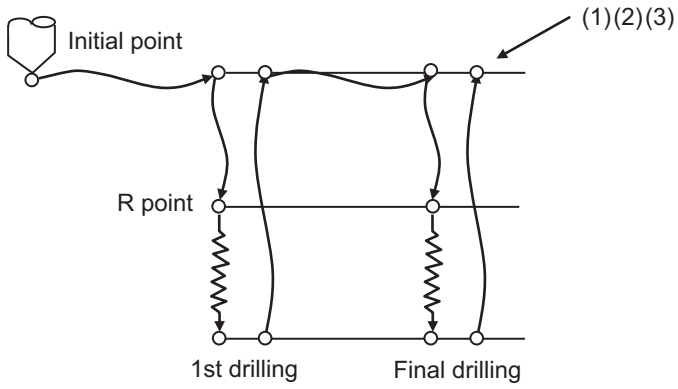
The chip removal operation during the final drilling

(1) The spindle reverse operation starts at the initial point after the final drilling or the R point, and waits until the spindle speed reaches the commanded speed set in the parameter "#8100 Chip removal speed". The reverse rotation speed is regarded to reach the commanded speed when the speed is 95% or more of the speed set in the parameter "#8100 Chip removal speed".

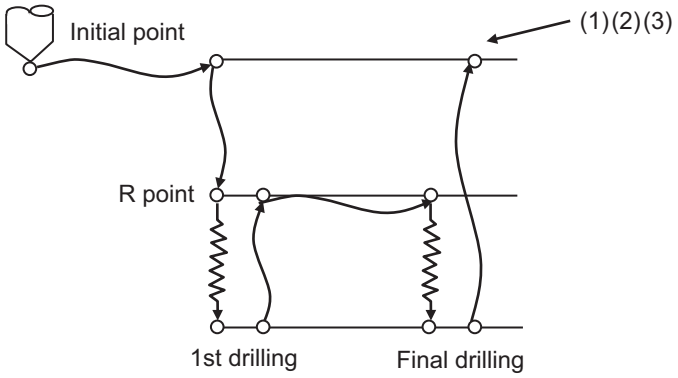
(2) After the spindle reverse speed reaches the commanded speed, the spindle rotates forward at the same initial point or R point.

(3) The next command is executed after the forward rotation speed is reached. If the spindle forward rotation speed does not reach the commanded speed yet, the drilling waits until the speed reaches the commanded value. The forward rotation speed is regarded to reach the commanded speed when the speed is 95% or more of the speed commanded for drilling.

[Initial point return]



[R point return]



If the NC is reset when the spindle is reversed, the rotation direction returns to the one commanded by the user PLC.

Precautions

(1) If the spindle is not rotating when drilling is performed, the chip removal operation is not performed.

(2) Rapid traverse overlap is temporarily disabled while the chip removal operation is being performed.

(3) The analog spindle is not compatible with this function.



Restrictions

(1) Even if multiple spindles are selected at the same time in combination with multiple spindle control II or multiple-axis synchronization control, only the spindle specified with the D command for chip removal is reversed. During the chip removal operation, multiple spindles are not reversed at the same time.

(2) Do not specify the synchronized spindle of spindle synchronization I or spindle synchronization II as the reversal spindle for chip removal.
If the synchronized spindle is commanded as the reversal spindle for chip removal, the machining program does not proceed as it continues to wait for the spindle to reach the reverse speed.

13.1.22 Cutting Reduction Amount Specification Method

Function and purpose

Hole drilling can be performed by specifying the cutting reduction amount while the depth of cut in each cutting operation (step) is reduced in the fixed cycle where the cutting amount per time is specified and the workpiece is cut multiple times till the bottom of the hole. This allows you to specify a larger cut depth in shallow areas where the load is low, so it is possible to reduce the number of cutting passes and efficiently perform machining.

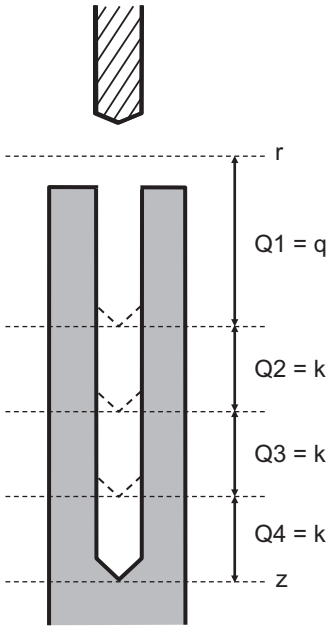
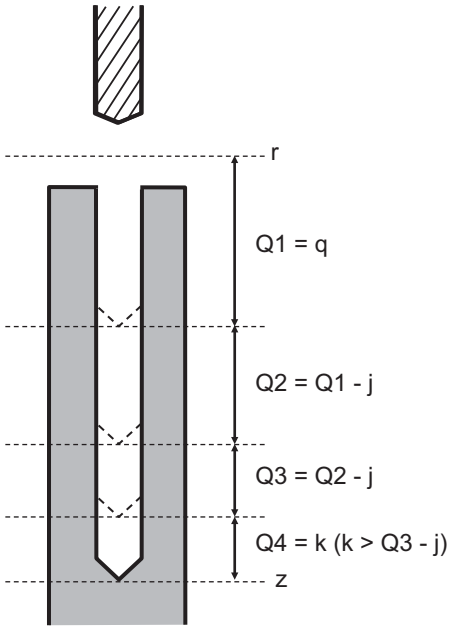
This function is available when the cut amount for each cutting pass (address Q) is commanded in the following fixed cycles.

- G83 (Deep-hole drilling cycle)
- G83 (Small-diameter deep-hole drilling cycle)
- G84 (Deep-hole/pecking tapping cycle)
- G73 (Stepping cycle)
- G74 (Deep-hole/pecking reverse tapping cycle)

This function is available in M800V/M80V series S/W version A9 or later.

Detailed description

The following two methods are valid depending on the cutting reduction amount and the minimum cutting amount.

(1) Cutting amount specification method	(2) Cutting amount reduction method
Specify the cut depth for the first step and the second and subsequent steps.	Specify the cutting reduction amount for the second and subsequent steps.
G83 Zz Rr Qq ,Kk Zz: Hole bottom position Rr: R point Qq: Cut amount for each cutting pass (cut depth for the first time) ,Kk: Minimum cutting amount (cut depth for the second and subsequent times)	G83 Zz Rr Qq Jj ,Kk Zz: Hole bottom position Rr: R point Qq: Cut amount for each cutting pass Jj: Cutting reduction amount ,Kk: Minimum cutting amount
	

Enabling conditions

Specify the cutting reduction amount and the minimum cutting amount with the machining parameters and command address to enable each method.

Cutting reduction amount: "#19444 Drill cut reduce", Address "J"

Minimum cutting amount: "#19445 Drill cut min.amt", Address ",K"

(1) Cutting amount specification method

This method is valid if all of the following conditions are satisfied.

- Minimum cutting amount is specified (other than "0").
- The minimum cutting amount is smaller than the cut amount for each cutting pass (address Q).

(2) Cutting amount reduction method

This method is valid if all of the following conditions are satisfied.

- Cutting reduction amount is specified (other than "0").
- The cutting amount specification method is valid.

Also, even when the cutting reduction amount has been specified, if the enabling conditions for the cutting amount specification method are not satisfied as shown below, the program error occurs.

- If the minimum cutting amount is "0", the program error (P33) occurs.
- If the minimum cutting amount is greater than the cut amount for each cutting pass, the program error (P35) occurs.

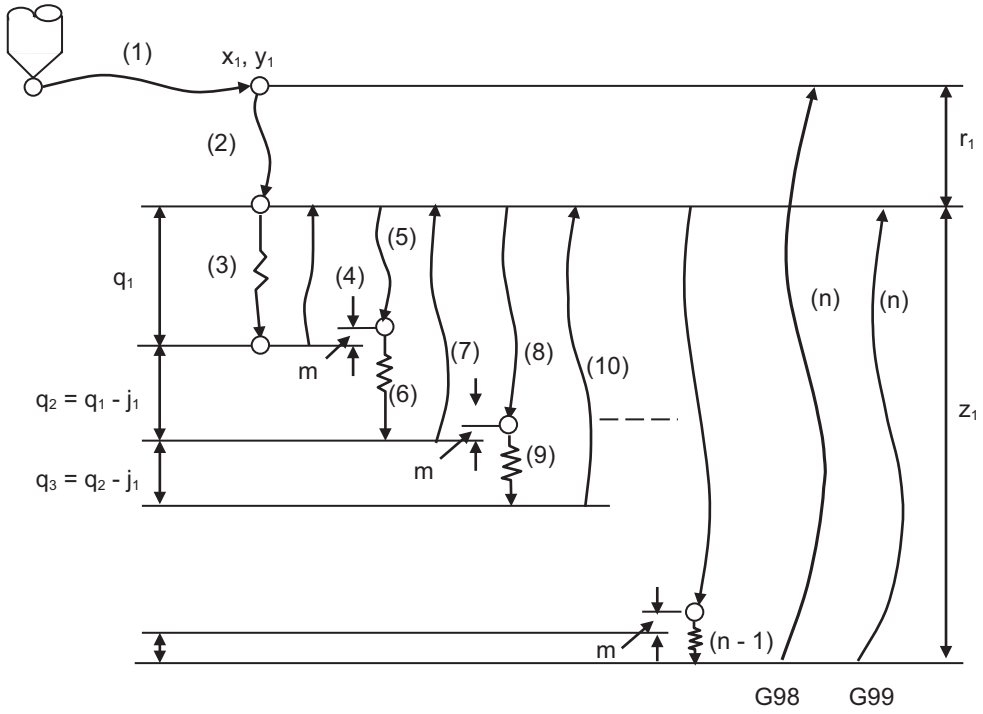
Cutting amount command address

The cutting amount command addresses are "Q" (cut amount for each cutting pass), "J" (cutting reduction amount) and ",K" (minimum cutting amount), and these are modal data.

Operation example

The following example shows how the cutting amount reduction method operates in the G83 deep-hole drilling cycle.

G83 Xx1 Yy1 Zz1 Rr1 Qq1 Ff1 Jj1 ,Kk1;



Operation pattern	Program	
(1)	G00 Xx1 Yy1 ;	
(2)	G00 Zr1 ;	
(3)	G01 Zq1 Ff1 ;	
(4)	G00 Z-q1 ;	
(5)	G00 Z[q1-m] ;	
(6)	G01 Z[q2+m] Ff1 ;	(q2 ≥ k1)
(7)	G00 Z-[q1+q2] ;	
(8)	G00 Z[q1+q2-m] ;	
(9)	G01 Z[q3+m] Ff1 ;	(q3 ≥ k1)
(10)	G00 Z-[q1+q2+q3] ;	
:		
(n)	G00 Z-[z1+r1] ;	G98 mode
	G00 Z-z1 ;	G99 mode

In the first step, the cut depth is "q1". In the second and subsequent steps, the cut depth decreases by "j1" each time a cutting is made. In the Nth step, the cut depth "qN" is "q1 - j1 * (N-1)".

If the cut depth "qN" in the second and subsequent steps falls below "k1", the cut depth is set to "k1". However, when the final cut is made to the hole bottom position, the length from the previous cut position to the hole bottom position is set as the cut depth, regardless of the value of "k1".

The commands and operations other than "j1" and "k1" related to the cut depth "qN" follow the explanation for each fixed cycle.

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

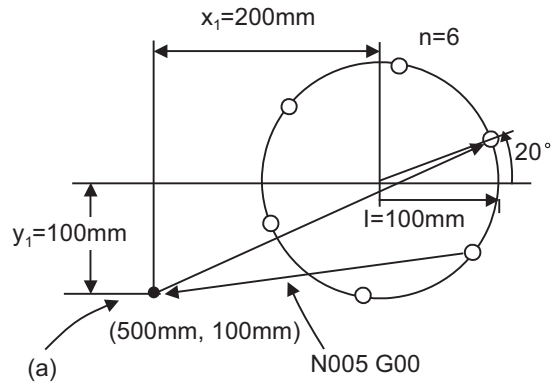
This function is used to drill "n" holes, dividing the circumference by "n", on the circumference with radius R centering the coordinates designated with X and Y. The drilling starts at the point which makes the angle θ with X axis. The hole drilling operation at each hole will follow the standard fixed 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 Jθ 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.
Jθ	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 position. Thus, use of the absolute 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

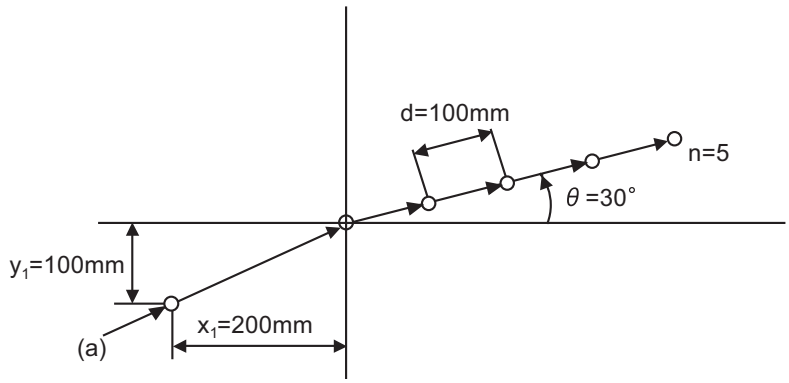
Using the position designated by X and Y as the start point, the n holes will be drilled with interval d in the direction which makes an angle θ with X axis. The hole drilling operation at each hole will follow the standard fixed cycle. The movement between hole positions will all be done in the G00 mode. G35 will not hold the data after the command is completed.

Command format

G35 Xx1 Yy1 Id Jθ 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.
Jθ	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

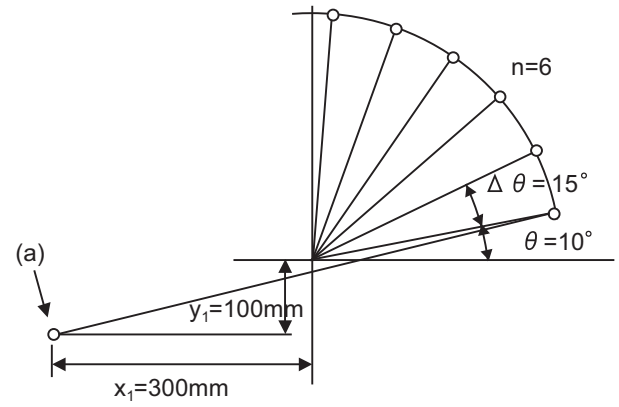
The "n" holes aligned with the angle interval Δθ will be drilled starting at the point which makes the angle θ with the X axis on the circumference with a radius R centering the coordinates designated with X and Y. The hole drilling operation at each hole will follow the standard fixed cycle.
The movement 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 Jθ PΔθ 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.
Jθ	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 Δθ	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

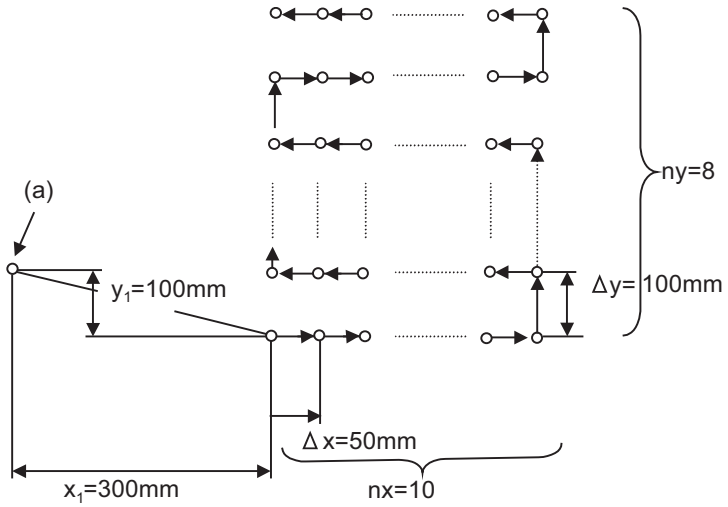
The nx points on a grid are drilled with an interval Δx parallel to the X axis, starting at the position designated with X, Y. The hole drilling operation at each hole will follow the standard fixed cycle.
The movement between hole positions will all be done in the G00 mode. G37.1 will not hold the data after the command is completed.

Command format

G37.1 Xx1 Yy1 IΔx Pnx JΔy Kny ;

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 nx 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.
Kny	No. of holes ny in the Y axis direction. The setting range is 1 to 9999.

Program example




```
G91 ;
G81    Z-10.000 R5.000 F20 ;
G37.1  X300.000 Y-100.000 I50.000 P10
       J100.000 K8 ;
```

(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

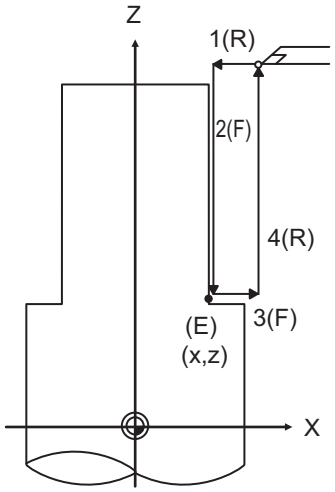
G174 X__ Z__ R__ F__ ;

X	X axis end point coordinate (*1)
Z	Z axis end point coordinate (*1)
R	Taper depth (radius designation, incremental position, 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)).

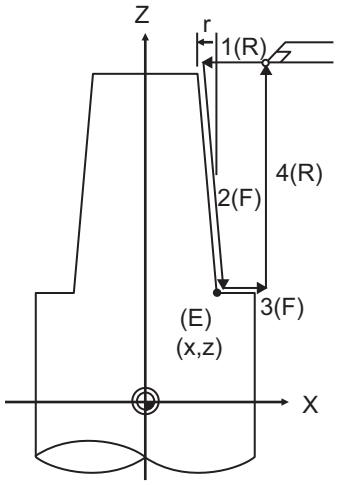
Detailed description

Straight cutting



(R) Rapid traverse (F) Cutting feed (E) End point coordinates

Taper cutting

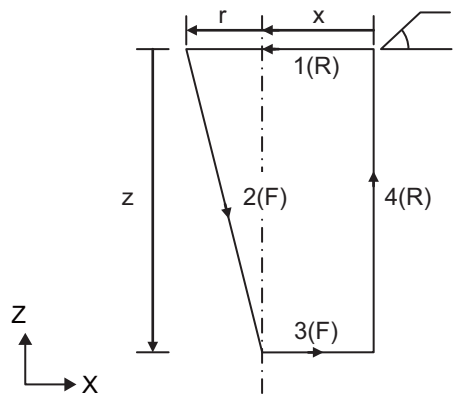
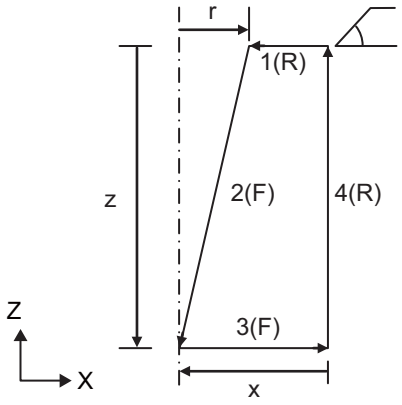
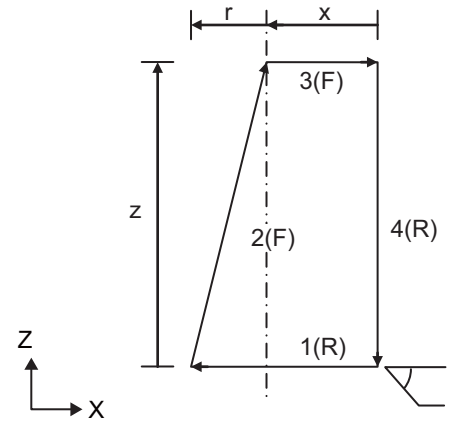
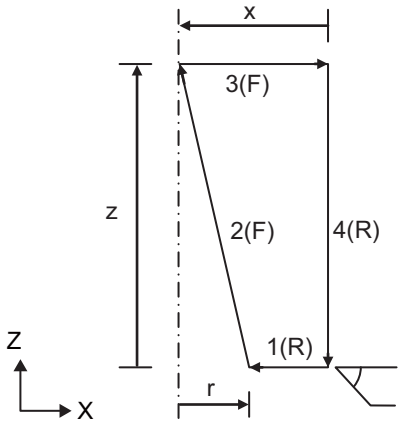


(R) Rapid traverse (F) Cutting feed (E) End point coordinates

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.

(a) z < 0, x < 0, r < 0	(b) z < 0, x < 0, r > 0
	
(c) z > 0, x < 0, r < 0	(d) z > 0, x <0 , r > 0
	

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__;
```

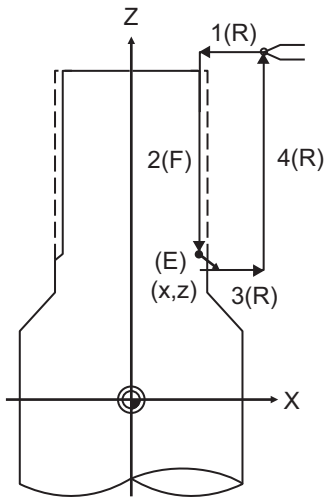
X	X axis end point coordinate (*1)
Z	Z axis end point coordinate (*1)
R	Taper depth (radius designation, incremental position, sign required)
F/E	Lead of long axis (axis which moves most) direction
Q	Thread cutting start shift angle

(*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

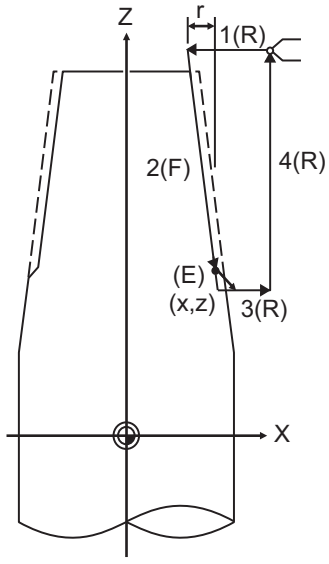
With a single block, the tool stops at the end points of operations 1, 3 and 4.



(R) Rapid traverse (F) Thread cutting cycle (E) End point coordinates

Taper thread cutting

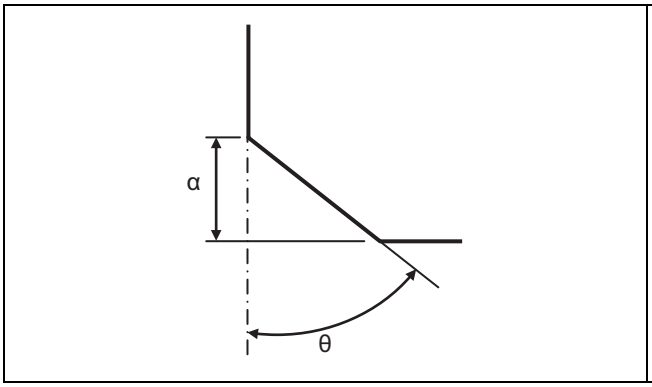
With a single block, the tool stops at the end points of operations 1, 3 and 4.



(R) Rapid traverse (F) Thread cutting cycle (E) End point coordinates

Detailed description

(1) Details for chamfering



α: Thread chamfering amount
This value is set in the parameter "#8014 CDZ-VALE". The available range is 0 to 12.7 leads. It can be set in 0.1L units.
Θ: Thread chamfering angle
This value is set in the parameter "#8015 CDZ-ANGLE". The setting range is 0 to 89°. It can be set in 1° units.

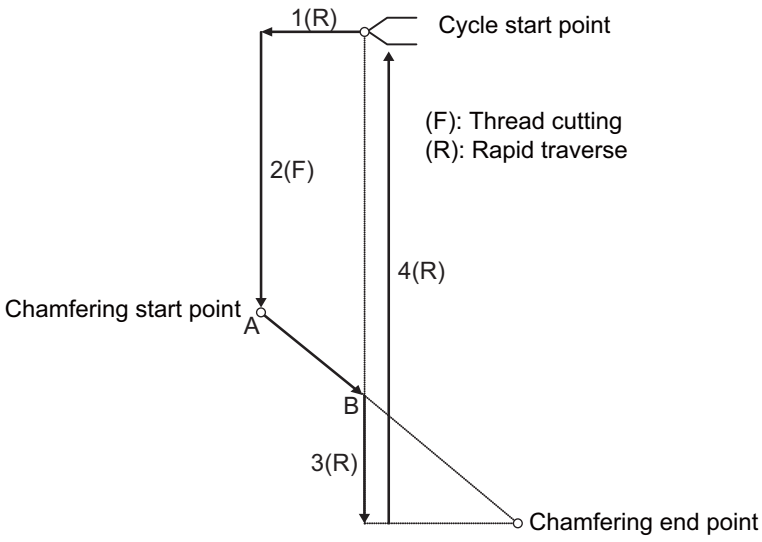
If thread chamfering amount is bigger than a thread lead length, a program error(P192) will occur before starting thread cutting.

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.

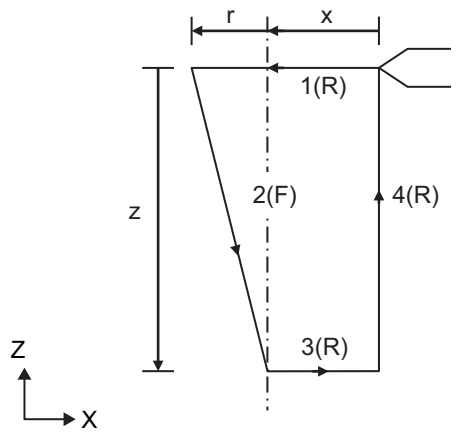
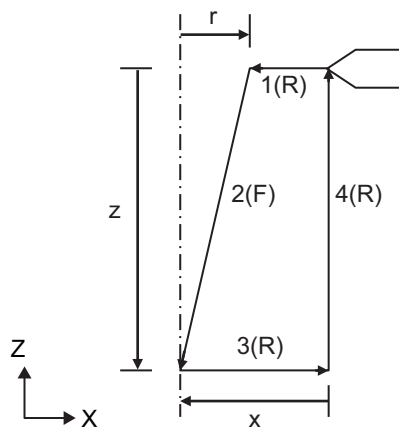
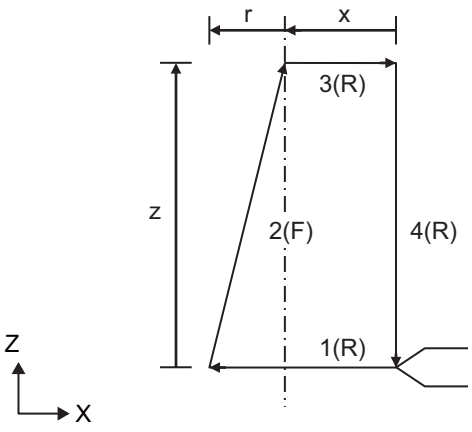
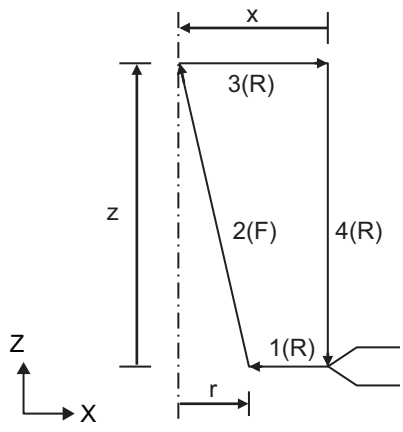
<#1270 ext06/bit4 = 1>
A program error (P192) will not occur. After the thread cutting tool retraction is stopped and the thread cutting block ends at the cycle start point ("A" in the figure), the axis moves to the end coordinate ("B" in the figure) at rapid traverse feed.

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.

(a) z < 0, x < 0, r < 0	**(b) z < 0, x < 0, r > 0**
	
(c) z > 0, x < 0, r < 0	**(d) z > 0, x < 0, r > 0**
	

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

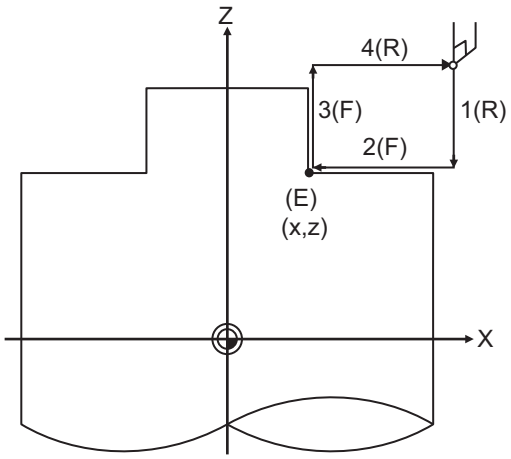
```
G176 X__ Z__ R__ F__ ;
```

X	X axis end point coordinate (*1)
Z	Z axis end point coordinate (*1)
R	Taper depth (radius designation, incremental position, 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)).

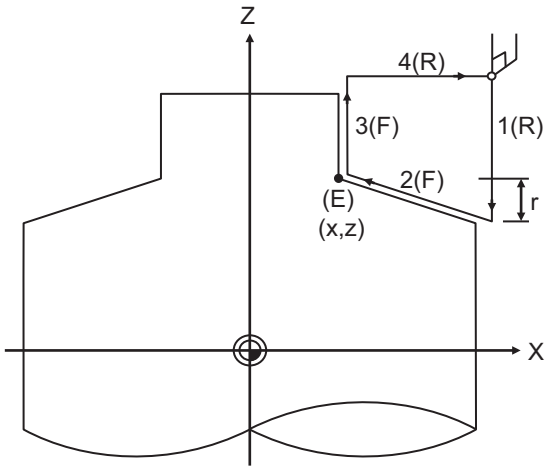
Detailed description

Straight cutting



(R) Rapid traverse (F) Cutting feed (E) End point coordinates

Taper cutting

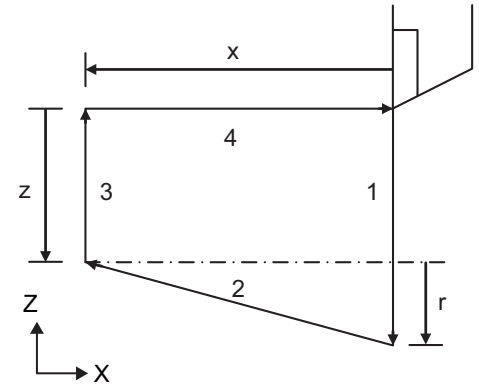
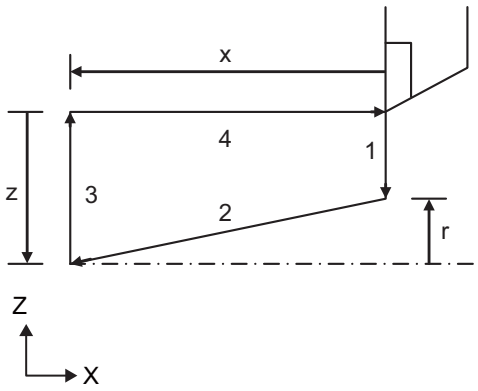
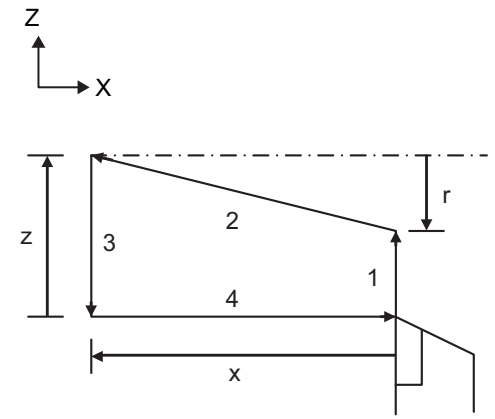
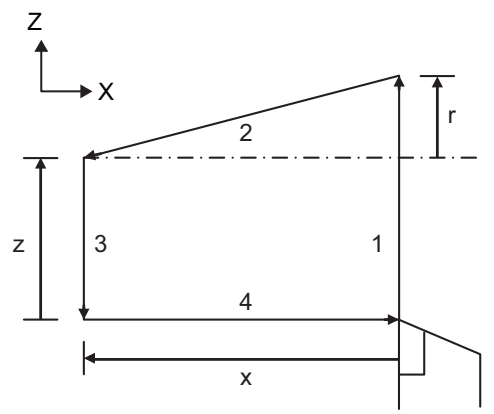


(R) Rapid traverse (F) Cutting feed (E) End point coordinates

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.

(a) z < 0, x < 0, r < 0	(b) z < 0, x < 0, r > 0
	
(c) z > 0, x < 0, r < 0	**(d) z > 0, x < 0, r > 0**
	

Program error (P191) will occur in (b) and (c) unless the following condition is satisfied.

$|z| \geq |r|$

14

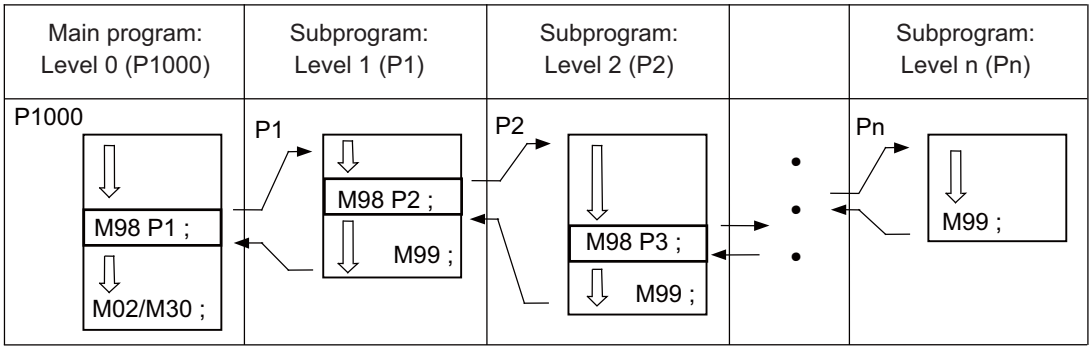
Macro Functions

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 operation (NC memory, NC memory 2, high-speed program server, SD card, hard disk or USB), and MDI 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	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".

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 commanded alone 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 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 memory, 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, the operation message will be displayed at registration.)

(5) 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

(6) 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

(7) To repeatedly use the subprogram, it can be repeated I1 times by programming M98 Pp1 LI1;.

(8) For a multi-part system, when there is no subprogram in the part system with the call command or when the subprogram is empty (EOR [%] only), the subprogram call operation changes according to the following parameters. (These parameters depend on the MTB specifications.)

#1285 ext21/ bit0	#1285 ext21/ bit2	Details
0	-	The subprograms registered in NC memory or NC memory 2 which is common for part systems is called.
1	0	The subprograms registered in NC memory or NC memory 2 of the selected part system is called. When there is not the specified subprogram in the selected part system, the subprogram of the first part system is called.
	1	The subprograms registered in NC memory or NC memory 2 of the selected part system is called.

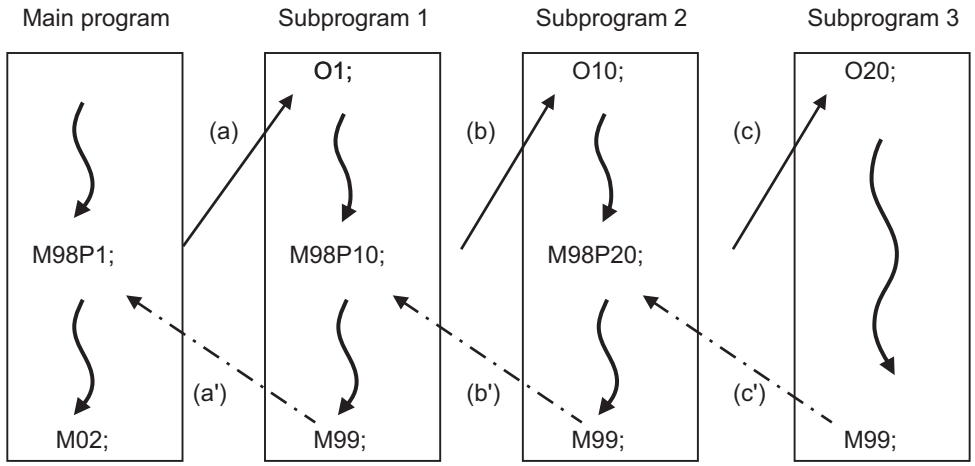
Subprogram calling/return block operation switching

The subprogram call command M98 (including the macro call command G65) is usually treated as an executable statement. It can be treated in the same manner as a macro statement depending on the MTB specifications. Refer to "14.4.7 Handling the Macro Call Command" for details.

Program example

Program example 1

When there are 3 subprogram calls (known as 3 nesting levels)



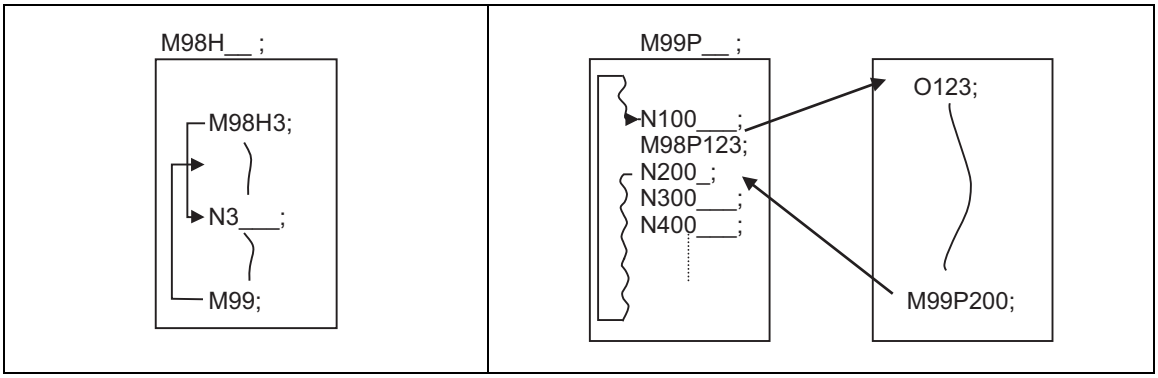
Sequence of execution: (a)-(b)-(c)-(c')-(b')-(a')

(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.

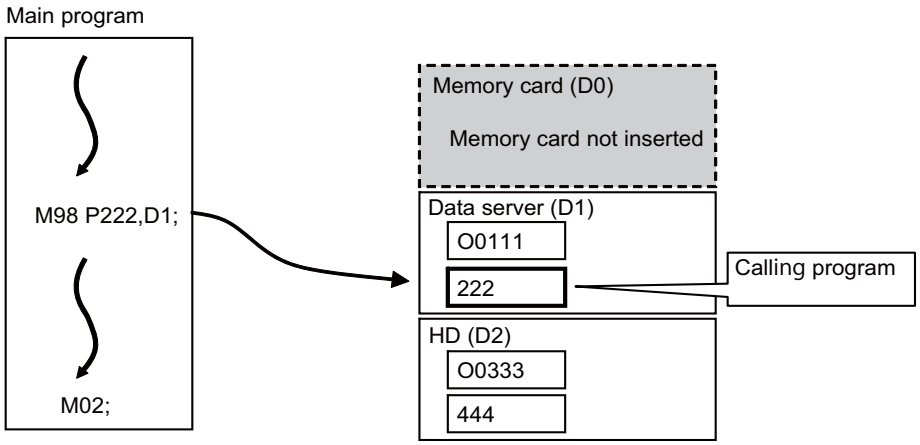
<Note>

- To prevent any unintended program call, avoid using program names that may be confused. (For example, 123, O0123 and O00000123 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 M8 series.)



[Parameter setting]

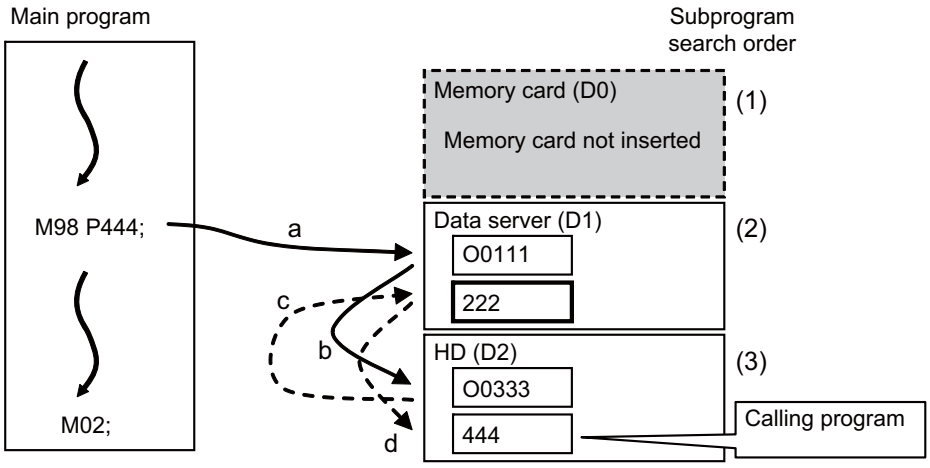
#8129 Subpro No. select = 1 (Four-digit program No. beginning with O No.)
#8880 Subpro stor D0 dev = R (Memory card)
#8882 Subpro stor D1 dev = D (Data server)
#8884 Subpro stor D2 dev = G (Hard disk)

(b) Without designation of device No.

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

(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 a device set with the parameter can be called as subprograms. To call a program in the device 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 the device to be called as a subprogram (Maximum 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. (Maximum 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

The registration destination (device and directory) of the program to be called with the M198 command can be switched with the parameter setting.

No.	Name	Details	Setting range (unit)
#8876	M198 pro stor: dev	Select the storage location (device) for the sub-program.	Drive name G: HD R: Memory card D: Data server N: USB memory
#8877	M198 pro stor: dir	Select the storage location (directory) for the subprogram.	Directory 48 characters

In subprogram call, the subprogram is searched for from the device and directory selected with these parameters. The following devices can be selected.

Setting value of #8876	Device	Drive
D	Data server	[M800VS/M80V] SD card on the back side of display unit [M800VW/M80VW] SD card in control unit
G	Hard disk	[M800VS/M80V] Local disk in IPC [M800VW/M80VW] Local disk in personal computer unit
R	Memory card	SD card on the front side of display unit
N	USB memory	USB memory on the front side of display unit

Note

- When the called subprogram is not in the selected storage location, a program error occurs.
- When #8876 is not set, which device to be searched depends on the model.
  M800VS/M80V Series: Memory card
  M800VW/M80VW Series: Data server
- For M800VW/M80VW Windows-based display units, "G" (HD) can be set.
  When a program in IPC connected to network is executed in M80V, "G" (HD) can be set.

(1) Up to 10 levels of programs can be called from the subprogram. If it is exceeded, the program error (P230) occurs.

(2) When "D" (Data server) is selected in the parameter #8876, the section from the head of the program to the first LF (*1) 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.

(*1) It is a line feed code "0x0A".

(3) 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 systems are reset when the error occurred, the programs will be displayed as only "%" except for the first part system.

(4) Refer to "14.1.1 Subprogram Call; M98, M99" for calling the subprogram by <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

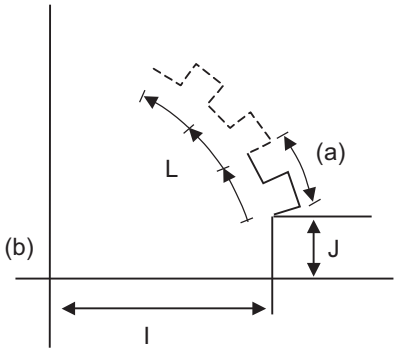
Subprogram call command

```
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 operation (NC memory, NC memory 2, high-speed program server, SD card, hard disk or USB), and MDI 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 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".

Detailed description



(a) Basic figure
(b) Center of rotation

(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 command and incremental command can be used in the figure rotation subprogram. Even if commanded with an absolute command, the rotation will be the same as when commanded with an incremental command.

(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.

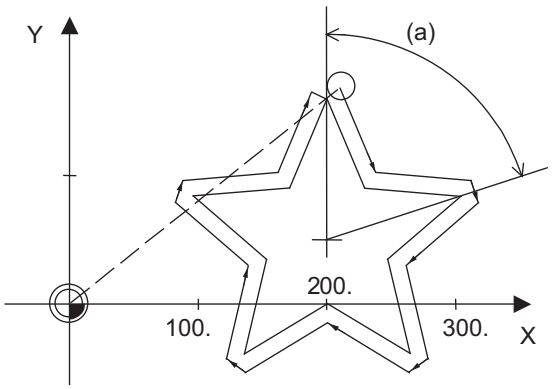
Program example

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 (P251) will occur if figure rotation is commanded during figure rotation.

(2) Figure rotation and program coordinate rotation cannot be commanded simultaneously. The program error (P252) 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 a reset is performed, the common variables can be set to <null> by setting the parameter ("#1128 RstVCl" and "#1129 PwrVCl").

Command format

#ΔΔΔ=○○○○○○○○ **;**

#ΔΔΔ = [Formula] ;

Detailed description

Variable expressions

		Example
#m	"m" indicates the value consisting of "0" to "9".	#100
#[f]	"f" indicates one of the following items 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) The program error (P241) occurs 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]

Kind of variables

The following table gives the kinds of variables.

The common variables are divided into the following two kinds.

Common variable 1: Variables that can be used commonly throughout the part systems.
Common variable 2: Variables that can be used commonly within the part system's program.

Kind		No.		Function
Common variable		Common variable 1	Common variable 2	♦Can be used commonly throughout main programs, sub programs, and macro programs. ♦When using common variables in the multi-part systems, the number of common variables shared between the part systems can be specified depending on the MTB specifications (parameter "#1052 MemVal"). (*2)
1 part system	200 sets	500 - 599	100 - 199	
	600 sets	500 - 999 100100 - 800199 (*4)	100 - 199	
	700 sets	400 - 999 (*1) 100100 - 800199 (*4)	100 - 199	
	8000 sets	400 - 999 (*1) 100100 - 800199 (*4) 900000 - 907399 (*3)	100 - 199	
Multi-part systems (n = number of part systems)	600 + 100 * n sets	400 - 999 (*1) 100100 - 800199 (*4)	100 - 199 *n	
	7900 + 100 * n sets	400 - 999 (*1) 100100 - 800199 (*4) 900000 - 907399 (*3)	100 - 199 *n	
Extended common variable				♦Can be used commonly throughout main programs, sub programs, and macro programs. This variable is common to all part systems. ♦Refer to "Kind of extended common variables" described later for details.
Extended common variable I		10000 - 49999		
Extended common variable II		10000 - 89999		
Extended common variable III		90000 - 97999		
Local variables		1 - 33		Can be used as local variables in macro programs.
System variables		1000 -		Application is fixed by system.
Fixed cycle variables		1 - 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 systems (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 variables, 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 variables 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 variable 1 (When "#1316 CrossCom" is set to "1")
Variable sets specification	600 sets (500 + 100 sets)	#100100 to #100199 (Equivalent to #100 to #199 of the 1st part system) #200100 to #200199 (Equivalent to #100 to #199 of the 2nd part system) #300100 to #300199 (Equivalent to #100 to #199 of the 3rd part system) #400100 to #400199 (Equivalent to #100 to #199 of the 4th part system) #500100 to #500199 (Equivalent to #100 to #199 of the 5th part system) #600100 to #600199 (Equivalent to #100 to #199 of the 6th part system) #700100 to #700199 (Equivalent to #100 to #199 of the 7th part system) #800100 to #800199 (Equivalent to #100 to #199 of the 8th part system)
	700 sets (600 + 100 sets)	
	8000 sets (7900 + 100 sets)	

(Example)

<1-part system>

#100100=200 ;	Equivalent to #100 = 200 ;
#200105=#100 ;	"200" is set in #200105.
#300110=#100100 ;	"200" is set in #300110.
#800199=#500120 ;	The variable value of #500120 is set in #800199.

<Multi-part system>

Each-part system common variables #100 to #199 in another part system can be used.

$1

#200100=-100 ;	"-100" is set in #100 of the 2nd part system.
#101=#200102 ;	The variable value of #102 of the 2nd part system is set in #101.
#300105=#200103 ;	The variable value of #103 of the 2nd part system is set in #105 of the 3rd part system.
#110=#500107 ;	The variable value of #500107 is set in #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.
- Even when the parameter "#1052 MemVal" is set to "1" (common variables common for part systems (number designation)), the designated number of variables becomes invalid, thus these common variables are handled in the same manner as when the parameter "#1052 MemVal" is set to "0".
- When the parameters "#1128 RstVCl" and "#1129 PwrVCl" are set to "1", the operation is as follows.
  "#1128 RstVCl"
  The common variables shared between the part systems equivalent to #100 to #199 of the reset part system are cleared.
  (Example) When resetting in 1st part system, #100100 to #100199 are cleared.
  When resetting in 2nd part system, #200100 to #200199 are cleared.
  "#1129 PwrVCl"
  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", the program error (P241) occurs.

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 is 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, #400 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 the variable is defined:

#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 the variable arithmetic formula is defined:

#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.".

(7) The significant digits of the variable are up to 15 decimal digits. Note that the arithmetic operation may cause an error.
For example, in the following formula, correct judgment cannot be made due to the error.

IF [#10 EQ #20]

Calculate with attention to the error when comparing variables. If the difference between the variables to be compared falls within the specified error range as in the following formula, consider them as equal. If the error is less than 0.01, describe the formula as follows.

IF [ABS [#10 - #20] LT 0.01]

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) occurs, 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) occurs.

(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 shared between the part systems, the variable values and variable names can be changed by the displayed part system.

Kind of extended common variables

Variable name/Number of variable sets	Variable No.	Function
Extended common variable I (10000 sets/40000 sets) (*1)	#10000 to #49999	⬧Extended common variable I includes variables that handle real values and variables that handle integer values. The number of each kind of variables is set by the parameter. ⬧When all variables are set to the real variable, up to 10000 sets can be used. When all variables are set to the integer variable, up to 40000 sets can be used. ⬧Integer variables can handle values from "-32768" to "32767". ⬧The variables are held even when the power is turned OFF. ⬧For the usage method, refer to "14.5.2.1 Extended Common Variable I/II".
Extended common variable II (80000 sets) (*1)	#10000 to #89999	⬧A part of program storage area allocated with the SRAM open parameter is used as variables. ⬧The variables handle real numbers. Up to 80000 sets are available. ⬧Set the SRAM open parameter number and the number of real type variables with the parameters. ⬧The variables are held even when the power is turned OFF. ⬧For the usage method, refer to "14.5.2.1 Extended Common Variable I/II".
Extended common variable III (2000 sets/8000 sets) (*1)	#90000 to #97999	⬧Variables are available in up to 8000 sets x 1000 files. Stored in the SD card (M800VW/M80VW Series) in the control unit or the SD card (M800VS/M80V Series) on the back of the display unit. (*2) ⬧Extended common variable III includes variables that handle real values and variables that handle integer values. The number of each kind of variables is set by the parameter. ⬧When all variables are set to the real variable, up to 2000 sets can be used. When all variables are set to the integer variable, up to 8000 sets can be used. The total number of real and integer variables is up to 8000 sets. ⬧Integer variables can handle values from "-32768" to "32767". ⬧For the usage method, refer to "14.5.2.2 Extended Common Variable III".

(*1) The number of variables for extended common variables I/II/III is determined by the parameter (depending on the MTB specifications).
[Extended common variable I]
Variables of the type of which the parameter setting value is not "0" are available.

Number of real type variables	#11823 ComVar1DecN
Number of integer type variables	#11824 ComVar1IntN
Maximum number of variables	(Number of real variables) x 8 + (Number of integer variables) x 2 ≤ 80000

[Extended common variable II]
When both parameters are not "0", extended common variable II is available.

Number of real type variables	#11826 exComVar2DecN
Maximum number of variables	Within the number of data items in the SRAM open area specified with "#11825 exComVar2SramOpen"

[Extended common variable III]
When the number of data files is not "0", variables of the type of which the number is not "0" are available.

Number of real type variables	#11827 exComVar3DecN
Number of integer type variables	#11828 exComVar3IntN
Maximum number of variables	(Number of real variables) x 8 + (Number of integer variables) x 2 ≤ 16000
Number of data files	#11829 exComVar3FileN

(*2) An extended common variable data file is generated on the SD card by setting the parameters #11827 to #11829 by the MTB.

Type of extended common variable

Extended common variable I/III provide the 8-byte real type and the 2-byte integer type.

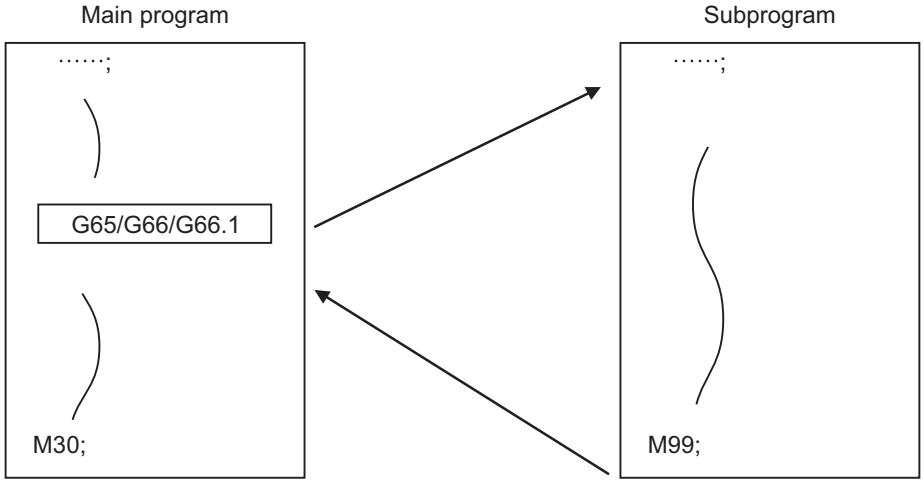
The extended common variable of 8-byte real type can represent <null>.
The extended common variable of 2-byte integer type cannot represent <null>. If <null> is entered, it is assumed to be "0". The values that can be set for the extended common variable of 2-byte integer type are from "-32768" to "32767". The numbers after the decimal point are rounded off.

ASCII codes can also be stored using an area of 2-byte integer type. A character string can be displayed on the custom screen by setting the ASCII code.

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))

Programs to be called by the macro call (G65/G66/G66.1) command are searched in the following order according to the setting of the parameter "#11053 UserProgramStorage" (User program storage destination).

#11053 UserProgramStorage	Storage device	Search order	
0 (Memory)	NC memory	NC memory	NC memory 2
	NC memory 2	NC memory 2	NC memory
	Other devices	NC memory	NC memory 2
1 (Hard disk)	All the devices	#12093 ExtSubProgram: dev #12094 ExtSubProgram: dir (Storage location of external device operation subprogram)	

In a multi-part system, when there is no macro program in the part system with the call command, the macro program call operation is switched depending on the parameter. (These parameters depend on the MTB specifications.)

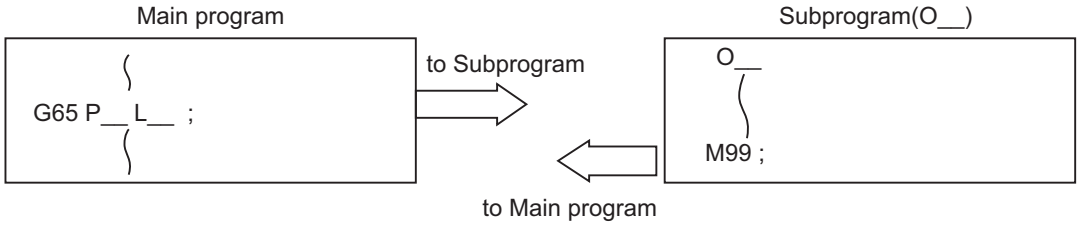
#1285 ext21/bit0	#1285 ext21/bit2	
	0	1
0	Searched according to the table of the storage device and search order above.	
1	Searched for the called program in the order of the selected part system to the first part system. The same device as the caller is searched in the order of the selected part system to the first part system. Then, the other device is searched.	The selected part system is searched. After the same device as the caller is searched, the other device is searched.

<Example> When "#1285 ext21/bit0" is set to "1" and "#1285 ext21/bit2" is set to "0"

When the storage device is NC memory 2, NC memory 2 is searched in the order of the selected part system and the first part system. Then, NC memory is searched in the order of the selected part system and the first part system.

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 ;

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	×	× (*1)
H	#11	○	○
I	#4	○	○
J	#5	○	○
K	#6	○	○
L	#12	×	× (*1)(*2)
M	#13	○	○
N	#14	×	× (*1)
O	#15	×	×
P	#16	×	× (*1)(*2)
Q	#17	○	○
R	#18	○	○
S	#19	○	○
T	#20	○	○
U	#21	○	○
V	#22	○	○
W	#23	○	○
X	#24	○	○
Y	#25	○	○
Z	#26	○	○

○: Available

×: Unavailable

(*1) Can be used while G66.1 command is modal

(*2) Can be used while the macro argument L/P valid function is enabled.

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

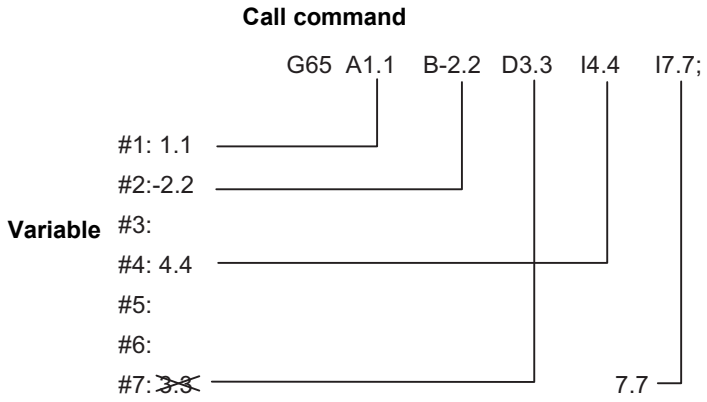
Note

(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.

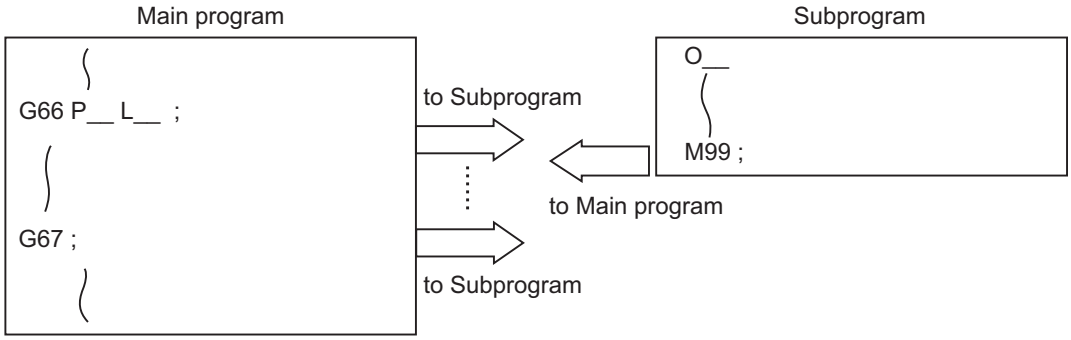
(2) If calling a subprogram numbered with O is enabled, a sub program number starting with O and specified by P command value is called with a priority.
However, when P command value is less than the digit number set with parameter "#8129 Subpro No. select", increase the digit number of command value 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 with 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

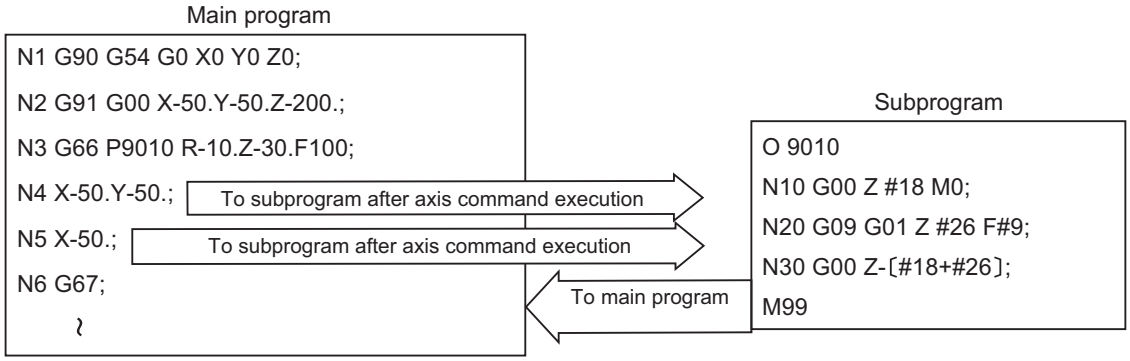
G67;

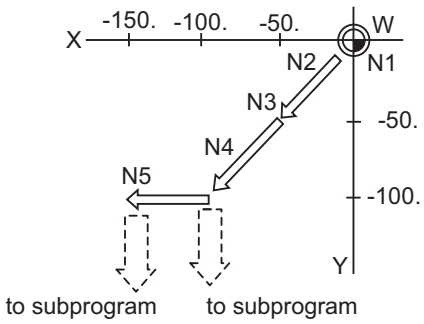
Detailed description

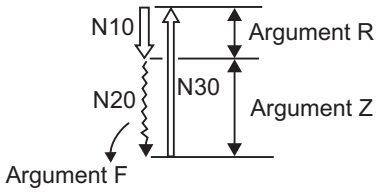
(1) When the G66 command is entered, the specified user macro program will be called after the movement command in a block with the movement commands has been executed, until the G67 (cancel) command is entered.

(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

Macro call via G code

```
G** P__ L__  argument ;
```

Macro call via G code with decimal point

```
G*.* P__ L__ ;
```

G** (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. (The parameter settings of the instructions depend on the MTB specifications.)

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 "G**" (or "G*.*") which conducts the macro call and the macro program number "P****" to be called is set by parameters.

(3) The number of available G codes is up to 538, which is total of the following ones.

- Among G01 through G9999, 10 G codes can be set individually, and 255 G codes by batch setting.
- Among G0.1 through G999.9, 10 G codes can be set individually, and 255 G codes by batch setting.
- Other 8 G codes: G200, G300, G400, G500, G600, G700, G800, G900

(4) G codes used in the system becomes available as G code macro when the parameter "#1081 Gmac_P" is set to "1". This parameter setting depends on the MTB specifications.

(5) 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.

(6) 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, the program error (P33) occurs.
This parameter setting depends on the MTB specifications (parameter "#1241 set13/bit5").

(7) The macro program to be called by the G code macro call needs to be registered in the NC memory or hard disk according to the parameter "#11053 UserProgramStorage". (This depends on the MTB specifications.)
When it is registered in a device other than the storage locations below, the program error (P232) occurs.

#11053 UserProgramStorage	Storage location
0 (Memory)	NC memory (NC memory 2 is not available.)
1 (Hard disk)	Storage location of external device operation subprogram (#12093, #12094)

Batch setting of macro call

(1) The batch setting of macro call via G code is available. The batch setting of up to 255 consecutive macro programs is possible by specifying the first G code, the first program number, the number of consecutive programs, and the type of macro call with the parameters. The same type applies to all macro programs. (It is the same for macro call via G code with decimal point.)

(Example 1) G code without decimal point

When the first G code is "152", the first program number is "7625", the number of consecutive programs is "10", and the type is "0".

G code	Equivalent command
G152	M98 P7625
G153	M98 P7626
G154	M98 P7627
:	:
G160	M98 P7633
G161	M98 P7634

(Example 2) G code with decimal point

When the first G code is "23.4", the first program number is "9012", the number of consecutive programs is "9", and the type is "2".

G code	Equivalent command
G23.4 argument	G66 P9012 argument
G23.5 argument	G66 P9013 argument
G23.6 argument	G66 P9014 argument
:	:
G24.1 argument	G66 P9019 argument
G24.2 argument	G66 P9020 argument

(2) The batch setting of macro call via G code needs to satisfy the following conditions. When the number exceeds the limit, the batch settings of macro call via G code are all invalid.

(a) G code without decimal point

- First G code + Number of consecutive programs - 1 ≤ 9999
- First program number + Number of consecutive programs - 1 ≤ 99999999

(b) G code with decimal point

- First G code + Number of consecutive programs / 10 - 0.1 ≤ 999.9
- First program number + Number of consecutive programs - 1 ≤ 99999999

(3) When setting of macro call via G code duplicates, the command is executed in the following priority from top to bottom.

(a) G code without decimal point

- Individual settings (#7201 to #7293)
- Batch settings (#7421 to #7424)
- Settings of G200, G300, ... , G900 (#7322 to #7393)

(b) G code with decimal point

- Individual settings (#56501 to #56593)
- Batch settings (#7431 to #7434)

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" and "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. Among M0 through M9999, 10 M codes can be registered individually, and the number of M codes specified by the parameter can be registered by batch setting. (Refer to the "Batch setting of macro call" section for batch setting.)
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 to 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) Even when the "Miscellaneous function lock" signal (YC5A) is ON, the macro call is performed.

(7) The macro program to be called by the miscellaneous command macro call needs to be registered in the NC memory or hard disk according to the parameter "#11053 UserProgramStorage". (This depends on the MTB specifications.)
When it is registered in a device other than the storage locations below, the program error (P232) occurs.

#11053 UserProgramStorage	Storage location
0 (Memory)	NC memory (NC memory 2 is not available.)
1 (Hard disk)	Storage location of external device operation subprogram (#12093, #12094)

(8) When the parameter "#7002 M[01] TYPE" or "#7442 Mmac[1] batch type" is set to "4" or "5", the macro call is not performed. (This setting value is only for L system.)

Macro argument L/P valid function

(1) 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.

(2) 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, the program error (P33) occurs.

(3) 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.

(4) 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, the program error (P33) occurs.
This parameter setting depends on the MTB specifications (parameter "#1241 set13"/bit5).

Batch setting of macro call

(1) The batch setting of macro call via M code is available. The batch setting of consecutive macro programs as many as specified is possible by specifying the first M code, the first program number, the number of consecutive programs, and the type of macro call with the parameters. The same type applies to all macro programs.

(Example)

When the first M code is "152", the first program number is "7625", the number of consecutive programs is "10", and the type is "0", the command is equivalent to the following command.

M code	Equivalent command
M152	M98 P7625
M153	M98 P7626
M154	M98 P7627
:	:
M160	M98 P7633
M161	M98 P7634

(2) The batch setting of macro call via M code needs to satisfy the following conditions. When the number exceeds the limit, the batch settings of macro call via M code are all invalid.

- First M code + Number of consecutive programs - 1 ≤ 9999
- First program number + Number of consecutive programs - 1 ≤ 99999999

(3) When setting of macro call via M code duplicates, the command is executed in the following priority from top to bottom.

- Individual settings (#7001 to #7093)
- Batch settings (#7441 to #7444, #7451 to #7454, #7461 to #7464)
- Smaller parameter number when the settings of the parameters #7441 to #7444, #7451 to #7454, and #7461 to #7464 duplicate

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

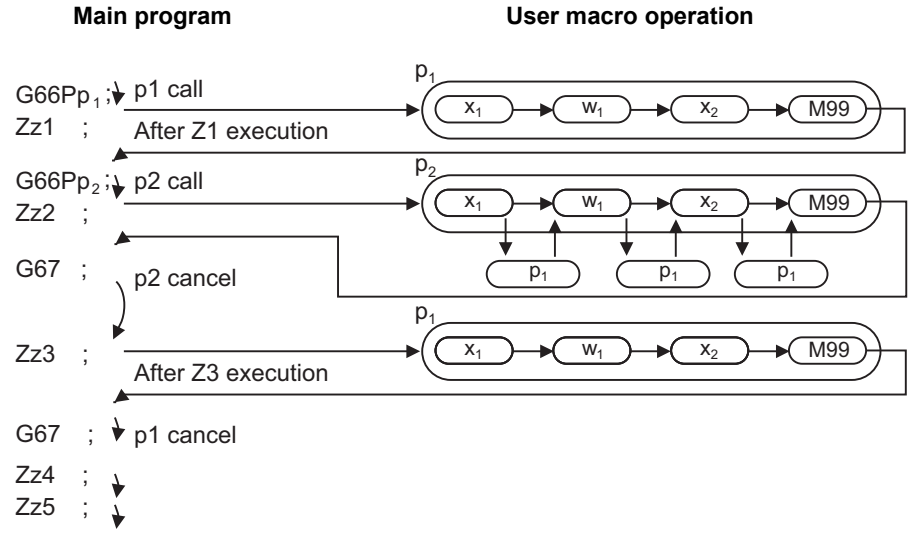
Up to 4 nesting levels are available for macro subprogram calls by simple call or modal call.
The argument for a macro call instruction is valid only within the called macro level. Since the nesting depth for macro calls extends up to 4 levels, the argument can be used as a local variable for the programs 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.

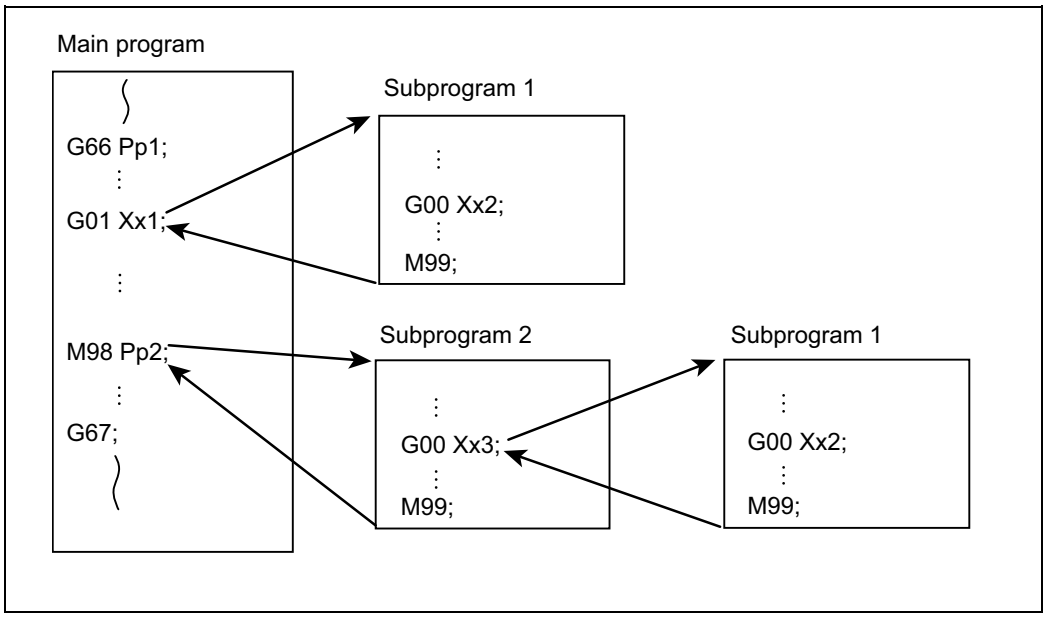
(2) With modal call A, the designated user macro subprogram is called every time a movement command is executed. However, when the G66 command is duplicated, the next user macro subprogram is called to movement commands in the macro every time an axis is moved.
User macro subprograms are called from 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 Handling the Macro Call Command

Function and purpose

Macro call commands (including the subprogram call command M98 and M198) are usually treated as executable statements; however, they can be treated in the same manner as macro statements by setting the parameter. A batch processing with commands at the call destination can be performed by treating as macro statements. Movement commands before and after the macro call command are processed continuously; therefore, continuous movement is possible without speed fluctuation.

The validity of this function depends on the MTB specifications (parameter "#1296 ext32/bit5").

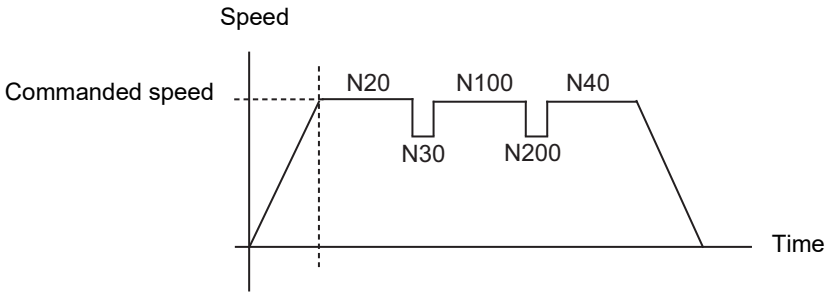
Detailed description

Example of differences depending on the setting of "#1296 ext32/bit5"

(1) Details of operation when "#1296 ext32/bit5" is set to "0" (Usual operation)

Call only the subprogram in the macro call command block.

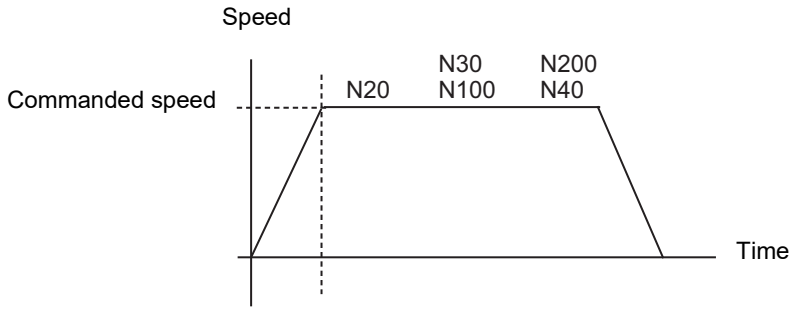
Main program	Subprogram	Operation
:		
N20 G01 X100. F1000 ;		
N30 G65 P100 ;		Call the subprogram "O100".
	O100	
	N100 X50. ;	Move to "X50." (execute "N100").
	N200 M99 ;	Return to the main program.
N40 X200. ;		Move to "X200." (execute "N40").
N50 M30 ;		



(2) Details of operation when "#1296 ext32/bit5" is set to "1"

Call the subprogram and execute the head block of the subprogram in the macro call command block.

Main program	Subprogram	Operation
:		
N20 G01 X100. F1000 ;		
N30 G65 P100 ;		(1) Execute as follows in "N30".
	O100	Call the subprogram "O100".
	N100 X50. ;	Move to "X50." (execute "N100").
	N200 M99 ;	(2) Execute as follows in "N200".
N40 X200. ;		Return to the main program.
		Move to "X200." (execute "N40").
N50 M30 ;		



Detailed description of the operation

(1) The parameter "#1296 ext32/bit5" is valid in the following macro call commands.

G65 Macro simple call (including MSTB macro)
G66.1 Macro modal call B (for each block)
M98 Subprogram call (including MSTB macro)
M198 Subprogram call (including MSTB macro)
M99 Return from subprogram

(2) When a command other than O, P, N, L and ",D" is issued in the subprogram call command block, the command is treated as an executable statement. The macro call command is treated as a macro statement even if any address is commanded. When the command is treated as an executable statement, the return command M99 is also treated as an executable statement.

(Example)
M98 P1000 G01 ; This is an executable statement because this includes the address G.

(3) The command is treated as an executable statement during single block operation.

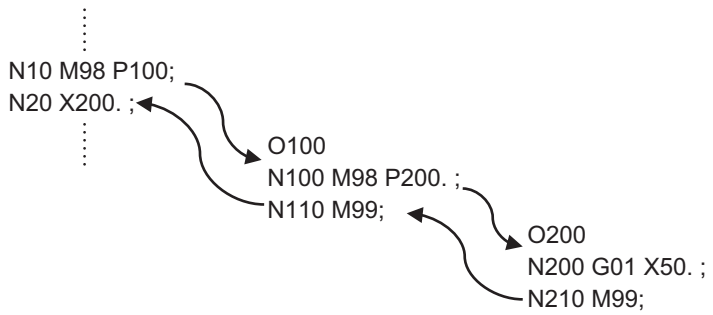
(4) The command is treated as an executable statement when the macro single is valid.

(5) The M99 command to repeat the main program is treated as an executable statement.

(6) This function is valid regardless of the nesting depth. When the macro call commands are successively issued (when the head block of the subprogram is the macro call command), the commands are treated as successive macro statements. However, in the following cases, the commands are treated as executable statements, but not as successive macro statements.

- ♦When the return command M99 from the subprogram is successively issued
- ♦When three or more macro call G65 or G66.1 commands are successively issued

(Example) When the return command from the subprogram is successively issued



Operation
(1) Execute as follows in "N10". Call the subprogram "O100". Call the subprogram "O200". Move to "X50." (execute "N200").
(2) Execute as follows in "N210". Return to the subprogram "O100".
(3) Execute as follows in "N110". Return to the main program. Move to "X200." (execute "N20").

(7) When the miscellaneous command macro call and G code macro call are commanded, the commands whose call type is M98, G65 or G66.1 are treated as macro statements, and the other commands are treated as executable statements.

(8) When the repetition is designated, only the last M99 is treated as a macro statement, and the other M99 commands are treated as executable statements.

(9) The command is handled as an executable statement in the following modes.

- ♦High-speed high-accuracy control II (G05P10000)
- ♦High-speed high-accuracy control III (G05P20000)
- ♦High-speed machining mode II (G5P2)

(10) If a block containing a G, N, O command or an axis name extension axis command is executed during G66.1 modal, the command is handled as an executable statement. Other commands are handled as macro statements.



Precautions

(1) When successive macro statements are commanded immediately before the macro call command is issued, a batch processing for the macro call command and the head block of the subprogram may not be performed depending on the number of macro statement blocks. (Macro statements in up to 32 blocks are processed in a batch.)

(2) When a program error occurs in the head block of the subprogram, the block where the program error occurs depends on the setting of the parameter "#1296 ext32/bit5".

When "#1296 ext32/bit5" is set to "0" (usual operation)

Main program	Subprogram	Operation
:		
N20 G01 X100. F1000 ;		
N30 M98 P100 ;		The program error (P34) occurs in "N30". (The error occurs after the subprogram "O100" is called.)
	O100	
	N100 G1234 ;	(Block with invalid G code)
	N200 M99 ;	
N40 X200. ;		
N50 M30 ;		

When "#1296 ext32/bit5" is set to "1"

Main program	Subprogram	Operation
:		
N20 G01 X100. F1000 ;		The program error (P34) occurs in "N20". (The error occurs after the "N20" block is completed.)
N30 M98 P100 ;		
	O100	
	N100 G1234 ;	(Block with invalid G code)
	N200 M99 ;	
N40 X200. ;		
N50 M30 ;		

(3) When the batch processing for the macro call command and the head block of the subprogram is performed in the tool radius compensation mode, movement path may differ from that when the batch processing is not performed.

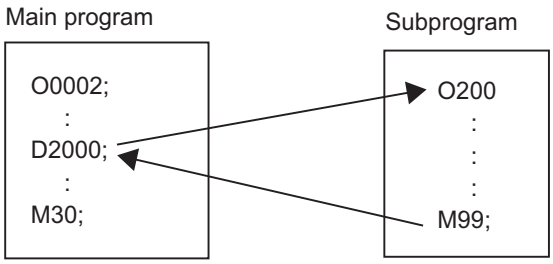
(4) When the movement amount of the block immediately before the macro call command is small, the speed may fluctuate in the macro call block.
In the same way, when the movement amount of the block immediately before the return block from the subprogram is small, the speed may fluctuate in the return block from the subprogram.

14.4.8 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

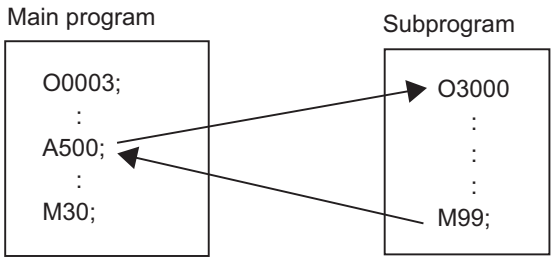


After outputting "2000" to common variable #146, the program No. 200 subprogram is called with the M98 subprogram call type.

Parameters

#7401 (ASCII call Valid/Invalid)	1 (Valid)
#7402 (ASCII code)	D
#7403 (Call type)	0 (M98 type)
#7404 (Program No.)	200
#7405 (Common 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 (Program No.)	3000
#7415 (Common 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>

♦The following addresses can be used.

A, B, D, F, H, I, J, K, M, Q, R, S, T

(4) 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")

(5) Register the macro program to be called by ASCII code macro in NC memory. When the program is registered to a device other than NC memory, including NC memory 2, the program error (P232) occurs.



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, M198 (subprogram call command)
M00 (program stop command)
M01 (optional stop command)
M02, M30 (end command)
M96, M97 (macro interruption command)

(2) When corresponding to miscellaneous code (M) call macro command

(3) When corresponding to ASCII code macro command

(4) Used as normal miscellaneous command

If "S", "T" and "B" are designated for the ASCII code address, commands will be identified with the following priority.

(1) When corresponding to miscellaneous code (S, T, B) call macro command

(2) When corresponding to ASCII code macro command

(3) 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)

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 PwrVCl".)

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.

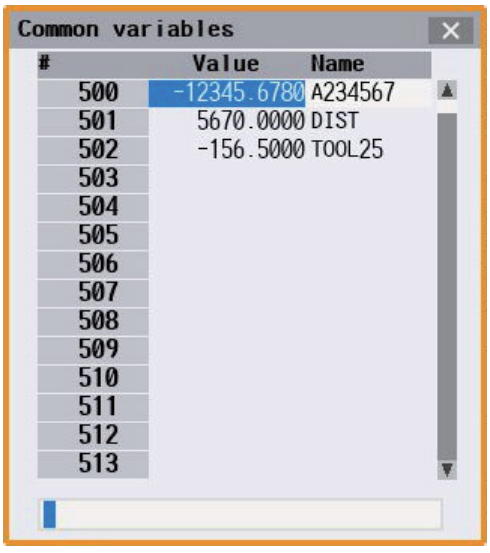
Variable name setting and quotation

Any name (variable name) can be given to common variables #500 to #599. It must be composed of not more than 7 alphanumerics and it must begin with a letter. Do not use "#" in variable names. It causes an alarm when the program is executed.

SETVNn [NAME1,NAME2, ...] ;	
n	Head No. of variable to be named (500 to 599)
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] ;



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.

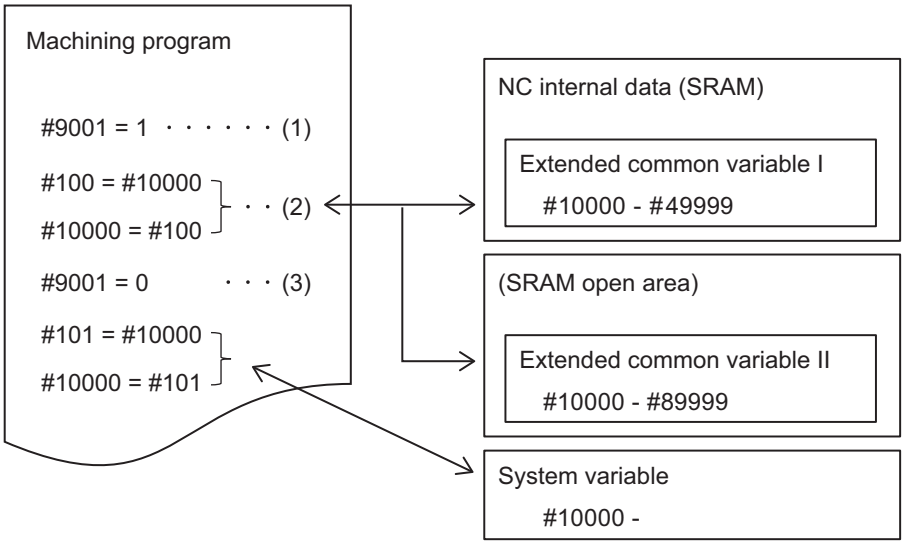
14.5.2 Extended Common Variable

14.5.2.1 Extended Common Variable I/II

Function and purpose

Extended common variables I/II (#10000 to #89999) are available in addition to common variables. Before using extended common variable I/II, it is necessary to set the system variable #9001 to "1" and switch to a mode in which extended common variable I/II can be commanded.

Reading/writing the extended common variable from the machining program



(1) Assign "1" to system variable #9001 from the machining program to set the mode in which extended common variable I or extended common variable II from #10000 to #89999 can be commanded.

(2) Extended common variables can be read from and written in the machining program.

(3) If "0" is assigned to system variable #9001, #10000 and subsequent variables are handled as system variables.

Detailed description

(1) Extended common variable I includes the real variables that handle real values and the integer variables that handle integer values. The number of real and integer variables depends on the MTB specifications. For details, refer to "Kind of extended common variables" in "14.2 Variable Commands".

(2) Extended common variables II all handle real values. The available number of variables depends on the MTB specifications. For details, refer to "Kind of extended common variables" in "14.2 Variable Commands".

(3) The extended common variable II is saved in the SRAM open area specified by the parameter "#11825 exCom-Var2SramOpen".

(4) The variable numbers of the extended common variables are assigned in the order of extended common variable I real number data, extended common variable II real number data, and extended common variable I integer data, starting from #10000.

(Example) Case where "1000" is set to the real number data for extended common variable I (#11823), "40000" to the real number data for extended common variable II (#11826), and "2000" to the integer data for extended common variable I (#11824).

#10000	Variables to store real number data of extended common variable I (1000 sets)
:	
#10999	
#11000	Variables to store real number data of extended common variable II (40000 sets)
:	
#50999	
#51000	Variables to store integer data of extended common variable I (2000 sets)
:	
#52999	

Note

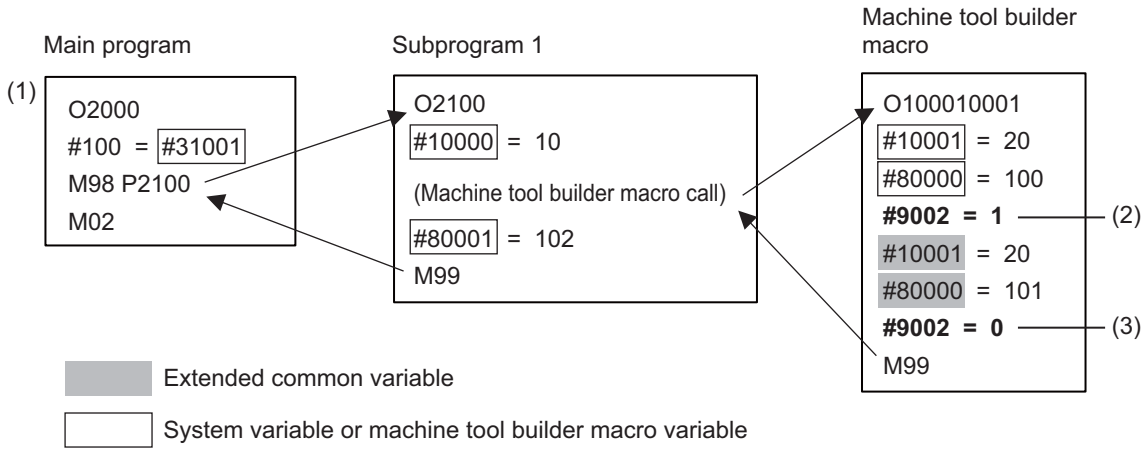
(1) The pre-reading of system variable #9001 is prohibited.

(2) If #9001 is commanded in the machine tool builder macro program, the program error (P241) occurs.

(3) System variable #9001 is set to "0" when the power is turned ON or reset.

Switching between extended common variables and system variables or machine tool builder macro variables (for machine tool builder macro program)

#10000 to #89999 are also used as system variables or machine tool builder macro variables. To use extended common variables in the machine tool builder macro program, switch them by system variable #9002.



(1) When the power is turned ON, system variable #9002 is "0". Commands #10000 to #89999 in the machine tool builder macro operate as system variables or machine tool builder macro variables.

(2) By the command "#9002 = 1", the system switches to a mode that uses extended common variables.

(3) By the command "#9002 = 0", the system switches to a mode that uses system variables or machine tool builder macro variables.

(4) System variable #9002 is set to "0" when NC is reset.

Note

(1) The pre-reading of system variable #9002 is prohibited.

(2) If system variable #9002 is commanded in the user program, the program error (P241) occurs.

14.5.2.2 Extended Common Variable III

Function and purpose

In addition to common variables, 8000 sets (#90000 to #97999) of extended common variable III can be used.

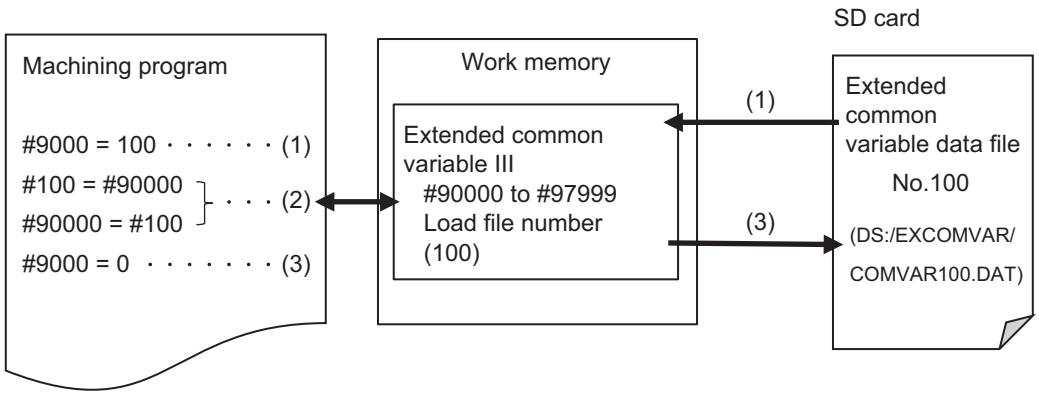
The extended common variable III is stored as the extended common variable data files on a dedicated SD card. The data files are read to the work memory using system variable #9000.

Up to 8 million sets (8000 sets x 1000 files) of variables can be used by preparing multiple extended common variable data files.

The extended common variable III is available only when the MTB specifications are enabled (parameters "#11827 exComVar3DecN", "#11828 exComVar3IntN", and "#11829 exComVar3FileN").

Detailed description

Reading/writing the extended common variable III from the machining program



(1) If the number (load file number) of the extended common variable data file saved on the SD card is set to the system variable #9000, the extended common variable data file can be copied (loaded) to the extended common variables III (#90000 to #97999) on the work memory.

(2) The variables on the work memory are read or written by the extended common variable III.

(3) By the command "#9000 = 0", the extended common variable III on the work memory is copied to (stored in) the extended common variable data file loaded in step (1). If you set a different load file number to # 9000 without the command "#9000 = 0", the value of the extended common variable III on the work memory is discarded and replaced with the value of the loaded extended common variable data file.

Details of extended common variable III

(1) The variable numbers of the extended common variable III are assigned in the order of the extended common variable III real number data, and the extended common variable III integer data, starting from #90000. The number of real and integer variables depends on the MTB specifications (parameters #11827 and #11828).

(Example) Case where "200" is set to the number of real variables (#11827) and "800" is set to the number of integer variables (#11828)

#90000	Variables to store real data (200 sets)
:	
#90199	
#90200	Variables to store integer data (800 sets)
:	
#90999	

(2) The extended common variable III, which is stored on the SD card, can contain up to 1000 files, regarding up to 8000 sets of variable data as one file. The number of files depends on the MTB specifications (parameter "#11829 exComVar3FileN" (number of extended common variable data files)).

(3) Before using the extended common variable III, you need to load the extended common variable data file on the SD card into the work memory area (extended common variable III) in the NC. The data file can be loaded by assigning the target extended common variable data file number to the system variable #9000.

(4) If the data file is written to the extended common variables III (#90000 to #97999) from the machining program, it is written to the work memory area (extended common variable III), so you need set "0" to the system variable #9000 to store the data file on the SD card.

(5) If you set or change the parameters (#11827 to #11829) that set the number of extended common variables III and the number of files, extended common variable data files are newly created at the next power-ON after all of them were deleted.
The real data of the newly created extended common variable data files is initialized with <null>, and the integer data of those files is initialized with "0".

14.5.2.3 Precautions for Extended Common Variable



Relationship with other functions

(1) Manual arbitrary reverse run

The manual arbitrary reverse run is prohibited for any of the following variables.

- Extended common variable III load file No.(#9000)
- Extended common variable III (#90000 to #97999)
- Extended common variable/system variable switching (#9001)
- Extended common variable/machine tool builder macro variable switching (#9002)
- Extended common variable I/II (#10000 to #89999)



Precautions

(1) Be sure to load the extended common variable data file in the machining program before using the extended common variable III. If the file is not loaded, the system works with the value of the last loaded variable after the power is turned ON.

(2) When the extended common variable data file is not loaded after power-ON, if the extended common variable III is used or if the variable store command "#9000 = 0" is executed, the program error (P247) occurs.

(3) Loading/storing the extended common variable III reduces the processing speed of other part systems, which may affect the machining accuracy. Do not load/store variables while other part systems are in machining operation.

(4) The system variable #9000 command, #9001 command, and #9002 command are pre-read prohibition commands that cancel the pre-read processing.

(5) A program error occurs in the following cases.

- A value outside the setting range was set to the integer-type extended common variable (P35).
- An extended common variable data file that does not exist was loaded (P246).
- A non-writable SD card was installed and the variable store command "#9000 = 0" was executed (P461).
- The number of assigned real and integer data items of variables included in the loaded extended common variable data file is different from the parameter (P248).

14.5.3 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]

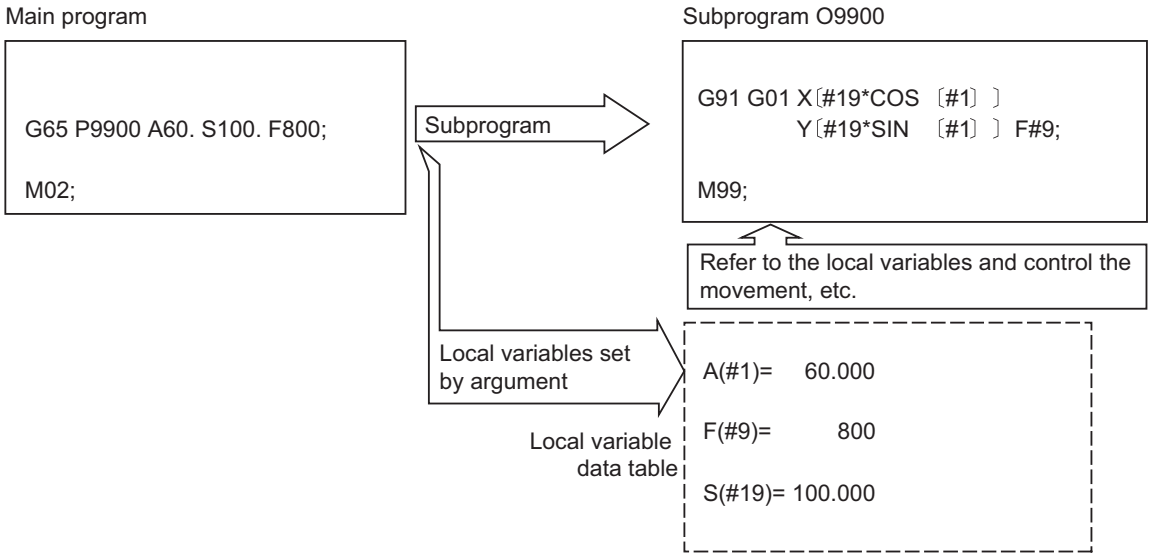
Argument designation II address	Variable in macro
A	#1
B	#2
C	#3
I1	#4
J1	#5
K1	#6
I2	#7
J2	#8
K2	#9
I3	#10
J3	#11
K3	#12
I4	#13
J4	#14
K4	#15
I5	#16

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

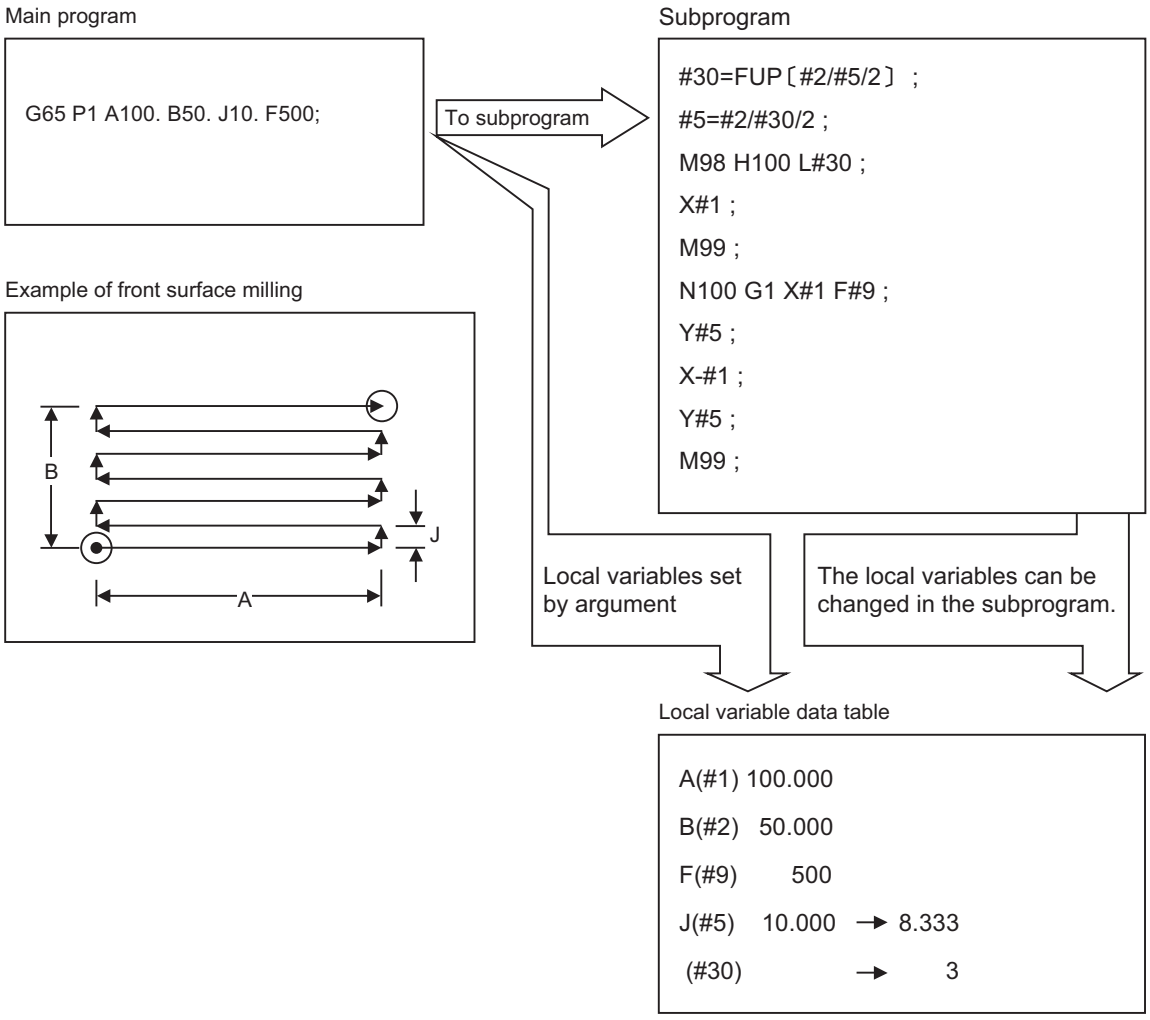
Note

- 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.)

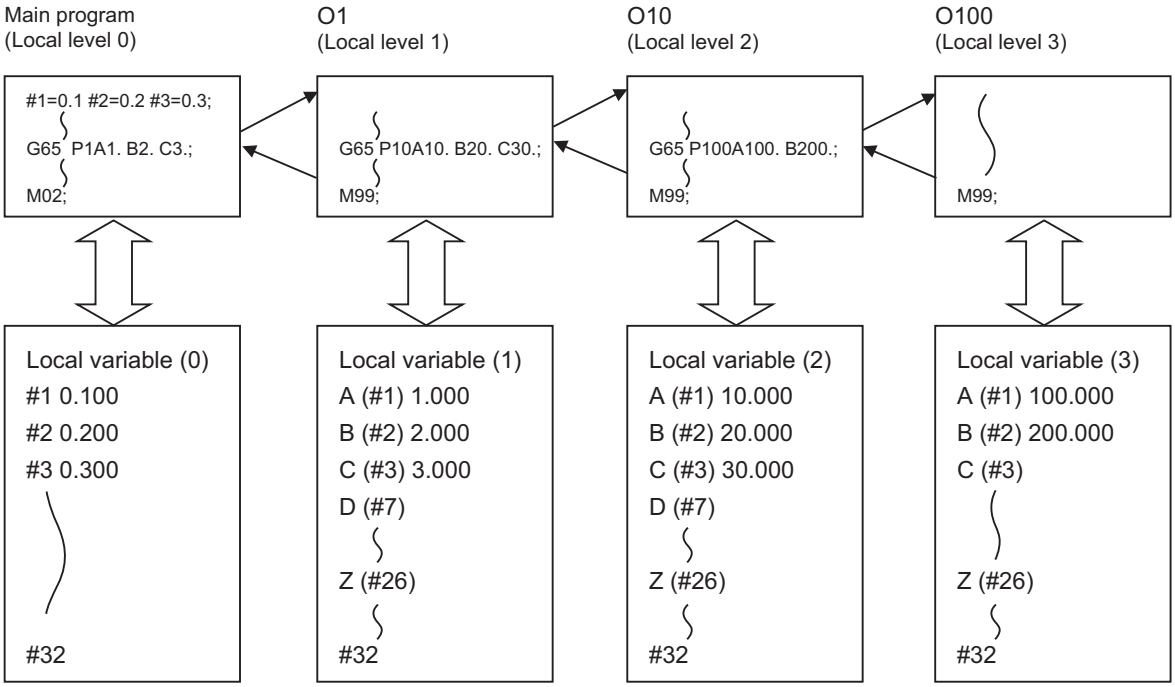


(2) The local variables can be used freely in that subprogram.



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.

14.5.4 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 [M8]

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 [ ].



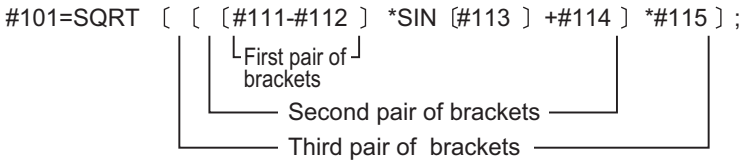
Detailed description

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.



(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.



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 #502 #503 #504 #505 #506	0.866 0.866 866.025 866.025 866.025 866.025
(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 #542 #543 #544 #545 #546	0.707 0.707 707.107 707.107 707.107 707.107

(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) Arctangent 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 fractions 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 [M8]	#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 0.01]

(2) If an operation command using a function is executed, a program error (P282) will occur when:

- A number that sets cosθ 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). [M8]
- A negative number has been set to argument 1 and a non-integer to argument 2 in the power command (POW). [M8]

(3) A method of processing macro operation (*1) can be changed. (This depends on the MTB specifications (parameter "1259 set31/bit7").)
When this is set to "0" (high speed), a high-speed macro operation is available because display update data will not be created.

(*1) This specifies whether to update the display data every time the certain number of macro blocks are executed for processing consecutive macro blocks.

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

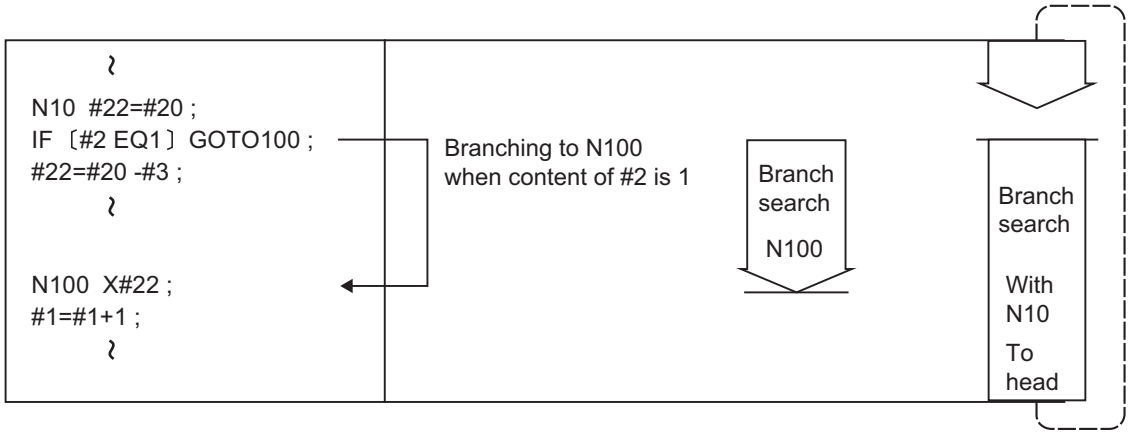
Branch (IF-GOTO-)

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 alone), 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).

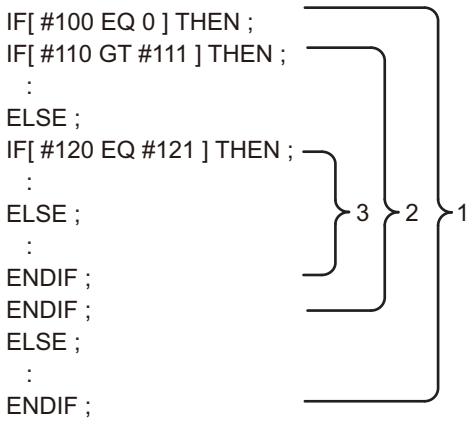
When ELSE-side processing is omitted:	IF[#100 EQ 0] THEN ; #100 = 2 ; G00 X#101 ; ENDIF ;
When THEN-side processing is omitted:	IF[#100 EQ 0] ELSE #110 =10 ;

(8) Formats (a) and (b) can be combined to issue commands.

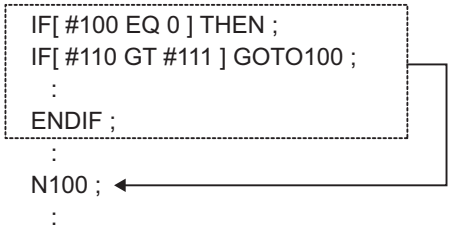
```
IF[ #100 EQ 0 ] THEN ;
#100 = 2 ;
G00 X#101 ;
ELSE #110 =10 ;
ENDIF ;
```

(9) If any of THEN, ELSE, and ENDIF is commanded with no IF command issued, a program error (P289) will occur.

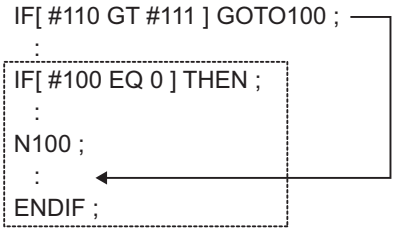
(10) The IF statement can be commanded up to 10 nesting levels.
When the nesting level exceeds 10, a program error (P288) will occur.
The following shows an example in which the nesting level is set to 3.



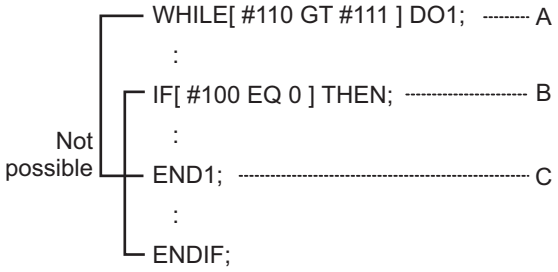
(11) You can set a branch from the inside of the IF to ENDIF range to the outside.



(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.

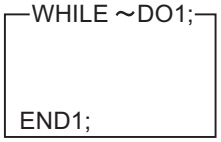
/ IF[#100 EQ 0] THEN #100 =10 ;	When "Optional block skip" signal is ON, IF statement will not be executed.

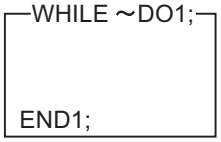
Repetitions (WHILE)

WHILE [conditional expression] DOm ; (m =1, 2, 3 ..... 127) :
:
END m ;

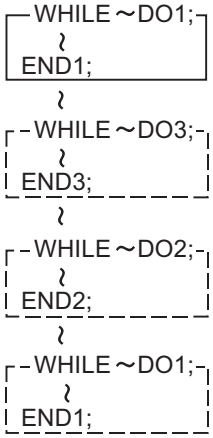
While the conditional expression is established, the blocks from the following block to ENDm are repeatedly executed; when it is not established, execution moves to the block following ENDm. DOm may come before WHILE. "WHILE [conditional expression] DOm" and "ENDm" must be used as a pair. If "WHILE [conditional 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.

(1) Same identification No. can be used any number of times.

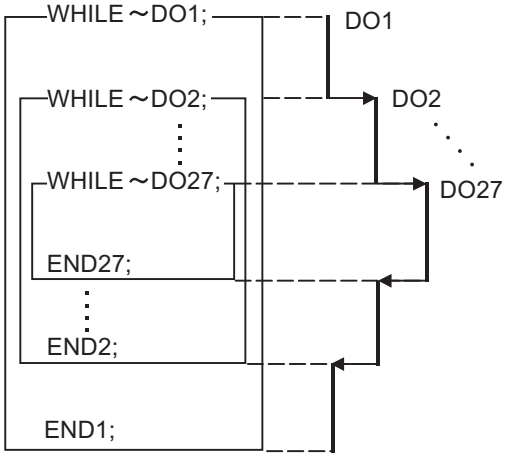




(2) Any number may be used as the WHILE-DOm identification No.



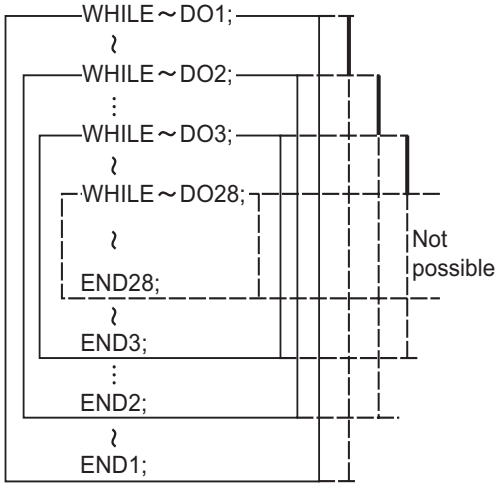
(3) Up to 27 nesting levels can be used for WHILE-DOm. Any number from 1 to 127 can be used for "m" as a repeating identification No. regardless of the nesting depth.



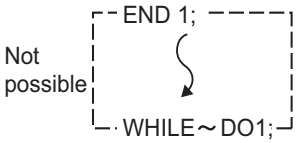
<Note>

•For nesting, "m" which has been used once cannot be used.

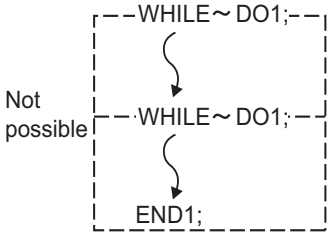
(4) The number of WHILE-DOm nesting levels cannot exceed 27.



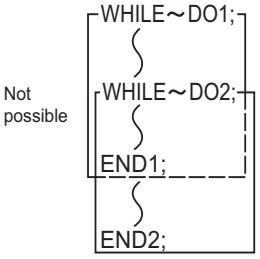
(5) WHILE - DOm must be designated first and ENDm last.



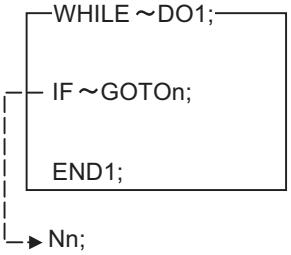
(6) WHILE - DOm and ENDm must correspond on a 1:1 (pairing) basis in a same program.



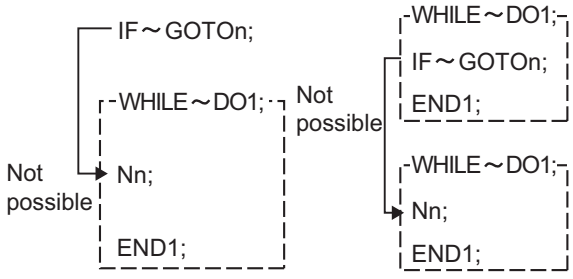
(7) Two WHILE - DOm's must not overlap.



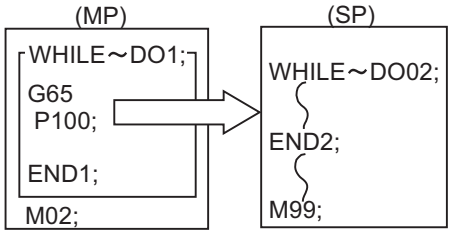
(8) Branching externally out of the WHILE - DOm range, is possible.



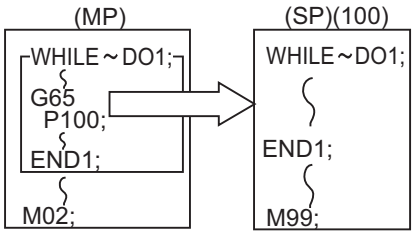
(9) No branching into WHILE - DOm, is possible.



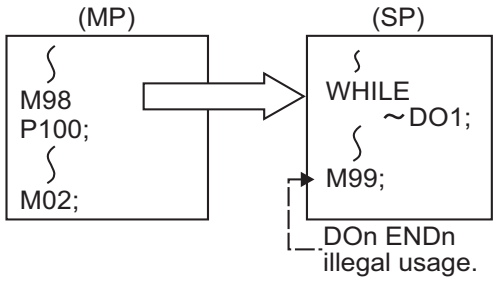
(10) Subprograms can be called by M98, G65 or G66 between WHILE - DOm's.



(11) Calls can be initiated by G65 or G66 between WHILE - DOm's and commands can be issued again from 1. Up to 27 nesting levels are possible for the main program and subprograms.



(12) A program error will occur in M99 if WHILE and END are not paired in the subprogram (including macro subprogram).



(MP) Main program (SP) Subprogram

Note

- Even if a fixed cycle containing WHILE is called, the nesting level will be counted up.

Precautions for control commands

(1) Depending on the settings of the following parameters, processing time reduction effect of macro commands becomes higher.

- #1754 cfgPR04/bit6 (Reduce processing time for block consisting only of sequence number/EOB)
- #1259 set31/bit7 (Macro processing method)

Processing time reduction effect (In the order from higher effect)	Parameter		
	#1754/bit6	#1259/bit7	#8101
1	1	0	0
2	1	1	0
3	0	- (*1)	0
4	- (*1)	- (*1)	1

(*1) The parameter value does not affect the processing time reduction effect.

Note

- When the parameter "#8101 MACRO SINGLE" is "1", there is no processing time reduction effect.
- The settings of the parameters #1754 and #1259 depend on the MTB specifications.

(2) When the parameter "#1754 cfgPR04/bit6" is "1", a block with N number command only is processed in a batch together with the executable statement in the next and subsequent blocks, in the same way as macro statement. When the parameter "#1754 cfgPR04/bit6" is "1", do not specify a sequence number in a block with N number command only by the following functions. The function does not work because the specified sequence number is not detected.

- Collation and stop
- Data sampling
- Real-time data sampling

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

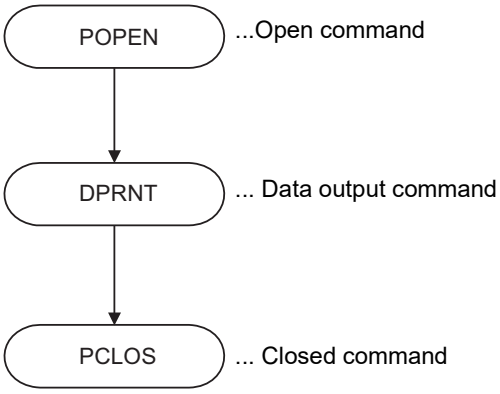
PCLOS

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 [l1#v1 [d1 c1] l2#v2 [d2 c2] .....] ;		
l1	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>

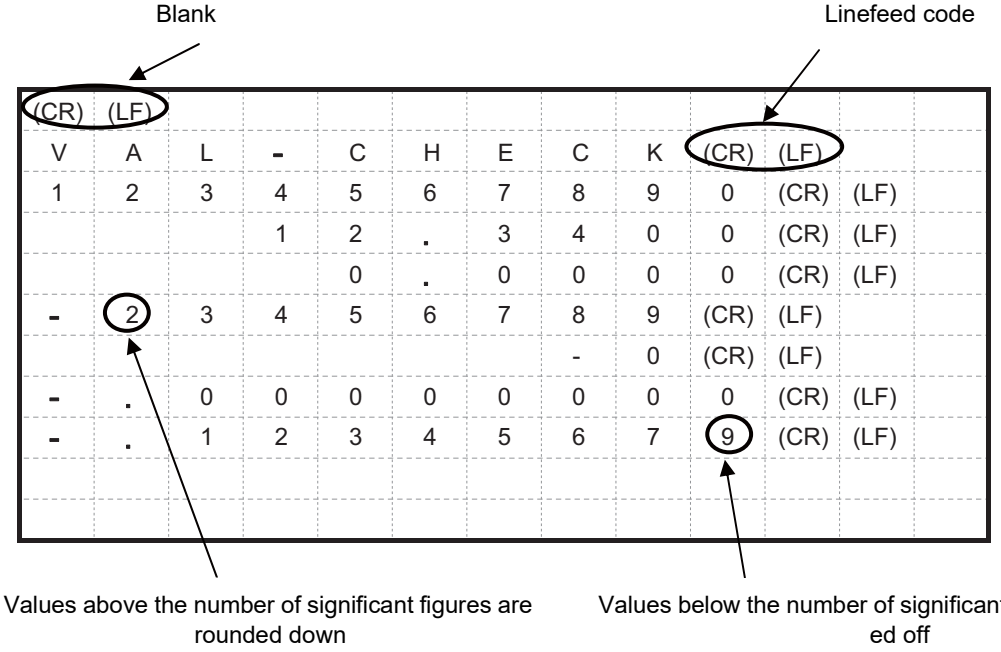
#1127 DPRINT (DPRINT alignment)	= 1 (Align the minimum digit and output)
#9007 MACRO PRINT PORT	= 9 (Output to a memory card by an external output command)
#9008 MACRO PRINT DEV.	= 0 (Device 0 is selected for an external output command)
#9054 MACRO PRINT FILE	= DPRNT_OUT (File name to store output data 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)>





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 M800V 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 combine conventional control commands such as movement commands and the M, S, T commands with macro commands such as the arithmetic, decision, branching for creating the machining programs. When the former commands are made into executable statements and the latter commands into macro statements, since macro statement processing is not directly related to machine control, it is an effective means for reducing machining time.
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 macro statements in a batch, and during a program check, set the parameter ON to execute the macro statements block by block. Setting can be chosen depending on the purpose.)

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 here refer to the following commands.

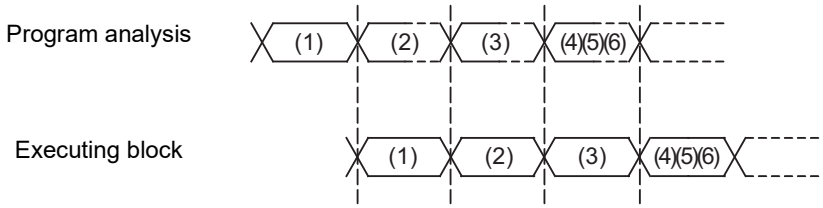
(a) Arithmetic commands (block including "=")
(b) Control commands (block including GOTO, DO-END, etc.)
(c) Macro call commands (Includes macro calls and cancel commands based on G codes (G65, G66, G66.1, G67).)

Execution statements refer to statements other than macro statements.

Flow of processing by the program example in the previous page

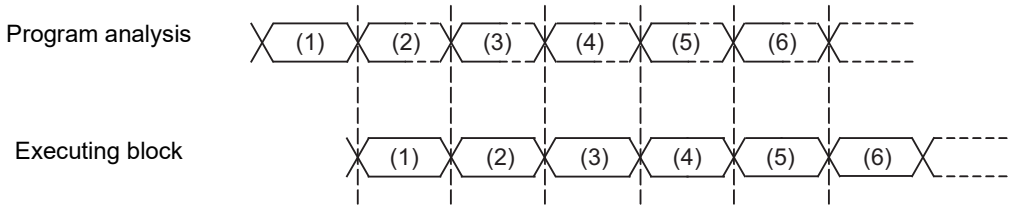
<Macro single OFF>

N4, N5 and N6 are processed in parallel with the control of the executable statement of N3. If the analysis of N4, N5, and N6 is in time during N3 control, the machine movement is 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. Therefore, 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.

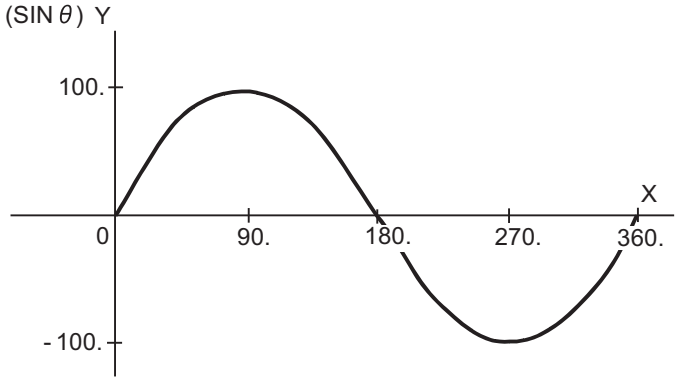
(Example 1) SIN curve

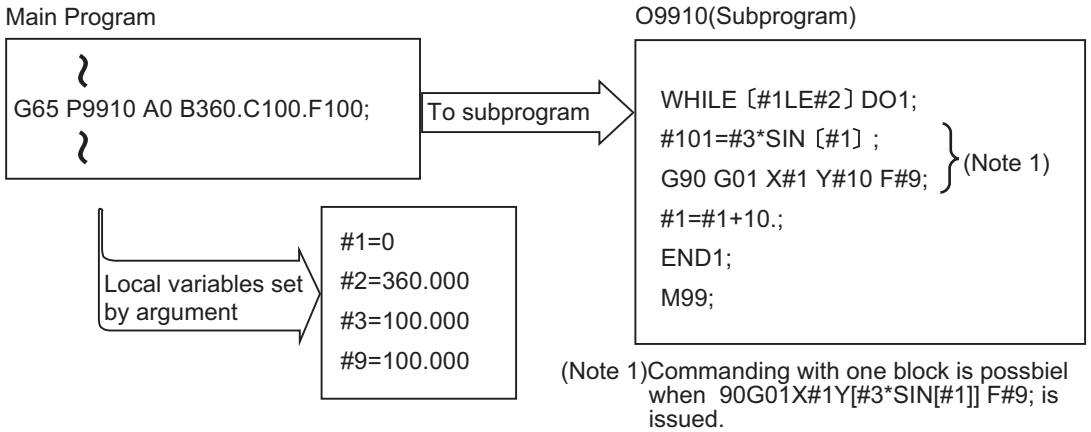
(Example 2) Bolt hole circle

(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

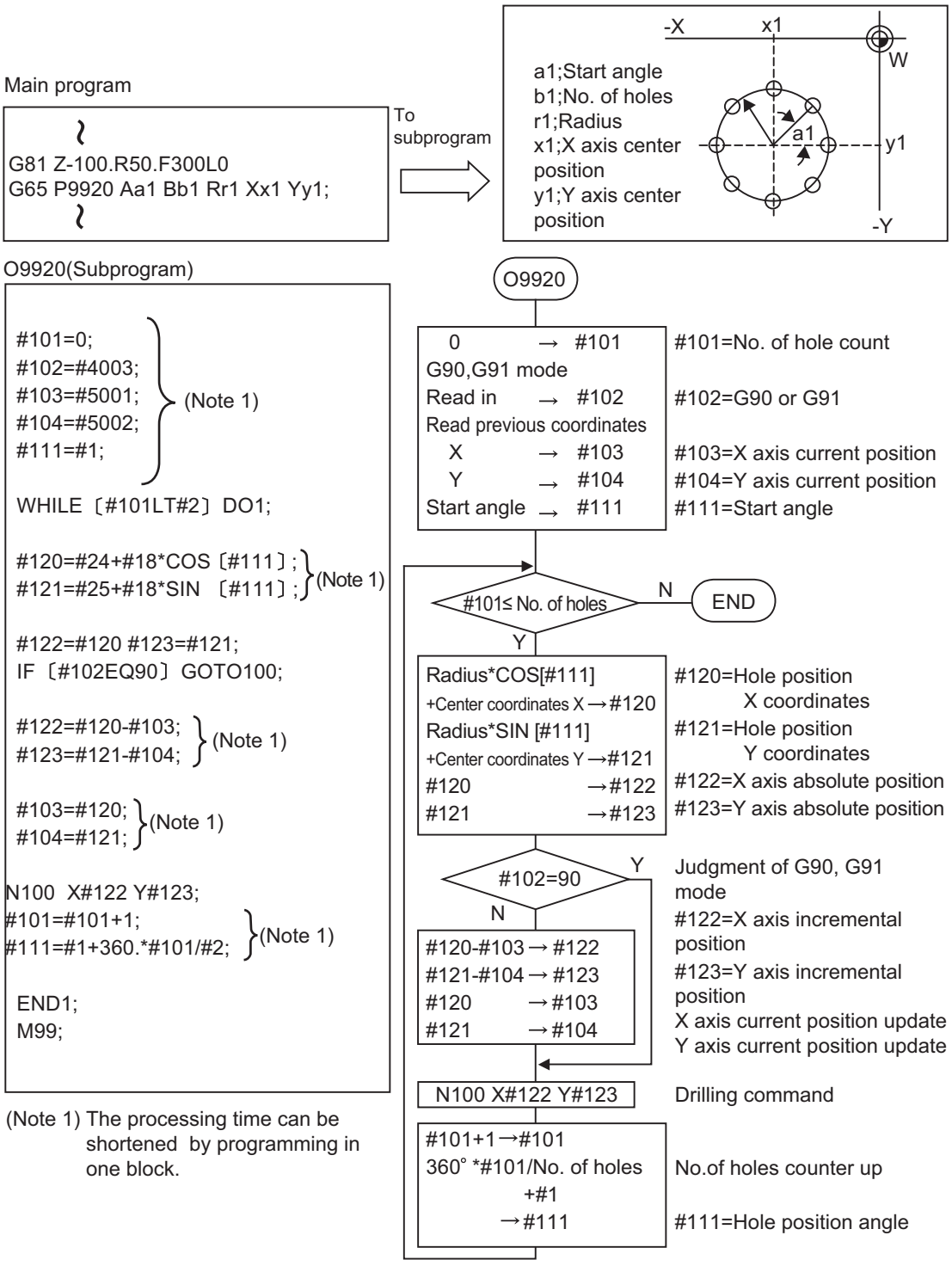


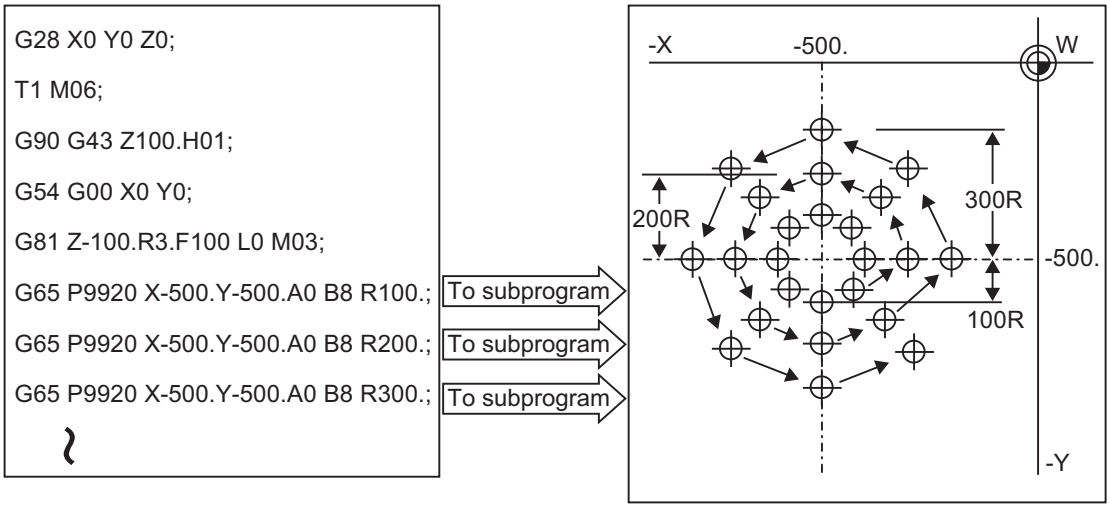


(Note 1)Commanding with one block is possbiel when 90G01X#1Y[#3*SIN[#1]] F#9; is issued.

(Example 2) Bolt hole circle

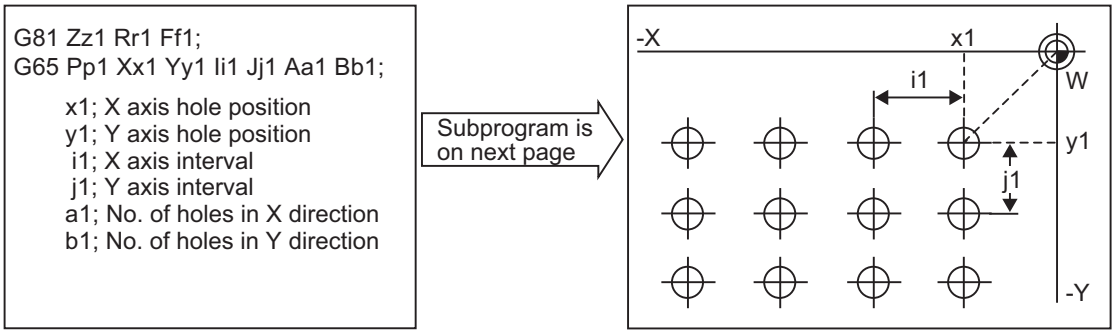
After defining the hole data with fixed cycle (G72 to G89), the macro command is issued as the hole position command.

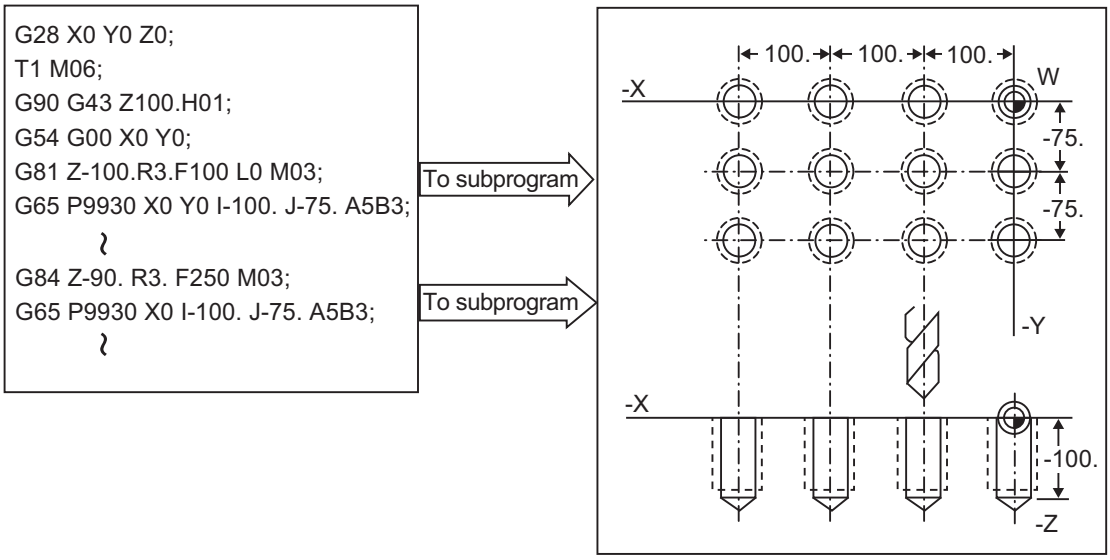




(Example 3) Grid

After defining the hole data with the fixed cycle (G72 to G89), macro call is commanded as a hole position command.





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;

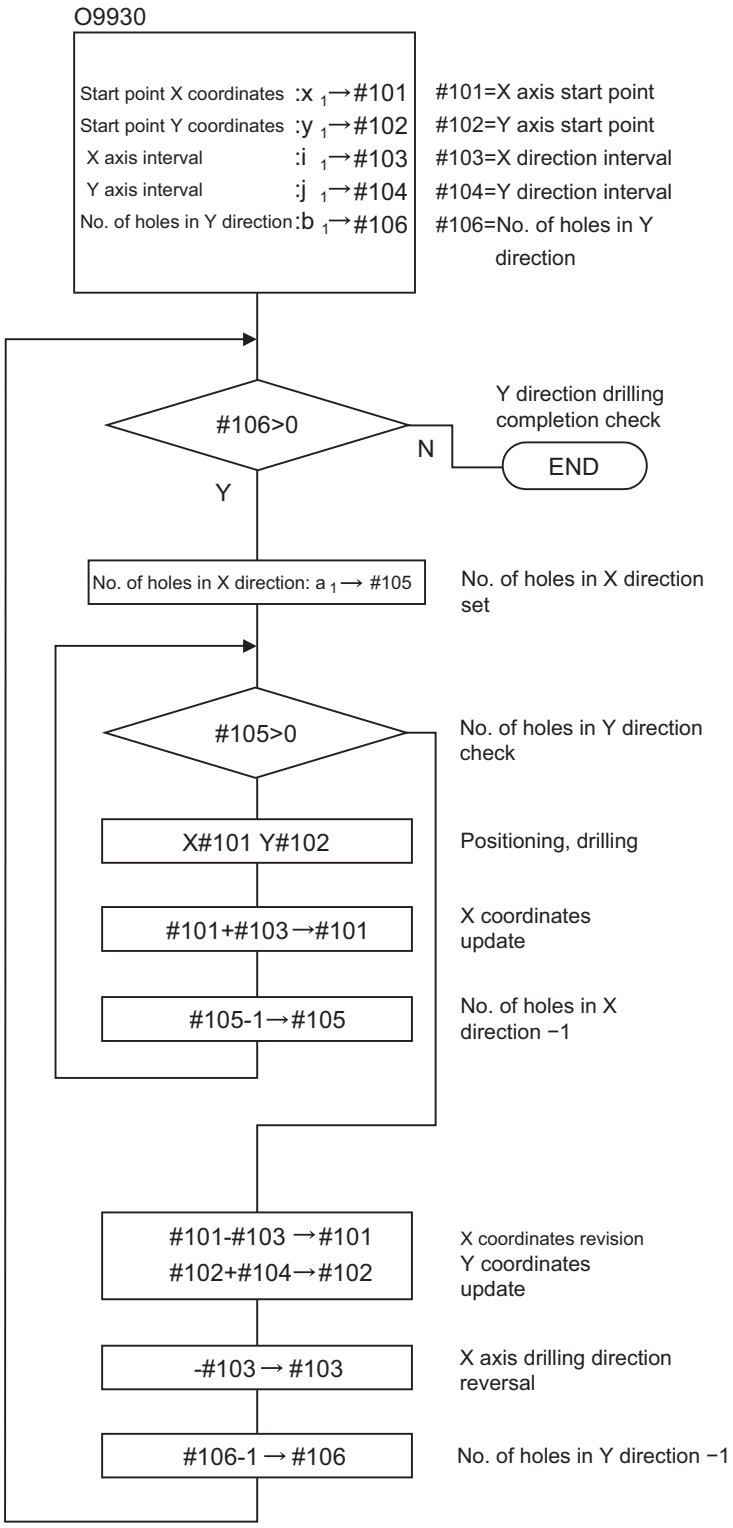
#103=-#103;
#106=#106-1;

END1;

M99;
```

(Note 1) (Note 1) (Note 1) (Note 1)

(Note 1) The processing time can be shortened by programming in one block.



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 Subpro No. select", increase the digit number of command value by adding leading zeros.
(Example) When parameter "#8129 Subpro No. select" is set to "1", call the subprogram "O0012" with "M96 P12" command.

(5) In the below 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 larger the digit number of the program number set with parameter "#8129 Subpro No. select".
- 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.

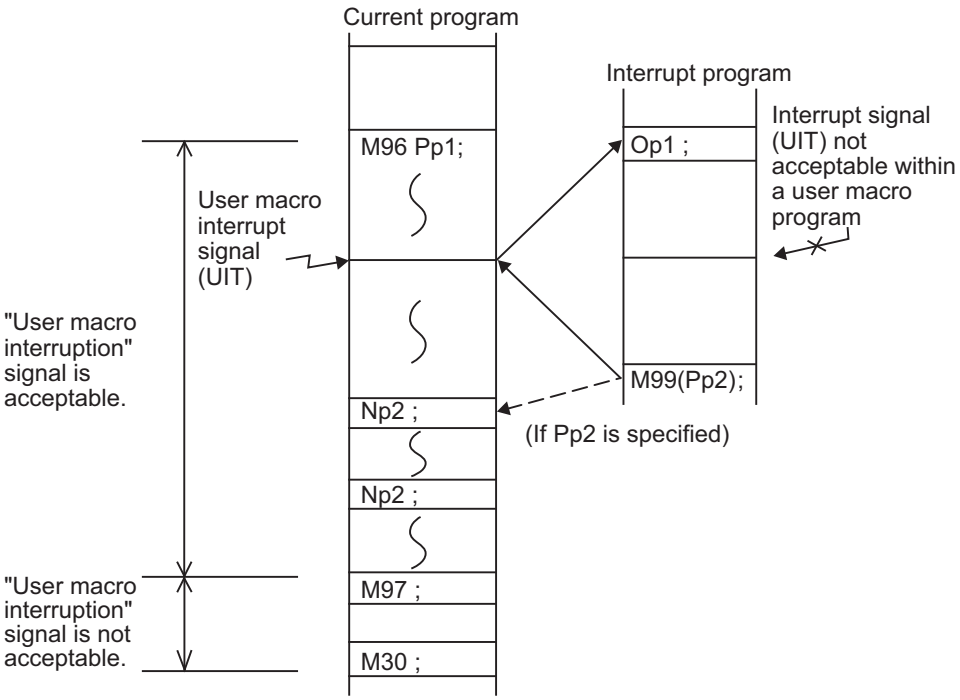
Note

(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 "M96 Pp1;" 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 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.

(4) If an interruption occurs during execution of a block that passes through a singular point in the tool center point control process or a tool axis direction control type 2 (G53.6) block, even if the movement command or miscellaneous function command (MSTB) is not designated in the interrupt program, the next block may be executed without reaching the end point of the interrupted block after the system returns from the interrupt program.

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]

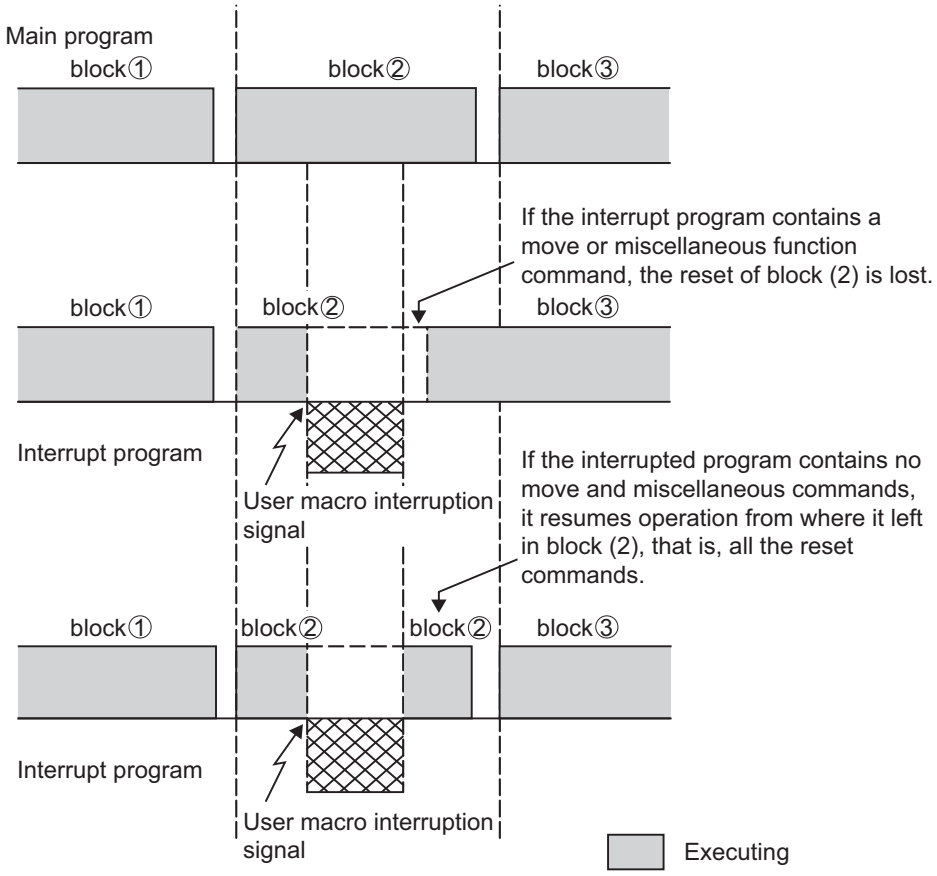
(1) When an interrupt signal (UIT) is input, the interrupt program is executed after the commands in the block executed at that time have been completed. Even if the interrupt program contains a move or miscellaneous function (MSTB) command, the same processing is performed.

(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.

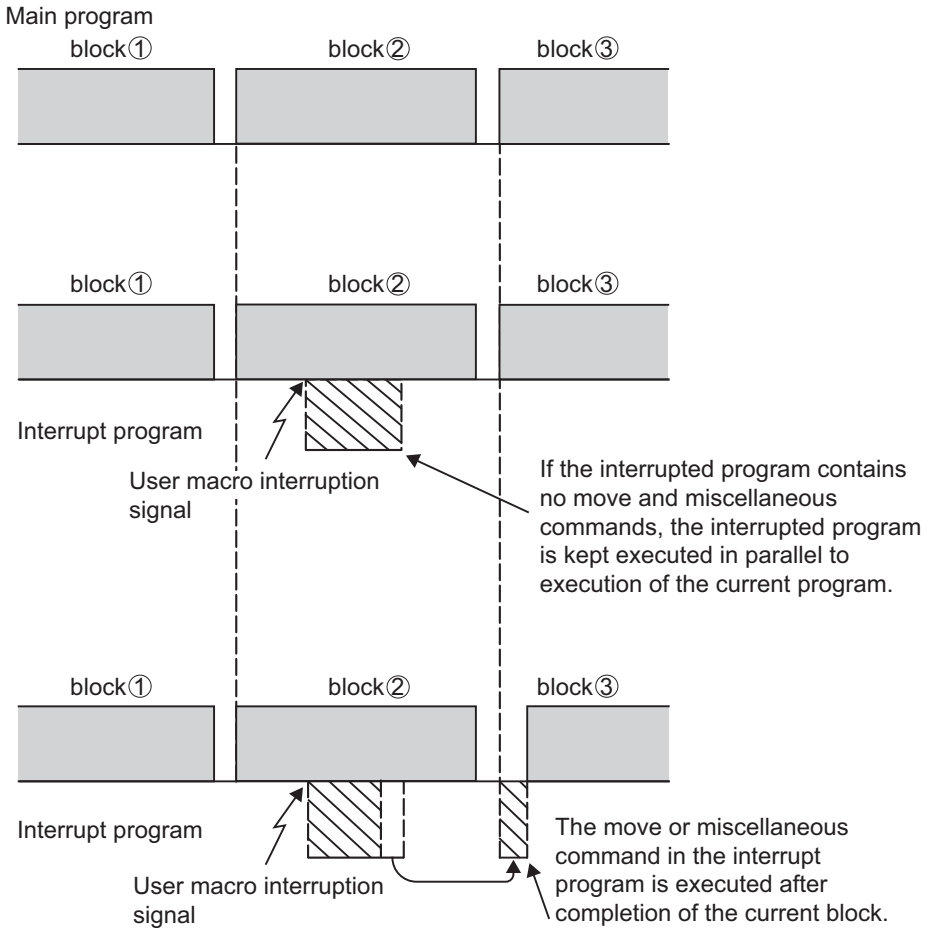
(3) If an interruption occurs during execution of a block that passes through a singular point in the tool center point control process or a tool axis direction control type 2 (G53.6) block, the block may be interrupted in the middle, not after the end of the block.

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]


Main program
block①
block②
block③
If the interrupt program contains a move or miscellaneous function command, the reset of block (2) is lost.
block①
block②
block③
Interrupt program
User macro interruption signal
If the interrupted program contains no move and miscellaneous commands, it resumes operation from where it left in block (2), that is, all the reset commands.
block①
block②
block②
block③
Interrupt program
User macro interruption signal
Executing

[Type 2]


Main program
block①
block②
block③
block①
block②
block③
Interrupt program
User macro interruption signal
If the interrupted program contains no move and miscellaneous commands, the interrupted program is kept executed in parallel to execution of the current program.
block①
block②
block③
Interrupt program
User macro interruption signal
The move or miscellaneous command in the interrupt program is executed after completion of the current block.

Calling method

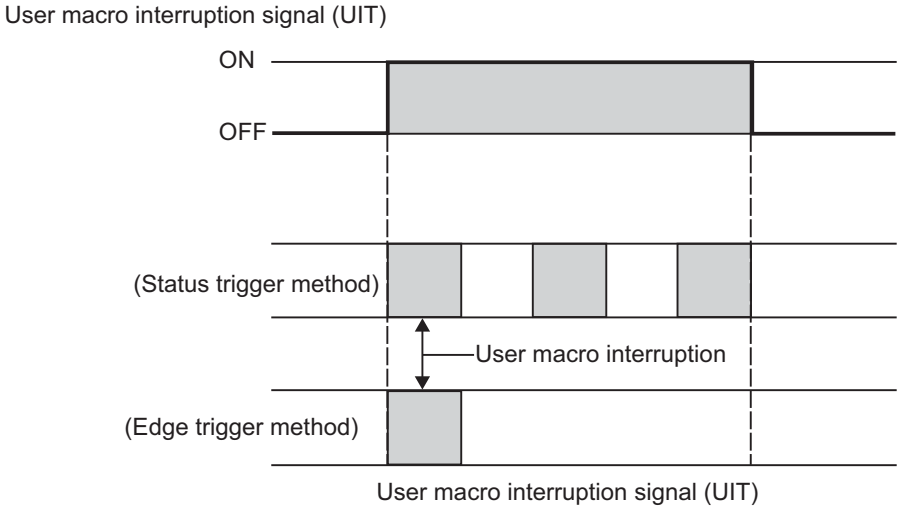
User macro interruption is classified into the following two types depending on the way an interrupt program is called. These two types of interrupt are selected by parameter "#8155 Sub-pro interrupt". This setting also involves the MTB settings (parameter "#1229 set01/bit0").
Both types of interrupt 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.



Returning from user macro interruption

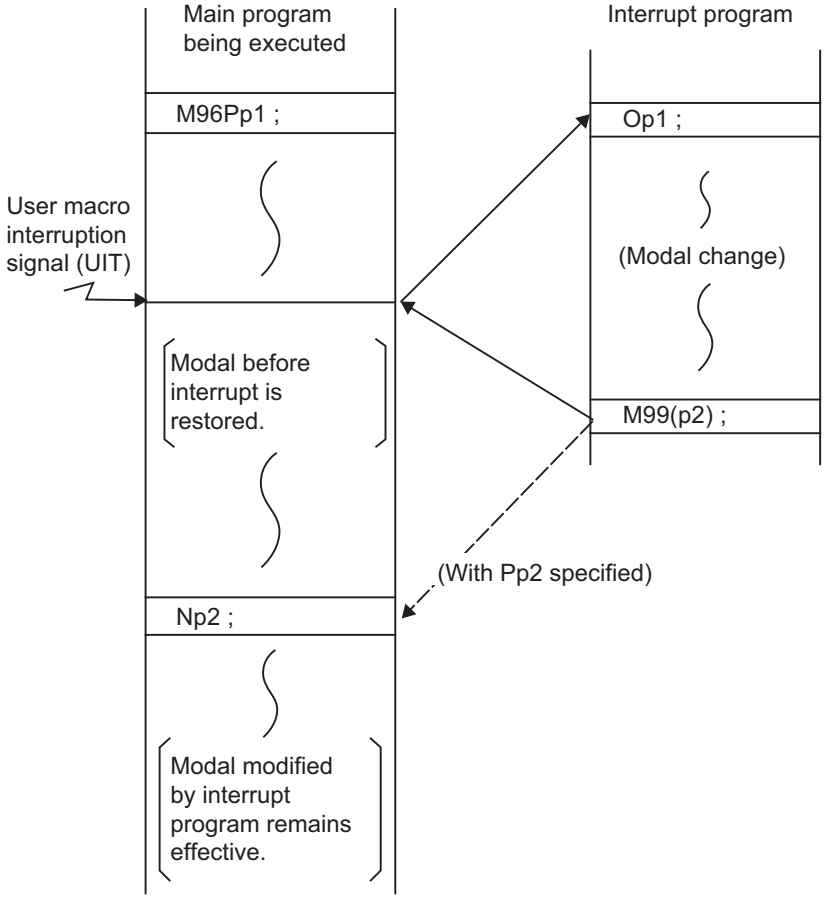
M99 (P__) ;

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. 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) For the commands shown below, modal information is not restored 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	
G40	-	3-dimensional tool radius compensation cancel	●
G41.2/G42.2	-	3-dimensional tool radius compensation (tool's vertical-direction compensation)	
G43.4/G43.5	-	Tool center point control	
G43.8/G43.9	-	Tool cutting point control	
G49	-	Tool center point control cancel	●
G53.1/G53.6	G53.1/G53.6	Tool axis direction control	
G54.4	-	Workpiece installation error compensation	
G68.2/G68.3/G68.4	G68.2/G68.4	Inclined surface machining	
G69	G69.1	Inclined surface machining cancel	●
G92	G92	Spindle clamp speed setting	
G92.1	G92.1	Workpiece coordinate system preset	
-	G110	Mixed control (cross axis control) I	
-	G111	Axis name switch	
-	G113	Spindle synchronization I cancel Tool spindle synchronization IA (spindle - spindle polygon) cancel Tool spindle synchronization II (hobbing) cancel	
-	G114.1	Spindle synchronization I	
-	G114.2	Tool spindle synchronization I A (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	
-	G174	Simple tool center point control	
-	G175	Simple tool center point control cancel	●
-	G176	Simple inclined surface control	
!	!	Timing synchronization (! code)	

The lathe-system commands are shown based on the G code system 3.
Hyphen "-" indicates that the specification is not available.

For the commands marked with “●” in the table above, modal information is restored after control has been returned from the interrupt program unless the modal information is changed between before and after the interruption.

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 variables	Modal information	
#4401 : #4421	G code (group 01) : G code (group 21)	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, the program error (P241) occurs.

(*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 a character string of 0x31, 0x32, 0x33. Thus, the value is as follows.

(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".

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: Subprogram type user macro interruption

0: Macro type user macro interruption

(2) Status trigger method validity "#1112 S_TRG" (*1)

1: Status trigger method

0: Edge trigger method

(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.

(7) Storage destination of user program "#11053 UserProgramStorage" (*1)

Specify the user program storage location.

0: Memory

1: Hard disk

(8) Storage location of external device operation subprogram "#12093 ExtSubProgram:dev", "#12094 ExtSubProgram:dir" (*1)

When a machining program in the external device is called as a subprogram, it is searched in the storage location (device and directory) specified with this parameter.

#12093: One alphabet letter for the device and ':' are set.

#12094: Directory 48 characters are set. (It cannot be set correctly if a space ( ) is used.)

(*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 interruption is performed during execution of the following functions, a sequence No. (M99P_;) must be specified in a command to return from the user macro interrupt program. If no sequence No. is specified, control cannot return to the main program normally.

- Nose R compensation
- Tool radius compensation
- 3-dimensional tool radius compensation

(3) The user macro interrupt program needs to be registered in the NC memory or hard disk according to the parameter "#11053 UserProgramStorage". (This depends on the MTB specifications.)
When it is registered in a device other than the storage locations below, the program error (P232) occurs.

#11053 UserProgramStorage	Storage location
0 (Memory)	NC memory (NC memory 2 is not available.)
1 (Hard disk)	Storage location of external device operation subprogram (#12093, #12094)

(4) If a macro interruption is performed during execution of the following functions, the "Reverse run control mode" signal (YD0A) must be disabled; however, the operation of this signal depends on the MTB specifications. If a macro interruption is performed during execution of these functions while the reverse run control mode is enabled, the operation error occurs.

- Inclined surface machining command
- Tool center point control
- Workpiece installation error compensation
- Tool cutting point control
- 3-dimensional tool radius compensation (tool's vertical-direction compensation)

The operation error that occurs here varies depending on the function that is executing at the macro interruption.
The operation error can be cleared by NC reset.

14.8 Variable Value Display in Program Being Executed

Function and purpose

If a variable, arithmetic operation, or function is specified in the numerical part following the address in the display of the running machining program, it is replaced with the calculated numerical value to be displayed.

This function is available when the parameter "#19431 Num display in prg" is "1".

Detailed description

(1) If a variable, arithmetic operation, or function is specified in the numerical part following the address, it is replaced with the calculated numerical values to be displayed.

(2) The following variables, arithmetic operation, and functions are replaced with the numerical value to be displayed.

Variable	Common variable, local variable, system variable, machine tool builder macro variable, and extended common variables
Arithmetic operation	Addition operation, multiplication operation (*1)
Function	Function (*1)

(*1) Refer to "(2) Addition operation", "(3) Multiplication operation", and "(4) Functions" in Command format of "14.6.1 Operation Commands".

(3) Values are replaced in a total of two blocks: the current block and the next block.

Precautions

(1) The replacement display is not performed until the automatic operation start is completed after the program operation search or program restart search is completed.

(2) If the variable following the address is <empty>, the address is not displayed.

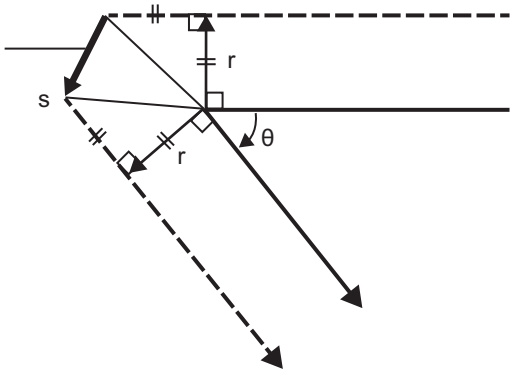
(3) When MACRO SINGLE is valid (#8101 is "1"), the numerical value replacement display is not performed in the IF and WHILE control instruction blocks and variable assignment blocks even if variables, arithmetic operation, or functions are specified. Also, if a block with only "; (EOB)" exists in the line preceding these blocks, "; (EOB)" is displayed.

(4) If MACRO SINGLE is invalid (#8101 is "0"), IF and WHILE control instruction blocks and variable assignment blocks are not displayed. Also, if a block with only "; (EOB)" exists in the line preceding these blocks, "; (EOB)" is not displayed.

(5) The replacement display is not performed for the block following the fixed cycle command or subprogram call command.

(6) Until the block inserted during the tool radius compensation and the nose radius compensation is operated, the replacement display is performed for the currently executed block. When the inserted block starts operation, the replacement display returns to the original display of variables, arithmetic operation, and functions.

The replacement display is not performed while the inserted block is in operation.



(7) In the operation up to the intermediate point of the reference position return command (G28, G29, G30), the replacement display is performed for the currently executed block. When the operation starts after the intermediate point, the replacement display returns to the original display of variables, arithmetic operation, and functions.

(8) Until the creep distance movement of the unidirectional positioning command (G60) starts, the replacement display is performed for the currently executed block. When the creep distance movement starts, the replacement display returns to the original display of variables, arithmetic operation, and functions.

(9) Until the chamfering or rounding operation for corner chamfering or corner rounding starts, the replacement display is performed for the currently executed block. When the chamfering or rounding starts, the replacement display returns to the original display of variables, arithmetic operation, and functions.

(10) When a program error occurs, the numerical value is not replaced in the block that causes the error.

(11) A value less than the input setting unit (#1003 iunit) is rounded off and displayed. Therefore, an error within ±1 may occur in the input setting unit for the program display, the actual movement amount operation, and the position counter display.

(12) When the optional block skip is enabled, the characters following "/" are not displayed.

(13) If the entire block becomes blank because the variable is <empty> or the optional block skip is replaced, the original program is displayed without being replaced.

(14) During arbitrary reverse run (reverse run/forward run), the replacement display is not performed for the block before the reverse run start block (*1). When the reverse run starts, the replacement display returns to the original display of variables, arithmetic operations, and function in the reverse run start block (*1) and the next block. After the reverse run signal is turned OFF, the program is executed in the forward direction. When the block following the reverse run start block is executed, the replacement display is performed again.

(*1) Reverse run start block: Block where the automatic operation start is performed after the reverse run signal is turned ON

(15) When the replacement display is performed, spaces in the block are deleted and displayed.

(16) The replacement display is not performed for the tape (RS-232C input) mode or computer link B.

(17) Variables, arithmetic operation, and functions in comments are not replaced. If the entire line is a comment line, it is handled in the same way as a block containing only "; (EOB)".

(18) During manual arbitrary reverse run (reverse run or forward run after reverse run), the replacement display is not performed for the block before the reverse run start block. For the forward run after the reverse run, the replacement display is not performed up to the block following the reverse run start block.

(19) System variables of position information (machine coordinates/workpiece coordinates/tool offset amount) are displayed as follows.

- When system variables are in the next block to be executed, the numerical value replacement display is not performed.
- When system variables are in the currently executed block, the variables are replaced with the numerical value to be displayed.



Operation example

The following example shows a case in which the machining program O100 is executed.

```
O100
   #100 = 4.
   #101 = 5.
   #102 = 10.
   #103 = 54.1
N01 G#100 X[#101 + 3.]
N02 M#102
N03 G#103 P1
   #110 = #101 + #102
   #111 = #0
   #112 = 1.2345
N04 G0 Z#110
N05 G0 X#111
N06 G0 Y#112
N07 G0 X[#112/0] Y#112
   M02
```

(Example 1) Display of G code, axis address, and M code
[N01 block running]

Machining program	Display after replacement
N01 G#100 X[#101 + 3.] N02 M#102 N03 G#103 P1 #110 = #101 + #102 #111 = #0 :	**N01 G4 X8.** **N02 M10** N03 G#103 P1 #110 = #101 + #102 #111 = #0 :

(Example 2) Display of macro statements
[N03 block running]

If MACRO SINGLE is enabled (#8101 is "1"), macro statements are displayed. If MACRO SINGLE is disabled (#8101 is "0"), macro statements are not displayed.

Machining program	Display after replacement	
	#8101 is "1"	#8101 is "0"
N03 G#103 P1 #110 = #101 + #102 #111 = #0 #112 = 1.2345 N04 G0 Z#110 :	**N03 G54.1 P1** **#110 = #101 + #102** #111 = #0 #112 = 1.2345 N04 G0 Z#110 :	**N03 G54.1 P1** **N04 G0 Z15.000** N05 G0 X#111 N06 G0 Y#112 N07 G0 X[#112/0] Y#112 M02

(Example 3) Display of an address commanded with a variable that is <empty>
[N04 block running]

#111 is <empty>; therefore, address X is also not displayed.

Machining program	Display after replacement
N04 G0 Z#110 N05 G0 X#111 N06 G0 Y#112 N07 G0 X[#112/0] Y#112 M02	**N04 G0 Z15.000** **N05 G0** N06 G0 Y#112 N07 G0 X[#112/0] Y#112 M02

(Example 4) Display when a value less than the input setting unit is commanded (when "#1003 iunit" is "B")
[N05 block running]

A value less than the input setting unit is rounded and displayed.

Machining program	Display after replacement
N05 G0 X#111 N06 G0 Y#112 N07 G0 X[#112/0] Y#112 M02	**N05 G0** **N06 G0 Y1.235** N07 G0 X[#112/0] Y#112 M02

(Example 5) Display of the block where the program error occurred
[N06 block running]

Replacement display is not performed because a program error occurs in block N07.

Machining program	Display after replacement
N06 G0 Y#112 N07 G0 X[#112/0] Y#112 M02	**N06 G0 Y1.235** **N07 G0 X[#112/0] Y#112** M02

15

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 both sides 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.

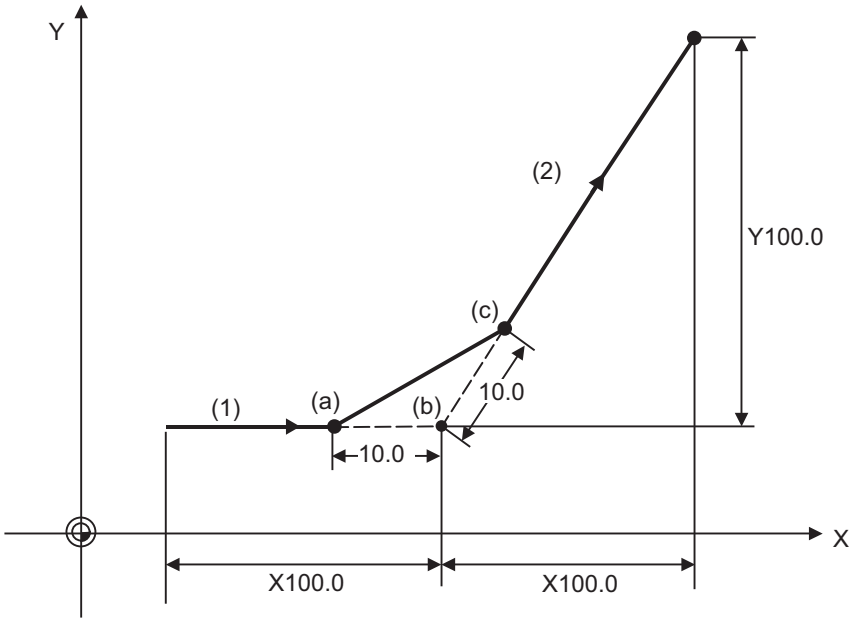
(6) Program error (P383) will occur when the movement amount in the corner chamfering block is less than the chamfering amount.

(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.

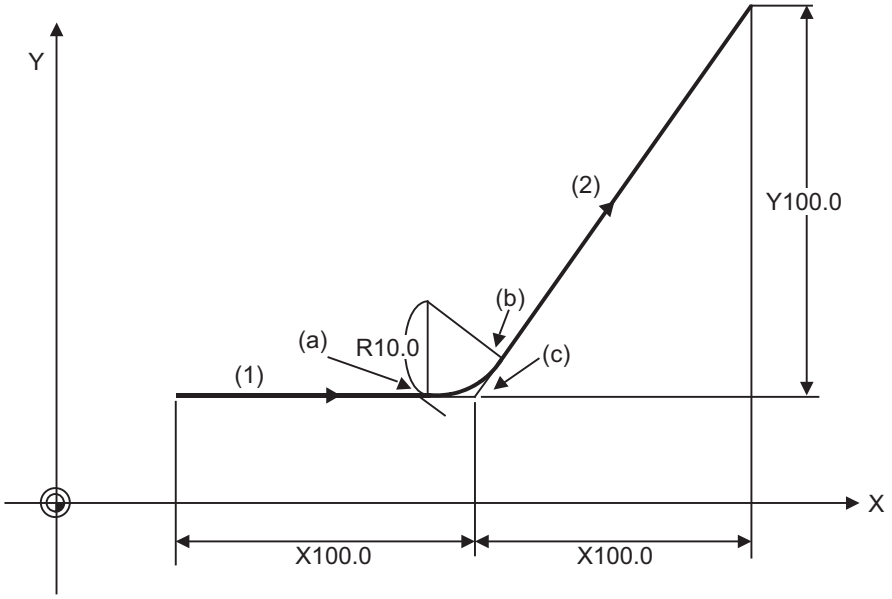
(5) Program error (P383) will occur when the movement amount in the corner rounding block is less than the R value.

(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.;



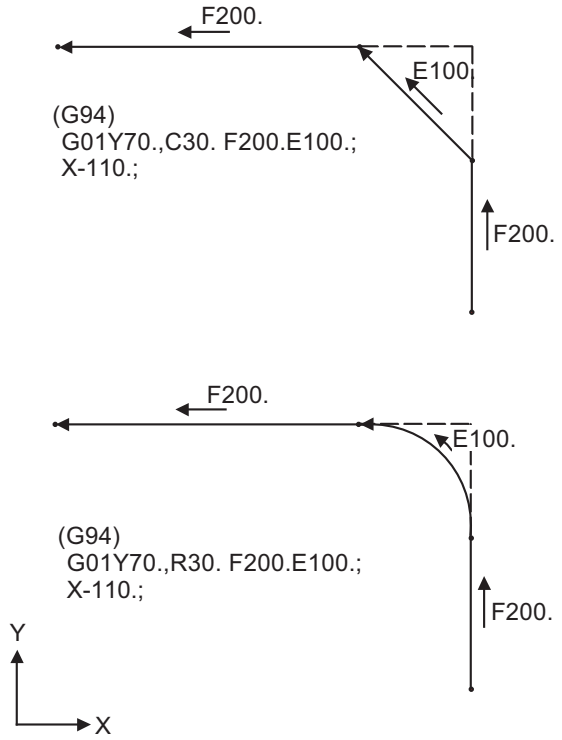
(a) Corner rounding start point
(b) Corner rounding end point
(c) Hypothetical corner intersection point

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.

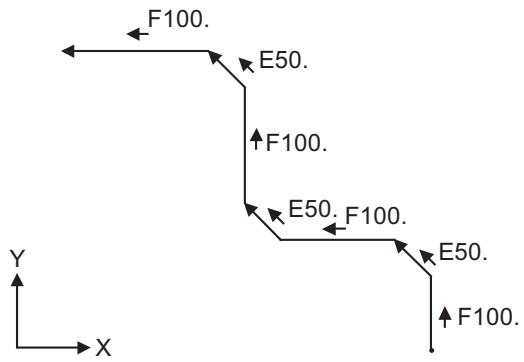
Example



Detailed description

(1) The E command is modal. It is also valid for the feed in the next corner chamfering/corner rounding section.

Example

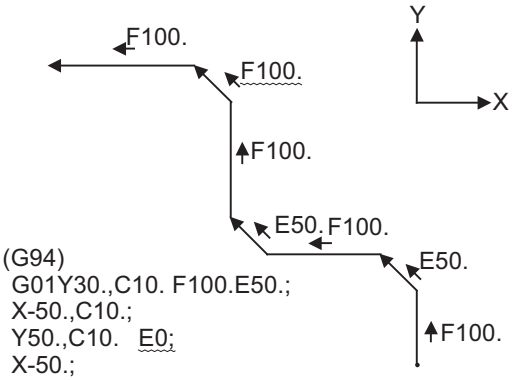



```
(G94)
G01Y30.,C10. F100.E50.;
X-50.,C10.;
Y50.,C10.;
X-50.;
```

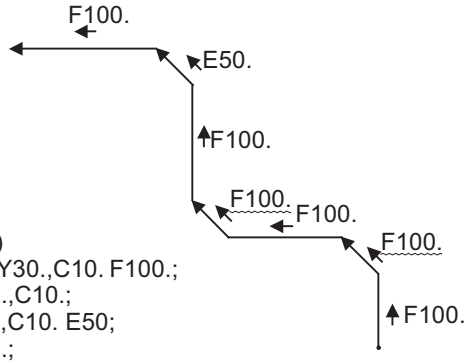
(2) E command modal has separate asynchronous feedrate modal and synchronous feedrate modal functions. Which one is validated depends on the asynchronous/synchronous mode (G94/G95).

(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




```
(G94)
G01Y30.,C10. F100.E50.;
X-50.,C10.;
Y50.,C10.  E0;
X-50.;
```




```
(G94)
G01Y30.,C10. F100.;
X-50.,C10.;
Y50.,C10. E50;
X-50.;
```

(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

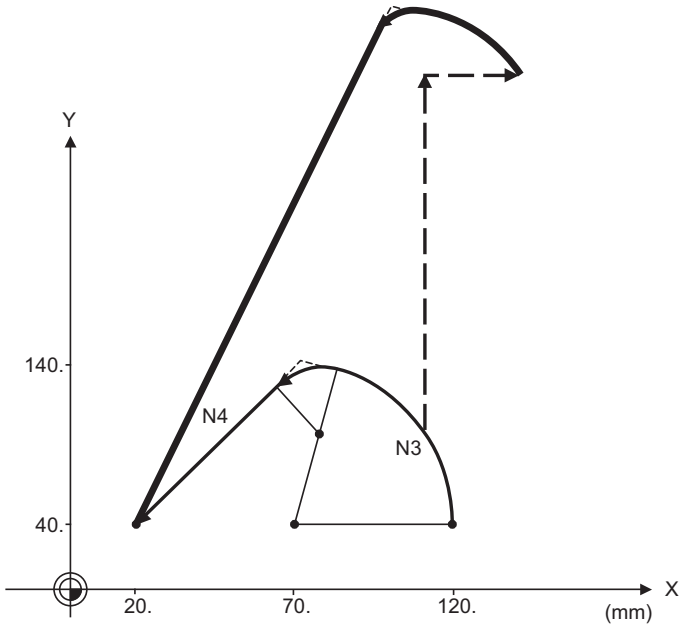
15.1.4 Interrupt during Corner Chamfering/Interrupt during Corner Rounding

Detailed description

(1) Shown below are the operations of manual interruption during corner chamfering or corner rounding.

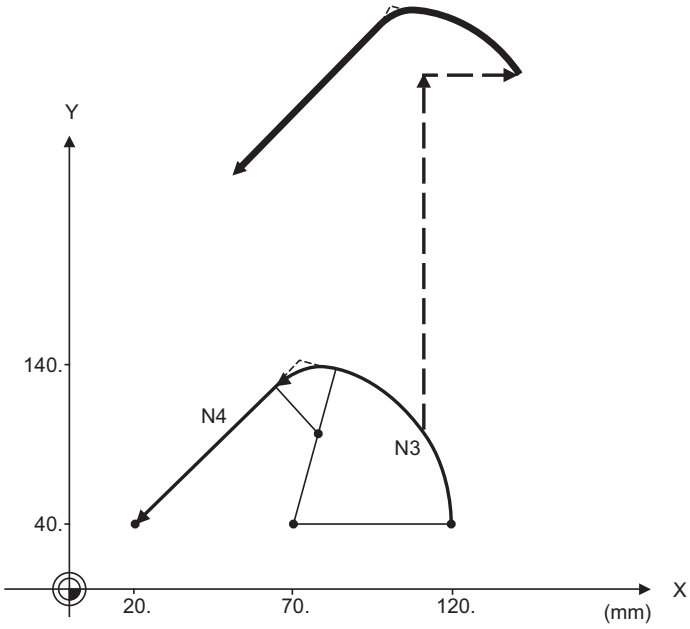
With an absolute command and manual absolute switch ON.

N1 G28 XY;
N2 G00 X120.Y20. ;
N3 G03 X70. Y70.I-50. ,R20. F100 ;
N4 G01 X20. Y20. ;



With an incremental 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 commanded in the corner chamfering block 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

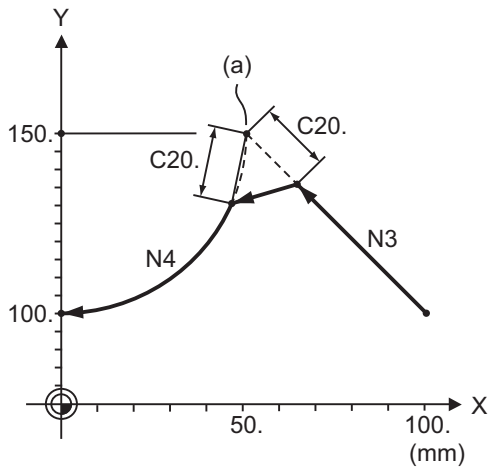
(1) Linear - arc

Absolute command

```
N1 G28 XY;
N2 G90 G00 X100. Y100.;
N3 G01 X50.Y150.,C20. F100;
N4 G02 X0 Y100. I-50. J0;
   :
```

Incremental 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

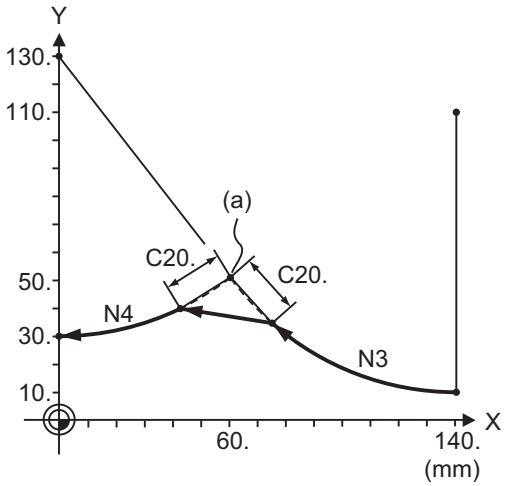
(2) Arc - arc

Absolute command

```
N1 G28 XY;
N2 G91 G00 X140. Y10.;
N3 G02 X60.Y50.I0 J100. ,C20. F100;
N4 X0 Y30.I-60.J80.;
   :
```

Incremental 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.

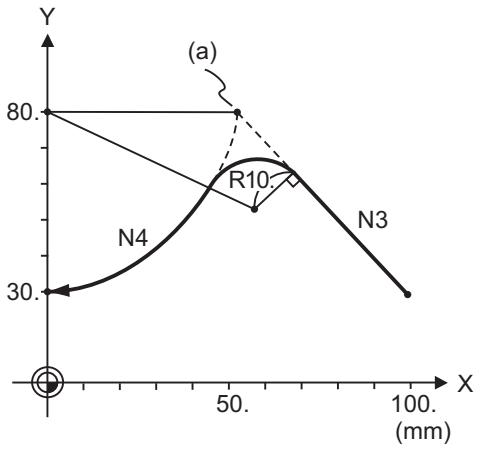
(9) Even if a diameter is commanded, it will be handled as a radial command value during corner rounding.

(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

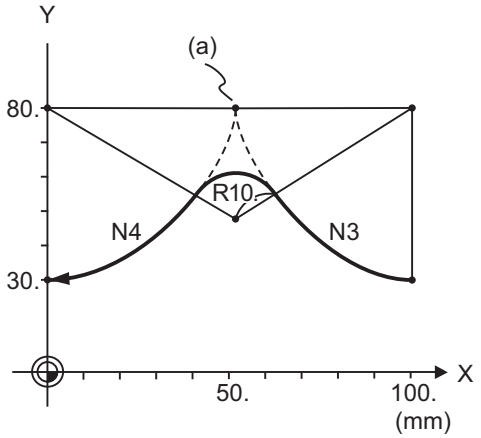
```
Absolute command
  N1 G28 XY;
  N2 G90 G00 X100. Y30.;
  N3 G01 X50.Y80.,R10. F100;
  N4 G02 X0 Y30. I-50.J0;
        :
Incremental command
  N1 G28 XY;
  N2 G91 G00 X100. Y30.;
  N3 G01 X-50.Y50.,R10. F100;
  N4 G02 X-50. Y-50. I-50.J0;
        :
```



(a) Hypothetical corner intersection point

(2) Arc - arc

```
Absolute command
  N1 G28 XY;
  N2 G90 G00 X100. Y30.;
  N3 G02 X50.Y80. R50.,R10.F100;
  N4 X0 Y30. R50.;
        :
Incremental 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

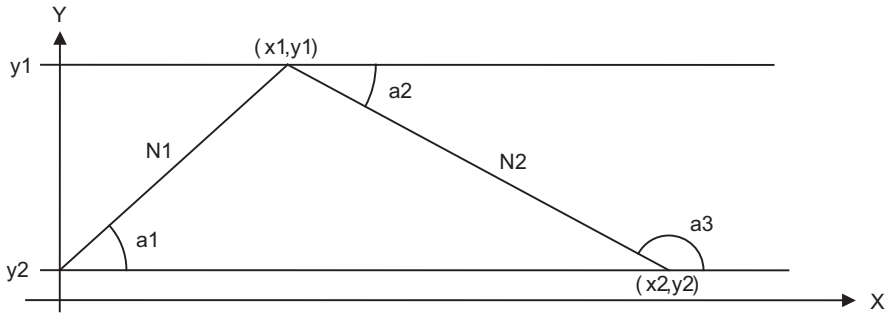
The end point coordinates are automatically calculated by commanding the linear angle and one of the end point coordinate axes.

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 I; G01 A_

Function and purpose

When it is difficult to calculate the intersection point of two straight lines of consecutive linear interpolation commands, the end point of the first straight line will be automatically calculated inside the NC and the movement command will be controlled, provided that the gradient of the first straight line as well as the end point coordinates and gradient of the second straight line are commanded.

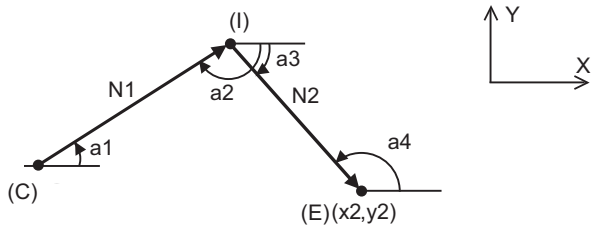
Note

(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 Aa4 (A-a3) Ff2;**

Aa1, A-a2, A-a3, Aa4	Angle
Ff1, Ff2	Speed
Xx2, Yy2	End point coordinates of the next block



(C) Current position
(E) End point coordinates
(I) Intersection point (calculated automatically)

Detailed description

(1) The gradient 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 (-).

(2) The range of gradient "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.

(3) The gradient of the line can be commanded on either the start or end point side. Whether designated gradient is the starting point or the end point will be automatically identified in NC.

(4) When the angle where the two straight lines intersect is less than 1°, program error (P392) occurs.

(5) The end point coordinates of the second block should be commanded with absolute position. If incremental command is used, program error (P393) occurs.

(6) The feedrate can be commanded for each block.

(7) Instead of G01, thread cutting (G33) or variable lead thread cutting (G34) can be specified as a linear path command.
Only G02/G03 commands can be specified as arc path commands.
If another G code is programmed in the second block, the program error (P394) is issued.

(8) Axes cannot be specified in the first block. If an axis is specified, the command is not treated as a geometric command, but a normal linear/arc path command.

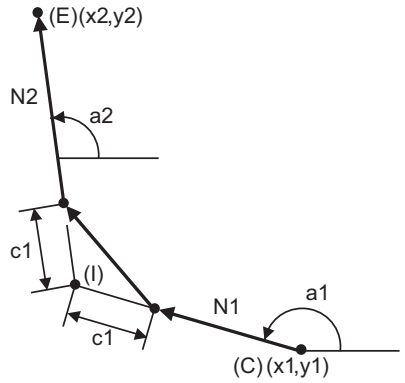
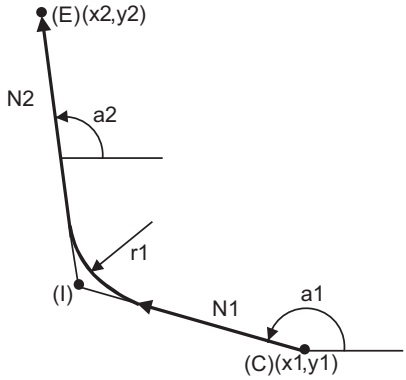
(9) If address "A" is used for an axis name or a second miscellaneous function, geometric I cannot be used. The commands are treated as normal linear path commands, not geometric commands.

Relationship with other functions

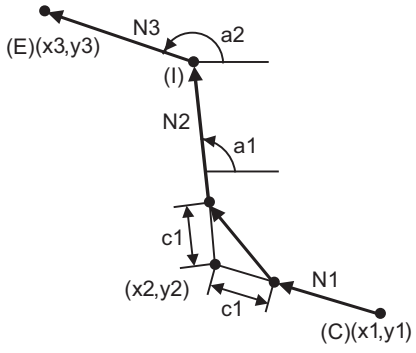
A description is provided using the following examples.

(C) Current position
(E) End point coordinates
(I) Intersection point (calculated automatically)

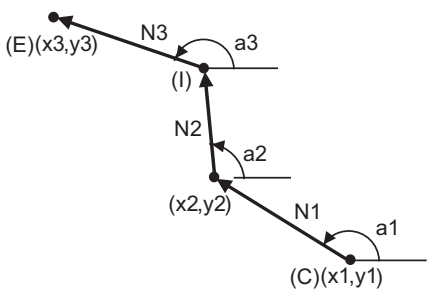
(1) Corner chamfering and corner rounding can be commanded after the angle command in the 1st block.

(Example 1) N1 Aa1 ,Cc1 ; N2 Xx2 Yy2 Aa2 ;	
(Example 2) N1 Aa1 ,Rr1 ; N2 Xx2 Yy2 Aa2 ;	

(2) The geometric command I can be issued after the corner chamfering or corner rounding command.

(Example 3) N1 Xx2 Yy2 ,Cc1 ; N2 Aa1 ; N3 Xx3 Yy3 Aa2 ;	

(3) The geometric command I can be issued after the linear angle command.

(Example 4) N1 Xx2 Aa1 ; N2 Aa2 ; N3 Xx3 Yy3 Aa3 ;	

15.5 Geometric IB

Function and purpose

Geometric IB is used to obtain the contact point or the intersection point for two travel commands in consecutive blocks when at least one of the commands is a circular path command. The center point of the circular arc or the slope angle of the straight line is required instead of the end point of the first block.

Note

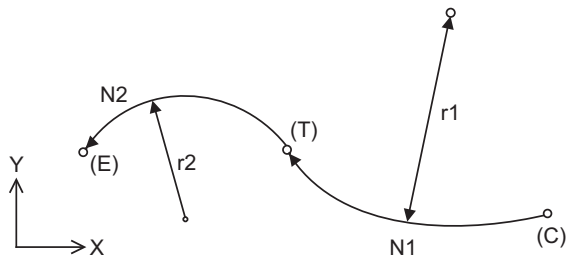
(1) If the parameter "#1082 Geomet" is not set to "2", geometric IB will not function.

(2) Instead of G01, thread cutting (G33) or variable lead thread cutting (G34) can be specified as a linear path command. Only G02/G03 commands can be specified as circular path commands.
If another G code is programmed in the second block, the program error (P394) occurs.

(3) Axes cannot be specified in the first block. If an axis is specified, the command is not treated as a geometric command, but a normal linear/circular path command.

(4) If address "A" is used for an axis name or a second miscellaneous function, a linear path command cannot be used. In such cases, the commands are treated as normal linear/circular path commands, and the intersection point of straight line and circular arc or the contact point between straight line and circular arc are not automatically calculated.

Contact point of two contacting arcs (Refer to 15.5.1.)



The following diagrams are described using the following examples.

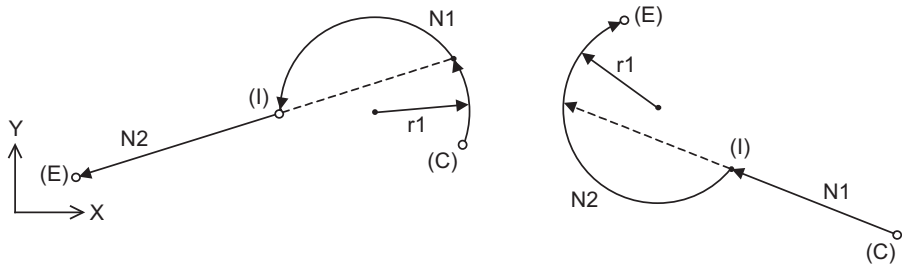
(C) Current position

(E) End point

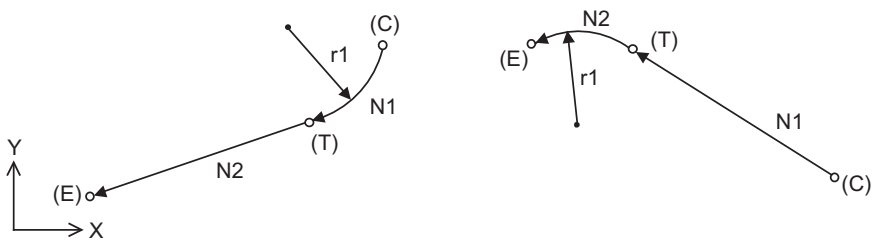
(I) Intersection point (calculated automatically)

(T) Contact point (calculated automatically)

Intersection point between linear and arc (or, arc and linear) (Refer to 15.5.2.)



Contact point between linear and arc (or, arc and linear) (Refer to 15.5.3.)



15.5.1 Geometric IB (Automatic Calculation of Contact Point of Two Circular Arcs); G02/G03 P_Q_ /R_

Function and purpose

When the contact point of two consecutive contacting circular arcs is not indicated in the drawing, it can be automatically calculated using any one of the following commands.

- Command the center coordinate position of the 1st arc as well as the end point (absolute position) and center coordinate position of the 2nd arc.
- Command the center coordinate position of the 1st arc and the radius of the 2nd arc.
- Command the radius of the 1st arc as well as the end point (absolute position) and center coordinate position of the 2nd arc.

Command format

```
N1 G02(G03) Ii1 Kk1 Ff1;
N2 G03(G02) Xx2 Yy2 Ii2 Kk2 Ff2;
```

```
N1 G02(G03) Ii1 Kk1 Ff1;
N2 G03(G02) Xx2 Yy2 Rr2 Ff2;
```

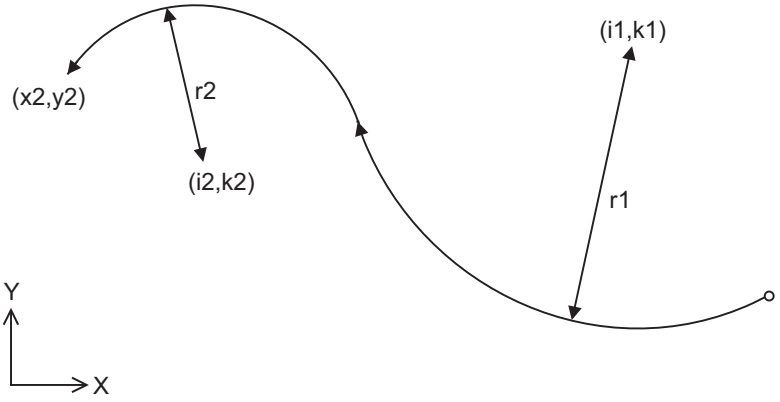
```
N1 G02(G03) Rr1 Ff1;
N2 G03(G02) Xx2 Yy2 Ii2 Kk2 Ff2;
```

I, K (*1)	Circle center coordinates (incremental position) (diameter/radius value command) in X and Z axes 1st block arc : Radius command incremental amount from the start point to the center 2nd block arc : Radius command incremental amount from the end point to the center
R	Arc radius (when a (-) sign is attached, the arc is judged to be 180° or more)

(*1) This command can be issued using P and Q (X and Z axes circle center coordinates (absolute position)) instead of I and K.

The circle center point for Y axis of G17 or G19 plane is specified with J (incremental position) or A (absolute position).

P, A, and Q are treated as addresses that are irrelevant to geometric command, not as a circle center point.



Detailed description

(1) The end point coordinates of the 2nd block should be commanded with the absolute position. A program error (P393) occurs before the 1st block if commanded with the incremental position.

(2) A program error (P390) occurs before the 1st block if there is no geometric IB specification.

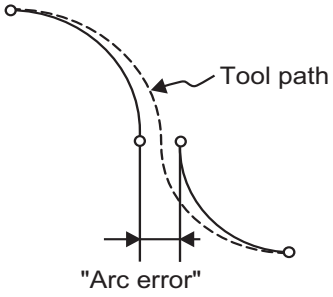
(3) In case of the 2nd block arc, a program error (P395) occurs before the 1st block if the R command (*1) or I/K (P/Q) command is not issued.

(*1) In this case, the 1st block must be set with the I or K (P or Q) command.

(4) A program error (P397) occurs before the 1st block if two arcs that do not contact are commanded.

(5) The accuracy to calculate the contact point is ±1 μm (fractions rounded up).

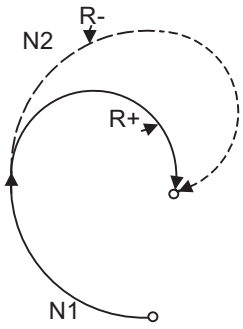
(6) The error range at calculating the contact point is set in parameter "#1084 RadErr".



(7) When I or K is omitted, the values are regarded as "I0" and "K0". P and Q cannot be omitted.

(8) If the start point and the end point of an arc block is identical, the R-designated arc command finishes immediately. To command a true circle, use the IK (PQ)-designated arc command.

(9) When the 2nd block arc inscribes the 1st block arc and the 2nd block is an R-designated arc, the path by the arc command depends on the R sign. When the R sign is positive, the path is set the inward turning arc command (refer to the path of "R+" in the figure). When the R sign is negative, the path is set to the outward turning arc command (refer to the path of "R-" in the figure).



(10) When the arc center of the 2nd block for geometric IB is commanded with IJK and the pitch is designated with address "P" or ",P", the helical interpolation is carried out for the arc of the 2nd block after geometric IB has been completed. Refer to "6.7 Helical Interpolation; G02, G03" for details.

(11) Single block operation stops at the 1st block.

(12) G codes of the G modal group 1 in the 1st/2nd block can be omitted.

15.5.2 Geometric IB (Automatic Calculation of Intersection Point between Line And Circular Arc) ; G01 A_ , G02/G03 P_Q_H_

Function and purpose

When the intersection point between a line and a circular arc is not indicated in the drawing though they intersect, 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 Ii2 Kk2 Hh2 (,Hh2) Ff2 ;

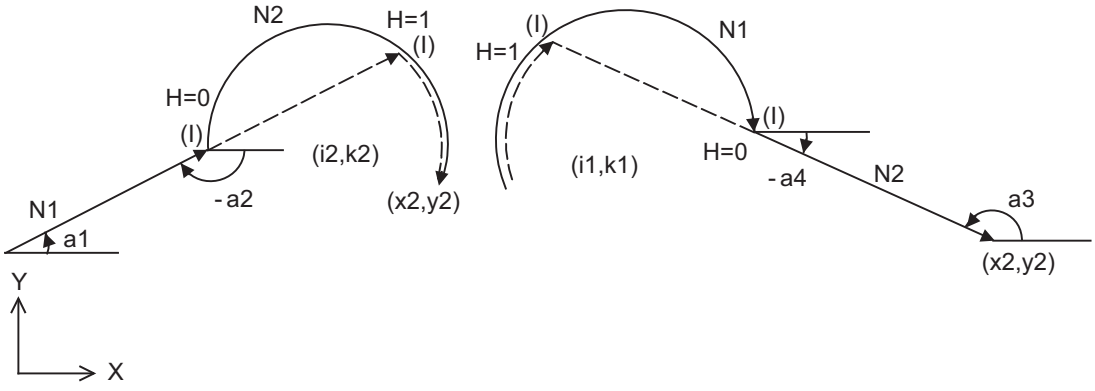
N1 G02(G03) Ii1 Kk1 Hh1 (,Hh1) Ff1 ;
N2 G01 Xx2 Yy2 Aa3 (A-a4) Ff2 ;

A	Linear angle (-360.000° to 360.000°)
I, K (*1)	Circular arc center coordinates (incremental position) (diameter/radius value command) in X and Z axes 1st block arc : Radius command incremental amount from the start point to the center 2nd block arc : Radius command incremental amount from the end point to the center
H (,H)	Selection of intersection point between line and circular arc 0: Intersection of the shorter line 1: Intersection of the longer line

(*1) This command can be issued using P and Q (X and Z axes circular arc center coordinates (absolute position)) instead of I and K.

The circular arc center point for Y axis of G17 or G19 plane is specified with J (incremental position) or A (absolute position).

P, A, and Q are treated as addresses that are irrelevant to geometric command, not as a circle center point.



Path at H = 0

Path at H = 1

(I) Intersection point (calculated automatically) at H = 0 or 1

Detailed description

(1) A program error (P390) occurs before the 1st block if there is no geometric IB specification.

(2) The gradient of the line is the angle to the positive (+) direction of its horizontal axis of the selected plane. The counterclockwise (CCW) direction is considered as positive (+) and the clockwise direction (CW) as negative (-).

(3) The gradient of the line can be commanded on either the start or end point side. Whether designated gradient is the starting point or the end point will be automatically identified.

(4) In case of the 2nd block arc, a program error (P395) occurs before the 1st block if there is no I/K (P/Q) command. A program error (P395) also occurs if there is no designation of "A" for the line.

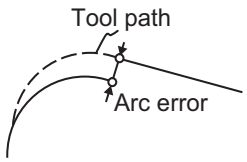
(5) The end point coordinates of the 2nd block should be commanded with the absolute position. A program error (P393) occurs before the 1st block if commanded with the incremental position.

(6) A program error (P397) occurs before the 1st block if a straight line and arc that do not contact or intersect are commanded.

(7) The accuracy to calculate the intersection point is ±1 μm (fractions rounded up).

(8) If the start point of an arc block is identical with the end point, the result is a true circle.

(9) The error range at calculating the intersection point is set in parameter "#1084 RadErr".

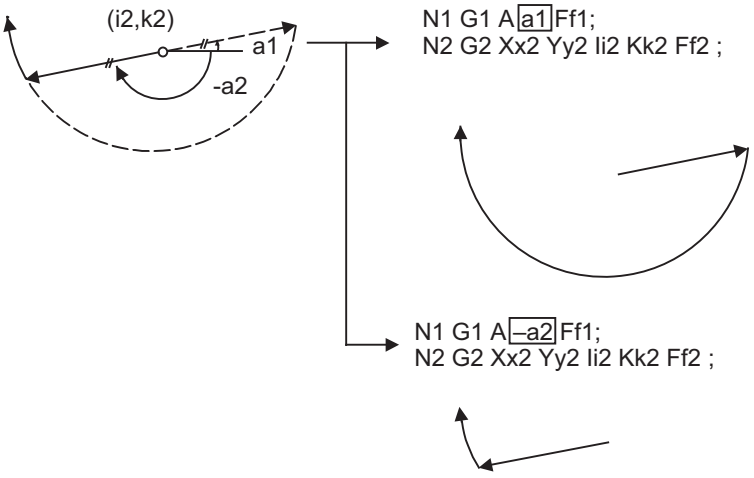


(10) When I or K is omitted, the values are regarded as "I0" and "K0". P and Q cannot be omitted.

(11) When H is omitted, the value is regarded as "H0".

(12) If R is commanded instead of P, Q (I, K) designation, the contact point between line and circular arc is calculated automatically.

(13) When the distance to the intersection from the line is the same as the distance from the arc (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.



(14) Addresses being used as axis names cannot be used as command addresses for angles, circular arc center coordinates or intersection selections.

(15) If address "H" is used as an axis name, the intersection point must be specified with ",H".

(16) If "H" and ",H" are programmed in the same block, ",H" is treated as specifying the intersection point.

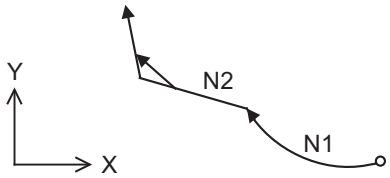
(17) When the circular arc center of the 2nd block for geometric IB is commanded with IJK and the pitch is designated with address "P" or ",P", the helical interpolation is carried out for the arc of the 2nd block after geometric IB has been completed. Refer to "6.7 Helical Interpolation; G02, G03" for details.

(18) Single block operation stops at the 1st block.

(19) G codes of the G modal group in the 1st block can be omitted.

(20) 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 Contact Point between Line And Circular Arc) ; G01 A_ , G02/G03 R_H_

Function and purpose

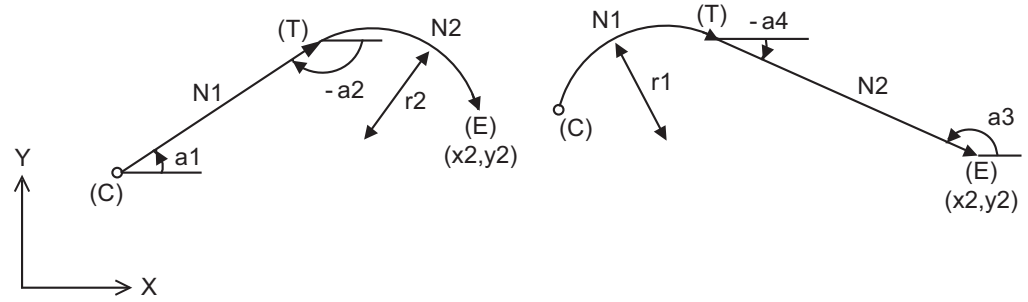
When the contact point between a line and a circular arc is not indicated in the drawing though they are in contact, it can be automatically calculated by commanding the following program.

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 arc radius



(C) Current position
(E) End point coordinates
(T) Contact point (calculated automatically)

Detailed description

(1) A program error (P390) occurs before the 1st block if there is no geometric IB specification.

(2) The gradient of the line is the angle to the positive (+) direction of its horizontal axis of the selected plane. The counterclockwise (CCW) direction is considered as positive (+) and the clockwise direction (CW) as negative (-).

(3) The gradient 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.

(4) 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) also occurs if there is no designation of "A" for the line.

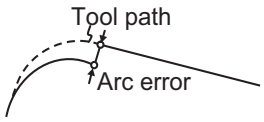
(5) The end point coordinates of the 2nd block should be commanded with the absolute position. A program error (P393) occurs before the 1st block if commanded with the incremental position.

(6) A program error (P397) occurs before the 1st block if a straight line and arc that do not contact are commanded.

(7) If the start point and the end point of an arc block is identical, the circular path command finishes immediately. A true circle cannot be specified.

(8) The accuracy to calculate the contact point is ±1 μm (fractions rounded up).

(9) The error range at calculating the contact point is set in parameter "#1084 RadErr".



(10) If I or K (P or Q) is commanded instead of the R designation, the contact point between line and circular arc is calculated automatically.

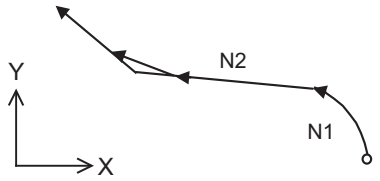
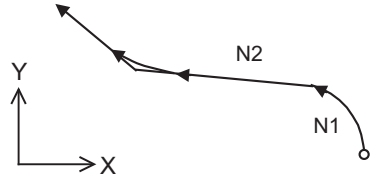
(11) When the arc center of the 2nd block for geometric IB is commanded with IJK and the pitch is designated with address "P" or ",P", the helical interpolation is carried out for the arc of the 2nd block after geometric IB has been completed. Refer to "6.7 Helical Interpolation; G02, G03" for details.

(12) Single block operation stops at the 1st block.

(13) G codes of the G modal group 1 in the 1st block can be omitted.

(14) When geometric IB is commanded, two blocks are pre-read.

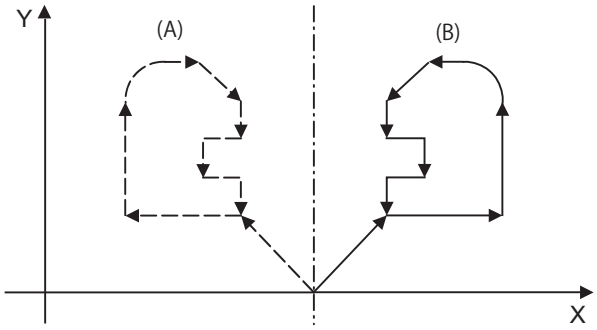
Relationship with other functions

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.



Mirror axis

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.

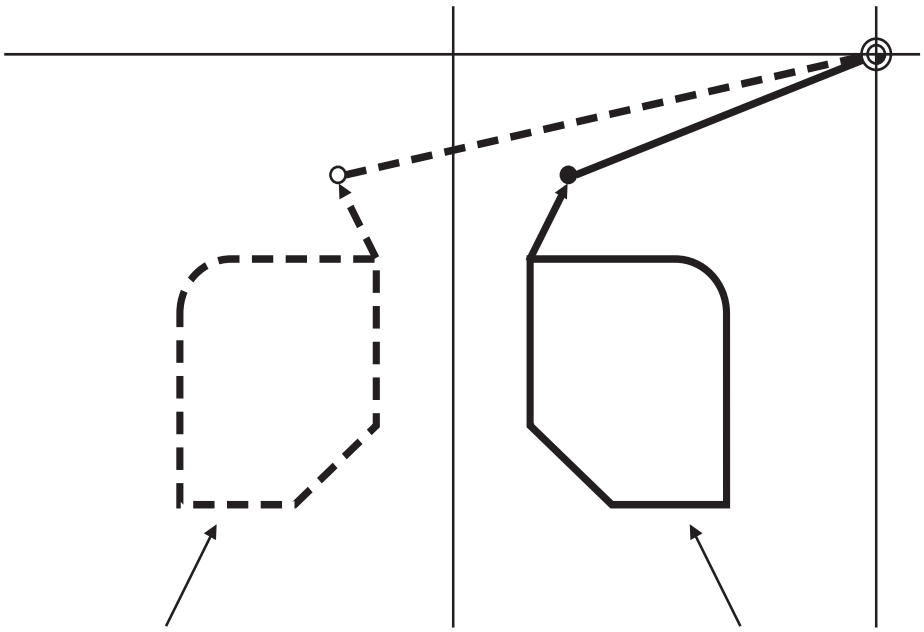
(2) At G50.1, command the axis for which mirror image is to be turned OFF. The values of x2, y2, and z2 will be ignored.

(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.



Path on which mirror is applied Mirror center Programmed path

○ Intermediate point when mirror is applied

● Intermediate point

(6) Return from zero point during mirror image

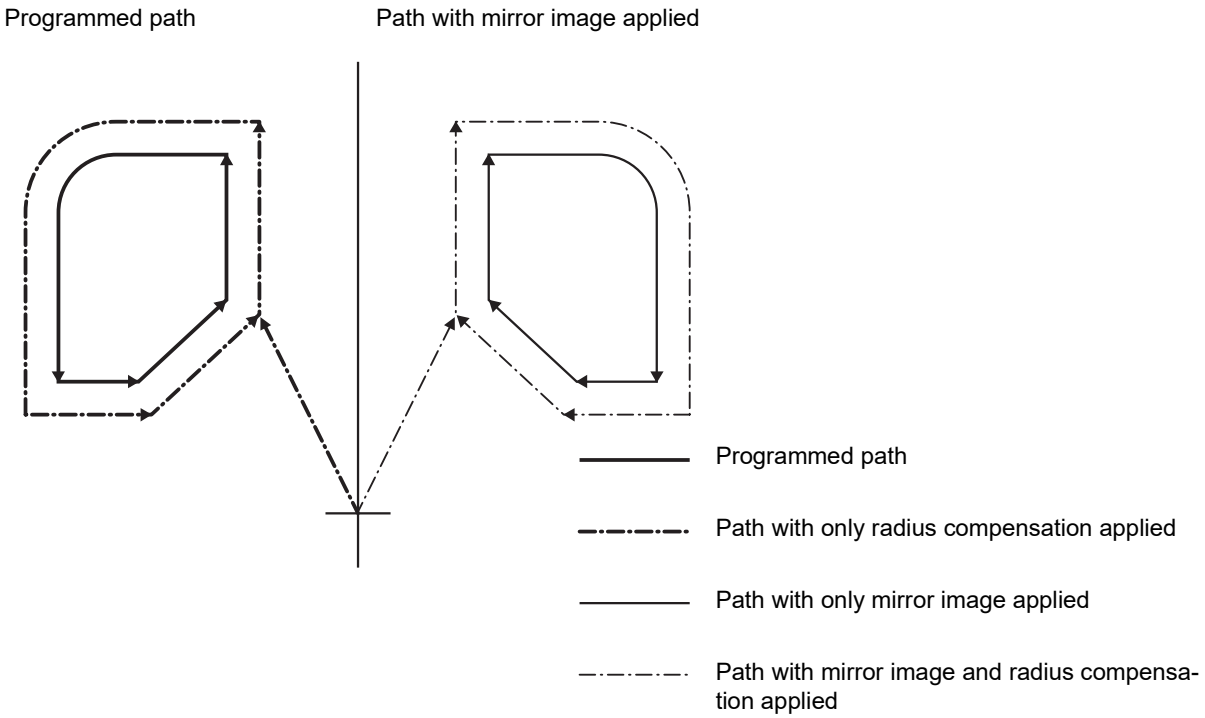
If the return command (G29) from the zero point is commanded during the mirror image, the mirror will be applied to the intermediate point.

(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.

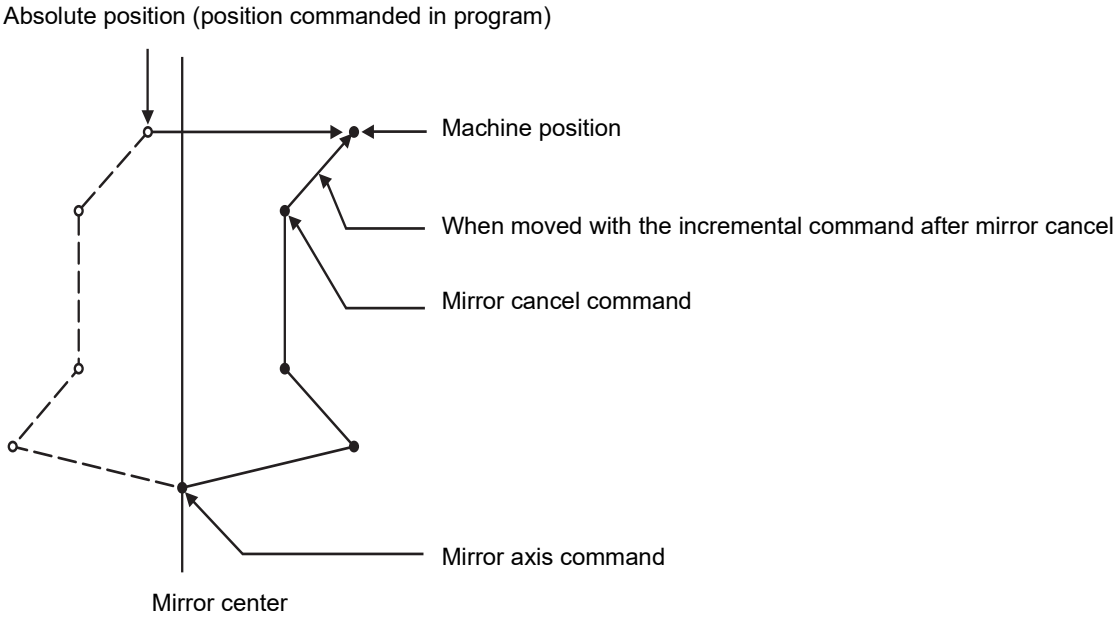


Precautions

⚠ CAUTION

⚠ Turn the mirror image ON and OFF at the mirror image center.

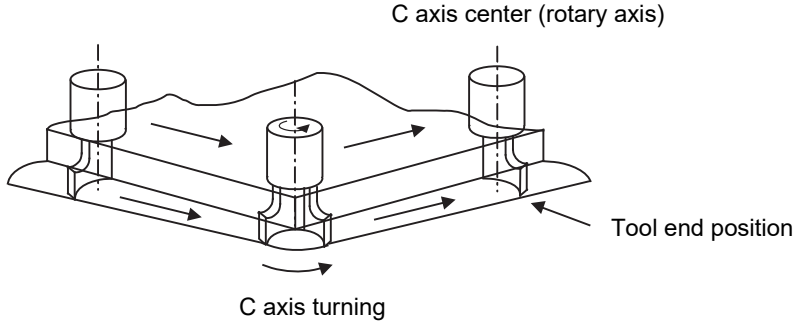
If mirror image is not canceled at the mirror center, the absolute position and machine position will deviate as shown below. (This state will last until an absolute command (positioning with G90 mode) is issued, or a reference position return with G28 or G30 is executed.) The mirror center is set with an absolute position, 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 command after canceling.



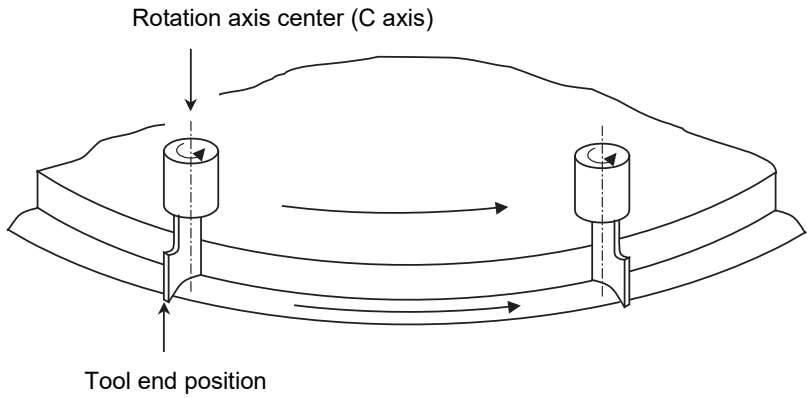
15.7 Normal Line Control; G40.1/G41.1/G42.1 (G150/G151/G152)

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

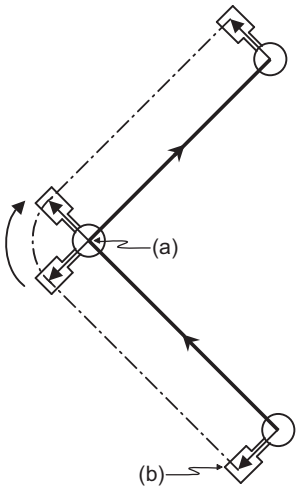
```
G41.1 (G151)X__ Y__ F__ ;
```

Normal line control right ON

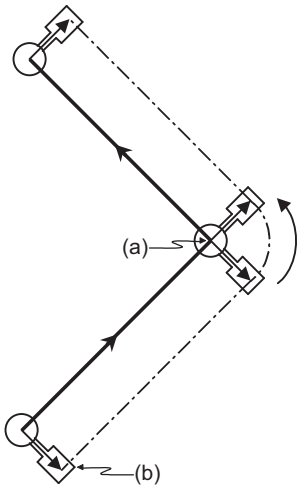
```
G42.1 (G152)X__ Y__ F__ ;
```

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



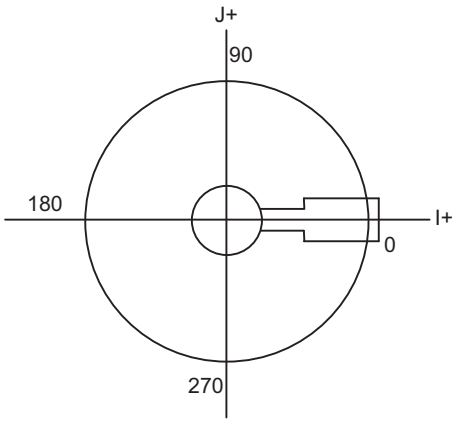
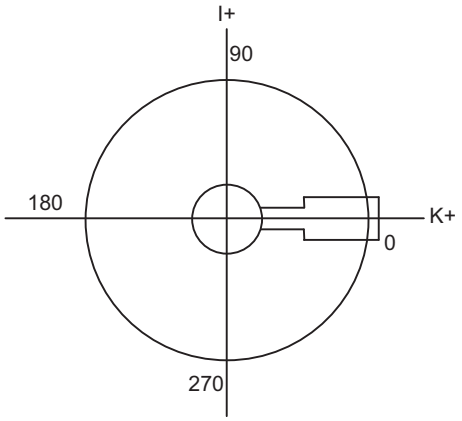
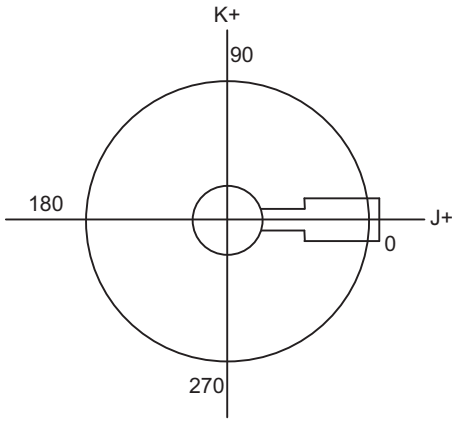
(a) Center of rotation
(b) Tool end(b) Tool end
——— Programmed path
– · – · – Tool end path

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

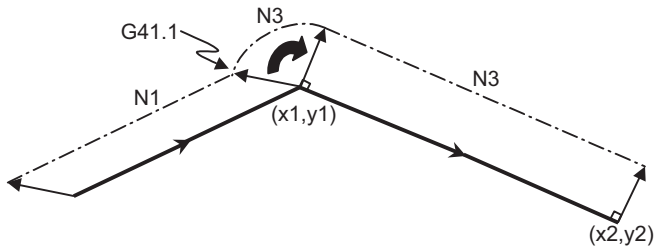
The normal line control angle is 0° (degree) when the tool is facing the horizontal axis (+ direction) direction.
The counterclockwise direction turning is + (plus), and the clockwise direction turning is - (minus).

G17 plane (I - J axes) ... The axis angle is 0°(degree) when the tool is facing the +I direction.	
G18 plane (K - I axes) ...The axis angle is 0°(degree) when the tool is facing the +K direction.	
G19 plane (J - K axes) ... The axis angle is 0°(degree) when the tool is facing the +J direction.	

Normal line control turning operation in respect to movement command

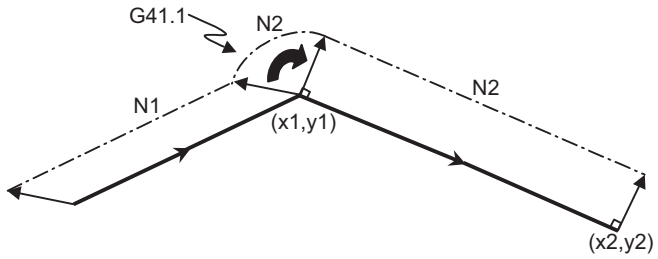
(1) Start-up

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 ;
N3 Xx2 Yy2 ;
:
```

Without movement commands in the G41.1 block



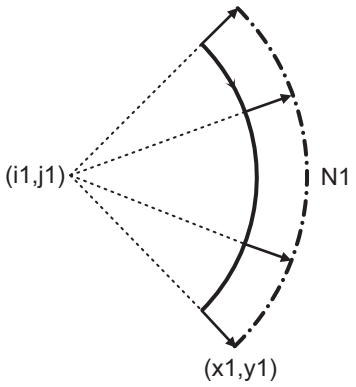

```
:
N1 G01 Xx1 Yy1 Ff1 ;
N2 G41.1 Xx2 Yy2 ;
:
```

With movement commands in the G41.1 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.

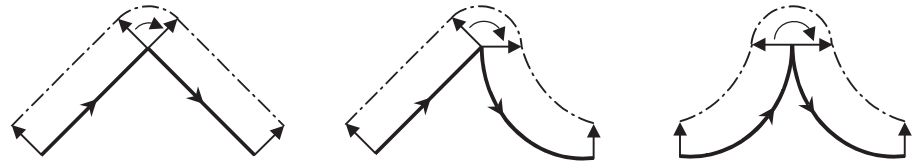
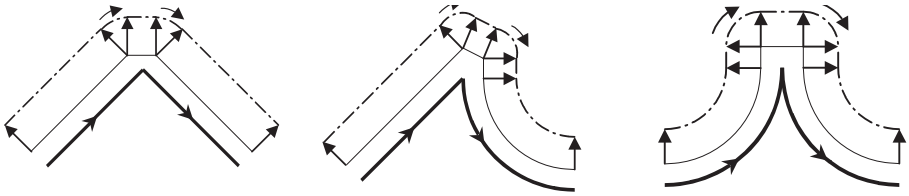



```
:
G41.1 ;
N1 G02 Xx1 Yy1 Ii1 Jj1 ;
:
```

Programmed path

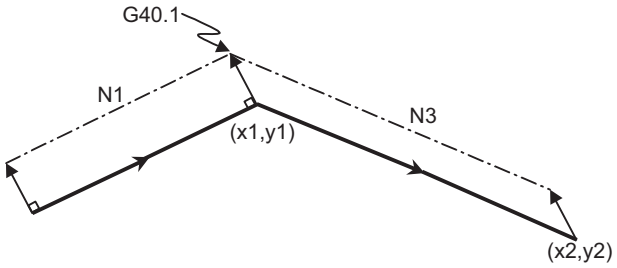
Tool end path

(b) Block seam

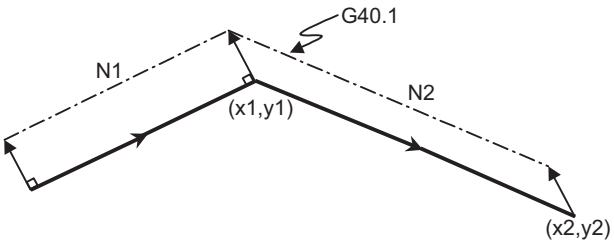
No tool radius compensation	After the normal line control axis is turned to be at the right angle of the plane selecting movement in the next block, the operation moves to the next block. **Liner - Liner** **Liner - Arc** **Arc - Arc**  Programmed path Tool end path
With tool radius compensation	If tool radius compensation is applied, normal line control is carried out along the path to which the tool radius compensation is applied. **Liner - Liner** **Liner - Arc** **Arc - Arc**  Programmed path Tool radius compensation path Tool end path

(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 ;          Without movement commands in the G40.1 block
N3 Xx2 Yy2 ;
:
```



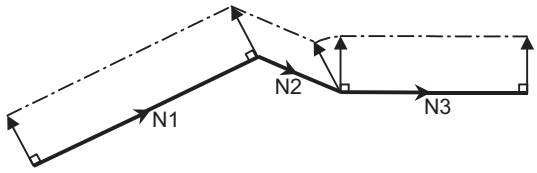

```
:
N1 G01 Xx1 Yy1 Ff1 ;
N2 G40.1 Xx2 Yy2 ;  With movement commands in the G40.1 block
:
```

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).

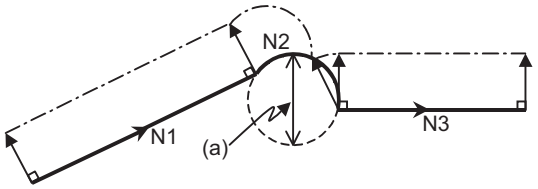
(1) For liner block;
When the movement amount of the N2 block is smaller than the parameter(#1535 C_leng), the normal line control axis is not turned at the seam between the N1 block and N2 block. It stays the same direction as the N1 block.

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. It stays the same direction as the N1 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

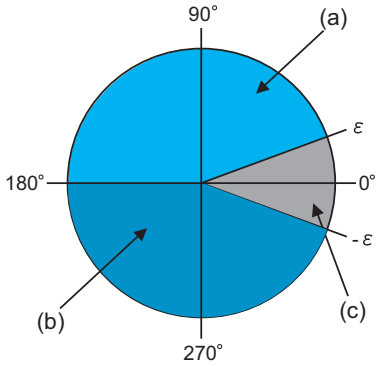
Note

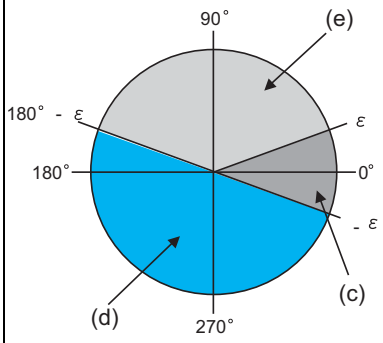
- Since operation fractions are created by calculating the intersection point of two segments, the turning operation may or may not be carried out when the parameter (#1535 C_leng) and the segment length are equal.

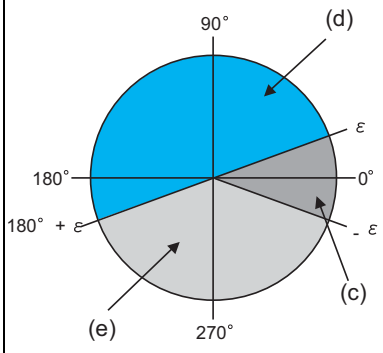
Normal line control axis turning direction at block seam

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.

<table>
<tr><th>Item</th><th>Normal line control type I</th><th>Normal line control type II</th></tr>
<tr><td>Normal line control axis turning direction at block seam</td><td>Direction that is 180° or less.
(shortcut direction)</td><td>G41.1: - direction (CW)
G42.1: + direction (CCW)</td></tr>
<tr><td>Normal line control axis turning angle at block seam</td><td>♦When | θ | < ε, turning is not performed.
θ: Turning angle
ε: Parameter (#1521 C_min)
♦When the turning angle is 180 degrees, the turning direction is undefined regardless of the command mode.

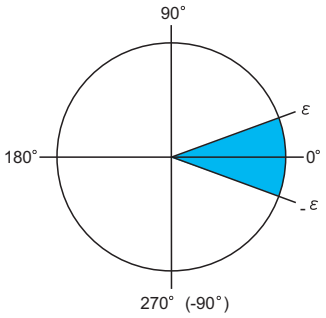
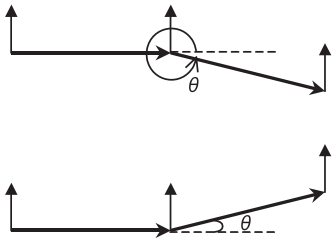
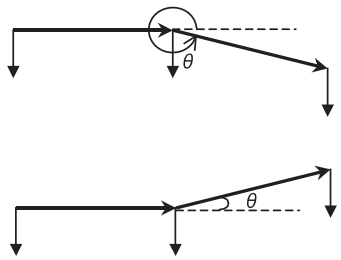
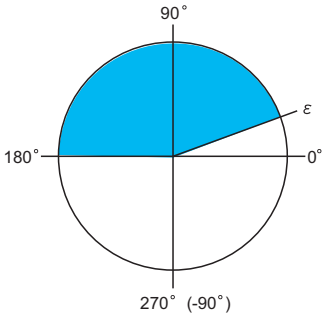
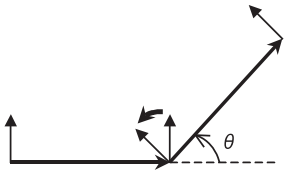
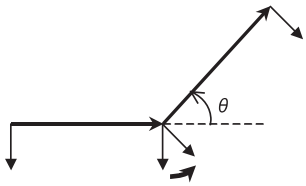
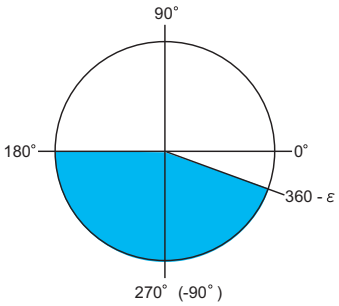
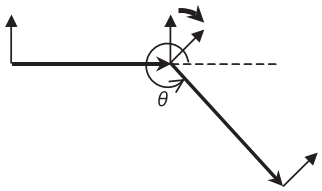
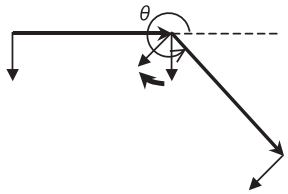
[G41.1/G42.1 When the normal line control axis is at 0°]


(a) The normal line control axis turns counterclockwise.
(b) The normal line control axis turns clockwise.
(c) The axis does not turn.</td><td>♦ When | θ | < ε, turning is not performed.
θ: Turning angle
ε: Parameter (#1521 C_min)
The operation error (0118) will occur in the following cases:
[For G41.1]
ε <= θ < 180° - ε
[For G42.1]
180° + ε < θ <= 360° - ε
[G41.1 When the normal line control axis is at 0°]


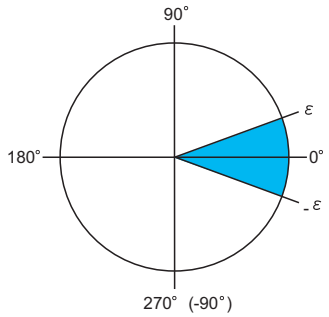
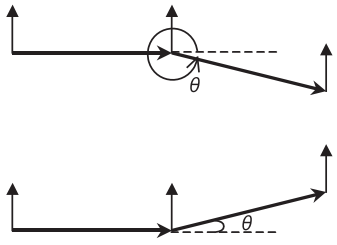
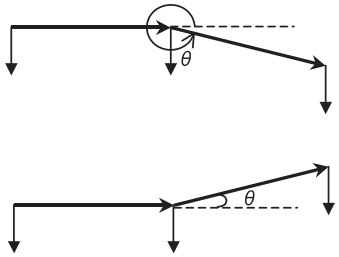
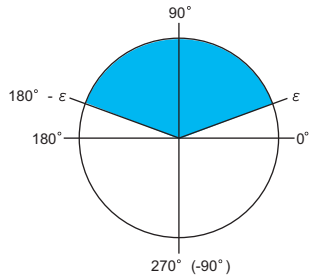
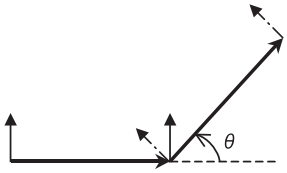
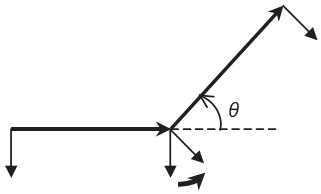
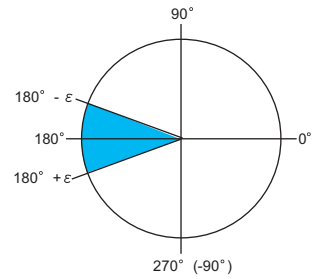
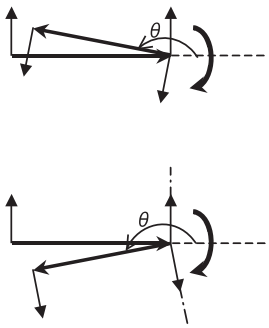
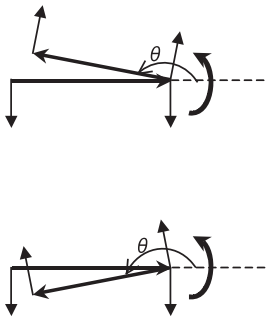
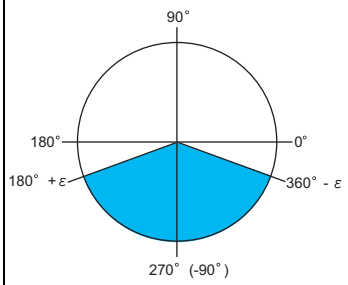
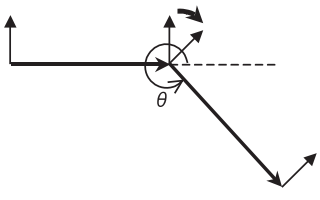
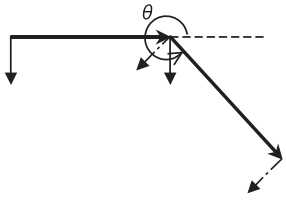
[G42.1 When the normal line control axis is at 0°]


(c) The axis does not turn.
(d) The normal line control axis turns.
(e) Operation error (0118)</td></tr>
</table>

(1) Normal line control type I

Normal line control axis turning angle at block seam: θ	G41.1	G42.1
1. -ε < θ < ε 	 No turning	 No turning
2. ε <= θ < 180° 		
3. 180° <= θ <= 360°- ε 	 Shortcut direction	 Shortcut direction

(2) Normal line control type II

Normal line control axis turning angle at block seam: θ	G41.1	G42.1
1. -ε < θ < ε 	 No turning	 No turning
2. ε <= θ < 180°- ε 	 Operation error (0118) (*1)	
3. 180°-ε <= θ <= 180°+ ε 		
4. 180°+ ε < θ <= 360°- ε 		 Operation error (0118) (*1)

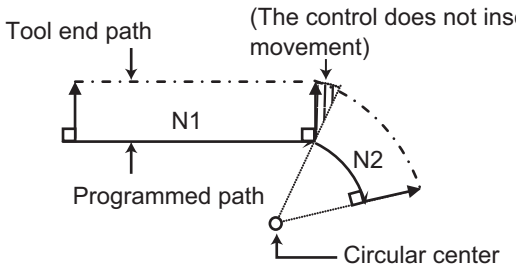
(*1) If the axis turns into the command direction, it turns inside the workpiece, causing an operation error.

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.

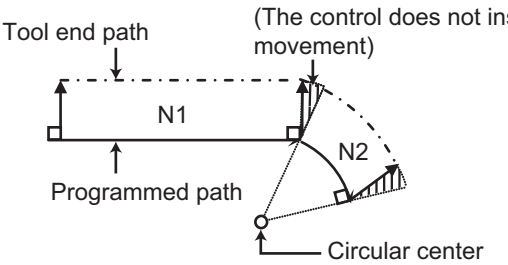
[The turning angle is interpolated up to the end point of the arc (“#12105 C_minTyp” = 0).]
The turning angle in the section in which the normal line control axis is not turned is interpolated up to the end point of the circular interpolation.

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
♦Dry run OFF The rapid traverse rate (#2001 rapid) is applied. Normal line control axis turning speed f = Rapid traverse rate * (Rapid traverse override) (° / min)	♦Dry run OFF Normal line control axis turning speed f = F * 180 / (π * R) * (Rapid traverse override) (°/min) When R = 0, follow the formula below. Normal line control axis turning speed f = F * (Rapid traverse override) (° /min) 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) <Note> (1) If the normal line control axis turning speed exceeds the rapid traverse rate (#2001 rapid), the rapid traverse rate will be applied.
♦Dry run ON The manual feedrate is applied. Normal line control axis turning speed f = Manual feedrate * (Cutting feed override) (° /min) <Note> (1) When the manual override valid is ON, the cutting feed override is valid. (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. (3) When the rapid traverse is ON, the dry run is invalid.	♦Dry run ON Normal line control axis turning speed f = F * 180 / (π * R) * (Rapid traverse override) (°/min) When R = 0, follow the formula below. Normal line control axis turning speed f = F * (Rapid traverse override) (° /min) 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) <Note> (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. (2) If the normal line control axis turning speed exceeds the rapid traverse rate (#2001 rapid), the rapid traverse rate will be applied. (3) When the rapid traverse is ON, the dry run is invalid.

(b) Cutting feed

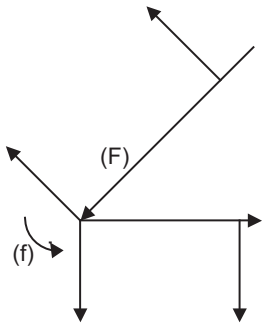
<table>
<tr><th>Normal line control type I</th><th>Normal line control type II</th></tr>
<tr><td>♦Dry run OFF

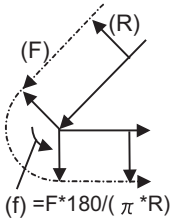
The normal line control axis turning speed set with the parameter (#1523 C_feed) is applied.
Normal line control axis turning speed f
= Parameter (#1523 C_feed) * (Cutting feed override) (° /min)</td><td rowspan="3">The feedrate at the tool nose is the F command. The normal line control axis turning speed is the normal line control axis speed that follows this F command.

Normal line control axis turning speed f
= F * 180 / (π * R) * (Cutting feed override) (°/min)

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)</td></tr>
<tr><td>♦Dry run ON (Rapid traverse ON)

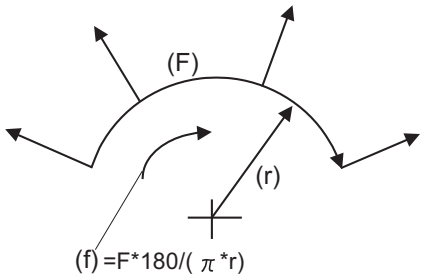
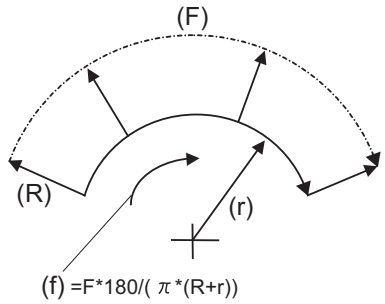
The cutting feed clamp speed (#2002 clamp) is applied.
Normal line control axis turning speed f
= Cutting feed clamp speed (°/min)</td></tr>
<tr><td>♦Dry run ON (Rapid traverse OFF)

The manual feedrate is applied.
Normal line control axis turning speed f
= Manual feedrate * (Cutting feed override) (° /min)
<Note>
(1) When the manual override valid is ON, the cutting feed override is valid.
(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.


F: Feedrate command
f: Normal line control axis turning speed</td><td><Note>
(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.
(2) When the dry run is ON, the normal line control axis turning speed is obtained by the same expression as the rapid traverse.


F: Feedrate command
f: Normal line control axis turning speed
R: Parameter (#8041 C-rot.R)</td></tr>
</table>

(2) Normal line control axis turning speed during circular interpolation

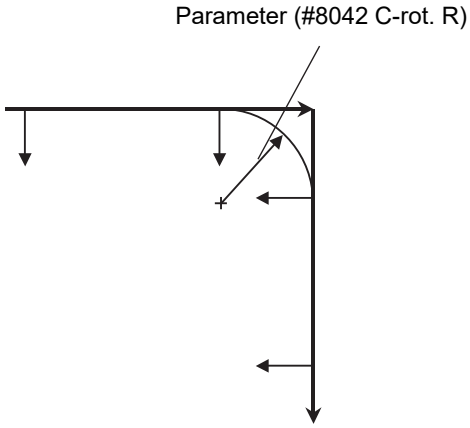
Normal line control type I	Normal line control type II
The normal line control axis turning speed is the rotation speed obtained by feedrate F. Normal line control axis turning speed f = F * 180 / (π * r) (° /min) F : Feed command speed (mm/min) r : Arc radius (mm)  	The feedrate at the tool nose is the F command. The normal line control axis turning speed is the rotation speed that follows this F command. Normal line control axis turning speed f = F * 180 / (π * (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) 

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

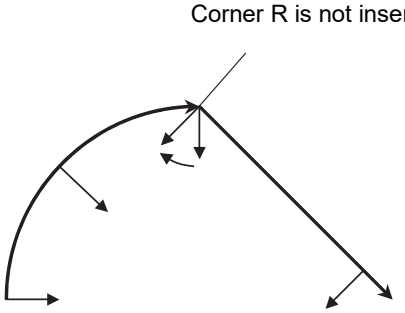
Automatic corner arc insertion function

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.

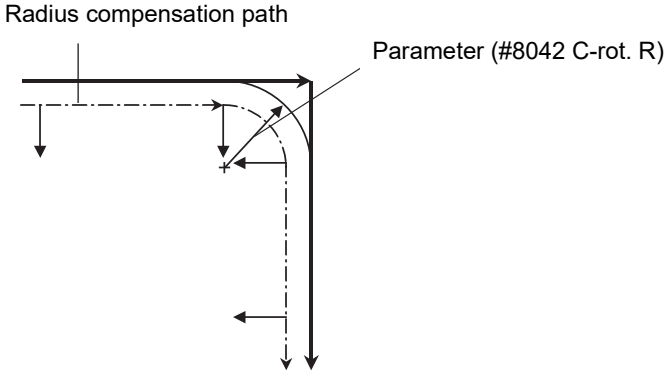


<Supplements>

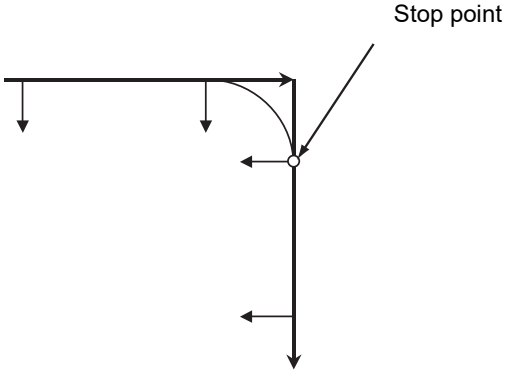
- The corner arc is not inserted in the following cases: linear and arc, arc and arc, linear and moveless or moveless and linear blocks or when a line is shorter than the radius of the arc to insert.



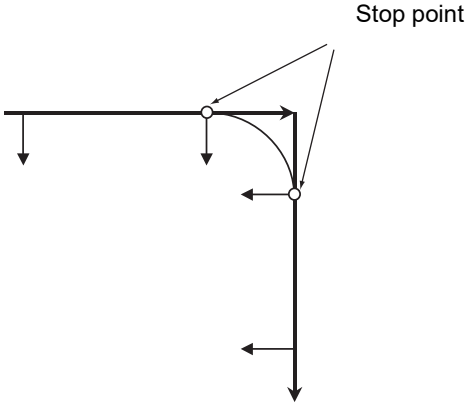
- During the radius compensation, the radius compensation is applied to the path that the corner arc is inserted.



♦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

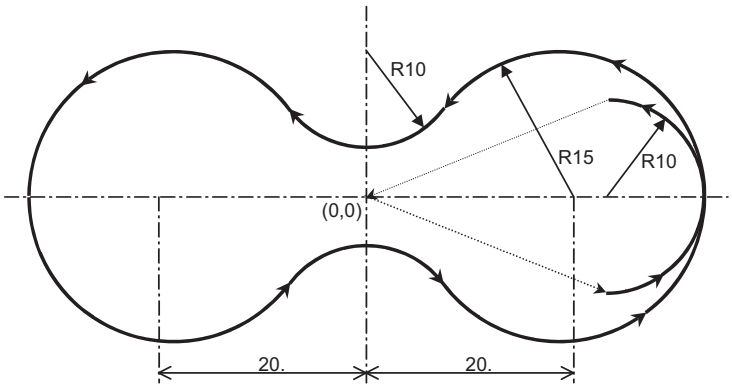
Normal line control type I

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

```
O500
    G91X0Y0;
    G28Z0;
    G28C0;
    G90G92G53X0Y0Z0;
    G00G54G43X35.Y0.Z100.H1;
    G00Z3.;
    G01Z0.1F6000;
    G42.1;
    M98P1001L510;
    M98P1002L2;
    G91G01Y10.Z0.05;
    G40.1;
    G90G00Z100.;
    G28X0Y0Z0;
    M02;
```

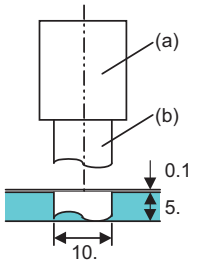
Sub-program

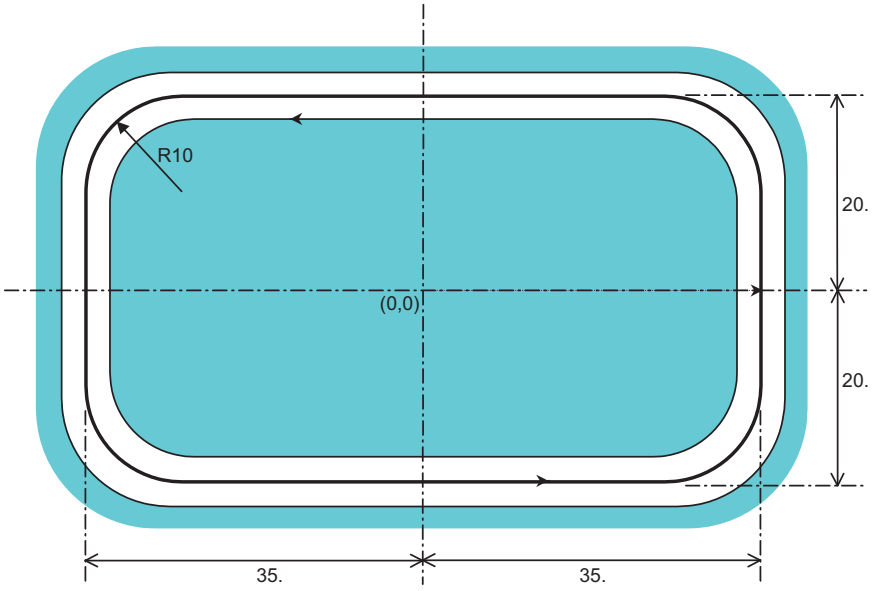
```
O1001
    G17G91G01Y20.,R10.Z-0.01;
    X-70.,R10.;
    Y-40.,R10.;
    X70.,R10.;
    Y20.;
    M99;
```

(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

(Example 2)

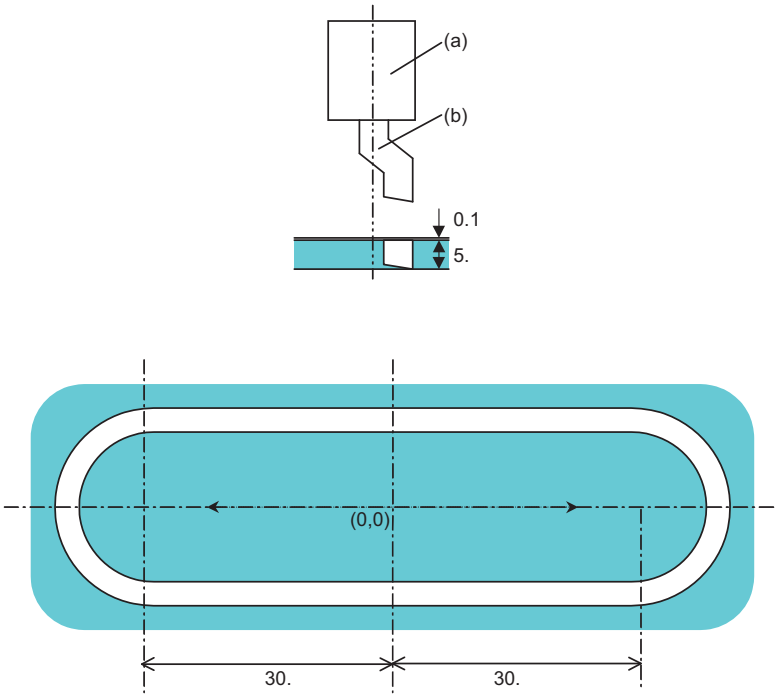
Main program

```
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.

Function name	Notes
Dry run	The feedrate is changed by the dry run signal even in respect to the turning movement of the normal line control axis.
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.

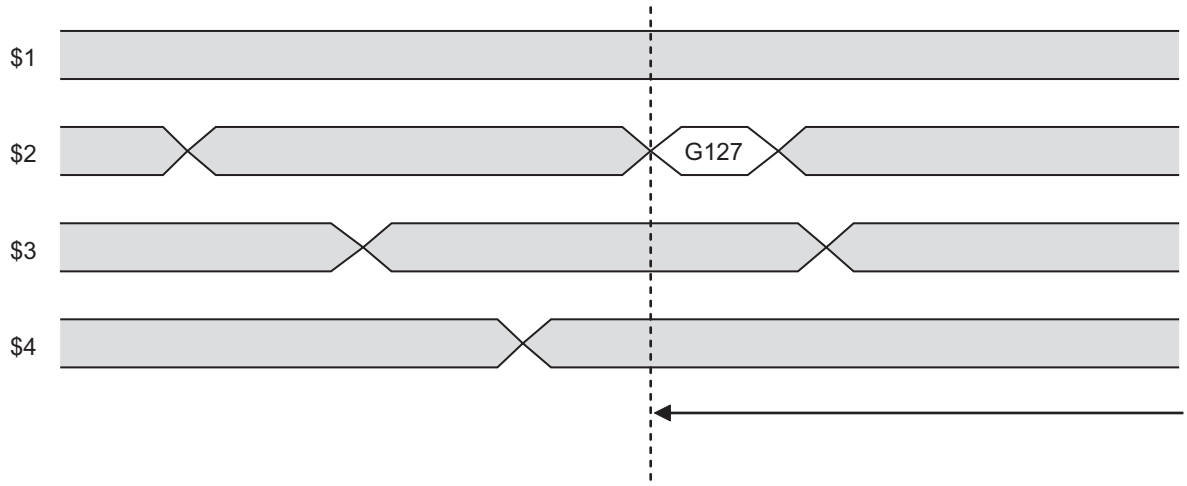
This function (G127) is available to prevent the program from backing to blocks before the commanded block when carrying out the manual arbitrary reverse run.
The detailed setting and operation vary depending on the machine specifications. Refer to the Instruction Manual issued by the MTB.
"Forward run" 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.
- Even if the G61.1 or G05, for which reverse run is prohibited, is not commanded in the machining program, when the initial high-accuracy control (#1148) is valid, reverse run is to be prohibited because of the modal of G61.1 (the value of #1148 is "1") or G05 (the value of #1148 is "2", "3" or "4").

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 Spiral/conical interpolation CW (type2)	○ *1	-
G03	Circular interpolation CCW Spiral/conical interpolation CCW (type2)	○ *1	-
G02.1	Spiral/conical interpolation CW (type1)	×*3	-
G03.1	Spiral/conical interpolation CCW (type1)	×*3	-
G02.2	Involute interpolation CW	×*3	-
G03.2	Involute interpolation CCW	×*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	This includes the initial high-accuracy control (#1148).
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	-

G code	Function name	Reverse run	Remarks
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	-
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	-

G code	Function name	Reverse run	Remarks
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.
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.

G code	Function name	Reverse run	Remarks
G60	Unidirectional positioning	× *3	-
G61	Exact stop check mode	○ *1	-
G61.1	High-accuracy control ON	× *4	This includes the initial high-accuracy control (#1148).
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	-
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.
G74.5 G74.6 G74.8	Fixed cycle (reverse Punchtap cycle)	×*4	The command is valid only in the actual cutting mode. The program error (P182) occurs in the dry run mode.
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.

G code	Function name	Reverse run	Remarks
G84	Fixed cycle (tapping)	○ *2	The reverse run is enabled, but the synchronous feed is invalid. Actual cutting mode available.
G84.5 G84.6 G84.8	Fixed cycle (Punchtap cycle)	×*4	The command is valid only in the actual cutting mode. The program error (P182) occurs in the dry run mode.
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 command	○ *2	Switched with the movement command just after commanded.
G91	Incremental 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	-
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.
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 - 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, G11

Function and purpose

The parameters set from the setting and display unit can be changed in the machining programs.

For commanding data with decimal point, and character string data.
The data's command range conforms to the setting range of each parameter.

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) G11 must be commanded alone in a block. A program error (P33 or P421) will occur if it is not commanded alone in a block.

(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.



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

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 ;	

15.9.2 Compensation Data Input by Program (Tool Compensation Amount) ; G10 L10/L11/L12/L13, G11

Function and purpose

The tool compensation can be set or changed by the program using the G10 command.

During the absolute command (G90) mode, the commanded offset amount serves as the new offset amount, whereas during the incremental command (G91) mode, the sum of present offset and the commanded offset serves as the new offset amount.

Command format

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_ ;	**Tool radius compensation (Shape compensation)**
G10 L13 P_ R_ ;	**Tool radius compensation (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 ;

Detailed description

(1) G10 is a non-modal command and is valid only in the commanded block.

(2) The G10 command does not perform any movement, but must not be used with G commands other than G90 or G91.

(3) Do not command G10 in the same block as the fixed cycle and subprogram call command. This will cause malfunctioning and program errors.

(4) The workpiece offset input command (L2 or L20) should not be issued in the same block as the tool compensation input command (L10).

(5) If an illegal L No. or compensation No. is commanded, the program errors (P172 and P170) occur respectively. If the offset amount exceeds the maximum command value, the program error (P35) occurs.

(6) Decimal point inputs can be used for the offset amount.

(7) If the G command that cannot be combined with G10 is issued in the same block, a program error (P45) occurs.

(8) A program error (P35) occurs for any value that does not match the compensation amount setting unit after command unit conversion.
With an incremental command, the setting range for the compensation amount is the sum of the present setting value and commanded value.

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)

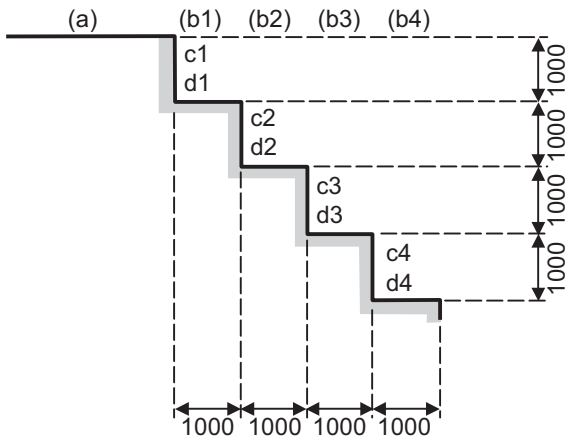
(Example 2) Assume that "H10 = -1000" is already set.

Main program

N1 G00 X100000 ;	a
N2 #1=-1.;	
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=#1-1.;	
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 ;



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 (Workpiece Offset Amount) ; G10 L2/L20, G11

Function and purpose

The workpiece offset can be set or changed by the program using the G10 command.

During the absolute command (G90) mode, the commanded offset amount serves as the new offset amount, whereas during the incremental command (G91) mode, the sum of present offset and the commanded offset serves as the new offset amount.

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 Other than 0 to 6: Program error
X, Y, Z	Offset amount of each axis

Note

(1) The compensation amount in the G91 will be an incremental amount and will be accumulated each time the program is executed. Command G90 or G91 before G10 as much as possible.

(2) When address P is omitted, set the offset amount in the currently selected workpiece coordinate system (G54 to G59). When the G54.1 modal is active, a program error (P35) occurs.

Extended workpiece coordinate system offset input (L20)

G90 (G91) G10 L20 P_ X_ Y_ Z_ ;	
P	"n" No. of G54.1 Pn (1 to 300)
X, Y, Z	Offset amount of each axis

Note

(1) The specifications of the extended workpiece coordinate system selection are required.

(2) When address P is omitted, set the offset amount in the currently selected "G54.1 Pn". When the G54 to G59 modals are active, a program error (P33) occurs.

Offset input to the currently selected workpiece coordinate system (When the L command is omitted)

G90 (G91) 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 G54.1 Pn modal	
	1 to 300	: Extended workpiece coordinate offset amount setting (G54.1 Pn)
	Other than 1 to 300	: Program error (P35)
X, Y, Z	Offset amount of each axis	

Note

(1) When both of addresses P and L are omitted, set the offset amount in the currently selected workpiece coordinate system (one of G54 to G59, or G54.1 Pn).

Compensation input cancel

G11 ;



Detailed description

(1) G10 is non-modal command and is valid only in the commanded block.

(2) The G10 command does not perform any movement, but must not be used with G commands other than G54 to G59, G90 or G91.

(3) Do not command G10 in the same block as the fixed cycle and subprogram call command. This will cause malfunctioning and program errors.

(4) The workpiece offset input command (L2 or L20) should not be issued in the same block as the tool compensation input command (L10).

(5) If an illegal L No. is commanded, the program error (P172) occurs.
If the offset amount exceeds the maximum command value, the program error (P35) occurs.

(6) Decimal point inputs can be used for the offset amount.

(7) 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.

(8) 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.

(9) L2/L20 can be omitted when the workpiece offset is input.

(10) When the P command is omitted for workpiece offset input, it will be handled as the currently selected workpiece compensation input.

(11) If the G command that cannot be combined with G10 is issued in the same block, a program error (P45) occurs.

(12) A program error (P35) occurs for any value that does not match the compensation amount setting unit after command unit conversion.
With an incremental command, the setting range for the compensation amount is the sum of the present setting value and commanded value.

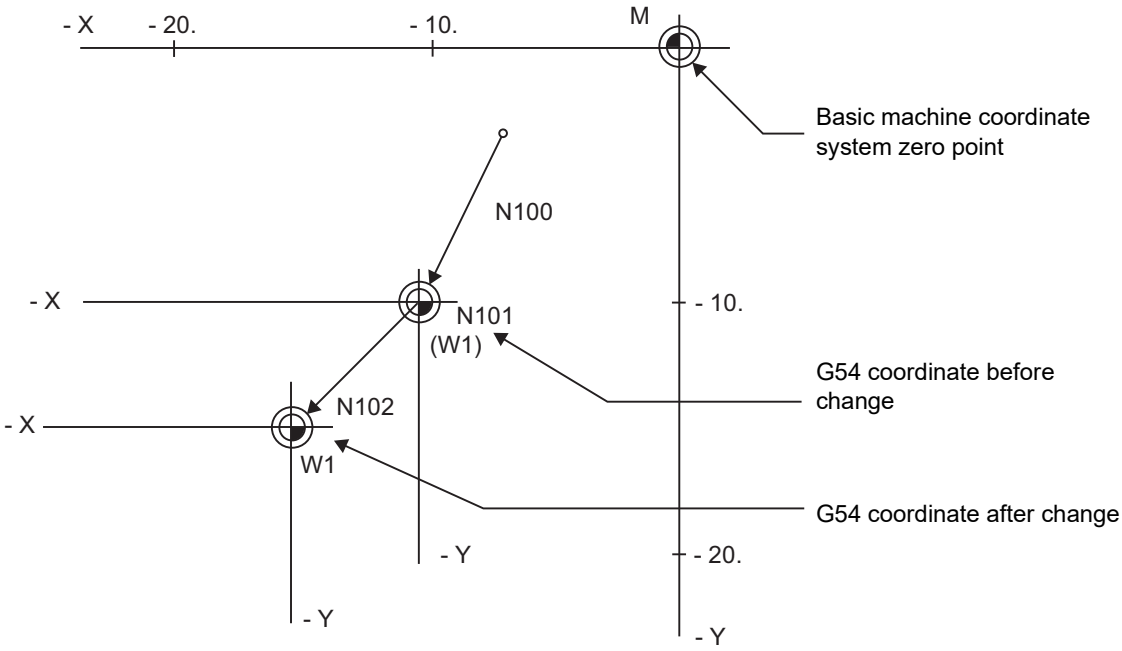
Program example

(1) 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

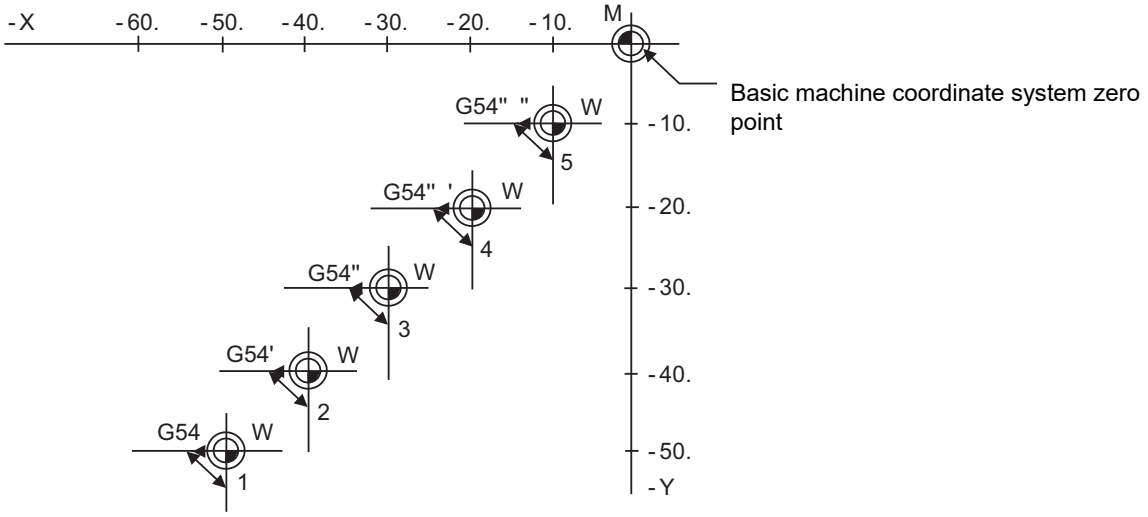
  Y = 0 Y = +5.000

When workpiece coordinate system offset amounts are 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 ;
```

(2) 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.

(2) It is recommended to command G10 in a block different from the one in which any of G54 to G59 and G54.1 is commanded. When G10 and any of G54 to G59 and G54.1 are commanded in the same block, it operates as follows.

When "#1274 ext10/bit5" is set to "0"

G10 G54 Pn Xx ;		Changes the modal to G54 and executes "G10 Pn".
G10 G54 Xx ;	(without "P")	Changes the modal to G54 and executes G10 for G54.
G10 G54.1 Pn Xx ;		Changes the modal to G54.1 Pn and executes G10 for "G54.1 Pn".
G10 G54.1 Xx ;	(without "P")	Program error (P33)

When "#1274 ext10/bit5" is set to "1" ("G54 Pn" is handled as "G54.1 Pn")

G10 G54 Pn Xx ;		Program error (P33)
G10 G55 Pn Xx ;		Changes the modal to G55 and executes "G10 Pn".
G10 G54 Xx ;	(without "P")	Changes the modal to G54 and executes G10 for G54.
G10 G54.1 Pn Xx ;		Program error (P33)
G10 G54.1 Xx ;	(without "P")	Program error (P33)

15.9.4 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 (G90) mode, the commanded tool compensation amount serves as a new one. During the incremental (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 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_I", "#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

15.9.5 Tool Shape Input by Program; G10 L100, G11

Function and purpose

This function sets tool shape data of the tool management screen by the machining program. Using this function saves the step to input tool shapes on the screen when the 3D checks is executed.

Command format

Tool shape settings from the program

G10 L100;	**Data setting start command**
P_ T_ K_ D_ H_ I_ J_ C_ ;	**Data setting command**

P	Data No.	Specify the data No. on the tool management screen. (Cannot be omitted.) The maximum value of data No. varies depending on the number of tool management data sets.
T	Tool No.	Specify the tool No. (Cannot be omitted.) 0 to 99999999 When "0" is specified, all the tool shape data of data No. specified by address P will be "0". In this case, only the tool shape data is changed.
K	Type	Designate the tool type using a numerical value. [Mill tool] 1: Ball end mill 2: Flat end mill 3: Drill 4: Radius end mill 5: Chamfer 6: Tap 7: Face mill
D	Shape data 1	Designate shape data of the tool. (Decimal point input enabled) The setting details of shape data differ depending on the tool type.
H	Shape data 2	
I	Shape data 3	Refer to the following "Correspondence between tool types and shape data" for the settings for each tool type.
J	Shape data 4	
C	Tool color	Specify the tool color. 1: Gray 2: Red 3: Yellow 4: Blue 5: Green 6: Light blue 7: Purple 8: Pink

G11;	**Data setting end command**

[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 radius (*1)						
2	Tool length						
3	-	-	Tool nose angle	Corner rounding	End angle	Pitch	Cutter length
4	-	-	-	-	End diameter (*1)	Thread diameter	Shank diameter

(*1) When "#8968 Tool shape radius" 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.

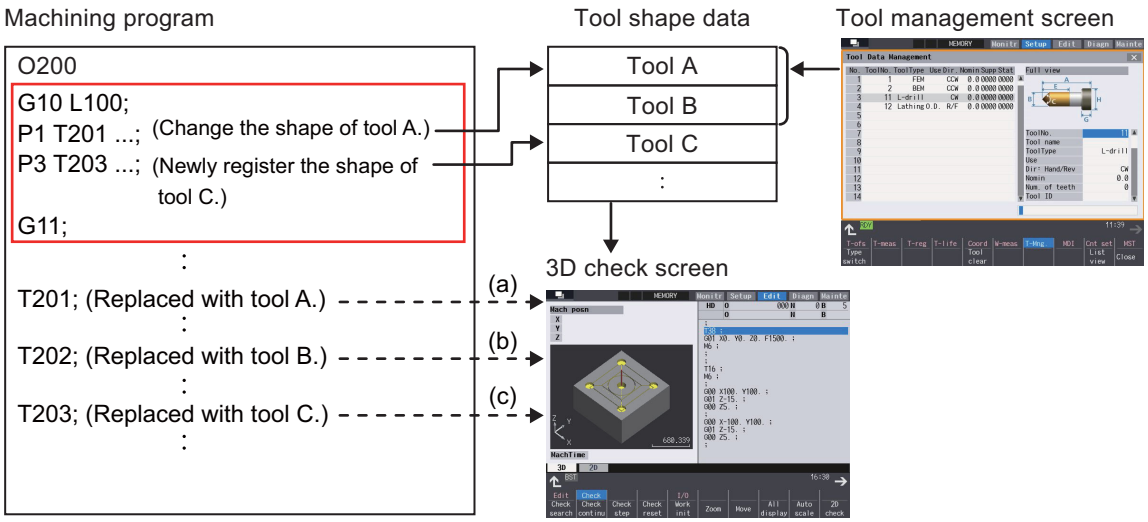
(3) For M80V Series, the tool shape data on the Tool management screen is rewritten during the graphic check.

(4) For M800VW Series, M800VS Series, and M80VW Series, this change is only reflected on the graphic check drawing. The tool shape data on the Tool management screen is not rewritten.

Detailed description

Tool shape settings from the program

This function sets the tool shape of the Tool management screen from the machining 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.



(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 alone in a 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 Decpt2" is valid for position commands (K address, W address).
Other command addresses comply with the minimum input unit ("#1015 cunit"). (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.

(7) The maximum value of "G10 L100" data No. (address P) is "80".

15.9.6 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 positions.
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**

- G10 must be commanded alone in a block, which also applies to G11.
  A program error (P423) will occur if it is not commanded alone in a block.
- 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**

- G10 must be commanded alone in a block, which also applies to G11.
  A program error (P423) will occur if it is not commanded alone in a block.
- 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 (θ1)
C: Longitude (θ2)
E to I: Vacuous

[M address: 2 (Latitude/Projection angle)]
B: Latitude (θ1)
C: Projection angle (θ2)
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 (θ1)
C: 2nd rotation angle (θ2)
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

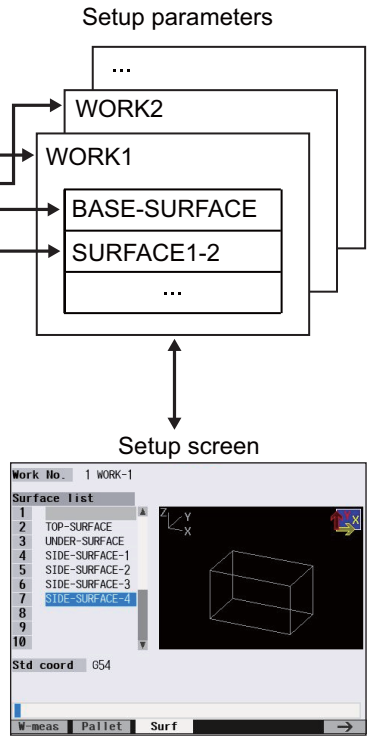
This function enables the R-Navi setup parameters to be configured from a machining program.
After the parameters have been configured from the program, you can check the values or select the machining surface from the setup screen.

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;
...
(Selecting workpiece: 1,  surface: 2)
G68.2 P10 Q1 D2;
```



Restrictions

(1) If the machining surface is selected or canceled while the block start interlock signal (*BSL) is turned OFF, the operation error (M01 0109) occurs. 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.

(2) The multiple-axis synchronization control cannot be used at the same time as R-Navi.
If the machining surface is selected during the multiple-axis synchronization control, the operation error (M01 0250) occurs.
If the multiple-axis synchronization control is performed while the machining surface is selected, the operation error (M50 5005) occurs.

15.10 Tool Life Management

15.10.1 Inputting the Tool Life Management Data by G10 L3 Command; G10 L3, G11

Function and purpose

Using the G10 command (non-modal command), the tool life management data can be registered, changed and added to, and preregistered groups can be deleted.

In tool life management II/III, it is possible to register, change, or add tool life management data and delete a registered group using the "G10L3" or "G10L30" command.
Such a command is not available in tool life management I. If commanded, a program error (P39) occurs.

Only group No. 1 can be used to register, change and add for the tool life management III.

Command format

Start of tool 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	Tool life
Q	Management method
T	Tool No. The spare tools are selected in the order of the tool Nos. registered here.
H	Tool length compensation No.
D	Tool radius compensation No.

Start of tool 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	Tool life
Q	Management method
T	Tool No.
H	Tool length compensation No.
D	Tool radius compensation No.

Start of tool life management data deletion

```
G10  L3 P2;
P_ ;          (First group)
P_ ;          (Next group)
```

P	Group No.

End of tool 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 life	(Ln)	0 to 65000 times (Managed by the number of times the tool was attached) 0 to 4000 minutes (Managed by the cutting hours)
Management method	(Qn)	1 to 3 1: Managed by the number of times the tool was attached 2: Managed by the cutting hours 3: Managed by the number of cuttings
Tool No.	(Tn)	1 to 99999999
Tool length compensation No.	(Hn)	0 to 999 (*)
Tool radius compensation No.	(Dn)	0 to 999 (*)

(*) The setting range of the tool compensation No. varies depending on the specification of the "number of tool offset sets".
If a value exceeding each command range is issued, a program error (P35) occurs.

Operation example

	Program example	Operation
Data registration	G10 L3 ;	After deleting all group data, registration starts.
	P10 L10 Q1 ;	Group No. "10" is registered.
	T10 H10 D10 ;	Tool No. "10" is registered in group No. "10".
	G11 ;	Registration ends.
	M02 ;	The program ends.
Group change, addition	G10 L3 P1 ;	Changing and addition of the group and tool starts.
	P10 L10 Q1 ; T10 H10 D10 ;	The change and addition operation takes place in the following manner. (1) When group No. "10" has not been registered. •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. •Tool No. "10" is additionally registered in group No. "10". (3) When group No. "10" and tool No. "10" have been both registered. •The tool No. "10" data is changed.
	G11;	The group and tool change and addition ends.
	M02 ;	The program ends.
Group deletion	G10 L3 P2 ;	The group deletion starts.
	P10 ;	The group No. "10" data is deleted.
	G11 ;	The group deletion ends.
	M02 ;	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 (non-modal 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 management method, the tool length compensation and tool radius compensation can be registered/changed with the tool compensation amount format.

Command format

Start of tool 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	Tool life
Q	Management method
T	Tool No. The spare tools are selected in the order of the tool Nos. registered here.
H	Tool length compensation No. or tool length compensation amount
R	Tool radius compensation No. or tool radius compensation amount

L_, Q_, T_, H_, and R_ cannot be omitted. If omitted, a program error (P33) occurs.

Start of tool 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	Tool life
Q	Tool length compensation data format, tool radius compensation data format, management method
T	Tool No.
H	Tool length compensation No. or tool length compensation amount
D	Tool radius compensation No. or tool radius compensation amount

L_, Q_, T_, H_, and R_ cannot be omitted. If omitted, a program error (P33) occurs.

Start of tool life management data deletion

```
G10  L30 P2;
P_ ;          (First group)
P_ ;          (Second group)
```

P	Group No.

End of tool 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
Management method	(Qabc)	abc: Three integer digits a. Tool length compensation data format 0: Compensation No. 1: Incremental compensation amount 2: Absolute compensation amount b. Tool radius compensation data format 0: Compensation No. 1: Incremental compensation amount 2: Absolute compensation amount c. Tool management method 0: Cutting hours 1: Number of mounts 2: Number of cuttings
Tool life	(Ln)	Refer to the table below.
Tool length compensation No./amount	(Hn)	
Tool radius compensation No./amount	(Rn)	

Management method (Qabc)			Tool length compensation (Hn)	Tool radius compensation (Rn)	Tool life (Ln)
a	b	c			
×	×	0	-	-	0 to 4000 (Cutting hours)
×	×	1	-	-	0 to 65000 (Number of times the tool was mounted)
×	×	2	-	-	0 to 65000 (Number of cuttings)
×	0	×	-	0 to 999 (Compensation No.) (*1)	-
×	1	×	-	±99999.999 (Incremental compensation amount)	-
×	2	×	-	±99999.999 (Absolute compensation amount)	-
0	×	×	0 to 999 (Compensation No.) (*1)	-	-
1	×	×	±99999.999 (Incremental compensation amount)	-	-
2	×	×	±99999.999 (Absolute compensation amount)	-	-

(*1) The setting range of the tool compensation No. varies depending on the specification of the "number of tool offset sets".
If a value exceeding each command range is issued, a program error (P35) occurs.

Operation example

	Program example	Operation
Data registration	G10 L30 ; P10 L10 Q001 ; T10 H10 R10 ; G11 ; M02 ;	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 ;	1. Changing and addition of the group and tool starts. 2. The change and addition operation takes place in the following manner. (1) When group No. "10" has not been registered. (a) Group No. "10" is additionally registered. About the change and addition tool; •Tool management method: count the number of cuttings •Tool length compensation: the incremental compensation amount method •Tool radius compensation: the absolute compensation amount method. (b) Tool No. "10" is registered in group No. "10". (c) "0.5" is registered to the length data that the compensation No. of the tool length shows, and "0.25" is registered to the radius data that the compensation No. of the tool radius shows. (2) When group No. "10" has been registered, but tool No. "10" has not been registered. •Tool No. "10" is additionally registered in group No. "10". (3) When group No. "10" and tool No. "10" have both been registered. •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 ;	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



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) occurs.

(3) When two or more addresses are commanded in one block, the latter address will be valid.

(4) If the tool life data (L_) is omitted in the "G10L3" command, the tool 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) occurs.

(10) The registered data is held even if the power is turned OFF.

(11) When "G10 L3" or "G10 L30" is commanded, the commanded group and tool will be registered after all of the registered data is erased.

(12) 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.

(13) 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.

(14) The commands of "G10L3" and "G10L30" cannot be used for the tool life management I. If commanded, the program error (P39) occurs.

(15) The setting range of the tool compensation No. depends on the MTB specifications.

(16) Only group No. 1 can be used to register, change and add for the tool life management III.

15.10.4 Tool Life Management Set Allocation 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.

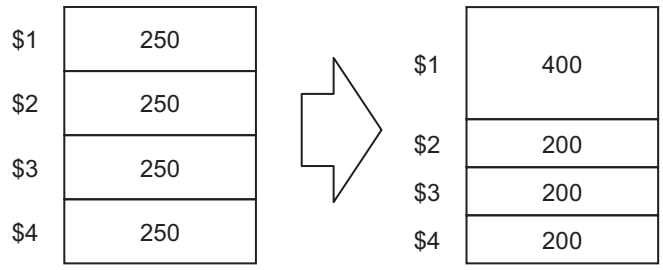
Fixed allocation: Automatically and evenly allocates the number of tool life management sets to each part system.

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 1000 sets.

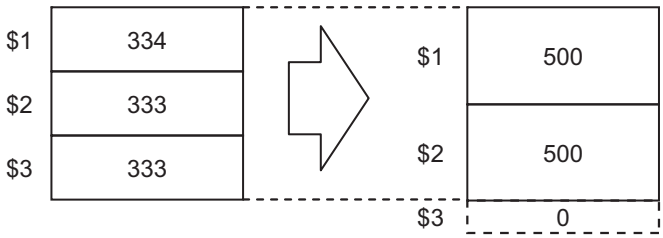
(1) Arbitrary allocation (with #1439 = 1)

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.

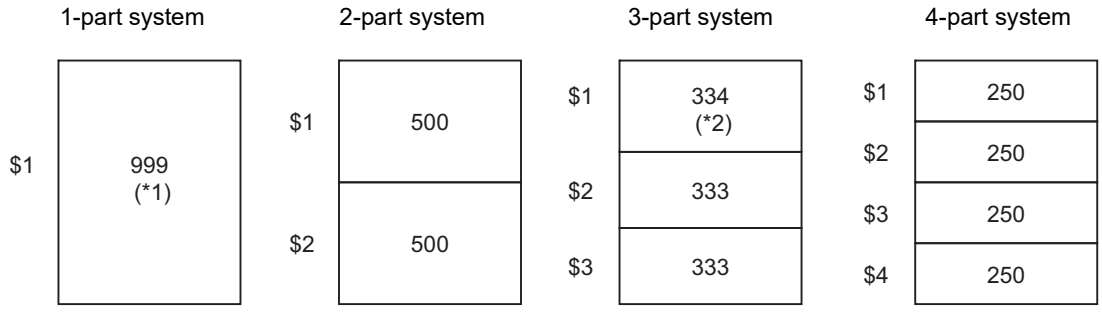
(a) When the number of tool life management sets is increased for the 1st part system ($1) of 4-part system



(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



(2) Fixed 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) For 1-part system, up to the number of tool life management sets in the system is available regardless of the parameter setting.

(2) 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.

(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, system alarm (Y05) is generated even if the specification allows arbitrary allocation.

(4) Even if the specification allows arbitrary allocation, fixed allocation is applied if the parameter "#12055 Tol-lifenum" is set to "0" for all part systems.

(5) 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 registered.

A message is displayed when it exceeds the number of tool life management sets, and the remaining tool life management data are not entered.

Set the unregistered tool life management data separately.

(6) When the number of tool life management sets is changed, erase all the tools (*1) on the maintenance screen, then reboot the NC and register the tool life data again. To change the number of tool life management sets, contact the MTB representative.

(*1) When all the tools are erased, the tool compensation data is also erased. Output the tool offset data file (TOOL.OFS) before erasing all the tools to back up the tool compensation data.

(7) For fixed allocation, when the number of the tool life management sets is "200" and the parameter "#1046 T-ofs disp type" (Tool compensation type switch) (M system) is set to "1", the number of tool life management sets in the system is "99".

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

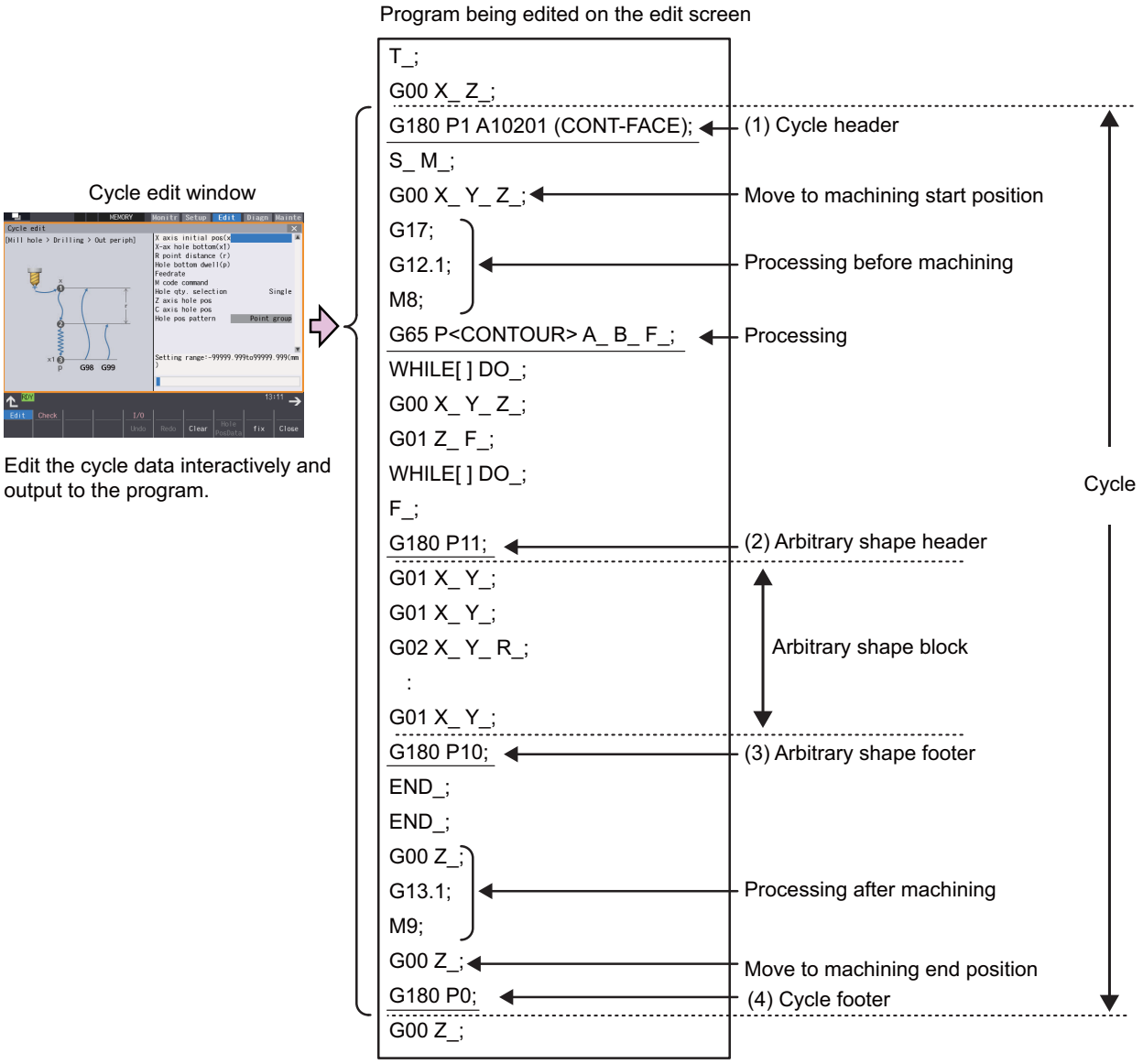
G180 P__ A__ ;

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:



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 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.

#	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 command axis name	U	W	H (*3)	U	W	U	W
1601	axnameEx	Axis name extension character	None	None	None	A	A	B	B
Absolute command axis name			X	Z	C	XA	ZA	XB	ZB
Incremental command axis name			U	W	H	UA	WA	UB	WB

(*3) "H" can be set to "#1014 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 command axis name (*1)	U	W	None	V
1601	axnameEx	Axis name extension character (*2)	A	A	B	None
Absolute command axis name			XA	ZA	ZB	Y
Incremental 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 user 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.

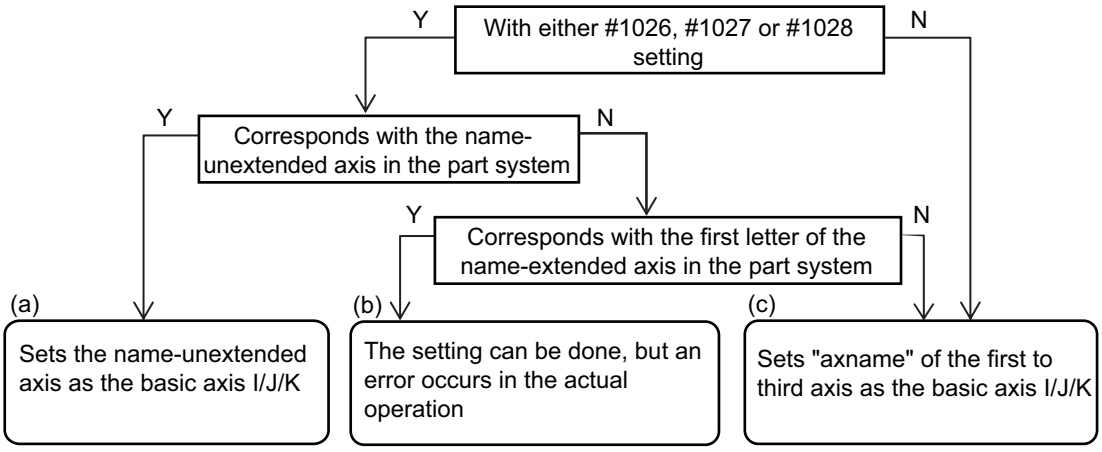
(b) When the NC is operated with the setting value of the base axis I, J, or K as follows, the program error (P11) occurs.

- 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
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

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

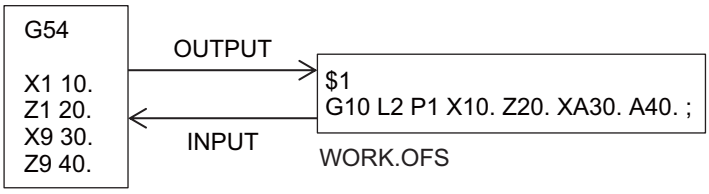
(2) G10 command in input/output file

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:

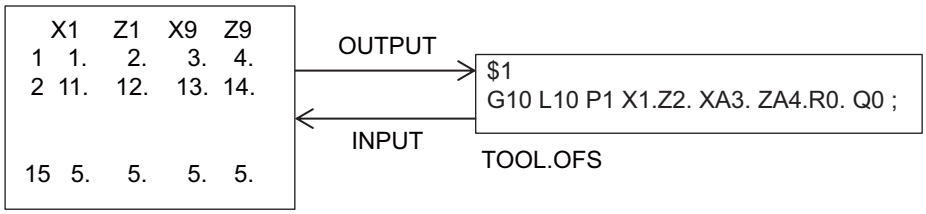
[Example of axis configuration]

$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]

Axis names	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 (Z23) occurs at the time of power ON.

$1	Normal axis configuration				Abnormal axis configuration (System error (Z23) 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.

(6) If the parameter "#1266 ext02/bit0" (Axis name extension valid) is set to "0" (invalid) and the program using the "Name-extended axis" is executed as it is, the machine may operate differently from the original command. For example, the command "XA10.000" for the "Name-extended axis" may be interpreted as "X0 A10.000". To prevent such misinterpretation in commands, it is recommended to use the parameter "#1227 aux11/bit4" as "1" (valid).

15.13 Vibration Cutting Control (VCC); G08.5



Function and purpose

The low-frequency vibration can be applied to the feed axis in the turning/drilling mode with the vibration cutting control (G08.5) command.

While cutting machining is performed, chips can be dispersed by applying vibrations in synchronization with the spindle rotation to the feed axis so that an air-cutting zone is made. Difficult-to-cut workpieces such as stainless steel can be machined easily and at high speed. In addition, this function can reduce the heat generated by machining and extend tool life. Only one axis vibrates in the commanded part system. Even if the cutting command is issued to two or more axes such as the taper, only the single specified axis vibrates.

This function may affect the surface quality of the workpiece as the axis vibrates, therefore, use it for machining such as rough machining.

Enabling conditions of the function

The following conditions must be satisfied to use this function. (This depends on the MTB specifications.)

(1) The vibration cutting expansion unit (FCU8-EX744) is mounted.

<Note>
- If G08.5 is commanded while the expansion unit is not mounted, the program error (P39) occurs.
- If the expansion unit fails in authentication, the system alarm (Z106) occurs when NC starts. When the system alarm (Z106) is displayed, contact your service representative. This alarm can be cleared by pressing the Reset button. However, if G08.5 is commanded, the program error (P39) occurs.

(2) The parameter "#12570 VibCutting_VCC" of the feed axis is set to "1". (The setting depends on the MTB specifications.)

<Note>
- When the parameter of the axis that enables the vibration cutting control during turning/drilling is set to "0", cutting machining is performed without vibration.
- The vibration cutting control is not available for the rotary axis or PLC axis. Even if the parameter is set to "1", the vibration cutting control is disabled.

(3) The parameter "#1037 cmdtyp" is set to "1" or "2" (G code system 1).

<Note>
- If the G08.5 command is issued in a system other than G code system 1, the program error (P34) occurs.

(4) OMR-FF that performs operation during vibration cutting is set.

(a) The parameter "#2313 SV113 SSF8/bit0" (OMR-FF control enabled) is set to "1".

(b) Set the parameter "#2306 SV106 PGM" (OMR-FF scale model gain).

(c) Set the parameter "#2139 omrff_off" to "1" (Temporarily disabled).

(By executing (c), OMR-FF is only enabled during vibration cutting mode. To enable OMR-FF in a mode other than the vibration cutting mode, set "0".)

<Note>
- If the parameter "#2313 SV113 SSF8/bit0" is set to "0", the operation error (M01 1302) is displayed. Vibration cutting is performed while OMR-FF is disabled.
- If OMR-FF is enabled in modes other than the vibration cutting mode ("#2139 omrff_off" is "0") and you want to set different scale model gains in the vibration cutting mode and in modes other than the vibration cutting mode, set the scale model gain during the vibration cutting mode to the parameter "#12583 VibPGMsp_VCC". However, if the difference between the parameter "#2306 SV106 PGM" and the parameter "#12583 VibPGMsp_VCC" is great, a load may be applied to the machine when the vibration cutting mode is started or canceled.

Spindle speed control method

There are the following two methods of the spindle rotation speed control method during vibration cutting mode.

[Fixed spindle speed selection method]

The operation is performed at the spindle rotation speed closest to the S command value of the program among the candidates of spindle rotation speed generated in the NC. The spindle rotation speed during vibration cutting has a slight difference from the S command value. The spindle rotation speed cannot be changed while the cutting block is being executed.

[Variable spindle speed method]

The spindle rotation speed is as specified with the S command value in the program. The spindle rotation speed can be changed while the cutting block is being executed.

The control method is selected as follows according to "#11686 Spec_VCC/bit6" (Spindle speed control method) and the status of the spindle selected at the start of vibration cutting mode.

Spindle selected at the start of vibration cutting mode	#11686 Spec_VCC/bit6 = 0 (The control method is automatically switched.)	#11686 Spec_VCC/bit6 = 1 (Variable spindle speed method is always in effect.)
During constant surface speed control (G96)	Variable spindle speed method	Variable spindle speed method
During constant surface speed control cancel (G97)	Fixed spindle speed selection method	Variable spindle speed method

Command format

The vibration cutting mode can be switched to ON or OFF by commanding G08.5 from the machining program.

When a cutting command is issued while the mode is ON, the tool performs cutting machining while vibrating.

Starting the vibration cutting mode

```
G08.5 P2 α0 I__ K__ ;
```

P	The vibration cutting mode start/cancel command (0, 2)
α	Axis address (Be sure to command "0" following the axis address.)
I	Frequency (0.10 to 31.50)
K	Amplitude feed ratio (0.01 to 9.99)

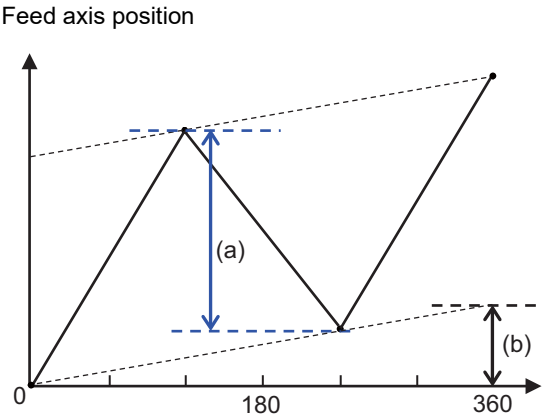
Canceling the vibration cutting mode

```
G08.5 P0 ;
```

Detailed description

Explanation of address

Address	Command range (unit)	Remarks
P	0/2	⬧Issues the vibration cutting mode start/cancel command. ⬧The command range is as follows. 0: Cancels the vibration cutting mode. 2: Vibration cutting mode ⬧When address "P" is omitted, the program error (P33) occurs. ⬧If an invalid value is commanded, the program error (P35) occurs.
α	X/Y/Z/U/V/W/A/B/C/H	⬧Commands the axis address that vibrates with the cutting command (G01/G02/G03) on two or more axes during the vibration cutting mode. The command value is held until the vibration cutting mode is canceled. ⬧For the axis address, specify the address set in "#1013 axname" or "#1014 incax". The axis to be vibrated is determined depending on the MTB specifications. ⬧If the commanded axis address does not exist in the commanded part system, the program error (P32) occurs. ⬧Only one axis can be commanded with the axis address. If two or more axes are commanded, the program error (P33) occurs. ⬧If address α is omitted at the start of the mode, the vibration cutting control is temporarily disabled for the cutting commands (G01/G02/G03) of two or more axes.
	0	⬧Command "0" following the axis address. ⬧If "0" is omitted, the program error (P33) occurs when the basic specification parameter "#1227 aux11/bit4" is set to "1" (word command value check: valid). ⬧If an invalid value is commanded, the program error (P35) occurs.
I	0.10 to 31.50	⬧Commands the frequency for one spindle rotation during vibration cutting mode. The command value is held until the vibration cutting mode is canceled. ⬧If the value of this address exceeds the commanded range or the setting value of the parameter "#11678 VibPerRevMax_VCC" (*1), the program error (P35) occurs. ⬧When address "I" is omitted at the start of the mode, the operation varies depending on the control method. [Fixed spindle speed selection method] The optimal spindle rotation speed is automatically selected for the command spindle rotation speed by a frequency that does not exceed the setting value of the parameter "#11678 VibPerRevMax_VCC". [Variable spindle speed method] The frequency is 0.5 times.

Address	Command range (unit)	Remarks
K	0.01 to 9.99	♦Commands the amplitude feed ratio during vibration cutting mode. The amplitude (retract amount at vibration cutting) in the mode is determined from the amplitude feed ratio and the feed value per spindle rotation. The command value is held until the vibration cutting mode is canceled.  (When the address "I" (Frequency) is set to "1.5") (a) Amplitude (retract amount) = Amplitude feed ratio x Feed amount for one spindle rotation (b) Feed amount for one rotation ♦If the value commanded to this address exceeds the commanded range or the setting value of the parameter "#11671 AmpFeedRateMax_VCC" (*1), the program error (P35) occurs. ♦When address K is omitted at the start of the mode, the setting value of the parameter "#11670 AmpFeedRate_VCC" (*1) is regarded as the amplitude feed ratio during mode. If the setting value of the parameter #11670 exceeds that of the parameter #11671, the program error (P35) occurs.

(*1) These settings depend on the MTB specifications.

Note

(1) If address "α", "I", or "K" is commanded to the "G08.5 P0" block, the program error (P32) occurs.

(2) G08.5 is an unmodal command of G code group 0.

(3) Do not set an integer value for address "I" (frequency). When an integer value is set, the chips cannot be dispersed as an air-cutting zone is not made. Theoretically, when the frequency is specified with "integer value + 0.5", the air-cutting area becomes the maximum.

(4) An axis that vibrates by vibration cutting operates at a speed (vibration feedrate) where the vibration components are added to the command speed. The vibration feedrate is clamped with the parameter "#12586 ActualClamp_VCC".

Command to invalidate the vibration cutting control temporarily

When the ",V0" command is issued in the cutting block (G01/G02/G03) during the vibration cutting mode, the vibration cutting control can be disabled temporarily.

G01 Z__ X__ F__ ,V0 ;	Linear interpolation command
G02 / G03 Z__ X__ I__ K__ F__ ,V0 ;	Circular interpolation command
G02 / G03 Z__ X__ R__ F__ ,V0 ;	R-specified circular interpolation command

Address	Command range	Remarks
,V	0	♦Address ",V" is valid only in the cutting command block. ♦Vibration cutting is performed when the ",V" command is omitted in the cutting block during the vibration cutting mode. ♦",V" command is ignored if a value other than "0" is commanded. ♦The ",V" command is ignored if it is commanded in a mode other than the vibration cutting mode.

Note

(1) If the next block of the vibration cutting enabled block is in the vibration cutting disabled state, the vibrations of the axis to be vibrated converges. The block in the vibration cutting disabled state starts moving without waiting for the vibration to converge.

(2) Vibration cutting is performed in the cutting block in the fixed cycle.

(3) In the block inserted by tool nose R offset or corner chamfering, vibration cutting is performed. The temporary invalid command of the cutting block is also applied to the inserted block in tool nose R offset and corner chamfering.

(4) While vibration cutting control is temporarily disabled, clamping of the vibration feedrate is also temporarily disabled.

Relationship with other functions

Relationship with other G code functions

Column A: Operation to be performed when the combined functions (G code shown in the left) are commanded during vibration cutting mode.

Column B: Operation to be performed when the vibration cutting mode start command is issued while the combined functions are enabled.

o: The command is available.

Δ: The command is available; however, the vibration cutting control is disabled.

x: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)

-: No combination

G code	Function name	A	B
G00	Positioning	Δ	○
G0.5 P1	Rapid traverse block overlap function: Valid	○	○
G0.5 P0	Rapid traverse block overlap: Invalid	○	○
G01	Linear interpolation	○	○
G02	Circular interpolation CW	○	○
	Helical interpolation CW	× (P790)	○
	Spiral interpolation CW (type2)	○	○
	Conical interpolation CW (type2)	× (P790)	○
G03	Circular interpolation CCW	○	○
	Helical interpolation CCW	× (P790)	○
	Spiral interpolation CCW (type2)	○	○
	Conical interpolation CCW (type2)	× (P790)	○
G02.1	Spiral interpolation CW (type1)	○	○
	Conical interpolation CW (type1)	× (P790)	○
G03.1	Spiral interpolation CCW (type1)	○	○
	Conical interpolation CCW (type1)	× (P790)	○
G02.2	Involute interpolation/Helical involute interpolation CW	× (P790)	× (P791)
G03.2	Involute interpolation/Helical involute interpolation CCW	× (P790)	× (P791)
G02.3	Exponential function interpolation positive rotation	× (P790)	× (P791)
G03.3	Exponential function interpolation negative rotation	× (P790)	× (P791)
G02.4	3-dimensional circular interpolation CW	× (P790)	× (P791)
G03.4	3-dimensional circular interpolation CCW	× (P790)	× (P791)
G04	Dwell	○	-
G05 P1	High-speed machining mode I ON	○	○
G05 P2	High-speed machining mode II ON		
G05 P10000	High-speed high-accuracy control II ON	○	○
G05 P20000	High-speed high-accuracy control III ON		
G05 P0	High-speed machining mode I/II OFF High-speed high-accuracy control II OFF	○	○
G05.1 Q1	High-speed high-accuracy control I ON	○	○
G05.1 Q2	Spline interpolation ON	× (P790)	× (P791)
G05.1 Q0	High-speed high-accuracy control I OFF	○	○
G06.2	NURBS interpolation	× (P790)	× (P32)
G07	Hypothetical axis interpolation	× (P790)	× (P791)
G07.1 G107	Cylindrical interpolation	× (P790)	× (P481)

G code	Function name	A	B
G08 P1	High-accuracy control ON	○	○
G08 P0	High-accuracy control OFF	○	○
G09	Exact stop check	○	-
G10 L70 G10 L100	Parameter input by program Tool shape input by program	○	× (P421)
G10 L101	Material shape input by program	○	× (P422)
G10 L2 G10 L20 G10 L10 G10 L11 G10 L12 G10 L13	Workpiece coordinate system offset input Extended workpiece coordinate system offset input Tool length compensation (Shape compensation) Tool length compensation (Wear compensation) Tool radius compensation (Shape compensation) Tool radius compensation (Wear compensation)	○	-
G10 L3	Tool life management data registration start	○	× (P33)
G10 L14	Programmable current limitation	○	-
G10 L110 G10 L111	R-Navi data input by program Start setting machining surface data	○	-
G10 I_J_ G10 K_	Parameter coordinate rotation input	× (P790)	-
G11	Data input by program cancel	○	○
G10.9	Diameter/Radius designation switch	○	○
G12	Circular cutting CW	× (P790)	-
G13	Circular cutting CCW	× (P790)	-
G12.1 G112	Polar coordinate interpolation ON	× (P790)	× (P481)
G13.1 G113	Polar coordinate interpolation cancel	○	○
G15	Polar coordinate command OFF	○	○
G16	Polar coordinate command ON	× (P790)	× (P791)
G17	X-Y plane selection	○	○
G18	Z-X plane selection	○	○
G19	X-Z plane selection	○	○
G20	Inch command	○	○
G21	Metric command	○	○
G22	Stroke check before travel ON	○	○
G23	Stroke check before travel OFF	○	○
G27	Reference position check	Δ	-
G28	Automatic reference position return	Δ	-
G29	Start position return	Δ	-
G30	2nd, 3rd and 4th reference position return	Δ	-
G30.1	Tool exchange position return 1	Δ	-
G30.2	Tool exchange position return 2	Δ	-
G30.3	Tool exchange position return 3	Δ	-
G30.4	Tool exchange position return 4	Δ	-
G30.5	Tool exchange position return 5	Δ	-
G30.6	Tool exchange position return 6	Δ	-
G31	Skip/Speed change skip Multi-step skip 2	× (P790)	-
G31.1	Multi-step skip 1-1	× (P790)	-
G31.2	Multi-step skip 1-2	× (P790)	-
G31.3	Multi-step skip 1-3	× (P790)	-
G33	Thread cutting	× (P790)	× (P791)

G code	Function name	A	B
G34	Special fixed cycle (bolt hole circle)	○	-
G35	Special fixed cycle (line at angle)	○	-
G36	Special fixed cycle (arc)	○	-
G37	Automatic tool length measurement	× (P790)	-
G37.1	Special fixed cycle (grid)	○	-
G38	Tool radius compensation vector designation	○	-
G39	Tool radius compensation corner arc	○	-
G40	Tool radius compensation (tool nose radius compensation) cancel 3-dimensional tool radius compensation cancel 3-dimensional tool radius compensation (Tool's vertical-direction compensation) cancel	○	○
G41	Tool radius compensation (tool nose radius compensation) left	○	○
	3-dimensional tool radius compensation (left)	× (P790)	× (P791)
G42	Tool radius compensation (tool nose radius compensation) right	○	○
	3-dimensional tool radius compensation (right)	× (P790)	× (P791)
G40.1 G150	Normal line control cancel	○	○
G41.1 G151	Normal line control left ON	× (P790)	× (P791)
G42.1 G152	Normal line control right ON	× (P790)	× (P791)
G41.2	3-dimensional tool radius compensation (Tool's vertical-direction compensation) (left)	× (P790)	× (P791)
G42.2	3-dimensional tool radius compensation (Tool's vertical-direction compensation) (right)	× (P790)	× (P791)
G43	Tool length compensation (+)	○	○
G44	Tool length compensation (-)	○	○
G43.1	Tool length compensation along the tool axis ON	× (P790)	× (P791)
G43.4	Tool center point control type 1 ON	× (P790)	× (P942)
G43.5	Tool center point control type 2 ON	× (P790)	× (P942)
G43.7	Tool position compensation start	○	○
G43.8	Tool cutting point control type 1	× (P790)	× (P912)
G43.9	Tool cutting point control type 2	× (P790)	× (P912)
G45	Tool position offset (extension)	○	-
G46	Tool position offset (reduction)	○	-
G47	Tool position offset (double elongation)	○	-
G48	Tool position offset (double contraction)	○	-
G49	Tool length compensation cancel Tool length compensation along the tool axis cancel Tool center point control cancel Tool position compensation cancel	○	○
G50	Scaling cancel	○	○
G51	Scaling ON	× (P790)	× (P791)
G50.1	Mirror image by G code cancel	○	○
G51.1	Mirror image by G code ON	○	○
G50.2 G250	Tool spindle synchronization IB/IC cancel	○	○
G51.2 G251	Tool spindle synchronization IB/IC	×	×

G code	Function name	A	B
G52	Local coordinate system setting	○	-
G53	Basic machine coordinate system selection	Δ	-
G53.1	Tool axis direction control (type 1)	× (P790)	-
G53.6	Tool axis direction control (type 2)	× (P790)	-
G54	Workpiece coordinate system selection 1	○	○
G55	Workpiece coordinate system selection 2	○	○
G56	Workpiece coordinate system selection 3	○	○
G57	Workpiece coordinate system selection 4	○	○
G58	Workpiece coordinate system selection 5	○	○
G59	Workpiece coordinate system selection 6	○	○
G54.1	Extended workpiece coordinate system selection	○	○
G54.2	Workpiece position offset for rotary axis	× (P790)	× (P791)
G54.4	Workpiece installation error compensation	× (P790)	× (P791)
G60	Unidirectional positioning	Δ	○ (*1)/-
G61	Exact stop check mode	○	○
G61.1	High-accuracy control ON	○	○
G61.2	High-accuracy spline	× (P790)	× (P791)
G61.4	Spline interpolation 2	× (P790)	× (P791)
G62	Automatic corner override	○	○
G63	Tapping mode	○	○
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	○	○
G68	Coordinate rotation by program ON 3-dimensional coordinate conversion mode ON	× (P790)	× (P791)
G68.2	Inclined surface machining command	× (P790)	× (P951)
	R-Navi data input (Selecting the registered machining surface)	○	-
G68.3	Inclined surface machining command (Based on tool axis direction)	× (P790)	× (P951)
G69	Coordinate rotation by program cancel 3-dimensional coordinate conversion mode OFF Inclined surface machining cancel R-Navi data input (Canceling the selected machining surface)	○	○
G70	User fixed cycle	× (P790)	× (P791)
G71	User fixed cycle	× (P790)	× (P791)
G72	User fixed cycle	× (P790)	× (P791)
G73	Fixed cycle (step)	○ (*2)	× (P791)
G74	Fixed cycle (reverse tapping)	× (P790)	× (P791)
G74.5 G74.6 G74.8	Reverse Punchtap cycle	× (P790)	× (P791)
G75	Fixed cycle (circle cutting cycle)	× (P790)	× (P791)
G76	Fixed cycle (fine boring)	× (P790)	× (P791)
G77	User fixed cycle	× (P790)	× (P791)
G78	User fixed cycle	× (P790)	× (P791)
G79	User fixed cycle	× (P790)	× (P791)

G code	Function name	A	B
G80	Fixed cycle cancel	○	○
G81	Fixed cycle (drill/spot drill)	○ (*2)	× (P791)
G82	Fixed cycle (drill/counter boring)	○ (*2)	× (P791)
G83	Fixed cycle (deep-hole drilling/small-diameter deep-hole drilling)	○ (*2)	× (P791)
G84	Fixed cycle (tapping)	× (P790)	× (P791)
G84.5 G84.6 G84.8	Punchtap cycle	× (P790)	× (P791)
G85	Fixed cycle (boring)	○ (*2)	× (P791)
G86	Fixed cycle (boring)	○ (*2)	× (P791)
G87	Fixed cycle (back boring)	× (P790)	× (P791)
G88	Fixed cycle (boring)	× (P790)	× (P791)
G89	Fixed cycle (boring)	○ (*2)	× (P791)
G90	Absolute command	○	○
G91	Incremental command	○	○
G92	Coordinate system setting Spindle clamp speed setting	○	-
G92.1	Workpiece coordinate system preset	○	-
G93	Inverse time feed	× (P790)	○
G94	Feed per minute (asynchronous feed)	× (P790)	○
G95	Feed per revolution (synchronous feed)	× (P790)	○
G96	Constant surface speed control ON	(*4)	○
G97	Constant surface speed control OFF	(*4)	○
G98	Fixed cycle (Initial level return)	○	○
G99	Fixed cycle (R point level return)	○	○
G100 - G225	User macro (G code call) Maximum 10	○	○
G113.1	Spindle synchronization I/tool spindle synchronization (IA/IB/II) cancel	○	○
G114.1	Spindle synchronization I	○ (*3)	○ (*3)
G114.2	Tool spindle synchronization IA (Spindle - spindle synchronization) mode ON	×	×
G114.3	Tool spindle synchronization II (Hobbing) ON	×	×
G115	Start point designation timing synchronization Type 1	○	-
G116	Start point designation timing synchronization Type 2	○	-
G120.1	Machining condition selection I	○	○
G121	Machining condition selection I cancel	○	-
G122	Sub part system control I	○	○
G127	Prohibits the reverse run in all part systems.	○	-
G130	Selection of axis (axes) for feedrate command	× (P790)	× (P791)
G136	Two-dimensional barcode engraving cycle	× (P790)	-
G140	Arbitrary axis exchange command	×	○
G141	Arbitrary axis exchange return command	×	○
G142	Arbitrary axis exchange/reference axis arrange return command	×	○
G145	Cancel sub part systems	○	○
G160	Torque limitation skip	× (P790)	-
G162	Spindle speed fluctuation detection	○	○
G162.1	Cutting load control	× (P790)	× (P791)
G163	Spindle speed fluctuation detection cancel	○	○

G code	Function name	A	B
G174	Longitudinal cutting fixed cycle	○ (*2)	○ (*2)
G175	Thread cutting fixed cycle	× (P790)	× (P791)
G176	Face cutting fixed cycle	○ (*2)	○ (*2)
G180	Interactive cycle insertion program	○	○
G186	Interference check III: Interfering object data valid command	○	-
G187	Thread milling cycle	× (P790)	-

(*1) When parameter "#1271/bit3" is set to "1" (This depends on the MTB specifications.)

(*2) The feed axis vibrates in the cutting block in the fixed cycle.

(*3) Only the reference spindle can be commanded.

(*4) The operation varies depending on the control method.
Refer to the "Constant surface speed control" section in this chapter.

Spindle function (S command)

[Fixed spindle speed selection method]

(1) The spindle rotation speed closest to the S command is selected from the vibration cutting conditions, and is set as the spindle rotation speed during the vibration cutting mode.

(2) If the vibration cutting mode is canceled, the spindle rotates with the issued S command.

(3) When the S command is issued to the selected spindle while the program is executing the cutting block during the vibration cutting mode, this function is enabled at the timing at which the cutting block is switched to the non-cutting block. However, the "S0" command is enabled immediately.

[Variable spindle speed method]

(1) The spindle rotation speed during vibration cutting mode is the same as the S command value.

(2) The S command to the selected spindle during vibration cutting mode is enabled immediately even while the cutting block is being executed.

Spindle synchronization, tool spindle synchronization IA/IB/IC/II

(1) If any of the following operations is performed for the spindle selected by the part system during the vibration cutting mode, the operation error (M01 1300) occurs.

- The target spindle is set as the synchronized spindle for spindle synchronization.
- The target spindle is set as the reference spindle or synchronized spindle for tool spindle synchronization IA/IB/IC (polygon).
- The target spindle is set as the hob axis for tool spindle synchronization II (hobbing).

(2) When any of the following spindles is selected in the part system at the start of the vibration cutting mode, the operation error (M01 1301) occurs.

- Synchronized spindle for spindle synchronization
- Reference spindle/synchronized spindle in tool spindle synchronization IA/IB/IC (polygon)
- Hob axis in tool spindle synchronization II (hobbing)

Constant surface speed control

[Fixed spindle speed selection method]

(1) If the constant surface speed control is commanded to the spindle selected by the part system during the vibration cutting mode, the operation error (M01 1300) occurs.

[Variable spindle speed method]

(1) When "#11686 Spec_VCC/bit6" is "0" (the spindle speed control method is automatically switched)

Constant surface speed control of the selected spindle cannot be turned OFF during vibration cutting mode. The selected spindle cannot be switched during vibration cutting mode. The operation error (M01 1300) occurs.

(2) When "#1447 G96_tmp_cancel" is "1" (temporary cancel of constant surface speed control is valid)

A spindle in constant surface speed control cannot be used as the selected spindle for vibration cutting control. The operation error (M01 1300 / M01 1301) occurs.

Spindle position control (Spindle/C axis control)

(1) The axis in the spindle mode can be set as the vibration cutting selection axis.

(2) If the spindle selected by the part system in the vibration cutting mode is changed to the C axis mode, the operation error (M01 1300) occurs.

(3) When the spindle selected by the part system is in the C axis mode at the start of the vibration cutting mode, the operation error (M01 1301) occurs.

(4) When the C axis is selected for spindle position control (spindle/C axis) in the part system during the vibration cutting mode, the normal position loop gain (SV003, SV004, SV057) is used for the NC axes other than the C axis even if "#3129 cax_spec/bit4" is "1" (*1).

(*1) When the vibration cutting mode is invalid and "#3129/bit4" is "1", the gains of the NC axes other than the C axis switch to the position loop gain (SV049, SV050, SV058) for spindle synchronization at the timing that the C axis is selected. (MTB specifications)

Spindle orientation

(1) If spindle orientation is performed to the spindle selected by the part system during the vibration cutting mode, the operation error (M01 1300) occurs.

(2) When the spindle selected by the part system is under spindle orientation at the start of the vibration cutting mode, the operation error (M01 1301) occurs.

Spindle override

[Fixed spindle speed selection method]

(1) When the spindle override is applied to the spindle selected by the part system during the vibration cutting mode, the spindle rotation speed closest to the spindle rotation speed specified after override is selected from the vibration cutting conditions, and is used as the spindle rotation speed in the mode.

(2) If override of the selected spindle is changed while the program is executing the cutting block during vibration cutting mode, it is enabled at the timing at which the cutting block is switched to the non-cutting block. However, the spindle override zero is enabled immediately.

[Variable spindle speed method]

(1) If override of the selected spindle is changed during vibration cutting mode, it is enabled immediately even while the cutting block is being executed.

Spindle clamp speed setting

[Fixed spindle speed selection method]

(1) When the spindle clamp speed is set to the spindle selected by the part system during vibration cutting mode, the optimal spindle rotation speed is selected from the range of the clamp speed. If there is no selectable spindle rotation speed due to the clamp speed setting, the operation error (M01 1304) occurs.

[Variable spindle speed method]

(1) The spindle rotation speed is clamped at a clamp speed lower than that of the spindle clamp speed by the frequency limit value.

Spindle forward run start/spindle reverse run start

(1) When the rotation direction of the selected spindle is changed while the program is executing the cutting block during the vibration cutting mode, it is enabled immediately. Do not change the spindle rotation direction during cutting.

Macro call, subprogram call

(1) The macro call (G65/G66/G66.1) and subprogram call (M98/M198) can be commanded during the vibration cutting mode. The status of the vibration cutting mode is also inherited to a macro or subprogram.

(2) The vibration cutting control (G08.5) can be commanded in a macro or subprogram. Even though the macro or subprogram is completed, the status of the vibration cutting mode is maintained.

Sub part system control I

(1) The sub part system control I (G122) can be commanded during the vibration cutting mode. However, the status of the vibration cutting mode is not inherited.

(2) The vibration cutting control (G08.5) can be commanded in a sub part system. If the vibration cutting mode is started in a sub part system, the mode is canceled when the sub part system is completed.

Arbitrary axis exchange, arbitrary axis exchange return, arbitrary axis exchange/reference axis arrange return

(1) When the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141) or the arbitrary axis exchange/reference axis arrange return command (G142) is issued in the vibration cutting mode, the program error (P790) occurs.

(2) The operation error (M01 1101) occurs if a command is issued to pull out the axis existing in the part system in the vibration cutting mode, or if a command is issued to return the axis to the part system in the vibration cutting mode.

(3) It is possible to issue the vibration cutting mode start command after the axis exchange or axis return.

Synchronous control

(1) When a cutting command is issued during the vibration cutting mode, operations are performed as follows.

- When a cutting command is issued to the reference axis for synchronization control, the reference axis vibrates, but the synchronized axis does not vibrate.

Multiple-axis synchronization control

(1) When a cutting command is issued during the vibration cutting mode, operations are performed as follows.

- When a cutting command is issued to the reference axis for multiple-axis synchronization control, the reference axis vibrates, but the synchronized axis does not vibrate.

Cutting feed override

(1) The cutting feed override can be applied during vibration cutting mode.
If the override is changed while the cutting block is being executed, the amplitude also changes by the change in the feedrate.
When the vibration feedrate exceeds the clamp value of the parameter "#12586 ActualClamp_VCC" due to the cutting feed override, the feedrate is clamped.

(2) During vibration cutting mode, the vibration feedrate clamp value (F command value) when the parameter "#8161 Vibe cut F value" is set to "1" is changed by the cutting feed override.

(3) For the feed per revolution (synchronous feed) that is activated under variable spindle speed method, both the command speed and the vibration feedrate increases when the spindle rotation speed is increased in the vibration cutting block. Then, the spindle speed is clamped to prevent the command speed and the vibration feedrate from exceeding parameter #12571 and #12586. During this time, the cutting override rate cannot be increased.

Deceleration check

(1) If any of the following conditions is enabled when the program executes a cutting block during vibration cutting mode, a deceleration check is performed at each joint between blocks before the next block is executed.

- When the "error detection" signal is ON
- When the exact stop check (G09) command is included in the same block
- When the exact stop check mode (G61) is selected
- When the next block is set to the non-cutting mode

When the block being executed is set to the vibration cutting valid block, confirm that the vibration of all axes in the system have converged before executing the next block.

(2) The deceleration check is performed during the vibration cutting mode even if the rapid traverse block overlap (G0.5) is in process.

(3) The programmable in-position check with the address ",I" is not available in vibration cutting mode.

Start point designation timing synchronization Type 1/2

(1) When the start point designation timing synchronization (G115/G116) is commanded during the vibration cutting mode, the function is enabled at the timing at which the vibration forward point reaches the specified position.

Stored stroke limit, Interference check III

(1) When the program executes the cutting block during the vibration cutting mode, the function is enabled at the timing when the vibration forward point is about to exceed the specified area.

Automatic operation handle interruption

(1) When the cutting block is executed during vibration cutting mode, an automatic operation handle interruption for the feed axis cannot be executed. The operation error (M01 1300) occurs.

NC axis/PLC axis switchover

(1) NC axes/PLC axes in PLC axis mode are the axes for which vibration cutting control is disabled when vibration cutting mode starts.

(2) NC axes for which vibration cutting control is enabled cannot be switched to PLC axes during vibration cutting mode. The operation error (M01 1250) occurs.

OMR-FF

(1) When the cutting block is being executed during vibration cutting mode, OMR-FF is enabled for all axes (excluding rotary axes) in the part system.
When the parameter "#2139 omrff_off" is set to "1" (OMR-FF temporarily disabled), OMR-FF is still enabled while the cutting block is being executed.

(2) When OMR-FF is disabled (#2313 SV113 bit0 = 0), vibration cutting is performed for the cutting block in the vibration cutting mode while OMR-FF is disabled. The operation error (M01 1302) occurs.

NC reset

(1) When the "NC reset 1" signal or the "NC reset 2" signal is input in the part system during the vibration cutting mode, the vibration cutting mode is canceled.

(2) When the "NC reset 1" signal or the "NC reset 2" signal is input while the program is executing the cutting block during the vibration cutting mode, reset processing is performed after vibration has converged.

Emergency stop

(1) When an emergency stop is commanded during the vibration cutting mode, the vibration cutting mode is canceled.

(2) If an emergency stop is commanded while the program is executing the cutting block during the vibration cutting mode, the feed axis stops immediately, and the program shifts to the emergency stop mode.

Manual arbitrary reverse run (Program check operation)

(1) If the vibration cutting control (G08.5) is commanded during a manual arbitrary reverse run, the spindle rotation speed is switched to the speed closest to the S command in the vibration cutting conditions, and the frequency, vibration frequency, and spindle rotation speed in the vibration cutting mode is notified to the ladder.

Inclined axis control

(1) When a cutting command including movement of inclined axis under inclined axis control is issued in a cutting block during the vibration cutting mode, the operation error (M01 1302) occurs and the cutting machining is performed without vibration.

(2) When the inclined axis under inclined axis control is specified as a vibration target axis at the start of the vibration cutting mode, the program error (P791) occurs.

Multiple-spindle control II, Spindle control by part system

(1) When multiple-spindle control II function is enabled, the selected spindle cannot be changed during vibration cutting mode. The operation error (M01 1300) occurs. Start vibration cutting mode after changing the selected spindle.

(2) When spindle control by part system is enabled, the S command that was commanded last in the selected part system is the enabled S command during vibration cutting mode. The S command is enabled at the switching from a cutting block to a non-cutting block. However, the "S0" command is enabled immediately.

Corner chamfering/Corner R

(1) It is possible to issue the corner chamfering or corner R command in the cutting block during the vibration cutting mode. In the inserted block for corner chamfering/corner R, machining is performed while the target axis is vibrating.

(2) The inserted block for corner chamfering/corner R inherits the vibration cutting control temporary invalid (,V0) command of the cutting block.

(3) If the vibration cutting mode is started or canceled in the next block for corner chamfering/corner R, the program error (P382) occurs.

Program restart

(1) If the program is restarted in the block in the vibration cutting mode, the program error (P49) occurs.

Spindle speed fluctuation detection

(1) If the actual spindle speed is different from the programmed speed due to an external factor such as a load fluctuation, the operation error (M01 1105) occurs; however, the automatic operation or the spindle does not stop.

Chopping

(1) Chopping cannot be executed during vibration cutting mode. The operation error (M01 1300) occurs.

(2) If the part system is in chopping when vibration cutting mode starts, the operation error (M01 1301) occurs.

Manual speed command

(1) The vibration target axis does not vibrate when manual speed command is enabled, even for a cutting block in vibration cutting mode.

(2) If manual speed command is enabled while the vibration target axis is vibrating, the vibration will stop, and handle mode or jog mode will be enabled after vibration convergence is completed.

NC Analyzer2

(1) When the parameter "#12583 VibPGMsp_VCC" is other than "0" and the vibration cutting mode command (G08.5) is issued, measurement using NC Analyzer2 cannot be performed. Use the data sampling function.

High-speed high-accuracy function (High-accuracy control, High-speed machining mode I/II, High-speed high-accuracy control I/II/III)

(1) The vibration cutting control can be commanded while the high-speed high-accuracy function is enabled. Also, the high-speed high-accuracy function can be commanded during the vibration cutting mode.

(2) During the vibration cutting mode, even during high-speed machining mode I/II or high-speed high-accuracy control I/II/III, the micro segment processing capability does not increase, and the given feedrate cannot be secured.

Coordinate rotation by parameter

(1) The coordinate rotation by parameter cannot be combined with the vibration cutting.

(2) If the vibration cutting is commanded in the status below, the program error (P791) occurs.

- The parameter "#8116 CoordRotPara invd" is "0" and the "#8627 Coord rot angle" is not "0".
- The parameter "#8116 CoordRotPara invd" is "0" and the parameters "#8625 Coord rot vctr(H)" and "#8626 Coord rot vctr(V)" are not "0".

Cutting load control

(1) If the cutting load control is combined with the vibration cutting control, the program error (P791 or P790) occurs.

Spindle-mode rotary axis control

(1) The spindle in the spindle-mode rotary axis control mode can be set as the vibration cutting selection axis.

(2) Do not turn the "Spindle-mode rotary axis control command" signal (YB40) to OFF (switch to the servo axis mode) for the selected spindle in the vibration cutting mode. The operation error (M01 1024) occurs and the spindle mode is maintained without switching to the servo axis mode. Also, the machining program continues without stopping.

(3) Follow the steps below to reset the operation error (M01 1024).

- Turn the "Spindle-mode rotary axis control command" signal (YB40) to ON (switch to the spindle mode).
- Reset

(4) If you turn OFF the vibration cutting mode (G08.5 P0) while the operation error (M01 1024) is occurring, the mode switches from the spindle mode to the servo axis mode and the operation error (M01 1024) is reset.

OMR-CC (Optimum machine response-contour control)

(1) OMR-CC cannot be used with vibration cutting control. Do not use them together.

Special fixed cycle

(1) Combine the special fixed cycle with the fixed cycle to use it.

(2) The special fixed cycle is a function that retains the drilling data of the fixed cycle such as G81. The axis does not vibrate when it is moving to the drilling position. The axis vibrates in the cutting block when drilling is performed.

Manual interruption

(1) When the program executes the cutting block during the vibration cutting mode, if the manual interruption is performed, the axis does not vibrate.

Tool handle feed & interruption

(1) When the program executes the cutting block during the vibration cutting mode, if an interruption amount distributed to each axis by the tool handle feed & interruption during automatic operation is generated for the commanded axis, the operation error (M01 1300) occurs and an interruption cannot be performed.

(2) When the program executes the cutting block during the vibration cutting mode, if the mode is switched from the automatic mode to the handle mode and the tool handle feed & interruption is performed, the axis does not vibrate.

Dry run, External deceleration

(1) The command speed of dry run and external deceleration are clamped with the parameter "#12571 Vib-Clamp_VCC".
The vibration feedrate is also clamped with the parameter "#12586 ActualClamp_VCC".
The operation error (M01 1303) occurs when the command speed is clamped, and the operation error (M01 1309) occurs when the vibration feedrate is clamped.

(2) For the feed per revolution (synchronous feed) that is activated under variable spindle speed method, both the command speed and the vibration feedrate increases when the spindle rotation speed is increased in the vibration cutting block. Then, the spindle speed is clamped to prevent the command speed and the vibration feedrate from exceeding parameter #12571 and #12586.

(3) When the parameter "#8161 Vibe cut F value" is "1", the vibration feedrate is clamped with the following values and the operation error (M01 1309) occurs.

During dry run: The vibration feedrate is clamped at the dry run speed.

During external deceleration: The vibration feedrate is clamped at the F command value.

Analog spindle I/F

[Fixed spindle speed selection method]

(1) If the analog spindle is the selected spindle for vibration cutting control, the spindle rotation speed during vibration cutting mode will switch to the spindle rotation speed calculated from the vibration cutting control conditions.

[Variable spindle speed method]

(1) If the spindle selected by the part system in vibration cutting mode is switched to the analog spindle, the operation error (M01 1300) occurs.

(2) If the selected spindle is the analog spindle at the start of vibration cutting mode, the operation error (M01 1301) occurs.



Precautions

(1) Make sure to issue the G08.5 as a single command. If another G code command is included in the same block, the program error (P34) occurs.

(2) During the vibration cutting mode, do not change the parameters related to vibration cutting using the function such as the parameter input by program command (G10). If any parameter is changed during the mode, the vibration cutting control cannot be correctly performed. Change parameters before issuing the vibration cutting mode start command.

(3) If the axis that moves between cutting blocks is exchanged, or if the next cutting block does not vibrate, the next block starts moving without waiting for the vibration to converge. Therefore, uncut parts may be left at the joint of the blocks. To start the next cutting block after the vibration has converged, command the exact stop check or dwell.

(4) Compared to the regular cutting, the vibration cutting control waits for the completion of vibration convergence when a cutting block is switched to a non-cutting block, which extends the machining time.

(5) When pocket machining is performed using the vibration cutting control, extra cuts may occur depending on the axis to be vibrated. Set the vibration target axis in consideration of the vibration direction, or temporarily disable the vibration cutting control.

(6) For vibration cutting, make sure that the total number of axes (axes with the parameter #12570 set to "1") for which vibration cutting of all the commanded part systems is valid does not exceed 7 axes. If the total number of valid axes exceeds 7 when "G08.5" is commanded, the operation error (M01 1301) occurs.

(7) The cutting feed acceleration/deceleration type in the vibration cutting mode is set to the step regardless of the presence or absence of vibration.

(8) If vibration cutting is commanded to the axis under synchronous control, only the reference axis vibrates. When the axes are physically connected like a gate, do not use such a command as it may cause a large shock to the machine.

(9) Do not use electromagnetic brake of a brake-installed motor for normal braking because it is for holding. The holding may not be possible due to the part life or the machine structure (*1). Install a stop device to ensure safety on the machine side.

(*1) An example case is that a ball screw and a servo motor are connected through a timing belt.

15.14 Two-dimensional Barcode Engraving Cycle; G136

Function and purpose

This function is used to machine a QR code that stores an arbitrary character string on the machined surface. When the character string to be stored in the QR code is specified, the QR code that stores the entered character string can be machined on the cylindrical surface or plane. In addition to the character string, the size of the QR code can be specified by commanding a specific address.

The QR code consists of a combination of white and black squares called cells.
In this function, the QR code is expressed as a collection of dots by performing a drilling operation in the black cell part. Therefore, the size of the cell is determined by the tool diameter and depth.

Term

The following describes the meanings of the terms used in this chapter.

Word	Meaning
QR code length	QR code side length
Cell interval	Distance between adjacent holes when machining a cell by drilling
Encoding mode	Encoding method used when characters are expressed as a QR code
Error correction level	Criteria that show to what extent the QR code can be read with respect to loss

Command format

G136 X_ Y_ Z_ α_ R_ D_ F_ L_ I_ P_ K_ E_ [ ] ;

X, Y, Z, α	Command position from the height of the machining surface
R	Retract height from the height of the machining surface
D	Depth
F	Feedrate in cutting feed
L	QR code length
I	Cell interval
P	QR code position (0: Top right, 1: Bottom right, 2: Top left, 3: Bottom left, 4: Center)
K	QR code angle
E	Error correction level (0: 7%, 1: 15%, 2: 25%, 3: 30%)
[ ]	The character string to be stored in the QR code (maximum 100 characters)

The G136 command is a non-modal command of the G code group. Command the G code for each block.

Explanation of address

Address	Meaning	Command range	Remarks
X, Y, Z, α (α is the additional axis.)	Commanded position	-	If omitted, the current position is set as the commanded position.
R	Retract height from the height of the machining surface	Numeric value of "0" or more	If omitted, the program error (P33) occurs.
D	Depth from the height of the machining surface	Numeric value of "0" or more	If omitted, the program error (P33) occurs.
F	Feedrate in cutting feed	Numeric value of "0" or more	If omitted, this function operates at the previously commanded cutting feedrate.
L	QR code length	Numeric value of "0" or more	When address "L" is commanded, the cell interval "I" is determined automatically. When address "I" is commanded while address "L" is omitted, the QR code length "L" is determined automatically. If either address "L" or address "I" is omitted, the program error (P33) occurs. (*1)
I	Cell interval	Numeric value of "0" or more	
P	QR code position	0 to 4 (Decimal part ignored)	0: Top right, 1: Bottom right, 2: Top left, 3: Bottom left, 4: Center If the value is omitted or is other than 1 to 4, this function operates assuming that "0" is specified. (*1)
K	QR code angle	±180.0000 (degrees)	If omitted, this function operates assuming that 0 degree is specified. (*1)
E	Error correction level	0 to 3 (Decimal part ignored)	0: 7%, 1: 15%, 2: 25%, 3: 30% If the value is omitted or is other than 0 to 2, this function operates assuming that "3" is specified. (*1)
[ ]	Character string to be stored in QR code	ASCII characters	Up to 100 characters can be stored. (*1)

(*1) Refer to the description below for details on the address.

Note

(1) If a command is given outside the command range, the operation described in the remarks is performed or the program error (P35) occurs.

(2) If a linefeed is inserted without closing square brackets "[", the program error (P225) occurs.

(3) In square brackets [ ], an operator is recognized as a character; thus, no operation can be performed.

(4) If an invalid character is entered in square brackets [ ], the program error (P225) occurs. At this time, the sub-error number indicates a character in which alarm is occurring for the entered character. However, if the entered character string exceeds the number of characters that can be stored in the QR code, "0" is selected as the sub-error number.

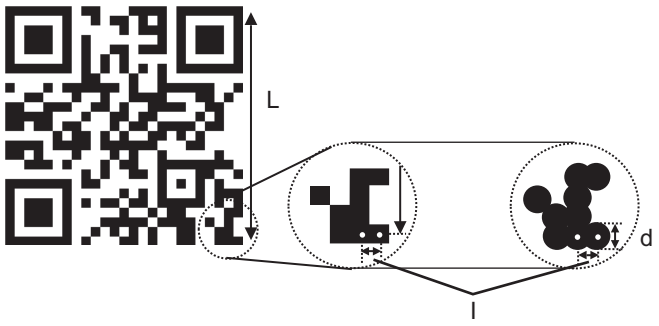
(5) Command G136 alone in a block. If it is commanded in the same block with other G codes, the program error (P33) occurs.

QR code length L and cell interval I

The QR code can be machined by specifying the QR code length and cell interval by commanding address L or address I. Here, the QR code length is defined as the distance from the center point of the cell at the four corners to the center point of the cell at the four corners located in the vertical or horizontal direction of the cells, and the cell interval is defined as the distance from the center point of the cell to the center point of the adjacent cell.

[Relationship between QR code length L, cell interval I, and tool diameter (hole diameter)]

In actual machining, the QR code is expressed by a collection of dots created by the drilling operation.



L: QR code length
I: Cell interval
d: Tool diameter (Hole diameter)

Partially enlarged view

Actual machining image

To machine a QR code that is easy to recognize, the tool diameter (hole diameter) must be approximately 1.0 to 1.1 times the cell interval, and an unmachined surface of 4 cells (cell interval × 4) or more must be secured around the QR code. The adjustment method is as follows.

<When commanding the QR code length (address L)>

(1) Obtain the number of cells from the encoding mode and the number of stored characters by referring to "(2) Number of QR code cells" in "QR code size" described later.

(2) Calculate the cell interval from the QR code length L specified by address L and the number of cells obtained in step (1) using the following formula.

(Formula) Cell interval = QR code length L / (Number of cells - 1)

(3) Select a tool and specify the depth so that the tool diameter (hole diameter) is approximately 1.0 to 1.1 times the cell interval calculated in step (2).

(4) Adjust the command position so that the unmachined surface for 4 cells is secured around the QR code based on the QR code length L specified by the address L.

<When commanding the cell interval (address I)>

(1) Obtain the number of cells from the encoding mode and the number of stored characters by referring to "(2) Number of QR code cells" in "QR code size" described later.

(2) Select the tool and specify the depth so that the tool radius (hole radius) is approximately 1.0 to 1.1 times the cell interval I commanded by address I.

(3) Calculate the QR code length from the cell interval I specified by address I and the number of cells obtained in step (1) using the following formula.

(Formula) QR code length = Cell interval I × (Number of cells - 1)

(4) Based on the QR code length calculated in step (3), adjust the command position so that the unmachined surface for 4 cells is secured around the QR code.

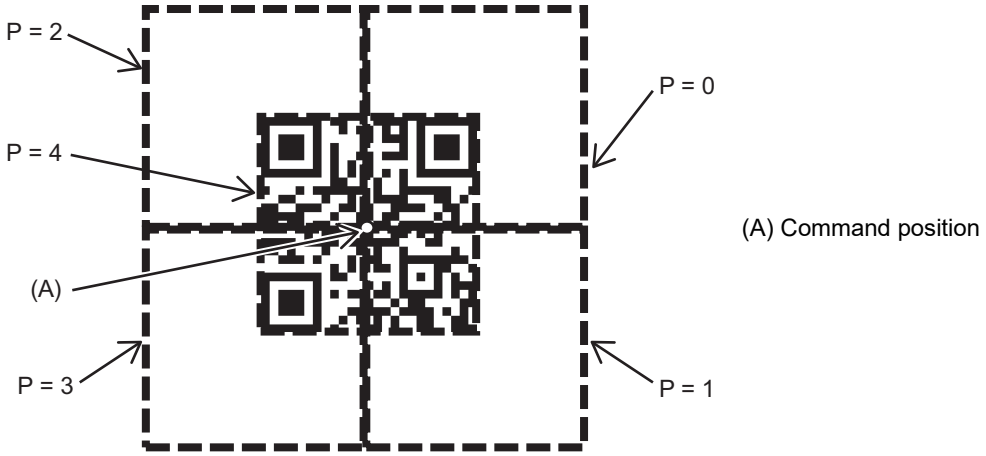
Note

(1) If QR code length L and cell interval I are specified at the same time, cell interval I is ignored.

QR code position P

By specifying a numeric value from "0" to "4" in address P, the QR code position for the command position can be changed to the position shown below.

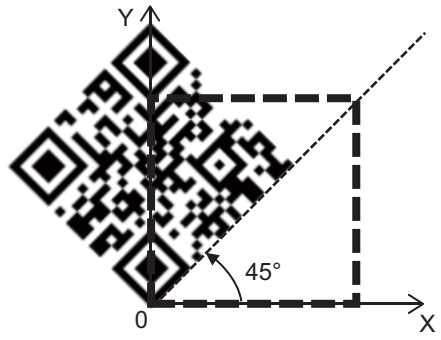
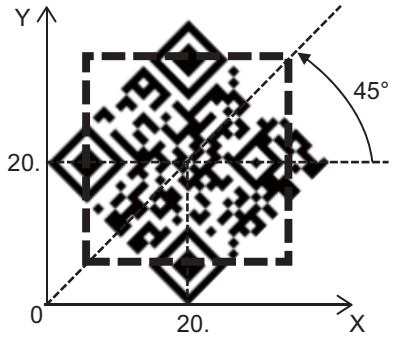
[Relationship between address P and QR code position]



QR code angle K

By commanding address K, the QR code can be rotated by angle K around the commanded position. The point around which the QR code rotates depends on address P.

QR code when the QR code angle (K) is set to 45 degrees

P = 0	P = 4
G136 X0 Y0 P0 K45. ;	G136 X20. Y20. P4 K45. ;
	

Error correction level E

The QR code has a built-in mechanism to recover data without loss even if it is partially damaged. Regarding this mechanism, the standard that indicates how much data can be lost to read the data is referred to as the error correction level.

The error correction level of the QR code to be machined can be specified by commanding the address E. The error correction level can be set in four steps. The table below shows the correspondence between each error correction level and the recoverable loss ratio.

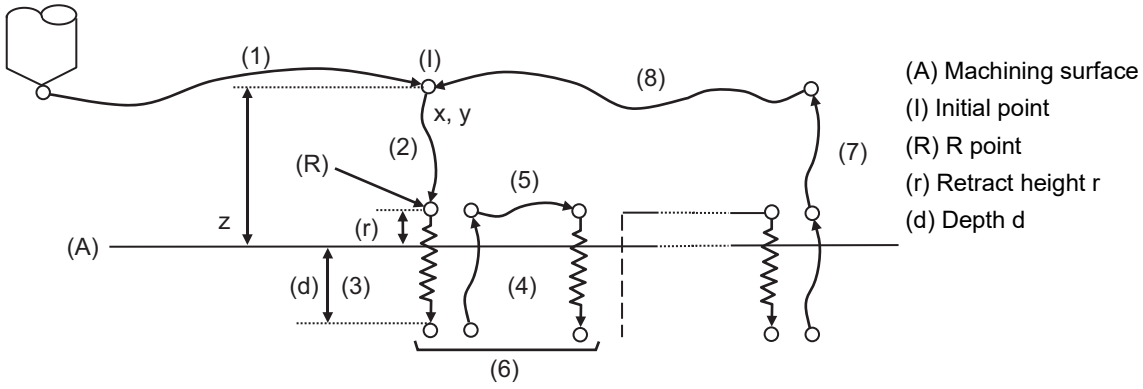
Error correction level	0	1	2	3
Recoverable loss rate	7%	15%	25%	30%

Detailed description

Description of operation

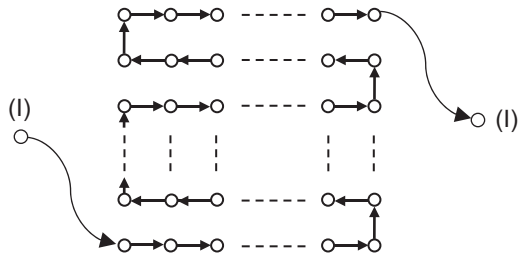
The movement command in the G136 command is as follows.

Operation of two-dimensional barcode engraving cycle



Operation pattern	Program	
(1)	G00 Xx Yy;	Axes other than the Z axis are positioned by rapid traverse. The position after the positioning has been completed by this operation is referred to as the initial point.
(2)	G00 Z(z + r);	The tool is positioned to the R point by rapid traverse.
(3)	G01 Z-(r + d);	Hole machining is conducted by cutting feed.
(4)	G00 Z(r + d);	The tool moves for the escape to the R point by rapid traverse.
(5)	-	The tool moves to the R point of the adjacent cells by rapid traverse.
(6)	-	Repeat steps (3) to (5) until QR code machining is completed.
(7)	G00 Z-(z + r);	The tool retracts to the height of the initial point by rapid traverse.
(8)	G00 Xx Yy;	The tool retracts to the initial point by rapid traverse.

The movement path from steps (2) to (8) is as shown below regardless of the P command.



(I) Initial point

Note

(1) In single block mode, machining stops at the end point of each operation of (1), (2), (4), (5), (7), and (8).

(2) Even if the orientation of the QR code is changed by changing the QR code angle or enabling the mirror image, machining is performed on a movement path where such that the cell at the bottom left of the QR code is the start point and the cell at the top right of the QR code is the end point.

(3) When a two-dimensional barcode engraving cycle is commanded while the rapid traverse block overlap is enabled, the in-position width for rapid traverse block overlap is applied in the rapid-traverse movement operations ((1), (2), (4), (5), (7), and (8)).
When using the rapid traverse overlap together, make the retract height sufficiently higher than the in-position width of the rapid traverse overlap.

Coordinate system setting

In this function, the plane to machine the QR code is determined by the plane selection commands G17, G18, and G19. Regarding the tool axis direction, position the tool axis in advance so that it is the axis perpendicular to the above plane (X, Y, Z or its parallel axis).

Plane selection	Plane to machine QR code
G17 (X-Y)	Xp-Yp
G18 (Z-X)	Zp-Xp
G19 (Y-Z)	Yp-Zp

Xp, Yp, and Zp indicate the basic axes X, Y, Z or the parallel axes of the basic axes, respectively.

Corresponding character

(1) ASCII characters

This function supports some of the characters included in ASCII. Specifically, the characters that can be stored in the QR code are the characters listed in the table below. Some characters cannot be entered using the Mitsubishi Electric standard keyboard, but can be stored in the QR code by entering them using an external device. A specific address needs to be commanded for some characters such as linefeeds and tabs.

(a) Characters that can be entered using the Mitsubishi Electric standard keyboard

!	#	$	(	)	*	+	,	-	.
/	0	1	2	3	4	5	6	7	8
9	:	=	@	A	B	C	D	E	F
G	H	I	J	K	L	M	N	O	P
Q	R	S	T	U	V	W	X	Y	Z
\	_	a	b	c	d	e	f	g	h
i	j	k	l	m	n	o	p	q	r
s	t	u	v	w	x	y	z	~	

(b) Characters that need to be entered using an external device, etc.

"	&	‘	?	^	`	{	|	}

(c) Characters that can be entered by commanding a specific address

SP	<	>	[	]	;	%
HT (Horizontal tab)		LF (Linefeed)		CRLF (Linefeed)		

Note

(1) Blank characters, "<", ">", "[", "]", and ";" can be entered using the Mitsubishi Electric standard keyboard. However, to store them in the QR code as characters, command a specific address and numeric value by referring to "(2) Characters by commanding a specific address" described later. If the above symbol is entered in square brackets [ ], the program error (P225 or P281) occurs. If a space character is entered in square brackets [ ], it is ignored.

(2) If a character that is not supported by this function is specified in square brackets [ ], the program error (P225) occurs.

(3) Up to 100 characters can be stored in the QR code. If the number of characters exceeds 100, the program error (P225) occurs. Note that when a variable command or a specific address is used to specify the characters to be stored, even if the number of characters in square brackets [ ] is 100 or less, if the number of characters actually stored exceeds 100, the program error (P225) occurs.

(2) Characters by commanding a specific address

Characters such as date, time and linefeed can be stored in the QR code by commanding a specific address in angle brackets <>. If it is necessary to enclose each address in angle brackets <> and if the address is any of Y, T, L, R, and S, command with a combination of "address" and "numeric value" as necessary.

Address	Stored contents	Description
<Y>	Year	Used to store the date and time in the QR code. If the number representing month and day is one digit, "0" is added at the beginning of the number.
<Y1>	Last two digits of the year	
<M>	Month	
<D>	Day	
<T>	Time in 24-hour notation	
<T1>	Time in 12-hour notation	
<N>	Linefeed code (LF)	Used to store control characters such as linefeeds and tabs in the QR code.
<C>	Linefeed code (CRLF) (*1)	
<H>	Tab code (HT)	
<L>	<	Used to store brackets in the QR code.
<R>	>	
<L1 >	[	
<R1 >	]	
<S>	Space character (space)	Used to store special ASCII characters as characters in the QR code.
<S1>	;	
<S2>	%	

(*1) The number of characters in the linefeed code (CRLF) is regarded as two characters.

<Example> If the current time is "April 1, 2019 13:00", the following numeric values are stored when a specific address is commanded.

Address	<Y>	<Y1>	<M>	<D>	<T>	<T1>
Stored contents	2019	19	04	01	13:00	01:00

Note

(1) If multiple addresses are specified in angle brackets <>, the program error (P225) occurs.

(2) If a numeric value outside the commanded range is specified in an address that needs to be commanded in combination with a numeric value, the program error (P225) occurs.

(3) If a combination of an address and a numeric value is specified in an address that cannot be specified in combination with a numeric value, the program error (P225) occurs.

(4) The date and time to be output to the QR code when the addresses Y, M, D, and T are commanded are the values specified at the start of G136 command execution.

(5) When a simulation is done using a background check and a 3D machine interference check, the correct date and time cannot be stored in the QR code. When a specific address is specified for the date or time, the numbers in the table below are stored instead of the actual date and time.

Address	<Y>	<Y1>	<M>	<D>	<T>	<T1>
Stored contents	2000	00	01	01	00:00	00:00

(3) Variable commands

By commanding a common variable, local variable, or system variable in angle brackets <>, the numeric value set in each variable can be stored in the QR code as a character string. For the available variables, refer to "14.2 Variable Commands".

With this function, it is possible to command significant figures when commanding variables. To specify significant figures, enter a slash "/", separate them with a period(s) ".", and command significant figures in the order of integer part and decimal part. If a slash "/" is not entered, the variable is stored as a numeric value that consists of a 5-digit integer part and a 3-digit decimal part.

The maximum number of significant digits is 8 digits for the integer part and 8 digits for the decimal part. At this time, the sign is not included in the number of significant digits. If the number of digits of the variable is smaller than the significant figure, add "0" to the upper digits in the integer part and the lower digits in the decimal part. If it is larger, the upper digits are rounded down in the integer part, and the lower digits are rounded down in the decimal part.

If a period "." or the number after a period "." is omitted, this function operates assuming that the decimal part does not contain any digits. If the number before a period "." is omitted or "0" is entered before a period ".", the value of the integer part is omitted. For the detailed conversion, refer to the table below.

Variable value	Contents of command	Numeric character to be stored in QR code
#100 = 12.345	<#100/1.2>	2.34
	<#100/4.5>	0012.34500
	<#100/2.0>, <#100/2.>, <#100/2>	12
	<#100/0.3>, <#100/.3>	.345
	<#100>	00012.345
#100 = -12.345	<#100/1.2>	-2.34
#100 = 12.345	<#100/>, <#100/.>, <#100/0.0>, <#100/.0>, <#100/0>	Program error (P225)
	<#100/9.0>, <#100/0.9>	Program error (P225)

Note

(1) Enclose each variable in angle brackets <>. If multiple variables are specified in angle brackets <>, the program error (P225) occurs.

(2) When both the decimal part and the integer part of a significant figure are "0", the program error (P225) occurs. Also, if nine or more digits are commanded, the program error (P225) occurs.

(3) Variable numbers cannot be specified by variables as "G136 [<## 100>]". If it is commanded, the program error (P225) occurs.

(4) This function does not support the numerical exponential notation.

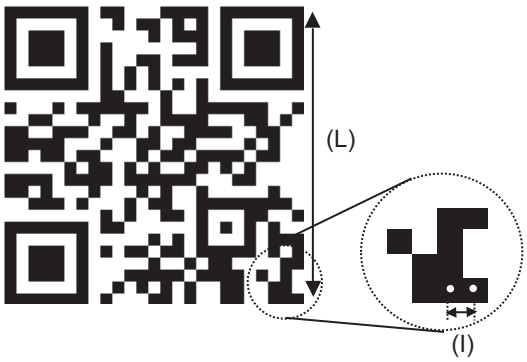
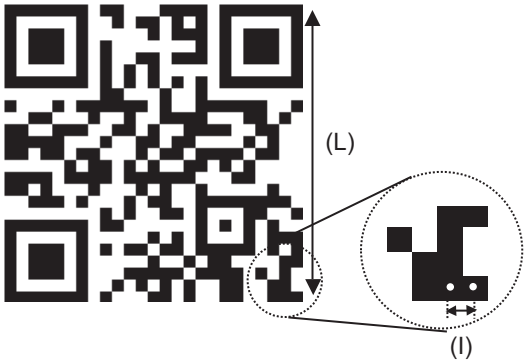
(5) The program error (P241) occurs if a write-only system variable is commanded or an unavailable number is commanded.

(6) When the variable is negative, the number of characters increases by one compared to the case where the variable is positive. Therefore, the number of cells in the QR code may differ depending on the sign of the variable even in the same machining program.

QR code size

The size of the QR code is determined by the QR code length L or cell interval I and the number of cells that make up the QR code. The number of cells is determined by the number of characters to be stored in the QR code, the error correction level, and the encoding mode described later.

When the QR code length L is specified, the cell interval I is automatically determined based on the number of cells required to store the entered character string. When the cell interval I is specified, the QR code length L is calculated by multiplying the cell interval I by the number of cells.

When QR code length L is specified	When cell interval I is specified
	
Cell interval I = QR code length L / (Number of cells - 1)	QR code length L = Cell interval I × (Number of cells - 1)

(1) Encoding mode

When characters are encoded into a QR code, the encoding method (encoding mode) for the QR code differs depending on the type of characters included in the entered character string. The encoding mode includes the numeric mode, alphanumeric mode, and 8-bit mode, and the types of characters that can be encoded in each encoding mode are shown in the table below.

This function automatically determines the encoding mode. The encoding mode is selected from the encoding modes that can encode the entered character string, in the order of the numeric mode, alphanumeric mode, and 8-bit mode.

Encoding mode	Type of encodable character
Numeric mode	Number only (0 to 9)
Alphanumeric mode	Numbers (0 to 9), uppercase alphabetic characters (A to Z), 9 types of symbols (space, $, %, *, +, -, ., /, :)
8-bit mode	All ASCII characters supported by this function

Note

(1) When a character or variable is specified by a specific address command, the encoding mode is selected depending on the type of the converted character.

<Example> If G136 [<Y>] is commanded, the entered character string is converted in the numeric mode.

(2) Number of QR code cells

The table below shows the maximum number of characters that can be stored in a specific number of cells, assuming that the total number of cells per side of the QR code is the number of cells.

Number of cells and maximum number of storable characters

Number of cells	Error correction level 0		
	Numeric mode	Alphanumeric mode	8-bit mode
21	41	25	17
25	77	47	32
29	100	77	53
33	-	100	78
37	-	-	100
41	-	-	-

Number of cells	Error correction level 1		
	Numeric mode	Alphanumeric mode	8-bit mode
21	34	20	14
25	63	38	26
29	100	61	42
33	-	90	62
37	-	100	84
41	-	-	100

Number of cells	Error correction level 2		
	Numeric mode	Alphanumeric mode	8-bit mode
21	27	16	11
25	48	29	20
29	77	47	32
33	100	67	46
37	-	87	60
41	-	100	74
45	-	-	86
49	-	-	100
53	-	-	-
57	-	-	-

Number of cells	Error correction level 3		
	Numeric mode	Alphanumeric mode	8-bit mode
21	17	10	7
25	34	20	14
29	58	35	24
33	82	50	34
37	100	64	44
41	-	84	58
45	-	93	64
49	-	100	84
53	-	-	98
57	-	-	100

<Example> When the character string "MitsubishiElectric" (18 characters) is stored in the QR code at error correction level 2, the number of cells can be derived by the following procedure:

(1) Refer to the table of error correction level "2".

(2) As the encoding mode is "8-bit mode", refer to the 8-bit mode columns in the table.
(The encoding mode is set to 8-bit mode because the QR code contains lowercase alphabetic characters that cannot be encoded in the numeric mode and alphanumeric mode.)

(3) As the number of characters is "18", refer to the cell that contains the smallest value among the numbers larger than "18". In this case, the cell with the number "20" is applicable.

(4) The number of cells is derived by referring to the value of the number of cells in the same row as the cell referred to in step (3). In this case, as "25" is applicable, the number of cells to be derived is "25".



Program example

Machine the QR code in the following conditions.

- Tool diameter: 1 mm
- Characters to be machined: MELCO
- Cell interval: 1 mm
- Error correction level: 3
- QR code size: 20 mm × 20 mm (Derived from the cell interval and error correction level.)

:	
G136 X25. Y25. Z0. I1.0 R5.0 D0.5 E3 [MELCO] F200 ;	Two-dimensional barcode engraving cycle
:	

Relationship with other functions

Relationship between two-dimensional barcode engraving cycle and G code functions

Column A: Operation when the two-dimensional barcode engraving cycle (G136) is commanded while the combined function is enabled

○: The two-dimensional barcode engraving cycle command can be issued, and the combined function is also enabled

Δ: The two-dimensional barcode engraving cycle command can be issued, while the combined function is disabled

x: The two-dimensional barcode engraving cycle command cannot be issued, and an alarm is to be generated. (The text in brackets refers to the number of the program error to be generated.)

□: Others

Group	G code	Function name	A
0	G05P0	High-speed machining mode II OFF High-speed high-accuracy control II OFF High-speed high-accuracy control III OFF	○
	G05P2	High-speed machining mode II ON	Δ
	G05P10000	High-speed high-accuracy control II ON	□ (*1)
	G05P20000	High-speed high-accuracy control III ON	□ (*1)
	G05.1Q0	High-speed high-accuracy control I OFF Spline interpolation OFF	○
	G05.1Q1	High-speed high-accuracy control I ON	□ (*1)
	G05.1Q2	Spline interpolation ON	Δ
	G07	Hypothetical axis interpolation	× (P29)
	G08P0	High-accuracy control OFF	○
	G08P1	High-accuracy control ON	○
	G60	Unidirectional positioning	○
	G65	User macro simple call	□ (*2)
	G122	Sub part system control I	○
	G144	Sub part system control II	○
1	G00	Positioning	Δ
	G01	Linear interpolation	Δ
	G02 G03	Circular interpolation	× (P29)
	G02.1 G03.1	Spiral interpolation	× (P29)
	G02.3 G03.3	Exponential interpolation	× (P29)
	G02.4 G03.4	3-dimensional circular interpolation	× (P29)
	G06.2	NURBS interpolation	× (P29)
	G33	Thread cutting	× (P29)
2	G17 - G19	Plane selection	○
3	G90	Absolute command	○
	G91	Incremental command	○
4	G22	Stroke check before travel ON	○
	G23	Stroke check before travel OFF	○

Group	G code	Function name	A
5	G93	Inverse time feed	× (P29)
	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	○
	G43.4 G43.5	Tool center point control	× (P942)
	G43.7	Tool position compensation	○
	G49	Tool length offset cancel	○
9	G80	Fixed cycle cancel	○
	Other than G80	Fixed cycle	× (P29)
10	G98	Fixed cycle (Initial level return)	○
	G99	Fixed cycle (R point level return)	○
11	G50	Scaling cancel	○
	G51	Scaling ON	× (P29)
12	G54 - G59 G54.1	Workpiece coordinate system selection	○
13	G61	Exact stop check mode	○
	G61.1	High-accuracy control	○
	G61.2	High-accuracy spline	□ (*3)
	G61.4	Spline interpolation 2	□ (*3)
	G62	Automatic corner override	○
	G63	Tapping mode	× (P29)
	G64	Cutting mode	○
14	G66 G66.1	User macro modal call	× (P29)
	G67	User macro modal call cancel	○
15	G40.1	Normal line control cancel	○
	G41.1 G42.1	Normal line control	× (P29)
16	G68	Coordinate rotation by program ON	○
	G68.2 G68.3	Inclined surface machining command	○
	G69	Coordinate rotation cancel	○
17	G96	Constant surface speed control ON	× (P29)
	G97	Constant surface speed control OFF	○
18	G15	Polar coordinate command OFF	○
	G16	Polar coordinate command ON	× (P29)
19	G50.1	Mirror image OFF	○
	G51.1	Mirror image ON	○
21	G07.1	Cylindrical interpolation	○
	G12.1	Polar coordinate interpolation ON	○
	G13.1	Polar coordinate interpolation OFF	○

Group	G code	Function name	A
27	G54.4P0	Workpiece installation error compensation cancel	○
	G54.4 P1 - P7	Workpiece installation error compensation	○

(*1) High-speed machining mode is disabled.

(*2) Can be commanded in a subprogram.

(*3) Operates as the G61.1 command modal is active.

Relationship between two-dimensional barcode engraving cycle and functions other than G codes

Column A: Operation when the two-dimensional barcode engraving cycle (G136) is commanded while the combined function is enabled

○: The two-dimensional barcode engraving cycle command can be issued, and the combined function is also enabled

Δ: The two-dimensional barcode engraving cycle command can be issued, while the combined function is disabled

x: The two-dimensional barcode engraving cycle command cannot be issued, and an alarm is to be generated. (The text in brackets refers to the number of the program error to be generated.)

□: Others

Function name	A
SSS ON	Δ
Mirror image by parameter setting ON	○
Mirror image by external input	○
Coordinate rotation by parameter	○
Subprogram call (M98)	□ (*1)
Figure rotation (M98 I_J_K_)	□ (*1)
Machine tool builder macro	□ (*2)
Manual interruption	× (M01 0181)
Automatic operation handle interruption	× (M01 0181)
PLC interruption	× (M01 0181)
MDI interruption	× (M01 0181)
Macro interruption (M96)	× (P29)
Arbitrary reverse run	Δ
Manual speed command	× (M01 0181)
Manual arbitrary reverse run	× (M01 0181)
Chopping	× (M01 0151)

(*1) Can be commanded in a subprogram.

(*2) Can be commanded in a program.



Precautions

(1) The two-dimensional barcode engraving cycle cannot be interrupted while it is being commanded. If the command is interrupted, the operation error (M01 0181) occurs.

(2) An operator enclosed in brackets [ ] is recognized as a character, so no operation can be performed.

(3) The QR code may not be recognized normally depending on the tool used for machining the QR code, the equipment used to read the QR code, and the material and shape of the workpiece.

(4) If the QR code is machined by this command during cylindrical interpolation, the QR code may not be recognized normally because the QR code is curved along the side surface of the cylinder.

(5) If the size of the hole to be machined is large, the cutting chip generated during hole drilling may deform the shape of the hole that makes up the QR code, and the QR code may not be correctly recognized. It is recommended that the machining conditions are set so that both the hole diameter and depth are within 1 mm.

(6) When the high-accuracy control mode is OFF, the tool may not reach the hole bottom position commanded in the hole drilling. Therefore, it is recommended to machine with the high-accuracy control mode ON.

(7) The two-dimensional barcode engraving cycle can be commanded during cylindrical interpolation. The command unit of rotary axis in G135 block during cylindrical interpolation is degrees (°). For details on the cylindrical interpolation, refer to "6.9 Cylindrical Interpolation; G07.1".

(8) When the parameter "#19431 Num display in prg" is set to "1", a character string specified in [ ] is displayed in the program buffer. It may be displayed differently from the actual character string due to the specifications of program buffer display. (Refer to "14.8 Variable Value Display in Program Being Executed" for variable value display in program being executed.)

Open source software

The QR Code generator library, which is the MIT license, is used for this function.

The copyright notice and permission notice of the QR Code generator library are described below.

QR Code generator library

Copyright (c) 2019 Project Nayuki

Released under the MIT license

Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER -26- IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

15.15 Text Engraving Cycle; G135



Function and purpose

This function enables to engrave arbitrary text on the machining surface. The specified text can be engraved on the plane or cylindrical surface. The size and angle of the text can be specified using specific addresses. The text thickness is determined by the tool tip shape and the engraving depth.



Command format

G135 X_ Y_ Z_ α_ R_ D_ F_ H_ L_ I_ J_ Q_ P_ K_ E_ [ ];

X, Y, Z, α	Machining start position (α is the additional axis.)
R	Retract height from the height of the machining surface
D	Engraving depth from the height of the machining surface
F	Feedrate in cutting feed
H	Text height
L	Text interval
I	Text arc center: Horizontal axis
J	Text arc center: Vertical axis
Q	Text arc direction (0: Clockwise; 1: Counterclockwise)
P	Text position (0: Center, 1: Bottom, 2: Top)
K	Text angle
E	Engraving speed adjustment coefficient
[ ]	Text to be machined

The text engraving cycle is a non-modal command or single command. Since it is the non-modal command, command the G code for each block. Also, since it is the single command, if another G code is commanded in the same block, the program error (P33) occurs.

Explanation of address

Address	Meaning	Command range	Remarks
X, Y, Z, α (α is the additional axis.)	Machining start position	-	If omitted, the current position is set as the machining start position. The coordinates follow the modal of G code group 3.
R	Retract height from the height of the machining surface	Numeric value of "0" or more (mm)	Cannot be omitted. If omitted, the program error (P33) occurs.
D	Engraving depth from the height of the machining surface	Numeric value of "0" or more (mm)	Cannot be omitted. If omitted, the program error (P33) occurs.
F	Feedrate in cutting feed	-	If omitted, this function operates at the previously commanded cutting feedrate.
H	Text height	0.001 to 99999.999 (mm)	Cannot be omitted. If omitted, the program error (P33) occurs. (*1)
L	Text interval	±99999.999 (mm)	If omitted, this function operates assuming that "0" is specified. (*1)
I	Text arc center: Horizontal axis	±99999.999 (mm)	If omitted, this function operates assuming that "0" is specified. If both "I" and "J" are "0", the text is arranged in a straight line. Command "I" as horizontal axis and "J" as vertical axis to any plane of G17/G18/G19. Command the incremental value from the machining start position. (*1)
J	Text arc center: Vertical axis	±99999.999 (mm)	
Q	Text arc direction	0, 1 (Decimal part ignored)	0: Clockwise; 1: Counterclockwise If omitted, this function operates assuming that "0" is specified. (*1)
P	Text position	0 to 2 (Decimal part ignored)	0: Center, 1: Bottom, 2: Top If omitted, this function operates assuming that "0" is specified. (*1)
K	Text angle	-180.000 to 180.000 (°)	If omitted, this function operates assuming that "0" is specified. (*1)
E	Engraving speed adjustment coefficient	1 to 999 (%)	This is a coefficient that adjusts the feedrate "F´" of the engraving path. Command the percentage (%) for the feedrate "F". If omitted, this function operates assuming that "100" is specified. (*1)
[ ]	Text to be machined	ASCII characters	Up to 100 characters can be specified. (*1)

(*1) Refer to the description below for details on the address.

Note

(1) If a command is given outside the command range, the operation described in the remarks is performed or the program error (P35) occurs.

(2) If a linefeed is inserted without closing square brackets "[ ]", the program error (P225) occurs.

(3) In square brackets [ ], an operator is recognized as a character; thus, no operation can be performed.

(4) If an invalid character is entered in square brackets [ ], the program error (P225) occurs.
At this time, the sub-error number indicates a character in which alarm is occurring for the entered character. However, if the entered character string exceeds the maximum number of characters that can be specified, "0" is selected as the sub-error number.

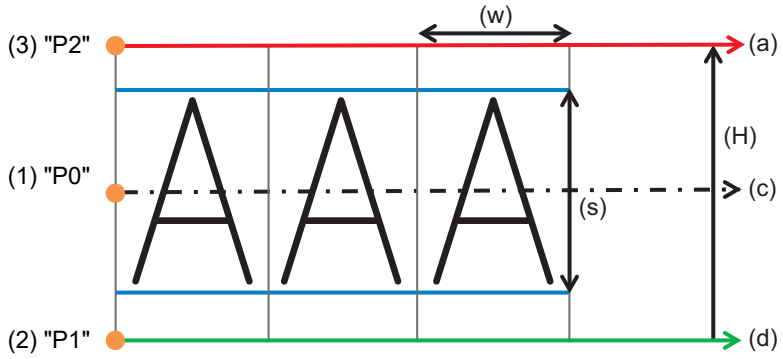
Text height H and text position P

Specify the text height with address "H". The text height is the distance from the descender line to the ascender line. Also, the line that passes between the descender line and the ascender line is referred to as the "center line". The machined text size is smaller than address "H". The text width is automatically calculated from the text height using the following formula.

"Text width" = "Text height" * 127 / 255

Specify the text position for the machining start position with address "P".

(1) "P0": The starting point on the center line is the machining start position.
(2) "P1": The starting point on the descender line is the machining start position.
(3) "P2": The starting point on the ascender line is the machining start position.



(H) Text height
(d) Descender line
(a) Ascender line
(c) Center line
(s) Text size
(w) Text width
(P) Machining start position

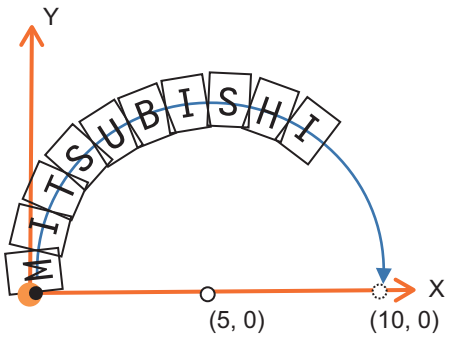
Arc centers I, J and text arc direction Q

To machine text in an arc shape, specify the center of the arc using addresses "I" and "J". Specify the distance from the machining start position to the arc center with the incremental values for the horizontal axis "I" and the vertical axis "J". For details on the combination of axes corresponding to the horizontal and vertical axes on the G17/G18/G19 plane, refer to "Coordinate system setting" described later.

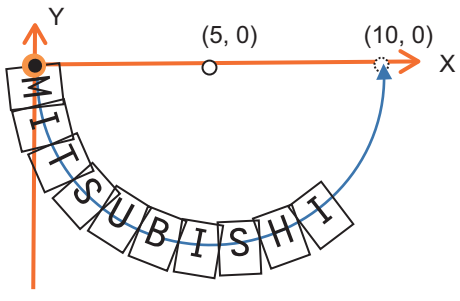
Specify the text rotation direction with address "Q".

(1) "Q0": Arrange text clockwise.
(2) "Q1": Arrange text counterclockwise.

[When commanding the rotation direction "Q" under the conditions that "I" is "5" and "J" is "0" on the G17 plane]



(1) "Q0"

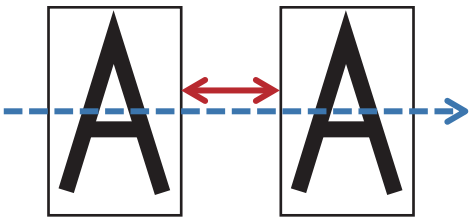


(2) "Q1"

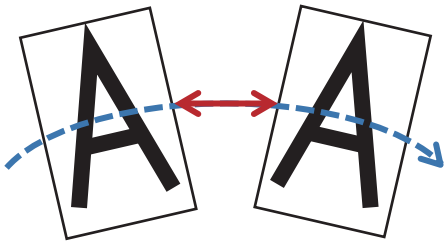
Text interval L

Specify the text interval with address "L".
When text is machined on a straight line, the text interval is the "distance of straight line".
When text is machined on an arc, the text interval is the "chord length". When text is machined on an arc, the chord to which the text interval is to be applied changes depending on the command of text position "P".

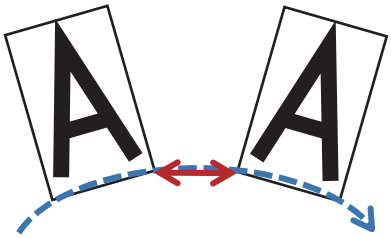
(1) When text is machined on a straight line



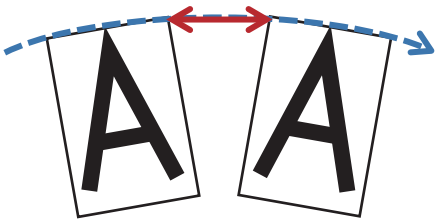
(2) When text is machined at the center of the arc ("P" is "0")



(3) When text is machined at the top of the arc ("P" is "1")



(4) When text is machined at the bottom of the arc ("P" is "2")

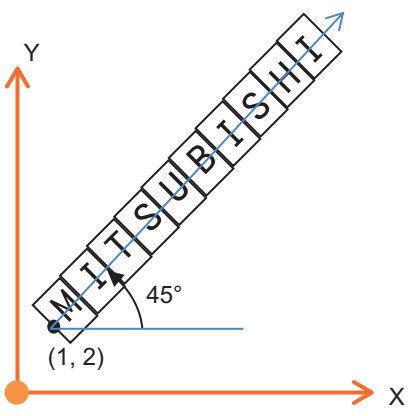


Note

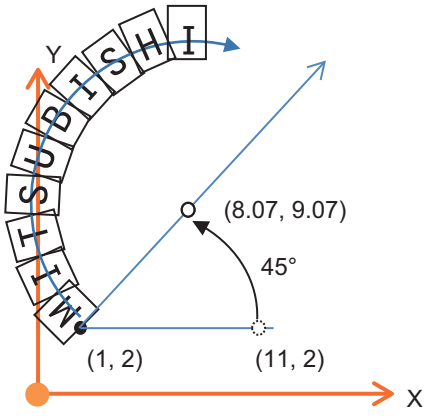
(1) You can specify a negative value for the text interval. In that case, the text interval is smaller than the text width, so text may overlap.

Text angle K

If you want to add an angle to the text, specify the angle with address "K". When the text is on an arc, the center point of the arc also rotates by angle "K".



(1) When the text is on a straight line

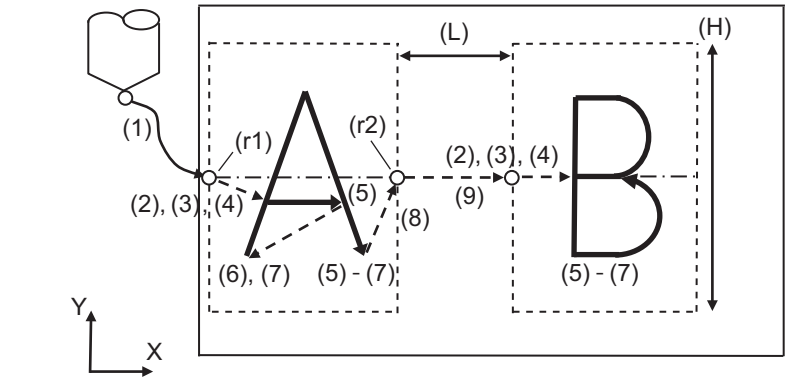


(2) When the text is on an arc

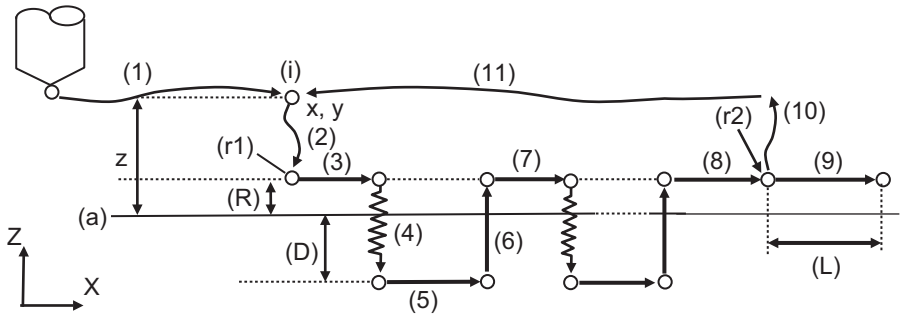
Detailed description

Details of operation

The movement command in the G135 command is as follows.



(a) Machining surface
(i) Initial point
(r1) R point
(r2) R2 point
(L) Text interval L
(H) Text height H
(R) Retract height R
(D) Depth D



Operation pattern	Operation
(1)	Axes other than the Z axis are positioned. This operation follows the modal of G code group 1. The position after the positioning has been completed by operation (1) is referred to as the "initial point".
(2)	The tool is positioned to the R point, which has been retracted by the retract height "R".
(3)	The tool is positioned at the first stroke by the cutting feed.
(4)	Text is engraved to the text depth "D". At this time, the feedrate "F´" can be adjusted using the address "E". If the value of address "E" is "α", "F´" is expressed by the following formula. F´ = F * α / 100
(5) - (7)	Text is machined by repeatedly machining, retracting, and positioning using the cutting feed.
(8)	The tool is positioned to the R2 point at the right end on the line that passes through the machining start point.
(9)	The tool moves by the text interval "L".
After that, steps (2) to (9) are repeated until all text has been machined.	
(10)	Only the Z axis is retracted to the same height as the initial point.
(11)	Axes other than the Z axis are positioned to the initial point.

Note

(1) Cutting feedrate "F" is the speed that is set after the cutting feed override was applied.

(2) As a result of calculation, if engraving feedrate "F´" is less than "1", "F"' operates as "1".

(3) During single block mode, the engraving pass and the positioning pass stop for each pass. However, in the machining path, a path that can be drawn in one stroke is considered as one operation and the block does not stop.

Coordinate system setting

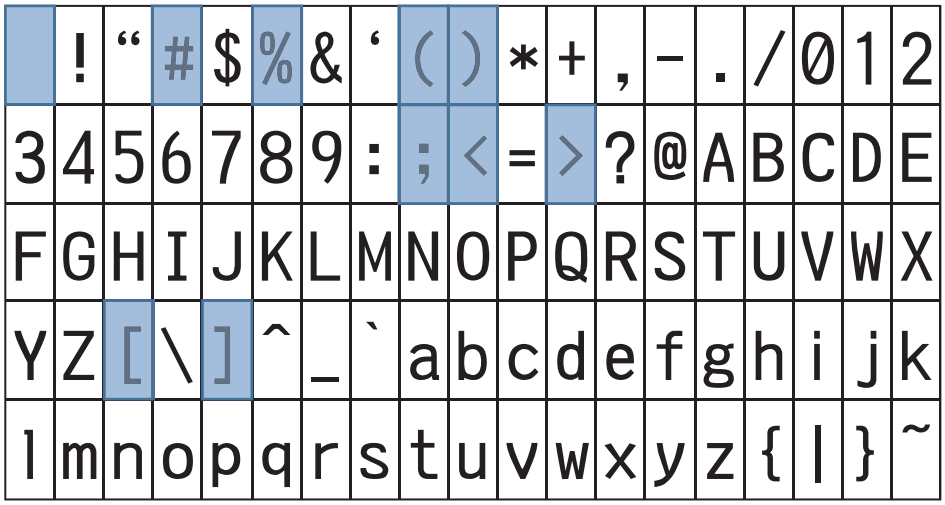
In this function, plane to machine the text is determined by the plane selection commands G17, G18, and G19. Regarding the tool axis direction, position the tool axis in advance so that it is the axis perpendicular to the selected plane (X, Y, Z or its parallel axis). Xp, Yp and Zp indicate the basic axes X, Y and Z or the parallel axes of the basic axes, respectively. "Plane to machine text" in the table below shows the axes in order of horizontal axis and vertical axis.

Plane selection	Plane to machine text
G17 (X-Y)	Xp-Yp
G18 (Z-X)	Zp-Xp
G19 (Y-Z)	Yp-Zp

Corresponding text

(1) ASCII characters

The table below shows the machinable characters. For characters that cannot be entered using the Mitsubishi Electric standard keyboard, create a program using an external device and enter them. (The shaded areas are not supported.)



Note

(1) If a character that is not supported is specified, the program error (P225) occurs.

(2) The maximum number of characters that can be specified is 100. If the number of characters exceeds 100, the program error (P225) occurs. Note that when a variable command or a specific address is used to specify the characters to be machined, even if the number of characters in square brackets [ ] is 100 or less, if the number of characters actually machined exceeds 100, the program error (P225) occurs.

(3) Blank characters (space), "<", ">", "[", "]", and ";" can be entered using the Mitsubishi standard keyboard; however, they cannot be machined. If the above symbol is entered in square brackets [ ], the program error (P225 or P281) occurs. If a space character is entered in square brackets [ ], it is ignored.

(4) The supported font is "Mitsubishi CNC Gothic". You cannot change the font.

(2) Characters by commanding a specific address

You can machine the date and time by specifying the address in the table below in square brackets [ ]. Each address must be enclosed in angle brackets < >.

Address	Meaning	Description
<Y>	Year	Machines the year with 4 digits.
<Y1>		Machines the last 2 digits of the year.
<M>	Month	Machines the month with 2 digits.
<D>	Day	Machines the day with 2 digits.
<T>	Time	Machines the time using 5-digit 24-hour notation (00:00).
<T1>		Machines the time using 5-digit 12-hour notation (00:00).

<Example> In the case of "November 2, 2021 16:00", the numbers machined when a specific address is commanded are as follows.

Address	<Y>	<Y1>	<M>	<D>	<T>	<T1>
Machining details	2021	21	11	02	16:00	04:00

Note

(1) If multiple addresses are specified in angle brackets <>, the program error (P225) occurs.

(2) If an address is combined with a numerical value and a combination other than the above is specified, the program error (P225) occurs.

(3) The value that is output with the characters by commanding a specific address is the value specified at the start of G135 command execution.

(4) In the background check simulation, it is not possible to output the correct date and time. The following fixed values are output regardless of the actual values.
Interference checks are also performed using the following fixed values in the pre-reading process in the 3D machine interference check. Note that they are different from the date and time to be machined.

Address	<Y>	<Y1>	<M>	<D>	<T>	<T1>
Machining details	2000	00	01	01	00:00	00:00

(3) Variable command

By writing "<#Variable number/Number of integer digits . Number of decimal digits>" in [ ], the numerical value set in the variable can be machined as a text. The specifiable variables are common variables, local variables, system variables, and extended common variables. For details on the variable command, refer to "14.2 Variable Commands".

If "/Number of integer digits . Number of decimal digits" is omitted, the variable is machined as a number with 5 digits in the integer part and 3 digits in the decimal part. The maximum number of significant digits is 8 digits for the integer part and 8 digits for the decimal part. At this time, the sign is not included in the number of significant digits. If the variable value is smaller than the specified number of digits, add "0" to the upper digits for the integer part and the lower digits for the decimal part. Also, if the variable value is larger than the specified number of digits, the upper digits are rounded down in the integer part, and the lower digits are rounded down in the decimal part.

If a period "." or the number after a period "." is omitted, this function operates assuming that the decimal part does not contain any digits. If the number before a period "." is omitted or "0" is entered before a period ".", the value of the integer part is not machined. For the detailed conversion by input/output, refer to the table below.

Variable value	Contents of command	Number to machine
#100 = 12.345	<#100/1.2>	2.34
	<#100/4.5>	0012.34500
	<#100/2.0>, <#100/2.>, <#100/2>	12
	<#100/0.3>, <#100/.3>	.345
	<#100>	00012.345
#100 = -12.345	<#100/1.2>	-2.34
#100 = 12.345	<#100/>, <#100/.>, <#100/0.0>, <#100/.0>, <#100/0>	Program error (P225)
	<#100/9.0>, <#100/0.9>	Program error (P225)

Note

(1) If multiple addresses are specified in angle brackets <>, the program error (P225) occurs.

(2) When both the decimal part and the integer part of a significant figure are "0", the program error (P225) occurs. Also, if nine or more digits are commanded, the program error (P225) occurs.

(3) Variable numbers cannot be specified by variables as "G135 [<## 100>]". If it is commanded, the program error (P225) occurs.

(4) This function does not support the numerical exponential notation.

(5) The program error (P241) occurs if a write-only system variable is commanded or an unavailable number is commanded.



Program example

Text engraving is performed with the following conditions.

♦The lower left point of the workpiece is the zero point in the workpiece coordinates.

	[mm]
Workpiece size	W200* D100* H20
Tool radius	2
Retraction height R	2.5
Engraving depth D	1
Text height H	20
Text interval L	5
Text to be machined	MITSUBISHI

G91 G28 Z0 ;	Zero point return of Z axis
G91 G28 X0 Y0 ;	Zero point return of X and Y axes
G90 G0 G54 G17 X0. Y0. ;	Workpiece coordinates and X-Y plane designation
G0 G43 H1 Z50. F3000 ;	Tool length compensation and approach to workpiece
G135 X20. Y50. Z0. R2.5 D1. H20. L5. [MITSUBISHI] ;	Text engraving cycle
G91 G0 Z50. ;	Tool retraction
G91 G28 Z0 ;	Zero point return of Z axis
G91 G28 X0 Y0 ;	Zero point return of X and Y axes
M30 ;	End command

Relationship with other functions

"G135" is a single command. If another G code is commanded in the same block, the program error (P33) occurs.

Relationship between text engraving cycle and other functions

Column A: Operation when the text engraving cycle (G135) is commanded while the combination function is enabled

○: The text engraving cycle command can be issued, and the combination function is also enabled.

Δ: The text engraving cycle command can be issued, while the combination function is disabled.

×: The text engraving cycle command cannot be issued, and an alarm occurs. (The text in parentheses ( ) refers to the number of the program error that occurs.)

□: Others

Group	G code	Function name	A
0	G05P0	High-speed machining mode II OFF High-speed high-accuracy control II OFF High-speed high-accuracy control III OFF	○
	G05P2	High-speed machining mode II ON	Δ
	G05P10000	High-speed high-accuracy control II ON	□ (*1)
	G05P20000	High-speed high-accuracy control III ON	□ (*1)
	G05.1Q0	High-speed high-accuracy control I OFF Spline interpolation OFF	○
	G05.1Q1	High-speed high-accuracy control I ON	□ (*1)
	G05.1Q2	Spline interpolation ON	Δ
	G07	Hypothetical axis interpolation	× (P29)
	G08P0	High-accuracy control OFF	○
	G08P1	High-accuracy control ON	○
	G60	Unidirectional positioning	○
	G65	User macro simple call	□ (*2)
	G122	Sub part system control I	○
	G144	Sub part system control II	○
1	G00	Positioning	Δ
	G01	Linear interpolation	Δ
	G02 G03	Circular interpolation	Δ
	G02.1 G03.1	Spiral interpolation	× (P29)
	G02.3 G03.3	Exponential interpolation	× (P29)
	G02.4 G03.4	3-dimensional circular interpolation	× (P29)
	G06.2	NURBS interpolation	× (P29)
	G33	Thread cutting	× (P29)
	G60	Unidirectional positioning	□ (*3)
2	G17 - G19	Plane selection	○
3	G90	Absolute command	○
	G91	Incremental command	○
4	G22	Stroke check before travel ON	○
	G23	Stroke check before travel OFF	○

Group	G code	Function name	A
5	G93	Inverse time feed	× (P29)
	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	○
	G43.4 G43.5	Tool center point control	× (P942)
	G43.7	Tool position compensation	○
	G49	Tool length offset cancel	○
9	G80	Fixed cycle cancel	○
	Group 9 Other than G80	Fixed cycle	× (P29)
10	G98	Fixed cycle (Initial level return)	○
	G99	Fixed cycle (R point level return)	○
11	G50	Scaling cancel	○
	G51	Scaling ON	× (P29)
12	G54 - G59 G54.1	Workpiece coordinate system selection	○
13	G61	Exact stop check mode	○
	G61.1	High-accuracy control	○
	G61.2	High-accuracy spline	□ (*4)
	G61.4	Spline interpolation 2	□ (*4)
	G62	Automatic corner override	○
	G63	Tapping mode	× (P29)
	G64	Cutting mode	○
14	G66 G66.1	User macro modal call	× (P29)
	G67	User macro modal call cancel	○
15	G40.1	Normal line control cancel	○
	G41.1 G42.1	Normal line control	× (P29)
16	G68	Coordinate rotation by program ON	○
	G68.2 G68.3	Inclined surface machining command	○
	G69	Coordinate rotation cancel	○
17	G96	Constant surface speed control ON	× (P29)
	G97	Constant surface speed control OFF	○
18	G15	Polar coordinate command OFF	○
	G16	Polar coordinate command ON	× (P29)
19	G50.1	Mirror image OFF	○
	G51.1	Mirror image ON	○
21	G07.1	Cylindrical interpolation	○
	G12.1	Polar coordinate interpolation ON	○
	G13.1	Polar coordinate interpolation OFF	○

Group	G code	Function name	A
27	G54.4P0	Workpiece installation error compensation cancel	○
	G54.4 P1 - P7	Workpiece installation error compensation	○

(*1) High-speed machining mode is disabled.

(*2) Can be commanded in a subprogram.

(*3) Only available for modal commands.

(*4) Operates as the G61.1 command modal is active.

Relationship between text engraving cycle and functions other than G codes

Column A: Operation when the text engraving cycle (G135) is commanded while the combination function is enabled

○: The text engraving cycle command can be issued, and the combination function is also enabled.

Δ: The text engraving cycle command can be issued, while the combination function is disabled.

×: The text engraving cycle command cannot be issued, and an alarm occurs. (The text in parentheses ( ) refers to the number of the program error that occurs.)

□: Others

Function name	A
SSS ON	Δ
Mirror image by parameter setting ON	○
Mirror image by external input	○
Coordinate rotation by parameter	○
Subprogram call (M98)	□ (*1)
Figure rotation (M98 I_J_K_)	□ (*1)
Machine tool builder macro	□ (*2)
Manual interruption	× (M01 0182)
Automatic operation handle interruption	× (M01 0182)
PLC interrupt	× (M01 0182)
MDI interruption	× (M01 0182)
Macro interruption (M96)	× (P29)
Arbitrary reverse run	Δ
Manual speed command	× (M01 0182)
Manual arbitrary reverse run	× (M01 0182)
Chopping	× (M01 0182)

(*1) Can be commanded in a subprogram.

(*2) Can be commanded in a program.



Precautions

(1) No interrupt is possible while the text engraving cycle command is in execution. If the command is interrupted, the operation error (M01 0182) occurs.

(2) When the high-accuracy control mode is OFF, the tool may not reach the engraving depth commanded in the machining. Therefore, it is recommended to machine with the high-accuracy control mode ON.

(3) When arranging text in an arc, specify the arc diameter to be at least half the character height and at least the character interval. If these are not satisfied, the program error (P229) occurs.

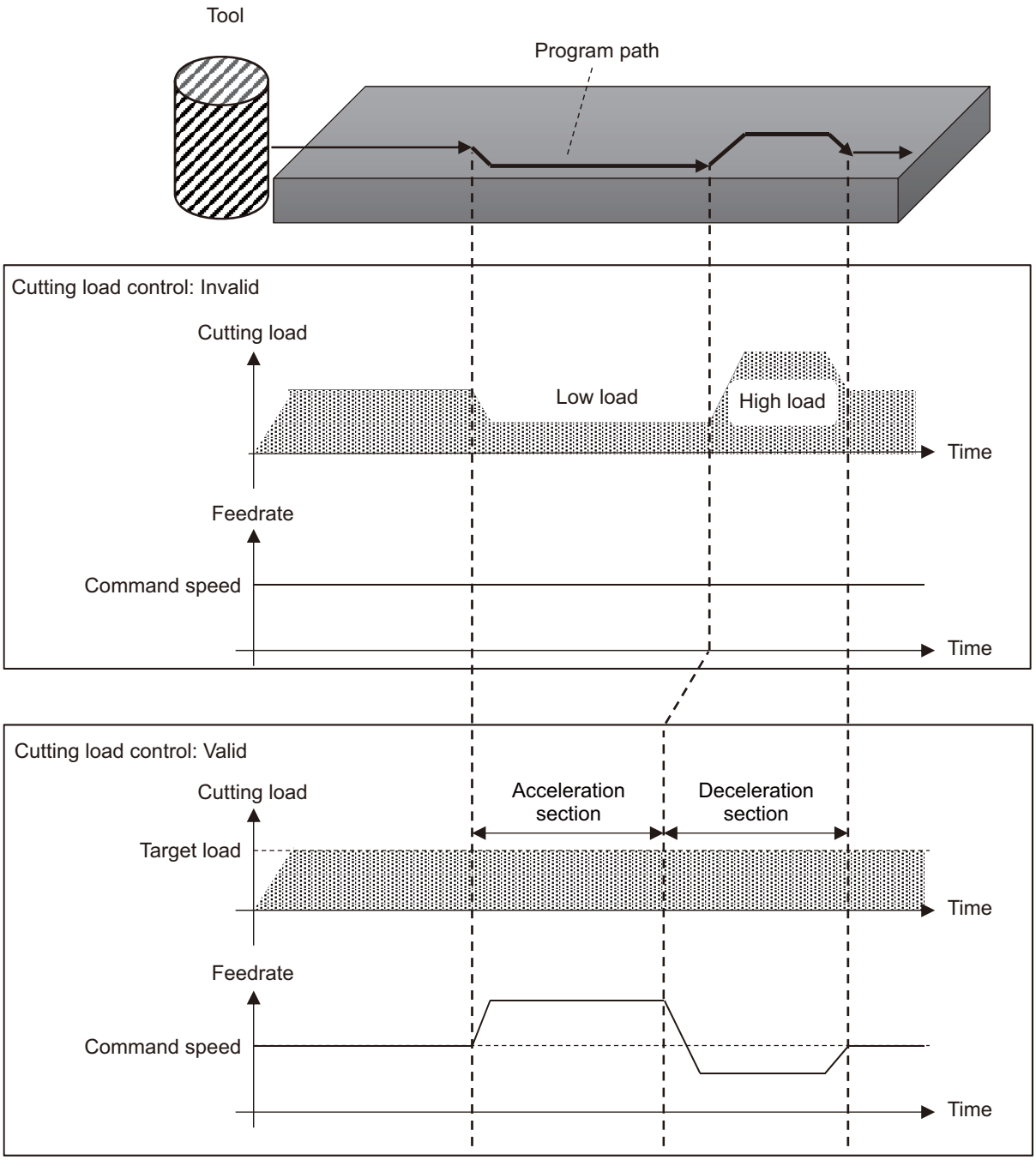
(4) By machining with this command during cylindrical interpolation, you can perform the text engraving machining on the cylindrical surface. The command unit of rotary axis in G135 block during cylindrical interpolation is degrees (°). For details on the cylindrical interpolation, refer to "6.9 Cylindrical Interpolation; G07.1".

(5) When the parameter "#19431 Num display in prg" is set to "1", a character string specified in [ ] is displayed in the program buffer. It may be displayed differently from the actual character string due to the specifications of program buffer display. (Refer to "14.8 Variable Value Display in Program Being Executed" for variable value display in program being executed.)

15.16 Cutting Load Control; G162.1

Function and purpose

This function automatically changes the feedrate so that the cutting load reaches the set target load. For machining with a low cutting load, the feedrate can be increased and the cycle time can be shortened. On the other hand, for machining with a high cutting load, the tool can be protected by reducing the feedrate and reducing the load on the tool.
This function may affect the surface quality of the workpiece as the feedrate increases or decreases; therefore, use it for the machining such as rough machining.



Term

The following describes the meanings of the terms used in this chapter.

Word	Meaning
Cutting load	This is the load value during cutting.
CLC	This stands for Cutting Load Control.

Command format

Cutting load control function: Valid

```
G162.1 P1 (L__);
```

Cutting load control function: Invalid

```
G162.1 P0;
```

L	Specification of parameter group number (1 to 8) Specify the number of the parameter set used in the cutting load control. If omitted, L1 (parameters #19200 to #19203) is specified.

Parameter group numbers and corresponding parameters

This function controls the cutting load according to the "CLC desired load", "CLC detect.load", "CLC max.feedrate", and "CLC min.feedrate" parameters. Eight sets of parameters can be specified as shown in the table below. The parameter suitable for the tool or workpiece can be selected by specifying the parameter group number of the set to be used in address L. The parameter group specification is only valid for the commanded part system.

It is necessary to set the parameters before using this function. For the setting method, refer to "Setting of the desired load and cutting detection load" described later.

Parameter name	Parameter group No.							
	1	2	3	4	5	6	7	8
CLC desired load	#19200	#19205	#19210	#19215	#19220	#19225	#19230	#19235
CLC detect.load	#19201	#19206	#19211	#19216	#19221	#19226	#19231	#19236
CLC max.feedrate	#19202	#19207	#19212	#19217	#19222	#19227	#19232	#19237
CLC min.feedrate	#19203	#19208	#19213	#19218	#19223	#19228	#19233	#19238

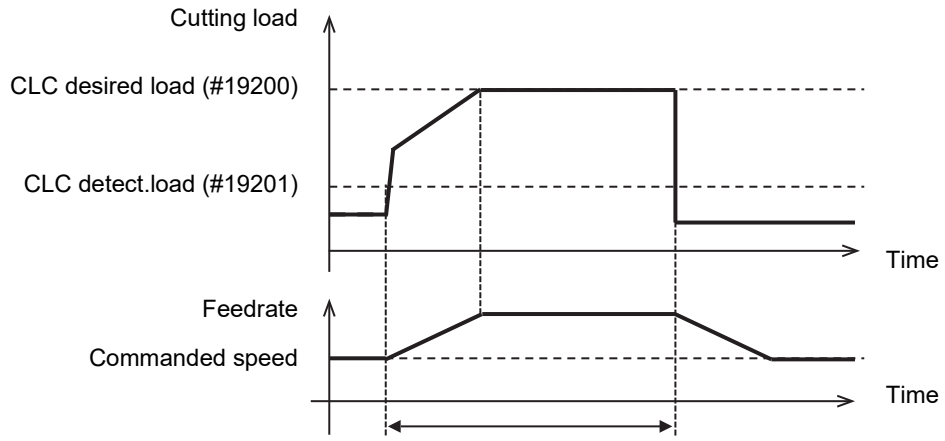
Detailed description

Operation during cutting load control

This function controls the feedrate so that the cutting load becomes the value of the "CLC desired load" (#19200). The feedrate is controlled between the "CLC max.feedrate" (#19202) and the "CLC min.feedrate" (#19203) that has been specified.

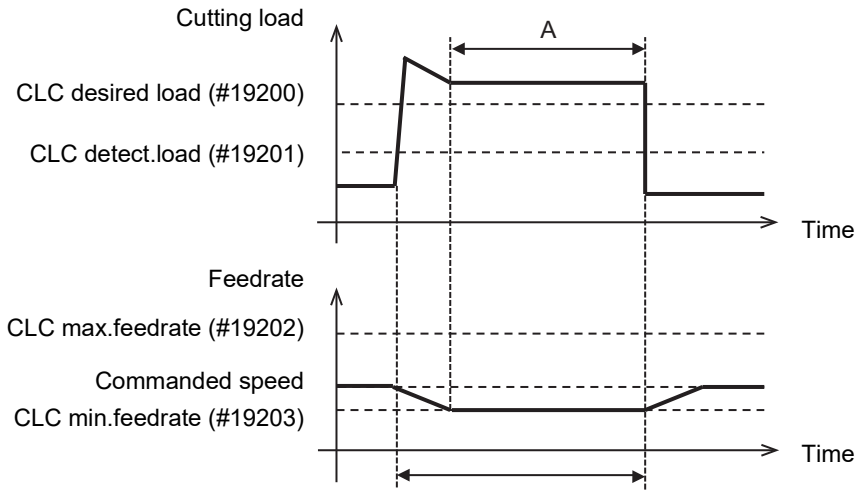
The cutting load control is enabled when the cutting load is greater than or equal to the "CLC detect.load", and is disabled when the cutting load is less than the "CLC detect.load".
There is a dead zone in the range of -0.5% to 0.5% with respect to the "CLC desired load", and the feedrate does not change while the cutting load is in the dead zone. For example, if the "CLC desired load" is 20% and the cutting load is 19.5% to 20.5%, the feedrate does not change.

The cutting load is the load value of the spindle specified in the "CLC: Spindle selection" signal (R22540).



Under cutting load control

Depending on the setting of parameters, the feedrate may be the same value as the "CLC max.feedrate" (#19202) or the "CLC min.feedrate" (#19203) and the cutting load may not reach the value of "CLC desired load" (#19200).



Under cutting load control

During the period indicated by "A" in the figure, the feedrate becomes the minimum feedrate, and the cutting load cannot be reduced.

Setting of desired load and cutting detection load

For the "CLC desired load" (#19200), set the cutting load to be constant during the cutting load control. For the "CLC detect.load" (#19201), set the load to start the cutting load control.

Determine the desired load and cutting detection load by referring to the spindle load value during operation of the machining program to be used. The maximum and minimum values are output to the R registers (R20564, R20565), as the spindle load value during operation.

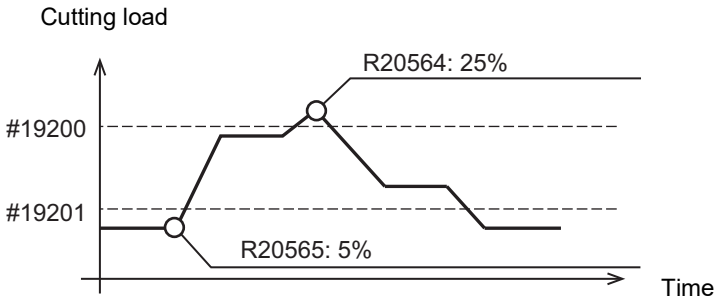
The cutting load during this function may exceed the value set to the "CLC desired load". When setting the "CLC desired load", set a value less than the maximum value of the spindle load value.

For the "CLC detect.load", set a value higher than the spindle load value during non-cutting time. If a value less than or equal to the spindle load value during non-cutting time is set, the feedrate increases during non-cutting time and the tool comes into contact with the workpiece at a speed faster than the commanded speed, which may damage the tool.

[Setting procedure]

(1) Perform machining with the cutting load control disabled.

(2) After machining is completed, check the maximum and minimum values of the spindle load value, and set the "CLC desired load" and "CLC detect.load".
For example, set the "CLC desired load" to approximately 80% of the maximum cutting load and set the "CLC detect.load" to a value that is +2% of the minimum value of the spindle load value.



In the case of this example, the settings are as follows:

As the maximum cutting load (R20564) is 25%, set the "CLC desired load" (#19200) to "20%", which is 80% of the maximum cutting load.

As the minimum cutting load (R20565) is 5%, set the "CLC detect.load" (#19201) to "7%" by adding 2% to the minimum cutting load.

Relationship with other functions

Mode in which the cutting load control is enabled

Cutting load control is enabled when it is commanded while any of the following modals is active. If a modal other than the following is active, the cutting load control is disabled even if it commanded. When a function other than the following functions is commanded while the cutting load control is enabled, the cutting load control is disabled.

Modal commands that enable the cutting load control

Command	G group	Function
G01	1	Linear interpolation
G02, G03		Circular interpolation
G02.1, G03.1		Spiral interpolation
G02.2, G03.2		Involute interpolation
G02.3, G03.3		Exponential interpolation
G02.4, G03.4		3-dimensional circular interpolation
G06.2		NURBS interpolation
G17, G18, G19	2	Plane selection (X-Y, Z-X, Y-Z)
G90, G91	3	Absolute command, Incremental command
G22, G23	4	Stroke check before travel ON/OFF
G93	5	Inverse time feed
G94		Feed per minute (asynchronous feed)
G20, G21	6	Inch command, Metric command
G40	7	Tool radius compensation cancel
G41, G42		Tool radius compensation
G41.2, G42.2		3-dimensional tool radius compensation
G43, G44	8	Tool length compensation (+), Tool length compensation (-)
G43.1		Tool length compensation along the tool axis
G43.4, G43.5		Tool center point control
G43.7		Tool position compensation
G43.8, G43.9		Tool cutting point control
G49		Tool length compensation cancel
G70 - G72, G75, G77 - G79	9	User fixed cycle
G80		Fixed cycle cancel
G98, G99	10	Hole drilling cycle I point return, R point return
G50, G51	11	Scaling ON/OFF
G54 - G59, G54.1	12	Workpiece coordinate system selection, Extended workpiece coordinate system selection
G61	13	Exact stop check mode
G61.1		High-accuracy control
G61.2		High-accuracy spline interpolation 1, 2
G61.4		Spline interpolation 2
G62		Automatic corner override
G64		Cutting mode
G66, G66.1, G67	14	User macro modal call A/B/cancel
G40.1	15	Normal line control cancel

Command	G group	Function
G68	16	Coordinate rotation, 3-dimensional coordinate conversion
G68.2, G68.3		Inclined surface machining, Inclined surface machining (Tool axis direction command)
G69		Coordinate rotation cancel, 3-dimensional coordinate conversion cancel
G97	17	Constant surface speed control OFF
G15, G16	18	Polar coordinate command ON/OFF
G50.1, G51.1	19	Mirror image by G code ON/OFF
G7.1	21	Cylindrical interpolation
G12.1		Polar coordinate interpolation
G13.1		Polar coordinate interpolation cancel
G54.2 P0, 1 - 8	23	Workpiece position offset for rotary axis ON/OFF
G54.4 P0, 1 - 7	27	Workpiece installation error compensation ON/OFF
G0.5 P0, P1	28	Rapid traverse block overlap ON/OFF

Non-modal commands can be issued even while this function is active; however, this function is disabled for blocks in which no axis movement is commanded. The non-modal commands for which this function is enabled are as follows.

Non-modal commands that enable the cutting load control

Command	G group	Function
G05 P0 / P1 / P2 /	0	High-speed machining mode I/II
P10000 / P20000		High-speed high-accuracy control II/III
G05.1 Q0 / Q1		High-speed high-accuracy control I
G08 P0 / P1		High-accuracy control
G09		Exact stop check
G12, G13		Circular cutting CW/CCW
G38		Tool radius compensation vector designation
G39		Tool radius compensation corner arc
G45 - 48		Tool position offset
G53		Machine coordinate system setting
G65		User macro simple call

Operations with other functions combined

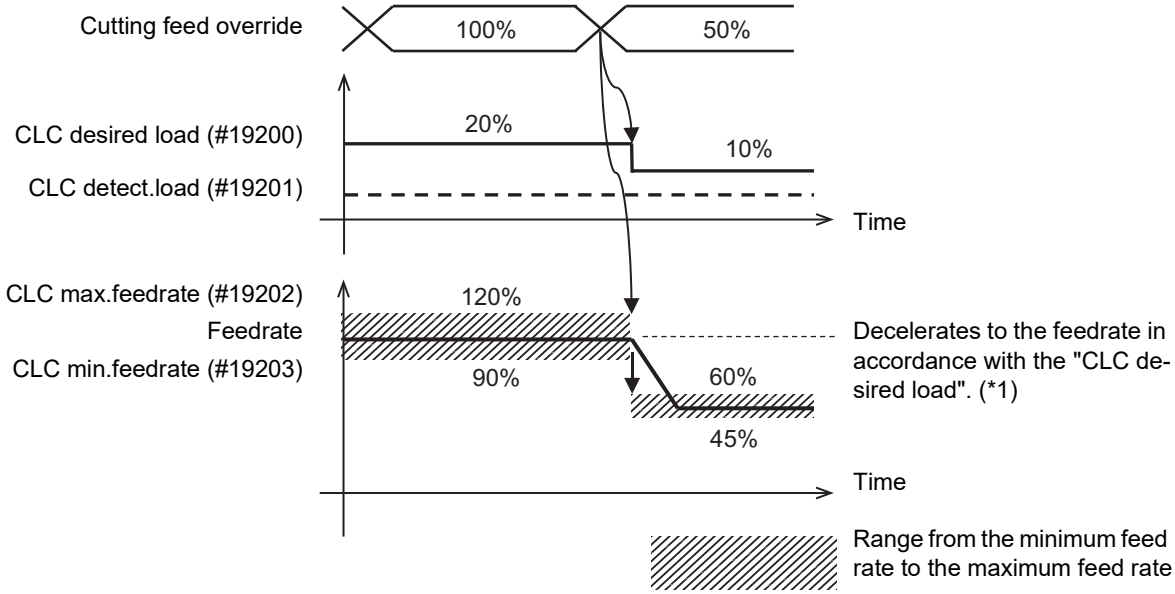
(1) Cutting feed override, 2nd cutting feedrate override

The "cutting feed override" (*FV1 to 16 or R2500) and the "2nd cutting feedrate override" (R2501) while this function is performed are applied to the "CLC desired load" (#19200), "CLC max.feedrate (#19202), and "CLC min.feedrate (#19203). At this time, the "CLC desired load" (#19200) is rounded down in 1% increments.

The cutting load control is disabled when the "CLC desired load" (#19200) to which the cutting feed override is applied becomes less than the "CLC detect.load" (#19201).

If the cutting feed override is 0%, the axis movement stops, and the operation error (M01 0102) is displayed.

Combined use with cutting feed override



(*1) The feedrate changes in accordance with the acceleration/deceleration setting after interpolation; therefore, it may exceed the range from the minimum feed rate to the maximum feed rate.

Parameter setting

CLC desired load (#19200)	20%
CLC detect.load (#19201)	7%
CLC max.feedrate (#19202)	120%
CLC min.feedrate (#19203)	90%

(2) Program format switch

The cutting load control is available while the program format is switched.

(3) Override cancel

When the "Override cancel" signal (OVC) is ON, this function is disabled, and the cutting feed override becomes 100%.

(4) Analog spindle I/F

If the spindle that detects the cutting load is an analog spindle, the cutting load control is disabled.

(5) Multiple-spindle control II

Set the "Cutting load spindle selection" signal (R22540) to change the combination of the spindle that detects the cutting load and the part system that changes the feedrate.

(6) Spindle position control (spindle/C axis control)

When a spindle whose cutting load is to be detected is under the spindle position control (spindle/C axis control), in spindle mode, cutting load control is enabled and controls the feedrate according to the cutting load. In the C-axis mode, cutting load control is disabled, and the feedrate is not changed from the commanded speed.

(7) Spindle-mode rotary axis control

When a spindle whose cutting load is to be detected is under the spindle-mode rotary axis control, cutting load control is disabled.

(8) Spindle synchronization control I/II, Tool spindle synchronization I/II

The cutting load control is disabled during spindle synchronization or tool spindle synchronization.

(9) Spindle oscillation

The cutting load control is disabled during the spindle oscillation.

(10) Manual interruption, Simultaneous operation of manual and automatic modes, Automatic operation handle interruption

The cutting load is controlled for the feedrate commanded by the automatic operation (tape, MDI, or memory), and the feedrate is changed.
The speed by manual operation (handle, step, jog, or manual reference point return) is not changed by this function.

(11) 3D interference check

The cutting load control is enabled during the 3D interference check. However, the feedrate may be clamped.

(12) Automatic corner override

The cutting load control is enabled even during the automatic corner override operation. Depending on the cutting load, the feedrate may increase at a location where the feedrate was decelerated by the automatic corner override.

(13) Program restart

After the restart search, the cutting load control is disabled while the axis returns to the restart position. This function is enabled after the axis is returned.

(14) F 1-digit feed

The cutting load control is enabled while the F 1-digit feed command is being performed. Note that speed cannot be changed with the manual handle.

(15) Dry run

The cutting load control is disabled while the dry run is being performed.

(16) Machine lock

The cutting load control is disabled while the machine lock is being performed.

(17) External deceleration

The cutting load control is disabled while the external deceleration is being performed.

(18) Manual speed command

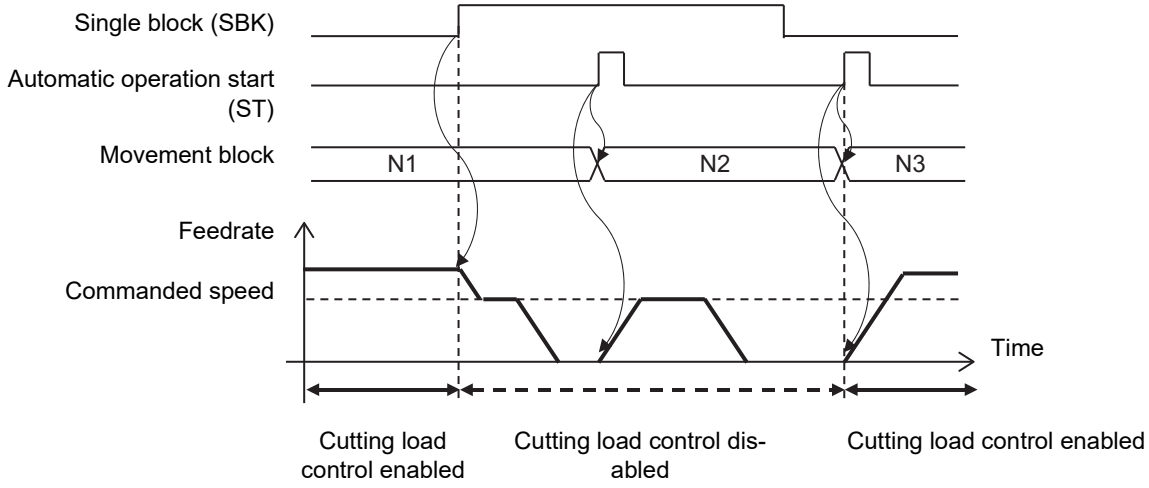
The cutting load control is disabled while the manual speed command is being performed.

(19) Manual arbitrary reverse run

The cutting load control is disabled while the arbitrary reverse run is being performed.

(20) Single block

The cutting load control is disabled by setting the "Single block" input signal ON.
The cutting load control is enabled by setting the "Single block" input signal OFF and starting cycle operation.





Precautions

(1) Issue the G162.1 command independently. If it is not commanded independently, the program error (P33) occurs.

(2) If address P is not specified in the G162.1 command block, the program error (P33) occurs. Also, if the value other than "0" or "1" is specified in address P, the program error (P35) occurs.

(3) If the value other than "1" to "8" is specified in the address L of the G162.1 command block, the program error (P35) occurs.

(4) If G162.1 is commanded when the specifications is not provided for the cutting load control function, the program error (P39) occurs.

(5) This function is disabled by NC reset.

(6) For the "CLC desired load" (#19200) and "CLC detect.load" (#19201), set the greater value than "0". For the "CLC desired load" (#19200), set the greater value than the "CLC detect.load" (#19201). When the "CLC desired load" (#19200) or the "CLC detect.load" (#19201) is "0", this function is disabled even when the "G162.1 P1" command is issued. Also, when the "CLC detect.load" (#19201) is set to a greater value than the "CLC desired load" (#19200), this function is disabled in the same manner.

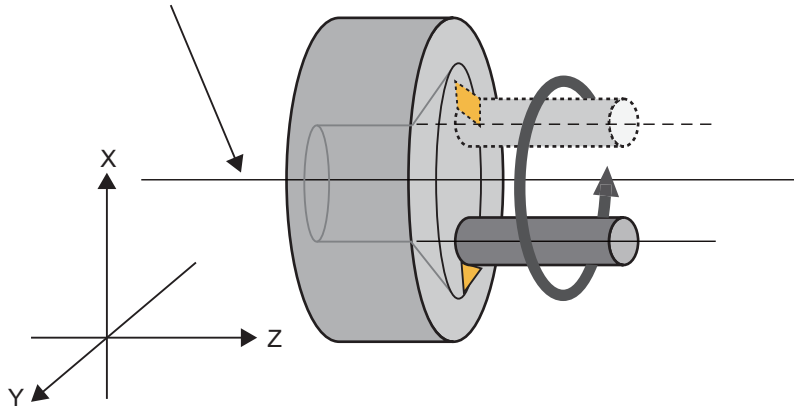
(7) For the "CLC max.feedrate" (#19202), set the greater value than the "CLC min.feedrate" (#19203). When the "CLC max.feedrate" (#19202) is set to be less than or equal to the "CLC min.feedrate", this function is disabled even when the "G162.1 P1" command is issued.

15.17 Orbit Machining; G08.6

Function and purpose

This function allows you to perform machining equivalent to turning on a lathe using a machining center.
The circular motion is performed while the tool nose is perpendicularly pressed against the machining surface by synchronizing the three basic axes with the rotation of the turning tool.
This function enables finish machining of different inner or outside diameter of a workpiece without changing the turning tools.

Circular motion center



Term

The following describes the meanings of the terms used in this chapter.

Term	Meaning
Orbit machining mode	This function is enabled and the orbit machining is available.
Orbit spindle	It is the tool spindle to perform the circular motion of turning tool in the orbit machining. It is specified with the parameter "#8183 Orbit_Spindle No.".
Circular motion	It is the circular motion of the orbit spindle on a plane perpendicular to the tool axis direction.
Circular motion center	It is the center position of the circular motion.
Circular motion plane	It is the plane to perform the circular motion. It is automatically selected by the plane selection (G17/G18/G19).
1st circular motion axis	It is the horizontal axis of the circular motion plane. The workpiece radius/diameter dimension is commanded. It can be commanded during the orbit machining mode.
2nd circular motion axis	It is the vertical axis of the circular motion plane. It cannot be commanded during the orbit machining mode.
Feed direction axis	It is the axis perpendicular to the circular motion plane and the horizontal axis of the selected plane. It can be commanded during the orbit machining mode.
Orbit machining axes	It is the generic name of axes used in the orbit machining. (Generic name of orbit spindle, 1st circular motion axis, 2nd circular motion axis and feed direction axis)

Enabling conditions of the function

This function is enabled when all the conditions shown below are satisfied.

Parameter	Setting value	Contents
#1007 System type select	0	Machining center system (M system) is selected.
#8183 Orbit_Spindle No.	1 to 8	The tool spindle is selected.
#3129 cax_spec/bit0	1	The program command method is used to switch between C axis mode and spindle mode of the tool spindle. (*1)
#1037 cmdtyp	1 or 2	Compensation data is type I or II. (Type III is not supported.) (*2)
#1046 T-ofs disp type	0	The tool offset display follows the type specified with the parameter "#1037 cmdtyp".

(*1) The tool spindle needs to be switched to the C axis mode before this function is executed.

(*2) When the tool nose radius compensation is used, select the type II. The optional specification of tool nose radius compensation is required.

Command format

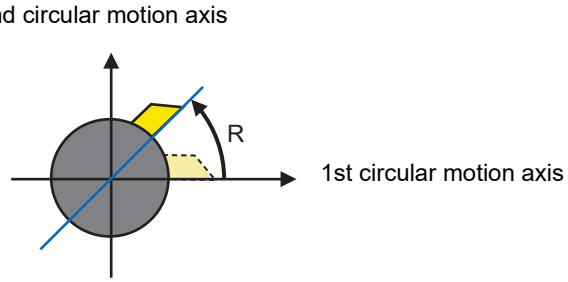
Orbit machining (G08.6) is a non-modal command of the G code group 00.
The orbit machining command (G08.6) must be issued alone in a block. If another command is in the same block, the program error (P994) occurs.

Orbit machining mode start

G08.6 Q__ R__;

Q	Select the machining surface of the orbit machining. Q1: Start the internal machining. Q2: Start the outside machining. If the address Q is omitted, the program error (P33) occurs.
R	Command the tool nose angle. (0.000 to 359.999) (*1) If the address R is omitted, it is assumed to be "0.000".

(*1) Command the tool nose angle to the 1st circular motion axis (positive direction) at the moment the orbit spindle is positioned with the "C0." command, for the address R.



Orbit machining mode cancel

G08.6 Q0;

Q	Cancel the orbit machining mode with the "Q0" command. If the address Q is omitted, the program error (P33) occurs.

Detailed description

Operation during the orbit machining mode

The mode turns to the orbit machining mode by commanding "G08.6 Q1 (or Q2)".
The "In orbit machining mode" signal turns ON during the orbit machining mode.

(1) Designation of circular motion plane, circular motion axis and feed direction axis

Before starting the orbit machining, select the plane to command the machining path with the plane selection command (G17/G18/G19). The circular motion plane, circular motion axis and feed direction axis used in the orbit machining mode are determined as follows according to the selected plane.

	G17 plane	G18 plane	G19 plane
Circular motion plane	Y-Z	X-Y	Z-X
1st circular motion axis	Y axis	X axis	Z axis
2nd circular motion axis	Z axis	Y axis	X axis
Feed direction axis	X axis	Z axis	Y axis

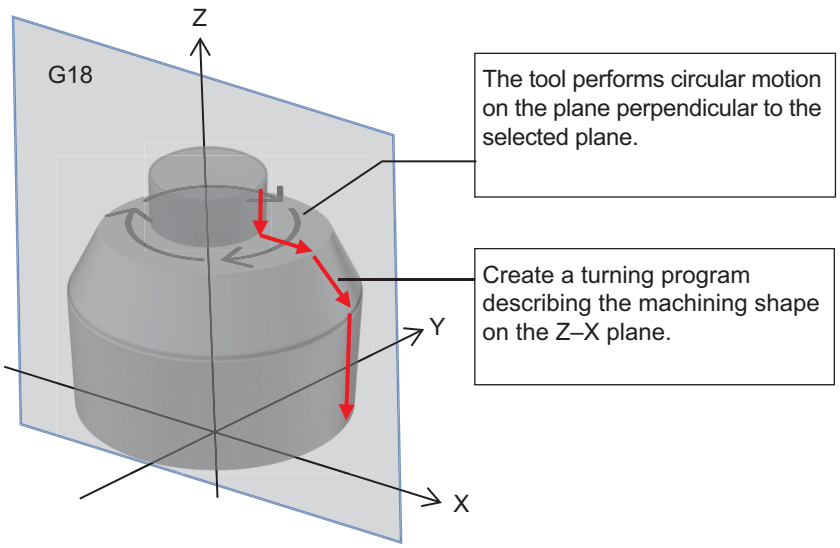
(2) Machining command

Command the machining path with the 1st circular motion axis and the feed direction axis during the orbit machining mode.
Use the diameter value to command the 1st circular motion axis.

[Example to command the machining path on the G18 plane]

X: 1st circular motion axis

Z: Feed direction axis



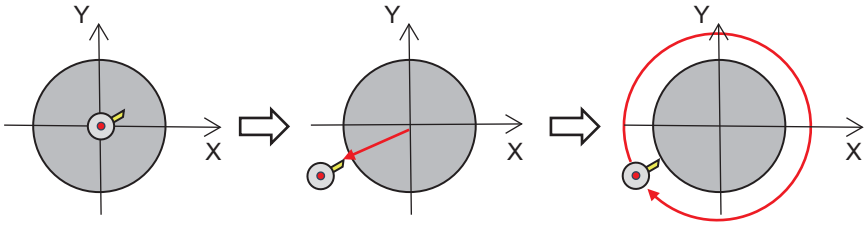
<Note>
- If a movement command is issued for the 2nd circular motion axis and the orbit spindle (C axis) during the orbit machining mode, the program error (P993) occurs.
- Even in orbit machining mode, movement commands can be issued to axes other than the orbit machining axes. In such cases, make sure to avoid mechanical interference with the circular motion of orbit machining.

(3) Circular motion center and circular motion start

The circular motion axis position of when "G08.6 Q1 (or Q2)" is commanded is the circular motion center. The circular motion center is the program zero point during the orbit machining mode.
Command the circular motion with the spindle command (S command, spindle forward rotation command, and spindle reverse rotation command) for the orbit spindle.
To control the circular motion radius, issue the movement command for the 1st circular motion axis. The orbit spindle moves so that the tool nose always points normal to the circular motion path regardless of whether the circular motion is in progress or stopped.

Once orbit machining mode is started, positioning the circular motion radius before the circular motion starts reduces cycle time compared with positioning it after the circular motion has started. This is because movement is performed without applying speed clamp based on the tolerable acceleration rate of orbit machining and based on the clamp speed of rapid traverse rate/cutting feedrate while the circular motion is stopped.

[Positioned before the circular motion starts] (outside machining)

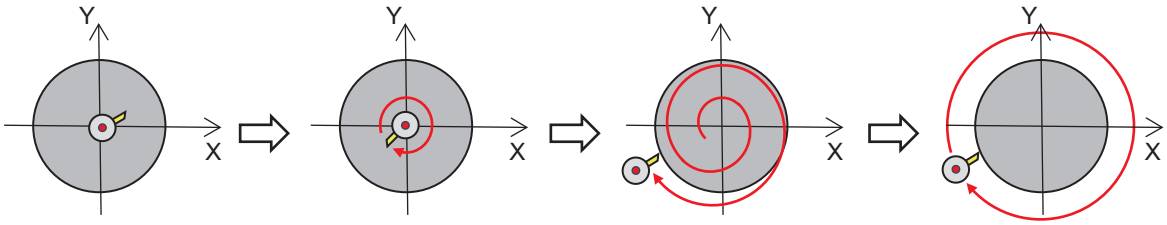


Orbit machining is started.

An axis movement command is issued to move in the positive (+) direction of the 1st circular motion axis.

Circular motion has started and is currently in progress.

[Positioned after the circular motion starts] (outside machining)



Orbit machining is started.

Circular motion starts.

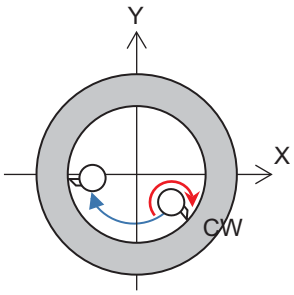
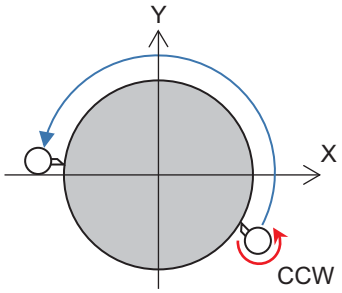
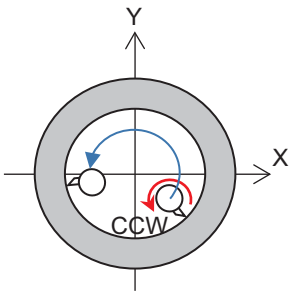
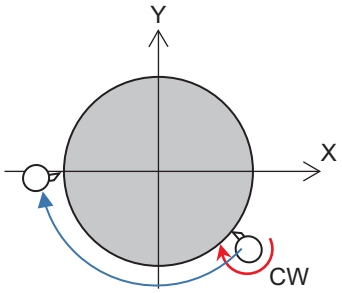
An axis movement command is issued to move in the positive (+) direction of the 1st circular motion axis.

Circular motion is currently in progress.

(4) Circular motion direction

Select the circular motion direction (spindle forward rotation/spindle reverse rotation) of the orbit spindle in consideration of direction (handedness) of the tool mounted on the orbit spindle and the address Q (internal/outside machining selection). Note that if a wrong circular motion direction is commanded, the tool may be damaged.

[Selection of the spindle rotation direction (when G18 is selected)]

Tool handedness	Q1 (Internal machining)	Q2 (Outside machining)
Right hand		
Left hand		

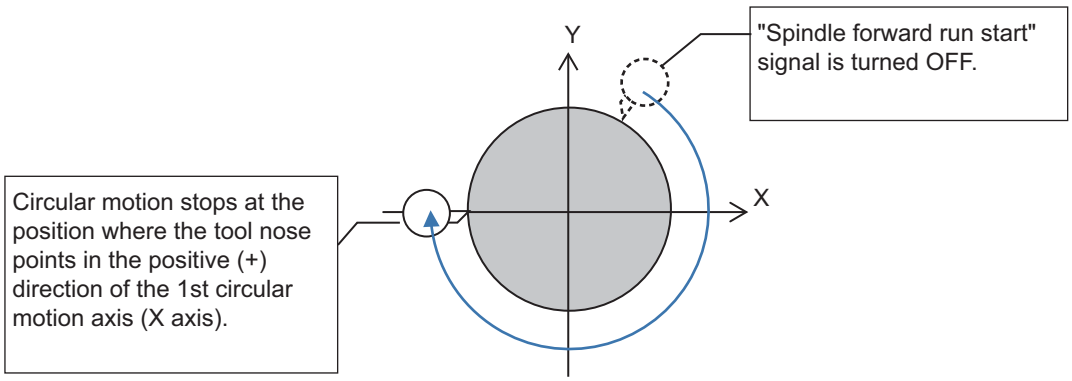
CW: Spindle forward rotation

CCW: Spindle reverse rotation

(5) Circular motion stop and stop position

To stop the circular motion, turn OFF both the "Spindle forward run start" signal and the "Spindle reverse run start" signal.
The stop position is where the tool nose points in the positive (+) direction of the 1st circular motion axis.

When the circular motion is stopped, it continues until the stop position is reached at the speed set in the parameter "#19448 Index speed_Orbit". When the parameter "#19448 Index speed_Orbit" is set to "0", the circular motion continues until the stop position is reached at the current circular motion speed.
When NC reset or emergency stop is performed, the circular motion immediately stops.
When the parameter "#8185 Contin circ motion" is "0" and feed hold stop or single block stop is performed, the circular motion immediately stops.



(6) Counter display

The display contents of the "Current posn" counter and the "Prog posn" counter (*1) are different during the orbit machining mode.

- For the 1st circular motion axis, the coordinate position where the circular motion center is "0" is displayed with the diameter value.
- The program position counter of the 2nd circular motion axis is always "0.000".

(*1) The "Prog posn" counter shows the machining program command position, and is not updated during the circular motion.

(7) Circular motion speed and cutting speed

Specify the circular motion speed with the S command for the orbit spindle. Circular motion is performed at the spindle speed set in the "Spindle command rotation speed output" signal. The cutting speed during the circular motion is calculated by the following formula.

Cutting speed = π x Circular motion diameter (diameter value) x Spindle rotation speed

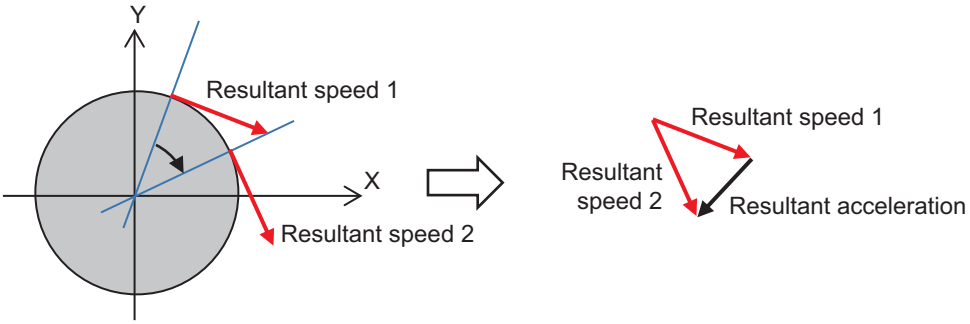
0.0 (m/min) is displayed in the spindle cutting speed (surface speed) on the Monitor screen.

Two types of speed clamp is applied during the orbit machining mode.

- Speed clamp based on tolerable acceleration rate
- Speed clamp based on rapid traverse/cutting feed clamp seed

[Speed clamp based on tolerable acceleration rate]

The circular motion speed is clamped so that the resultant acceleration rate (resultant speed change amount) of the circular motion axis does not exceed the tolerable acceleration rate during the circular motion.
The tolerable acceleration rate is calculated based on the circular motion axis parameters "#12166 orbit_F" (Orbit machining: Maximum tolerable acceleration (feed rate)) and "#12167 orbit_tL" (Orbit machining: Maximum tolerable acceleration (time constant)).
When the parameter "#12166 orbit_F" or "#12167 orbit_tL" is "0", the speed is not clamped based on tolerable acceleration rate.
These parameter settings depend on the MTB specifications.



[Speed clamp based on cutting feed clamp speed]

The circular motion speed is clamped so that the maximum feedrate of each axis does not exceed the value of the parameter "#12561 orbit_clamp" (Orbit machining: Cutting feedrate limit).
When the parameter "#12561 orbit_clamp" is "0", the speed is clamped based on the value of the parameter "#2002 clamp" or "#2110 Clamp(H-precision)".

These parameter settings depend on the MTB specifications.

(8) Adjusting the machining accuracy

Orbit machining is recommended to use with the high-accuracy control and OMR-FF. Inward deviation caused by servo response delay can be reduced. OMR-FF depends on the MTB specifications.

Operation when the orbit machining mode is canceled

The orbit machining mode is canceled with the "G08.6 Q0" command.
The "Prog posn" counters of the 1st and 2nd circular motion axes return to the positions on the workpiece coordinate system. The 1st circular motion axis display returns to the radius value.
When the orbit machining mode is canceled during the circular motion, the circular motion is stopped.
The "In orbit machining mode" signal turns OFF after the orbit spindle stops.



Program example

[Machining condition]

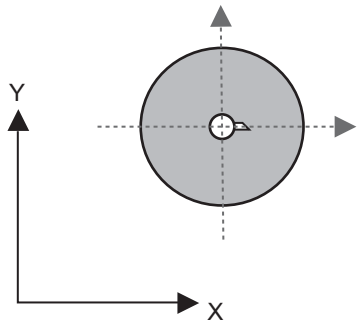
The machining conditions for the machining program example are as follows.

- Machine configuration with 1 part system, X, Y and Z axes and spindle/C axis
- The G18 plane is turned, and the outside of the workpiece is machined.
- A right hand tool is used. (Tool No.: 01)
- Tool compensation No. used for tool nose radius compensation: 11

[Machining program example]

N001 G28 Z0.;	
N002 G28 X0. Y0.;	
N003 G49 G40;	
N004 T01 M06;	The tool is changed to a turning tool.
N005 G43 Z100. H01;	The tool length is compensated in the tool axis direction.
N006 G90 G00 X100. Y150.;	The tool is positioned at the center position (workpiece center, etc.) of the circular motion. (*1)
N007 Z75.;	
N008 G00 C0.;	The orbit spindle is switched to the C axis mode, and is positioned at "C0.".
N009 G18;	The machining plane is selected. The plane commanded here is different from the circular motion plane. (*2)
N101 **G08.6 Q2 R0.;**	The orbit machining (outside machining) starts. (*3)
N102 G43 X100. H02;	Compensation in the tool radial direction is performed. Compensation is performed for the 1st circular motion axis with the tool length compensation command.
N103 #23011=3; N104 G42 D11;	Specify the hypothetical tool nose point with the system variable #23000+n ("n" is the compensation No.), and then command the tool nose radius compensation.
N105 G96 S60 P1;	The constant surface speed control is commanded. (*4)
N201 G00 X40.;	The tool moves to the machining start position by rapid traverse. (*5)
N202 M04;	The spindle reverse rotation is commanded.
N203 G95 G01 X50. Z65. F0.1;	The turning machining is commanded. (*6)
N204 G01 Z50.;	
N205 G02 X70. Z40. R15.;	
N206 G01 Z20.;	
N207 G01 X80.;	
N301 M05;	The spindle rotation stops, and the circular motion stops. (*7)
N302 G97;	
N303 G94 G40;	
N304 **G08.6 Q0;**	The orbit machining is ended. (*8)
N401 G00 Z100.;	The orbit spindle is retracted. (*6)
N402 X100.;	
N403 G49;	

(*1) During the orbit machining mode, the circular motion center is used as the program coordinate zero point.

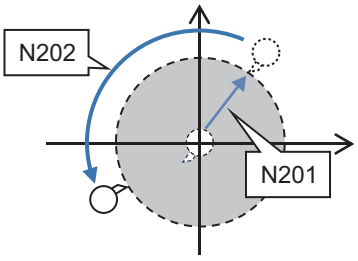


(*2) For details, refer to "Designation of circular motion plane, circular motion axis and feed direction axis" in "Detailed description".

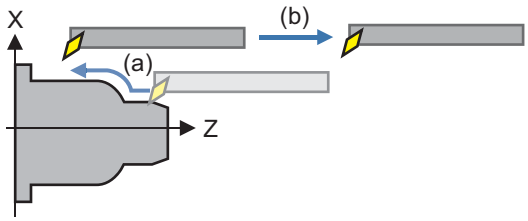
(*3) The "In orbit machining mode" signal turns ON. The program position changes to the coordinate where the circular motion center is used as the zero point.

(*4) Specify the 1st circular motion axis for the address P (constant surface speed control axis).
The spindle speed clamp command is not required during the orbit machining mode.

(*5) The axis can move quickly on the circular motion path by a movement command while the circular motion is stopped. It moves so that the tool nose faces the machining surface perpendicular to the circular motion center while the circular motion is stopped.



(*6) (a) Interpolation is performed on the selected plane (Z-X plane) by the G00/G01/G02/G03 commands.
The diameter value is used to command the 1st circular motion axis (X axis) during the orbit machining mode.
(b) The orbit spindle is retracted when the machining ends.



(*7) The orbit spindle decelerates to stop at the position where the tool nose points in the positive (+) direction of the 1st circular motion axis.

(*8) The "In orbit machining mode" signal turns OFF after the orbit spindle stops.

Relationship with other functions

Relationship between the orbit machining and G code functions

The table below shows the G codes available during the orbit machining mode.

If a G code not listed below is commanded during the orbit machining mode, the program error (P993) occurs.
If the orbit machining mode is commanded during operation of a G code not listed below, the program error (P994) occurs.

Column A: Operation when the combination function is commanded while the orbit machining mode is enabled
Column B: Operation when the orbit machining (G08.6) is commanded while the combination function is enabled
○: Combination is available.
X: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)
-: No combination
Δ: Others

Group	G code	Function name	A	B
0	G04	Dwell	○	-
	G05	High-speed machining mode I/II High-speed high-accuracy control II/III	Δ (*1)	Δ (*1)
	G05.1	High-speed high-accuracy control I Spline interpolation	Δ (*1)	Δ (*1)
	G08	High-accuracy control	○	○
	G52	Local coordinate system setting	× (P993)	○
	G92	Spindle clamp speed setting	○	○
	G162	Spindle speed fluctuation detection	○	○
	G163	Spindle speed fluctuation detection cancel	○	○
	G186	Interference check III interfering object data enable command	○	-
1	G00	Positioning	○	○
	G01	Linear interpolation	○	○
	G02	Circular interpolation CW	○	○
	G03	Circular interpolation CCW	○	○
2	G17	X-Y plane selection	× (P993)	○
	G18	Z-X plane selection	× (P993)	○
	G19	Y-Z plane selection	× (P993)	○
3	G90	Absolute command	○	○
	G91	Incremental command	○	○
4	G22	Stroke check before travel ON	Δ (*2)	Δ (*2)
	G23	Stroke check before travel cancel	○	○
5	G94	Feed per minute (asynchronous feed)	○	○
	G95	Feed per revolution (synchronous feed)	○	○
6	G20	Inch command	○	○
	G21	Metric command	○	○
7	G40	Tool radius compensation (tool nose radius compensation) cancel	○	○
	G41	Tool radius compensation (tool nose radius compensation) left	○	× (P994)
	G42	Tool radius compensation (tool nose radius compensation) right	○	× (P994)

Group	G code	Function name	A	B
8	G43	Tool length compensation (+)	Δ (*3)	○
	G44	Tool length compensation (-)	Δ (*3)	○
	G49	Tool length compensation cancel	Δ (*3)	○
13	G64	Cutting mode	○	○
	G61.1	High-accuracy control ON	○	○
17	G96	Constant surface speed control ON	○	× (P994)
	G97	Constant surface speed control OFF	○	○
28	G0.5	Rapid traverse block overlap	Δ (*4)	Δ (*4)

(*1) The high-speed machining mode is temporarily canceled during the orbit machining.

(*2) The stroke check before travel (G22) is enabled only for a movement command. The amount moved by the circular motion cannot be checked.

(*3) If the 2nd circular motion axis is commanded, the program error (P993) occurs.

(*4) The rapid traverse block overlap is temporarily canceled during the orbit machining.

Spindle override

The spindle override during the orbit machining is enabled for the circular motion speed.
When the spindle override is set to 0%, the circular motion immediately stops.

Rapid traverse override/cutting feed override

The rapid traverse override and cutting feed override are enabled during the orbit machining. Even when the rapid traverse override or cutting feed override is changed, the circular motion axis speed does not change.

Constant surface speed control

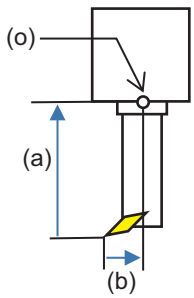
When the constant surface speed control command (G96) is issued during the orbit machining mode, the surface speed of the circular motion can be kept.
This enables cutting machining with a constant cutting speed even when the circular motion radius is changed by the movement command of the 1st circular motion axis.

To command the constant surface speed control, issue the constant surface speed control command (G96) after the orbit machining mode is started, and cancel the orbit machining mode after the constant surface speed control cancel command (G97) is issued.
Command the 1st circular motion axis for the address P (constant surface speed control axis) of the constant surface speed control command (G96).
The spindle speed clamp command (G92) is not required for the constant surface speed control during the orbit machining mode. Even when the constant surface speed control command (G96) is issued without the spindle speed clamp command (G92), the program error (P134) does not occur.
When the spindle speed clamp command (G92) is issued, the circular motion speed is clamped according to the command.

Tool length compensation

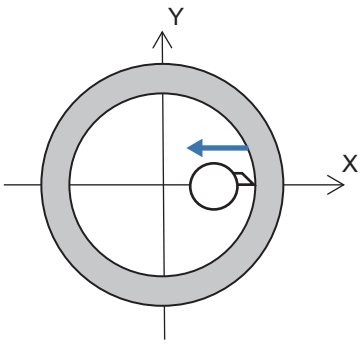
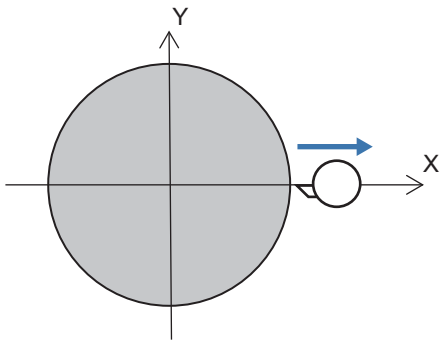
The tool length compensation is performed with the tool length compensation command (G43/G44) in two steps: in the tool axis direction and in the tool radial direction.

For the tool axis direction, perform the compensation before the "G08.6 Q1 (or Q2)" command.
For the tool radial direction, perform the compensation for the 1st circular motion axis after the "G08.6 Q1 (or Q2)" command.



(o) Reference position (basic point)
(a) Tool length compensation amount in the tool axis direction
(b) Tool length compensation amount in the tool radial direction

Change the compensation direction in the tool radial direction between internal machining and outside machining as shown below.

Internal machining G44 Tool length compensation (-)	Outside machining G43 Tool length compensation (+)
	

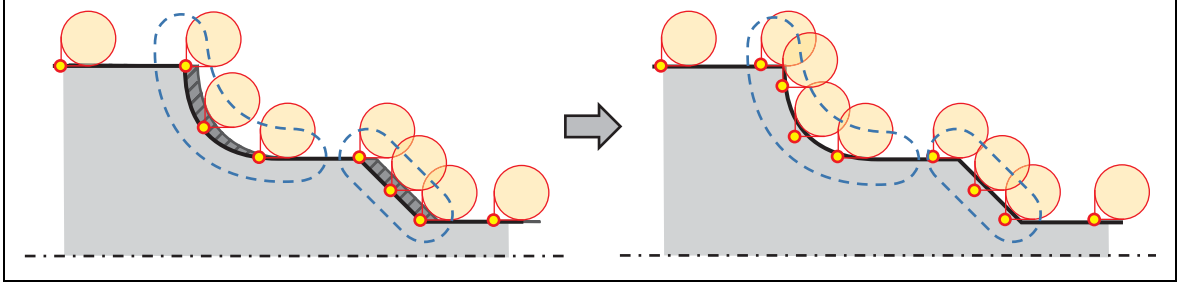
When the parameter "#1080 Dril_Z" (Specify boring axis) is "1", the tool length compensation for an axis other than Z axis cannot be performed. Therefore, set the parameter "#1080 Dril_Z" to "0" with the parameter input by program (G10 L70), etc., and then perform the tool length compensation.
Return the parameter "#1080 Dril_Z" to "1" after the orbit machining as necessary.
When the tool length compensation is commanded for two or more axes and the tool length compensation cancel command (G49) is issued, the tool length compensation for all the axes is canceled.

Tool nose radius compensation

When the tool compensation type is II, the tool nose radius compensation (G41/G42/G40) can be used during the orbit machining.
The turning tool to be used for the orbit machining has a rounded nose. When only the tool compensation amount in the vertical axis or horizontal axis direction entered for tool measurement is applied, the machining path is generated as if machining is performed with the square-cornered (unrounded) and sharp tool nose (hypothetical tool nose point). Uncut parts are left on the tapered or arc-shape part because the hypothetical tool nose is slightly different from the actual tool nose shape.
When the tool nose radius compensation is applied, uncut parts left on a tapered or an arc shape part can be reduced.

The figure below shows the program path and the tool nose path.

When the tool nose radius compensation is not performed	When the tool nose radius compensation is performed



For tool nose radius compensation, refer to "12.4 Tool Nose Radius Compensation (for Machining Center System)".

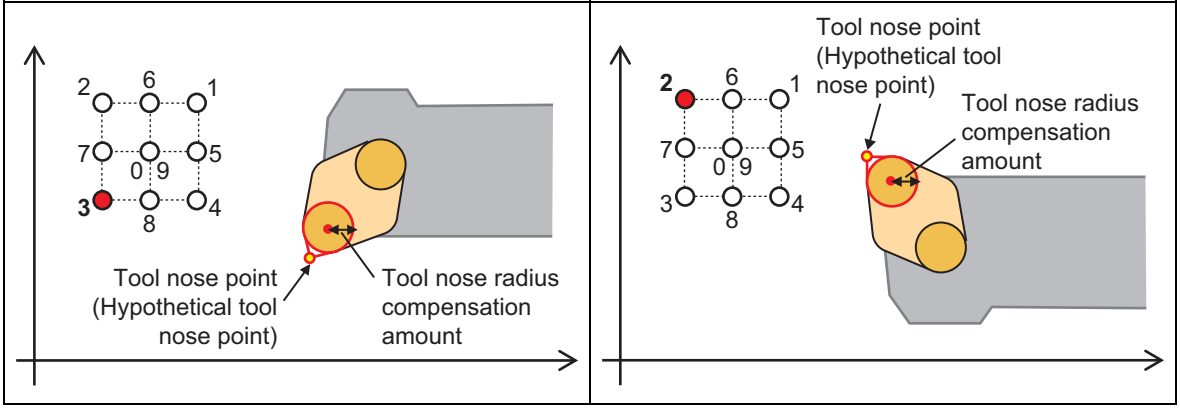
(1) Tool nose radius compensation amount

Set the tool nose radius compensation amount for the orbit machining to the "Radius" item of the Tool Offset screen.

(2) Hypothetical tool nose point

The tool nose is generally rounded. Set the hypothetical tool nose point to indicate the tool nose position on the program. Set the hypothetical tool nose point position in the tool advance direction (machining direction of feed direction axis).

[Outside machining] (Hypothetical tool nose point: 3)	[Internal machining] (Hypothetical tool nose point: 2)



The hypothetical tool nose point is set with the system variable "#23000+n". "n" must be the same value as the compensation No. of the tool nose radius compensation amount.
The hypothetical tool nose point cannot be set with the compensation data input by program (G10) or on the PLC window.

(Example) Internal machining where the tool compensation No. of the tool nose radius compensation is "11" and the hypothetical tool nose point is set to "2"

#23011 = 2;	Set "2" for the hypothetical tool nose point for the tool compensation No. "11".
G41 D11;	Start the tool nose radius compensation using the tool compensation No. "11".

<Note>
- Perform tool nose radius compensation start (G41/G42) and cancel (G40) during the orbit machining mode.
- Tool radius compensation (G41/G42/G40) cannot be commanded during the orbit machining. If it is commanded, the program error (P993) occurs. Whether tool nose radius compensation or tool radius compensation is enabled depends on the following conditions.

Tool nose radius compensation	The tool compensation type is II, and the hypothetical tool nose point is selected with the system variable "#23000+n" (the value is "1" to "9").
Tool radius compensation	The tool compensation type is not II, or the hypothetical tool nose point is not selected with the system variable "#23000+n" (the system variable value is "0").

- The tool compensation operation after the orbit machining cancel varies depending on whether the optional specification of tool nose radius compensation for Machining Center System is enabled or disabled.
  When the optional specification is enabled, the operation follows the specification of tool nose radius compensation for Machining Center System.
  When the optional specification is disabled, the G41/G42 command operates as tool radius compensation regardless of the setting of the system variable "#23000+n".
- When NC reset or emergency stop is performed during the orbit machining, the tool nose radius compensation modal is canceled.

Interlock

When one of orbit machining axes is interlocked, all other orbit machining axes stop due to the interlock.

Machine lock

To stop the circular motion by the "Automatic machine lock" signal, turn ON the "Automatic machine lock" signal for all of the orbit machining axes.

Single block/feed hold/dwell

Whether to stop or continue the circular motion of the orbit machining during single block stop, during feed hold or during dwell can be selected with the parameter "#8185 Contin circ motion".

	Parameter "#8185 Contin circ motion"	
	0	**1**
During single block stop	Stop	Continue
During feed hold	Stop	Continue
During dwell	Stop	Continue

NC reset, Emergency stop

When the NC reset or emergency stop is performed during the circular motion, the orbit spindle immediately decelerates to stop.
The "In orbit machining mode" signal turns OFF after the orbit spindle stops.

MDI interruption, PLC interruption, Macro interruption

MDI interruption, PLC interruption or macro interruption cannot be performed during the orbit machining mode.
If the MDI interruption or PLC interruption is performed during the orbit machining mode, the operation error (M01 1609) occurs.
If the macro interruption is enabled during the orbit machining mode, the program error (P993) occurs.
If the orbit machining (G08.6) is commanded while the macro interruption is enabled, the program error (P994) occurs.

Manual arbitrary reverse run, Arbitrary reverse run, Manual speed command

Forward run is available during the orbit machining mode. However, reverse run is not available. Reverse run is prohibited.

Mirror image by parameter setting, Mirror image by external input

If the orbit machining (G08.6) is commanded during the mirror image by parameter setting or mirror image by external input, the program error (P994) occurs.
If the mirror image by parameter setting or mirror image by external input is turned ON during the orbit machining, the operation error (M01 1609) occurs.

Coordinate rotation by parameter

If the orbit machining (G08.6) is commanded during the coordinate rotation by parameter, the program error (P994) occurs.

Dry run

The dry run during the orbit machining mode is enabled for the feedrates of the 1st circular motion axis and the feed direction axis. It does not affect the circular motion speed.

Manual interruption

Manual interruption cannot be performed for orbit spindle, 1st and 2nd circular motion axes during the orbit machining. If it is performed, the operation error (M01 1609) occurs.

Manual reference position return

Manual reference position return cannot be performed for orbit machining axes during the orbit machining mode. If it is performed, the operation error (M01 1609) occurs.

Feed per revolution (synchronous feed)

The feedrate according to the circular motion speed is used for the orbit machining during feed per revolution (G95).

Arbitrary axis exchange control

Arbitrary axis exchange control cannot be commanded during the orbit machining mode. If it is commanded, the program error (P993) occurs.
If arbitrary axis exchange control is commanded for an axis in the part system during the orbit machining mode, the operation error (M01 1101) occurs.
The orbit machining can be commanded while the axis is exchanged with the arbitrary axis exchange control.

Program restart

If the restart search from the block of the orbit machining command (G08.6) is attempted, the program error (P49) occurs.

Graphic check

Only the position commanded by the machining program is drawn.
The circular motion is not drawn.

Graphic trace

The machine coordinate position is used to draw. The circular motion is also drawn.

3D machine interference check, Real-time 3D machine interference check

If the orbit machining (G08.6) is commanded during 3D machine interference check or real-time 3D machine interference check, the program error (P994) occurs. Even when the [Colli check] menu is pressed on the 3D monitor screen during the orbit machining and single block stop, the interference check is not enabled.

External deceleration

The external deceleration during the orbit machining is enabled for feedrates of the 1st circular motion axis and the feed direction axis commanded in the turning machining program. The circular motion speeds of the 1st and 2nd circular motion axes do not decelerate.

Smart safety observation

The safety speed cannot be clamped during the orbit machining mode. The following operation is performed.

(1) If the "SLS observation request (control axis)" signal is turned ON during the orbit machining command, the system alarm (Z25) occurs, the safety speed is not clamped, and the machining continues.

(2) If the orbit machining is commanded from the operating program while the safety speed is clamped, the block stops and the stop code (T03 0308) occurs.

(3) If the cycle start is performed from the orbit machining block during the safety speed clamp, the stop code (T01 165) occurs and the cycle start cannot be performed.



Precautions

(1) Do not command the tool change during the circular motion. Doing so may cause tool interference.

(2) Set the same value for the position loop gains of circular motion axis and orbit spindle. If a different value is set, synchronization of the circular motion may be collapsed.

(3) For the orbit spindle rotation direction, refer to "(4) Circular motion direction" in "Detailed description". If a wrong rotation direction is commanded, the tool may be damaged.

(4) Interference check by the 3D machine interference check is not performed during the orbit machining. Prevent machine interference when an axis other than the orbit machining axes is commanded during the circular motion, etc.

(5) To perform the manual interruption during the orbit machining mode, keep the "Manual absolute" signal ON. If the "Manual absolute" signal is OFF, the tool, etc. may be damaged in the circular motion because the coordinate system is shifted.

(6) When the parameter "#1080 Dril_Z" (Specify boring axis) is "1", the tool length compensation for an axis other than Z axis cannot be performed. Therefore, set the parameter "#1080 Dril_Z" to "0" with the parameter input by program (G10 L70), etc., and then perform the tool length compensation.
Return the parameter "#1080 Dril_Z" to "1" after the orbit machining as necessary.

(7) To use the tool nose radius compensation, command the tool nose radius compensation start/cancel during the orbit machining mode. The tool nose radius compensation cannot be commanded between orbit machining commands.

(8) Even when the S command is issued for the orbit spindle during the orbit machining, the mode does not change to the spindle mode. Perform the spindle command in the C axis mode during the orbit machining.

(9) The rapid traverse acceleration/deceleration time constant is used to perform the acceleration/deceleration operation regardless of the G code group 01 modal in the cutting mode (G64) during the orbit machining.

16

Multi-part System Control

16.1 Timing Synchronization Operation

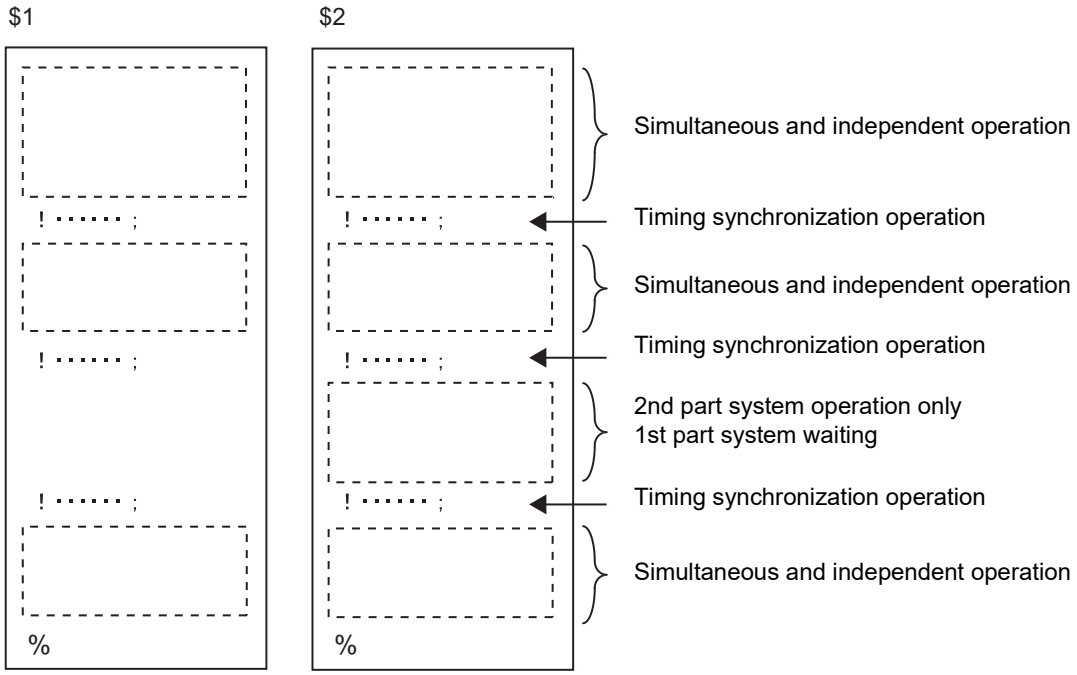
⚠ CAUTION

⚠ When programming a multi-part system, carefully observe the movements caused by other part systems' programs.

16.1.1 Timing Synchronization Operation (! code) !n (!m ...) L

Function and purpose

Multiple machining programs can be operated independently at same time for multi-axis and multi-part system mixed control CNC. This 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.
This function is enabled in two or more part systems.



Command format

!n (!m ...) L_ ;

!n, !m, ...	Timing synchronization operation (!) and part system No. (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

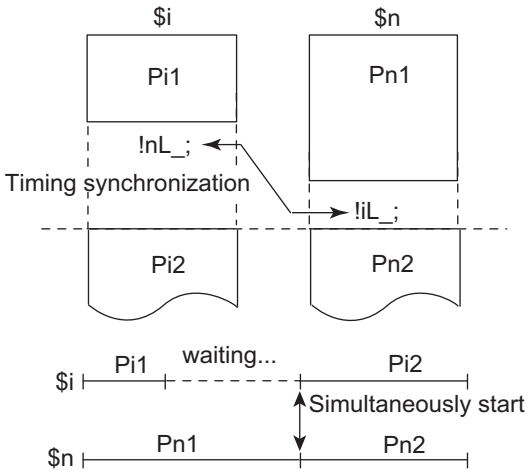
Timing synchronization is also possible among three or more part systems.

Detailed description

Timing synchronization between part systems during automatic operation

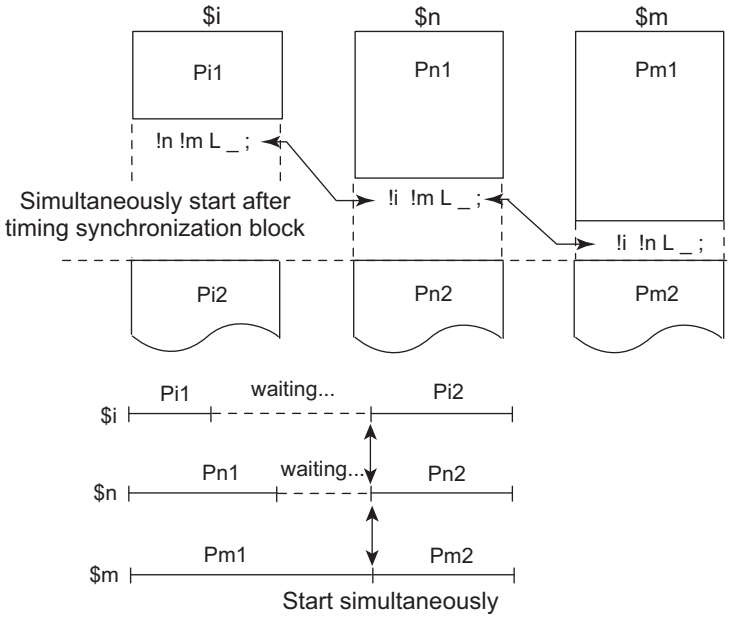
If "!n L_" is commanded from a part system "i", operation of the part system "i" program will wait until "!i L_" is commanded from the part system "n" program.
When "!i L_" is commanded, the programs for the two part systems will start simultaneously.

(1) Timing synchronization between two part systems



(2) Timing synchronization between three part systems and more

Similarly with three part systems or more, when all part systems that are included in the timing synchronization operation reach timing synchronization block, these part systems start operating from the next block simultaneously.



Ignore timing synchronization

The timing synchronization operation of a specific part system is ignored depending on the MTB specifications.
The behavior depends on the MTB specifications. (Combination of the "Waiting ignore" signal (YCD0) and the parameter "#1279 ext15/bit0")
Refer to "16.1.5 Timing Synchronization When Timing Synchronization Ignore Is Set" for details.

Specify part system for timing synchronization

(1) When the part system No. is omitted in the timing synchronization operation, it follows the setting of the parameter "#19419 Timing sync system".

(Example) When #19419 is "3", "!;" is equivalent to "!3;".

(2) If a nonexistent part system No. is specified or a part system No. with a decimal point is specified in the timing synchronization operation, the program error (P35) occurs.

(Example) If "!2.;" is specified, the program error (P35) occurs.

(3) 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.

(Example) To command from the part system "i", a command like "!i!n!mL1;" is possible.

(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".

Command in the same block as timing synchronization operation

(1) The timing synchronization operation command is normally issued alone in a 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 (parameter "#1093 Wmvfin").
When "#1093 Wmvfin" is "0", the timing synchronization operation is performed before executing movement command.
When "#1093 Wmvfin" is "1", the timing synchronization operation is performed after executing movement command.

(2) 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.



Precautions

(1) When the M code can be used, both the M code and ! code can be used.
Refer to "16.1.4 Timing Synchronization Operation Function Using M codes ; M***" for details.

(2) While the timing synchronization operation M code is valid, if one part system is standing by with an M code, the operation error (M01 1030) occurs 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, the operation error (M01 1030) occurs if there is an M code timing synchronization operation command in the other part system.

(4) "SYN" appears in the operation status section during timing synchronization operation. Also, the "Timing synchronization between part systems" signal (XC34) is output to PLC.

(5) The timing synchronization operation ("!" code, M code) in the machining program can be ignored with the "Waiting ignore" signal (YCD0). (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.
Refer to "16.1.5 Timing Synchronization When Timing Synchronization Ignore Is Set" for details.

(6) When the timing synchronization operation between part systems and single block operation are used simultaneously, the next block stands by until the cycle start signal is input in the part system in which the single block mode is ON. Therefore, the operation may stop at the block without the timing synchronization operation code of the part system in which the single block is OFF, where it is not supposed to stop initially.

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

```
!n 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)

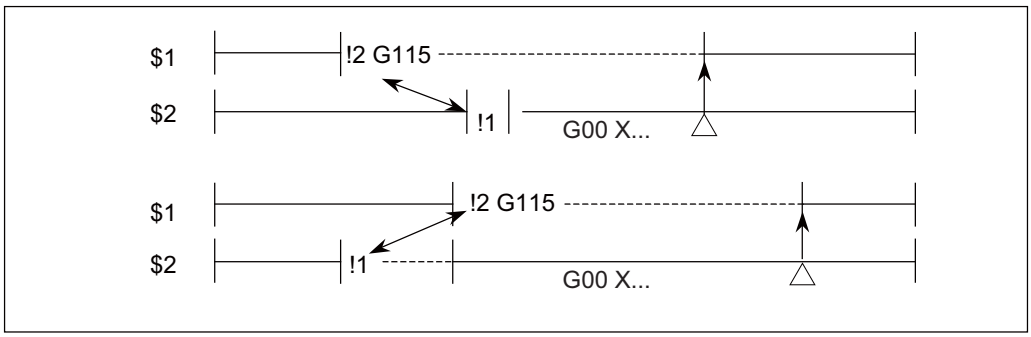
Detailed description

(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.

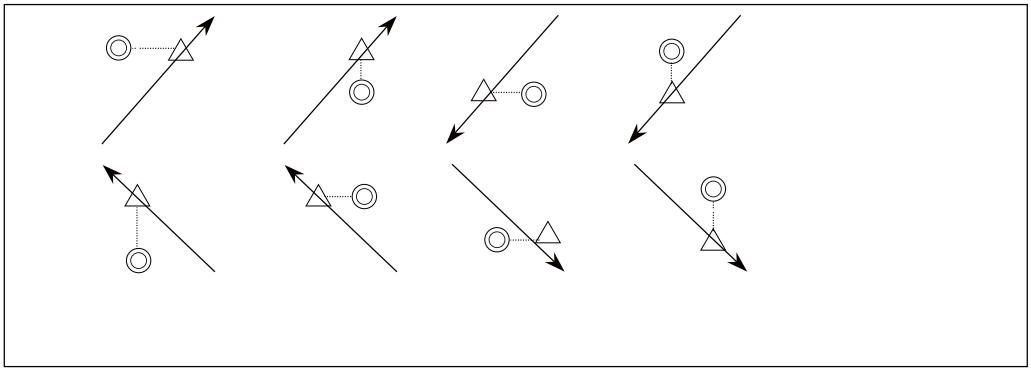
(3)The other part system starts first when timing synchronization operation is executed.

(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.



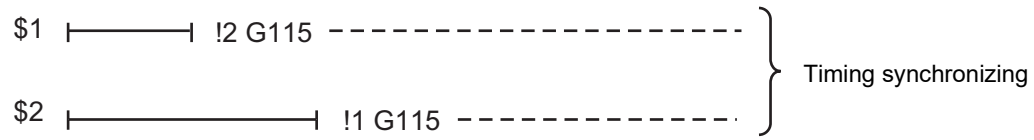
Movement ◎ Designated start point △ Actual start point

(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)

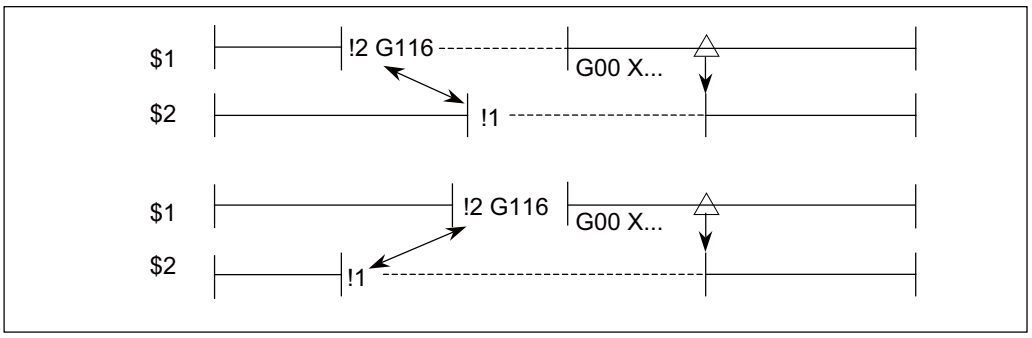
Detailed description

(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.

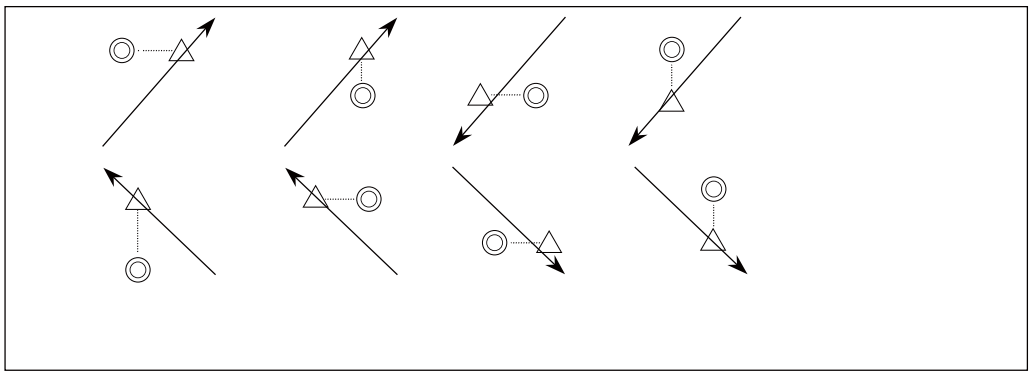
(3)The own part system starts first when timing synchronization operation is executed.

(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.



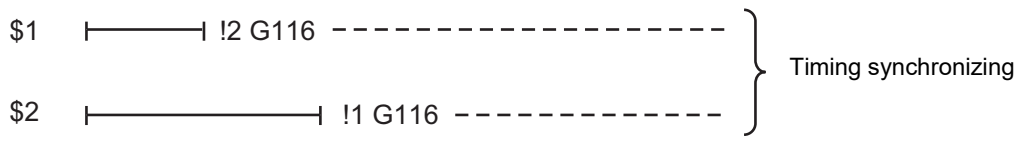
Movement ◎ Designated start point △ Actual start point

(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

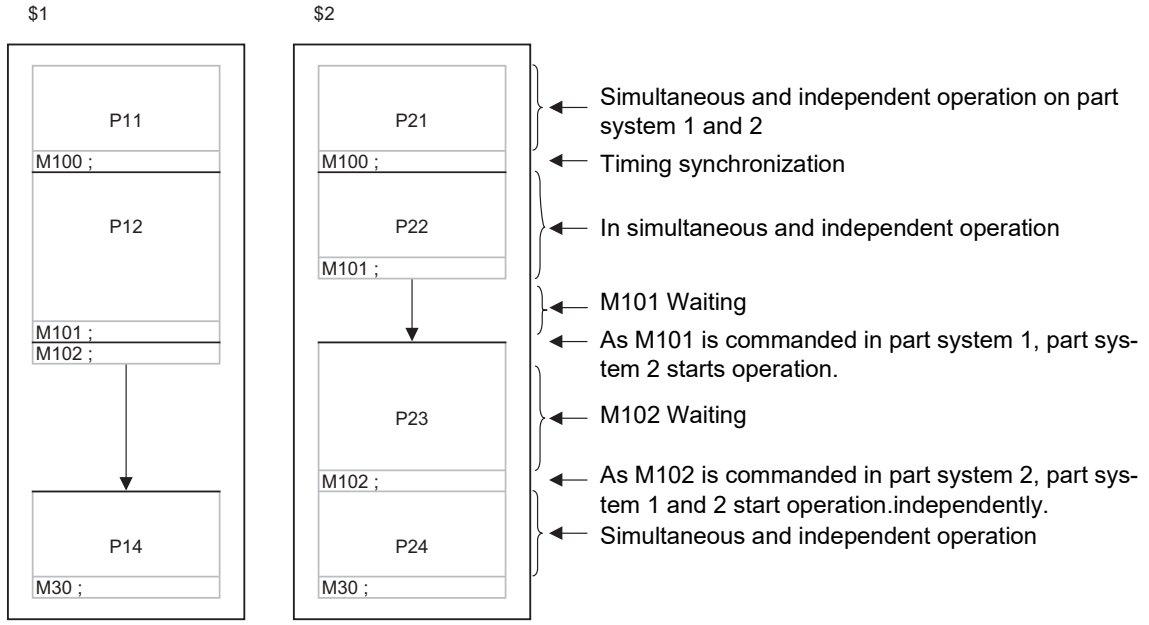
M* ;**

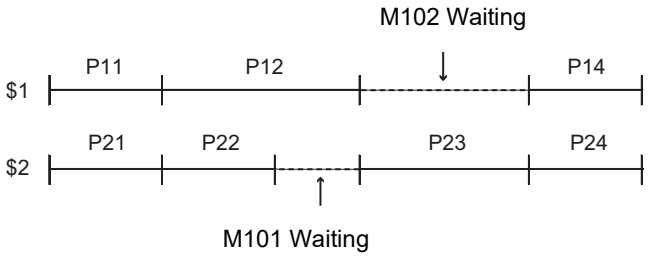
***	Timing synchronization operation M code

M code used for timing synchronization depends on the MTB specifications (parameter "#1310 WtMmin)", "#1311 WtMmax").

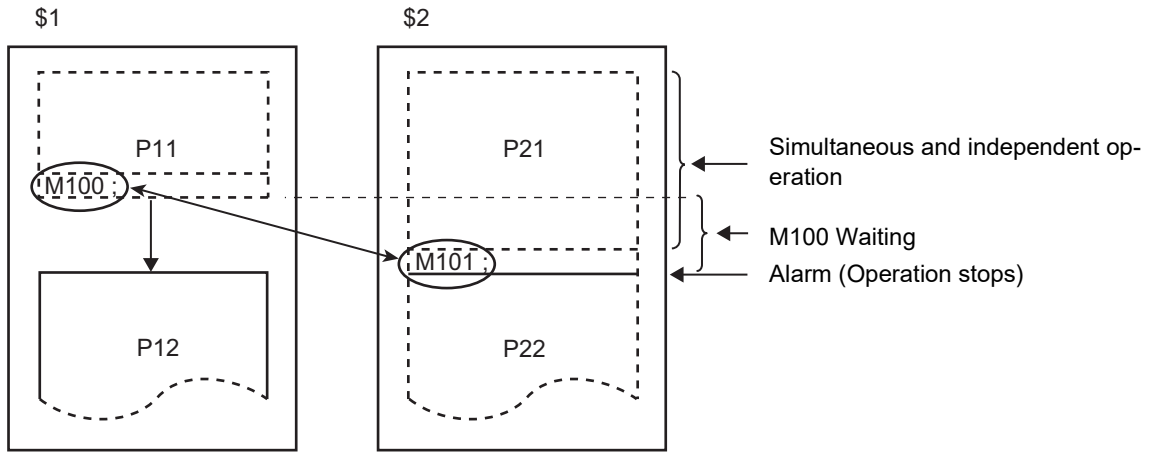
Detailed description

(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, the operation error (M01 1030) occurs 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.

(a) M code range designation parameter (M code minimum value <= M code <= M code maximum value)

#	Item		Details	Setting range
1310	WtMmin	Timing synchronization M code ABS. MIN.	The minimum value of the M code. If the setting value is "0", the timing synchronization operation M code will be ignored.	0, 100 ~ 99999999
1311	WtMmax	Timing synchronization M code ABS. MAX.	The maximum value of the M code. If the setting value is "0", the timing synchronization operation M code will be ignored.	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 ! 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	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.	0 / 1

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 ("!" 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	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.	0 / 1



Relation with other functions

Refer to "16.1.1 Timing Synchronization Operation (! code) !n (!m ...) L".



Precautions

For precautions for time synchronization, also refer to "16.1.1 Timing Synchronization Operation (! code) !n (!m ...) L".

(1) During timing synchronization operation with the M code, always command the M code alone in a block.

(2) While standing by after commanding the timing synchronization operation M code in one part system, the operation error (M01 1030) occurs if a different M code is commanded in the other part system. Operation will stop in both part systems.

(3) Unlike other M codes, the timing synchronization operation M code does not output code signals and strobe signals.

(4) When the M code can be used, both the M code and ! code can be used.

(5) While the timing synchronization operation M code is valid, if one part system is standing by with an M code, the operation error (M01 1030) occurs if there is a ! code timing synchronization operation command in the other part system.

(6) While the timing synchronization operation M code is valid, if one part system is standing by with a ! code, the operation error (M01 1030) occurs if there is an M code timing synchronization operation command in the other part system.

(7) The G115 and G116 commands cannot be used when waiting with the M code.

(8) If there is a timing synchronization operation with M code after the 3rd part system, the program error (P511) occurs.

(9) 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.

16.1.5 Timing Synchronization When Timing Synchronization Ignore Is Set

Function and purpose

While the "Waiting ignore" signal (YCD0) is ON, timing synchronization operation of that part system is ignored. (This depends on the MTB specifications.)
With a 2-part system, if the "Waiting 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

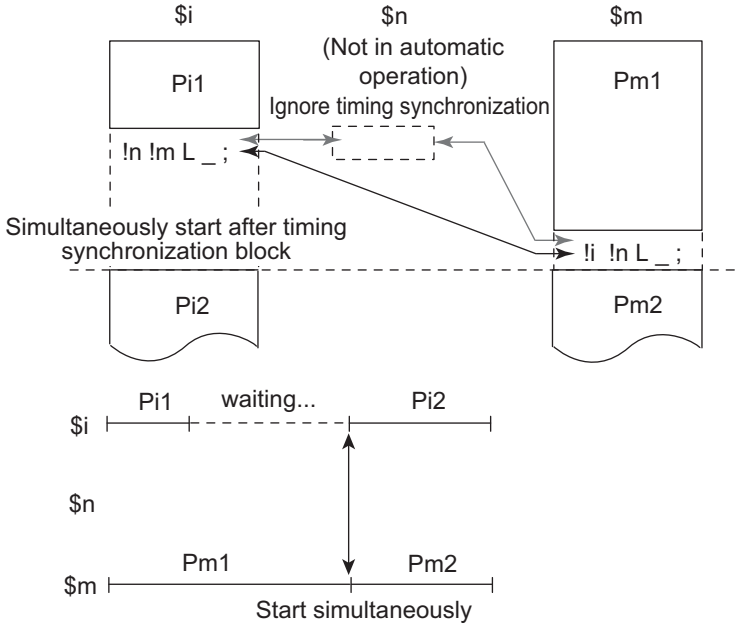
Note

(1) For sub part system control function, refer to "16.4 Sub Part System Control".

		"Waiting 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)

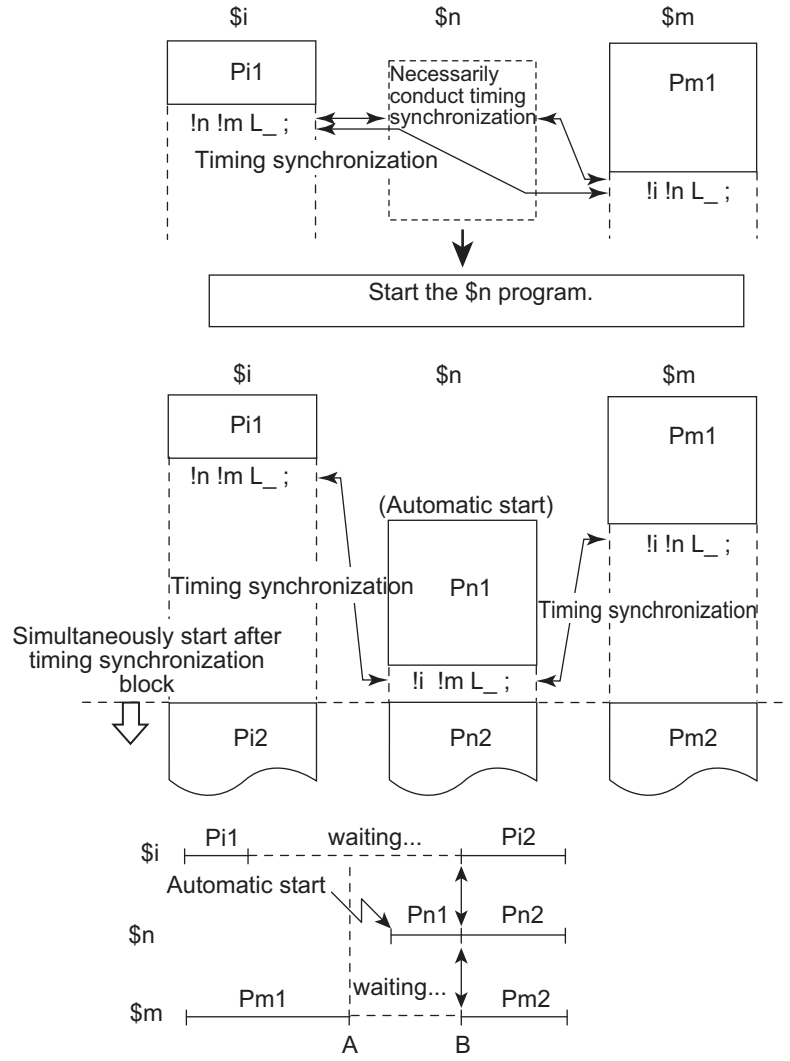
The following operation diagram gives an example of ! code.

(1) A case that "Ignores the timing synchronization with a part system not in automatic operation"



$n (part system "n") is not in automatic operation. Even when the "!n" is commanded in a machining program of another part system ($i or $m), the timing synchronization with the part system "n" is ignored.

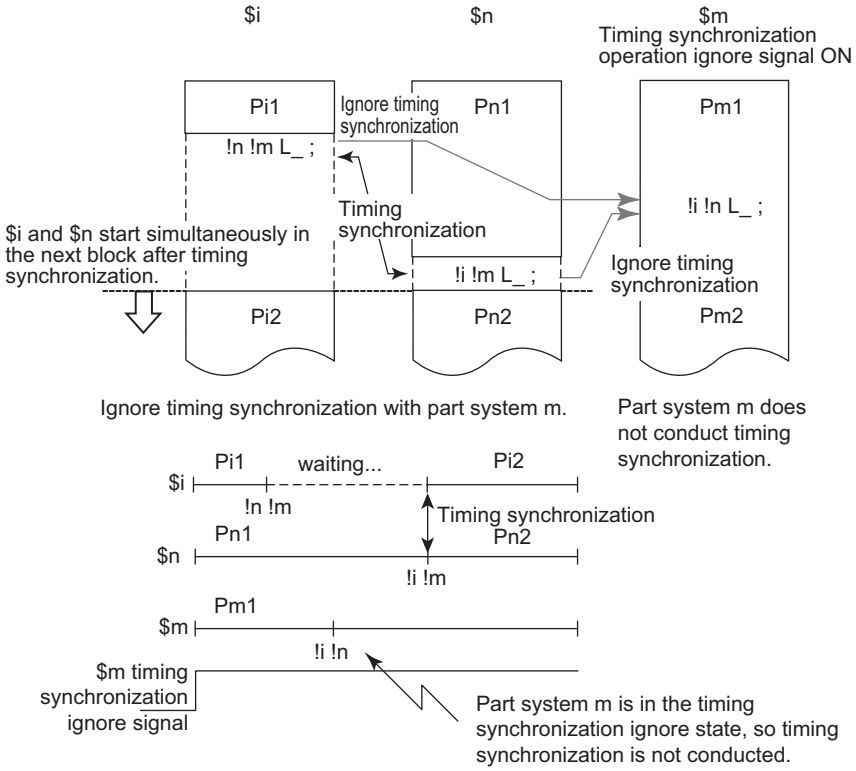
(2) A case that "Does not ignore the timing synchronization regardless of whether or not in automatic operation"



A: When timing synchronization operation between part systems (parameter "#1279 ext15/bit0" is set to "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" for details of the arbitrary axis exchange.

Note

(1) When performing arbitrary axis exchange with M80V TypeA and M80VW, set the parameter "#1431 Ax_Chg" to "1". If the arbitrary axis exchange command is issued while the parameter is "0", a program error (P39) occurs.

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" is set to "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 $1** X	Y	Z	**Control axes $2** X	Y	C	**Uncontrol axes**
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; :		X1	Y2	-	X2	-	C2	Y1,Z1
	G140 Y=Z1; (e) G00 Y10.; G01 Y8. F0.05; :				-	Z1	-	Y1,X2,C2
G140 X=X1 Y=Y1 Z=Z1; (c) G00 X20. Y15.; G01 X15. F5; :		X1	Y1	Z1	-	-	-	X2,Y2,C2
	G140 X=X2 Y=Y2 C=C2; (f) G00 X0; :				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" is set to "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 $1 X	$1 Y	$1 Z	$2 X	$2 Y	$2 C	Uncontrol axes
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; :		X1	Y2	Z1	X2	-	C2	Y1
	G140 Y=Y1; (e) G00 Y10.; G01 Y8. F0.05; :				X2	Y1	C2	-
G140 Y=Y1 ; (c) G00 X20. Y15.; G01 X15. F5; :		X1	Y1	Z1	X2	-	C2	Y2
	G140 X=X2 Y=Y2 C=C2; (f) G00 X0; :				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).

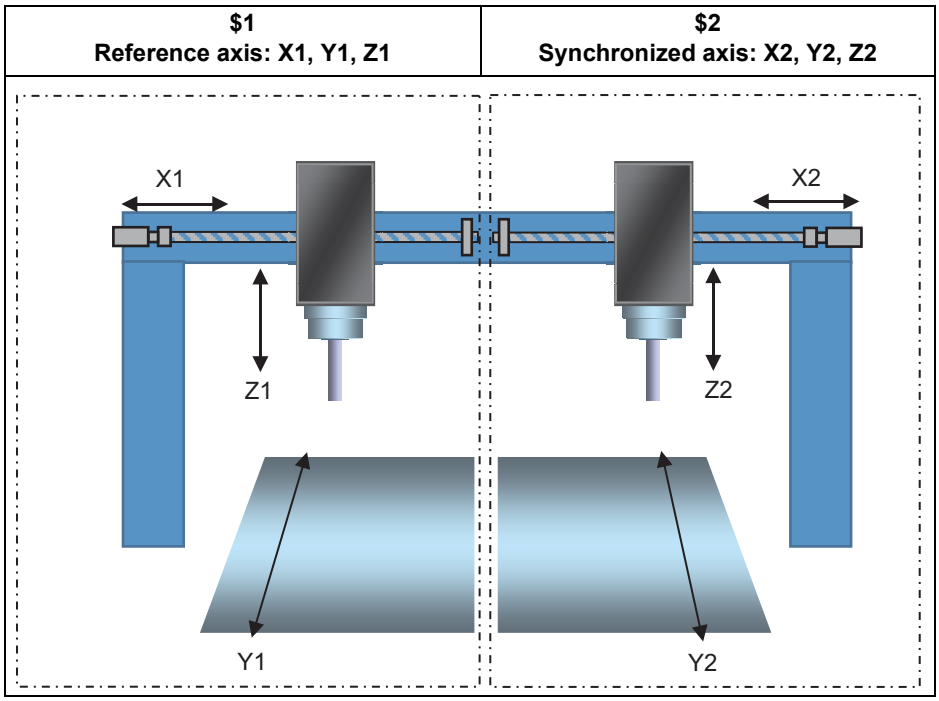
When the conditions for unavailable state of axis exchange fall through, no axis exchange mode will be cancelled. It will be cancelled when a "reset" signal or emergency stop is entered.

16.3 Control Axis Synchronization between Part Systems ; G125

Function and purpose

This function enables an arbitrary control axis in the other part system to move in synchronization with the movement command assigned to an arbitrary control axis.
The direction in which the axis is to move synchronously can also be reversed.

The following example indicates the machining to synchronize X2 axis to X1 axis, Y2 axis to Y1 axis, and Z2 axis to Z1 axis in the 1st part system (X1, Y1, Z1) and 2nd part system (X2, Y2, Z2).



While the synchronization, synchronized axis synchronizes and moves in accordance with the reference axis regardless of conducting automatic operation or manual operation. The movement command cannot be issued to the synchronized axis.

There are two types of control axis synchronization between part systems depending on the command method: Control axis synchronization between part systems I (by G command) and control axis synchronization between part systems II (by PLC signal). The method to be selected depends on the MTB specifications (parameter "#1280 ext16/bit5").
This manual describes the control axis synchronization between part systems I (by G command).

Term

The following describes the meanings of the terms used in this manual.

Word	Meaning
Reference axis	Reference axis in control axis synchronization between part systems (This axis moves only by its own axis command.)
Synchronization axis	Axis that moves synchronously with the reference axis in control axis synchronization between part systems (This axis moves with a command from the reference axis.)
Axes related to control axis synchronization between part systems	Reference axis and synchronized axis for control axis synchronization between part systems

Enabling conditions of the function

(1) This function can be used in multi-part systems of two or more part systems.

Command format

Synchronization start

G125 Synchronized axis name = Reference axis name ;

Synchronized axis name	Synchronized axis name (Axis name set in the parameter "#1022 axname2")
Reference axis name	Reference axis name (Axis name set in the parameter "#1022 axname2)

Synchronization end

G125 Synchronized axis name ;

Synchronized axis name	Synchronized axis name (Axis name set in the parameter "#1022 axname2")



Detailed description

This example describes a case to synchronize the Z2 axis with the Z1 axis.

Synchronization start	(1) Waits until all the axes stop in the part systems including the Z1 axis and the part systems including the Z2 axis. (2) Starts the 2-axis synchronization control using the Z1 axis as the reference axis and the Z2 axis as the synchronized axis.
Operation during synchronization	As the reference axis moves, the workpiece coordinate and machine coordinate of the synchronized axis move.
At execution of synchronization end	(1) Waits until all the axes stop in the part systems including the Z1 axis and the part systems including the Z2 axis. (2) Ends the synchronization control.

Synchronous direction

The synchronous direction is determined by the command and parameter.

[Designation with command]

In synchronization start command, if a code "-" is added to the beginning of the reference axis name, the synchronous direction will be reversed.

[Designation with parameter]

Follow the setting of the parameter "#2087 syncnt". (The setting depends on the MTB specifications.)

0: Moves in the same direction as the reference axis.

1: Moves in the opposite direction from the reference axis.

#2087 syncnt	"-" sign	
	Yes	None
0	Moves in the opposite direction from the reference axis.	Moves in the same direction as the reference axis.
1	Moves in the same direction as the reference axis.	Moves in the opposite direction from the reference axis.

Rapid traverse rate and cutting feedrate clamp speed of the reference axis and synchronized axis

The synchronized axis moves with the speed commanded of the reference axis; therefore, the same parameter setting value must be designated for the rapid traverse rate and cutting feedrate clamp speed of the reference axis and synchronized axis.

Time constant of the reference axis and synchronized axis

To match the acceleration/deceleration between the reference axis and the synchronized axis, the same time constant parameter setting value must be designated for the reference axis and the synchronized axis.

Synchronization deviation detection

The synchronization deviation is detected during the control axis synchronization between part systems.

The deviation of the synchronized axis feedback value against the reference axis feedback value is calculated as the synchronization deviation amount.

The synchronization deviation amount is output to R register (R5076 to R5107) which correspond to each synchronized axis.

If the synchronization deviation amount exceeds the value of parameter "#2024 synerr", the operation error (M01 0051) occurs. Cancel the control axis synchronization between part systems of the error axis to clear the error. However, when the parameter "#2024 synerr" is set to "0", the range of the synchronization deviation amount will not be checked.



Program example

When commanding from the part system containing the reference axis

$1			$2	
Program	**Operation**		**Program**	**Operation**
:			:	
N10 !L1;	(a)	---	N20 !L1;	(a)
N11 G125 Z2=Z1;	Starts synchronizing the Z2 axis with the Z1 axis.		:	
N12 !L2;	(a)	---	N21 !L2;	(a)
N13 G01 Z50.F100; N14 X4. Z4.; :	Synchronizes between the Z1 axis (reference axis) and Z2 axis (synchronized axis).		: : :	No movement command is issued to the Z2 axis (synchronized axis).
N15 !L3;	(a)	---	N24 !L3;	(a)
N16 G125 Z2;	Cancels the synchronization between the Z1 axis and Z2 axis.		:	
N17 !L4;	(a)	---	N25 !L4;	(a)
:			:	

(a) Timing synchronization operation

When commanding from the part system containing the synchronized axis

$1			$2	
Program	**Operation**		**Program**	**Operation**
:			:	
N10 !L1;	(a)	---	N20 !L1;	(a)
:			N21 G125 Z2=Z1;	Starts synchronizing the Z2 axis with the Z1 axis.
N12 !L2;	(a)	---	N21 !L2;	(a)
N13 G01 Z50.F100; N14 X4. Z4.; :	Synchronizes between the Z1 axis (reference axis) and Z2 axis (synchronized axis).		: : :	No movement command is issued to the Z2 axis (synchronized axis).
N15 !L3;	(a)	---	N24 !L3;	(a)
:			N25 G125 Z2;	Cancels the synchronization between the Z1 axis and Z2 axis.
N17 !L4;	(a)	---	N26 !L4;	(a)
:			:	

(a) Timing synchronization operation

When commanding from a part system containing neither the synchronized axis nor reference axis

$1			$2			$3	
Program	**Operation**		**Program**	**Operation**		**Program**	**Operation**
:			:			:	
N10 !2!3L1;	(a)	---	N20 !1!3L1;	(a)	---	N30 !1!2L1;	(a)
:			:			N31 G125 Z2=Z1;	Starts synchronizing the Z2 axis with the Z1 axis.
N11 !2!3L2;	(a)	---	N21 !1!3L2;	(a)	---	N32 !1!2L2;	(a)
N12 G01 Z50 F100; N13 X4.Z4.; :	Synchronizes between the Z1 axis (reference axis) and Z2 axis (synchronized axis).		: : :	No movement command is issued to the Z2 axis (synchronized axis).			
N14 !2!3L3;	(a)	---	N24 !1!3L3;	(a)	---	N35 !1!2L3;	(a)
:			:			N36 G125 Z2;	Cancels the synchronization between the Z1 axis and Z2 axis.
N15 !2!3L4;	(a)	---	N25 !1!3L4;	(a)	---	N37 !1!2L4;	(a)
:			:			:	

(a) Timing synchronization operation

Relationship with other functions

Functions that are not available during control axis synchronization between part systems

(1) The operation error (M01 1038) occurs in the following cases:

(a) The following commands are issued to the synchronized axis.
These commands can be issued to the reference axis.

- Mirror image
- Linear interpolation, circular interpolation (Rapid traverse rate, Cutting feedrate)
- Unidirectional positioning
- Helical interpolation
- Spiral/Conical interpolation
- Cylindrical interpolation
- Polar coordinate interpolation
- Exponential interpolation
- Involute interpolation
- Spline interpolation
- NURBS interpolation
- 3-dimensional circular interpolation
- Spline interpolation 2
- Thread cutting
- Synchronous tap
- Corner chamfering/Corner R
  (The same processing is also performed when the synchronized-axis movement command is issued in the block following corner chamfering or corner rounding.)
- Tool length / wear compensation
- Tool position offset
- Tool position compensation
- Tool radius compensation
- Nose R compensation
- Machine coordinate system command
- Reference position check
- Manual operation
- Fixed cycle
- Rotation center error compensation (compensation for the angular deviation)
- Tool exchange position return
- Skip

(b) The following commands are issued to the axes related to synchronization between part systems.

- Spatial error compensation

(2) In the following cases, the operation error (M01 1036) occurs.

(a) The following command is issued to the axes related to control axis synchronization between part systems.

- Slave axis of synchronous control
- Secondary axis of multiple-axis synchronization control

(b) The following commands are issued to the synchronized axis of control axis synchronization between part systems.

- Master axis of synchronous control
- Primary axis of multiple-axis synchronization control

(3) In the following cases, a program error occurs.

(a) The following commands are issued to part systems containing the reference axis.

- Tool center point control: (P941)

(b) The following commands are issued to part systems containing the synchronized axis.

- Inclined surface machining: (P952)
- Tool cutting point control: (P911)

Axes that cannot be designated as the axis related to the control axis synchronization between part systems

(1) The operation error (M01 1037) occurs in the following cases:

(a) Any of the following axes is designated as the synchronized axis.

- Reference axis in control axis synchronization between part systems mode
- Master axis in synchronous control mode
- Main axis in simple synchronous control mode
- Primary axis in multiple-axis synchronization control mode

(b) Any of the following axes is designated as the axis related to the control axis synchronization between part systems.

- Slave axis in synchronization control mode
- Sub axis in simple synchronous control mode
- Secondary axis in multiple-axis synchronization control
- Synchronized axis in control axis synchronization between part systems mode
- Reference axis and synchronized axis being synchronized with auxiliary axis
- Axis with servo OFF in spindle position control
- Axis with no zero point established
- Axis in the spindle mode in spindle-mode rotary axis control
- Axis in the loader control part system
- Axis in the part system where laser processing control is enabled

(c) An axis in a part system in which the following functions are being executed is designated as the axis related to the control axis synchronization between part systems. (*1)

(*1) The program error (P521) occurs in synchronization between part systems I, and the operation error (M01 1037) occurs in synchronization between part systems II.

- Rotation center error compensation
- Spatial error compensation

(d) An axis in a part system where any of the following functions is being executed is designated as the reference axis.

- Tool center point control
- Tool cutting point control

(e) An axis in a part system where any of the following functions is being executed is designated as the synchronized axis.

- Inclined surface machining
- Tool center point control
- Tool cutting point control
- 3-dimensional tool radius compensation (tool's vertical-direction compensation)

Other functions

Function name	Operation
Thread cutting	Commands can be issued to the reference axis for control axis synchronization between part systems. If the feedrate exceeds the clamp speed of the reference axis or synchronized axis, the operation error (M01 0107) occurs, and thread cutting will not start.
Reference position return	When the reference position return is commanded to the reference axis, the synchronized axis only synchronizes with the reference axis movement and moves, and the reference position return is not carried out.
Stroke end	These functions are checked at the machine position. When an alarm occurs in an axis in a part system containing the axis related to the control axis synchronization between part systems while these functions are being executed, the part systems containing the reference axis or synchronized axis stop.
Stored stroke limit	
Graphic check	Start or cancel of the synchronization by the G125 command is invalid.
Program restart	

Function name	Operation
Arbitrary axis exchange control	Commands can be issued to the axis related to the control axis synchronization between part systems. If an uncontrolled axis is set as the axis related to the control axis synchronization between part systems by the arbitrary axis exchange function, an axis in a part system containing the uncontrolled axis will be checked about whether or not the axis stops at the start or end of synchronization.
Manual arbitrary reverse run	The G125 command cannot run reversed.
Arbitrary reverse run	The G125 command cannot run reversed.
Inclined axis control	Operation is different between the reference axis and inclined axis for inclined axis control. Whether or not the target is the reference axis or inclined axis for inclined axis control is determined by the setting value of the parameter "#2071 s_axis" regardless of "Inclined axis control valid" signal (YC35). [Reference axis for inclined axis control] The control axis synchronization between part systems can be commanded to the reference axis for inclined axis control ("#2071 s_axis" = "2"). However, the movement command cannot be issued to the inclined axis. If the movement command is issued to the inclined axis, the operation error (M01 0005) occurs. [Inclined axis for inclined axis control] When the control axis synchronization between part systems is commanded to the inclined axis for inclined axis control ("#2071 s_axis" = 1), operation is different between control axis synchronization between part systems I and control axis synchronization between part systems II. ♦Control axis synchronization between part systems I The program error (P521) occurs. ♦Control axis synchronization between part systems II The operation error (M01 1037) occurs.
Machine lock	When the reference axis is in automatic machine lock or manual machine lock, the synchronized axis is also in machine lock. When the automatic machine lock state (enabled/disabled) is the same between the reference axis and the synchronized axis, start or end the control axis synchronization between part systems. If it is started or ended while the state is different, only the block of the synchronized axis stops, and the synchronous relationship between the reference axis and synchronized axis cannot be maintained.
Servo OFF	When the state of the reference axis is servo OFF, that of the synchronized axis is also servo OFF. When "#1064 svof" is set to "1" (error correction), do not turn the axes servo OFF during synchronization. If they are servo OFF, the synchronous relationship between the reference axis and synchronized axis cannot be maintained by the error correction operation.
NC reset (Reset 1/2, Reset & rewind)	When a reset is applied during control axis synchronization between part systems, select whether to cancel the control axis synchronization between part systems using the parameter "#1280 ext16/bit3". Apply the reset to all part systems which are related to the synchronization at a time.
Emergency stop	When the emergency stop occurs during the control axis synchronization between part systems, the control axis synchronization between part systems will be canceled.

Function name	Operation
High-speed machining mode II High-speed high-accuracy control I/II/III	The control axis synchronization between part systems can be combined with high-speed machining mode II or high-speed high-accuracy control I/II/III. When the high accuracy parameter is different between part systems, the synchronized axis moves as follows. ♦Even if the parameters "#1206 G1bF", "#1207 G1btL", and "#1568 SfiltG1" are different between part systems, the synchronized axis moves synchronously with the reference axis. ♦For the parameters "#1570 Sfilt2" and "#2010 fwd_g", the value set to the synchronized axis is valid.
Rotation center error compensation	♦While the control axis synchronization between part systems is being executed, rotation center error compensation amount of the reference axis is applied to the synchronized axis. ♦Execute the rotation center error compensation to the reference axis. When the rotation center error compensation is executed to the synchronized axis, relative positional relation between the reference axis and the synchronized axis may collapsed because the compensation is applied only to the synchronized axis. ♦When the rotation center error compensation is commanded to both the reference axis and synchronized axis, the operation error (M01 1038) occurs. The alarm can be canceled by turning OFF the "Rotation center error compensation enabled" signal (RCEE). ♦The operation when both functions are used together depends on the MTB specifications (parameter "#1450 5axis_Spec/bit5"). Refer to the "Instruction Manual" for details.
Constant-gradient acceleration/deceleration in manual feed	To match the acceleration/deceleration between the reference axis and the synchronized axis, the same setting value of the parameter "#2703 manual_acc" (Enable constant-gradient acceleration/deceleration in manual feed) must be designated for the reference axis and the synchronized axis. If the jog feed is commanded to the reference axis when the setting value is different between the axes, the operation alarm (M01 1036) occurs.
Vibration cutting control	♦When a cutting command is issued to the reference axis for control axis synchronization between part systems during the vibration cutting mode, the reference axis vibrates, but the synchronized axis does not vibrate. ♦When there is a synchronized axis in the part system during the vibration cutting mode, the operation error (M01 1302) occurs and cutting is performed without vibration cutting control.
Chopping	♦Do not set the reference axis or synchronized axis as the chopping axis. ♦When the control axis synchronization between part systems is started and the part system that contains the reference axis or synchronized axis has an axis during chopping, the operation error (M01 1037) occurs. The operation is the same when this function is canceled.



Precautions

Common to control axis synchronization between part systems I and II

(1) The number of combinations is not limited unless the reference axis is duplicated with the synchronized axis.

(2) When the control axis synchronization between part systems are started/canceled, the related two part systems wait for the completion of the currently executing block, and the synchronization starts/cancels after all the axes reached "smoothing zero" in both part systems.

(3) As the state of two target part systems affects the control axis synchronization between part systems, be aware of the timing between the part systems.
Therefore, command the timing synchronization operation before or after commanding the synchronization start/cancel.

(4) Do not change the related parameters during the control axis synchronization between part systems.

(5) The target control axis at the start of the control axis synchronization between part systems should be in a state where the reference position return after power ON is complete or the absolute position is established.

Precautions for control axis synchronization between part systems I

(1) If G125 is commanded where control axis synchronization between part systems II is selected (*1), the program error (P610) occurs.

(*1) This setting depends on the MTB specifications (parameter "#1280 ext16/bit5"= 0).

(2) G125 can be commanded from any part system.

(3) The synchronization start/end command can also be issued from a part system which does not contain the axis (reference axis or synchronized axis) related to the control axis synchronization between part systems.

(4) If an axis other than synchronized axis is specified at the synchronization end command, that command is ignored.

(5) When only the G125 command is issued, it is ignored.

(6) G125 must be commanded alone in a block.
The program error (P33) occurs if it is not commanded alone in a block.

(7) When designating the "reference axis name" and "synchronized axis name" in the G125 command, command the names specified in the parameter "#1022 axname2".
If an unspecified name is commanded, the program error (P521) occurs.

(8) If the same axis name is given to the reference axis and the synchronized axis, the program error (P521) occurs.

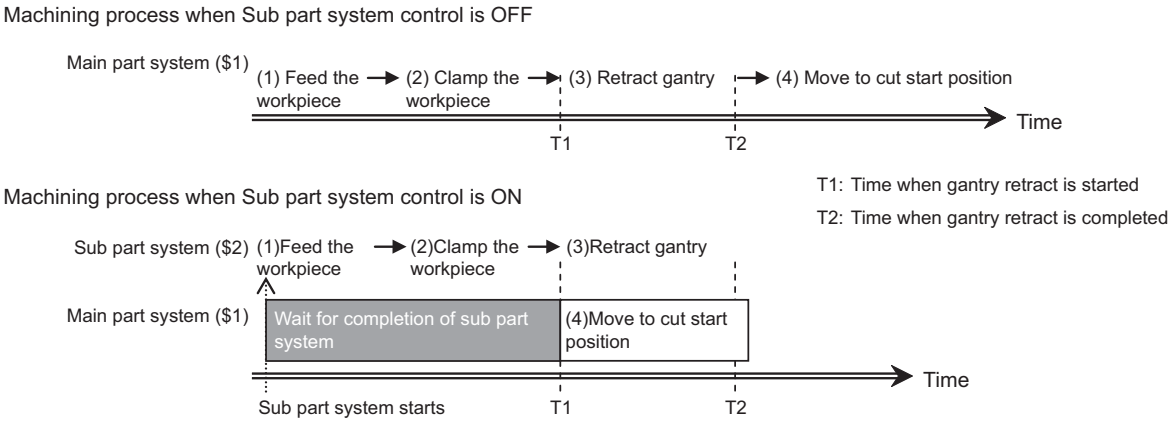
(9) Two axes in the same part system cannot be commanded as a reference axis and synchronized axis.
If this type of command is given, the program error (P521) occurs.

16.4 Sub Part System Control

16.4.1 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.



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 M80V series.

[Condition 1]

This condition must be satisfied only for the M80V 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) 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)
              It must be commanded alone.
```

Cancel the standby status for completion of sub part system (command in the sub part system side when the D0 command is issued)

```
G145;
```

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. •The block with sequence No. of the sub part system specified with the address Q must be commanded alone. •If there is no command, the program will run up to "M99".
K	Number of repetitions (*3)	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. In this case, the "In reset" signal (XC15) turns ON. To initialize the G command modal by the reset when a sub part system is completed, command "H1" while the parameter "#1151 rstinit" is set to "1".

(*3) When the machining program in the sub part system is repeated by the K command, the sub part system does not end and reset is not performed.

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.

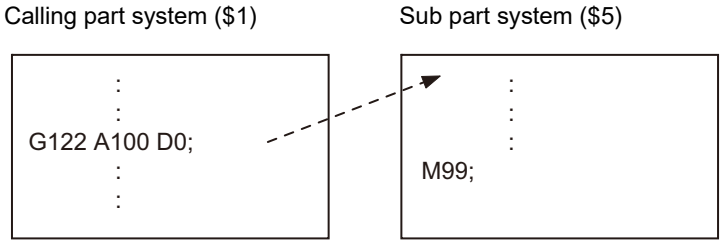
Activation part system of a sub part system

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")

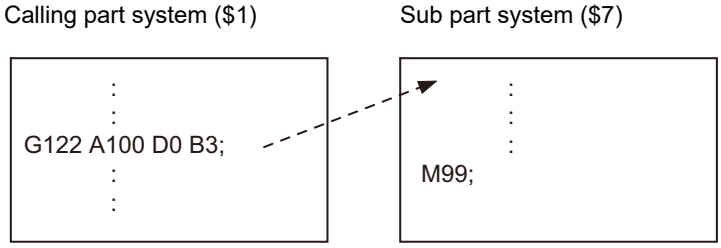
(Example 1) and (Example 2) show the operations when parameters are set as shown below.
The available number of part systems depends on your machine's specifications.

#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.



(Example 2) Sub part system identification No. (the part system No. to be activated and correspondence) can be specified with the B command.



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.

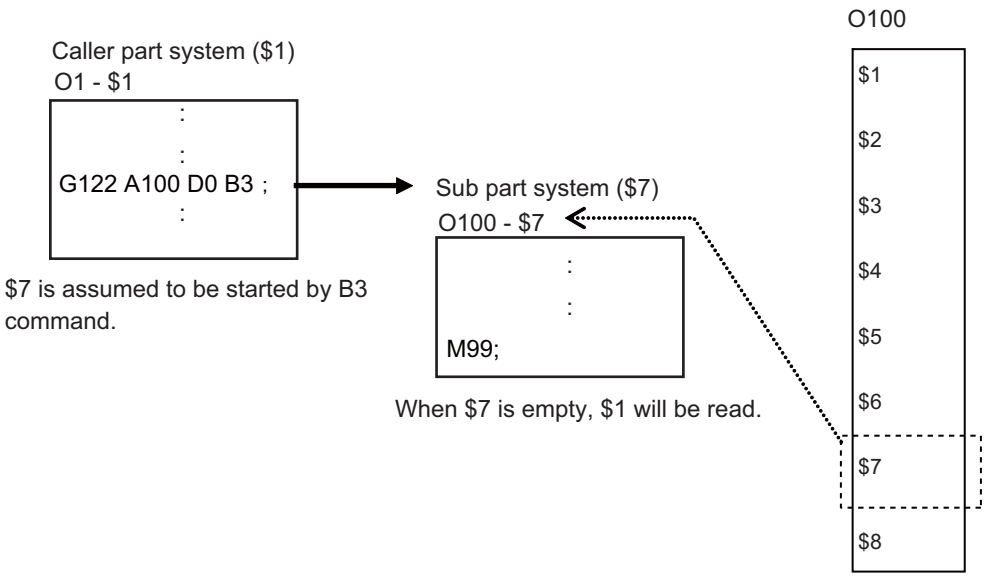
Sub part system operation programs need to be stored in the memory or an external device. The storage locations are shown in the table below.

#11053 UserProgram-Storage	#11085 ExtSBSPrgSelect	Program storage location
0 (Memory)	0 / 1	NC memory (NC memory 2 is unavailable)
1 (Hard disk)	0 (Main program storage location in external device)	The storage location is in accordance with the following parameters for the sub part system. #12091 ExtMainProgram: dev #12092 ExtMainProgram: dir
	1 (Sub program storage location in external device)	The storage location is in accordance with the following parameters for the sub part system. #12093 ExtSubProgram: dev #12094 ExtSubProgram: dir

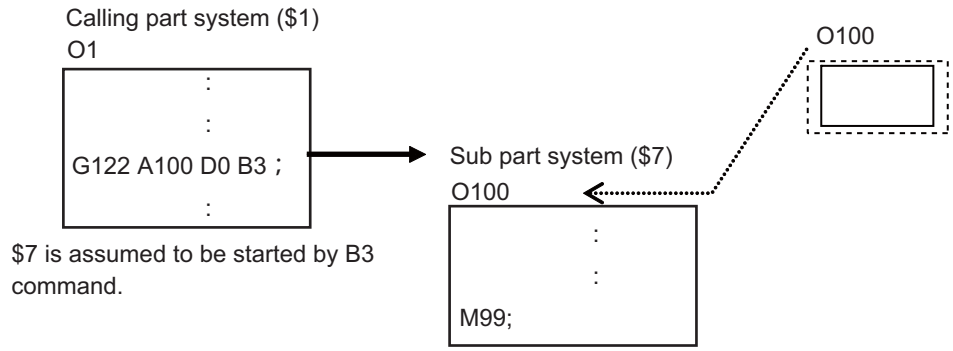
For a multi-part system, the program to be operated is switched depending on the parameter settings (program management method). (This depends on the MTB specifications.)

#1285 ext21/bit0	#1285 ext21/bit2	
	0	1
1	In a specified program, a part indicated with the number of the main part system is operated. (In the figure (1) below, when the seventh part system is designated as the sub part system, the part provided for $7 in the "O100" program will be operated.)	
	When the program of the sub part system's number is empty, the program of the first part system in the specified program is operated. When the program of the first part system in the specified program is also empty, the program error (P461) occurs.	When the program of the sub part system's number in the specified program is empty, the program error (P461) occurs. The program of the first part system in the specified program will not be called.
0	The specified program is operated.	

(1) When the multi-part system program management is enabled ("#1285 ext21/bit0" is set to "1")

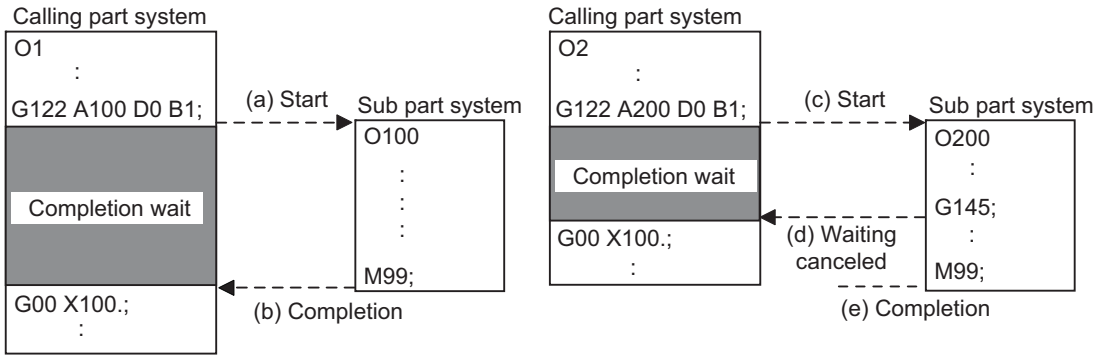


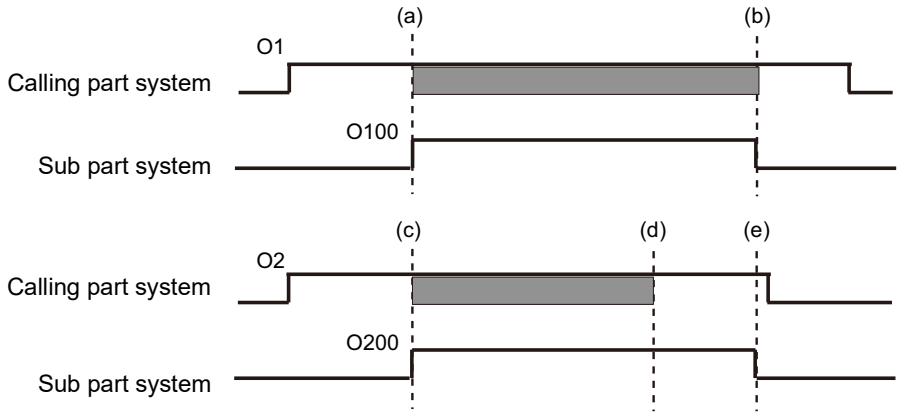
(2) When the multi-part system program management is disabled ("#1285 ext21/bit0" is set to "0")



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.

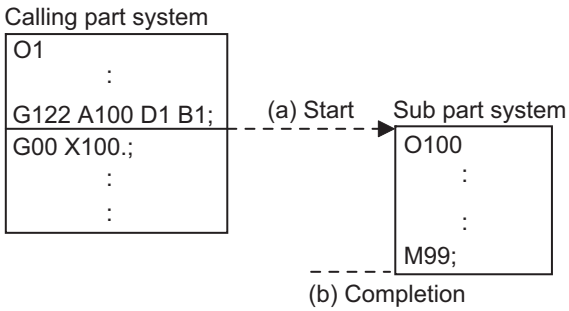


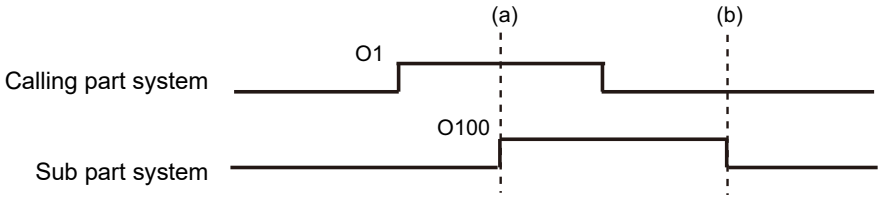


: Completion wait

Activation of a sub part system with parallel processing mode (D=1)

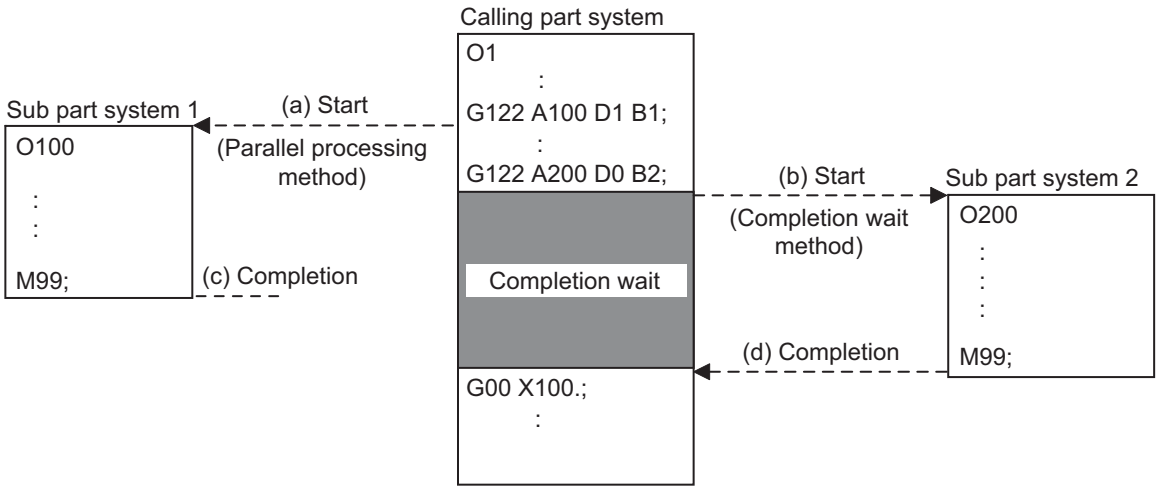
If "1" is designated for command address D when the sub part system control I command is issued, the following blocks of the calling part system and the first and the following blocks of the sub part system will be operated in parallel.
The following shows the operation and the activation timing of each part system.

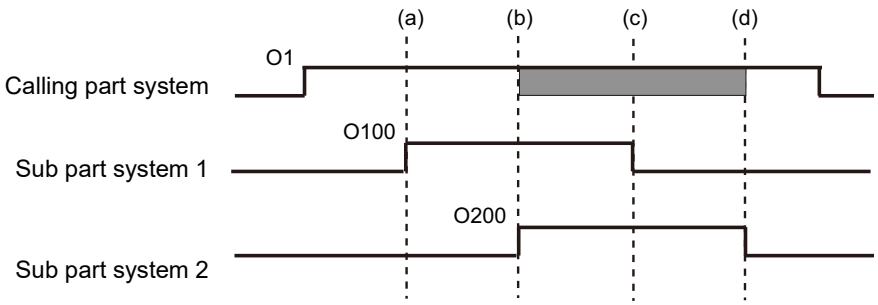




Activation of multiple sub part systems

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.

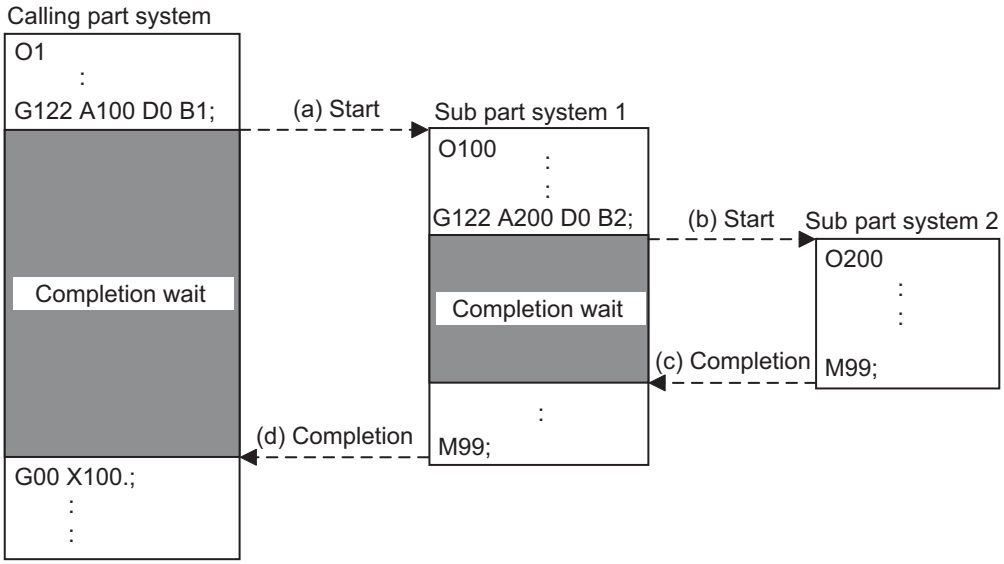


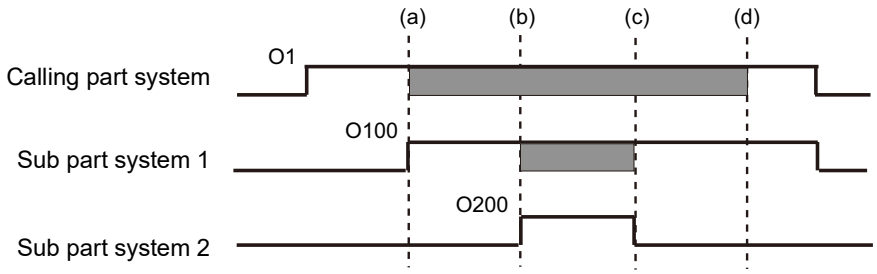


: Completion wait

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.

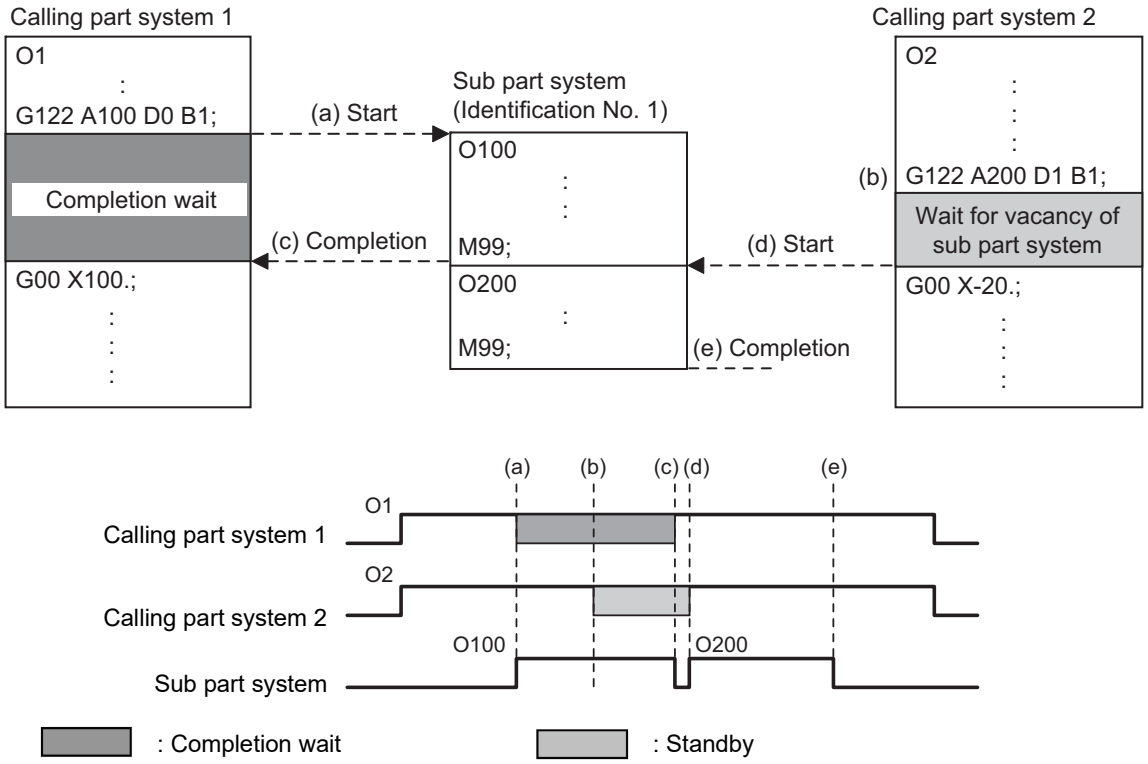




: Completion wait

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

(1) Machining process when sub part system control is OFF

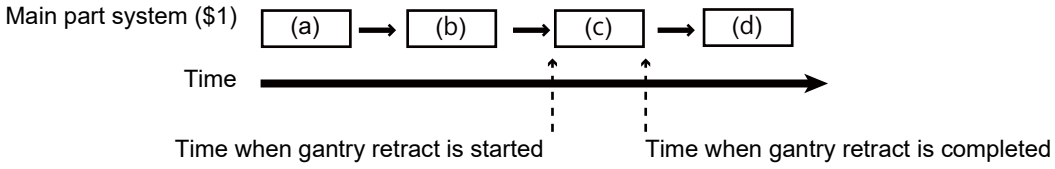
Main part system ($1)

```
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

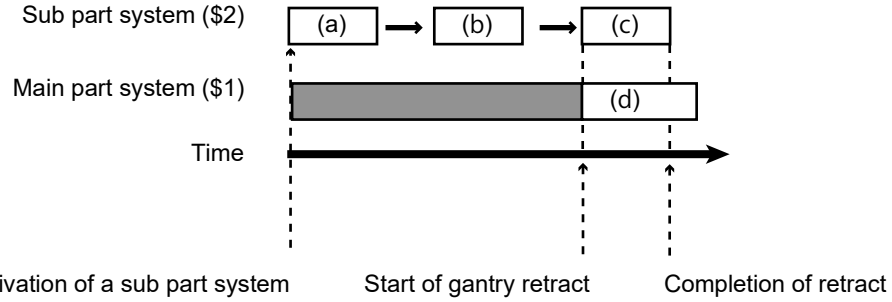
Main part system ($1)

```
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



: Completion wait

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.

Main part system ($3)

```
        :
G122 A100 D1 B1;
G122 A200 D1 B2;
        :
!5!6;
G00 X100.;
        :
        :
```

Timing synchronization with 5th and 6th part systems

Sub part system 1 ($5)

```
        :
        :
        :
        :
!3!6;
        :
        :
        :
```

Timing synchronization with 3rd and 6th part systems

Sub part system 2 ($6)

```
        :
        :
        :
        :
!3!5;
        :
        :
        :
```

Timing synchronization with 3rd and 5th part systems

(Example 2) Timing synchronization by designating a sub part system identification No.

Main part system ($3)

```
        :
G122 A100 D1 B1;
G122 A200 D1 B2;
        :
![1]![2];
G00 X100.;
        :
```

Timing synchronization with the following part system

- Sub part system of identification No. 1
- Sub part system of identification No. 2

Sub part system 1 ($5, identification No. 1)

```
        :
        :
        :
        :
![0]![2];
        :
        :
```

Timing synchronization with the following part system

- Calling part system ($3)
- Sub part system of identification No. 2

Sub part system 2 ($6, identification No. 2)

```
        :
        :
        :
        :
![0]![1];
        :
        :
```

Timing synchronization with the following part system

- Calling part system ($3)
- Sub part system of identification No. 1

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).

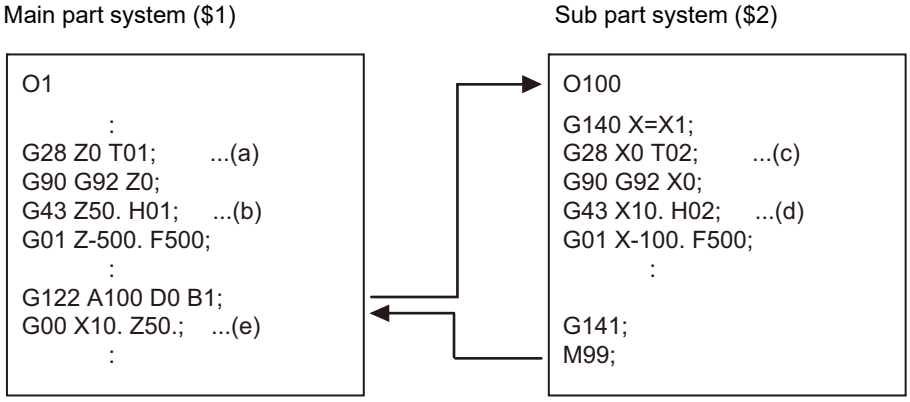
Tool functions

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 : M99;	Program operated in sub part system

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

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 M80V 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".
- 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.

16.5 Loader Control Part System



Function and purpose

This function operates a part system which is dedicated to a carrier loader (hereinafter referred to as "loader") in the machining center system. This function can be used to isolate the loader system from other controls, allowing your machine to operate in automatic operation with CNC machining program.

Compared to when a loader is operated in a workpiece machining part system, the cycle time can be improved because the loader-specific process (such as placing the next workpiece and carrying the machined workpiece) is performed independently while a workpiece is being machined.

[Example of system configuration]

1st part system	X1, Y1, Z1, A1, C1
2nd part system	X2, Y2, Z2, A2, C2
Loader-dedicated part system	X3, Z3

The validity of this function and the designation of loader-dedicated part system depend on the MTB specifications (parameter "#11811 LDR Dedicated Sys" (Loader-dedicated part system (For M system only))).
In the loader-dedicated part system, the functions that can be used are limited to loader functions.



Detailed description

G code function list

The following is the list of G code functions which are available in the loader-dedicated part system.

G Code	Group	Function
G00	01	Positioning
G0.5	28	Rapid traverse block overlap
G01	01	Linear interpolation
G04	00	Dwell
G09	00	Exact stop check
G10	00	Programmable data input
G11	00	Programmable data input cancel
G17	02	X-Y plane
G18	02	Z-X plane
G19	02	Y-Z plane
G20	06	Inch command
G21	06	Metric command
G22	04	Stroke check before travel ON
G23	04	Stroke check before travel OFF
G27	00	Reference position check
G28	00	Reference position return
G29	00	Start position return
G30	00	2nd to 4th reference position return
G30.1	00	Tool exchange position return 1
G30.2	00	Tool exchange position return 2
G30.3	00	Tool exchange position return 3
G30.4	00	Tool exchange position return 4

G Code	Group	Function
G30.5	00	Tool exchange position return 5
G30.6	00	Tool exchange position return 6
G31	00	Skip/Multi-step skip 2
G31.1	00	Multi-step skip 1-1
G31.2	00	Multi-step skip 1-2
G31.3	00	Multi-step skip 1-3
G50	11	Scaling cancel
G51	11	Scaling ON
G52	00	Local coordinate system setting
G53	00	Basic machine coordinate system selection
G54	12	Workpiece coordinate system 1 selection
G55	12	Workpiece coordinate system 2 selection
G56	12	Workpiece coordinate system 3 selection
G57	12	Workpiece coordinate system 4 selection
G58	12	Workpiece coordinate system 5 selection
G59	12	Workpiece coordinate system 6 selection
G54.1	12	Extended workpiece coordinate system selection
G60	00	Unidirectional positioning
G60	01	Unidirectional positioning (Modal)
G61	13	Exact stop check mode
G64	13	Cutting mode
G65	00	User macro simple call
G66	14	User macro modal call A
G66.1	14	User macro modal call B
G67	14	User macro modal call cancel
G90	03	Absolute command
G91	03	Incremental command
G92	00	Coordinate system setting/Spindle clamp speed setting
G92.1	00	Workpiece coordinate system preset
G93	05	Inverse time feed
G94	05	Feed per minute
G100 - G225	00	User macro (G code call) Maximum 10
G115	00	Start point timing synchronization
G116	00	Start point timing synchronization
G127	00	Manual arbitrary reverse run prohibition
G130	00	Selection of axis (axes) for feedrate command
G160	00	Torque limitation skip
G186	00	Interference check III setting data valid

Functions that cannot be used

In the loader-dedicated part system, functions such as spindle functions and tool functions cannot be used. Also, G codes other than those in the "G code function list" cannot be used.

(1) Mirror image

Even when the parameter "#8211 MIRR. IMAGE" is enabled for the axis to be used in the loader-dedicated part system, the parameter is ignored.
Also, even when the "Mirror image 1st axis" to "Mirror image 8th axis" signals which are PLC output signals for the loader-dedicated part system are turned ON, the signals are ignored.

(2) Chopping

In the loader-dedicated part system, chopping cannot be used. Even when the "Chopping" signal (CHPS) is turned ON in the loader-dedicated part system, the chopping mode is not entered, and the error No. (20) which shows that there is no specification for chopping is output to "Chopping error No." (R554).

(3) High-accuracy control function, high-speed high-accuracy control function

In the loader-dedicated part system, the high-accuracy control function and high-speed high-accuracy control function cannot be used. Even when the parameter "#1148 I_G611" (Initial hi-precis) is set to a value other than "0", the operation is performed in the cutting mode. Even when the parameter "#8040 High-SpeedAcc" is set to "1", the high-speed high-accuracy control cannot be used. Even when the parameter "#8090 SSS ON" is set to "1", the SSS control is not enabled. Even when the parameter "#1205 G0bdcc" (Acceleration and deceleration before G0 interpolation) is set to "1", the acceleration/deceleration after interpolation is applied to the acceleration/deceleration in G00.

(4) Tool handle feed & interruption

In the loader-dedicated part system, the tool handle feed & interruption function cannot be used.

(5) Rapid traverse constant-gradient multi-step acceleration/deceleration

In the loader-dedicated part system, the rapid traverse constant-gradient multi-step acceleration/deceleration function cannot be used. Even when the parameter "#1205 G0bdcc" (Acceleration and deceleration before G0 interpolation) is set to "2", the rapid traverse constant-gradient multi-step acceleration/deceleration is not enabled in the loader-dedicated part system.



Precautions

(1) When the system is configured with three or more part systems including the loader-dedicated part system, set the parameter "#1468 ctrl period" to "1". Otherwise, the machining accuracy may decrease in the main part system.

17

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 micro segments at high speed.
A higher micro 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 micro 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.

High-speed machining mode I

G01 block micro segment processing capability for 1 mm segment (unit: kBPM)

Maximum feedrate when 1 mm segment G01 block is executed (kBPM)			
M850V/M830V	M80VW	M80V	
		TypeA	TypeB
33.7	33.7	33.7	16.8

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 functions.

High-speed machining mode II

G01 block micro segment processing capability for 1 mm segment (unit: kBPM)

	Specified number of part systems (#8040 = 1)	Maximum feedrate when 1 mm segment G01 block is executed (kBPM)		
		M850V/M830V	M80VW/M80V TypeA	M80V TypeB
♦1-part system (8 axes or less)	1 part system	168 135 (*3) 101 (*1) 67.5 (*2)(*4)	101 67.5 (*1) 33.7 (*2)(*4)	67.5
♦1-part system (9 axes or more) ♦2-part system	1 part system	135 101 (*3) 67.5 (*1) 33.7 (*2)(*4)	101 67.5 (*1) 33.7 (*2)(*4)	67.5
	2 part systems	67.5 33.7 (*2)	33.7	-
♦3-part system	1 part system	33.7 (*5)	-	-
	2 part systems	33.7 (*5)	-	-
♦4-part system (16 axes or less)	1 part system	33.7 (*5)	33.7	-
	2 part systems	33.7 (*5)	33.7	-
♦4-part system (17 axes or more) ♦5-part system or more	1 part system	16.8 (*5)	-	-
	2 part systems	16.8 (*5)	-	-

(*1) This is the micro segment processing capability in the time constant expansion system or in any of the following modes.

- ♦Tool center point control (G43.4/G43.5)
- ♦Inclined surface machining (G68.2/G68.3)
- ♦Workpiece installation error compensation (G54.4P1 to P7)
- ♦3-dimensional tool radius compensation (tool's vertical-direction compensation) (G41.2/G42.2)
- ♦Spatial error compensation
- ♦Variable-acceleration pre-interpolation acceleration/deceleration
- ♦Spline interpolation 2 (G61.4)
- ♦3D machine interference check
- ♦Tool cutting point control (G43.8/G43.9)
- ♦Real-time 3D machine interference check

(*2) This is the micro segment processing capability when the smooth fairing function is enabled ("#8033 Fairing ON" is set to "2").

(*3) This is the micro segment processing capability when the tolerance control is enabled ("#12066 Tolerance ctrl ON" is set to "1").

(*4) When the smooth fairing function is enabled ("#8033 Fairing ON" is set to "2") and the parameter "#1468 ctrl period" is set to "1", the program error (P29) occurs.

(*5) Optional specification of part systems is required separately.

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 / Tool nose radius compensation is canceled (G40)
- ♦The parameter "#1468 ctrl period" is set to "0". (This depends on the MTB specifications.)

When the above conditions are not satisfied, the given feedrate in the table may not be secured.

(2) The performance in the table may vary depending on the combination with other functions.

(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 alone in a block. A program error (P33) occurs if a movement or other command is additionally issued in a G05 command block. A program error (P33) also occurs 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 commanded while high-speed machining mode II is valid, the micro segment processing capability decreases. 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 micro segment processing capability does not decrease.

(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.).

(5) If geometric command is programmed while high-speed machining mode II is active, a program error (P33) occurs.



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 combination 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 combination function is enabled

○: The high-speed machining mode II and the combination function are both enabled

Δ: The high-speed machining mode II is temporarily canceled, while the combination 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	□ (*3)	□ (*3)
	G05P10000	High-speed high-accuracy control II ON	□ (*4)	□ (*2)
	G05P20000	High-speed high-accuracy control III ON	□ (*5)	□ (*6)
	G05.1Q0	High-speed high-accuracy control I OFF Spline interpolation OFF	□ (*3)	□ (*2)
	G05.1Q1	High-speed high-accuracy control I ON	□ (*7)	□ (*2)
	G05.1Q2	Spline interpolation ON	○ (*12)	○ (*12)
	G07	Hypothetical axis interpolation	Δ (*12)	Δ (*12)
	G08P0	High-accuracy control OFF	□ (*3)	□ (*2)
	G08P1	High-accuracy control ON	□ (*8)	□ (*8)
	G09	Exact stop check	Δ	-
	G10 I_J_ G10 K_	Parameter coordinate rotation input	Δ (*12)	- (*12)
	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 Multi-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	-
	G53	Machine coordinate system selection	Δ	-
	G60	Unidirectional positioning	Δ	-

Group	G code	Function name	A	B
0	G65	User macro simple call	□ (*9)	□ (*10)
	G92	Coordinate system setting	Δ	-
	G92.1	Workpiece coordinate system preset	Δ (*12)	- (*12)
	G122	Sub part system control I	Δ (*12)	□ (*11)(*12)
1	G00	Positioning	Δ	Δ
	G01	Linear interpolation	○	○
	G02 G03	Circular interpolation	○	○
	G02.1 G03.1	Spiral interpolation	Δ (*12)	Δ (*12)
	G02.3 G03.3	Exponential interpolation	Δ (*12)	Δ (*12)
	G02.4 G03.4	3-dimensional circular interpolation	Δ (*12)	Δ (*12)
	G06.2	NURBS interpolation	○ (*12)	○ (*12)
	G33	Thread cutting	Δ	Δ
2	G17 - G19	Plane selection	○	○
3	G90	Absolute command	○	○
	G91	Incremental command	○	○
4	G22	Stroke check before travel ON	Δ	Δ
	G23	Stroke check before travel OFF	○	○
5	G93	Inverse time feed	Δ (*12)	Δ (*12)
	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	○ (*12)	○ (*12)
	G43.4 G43.5	Tool center point control	○ (*12)	○ (*12)
	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	○	○
	G61.2	High-accuracy spline	○ (*12)	○ (*12)
	G61.4	Spline interpolation 2	○ (*12)	○ (*12)
	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)(*12)	× (P29)(*12)
16	G68	Coordinate rotation by program ON	○	○
		3-dimensional coordinate conversion ON	× (P922)	× (P921)
	G68.2 G68.3	Inclined surface machining command	○ (*12)	○ (*12)
	G69	Coordinate rotation cancel	○	○
		3-dimensional coordinate conversion OFF	○	○
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	× (P34)	× (P481)
	G12.1	Polar coordinate interpolation ON	× (P34)	× (P481)
	G13.1	Polar coordinate interpolation OFF	○	○
27	G54.4P0	Workpiece installation error compensation cancel	○	○
	G54.4 P1 - P7	Workpiece installation error compensation	○ (*12)	○ (*12)

(*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 high-accuracy control II.

(*5) Enables the high-speed high-accuracy control III.

(*6) High-speed high-accuracy control III continues.

(*7) Enables the high-speed high-accuracy control I.

(*8) Enables the high-speed machining mode II and high-accuracy control.

(*9) Enables the high-speed machining mode II in a macro program.

(*10) Enables the high-speed machining mode II if "G05P2" is commanded in a macro program.

(*11) Enables the high-speed machining mode II if "G05P2" is commanded in a sub part system.

(*12) In M80V TypeB, the following program errors occur depending on the G codes.

G code	Program error
G05.1Q2, G92.1, G122, G61.2, G61.4	P39
G54.4 P1 - P7	P34
G07	P80
G02.1, G03.1	P73
G02.3, G03.3	P611
G02.4, G03.4	P76

G code	Program error
G10 I_J_/G10 K_	P260
G06.2	P550
G93	P124
G41.1, G42.1	P900
G43.1	P930
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 combination 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 combination function is enabled

○: The high-speed machining mode II and the combination function are both enabled

Δ: The high-speed machining mode II is temporarily canceled, while the combination 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
Mirror image by external input 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)	-
Machine tool builder macro	□ (*4)	□ (*5)
Macro interruption	□ (*6)	□ (*7)
Corner chamfering/Corner rounding	Δ	-
Linear angle command	Δ	-
Geometric command	× (P33)	-
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 M80V TypeB, 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) occurs.

(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 alone in a 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) occurs.
If there is no P command, a program error (P33) occurs.

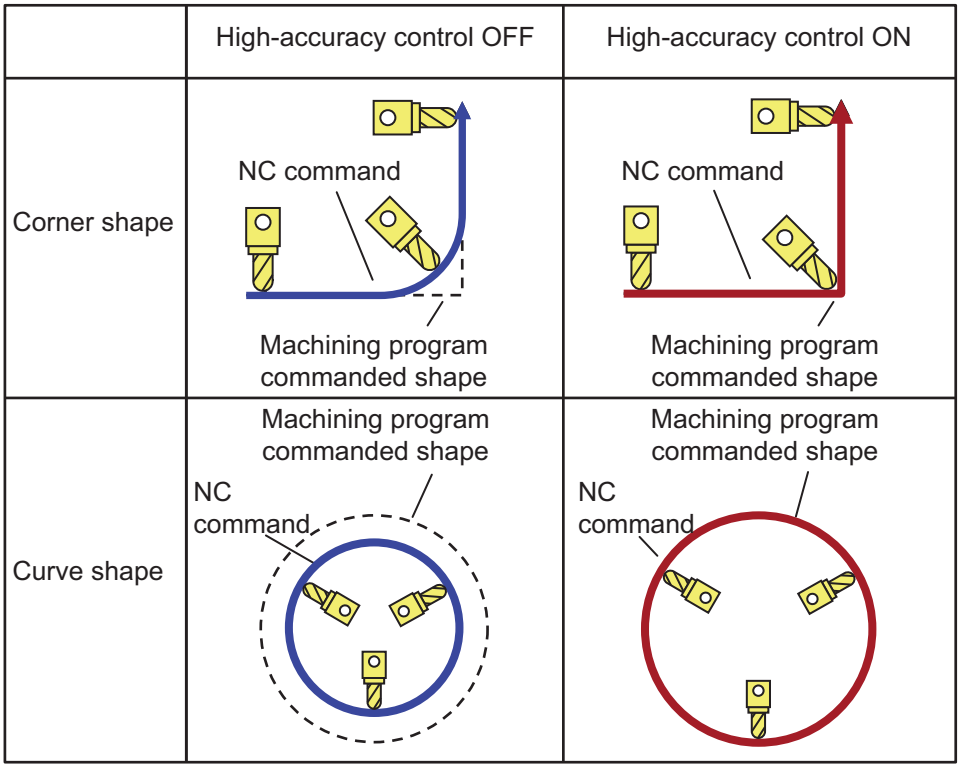
(8) The machining speed may be suppressed depending on the number of characters in one block.

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 and deceleration is automatically performed according to a machining shape so that the machining error is inhibited with minimizing an extension of machining time.



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

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) When either "G08P1" or "G61.1" is commanded, a modal in the G code group 00 is changed to "G08P1", and a modal in the G code group 13 is changed to "G61.1".
When either "G08P0" or "G command other than G61.1 in the G code group 13" is commanded, a modal in the G code group 00 is changed to "G08P0". A modal in the G code group 13 is changed as follows.

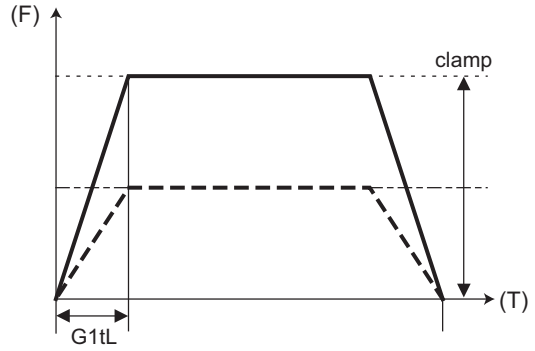
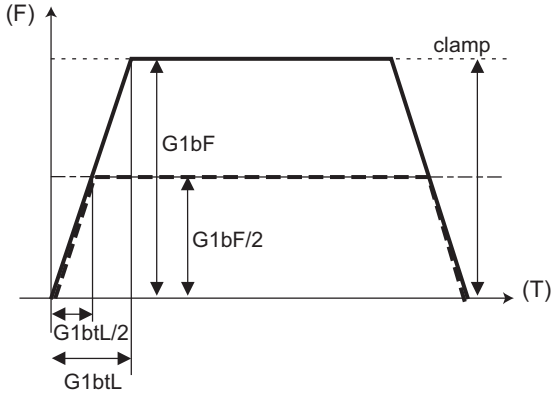
- When "G08P0" is commanded, it is changed to "G64".
- When "G command other than G61.1 in the G code group 13" is commanded, it is changed to the "commanded mode".

Detailed description

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-gradient acceleration/deceleration is performed for the acceleration/deceleration before interpolation.

(1) Basic patterns of acceleration/deceleration control in linear interpolation commands

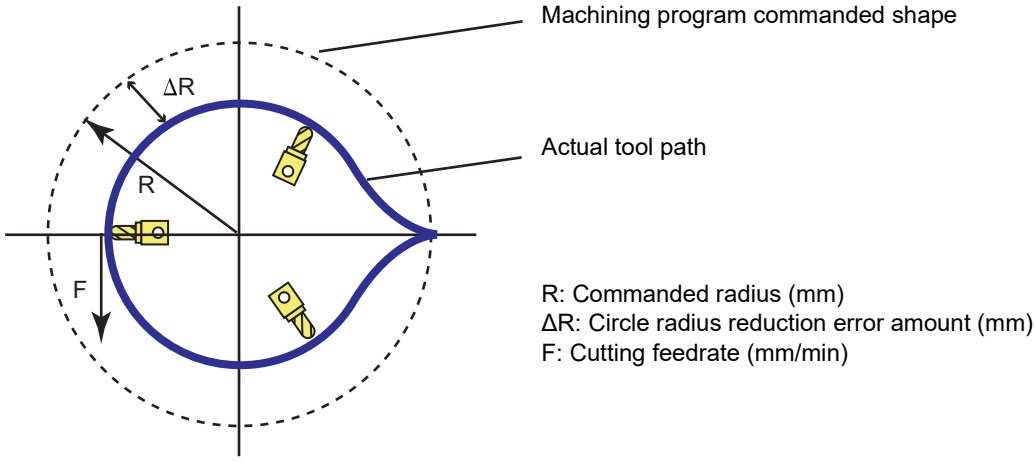
	Acceleration/deceleration waveform pattern	
Acceleration/deceleration after interpolation (Normal mode)		(a) Because of the acceleration/deceleration that controls the acceleration time to achieve the commanded speed at a constant level (acceleration/deceleration with fixed time constant), the acceleration/deceleration 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)
Pre-interpolation acceleration/deceleration (High-accuracy control mode)	 (F) Resultant speed (T) Time	(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-gradient 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 gradient of the acceleration/deceleration time. The actual cutting feed maximum speed is clamped by the parameter "#2002 clamp" value.

(2) Path control in circular interpolation commands

When commanding circular interpolation with the conventional "acceleration/deceleration after interpolation" 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 "acceleration/deceleration after interpolation" 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 acceleration/deceleration after interpolation (normal mode), ΔR is caused by acceleration/deceleration of NC and lag of servo system. Pre-interpolation acceleration/deceleration (high-accuracy control mode), 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.

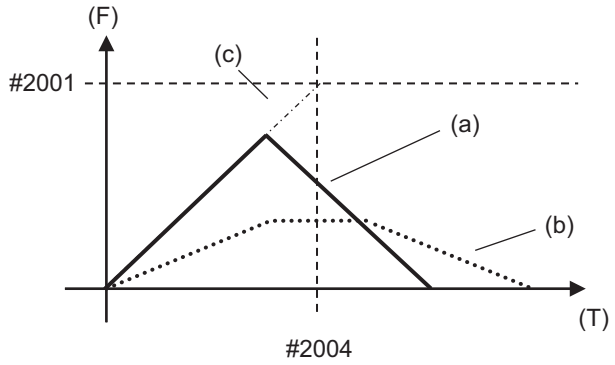
The compensation amount of the circle radius reduction error (ΔR) is theoretically calculated as shown in the following table.

"Acceleration/deceleration after interpolation" control (normal mode)	**"Pre-interpolation acceleration/deceleration" control (high-accuracy control mode)**
Linear acceleration/deceleration $\Delta R = \frac{1}{2R}\left(\frac{1}{12}Ts^2 + Tp^2\right)\left(\frac{F}{60}\right)^2$ Exponential function acceleration/deceleration $\Delta R = \frac{1}{2R}\left(Ts^2 + Tp^2\right)\left(\frac{F}{60}\right)^2$	Linear acceleration/deceleration $\Delta R = \frac{1}{2R}\left\{Tp^2\left(1 - Kf^2\right)\right\}\left(\frac{F}{60}\right)^2$ (a) Because the item Ts can be ignored by using the pre-interpolation acceleration/deceleration control method, the radius reduction error amount can be reduced. (b) Item Tp can be negated by making Kf = 1.

Ts: Acceleration/deceleration time constant in the NC (s)
Tp: Servo system position loop time constant (s) (inverse number to "#2203 PGN1")
Kf: Feed forward coefficient
Kf = fwd_g / 100 (fwd_g: #2010 Feed forward gain)

Rapid traverse pre-interpolation acceleration/deceleration

When "#1205 G0bdcc" (G0 pre-interpolation acceleration/deceleration) is "1", pre-interpolation acceleration/deceleration is also enabled for rapid traverse movement. In this case, acceleration/deceleration control is performed so that the acceleration rate of each axis does not exceed the gradient determined by parameters "#2001 rapid" (rapid traverse rate) and "#2004 G0tL" (G0 time constant (linear)).



(a) Velocity waveform of pre-interpolation acceleration/deceleration

(b) Velocity waveform of acceleration/deceleration after interpolation

(c) Gradient of pre-interpolation acceleration/deceleration

(F) Speed

(T) Time

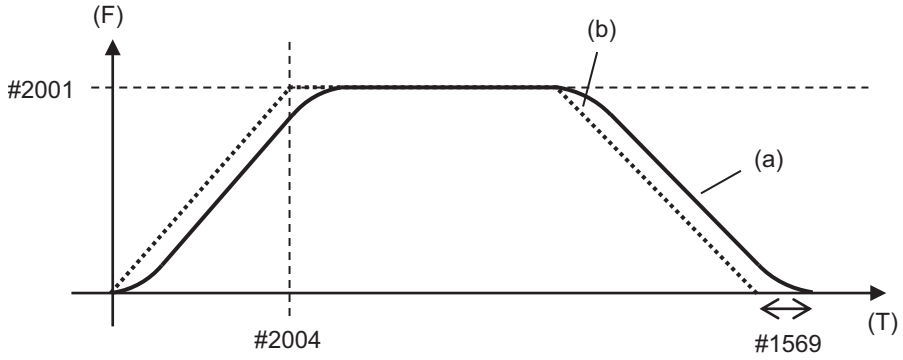
When the gradient of each axis is different, the most moderate gradient among them is used in pre-interpolation acceleration/deceleration.
Because acceleration/deceleration control is performed with a constant gradient, the positioning time for a G00 fine-segment block is reduced.

Note

(1) When "#1086 G0Intp" (G00 non-interpolation) is "1", acceleration/deceleration after interpolation is applied to G00.

(2) When "#8090 SSS ON" is "1", pre-interpolation acceleration/deceleration is applied to G00 regardless of the setting in "#1205 G0bdcc" (Acceleration and deceleration before G0 interpolation).

(3) When "#1569 SfiltG0" (G00 soft acceleration/deceleration filter) is set to a value other than "0", cycle time may become longer when pre-interpolation acceleration/deceleration is applied to linear acceleration/deceleration than when acceleration/deceleration after interpolation is applied.



(a) Velocity waveform of pre-interpolation acceleration/deceleration

(b) Velocity waveform of acceleration/deceleration after interpolation

(F) Speed

(T) Time

Optimum speed control

When the moving direction is changed on the corner, arc, etc., acceleration rate corresponding to the amount of change and the feedrate is generated. When the acceleration rate 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 rate 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 axis-specific acceleration tolerance control.

The control method to be used depends on the MTB specifications (parameter "#12053 EachAxAccCntrl").

[#12053 EachAxAccCntrl = 0]
Optimum corner deceleration is performed.

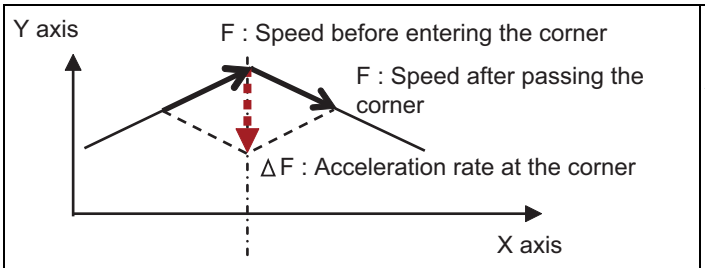
[#12053 EachAxAccCntrl = 1]
Axis-specific acceleration tolerance control is performed.
When the specification of axis-specific acceleration tolerance control is disabled, optimum corner deceleration is performed regardless of the parameter setting.

Arc speed clamp

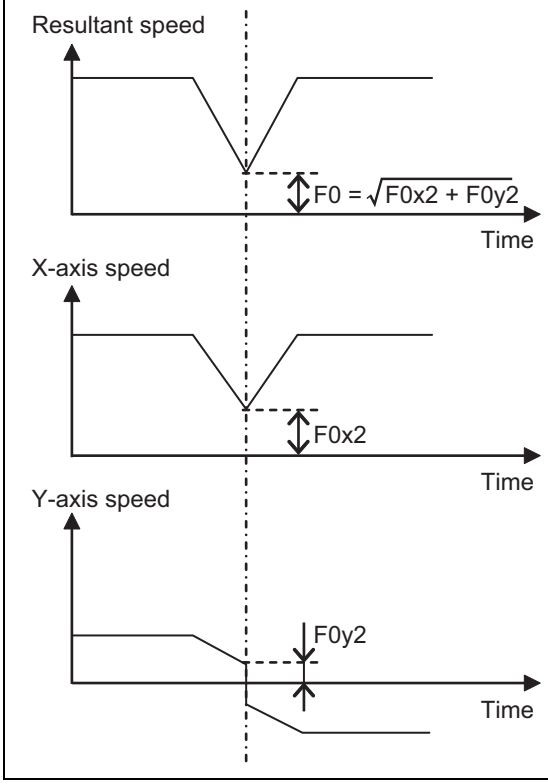
Controls deceleration so that the combined acceleration rate on an arc is kept below the tolerable acceleration rate common to all axes. This can suppress path errors (circle radius reduction error amount) on an arc to a certain level.

(1) Optimum corner deceleration

Highly accurate edge machining can be achieved by controlling deceleration so that the combined acceleration rate at the seam between blocks is kept under the tolerable acceleration rate 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 rate common to all axes. The machine decelerates to the speed in advance, and then accelerates back to the command speed after passing the corner.



When a corner with the corner angle θ is passed at speed F, the acceleration rate ΔF occurs according to θ and F.



The corner speed F is controlled so that ΔF generated above does not exceed the tolerable acceleration rate common to all axes.
The speed pattern is as shown on the left.

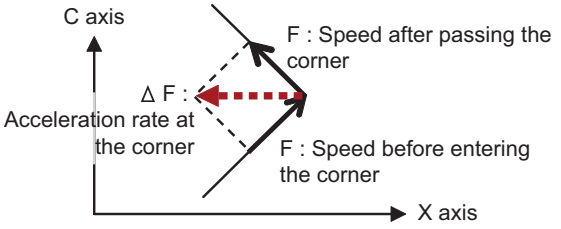
Optimum corner deceleration is not carried out when blocks are smoothly connected, because deceleration is not necessary. The criteria for whether the connection is smooth or not can be designated by the machining parameter "#8020 DCC ANGLE". If the corner angle is equal to or less than the corner deceleration 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 rate 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

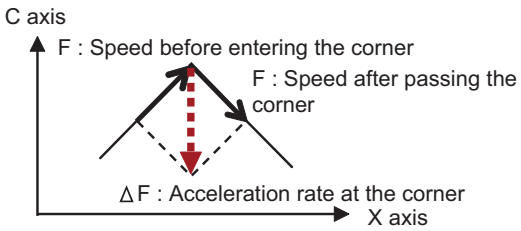
$$\text{Tolerable acceleration rate for all axes } (mm/s^2) = \frac{\text{G1bF (mm/min)} / 60}{\text{G1btL (ms)} / 1000} \times \frac{100 - \text{R COMP}}{100}$$

(2) Axis-specific acceleration tolerance control (optimum acceleration control)

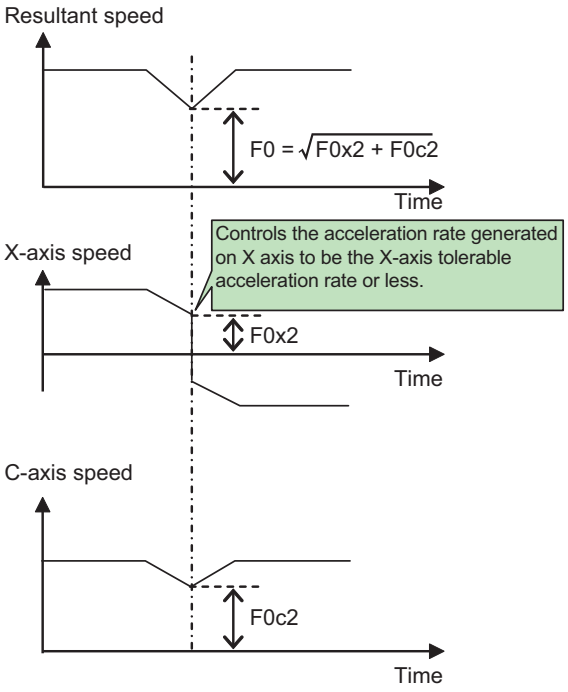
The acceleration rate to be generated at a seem between blocks is evaluated for each axis to control deceleration so that the seam is passed at the optimum speed. This enables highly accurate edge machining.
The optimum deceleration speed is calculated so that the acceleration rate of each axis to be generated at the seam is equal to or less than the tolerable acceleration rate 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 rate for a specific axis (rotary axis). This means that the deceleration speed can be raised at a corner where acceleration rate is generated only for an axis with a high tolerable acceleration rate, leading to a reduced cycle time.
If acceleration rate 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 rate to be generated at the X or C axis does not exceed the tolerable acceleration rate for the X or C axis, respectively. If the tolerable acceleration rate for the X axis is higher than that for the C axis, a higher deceleration speed can be used for a path where acceleration rate is generated only for the X axis than where acceleration rate is generated only for the C axis. In this case, the speed patterns are as shown in Figures (c) and (d) below:



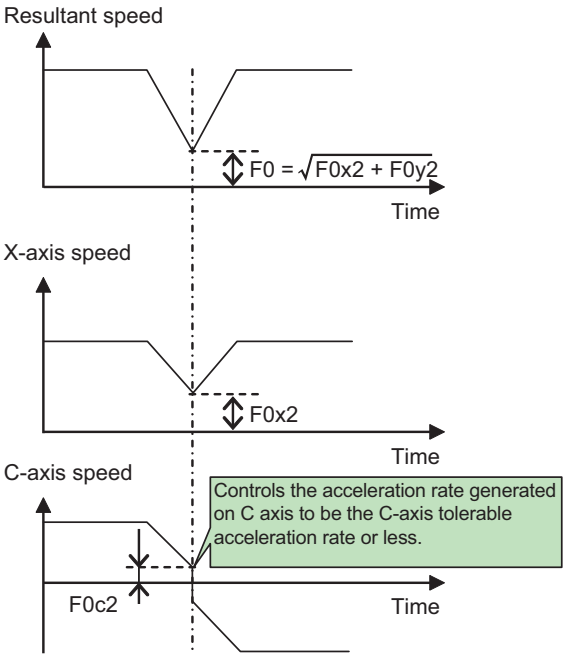
(a) Corner shape which generates the acceleration rate on X axis (linear axis)



(b) Corner shape which generates the acceleration rate on C axis (rotary axis)



(c) Speed pattern which generates the acceleration rate on X axis (linear axis)



(d) Speed pattern which generates the acceleration rate on C axis (rotary axis)

Deceleration is not carried out when blocks are smoothly connected (when the acceleration rate to be generated for each axis is equal to or lower than the tolerable acceleration rate 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 rate can be adjusted for each axis using "#2159 compx" (accuracy coefficient for each axis), and the tolerable acceleration rate for each axis can be obtained with the following formula. It is necessary, however, to set the same tolerable acceleration rate for all base axes because an arc shape is distorted if it differs among them. If G1bFx is "0" (not set), the tolerable acceleration rate is calculated using "#2001 rapid" (rapid traverse rate). And if G1btLx is "0" (not set), the tolerable acceleration rate is calculated using "#2004 G0tL" (G0 time constant (linear)).
If G1bFx and G1btLx are 0 for all base axes, the tolerable acceleration rate for the base axes are unified to the lowest one.

#8021 COMP CHANGE	Accuracy coefficient used
0	#8019 R COMP
1	#8022 CORNER COMP

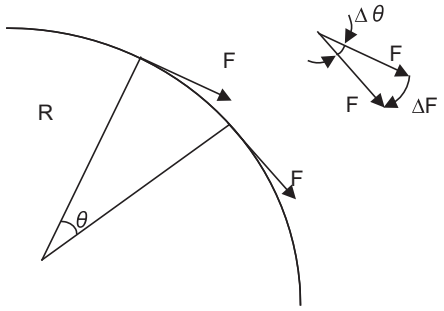
$$\text{Tolerable acceleration rate for each axes } (mm/s^2) = \frac{\text{G1bFx (mm/min)} / 60}{\text{G1btLx (ms)} / 1000} \times \frac{100 - \text{R COMP}}{100} \times \frac{100 - \text{compx}}{100}$$

(3) Arc speed clamp

During circular interpolation, even when moving at a constant speed, acceleration rate 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 rate 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 rate ΔF (mm/s^2) 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 rate ΔF lower than the tolerable acceleration rate common to all axes Ac (mm/s^2) can be obtained with the following formula:



F: Commanded speed (mm/min)
R: Commanded arc radius (mm)
Δθ: 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 rate common to all axes Ac (mm/s^2).

$$F' \leqq \sqrt{R * Ac * 60}$$

$$\Delta F' = \frac{G1bF(mm/min)}{G1btL(ms)}$$

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, Tp is the servo system position loop time constant (s) and Kf is the feed forward coefficient.

Tp is the inverse number to "#2203 PGN1" (position loop gain) (Tp = 1 / PGN1) and Kf is a ratio of "#2010 fws_g" (feed forward gain) (Kf = fwd_g / 100), both of which depend on the MTB specifications.

$$\Delta R = \frac{1}{2R}\left\{Tp^2\left(1 - Kf^2\right)\right\}\left(\frac{F}{60}\right)^2 = \frac{AC}{2}\left\{Tp^2\left(1 - Kf^2\right)\right\}$$

ΔR : Arc radius reduction error amount
Tp : Position loop gain time constant of servo system
Kf : 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.

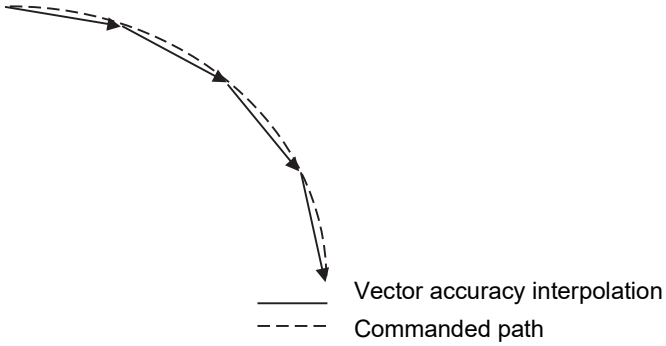
As shown below, different accuracy coefficients can be used depending on the parameter "#8021 COMP_CHANGE", and the tolerable acceleration rate common to all axes can be obtained with the following formula:

#8021 COMP CHANGE	Accuracy coefficient used
0	#8019 R COMP
1	#8022 CORNER COMP #8023 CURVE COMP

$$\text{Tolerable acceleration rate for all axes (mm/s}^2\text{)} = \frac{G1bF\ (mm/min) / 60}{G1btL\ (ms) / 1000} \times \frac{100 - R\ COMP}{100}$$

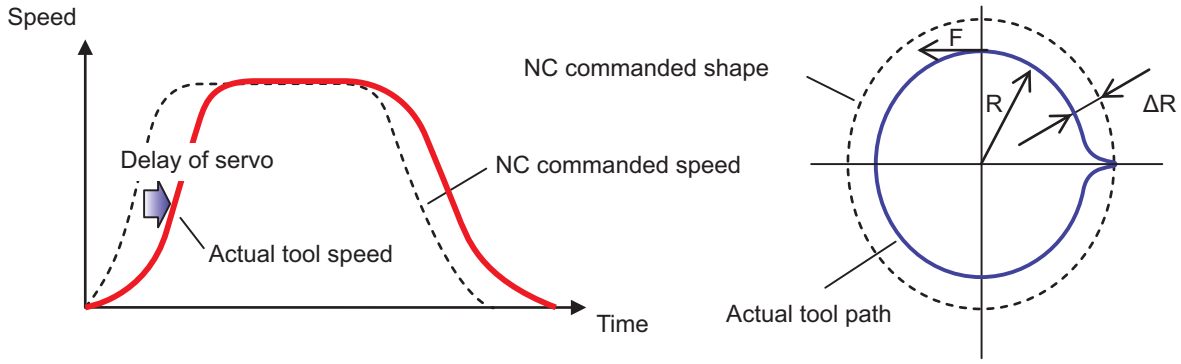
Vector accuracy interpolation

When a micro 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.

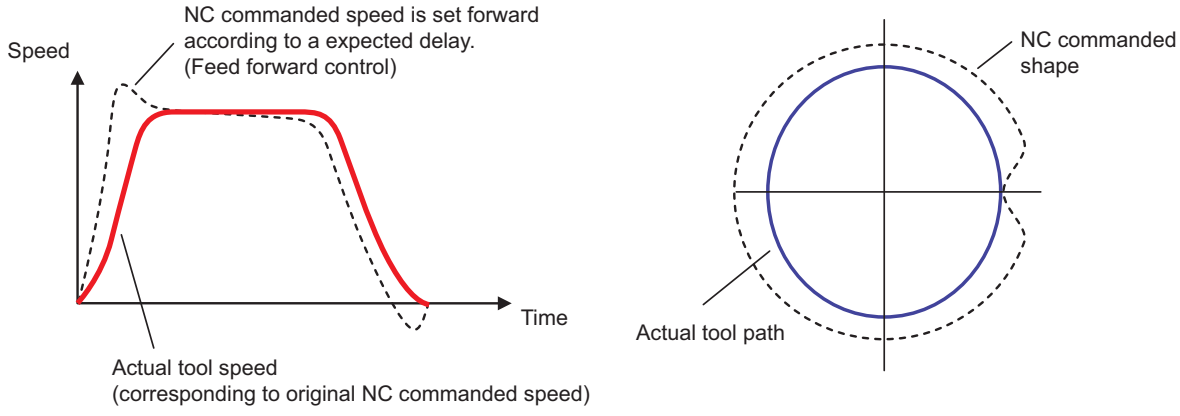


Feed forward control

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

$$\Delta R = \frac{1}{2R}\left\{Tp^2\left(1 - Kf^2\right)\right\}\left(\frac{F}{60}\right)^2$$

Here, Tp is the servo system position loop time constant (s) and Kf is the feed forward coefficient. Tp is the inverse number to "#2203 PGN1" (position loop gain) (Tp = 1 / PGN1) and Kf is a ratio of "#2010 fws_g" (feed forward gain) (Kf = fwd_g / 100), both of which depend on the MTB specifications.

Combination with the smooth high gain (SHG) control function

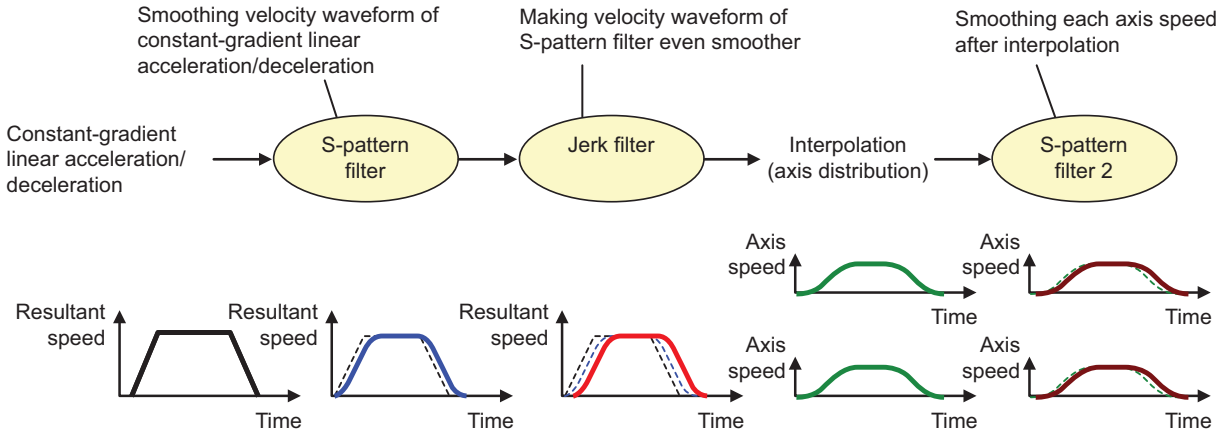
Feed forward control can inhibit path errors more effectively by increasing the feed forward coefficient. In some cases, however, the coefficient cannot be increased because a greater coefficient may cause machine vibrations. In this case, use this function together with the smooth high gain (SHG) control 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

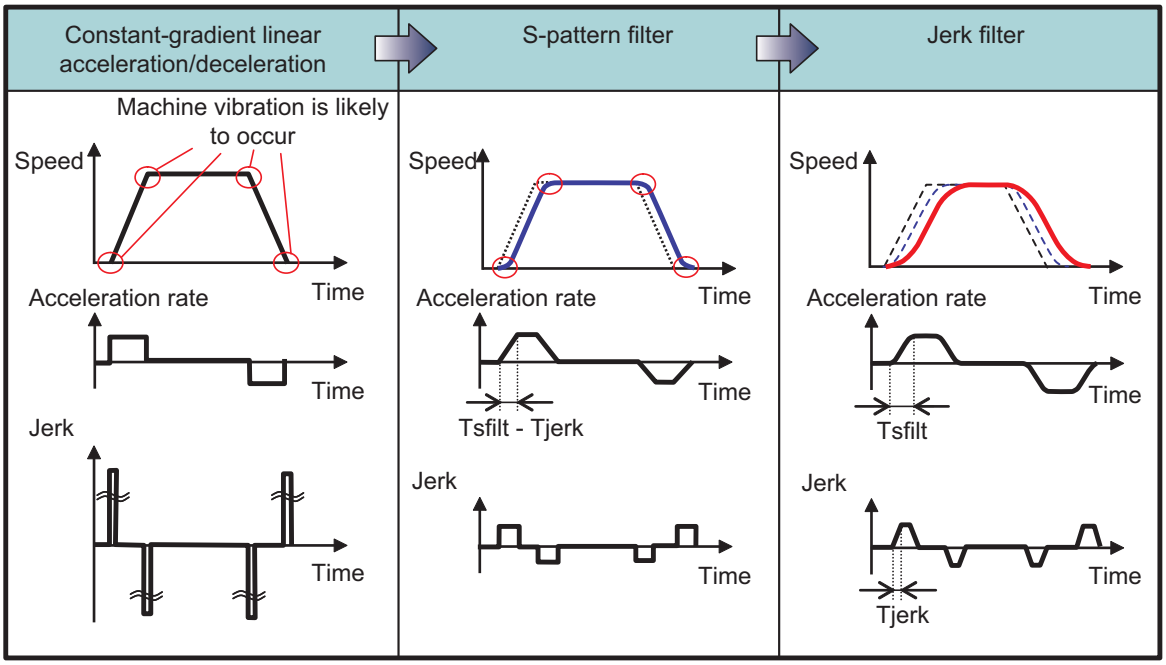


(1) G01/G00 S-pattern filter

This function inhibits the machine vibration by smoothing a velocity waveform generated by constant-gradient linear acceleration/deceleration.
Constant-gradient linear acceleration/deceleration generates continuous velocity waveforms, but makes the acceleration rate discontinuous. As a result, machine vibrations may easily occur when there are discontinuities in acceleration rate, which may cause scratches or streaks on the machining surface. The S-pattern filter can make the velocity waveform even smoother and eliminate acceleration rate 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 increases 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 rate, 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 rate. 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 increases 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 fn (Hz) is known, vibrations can be inhibited effectively by setting the vibration period Tn (ms) obtained with the following formula for the S-pattern filter time constant:

$$Tn = \frac{1000}{fn} \text{ (ms)}$$

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
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 axis-specific acceleration tolerance control (optimum acceleration control) or variable-acceleration pre-interpolation acceleration/deceleration specification is valid.

(2) A program error will occur if high-accuracy control is commanded in the following modes.
- ♦During milling -> Program error (P481) (*2)
- ♦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 will occur if the following commands are issued during the high-accuracy control mode.
- ♦Milling -> Program error (P126) (*2)
- ♦Cylindrical interpolation -> Program error (P126) (*2)
- ♦Polar coordinate interpolation -> Program error (P126) (*2)
- ♦Normal line control -> Program error (P29)

(*2) An error will not occur if the axis-specific acceleration tolerance control (optimum acceleration control) or variable-acceleration pre-interpolation acceleration/deceleration specification is 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

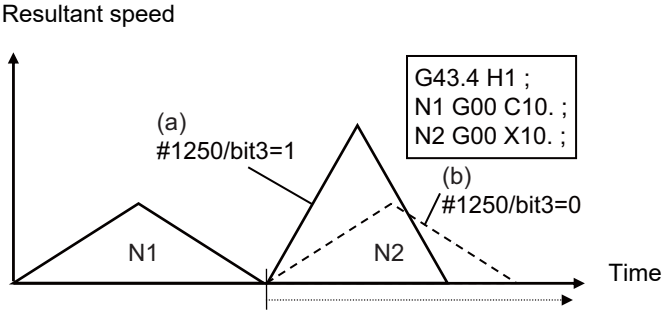
A	B	Operation when B is commanded during A command
G61.1/G08P1	G61.1	Continues high-accuracy control.
	G61,G62,G63,G64	Cancels high-accuracy control and operates in the commanded mode.
	G61.2	Operates in the high-accuracy spline interpolation mode.
	G8P1	Continues high-accuracy control.
	G8P0	Cancels 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.

A	B	Operation when B is commanded during A command
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.
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 rate switching during tool center point control, workpiece installation error compensation, or inclined surface machining command

The acceleration rate 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 rate is used depends on the MTB specifications (parameter "#1250 set22/bit3").

Normally, the acceleration rate of the cutting feed (G01) is used and the acceleration rate in the route direction becomes constant. If the acceleration rate of the rapid traverse (G00) is used, the acceleration rate 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). 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).



During tool center point control

(a) Acceleration rate of rapid traverse (G00)

(b) Acceleration rate of cutting feed (G01)

When at least one of the functions shown in the table below is commanded, positioning is performed with the acceleration rate of the rapid traverse (G00) in the following conditions:

(1) The parameter "#1250 set22/bit3" is set to "1".

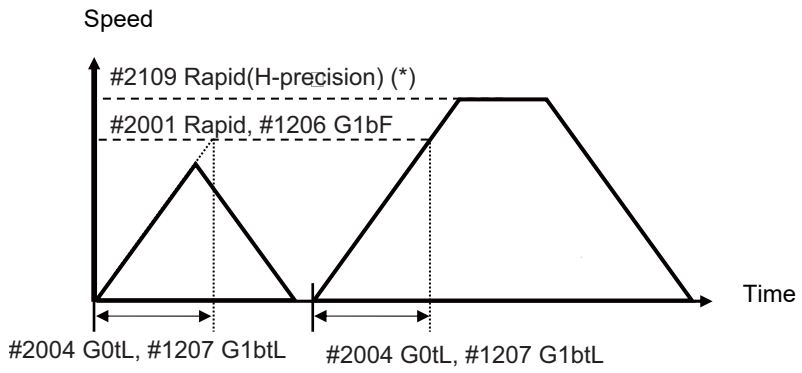
(2) SSS control is being executed.

If the above conditions are not satisfied, the system runs with the acceleration rate 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 rate designation methods under the SSS control. The acceleration rate is calculated based on the designated parameters to perform operations. Designate the parameters to determine the rapid traverse acceleration rate, referring to the table below.

#1250 bit3	Functions in the table above	Rapid traverse constant-gradient multi-step acceleration/deceleration	Acceleration/deceleration pattern	Acceleration rate designation method
0	Not commanded.	Invalid	Constant-gradient acceleration/deceleration	Designated with the parameters "#2001 Rapid" and "#2004 G0tL".
		Valid	Constant-gradient multi-step acceleration/deceleration	Follows the rapid traverse constant-gradient multi-step acceleration/deceleration specifications.
	Commanded.	Invalid/Valid	Constant-gradient acceleration/deceleration	Designated with the parameters "#1206 G1bF" and "#1207 G1btL".
1	Not commanded.	Invalid	Constant-gradient acceleration/deceleration	Designated with the parameters "#2001 Rapid" and "#2004 G0tL".
		Valid	Constant-gradient multi-step acceleration/deceleration	Follows the rapid traverse constant-gradient multi-step acceleration/deceleration specifications.
	Commanded.	Invalid/Valid	Constant-gradient acceleration/deceleration	Designated with the parameters "#2001 Rapid" and "#2004 G0tL".

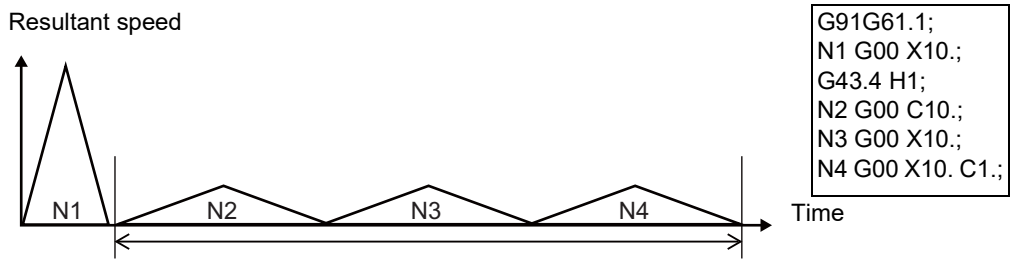


(*) When "#2109 Rapid(H-precision)" (rapid traverse rate for high-accuracy control mode) is set to "0", "#2001 rapid" is used.

The following describes the relationship between the setting values of the parameter "#1250 set22/bit3" and the applied acceleration rate 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"

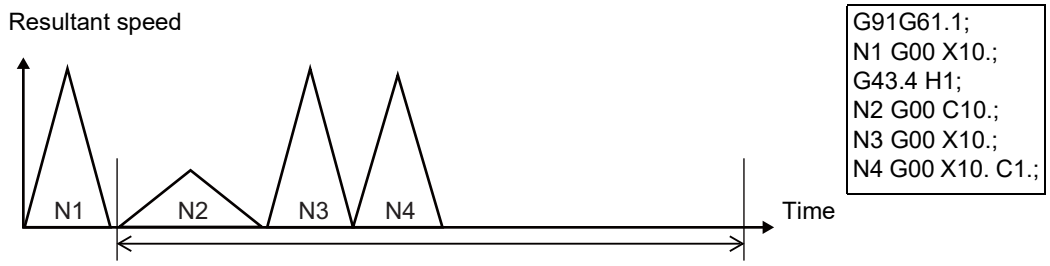


During tool center point control

N1: Runs with the acceleration rate calculated from "#2001" and "#2004" of the X axis.

N2-N4: Runs with the acceleration rate calculated from "#1206" and "#1207".

(2) Acceleration/deceleration pattern when "#1250 bit3" is set to "1"



During tool center point control

N1: Runs with the acceleration rate calculated from "#2001" and "#2004" of the X axis.

N2-N4: Runs with the acceleration rate calculated from "#2001" and "#2004" of each axis.
(Calculate the optimum acceleration rate 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) "G08P1" must be commanded alone in a block, which also applies to "G08P0".

(3) 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.

(4) 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.

(5) 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.

(6) 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.

(7) When there are high-accuracy acceleration/deceleration time constant expansion specifications, the sampling buffer area may be smaller.

(8) 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.

(9) 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.

(10) 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.

(11) 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.

(12) When SSS is enabled, the feedrate is controlled so that it will be the optimum value based on the global path information. This means that the actual feedrate may be different from the speed commanded in the machining program.

(13) When the "Manual/Automatic simultaneous valid n-th axis" signal (Y920) is changed during the execution of the movement blocks for the pre-interpolation acceleration/deceleration, the change will not be enabled immediately even if the axis is not moving. The change is enabled when all the axes in the part system decelerate and stop.

17.2.2 SSS Control



Function and purpose

This function runs a machining program that approximates a freely curved surface with micro 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 micro 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".

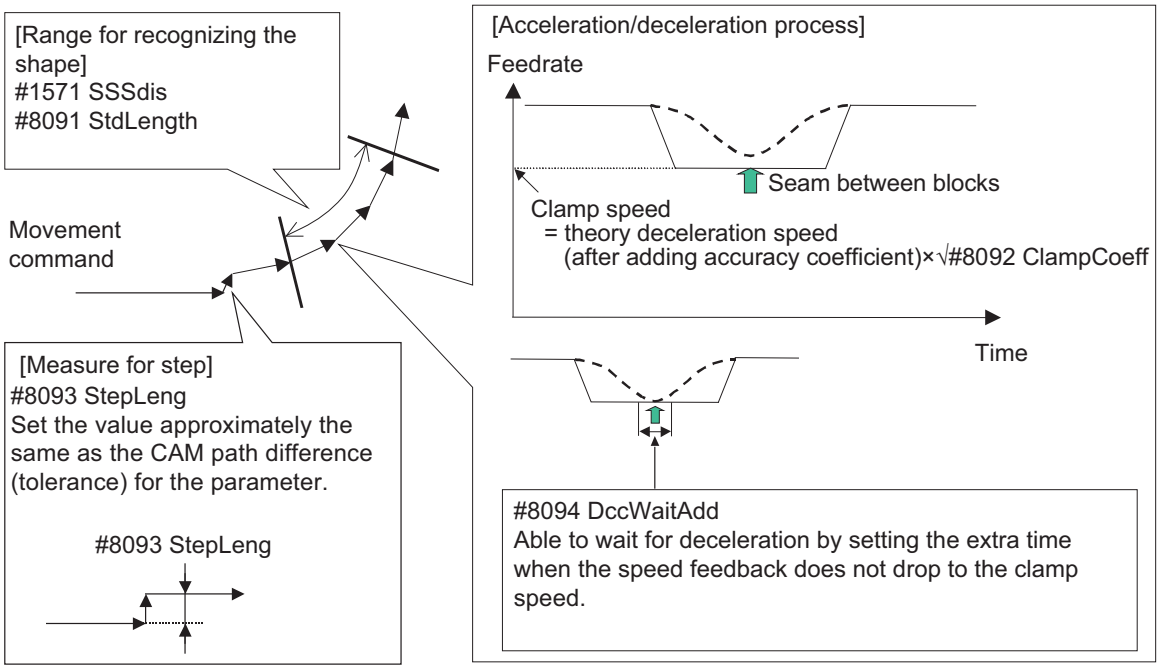
(2) Basic specification parameters (depend on the MTB specifications)

#	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
- Hypothetical axis interpolation
- Automatic tool length measurement
- Tool length compensation along the tool axis

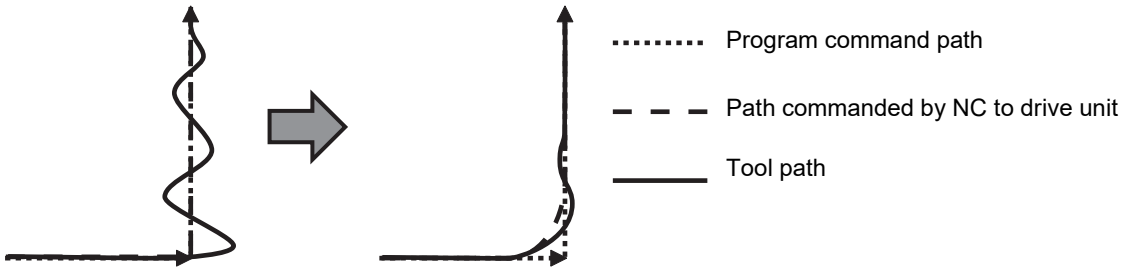
(8) There are some restrictions for each high-accuracy control. Refer to each section for restrictions.
- "17.2 High-accuracy Control"
- "17.3 High-speed High-accuracy Control"

(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.



Tolerance control: Invalid Tolerance control: Valid

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)

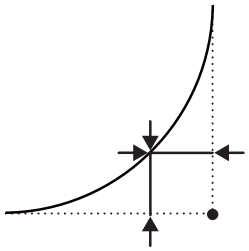
- The range of the command value is 0.000 to 100.000. If a value exceeding the range is commanded, a program error (P35) will occur.
- The tolerance designated by ",K" is applied to all axes in the part system.
- When "0" is designated or ",K" is omitted, the program runs based on the value of the parameter "#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.

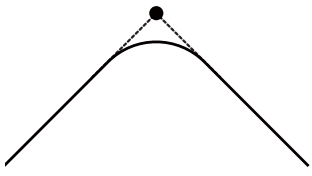
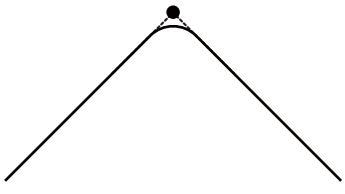
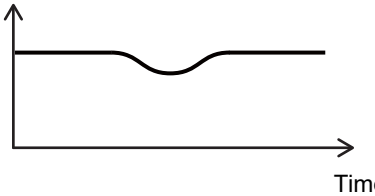
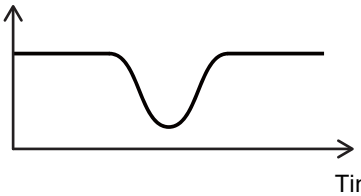
Detailed description

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 axis-specific acceleration tolerance control, 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).

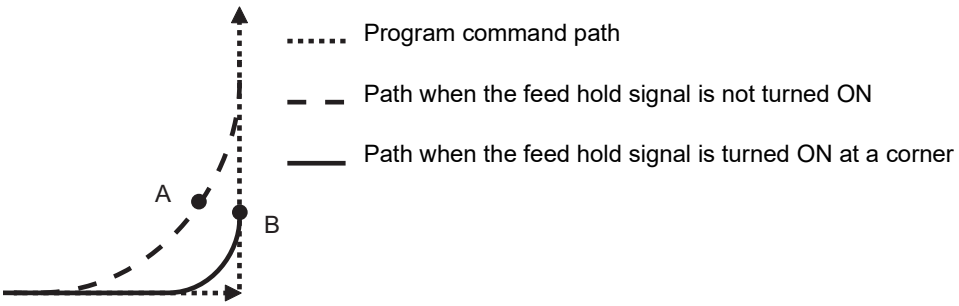
(b) Under single block operation

(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
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 the "Automatic operation "pause" command" signal (YC11) 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.



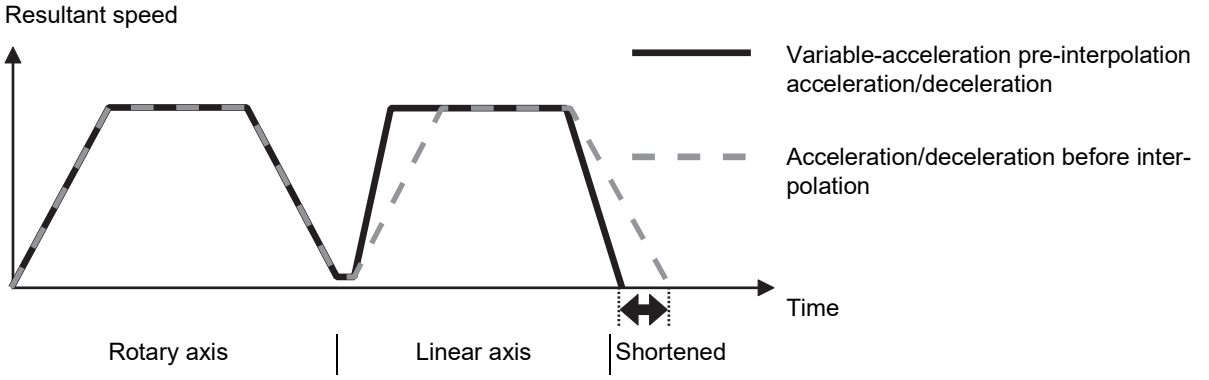
(4) When the tolerance control is enabled (#12066 = "1"), the maximum value of the micro segment processing capability is as follows.

Model	Maximum value of the micro segment processing capability
M800V Series	135 m/min
M80VW/M80V TypeA	100 m/min
M80V TypeB	67.5 m/min

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 rate common to all axes. Therefore, if the high responsiveness and low responsiveness coexist in axes, the acceleration rate 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 rate to each axis. This means that it is possible to set a higher acceleration rate 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 rate 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 rate is calculated using "#1206 G1bF" (maximum speed).
And for an axis with G1btLx = 0 (not set), the acceleration rate 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 rate for "#1206 G1bF" and "#1207 G1btL".

#1206 G1bF	10000 (mm/min)
#1207 G1btL	100 (ms)

It is assumed that only the acceleration rate for the rotary axis is set for "#2157 G1bFx" and "#2158 G1btLx". ("#1206 G1bF" and "#1207 G1btL" are used by not setting the acceleration rate for the linear axis.)

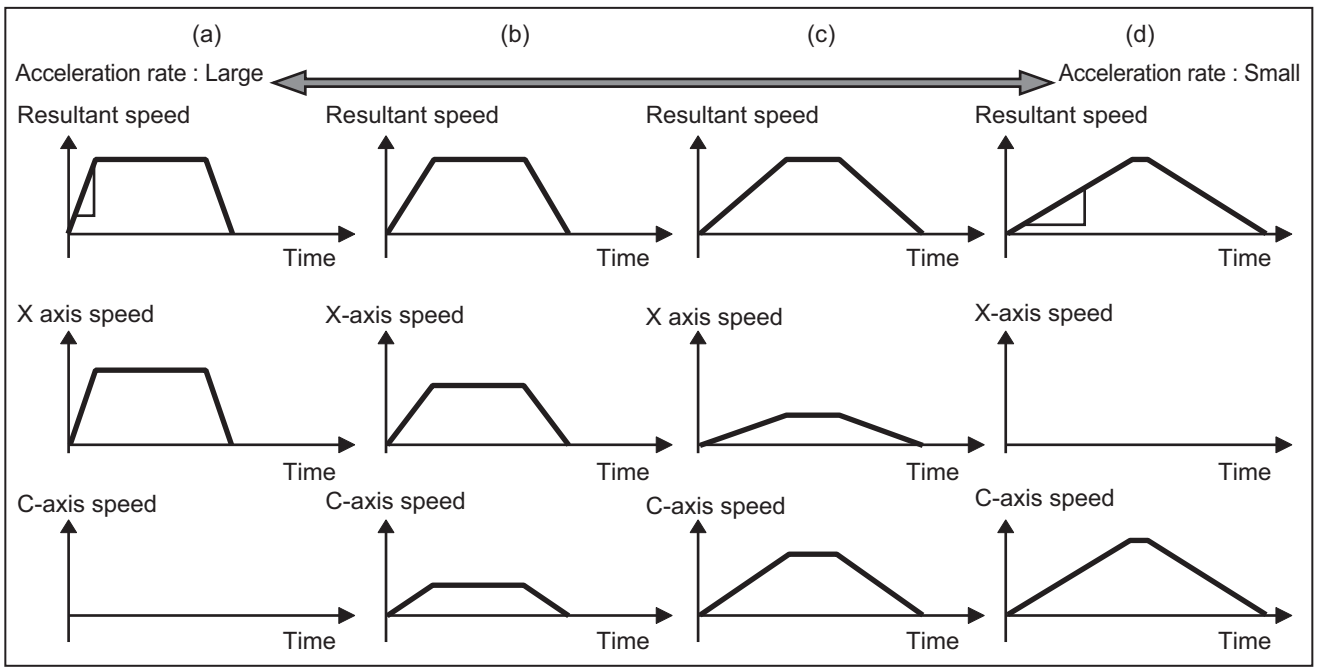
	X	Y	Z	C
#2157 G1bFx	0 (not set)	0 (not set)	0 (not set)	10000 (mm/min)
#2158 G1btLx	0 (not set)	0 (not set)	0 (not set)	500 (ms)

The figure below shows movements with the above settings.

(1) If only the X axis moves, acceleration/deceleration is performed at the acceleration rate set for the X axis. ... (a)

(2) If only the C axis moves, acceleration/deceleration is performed at the acceleration rate set for the C axis. ... (d)

(3) If both of the X and C axes move, acceleration/deceleration is performed at the optimum acceleration rate calculated within the range that the acceleration rate 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)



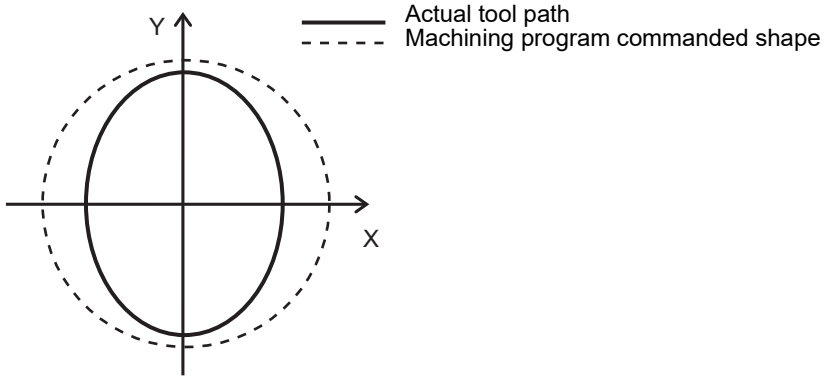
Precautions

(1) Under variable-acceleration pre-interpolation acceleration/deceleration, corner deceleration is realized with the axis-specific acceleration tolerance control.
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 Axis-specific acceleration tolerance control ON	0	1	0	1
Corner deceleration pattern	Optimum corner deceleration	Axis-specific acceleration tolerance control		
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 the axis-specific acceleration tolerance control. In this mode, the acceleration rate 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 rate for base axes I, J, and K. A different acceleration rate causes a distorted shape against an arc command.
The figure below shows an example where the acceleration rate in the Y direction is greater than that in the X direction.



17.2.5 Initial High-accuracy Control

If "#1148 I_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.

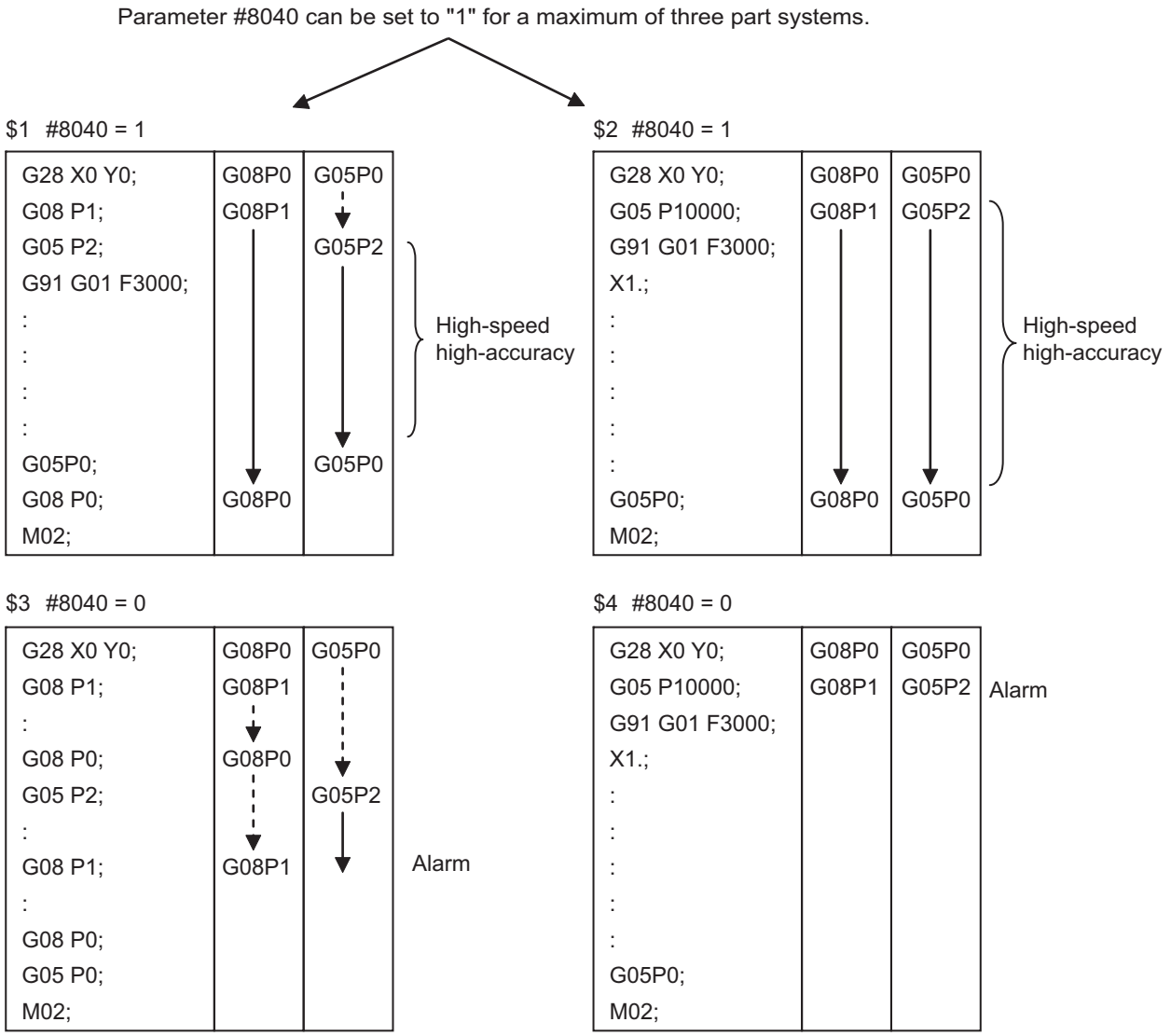
#1148 setting value	Modes enabled at power ON
0	G08P0/G64 (cutting mode) command
1	G08P1/G61.1 (high-accuracy control mode) command
2	G05.1Q1 (high-speed high-accuracy control I mode) command
3	G05P10000 (high-speed high-accuracy control II mode) command
4	G05P20000 (high-speed high-accuracy control 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 High-speed Machining Mode and High-accuracy Control in Multi-part System

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) 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 the parameter "#8040 High-SpeedAcc" can be set to "1" for a maximum of three part systems. If this parameter is set to "1" for four or more part systems, the MCP alarm (Y51 0032) occurs.
If the parameter "#8040 High-SpeedAcc" is set to "0" for all part systems, the simultaneous usage of high-accuracy control and high-speed machining mode is available in the 1st and 2nd part systems.



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 your machine's 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 micro 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 micro 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 micro 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.

Micro segment processing capability

The tables (1) to (3) describe the micro segment processing capability while the high-speed high-accuracy control is enabled. The units of values in the tables is kBPM (kilo Blocks Per Minute), which expresses the number of machining program blocks which can be processed per minute.
The table (1) shows the maximum values of the micro segment processing capability. The micro segment processing capability may decelerate in the conditions with high processing load as follows:

- Four or more axes are being commanded at the same time.
- A macro command or a variable command is included in a command.
- A compensation function such as the tool radius compensation is used.
- A large number of axes are installed.

Micro segment processing capability may be lowered when the acceleration rate and speed are set to such values that the time duration to reach the specified speed exceeds the maximum value (5000 ms) of acceleration/deceleration time constant before interpolation (parameter "#1207 G1btL").

(1) High-speed high-accuracy control I

Number of part systems/ number of axes	Number of part systems (#8040=1)	M850V/M830V	M80VW/M80V TypeA	M80V TypeB
⬧1-part system (8 axes or less)	1 part system	67.5	33.7	33.7
⬧1-part system (9 axes or more) ⬧2-part system	1 part system	67.5	33.7	33.7
	2 part systems	33.7	16.8	-
⬧3-part system	1 part system	33.7 (*7)	16.8	-
	2 part systems	33.7 (*7)	16.8	-
⬧4-part system (16 axes or less)	1 part system	33.7 (*7)	16.8	-
	2 part systems	33.7 (*7)	16.8	-
	3 part systems	16.8 (*7)	16.8	-
⬧4-part system (17 axes or more) ⬧5-part system or more	1 part system	16.8 (*7)	-	-
	2 part systems	16.8 (*7)	-	-
	3 part systems	16.8 (*7)	-	-

(2) High-speed high-accuracy control II

Number of part systems/ number of axes	Number of part systems (#8040=1)	M850V/M830V	M80VW/M80V TypeA	M80V TypeB
⬧1-part system (8 axes or less)	1 part system	168 135 (*6) 101 (*2) 67.5 (*4)(*5)	101 67.5 (*2) 33.7 (*4)(*5)	67.5
⬧1-part system (9 axes or more) ⬧2-part system	1 part system	135 101 (*6) 67.5 (*2) 33.7 (*4)(*5)	101 67.5 (*2) 33.7 (*4)(*5)	67.5
	2 part systems	67.5 33.7 (*4)	33.7	-
⬧3-part system	1 part system	33.7 (*7)	33.7	-
	2 part systems	33.7 (*7)	33.7	-
⬧4-part system (16 axes or less)	1 part system	33.7 (*7)	33.7	-
	2 part systems	33.7 (*7)	33.7	-
	3 part systems	16.8 (*7)	16.8	-
⬧4-part system (17 axes or more) ⬧5-part system or more	1 part system	16.8 (*7)	-	-
	2 part systems	16.8 (*7)	-	-
	3 part systems	16.8 (*7)	-	-

(3) High-speed high-accuracy control III

Number of part systems/ number of axes	Number of part systems (#8040=1)	M850V/M830V	M80VW/M80V TypeA	M80V TypeB
⬧1-part system (8 axes or less)	1 part system	540 (*1) 404 (*3) 270 101 (*2) 67.5 (*4)(*5)	202 67.5 (*2) 33.7 (*4)(*5)	-
⬧1-part system (9 axes or more) ⬧2-part system	1 part system	168 67.5 (*2) 33.7 (*4)(*5)	135 67.5 (*2) 33.7 (*4)(*5)	-
	2 part systems	101 67.5 (*2) 33.7 (*4)	67.5 33.7 (*2)(*4)	-
⬧3-part system	1 part system	33.7 (*7)	33.7	-
	2 part systems	33.7 (*7)	33.7	-
⬧4-part system (16 axes or less)	1 part system	33.7 (*7)	33.7	-
	2 part systems	33.7 (*7)	33.7	-
	3 part systems	16.8 (*7)	16.8	-
⬧4-part system (17 axes or more) ⬧5-part system or more	1 part system	16.8 (*7)	-	-
	2 part systems	16.8 (*7)	-	-
	3 part systems	16.8 (*7)	-	-

(*1) When the parameter "#1468 ctrl period" is set to "1", 540 kBPM is applied here.

(*2) This is the micro segment processing capability in the time constant expansion system or in any of the following modes.
⬧Tool center point control (G43.4/G43.5)
⬧Inclined surface machining (G68.2/G68.3)
⬧Workpiece installation error compensation (G54.4P1 to P7)
⬧3-dimensional tool radius compensation (Tool's vertical-direction compensation) (G41.2/G42.2)
⬧Spatial error compensation (High-speed high-accuracy control II only)
⬧Variable-acceleration pre-interpolation acceleration/deceleration (High-speed high-accuracy control II only)
⬧Spline interpolation 2 (G61.4)
⬧3D machine interference check (High-speed high-accuracy control II only)
⬧Tool cutting point control (G43.8/G43.9)
⬧Rotation center error compensation (High-speed high-accuracy control II only)
⬧Real-time 3D machine interference check (High-speed high-accuracy control II only)

(*3) This is the micro segment processing capability in any of the following modes when the parameter "#1468 ctrl period" is set to "1".
⬧Spatial error compensation
⬧Variable-acceleration pre-interpolation acceleration/deceleration
⬧3D machine interference check
⬧Rotation center error compensation
⬧Real-time 3D machine interference check (High-speed high-accuracy control II only)

(*4) This is the micro segment processing capability when the smooth fairing function is enabled ("#8033 Fairing ON" is set to "2").

(*5) When the smooth fairing function is enabled ("#8033 Fairing ON" is set to "2") and the parameter "#1468 ctrl period" is set to "1", the program error (P29) occurs.

(*6) This is the micro segment processing capability when the tolerance control is enabled ("#12066 Tolerance ctrl ON" is set to "1").

(*7) Optional specification of part system is required separately.

Multi-part system simultaneous high-accuracy control

High-speed high-accuracy control I, II, III can be used simultaneously in up to three 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.

The parameter "#8040 High-SpeedAcc" can be set to "1" for up to three part systems. If four or more part systems are set to "1", an MCP alarm (Y51 0032) occurs. When "1" is set for two or more part systems, the micro 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) 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) is issued from the machining program. Alternatively, command "G05 P2" (high-speed machining mode II) is issued during high-accuracy control.

(*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 combination 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 combination function is enabled
○: The high-speed high-accuracy control I and the combination function are both enabled
Δ: The high-speed high-accuracy control I is temporarily canceled, while the combination function is enabled
×: 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	× (P34)	□ (*2)
	G05P2	High-speed machining mode II ON	□ (*4)	□ (*2)
	G05P10000	High-speed high-accuracy control II ON	× (P34)	× (P34)
	G05P20000	High-speed high-accuracy control III ON	× (P34)	× (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	Δ (*10)	Δ (*10)
	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 (*10)	- (*10)
	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	-

Group	G code	Function name	A	B
0	G50.2/G250	Tool spindle synchronization IB (spindle-spindle, polygon)/IIC (spindle-NC Axis, polygon) cancel	○	○
	G51.2/G251	Tool spindle synchronization IB (spindle-spindle, polygon) ON	○	○
		Tool spindle Synchronization IC (spindle-NC Axis, polygon) ON	Δ	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)
	G113.1	Spindle synchronization OFF	○	○
	G114.1	Spindle synchronization ON	○	○
	G114.2	Tool spindle synchronization IA (spindle-spindle, polygon mode)	○	○
	G114.3	Tool spindle synchronization II (hobbing)	Δ	Δ
	G122	Sub part system control I	Δ (*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 command	○	○
	G91	Incremental 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)

Group	G code	Function name	A	B
8	G43 G44	Tool length offset	○	× (P29)
	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	○	× (P34)
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	○	○
		3-dimensional coordinate conversion ON	○	○
	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 M80V TypeB, the following program errors occur depending on the G codes.

G code	Program error
G05.1Q2, G92.1, G122, G61.2, G61.4	P39
G144, G54.4 P1-P7	P34
G07	P80
G02.1, G03.1	P73
G02.3, G03.3	P611
G02.4, G03.4	P76

G code	Program error
G10 I_J_/G10 K_	P260
G06.2	P550
G93	P124
G41.1, G42.1	P900
G43.1	P930
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 combination 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 combination function is enabled
○: The high-speed high-accuracy control I and the combination function are both enabled
Δ: The high-speed high-accuracy control I is temporarily canceled, while the combination function is enabled
×: 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	-	× (P34)
Mirror image by external input	-	× (P34)
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	○	-
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 M80V TypeB, 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 combination 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 combination function is enabled

○: The high-speed high-accuracy control II and the combination function are both enabled

Δ: The high-speed high-accuracy control II is temporarily canceled, while the combination function is enabled

×: 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	× (P34)	× (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 designation	Δ	-
	G39	Tool radius compensation corner circular command	Δ	-

Group	G code	Function name	A	B
0	G50.2/G250	Tool spindle synchronization IB (spindle-spindle, polygon)/IIC (spindle-NC Axis, polygon) cancel	○	○
	G51.2/G251	Tool spindle synchronization IB (spindle-spindle, polygon) ON	○	○
		Tool spindle Synchronization IC (spindle-NC Axis, polygon) ON	Δ	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 (*8)	- (*8)
	G113.1	Spindle synchronization OFF	○	○
	G114.1	Spindle synchronization ON	○	○
	G114.2	Tool spindle synchronization IA (spindle-spindle, polygon mode)	○	○
	G114.3	Tool spindle synchronization II (hobbing)	Δ	Δ
	G122	Sub part system control I	Δ (*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 command	○	○
	G91	Incremental 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	○	○

Group	G code	Function name	A	B
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	○	○
13	G61	Exact stop check mode	Δ	Δ
	G61.1	High-accuracy control	□ (*3)	□ (*2)
	G61.2	High-accuracy spline	× (P29) (*8)	× (P29) (*8)
	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	○	○
		3-dimensional coordinate conversion ON	×(P922)	×(P921)
	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	× (P34)	× (P481)
	G12.1	Polar coordinate interpolation ON	× (P34)	× (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 M80V TypeB, the following program errors occur depending on the G codes.

G code	Program error
G05.1Q2, G92.1, G122, G61.2, G61.4	P39
G54.4 P1-P7	P34
G07	P80
G02.1, G03.1	P73
G02.3, G03.3	P611
G02.4, G03.4	P76

G code	Program error
G10 I_J_/G10 K_	P260
G06.2	P550
G93	P124
G41.1, G42.1	P900
G43.1	P930
G43.4, G43.5	P940
G68.2, G68.3	P950

(2) Relationship between the high-speed high-accuracy control II and functions other than G codes
Column A: Operation when the combination 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 combination function is enabled
○: The high-speed high-accuracy control II and the combination function are both enabled
Δ: The high-speed high-accuracy control II is temporarily canceled, while the combination function is enabled
×: 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
Mirror image by external input	-	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 M80V TypeB, 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 combination 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 combination function is enabled

○: The high-speed high-accuracy control III and the combination function are both enabled

Δ: The high-speed high-accuracy control III is temporarily canceled, while the combination function is enabled

×: 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	× (P34)	× (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_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	-
	G50.2/G250	Tool spindle synchronization IB (spindle-spindle, polygon)/IIC (spindle-NC Axis, polygon) cancel	○	○
	G51.2/G251	Tool spindle synchronization IB (spindle-spindle, polygon) ON	○	○
		Tool spindle Synchronization IC (spindle-NC Axis, polygon) ON	Δ	Δ

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	□ (*5)	□ (*6)
	G92	Coordinate system setting	Δ	-
	G92.1	Workpiece coordinate system preset	Δ	-
	G113.1	Spindle synchronization OFF	○	○
	G114.1	Spindle synchronization ON	○	○
	G114.2	Tool spindle synchronization IA (spindle-spindle, polygon mode)	○	○
	G114.3	Tool spindle synchronization II (hobbing)	Δ	Δ
	G122	Sub part system control I	Δ	□ (*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 command	○	○
	G91	Incremental 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	□ (*9)	○
	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)

Group	G code	Function name	A	B
12	G54-G59 G54.1	Workpiece coordinate system selection	○	○
13	G61	Exact stop check mode	□ (*4)	□ (*4)
	G61.1	High-accuracy control	□ (*4)	□ (*2)
	G61.2	High-accuracy spline	× (P29)	× (P29)
	G61.4	Spline interpolation 2	□ (*4)	□ (*4)
	G62	Automatic corner override	□ (*4)	□ (*4)
	G63	Tapping mode	□ (*4)	□ (*4)
	G64	Cutting mode	□ (*4)	□ (*2)
14	G66 G66.1	User macro modal call	Δ	□ (*4)
	G67	User macro modal call cancel	○	○
15	G40.1	Normal line control cancel	× (P29)	○
	G41.1 G42.1	Normal line control	× (P29)	□ (*4)
16	G68	Coordinate rotation by program ON	□ (*4)	□ (*4)
		3-dimensional coordinate conversion ON	× (P922)	× (P921)
	G68.2 G68.3	Inclined surface machining command	Δ	□ (*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	□ (*4)
19	G50.1	Mirror image OFF	○	○
	G51.1	Mirror image ON	Δ	□ (*4)
21	G07.1	Cylindrical interpolation	× (P34)	× (P481)
	G12.1	Polar coordinate interpolation ON	× (P34)	× (P481)
	G13.1	Polar coordinate interpolation OFF	○	○
27	G54.4P0	Workpiece installation error compensation cancel	○	○
	G54.4P1-P7	Workpiece installation error compensation	Δ	□ (*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.

(*9) When the parameter "#1271 ext07/bit4" is set to "1", the compensation vector is retained until positioning is commanded even after the cancel command (G40) has been issued for the tool radius compensation. This condition enables the high-speed high-accuracy control II. When the compensation vector is set to "0" in accordance with the positioning command, the high-speed high-accuracy control III is enabled.

(2) Relationship between the high-speed high-accuracy control III and functions other than G codes
Column A: Operation when the combination 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 combination function is enabled
○: The high-speed high-accuracy control III and the combination function are both enabled
Δ: The high-speed high-accuracy control III is temporarily canceled, while the combination function is enabled
×: 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
Mirror image by external input	-	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.

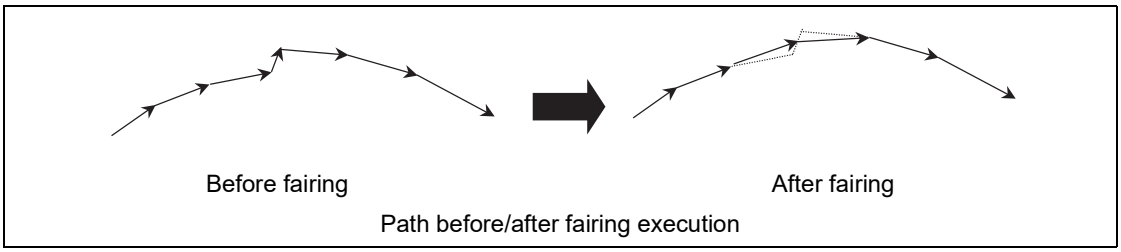
(*10) The high-speed high-accuracy control III is disabled even if "G05P20000" is commanded in a figure rotation subprogram.

17.3.2 Fairing

Function and purpose

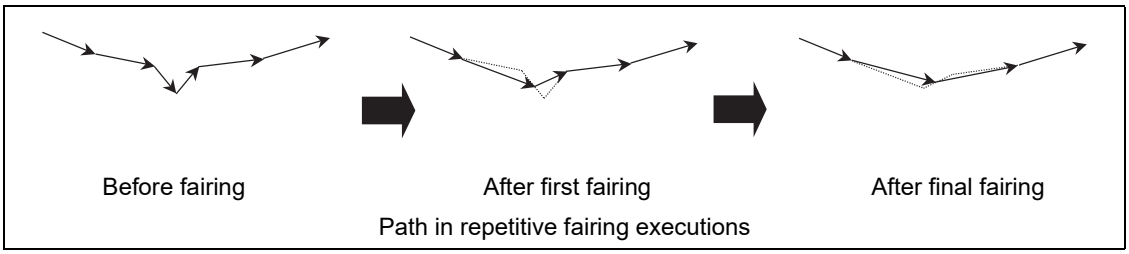
This function is an additional function when the high-speed high-accuracy control II or III mode is ON
If there is a protrusion in a path (zigzagging path) in a machining program generated with a CAM, etc., this function can be used to eliminate the protruding path smaller than the setting value so that 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.



Path before/after fairing execution

If there is any protruding path after fairing, fairing is repeated.



Path in repetitive fairing executions

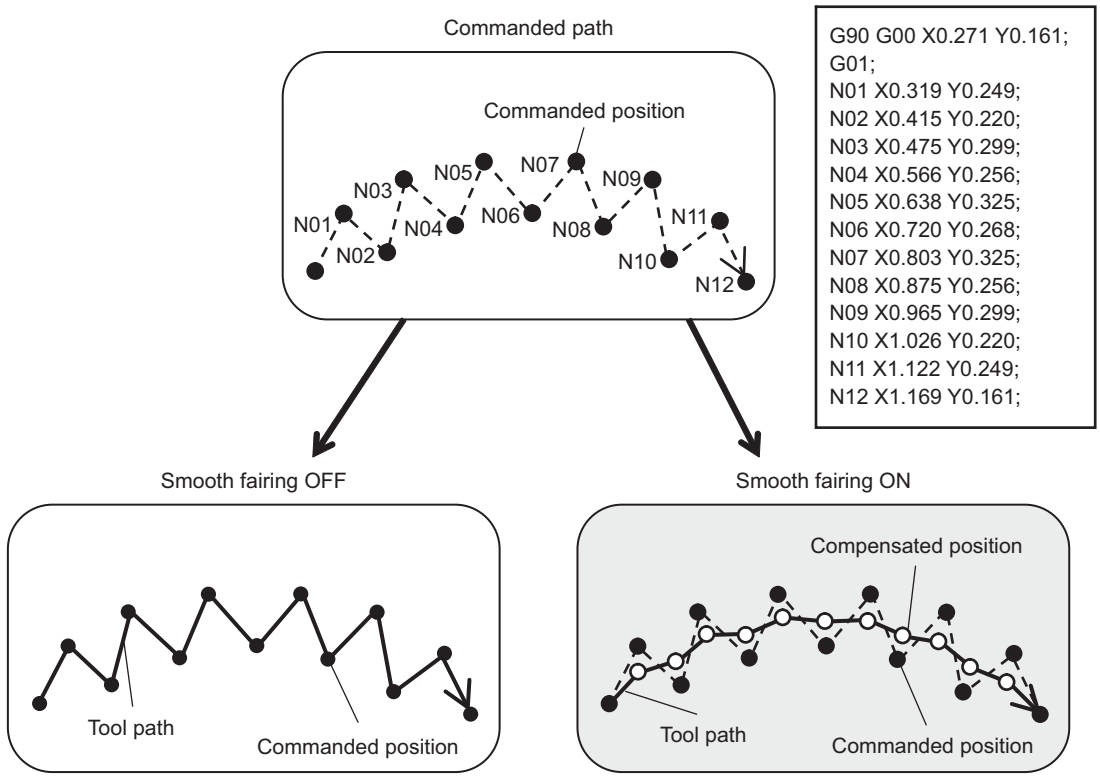
17.3.3 Smooth Fairing

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 smoothen by compensating commanded positions of a machining program.
This function is useful when executing a micro 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

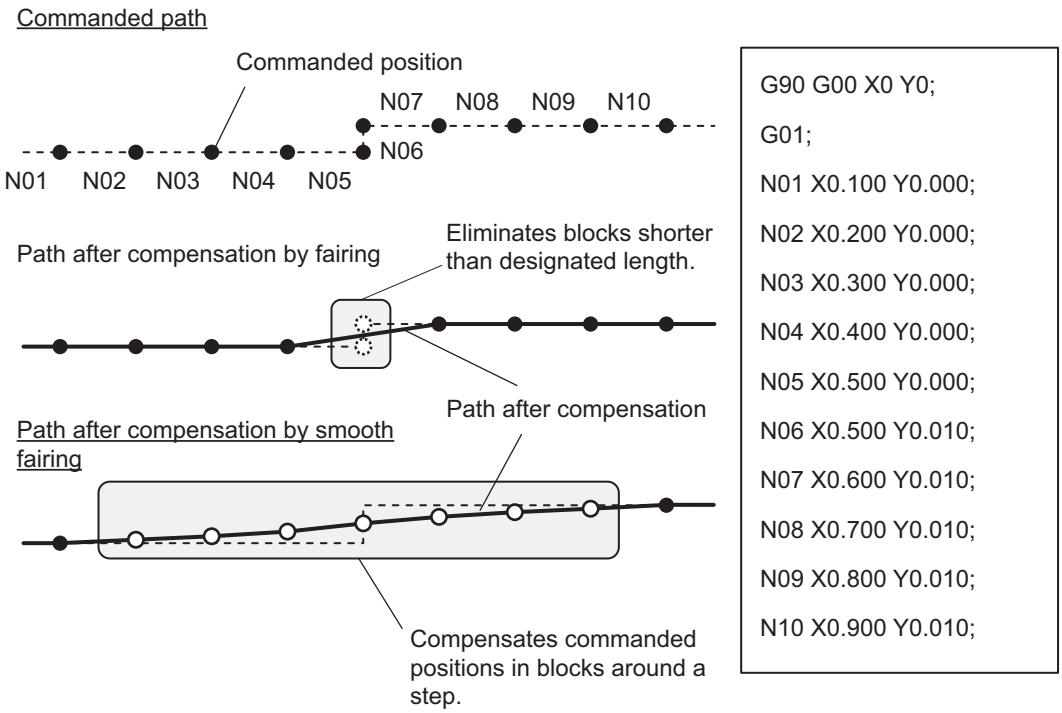
(1) High-speed high-accuracy control III functions as high-speed high-accuracy control II while smooth fairing is ON.



Faring and smooth faring 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 micro 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".
- One of "G05 P20000 ,R1", "G05 P10000 ,R1", and "G05 P2 ,R1" is commanded.

(*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 G05 P10000 G05 P2 Command	No ,R	×	○	●
	,R0	×	×	×
	,R1	●	●	●

●: Smooth fairing ON, ○: 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

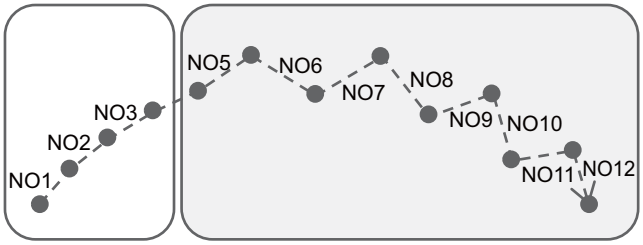
Details of Operation

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.

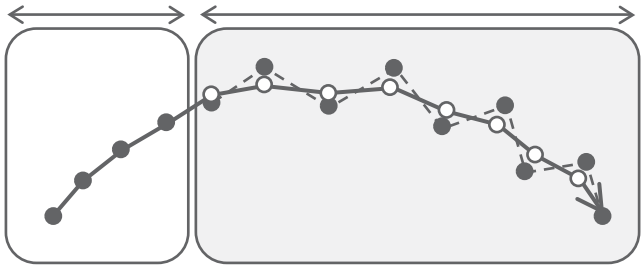
[Commanded path] (indicated by dashed lines)




```
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.

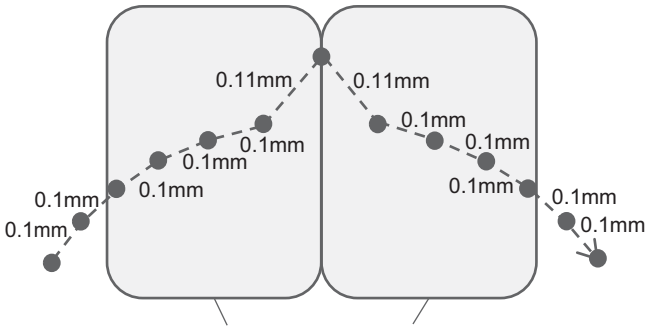


○: Compensated position

●: 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:

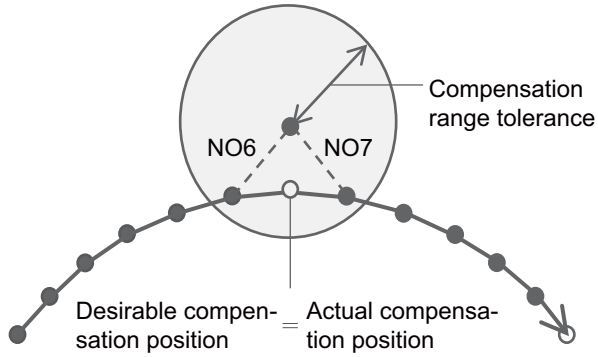


The path is recognized in the range of 0.5 mm forward and 0.5 mm backward of the commanded position.

```
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;
```

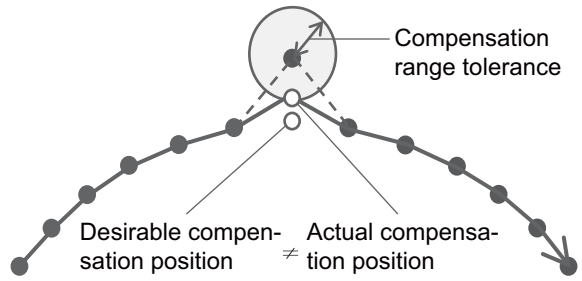
(3) The upper limit of the compensation distance can be determined so that the compensated position does not deviate from the commanded position significantly. Designate this upper limit in the parameter "#8039 Comp. range limit". Ordinarily, designate the tolerance that is designated when generating 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;
```

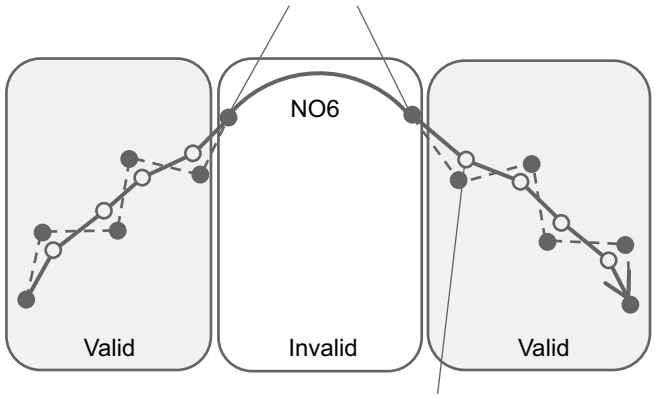
(b) When the compensation distance tolerance is low:




```
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.

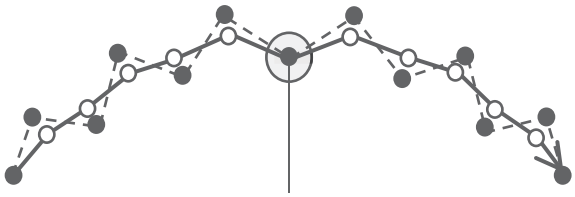



```
G90 G00 X0.0 Y0.0;
G01;
N01 G01 X0.039 Y0.077;
N02 G01 X0.139 Y0.080;
N03 G01 X0.172 Y0.174;
N04 G01 X0.271 Y0.161;
N05 G01 X0.319 Y0.249;
N06 G02 X1.122 Y0.249 R0.5;
N07 G01 X1.169 Y0.161;
N08 G01 X1.268 Y0.174;
N09 G01 X1.301 Y0.080;
N10 G01 X1.401 Y0.077;
N11 G01 X1.441 Y0.000;
```

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/incremental 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: the block where G90 is commanded alone).

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 command/incremental 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	×	×	×
	G61.2	○	○	○
G05 P2	G61.1	×	×	●
	G61.2	○	○	●
G05 P10000 G05 P20000	G5.1 Q0	×	○	●
	G5.1 Q2	×	×	●

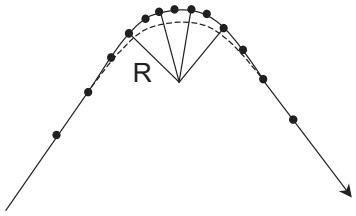
●: Smooth fairing ON, ○: Fairing ON, X: Both OFF

17.3.4 Cutting Speed Clamp with Acceleration Rate Judgment

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 rate 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 value of the acceleration rate is calculated from the parameter "#1206 G1bF" and "#1207 G1btL" setting values. (Tolerable acceleration rate = #1206/#1207)

Related parameter		Details
#8034	AccClampt ON	0 : Clamp the cutting speed with parameter "#2002 clamp" (*1) or the corner deceleration function. 1 : Clamp the cutting speed with acceleration rate judgment.



Speed control by curvature

If the tool moves along the large curvature section without deceleration, a large acceleration rate is generated resulting in a path error by curving inward.

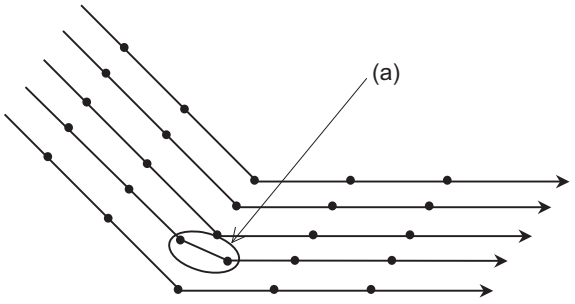
(*1) When a speed is set in "#2109 Clamp(H-precision)", clamp is executed at that speed. When the setting value is "0", clamp is executed with "#2002 clamp".

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 rate 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.



High-speed mode corner deceleration

(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 INP" (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.

(10) When the fairing is valid (#8033 = "1"), if consecutive fairing is executed by the machining program, the micro segment processing capability may become lower than that in the tables (1) to (3) of "Micro segment processing capability" in "17.3.1 High-speed High-accuracy Control I, II, III ; G05.1 Q1/Q0, G05 P10000/P0, G05 P20000/P0".

(11) When the "Manual/Automatic simultaneous valid n-th axis" signal (Y920) is changed during the execution of the movement blocks for the pre-interpolation acceleration/deceleration, the change will not be enabled immediately even if the axes are not moving. The change will be enabled when all the axes in the part system decelerate and stop.

Common precautions on high-speed high-accuracy control II/III (*1)

(*1) "III" is a function only for the machining center.

(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 micro segment processing capability may decelerate. 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 micro segment processing capability does not decelerate.

(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 (P282) 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;" must be commanded alone in a block, which also applies to "G05.1Q0;". If a sequence number other than "N" is commanded, a program error (P33) will occur.

(3) A 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.

(5) To command the high-speed high-accuracy control I during cylindrical interpolation for machining center system, command "G05.1 Q0" to cancel high-speed high-accuracy control I before canceling cylindrical interpolation (G07.1 C0). If the cylindrical interpolation is canceled during the high-speed high-accuracy control I, the program error (P485) occurs.

Precautions on high-speed high-accuracy control II

(1) "G05P10000;" must be commanded alone in a block, which also applies to "G05P0;". If a sequence number other than "N" is commanded, the program error (P33) will occur.

(2) A program error (P33) will occur if the G05 command block does not contain a P command.

(3) The 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 the 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.

(8) A program error (P922) will occur if 3-dimensional coordinate conversion is issued during the high-speed high-accuracy control II.

(9) A program error (P921) will occur if the high-speed high-accuracy control II is commanded during 3-dimensional coordinate conversion.

(10) During coordinate rotation by program (G68) and when the parameter "#19003 PRG coord rot type" is set to "0", if the external workpiece offset is rewritten by the system variables (#5201 or later) during fairing, the next block is executed after decelaration and stop.

Precautions on high-speed high-accuracy control III

(1) "G05P20000;" must be commanded alone in a block, which also applies to "G05P0;". If a sequence number other than "N" is commanded, a program error (P33) will occur.

(2) A program error (P33) will occur if the G05 command block does not contain a P command.

(3) A program error (P33) will occur if the geometric command is issued during the high-speed high-accuracy control III.

(4) 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.

(5) 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.

(6) 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, however, the micro segment processing capability may decelerate.

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 micro segment processing capability described in the tables (1) to (3) of "Micro segment processing capability" in "17.3.1 High-speed High-accuracy Control I, II, III ; G05.1 Q1/Q0, G05 P10000/P0, G05 P20000/P0" may decelerate.

(*3) When the parameter "#1271 ext07/bit4" is "1", the tool radius compensation vector is retained until the positioning command is issued even after the cancel command (G40). In this state, high-speed high-accuracy control II is enabled. When the compensation vector changes to "0" by the positioning command, the high-speed high-accuracy control III is enabled.

(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

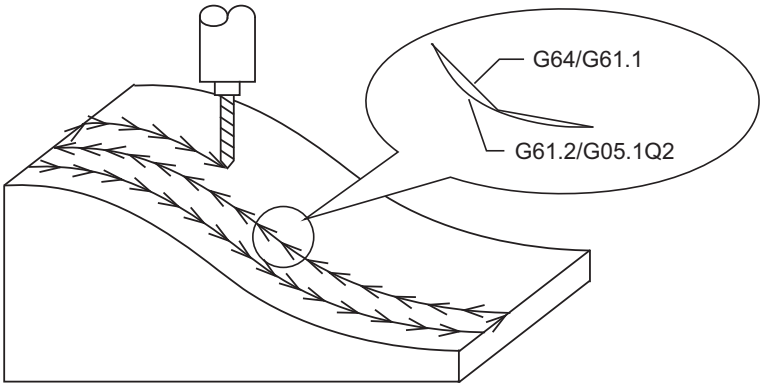
(7) A program error (P922) will occur if 3-dimensional coordinate conversion is issued during the high-speed high-accuracy control III.

(8) A program error (P921) will occur if the high-speed high-accuracy control III is commanded during 3-dimensional coordinate conversion.

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 micro 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 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-gradient 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

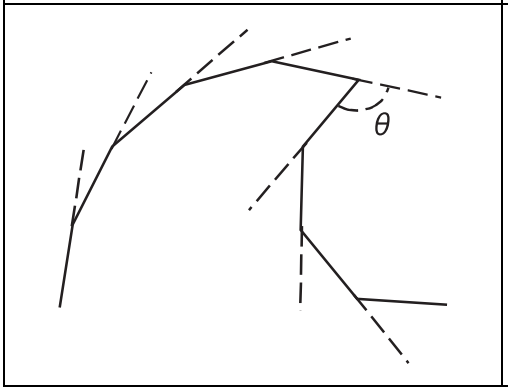
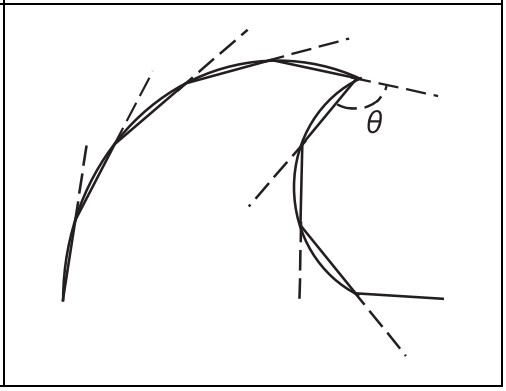
Temporary cancellation of spline interpolation

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.

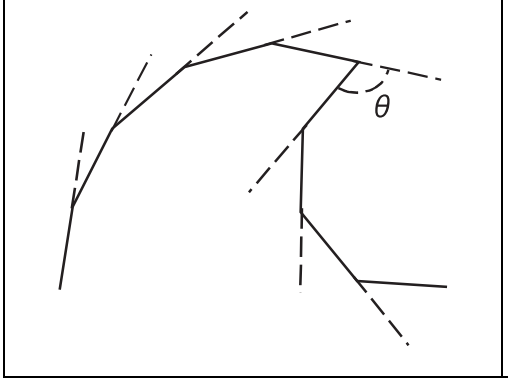
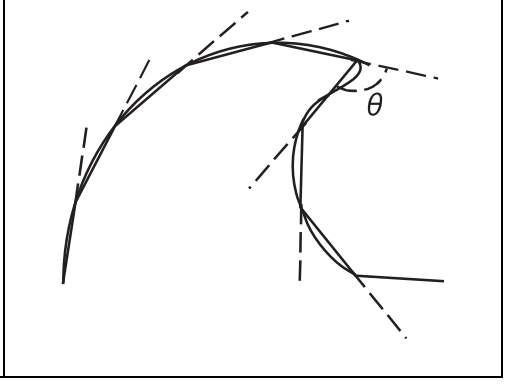
(1) Cancel angle

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°

Programmed command	Spline interpolation path
	

(Example 2) Cancel angle = 0°

Programmed command	Spline interpolation path
	

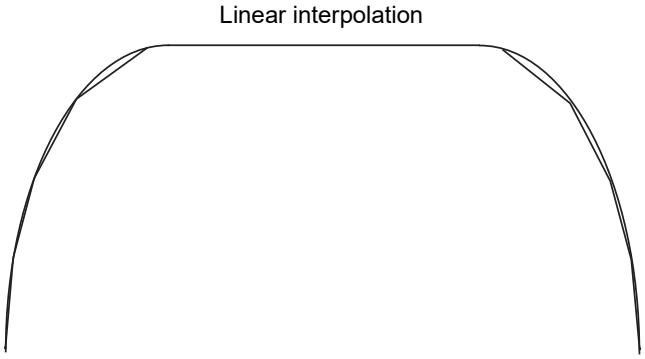
<Note>

- 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." >= "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) Micro segment length

If the movement amount in a block is longer than the parameter "#8030 MINUTE LENGS", the spline function will be temporarily canceled, and the linear interpolation will be executed. When this parameter is not set (= 0), the micro segment length will be 1mm.

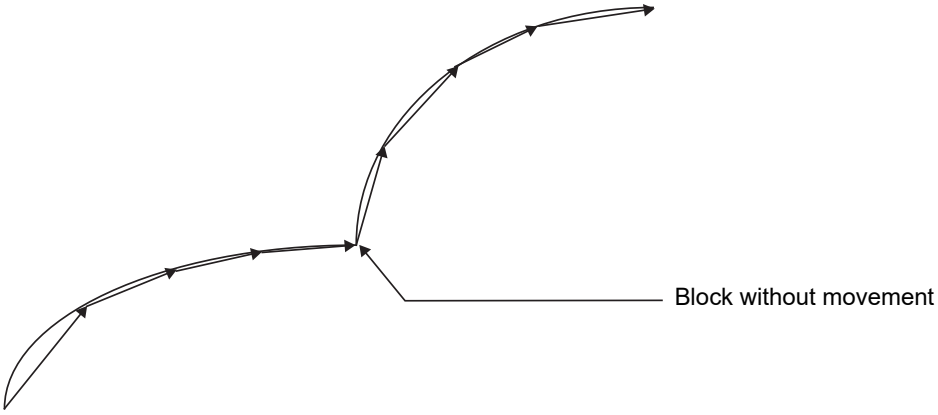
When the segment length in a block > micro segment length (#8030 MINUTE LENGS), the linear interpolation will be executed.



If the micro 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.



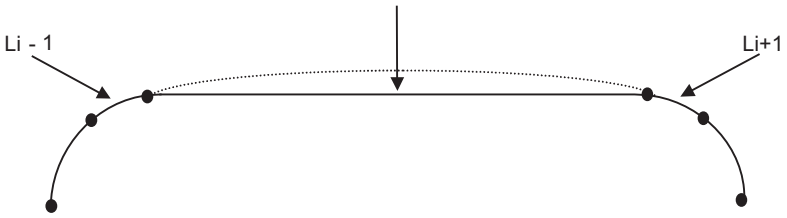
(4) When a block markedly longer than other blocks exists in spline function

Given that the i-th block length is Li 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:

Li > Li-1 x 8 or Li > Li+1 x 8

However, if the parameter "#8030 MINUTE LENGS" is set to "-1", the mode will not be canceled.

Li > Li-1 × 8 or Li > Li + 1 × 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 micro 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 micro 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.

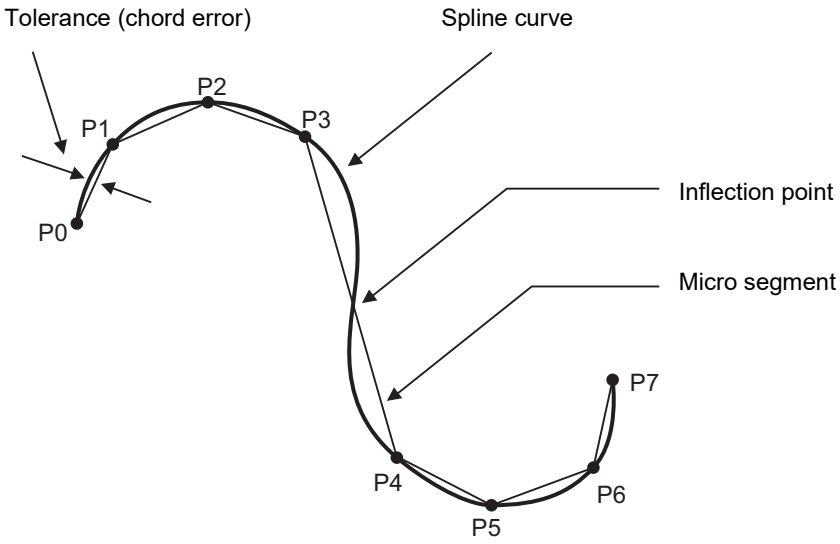


Fig. 1 Spline curve before error correction

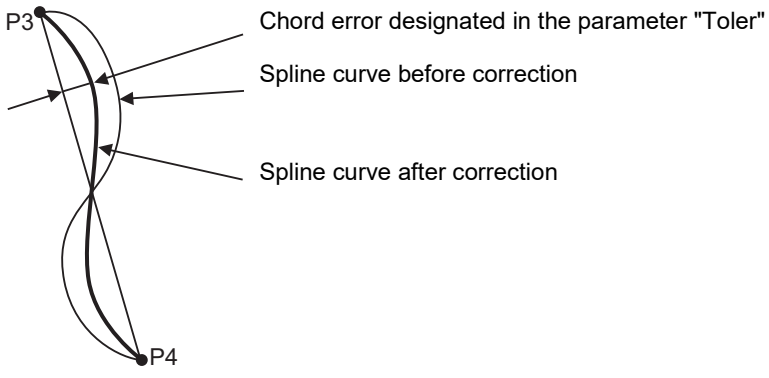


Fig. 2 Spline curve after error correction

In parameter "#8027 Toler-1", set the tolerance when developed into micro 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 micro 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.

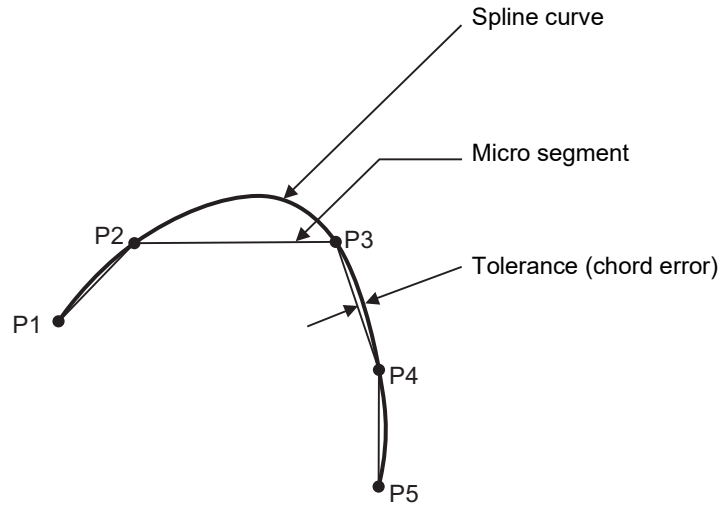


Fig. 3 Spline curve before error correction

Chord error designated in the parameter "Toler-2"

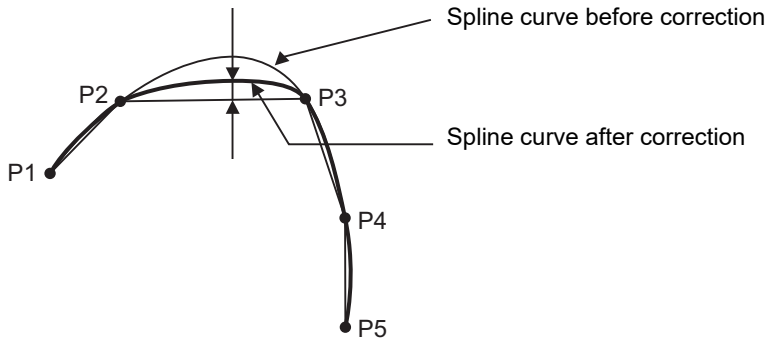
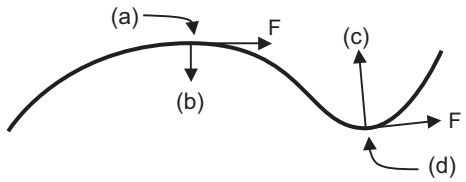


Fig. 4 Spline curve after error correction

In parameter "Toler-2", set the tolerance when developed into micro segments with the CAM.

Curvature speed clamp

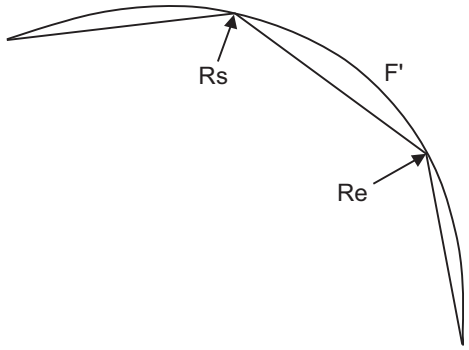
The commanded speed F for the spline function during a segment linear arc will be the speed commanded in the previously set modal. However, if the axis is fed with the same speed, excessive acceleration rate may occur at the sections where the curvature is large (where curvature radius is small) as shown below. Thus, the speed clamp will be applied.



(a) Curvature small
(b) Acceleration rate small
(c) Acceleration rate large
(d) Curvature large
F: Feed command speed (mm/min)

Change of acceleration rate by 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 Rs at the curve block start point and the curvature radius Re 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.



Rs : Block start point curvature radius (mm)
Re : Block end point curvature radius (mm)
R : Block main curvature radius (mm) (smaller one of Rs and Re)
ΔV : Tolerable value of pre-interpolation acceleration/deceleration
F : Clamp speed (mm/min)

$$F' = \sqrt{R \times \Delta V \times 60 \times 1000} \times \frac{100 - Ks}{100} \quad \cdots (1)$$

$$\Delta V = \frac{G1bF(mm/min)}{G1btL(ms)}$$

G1bF : Target pre-interpolation acceleration/deceleration
G1btL : Acceleration/deceleration time to reach the target speed
Ks: 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 LENGS".
- ⬧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 LENGS", 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) "G05.1 Q2" must be commanded alone in a block, which also applies to "G05.1 Q0".
A program error (P33) will occur if it is not commanded alone in a block.

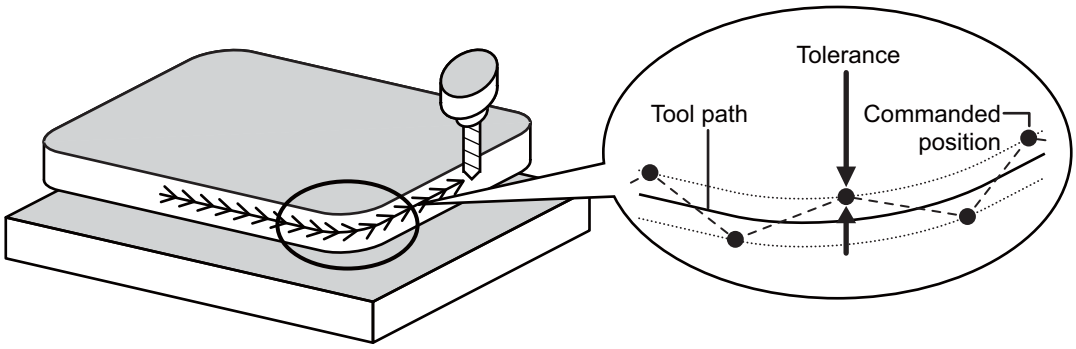
(4) The program error (P33) will occur if the G05.1 command block does not contain a Q command.

(5) A program error (P34) will occur if the number of axis in the part system does not exceed 3.

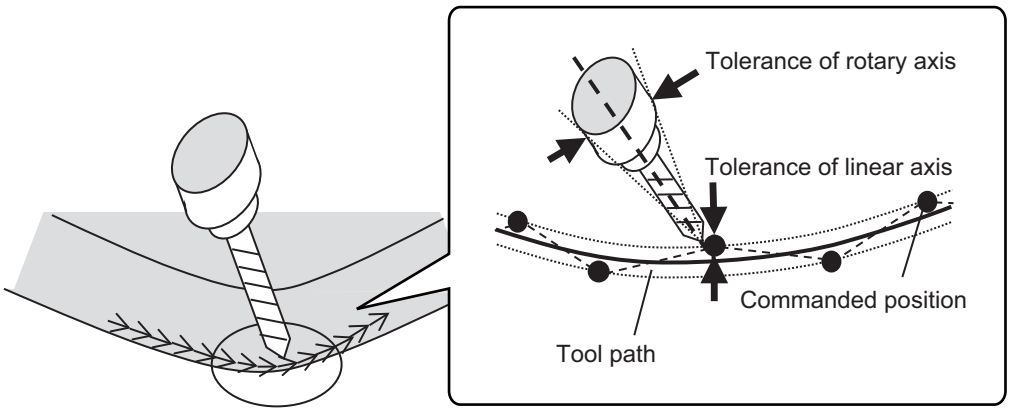
17.5 Spline Interpolation 2; G61.4

Function and purpose

This function automatically generates a curve that smoothly passes within the tolerance range and interpolates the path along this curve. By using this function, smooth machining can be achieved even when the speed commanded in the program is low or the program has minute steps.
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, or tool cutting 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.



Enabling conditions

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) The tool center point control ("G43.4" or "G43.5") or the tool cutting point control ("G43.8" or "G43.9) is commanded (*3).

(*1) If G61.4 is commanded while tolerance control is invalid, the program error (P34) occurs.

(*2) If G61.4 is commanded while the specifications of spline interpolation 2 are not defined, the program error (P39) occurs.

(*3) If G43.4 or G43.5 is given while the tool center point control specification is not provided, the program error (P39) occurs. When the tool cutting point control is not available and G43.8 or G43.9 is commanded, the program error (P910) occurs.
For details on spline interpolation 2 using 5 axes, refer to "Behavior under tool center point control" in "Details of operation".

Enabling conditions with parameters

When items (1) to (3) in the above "Enabling conditions" are satisfied, G group 13 modal can be set to spline interpolation 2 immediately after the power is turned ON next time by setting "1" in the parameter "#8180 Ini spline intr-pl2".
When #8180 is "1", the G group 13 modal changes to spline interpolation 2 (G61.4) regardless of the parameter "#1148 Initial hi-precis".

[When #8180 is "1"]

#1148 setting value		Modal status at power-on
0	Cutting mode	Spline interpolation 2 mode
1	High-accuracy mode	Spline interpolation 2 mode
2	High-speed high-accuracy control I mode	Spline interpolation 2 mode High-speed high-accuracy control I mode
3	High-speed high-accuracy control II mode	Spline interpolation 2 mode High-speed high-accuracy control II mode
4	High-speed high-accuracy control III mode	Spline interpolation 2 mode High-speed high-accuracy control III mode

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 (High-accuracy spline interpolation)
- 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, the program error (P35) occurs.

(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]

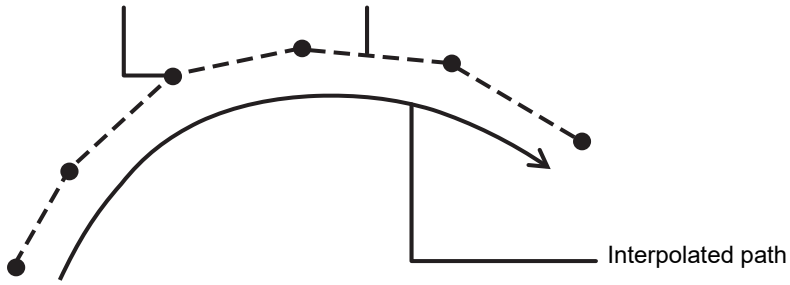
: G91 ;	
G61.4 ,K0.02;	Designate the tolerance 0.02 (mm).
G01 X0.1 Z0.1 F1000 ; X0.1 Z-0.2 ; Y0.1 ;	**Tolerance: 0.02 (mm)**
G61.4 ,K0;	Designate the tolerance 0 [mm].
X-0.1 Z-0.05 ; X-0.1 Z-0.3 ;	**Tolerance: Follows parameter "#2659 tolerance".**
G64 ; :	

Details of operation

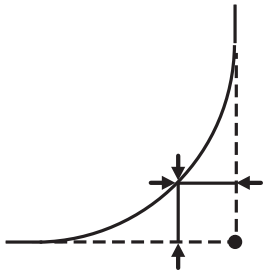
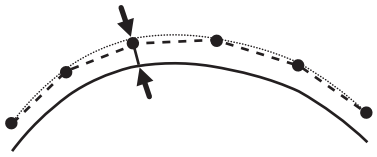
Basic operations

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.

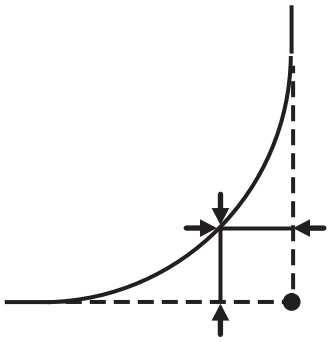
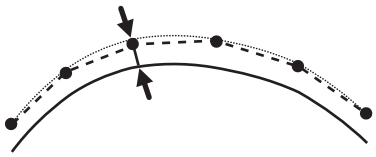
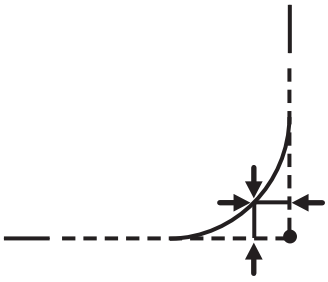
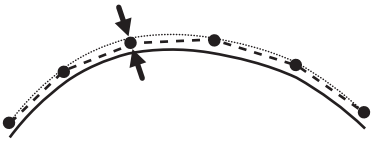
Program command point Program command path



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.

[For corner]	[For curve]
	

The interpolated path varies depending on the tolerance as shown below.

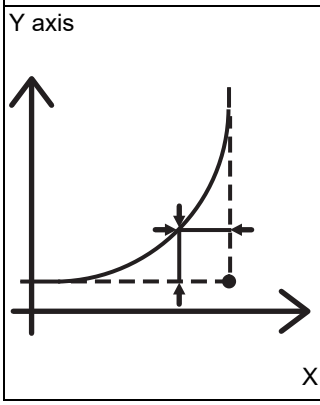
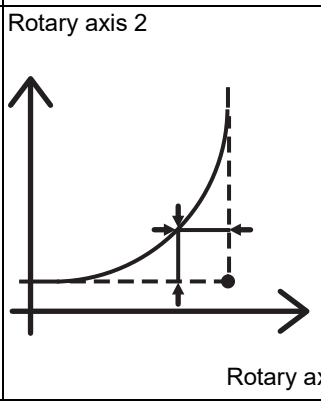
	For corner	For curve
Tolerance: High		
Tolerance: Low		

Behavior under tool center point control/tool cutting point control

Spline interpolation 2 can be used together with tool center point control (G43.4/G43.5) or tool cutting point control (G43.8/G43.9). 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 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.

Tool center point	Tool posture
	

The points to note when tool center point control /tool cutting point control is used together with spline interpolation 2 are as follows:

(1) The tool center point control specification is necessary to enable the tool center point control.

(2) The tool cutting point control specification is necessary to enable the tool cutting point control.

(3) 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.

(4) Any G code that is unavailable under tool center point control cannot be commanded.

(5) The combination with the tool center point control is supported only in M850VW/M850VS. When the spline interpolation 2 and the tool center point control (G43.4/G43.5) are used in combination with a model other than M850VW/M850VS, the program error (P39) occurs.

[Program example]

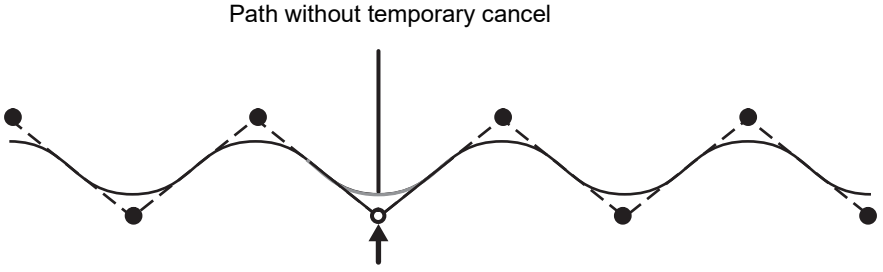
: G91 ;	
G43.4 H1;	Tool center point control ON
G61.4 ;	Spline interpolation 2 mode ON
G01 X0.1 Z0.1 F1000 ; X0.1 Z-0.2 ; Y0.1 ; X-0.1 Z-0.05 ; X-0.1 Z-0.3 ;	Spline interpolation 2 mode using 5 axes
G61.1 ;	Spline interpolation 2 mode OFF
G49 ; :	Tool center point control OFF

Temporary cancel

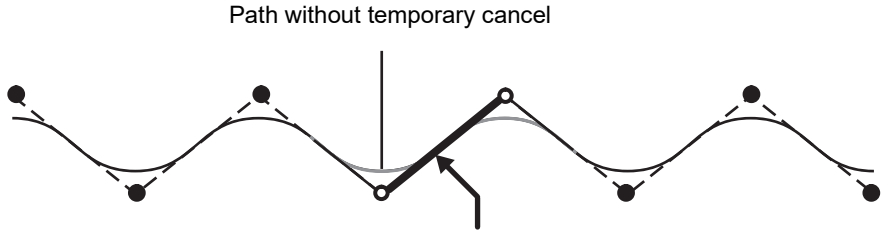
While spline interpolation 2 is enabled, it may be canceled temporarily depending on commands.
If spline interpolation 2 is canceled temporarily, the axis moves to the commanded position. After this, when a temporary cancel cause is removed, spline interpolation 2 restarts.

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



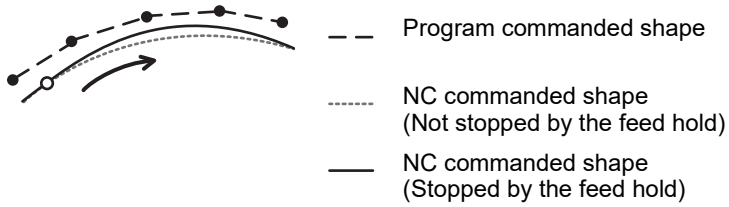
Block without movement by temporary cancel



Block with movement by temporary cancel

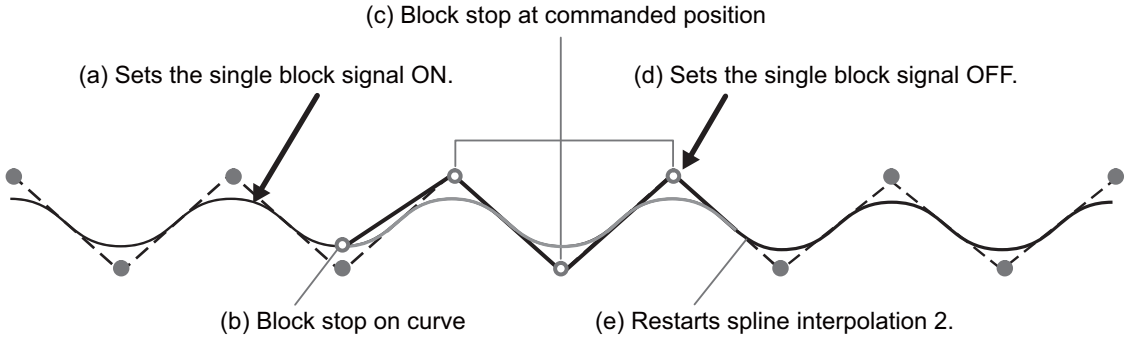
Feed hold

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, the operation error (M01 0180) occurs and the interrupt operation is 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

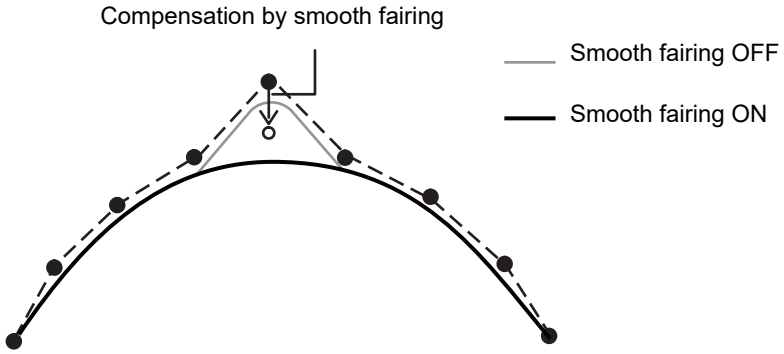
During single block operation, spline interpolation 2 is canceled temporarily. In this period, linear interpolation is carried out at the commanded position. If single block is set to ON during continuous operation, the currently processed block stops on a curve, and the next and subsequent blocks stop 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/tool cutting point control, smooth fairing is disabled.



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 micro segment processing capability is limited.
Spline interpolation 2 used in combination with tool center point control/tool cutting point control cannot be used together with high-speed high-accuracy control III.

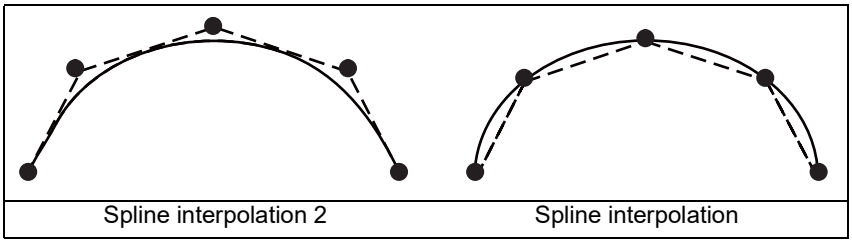
Spline interpolation

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 LENGS #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/tool cutting 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/tool cutting point control is enabled.

The points to note when tool center point control/tool cutting 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/tool cutting point control mode, the operation is performed in the G61.1 mode. G61.4 mode is activated after tool center point control/tool cutting 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]

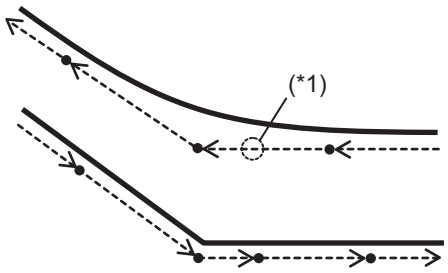
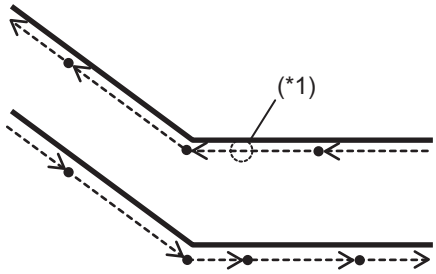
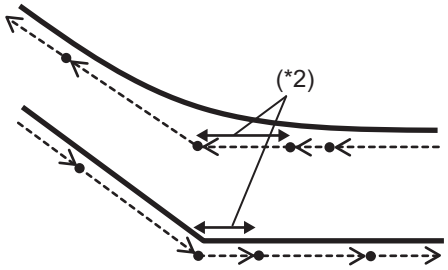
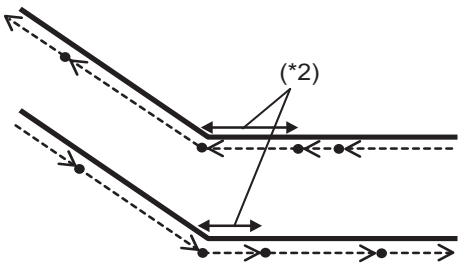
G code	Function name	When the G codes shown on the left are commanded while spline interpolation 2 is enabled	When spline interpolation 2 is enabled while the functions shown on the left are enabled
G43.4 G43.5	Tool center point control	Operation is performed in spline interpolation 2 mode using five axes.	Operation is performed in spline interpolation 2 mode using five axes.
G43.8 G43.9	Tool cutting point control		
G68.2 G68.3	Inclined surface machining command	♦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.	♦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 - 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) occurs.

(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 high	(b) When the tolerance amount is low
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 micro 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.
There are two types of spline interpolation command format: G61.2 and G05.1Q2.
Both formats can be used 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 "17.4 Spline Interpolation ; G05.1 Q2/Q0".

Command format

Spline mode ON

G61.2 X__ Y__ Z__ F__ ; or G61.2 ;

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 "17.3.2 Fairing".
(2) Spline interpolation
Refer to "Detailed description" of "17.4 Spline Interpolation ; G05.1 Q2/Q0".



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 LENGS".
- ♦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 LENGS", 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 cond" (selecting) 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) G120.1 must be commanded alone in a block, which also applies to G121. If it is not commanded alone in a block, a program error (P33) occurs.
(4) Address P in G120.1 command cannot be omitted. If omitted, a program error (P33) occurs.
(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 the address P is not within the range of 0 to 3 or the address Q is not within the range of 1 to 3 in G120.1 command, a program error (P35) occurs.
(8) When address P is set to "0" and address Q is either omitted or set to the number 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" (selecting) 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, the operation depends on the MTB specifications (parameter "#1712 cfg12/bit5").

#1712/bit5	Contents
0	Switches to the machining condition parameter group selected in the "Machining cond" (selecting) screen.
1	Retains the machining condition parameter group switched by the G120.1 command.

(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.

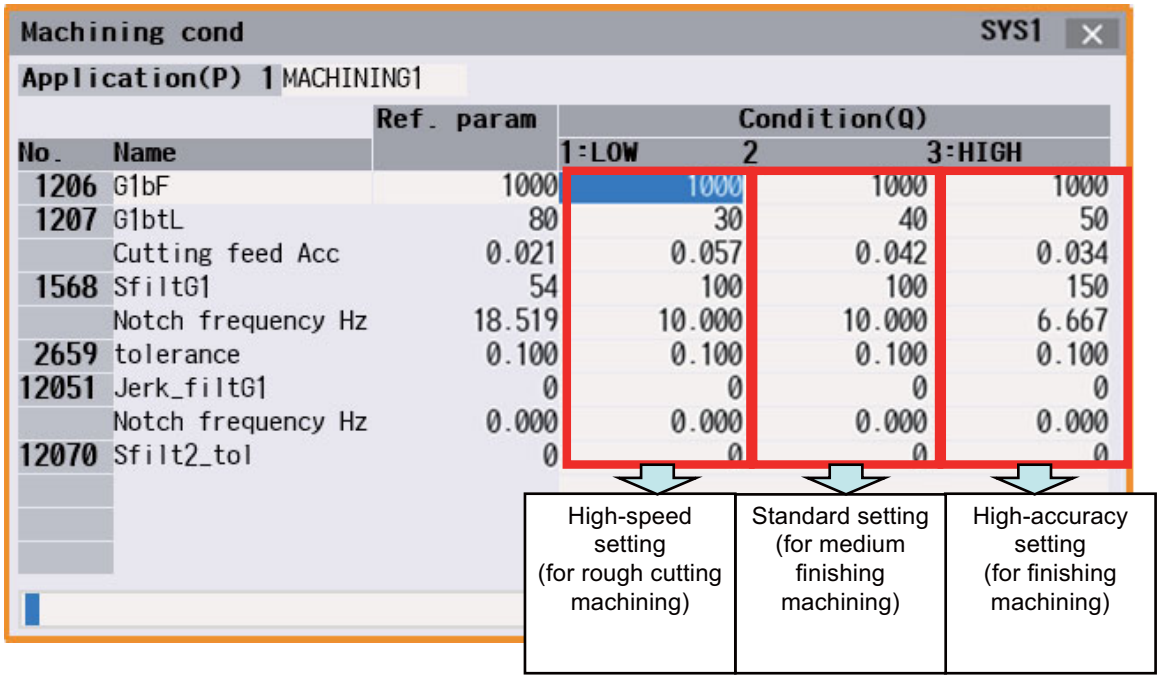
(13) It is switched to the selected machining condition parameter group in the "Machining cond" (selecting) screen by program end (M02 and M30).

(14) If G120.1 and G121 are commanded without initializing the machining condition parameter group, a program error (P128) occurs.

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").

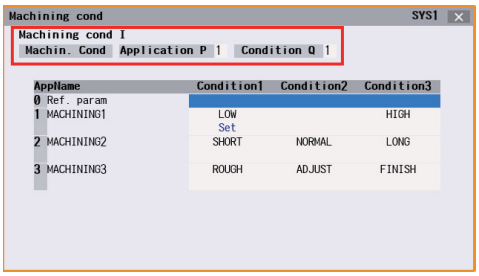


(1) When "Application1" and "Condition1" from the machining condition parameter group are selected in "Machining cond" (selecting) screen before running the program.

```
N1 G91 G28 Z0;
N2 G28 X0 Y0;
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.

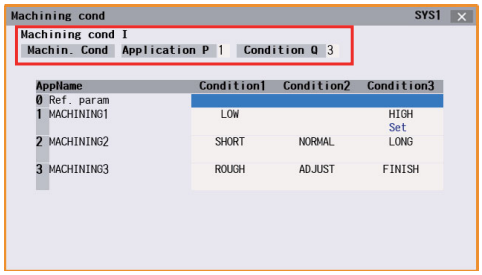
Machining condition parameters Operate with (machining usage 1/ condition 1)

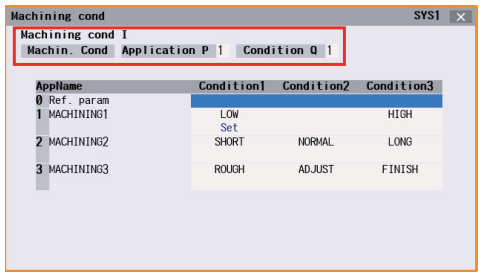



```
N1504 G90 G54 G00 X2. Y2.;
N1505 G43 H1 Z50.;
N1506 G90 G01 Z-8. F3000;
N1507 M3 S10000;
N1508 F1200;
N1509 G05 P10000;
N1510 G01 X2.099 Y1.997;
N1511 X2.199 Y1.990;
:
N2999 G05 P0;
N3000 G91 G28 Z0;
N3001 G28 X0 Y0;
N3002 M5;
N3003 M30;
```

... Return to the selected machining condition parameter group in "Machining cond" (selecting) screen at the program end.

Machining condition parameters Operate with (machining usage 1/ condition 3)





Relationship with other functions

(1) For M800V/M80V Series S/W version A8 or earlier, G code modal that cause a program error when commanding G120.1 and G121 are listed below.
They can be commanded in M800V/M80V Series S/W version A9 or later. (No program error occurs.)

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
G43.1	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) It is switched to the reference parameter by turning the power ON again.

(3) When G120.1 or G121 command is executed, the timing to switch the parameter depends on "#1254 set26/bit3". (This depends on the MTB specifications.)
When "0" is set, it is switched after smoothing zero for all axes including PLC axes are made in all the part systems.
When "1" is set, it is switched after smoothing zero for all axes are made in the part system where the command is issued.

(4) 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).

(5) If the machining condition parameters are switched, the setting values of the parameters "#2010 fwd_g" and "#2659 tolerance" are identical for all the NC axes in the switched part system.

(6) The machining condition parameters are not switched for the operation search. When the restart search is performed for the block after the G120.1 or G121 command, the machining condition parameter group is switched. However, when the machining condition parameter group switch by PLC signal is performed by M code, the M strobe does not turn ON in the restart search, and the machining condition parameter group is not switched. When the restart search is performed for the block after the M code to switch the machining condition parameter group, execute the applicable M code from the [MSTB history] menu and switch the machining condition parameter group.

(7) When the following conditions are satisfied simultaneously, the variable-acceleration pre-interpolation acceleration/deceleration is disabled automatically. The operation error (M01 0136) is not displayed.
・Variable-acceleration pre-interpolation acceleration/deceleration is ON.
・SSS control has been switched from ON to OFF by machining condition selection I.

(8) When the axis is exchanged by the cross axis control or arbitrary axis exchange and the machining condition parameter group is switched, the machining condition parameters are applied to the part system after the exchange.

18

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 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.

This function cannot be combined with the following spindle synchronization functions while it is active. An operation error (M01 1005) occurs.

- Spindle synchronization I
- Spindle synchronization II
- Tool spindle synchronization IA
- Tool spindle synchronization IB (IC)
- Tool spindle synchronization II

Command format

Spindle synchronization command

G114.1 H__ D__ R__ A__ ;	**Spindle synchronization start command**

H	Reference spindle specification
D	Synchronized spindle specification
R	Phase shift amount of synchronized spindle
A	Spindle synchronization acceleration/deceleration time constant

The spindle synchronization ON (G114.1) command designates the reference spindle and synchronized spindle, and synchronizes the two designated spindles. By commanding the phase shift amount of synchronized spindle, the phases of the reference spindle and synchronized spindle can be aligned.

Address	Meaning	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	• 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	• 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	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})	♦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 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)	♦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 (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.
The "Spindle rotation speed synchronization completion" signal (FSPRV) and the "Spindle phase synchronization completion" signal (FSPPH) are turned OFF.

(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.

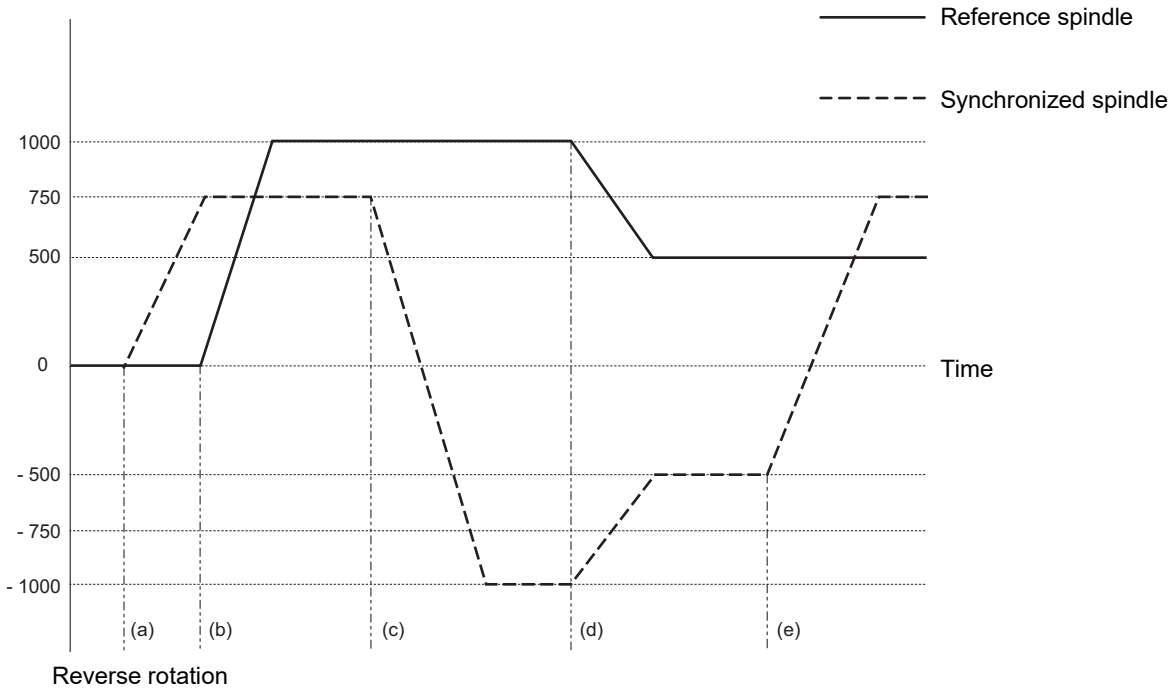
(2) If the reference spindle's commanded rotation speed is changed during the rotation synchronization state, acceleration/deceleration will be carried out while maintaining the synchronization state following the spindle acceleration/deceleration time constants set in the parameters, and the commanded 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.

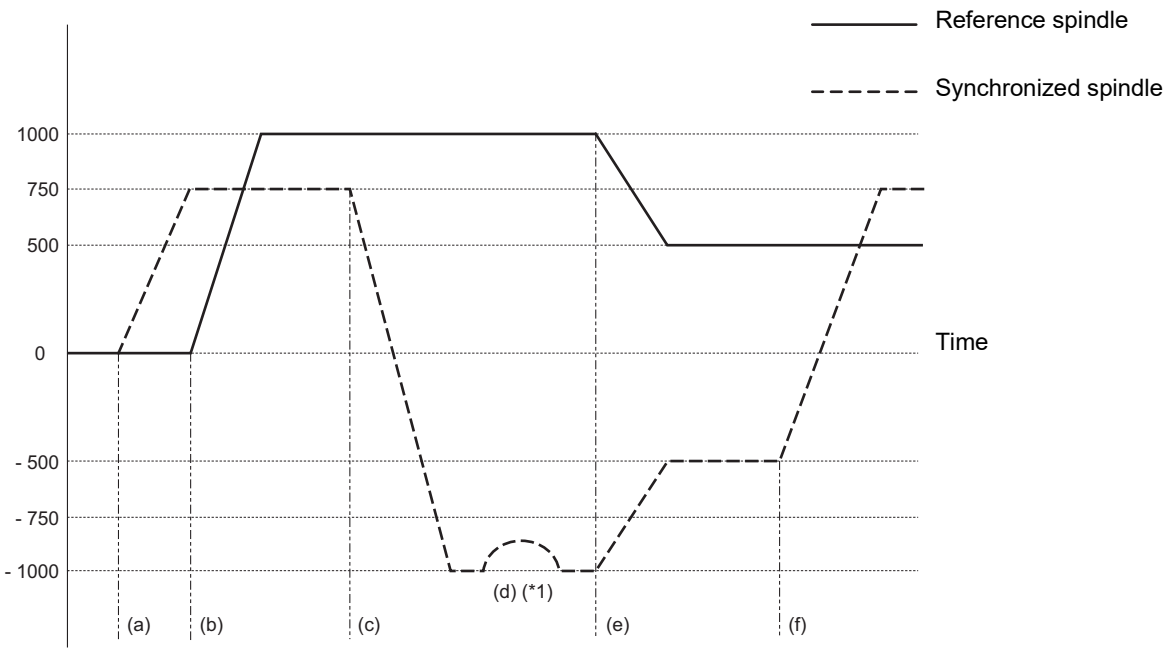
(2) If the reference spindle's commanded rotation speed is changed during the phase synchronization state, acceleration/deceleration will be carried out while maintaining the synchronization state following the spindle acceleration/deceleration time constants set in the parameters, and the commanded 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>



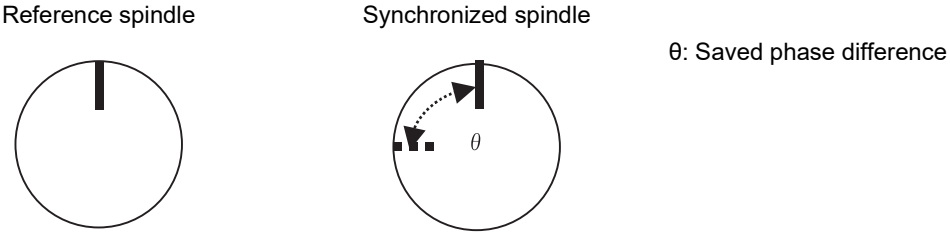
(*1) Phase synchronization is performed with the step alignment method (without acceleration/deceleration) when "#3130 syn_spec/bit1" = "0", and with the multi-step acceleration/deceleration method (described later) when "#3130 syn_spec/bit1" = "1".

Spindle synchronization phase shift amount calculation function

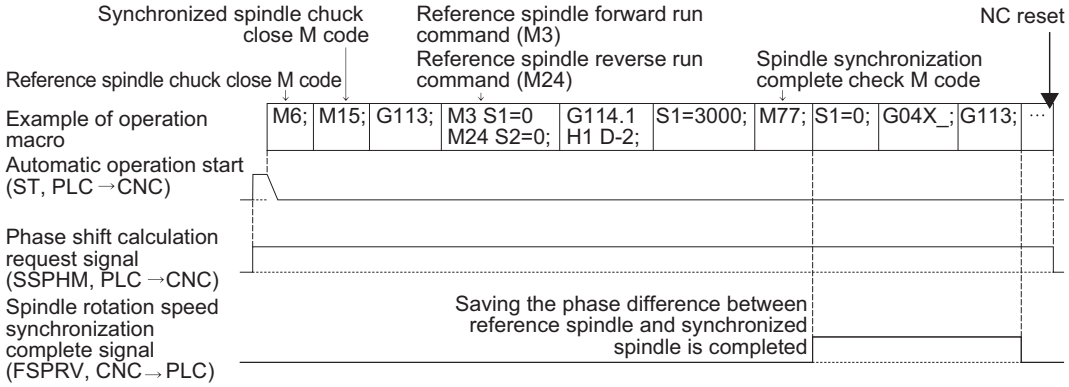
The spindle phase shift amount calculation function obtains and saves the phase difference of the reference spindle and synchronized spindle by turning the "PLC" signal ON when the phase synchronization command is executed. When the phase is positioned to the automatically saved phase difference before executing the phase synchronization control command, phases can be aligned easier when re-grasping profile materials.

[Saving the phase difference between reference spindle and synchronized spindle]

(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 command (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.



<Example of operation>



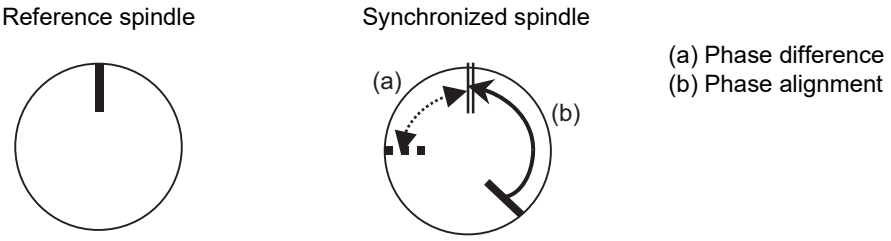
[Automatic phase alignment of reference spindle and synchronized spindle]

(1) Turn the "phase offset request" signal ON.

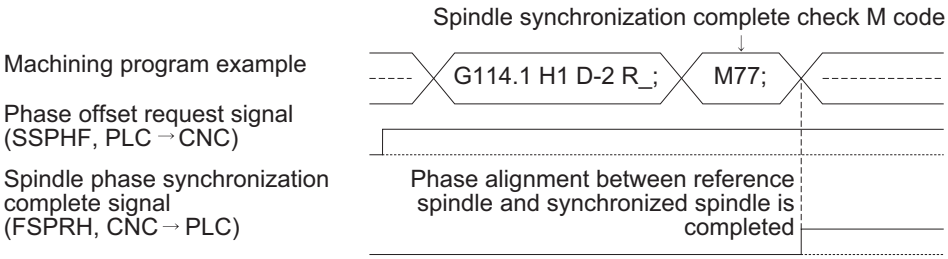
(2) Issue the phase synchronization command (with R command).

<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 phase shift amount of synchronized spindle, 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
= [Time constant without multi-step acceleration/deceleration] * [magnification of time constant in each step]
* [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 for spindle to rotate with the rotation speed set in "sptc 1" from stopped state (a)
= spt * sptc1 / slimit

Time required for spindle to reach the rotation speed set in "sptc2" from the speed of "sptc1" (b)
= spt * spdiv1 * (sptc2 - sptc1) / slimit

Time required for spindle to reach the rotation speed set in "sptc3" from the speed of "sptc2" (c)
= spt * spdiv2 * (sptc3 - sptc2) / slimit

Time required for spindle to reach the rotation speed set in "sptc4" from the speed of "sptc3" (d)
= spt * spdiv3 * (sptc4 - sptc3) / slimit

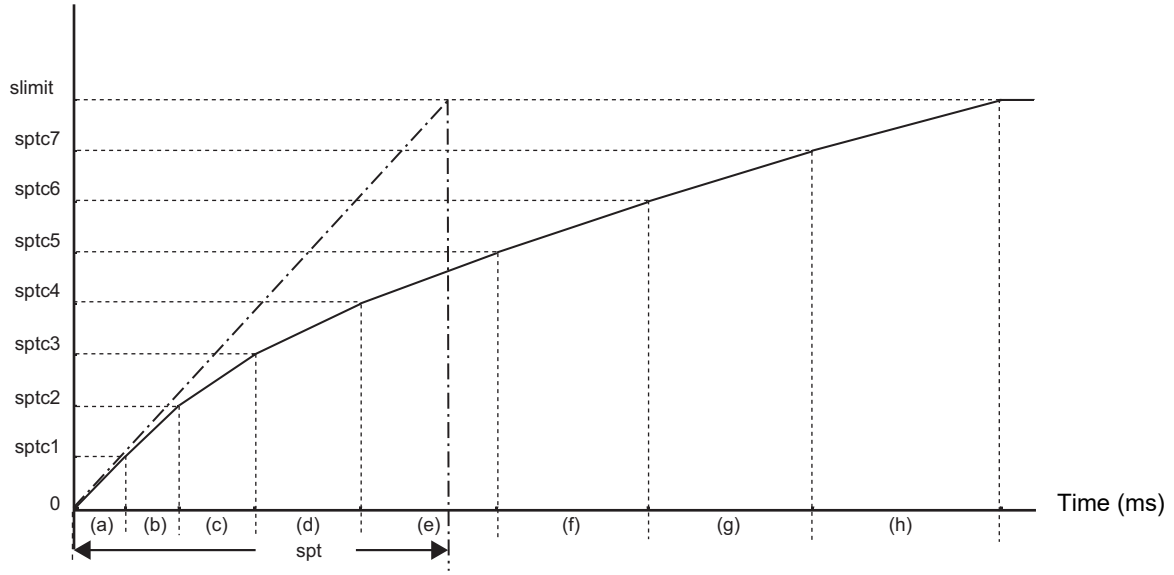
Time required for spindle to reach the rotation speed set in "sptc5" from the speed of "sptc4" (e)
= spt * spdiv4 * (sptc5 - sptc4) / slimit

Time required for spindle to reach the rotation speed set in "sptc6" from the speed of "sptc5" (f)
= spt * spdiv5 * (sptc6 - sptc5) / slimit

Time required for spindle to reach the rotation speed set in "sptc7" from the speed of "sptc6" (g)
= spt * spdiv6 * (sptc7 - sptc6) / slimit

Time required for spindle to reach the rotation speed set in "slimit" from the speed of "sptc7" (h)
= spt * spdiv7 * (slimit - sptc7) / 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" signal (ORC) for the reference spindle in the C axis mode is ignored during spindle synchronization. However, if the spindle position control command (C axis mode switch command) is issued during spindle orientation, the mode is switched to the C axis mode.

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 axes 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. If such a command is issued, an operation error (M01 1007) occurs, causing the automatic operation to be paused.
- You cannot command a synchronous tapping that uses the reference spindle or synchronized spindle of spindle synchronization I. If such a command is issued, an operation error (M01 1139) occurs, causing the automatic operation to be paused.

Spindle-mode rotary axis control

The spindle-mode rotary axis cannot be commanded for the reference spindle or synchronized spindle of spindle synchronization I/II, tool spindle synchronization IA (spindle-spindle, polygon), or tool spindle synchronization IB (spindle-spindle, polygon). If such a command is issued, an operation error (M01 1024) occurs, causing the automatic operation to be paused.



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. However, the modal will be updated, therefore 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. However, the modal will be updated, therefore this command 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) occurs.

(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) occurs.

(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 spindle phases between reference spindle and synchronized spindle 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 phase difference between reference spindle and synchronized spindle 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 phase difference between reference spindle and synchronized spindle saved in the NC is held until the next spindle synchronous phase shift calculation (rotation synchronization command is completed with the "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) occurs 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.)

Cautions on programming

(1) To enter the rotation synchronization mode while the reference spindle and synchronized spindle are chucking the same workpiece, turn the reference spindle and synchronized spindle rotation commands ON before turning the spindle synchronization mode ON.

(2) To chuck the same workpiece with the reference spindle and synchronized spindle in the phase synchronization mode, align the phases before chucking.

(3) (Only for M8 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 spindle-mode servo, multi-step acceleration/deceleration is applied regardless of the parameter setting.

⚠ CAUTION

⚠ Do not make the synchronized spindle rotation command OFF with one workpiece chucked by the reference spindle and synchronized spindle during the spindle synchronization mode. Failure to observe this may cause the synchronized spindle stop, and hazardous situation.

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.

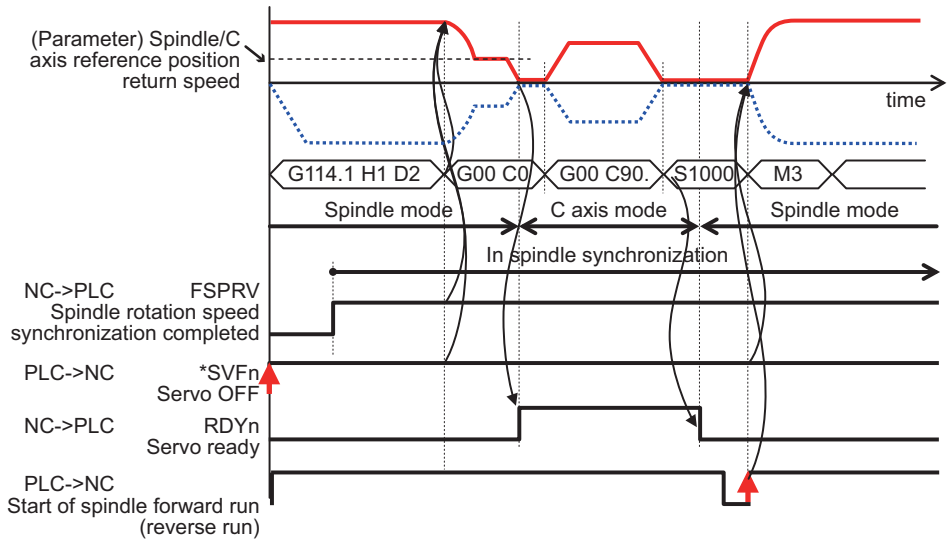
Detailed description

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 position address or absolute command (G90). If the incremental position address or incremental 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 to OFF>

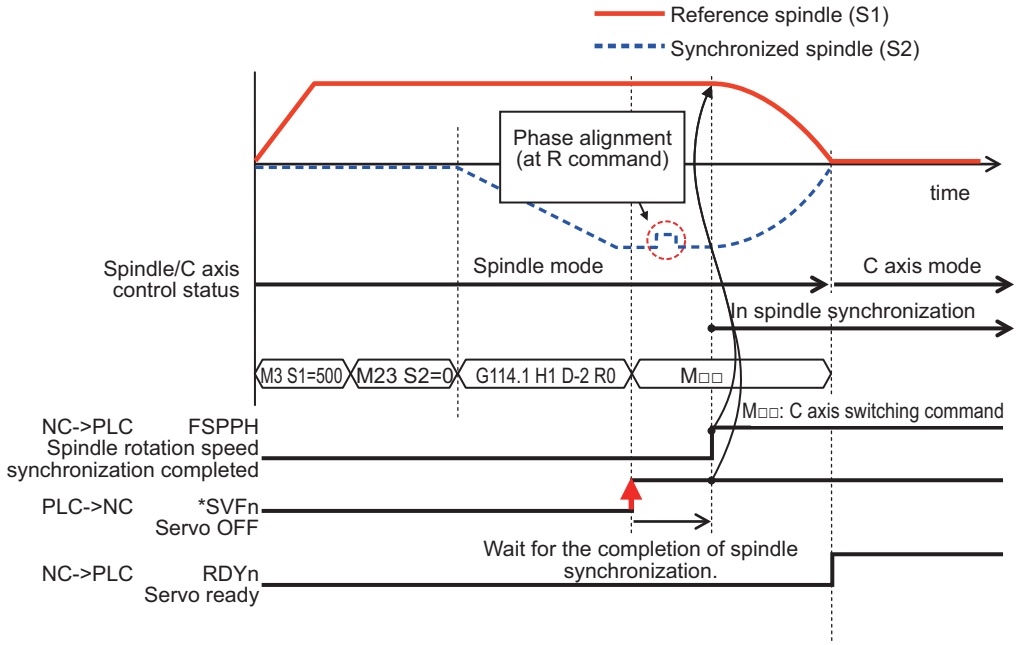
(1) The mode cannot be switched from the spindle mode to the C axis mode or from the C axis mode to the spindle mode.

(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).

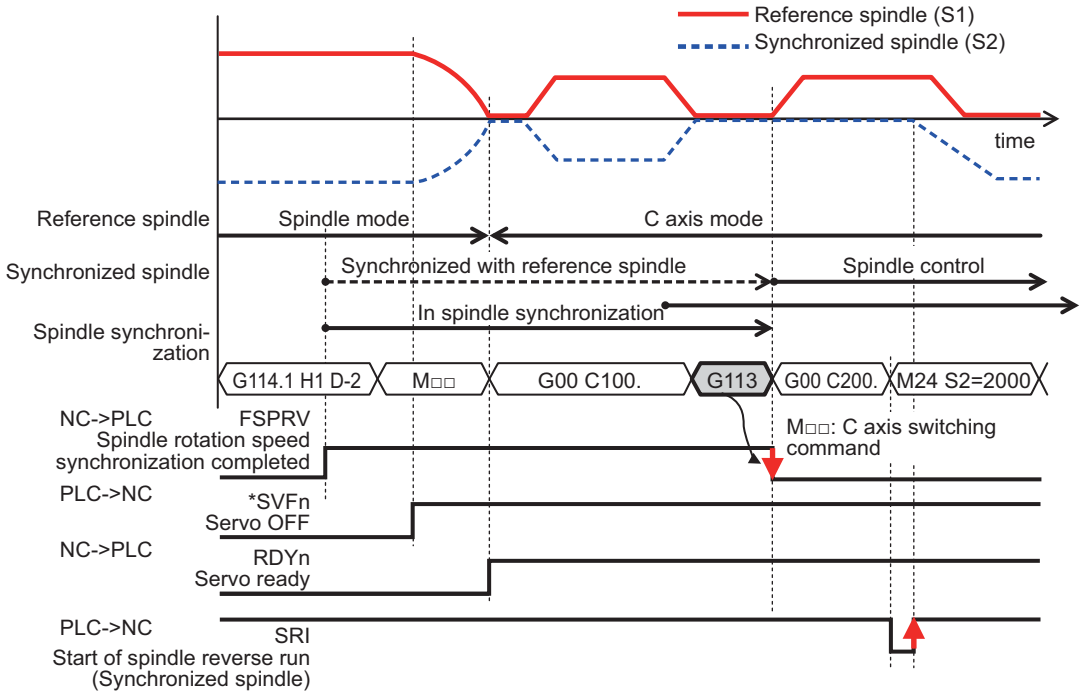
C axis mode selection command in spindle synchronization incomplete state

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 (FSPPH)) 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. ⬧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.
Inclined surface machining command (G68.2)	
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).

C axis rapid traverse rate (°/min) ≤ Highest spindle rotation speed (r/min) × 360(°)

(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 Tool Spindle Synchronization I

18.2.1 Tool Spindle Synchronization IA (Spindle-Spindle, Polygon) ; G114.2, G113.1

Function and purpose

In a machine having a rotary tool axis and having a spindle controlled as the workpiece axis, polygon machining between spindles can be carried out by controlling the rotation of the workpiece axis in synchronization with the rotary tool axis rotation.

This function cannot be combined with the following spindle synchronization functions while it is active. An operation error (M01 1005) occurs.

- Spindle synchronization I
- Spindle synchronization II
- Tool spindle synchronization IA
- Tool spindle synchronization IB (IC)
- Tool Spindle Synchronization II

Command format

Tool spindle synchronization IA (Spindle-spindle, polygon mode) ON

```
G114.2 H__ D__ E__ L__ R__ ;
```

H	Rotary tool axis selection (Reference spindle)
D	Workpiece axis selection (Synchronized spindle)
E	Rotary tool axis rotation ratio designation
L	Workpiece axis rotation ratio designation
R	Phase shift amount of synchronized spindle

Tool spindle synchronization IA (Spindle-spindle, polygon mode) OFF

```
G113.1 ;
```

Note

(1) An axis address that involves any travel cannot be put in the same block as for the tool spindle synchronization IA cancel command. Doing so causes the program error (P33) when the cancel command is issued, which causes automatic operation to pause.

Explanation of address

Tool spindle synchronization IA ON (G114.2) command sets the polygon machining mode that rotates the two axes in synchronization with differing speeds by designating the rotary tool axis and workpiece axes and the rotation ratio (Number of the rotary tool gear teeth and workpiece corners) of the two designated spindles (spindle and spindle). Tool spindle synchronization IA OFF (G113.1) cancels the synchronous state of two synchronously rotating spindles using the spindle synchronization command.

Address	Meaning	Command range (unit)	Remarks
H	Rotary tool axis selection Command the spindle number or spindle name of the rotary tool axis from multiple spindles. (*1)	Spindle number: 1 to n (n: Maximum number of available spindles) Spindle name: 1 to 9	⬧If a value exceeding the command range is commanded, a program error (P35) will occur. ⬧If there is no command, a program error (P33) will occur. ⬧If the same value as the D command is commanded, a program error (P33) will occur. ⬧If an analog-connected spindle is selected, a program error (P700) will occur. (*2)
D	Workpiece axis selection Commands the spindle number or spindle name of the workpiece axis of two spindles. (*1)	Spindle number: 1 to n or -1 to -n (n: Maximum number of available spindles) Spindle name: 1 to 9 or -1 to -9	⬧If a value exceeding the command range is commanded, a program error (P35) will occur. ⬧If there is no command, a program error (P33) will occur. ⬧The rotation direction of the workpiece axis in respect to the rotary tool axis is commanded with the D sign. ⬧If the same value as the H command is commanded, a program error (P33) will occur. ⬧If an analog-connected spindle is selected, a program error (P700) will occur. (*2)
E	Rotary tool axis rotation ratio designation Set the rotation ratio (Number of rotary tool gear teeth) of the rotary tool axis.	1 to 999	⬧If a value exceeding the command range is commanded, a program error (P35) will occur. ⬧If there is no command, the rotation ratio will be interpreted as 1.
L	Workpiece axis rotation ratio designation Set the rotation ratio (number of workpiece corners) of the workpiece axis.	1 to 999	⬧If a value exceeding the command range is commanded, a program error (P35) will occur. ⬧If there is no command, the rotation ratio will be interpreted as 1.
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 (°)	⬧If a value exceeding the command range is commanded, a program error (P35) will occur. ⬧The commanded shift amount will be applied in the clockwise direction in respect to the spindle. ⬧Minimum resolution of commanded shift amount Semi-close case 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.

(*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.

Detailed description

Rotary axis and rotation direction

The rotary tool axis and workpiece axis rotation speed and rotation direction during tool spindle synchronization IA command are as follows.

(1) The rotation speed and rotation direction of the rotary tool axis are the rotation speed commanded with the S command and the rotation direction commanded with the M command, etc., for the spindle selected as the rotary tool axis.

(2) The workpiece axis rotation speed is determined by the number of the rotary tool gear teeth workpiece corners commanded with G114.2.

$$Sw = Sh * \frac{E}{L}$$

Sw: Workpiece axis rotation speed (r/min)
Sh: Rotary tool axis rotation speed (r/min)
E: Rotary tool axis rotation ratio (Number of rotary tool gear teeth)
L: Workpiece axis rotation ratio (Number of workpiece corners)

(3) The workpiece axis rotation direction is determined by the sign of the address D commanded with G114.2. In other words, when the "D" sign is "+", the workpiece axis rotates in the same direction as the rotary tool axis, and when "-", the workpiece axis rotates in the reverse direction of the rotary tool axis.

(4) After tool spindle synchronization IA is commanded, the relation of the rotary tool axis and workpiece axis rotation is held in all automatic or manual operation modes until spindle synchronization cancel (G113) is commanded, the "spindle synchronization cancel" signal is input, or reset (reset 1, reset 2, reset & rewind) is executed when "#1239 set11/bit3" is set to "1".
Even during feed hold, the rotary tool axis and workpiece axis synchronization state is held.

Spindle operation for spindle-spindle polygon

(1) Even if the forward run command and reverse run command are not issued to the workpiece axis when the tool spindle synchronization IA mode is commanded, the workpiece axis starts rotation synchronously with the rotary tool axis if the rotation command is issued to the rotary tool axis.

(2) The rotation command (S command) and constant surface speed control are invalid in respect to the workpiece axis during the tool spindle synchronization IA mode. Note that the modal will be updated, so these will be effective after the spindle synchronization is canceled.

(3) If the rotation speed commanded to the workpiece axis exceeds the maximum rotation speed of the rotary tool axis or the maximum clamp speed designated by address S following G92, the workpiece axis rotation speed is clamped to prevent the rotation speed from exceeding those maximum speeds.

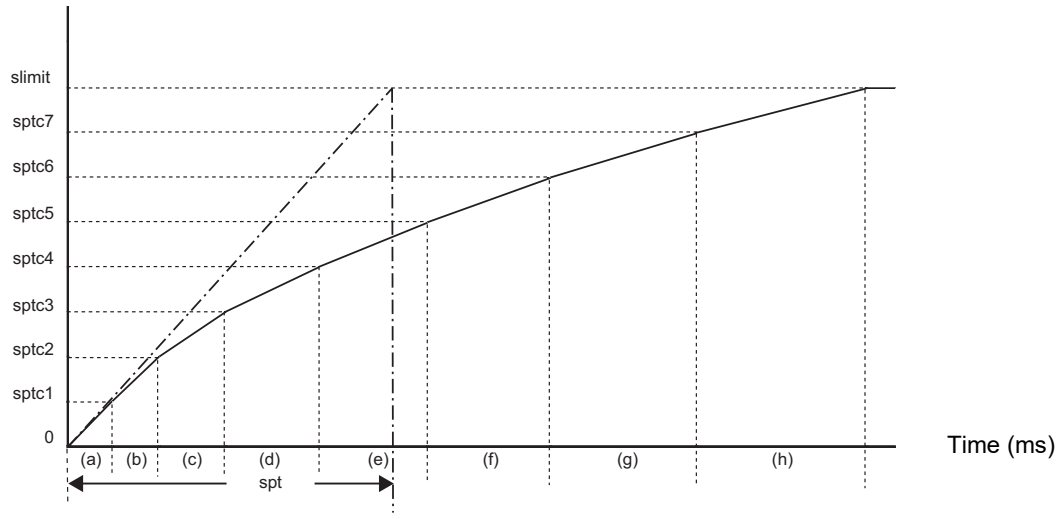
Multi-step acceleration/deceleration control

(1) Acceleration/deceleration time constants for up to eight steps can be selected according to the spindle rotation speed for the acceleration/deceleration during spindle-spindle polygon machining.
The acceleration/deceleration in each step is as follows.

Time required from minimum rotation speed to maximum rotation speed in each step
= [Time constant without multi-step acceleration/deceleration] * [magnification of time constant in each step]
* [Rate of rotation speed width in each step respect to rotation speed width up to limit rotation speed]

Time required to rotate to sptc1 set rotation speed from stopped state (a)
= spt * sptc1 / slimit
Time required to reach sptc2 set rotation speed from sptc1 (b)
= spt * spdiv1 * (sptc2 - sptc1) / slimit
Time required to reach sptc3 set rotation speed from sptc2 (c)
= spt * spdiv2 * (sptc3 - sptc2) / slimit
Time required to reach sptc4 set rotation speed from sptc3 (d)
= spt * spdiv3 * (sptc4 - sptc3) / slimit
Time required to reach sptc5 set rotation speed from sptc4 (e)
= spt * spdiv4 * (sptc5 - sptc4) / slimit
Time required to reach sptc6 set rotation speed from sptc5 (f)
= spt * spdiv5 * (sptc6 - sptc5) / slimit
Time required to reach sptc7 set rotation speed from sptc6 (g)
= spt * spdiv6 * (sptc7 - sptc6) / slimit
Time required to reach slimit set rotation speed from sptc7 (h)
= spt * spdiv7 * (slimit - sptc7) / slimit

Rotation speed (r/min)



To decrease the number of acceleration/deceleration steps, set one of the followings 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

(2) The rotary tool axis accelerates/decelerates linearly according to the spindle synchronous acceleration/deceleration time constant (spt) setting value of the spindle selected as the rotary tool axis and workpiece axis, whichever is larger.

(3) If the rotary tool axis command rotation speed is changed during spindle synchronization, the axis will accelerate/decelerate to the commanded rotation speed according to the spindle acceleration/deceleration set in the parameters while maintaining the synchronized state.

Phase alignment control

(1) If the tool spindle synchronization IA command (with R designation) is commanded with the G114.2 command, the synchronized spindle rotating at an arbitrary rotation speed will accelerate or decelerate to the rotation speed following the reference spindle and synchronized spindle rotation ratio command, and the spindle synchronization state will be entered. The spindles will then enter the spindle synchronization state. After that, the phases will be aligned to realize the rotation phase commanded with the R address.

(2) The spindle synchronization phase shift amount is commanded as the shift amount from the synchronized spindle's (workpiece axis) reference position ("one rotation" signal). There is not the shift amount in respect to the reference spindle (rotary tool axis).



Relationship with other functions

Spindle-mode rotary axis control

If the spindle-mode rotary axis is commanded for the reference spindle or synchronized spindle of spindle synchronization I/II, tool spindle synchronization IA (spindle-spindle, polygon), or tool spindle synchronization IB (spindle-spindle, polygon), an operation error (M01 1024) occurs.

Tool center point control

Tool spindle synchronization IA is not available during tool center point control. A program error (P942) occurs. However, the tool center point control command can be issued during tool spindle synchronization IA.

Inclined surface machining

Tool spindle synchronization IA is not available during execution of the inclined surface machining command. A program error (P951) occurs. However, the inclined surface machining command can be issued during tool spindle synchronization IA.



Precautions

Precautions for programming

(1) Always command G114.2 alone in a block.

(2) Do not issue S command in the same block as G114.2.

(3) The tool spindle synchronization IA (spindle-spindle, polygon) mode cannot be commanded during the spindle synchronization mode commanded with G114.*. An operation error (M01 1005) will occur.

(4) If spindle-spindle polygon machining is commanded while the "phase shift calculation request" signal SSPHM is ON, an operation error (M01 1106) will occur.

(5) Tool spindle synchronization IA (G114.2) cannot be executed using the spindle that is used in the synchronous tapping. An operation error (M01 1007) will occur.
Also, the synchronous tapping cannot be commanded using the spindle which is used in G114.2 command. An operation error (M01 1139) will occur.

(6) When the spindle/C axis is used for the spindle-spindle polygon machining cannot be executed by designating the C axis mode spindle with the G114.2 command. An operation error (M01 1026) will occur.

(7) After G114.2 is commanded, the cutting feed block will not start until synchronization is established. Operation will stop with an operation error (M01 1033).

Restrictions regarding phase alignment control

(1) Make sure that the rotation ratio of spindle (and rotary tool axis spindle) actual rotation speed and encoder rotation speed has the following relation.
Spindle rotation speed/encoder rotation speed = "n" ("n" is an integer of "1" or more)
If this relationship is not established, the encoder's reference position will not stay at a constant position on the spindle, and thus the phase (position) will deviate with each phase alignment command.
Note that even in this case, as shown below, if the number of rotary tool gear teeth (Number of workpiece corners) is equivalent to the rotation ratio, the blade and workpiece phase (position) will not deviate.
(Rotary tool axis spindle rotation speed * Number of rotary tool gear teeth) /encoder rotation speed = "n"
("n" is an integer of "1" or more)

(2) During phase alignment control, phase alignment is carried out following each spindle encoder's reference position.
Thus, if the positional relation of the workpiece and reference position (rotary tool and reference position) deviates when the power is turned OFF/ON or when the tool is changed, etc., the phase will deviate.

18.2.2 Tool Spindle Synchronization IB (Spindle-Spindle, Polygon) ; G51.2/G50.2 or G251/G250

Function and purpose

In a machine having a workpiece axis and having a spindle controlled as the rotary tool axis, polygon machining between spindles can be carried out by controlling the rotation of the rotary tool axis in synchronization with the workpiece axis rotation.
Tool spindle synchronization IB and tool spindle synchronization IC are switched depending on the setting of the parameter (#1501).

#1501 polyax =	0	: Tool spindle synchronization IB
	Other than 0	: Tool spindle synchronization IC

This function is valid when the G code system is 1.

This function cannot be combined with the following spindle synchronization functions while it is active. An operation error (M01 1005) occurs.

- Spindle synchronization I
- Spindle synchronization II
- Tool spindle synchronization IA
- Tool spindle synchronization IB (IC)
- Tool Spindle Synchronization II

Command format

Tool spindle synchronization IB (Spindle-spindle, polygon mode) ON (or G251)

```
G51.2 H__D__P__Q__R__ ;
```

H	Workpiece axis selection (Reference spindle)
D	Rotary tool axis selection (Synchronized spindle)
P	Workpiece axis rotation ratio designation
Q	Rotary tool axis rotation ratio designation
R	Phase shift amount of synchronized spindle

Tool spindle synchronization IB (Spindle-spindle, polygon mode) Cancel (or G250)

```
G50.2 ;
```

Note

(1) An axis address that involves any travel cannot be put in the same block as the tool spindle synchronization IB mode cancel command. Doing so causes the program error (P33) when the cancel command is issued, which causes automatic operation to pause.

Detailed explanation of command format

Tool spindle synchronization IB ON (G51.2) command sets the polygon machining mode that rotates the two axes in synchronization with differing speeds by designating the rotary tool axis and workpiece axes and the rotation ratio (Number of the rotary tool gear teeth and workpiece corners) of the two designated spindle (spindle and spindle). Tool spindle synchronization IB OFF (G50.2) command cancels the synchronous state of rotating two spindles by the tool spindle synchronization command.

The spindle-spindle polygon machining mode is also canceled in the following cases.

- Power OFF
- Emergency stop
- Reset (reset 1, reset 2, reset & rewind)
  (only when #1239 set11/bit3 = 1)
- "Spindle-spindle polygon machining cancel" signal
- Spindle synchronization cancel command (G113.1)
- "Spindle synchronization cancel" signal (SPSYC)

The detail of each address of the command format is as follows.

Address	Meaning	Command range (unit)	Remarks
H	Workpiece axis selection Command the spindle number of the workpiece axis. (*1)	Spindle No.: 1 to n (n: Maximum number of available spindles) Spindle name: 1 to 9	•If a value exceeding the command range is commanded, a program error (P35) will occur. • If the same value as the D command is commanded, a program error (P33) will occur. •If an analog-connected spindle is selected, a program error (P33) will occur. (*2) •If this address is omitted, the spindle number or spindle name specified in the parameter is designated.
D	Rotary tool axis selection Command the spindle number of the rotary tool axis. (*1)	Spindle No.: 1 to n (n: Maximum number of available spindles) Spindle name: 1 to 9	•If a value exceeding the command range is commanded, a program error (P35) will occur. • If the same value as the H command is commanded, a program error (P33) will occur. •If an analog-connected spindle is selected, a program error (P33) will occur. (*2) •If this address is omitted, the spindle number or spindle name specified in the parameter is designated.
P	Workpiece axis rotation ratio designation Set the rotation ratio (number of workpiece corners) of the workpiece axis.	1 to 999	•If a value exceeding the command range is commanded, a program error (P35) will occur.
Q	Rotary tool axis rotation ratio designation Command the rotary tool axis rotation ratio (number of tool teeth).	1 to 999 -1 to -999	•If a value exceeding the command range is commanded, a program error (P35) will occur. •If a negative "-" sign is commanded, the rotary tool axis will rotate in the direction opposite to the workpiece axis.
R	Phase shift amount of synchronized spindle Command the shift amount designation from the reference position ("one rotation" signal) of the rotary tool axis spindle.	0 to 359.999 (°)	•If a value exceeding the command range is commanded, a program error (P35) will occur. •The commanded shift amount will be applied in the clockwise direction in respect to the spindle. •Minimum resolution of commanded shift amount Semi-close case 360/4096 [°] Full-close case (360/4096) * K [°]) (K: Gear ratio of spindle and encoder) •If there is no R command, the phase will be handled as R0. (only when #1239 set11/bit4 = 0)

(*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.

Detailed description

Rotary axis and rotation direction

The workpiece axis and rotary tool axis rotation speed and rotation direction during spindle-spindle polygon machining are as follows.

(1) The workpiece axis rotation speed and rotation direction are rotation speed commanded with the S command and the rotation direction commanded with the M command, etc., for the spindle selected as the workpiece axis.

(2) The rotary tool axis rotation speed is determined by the number of the rotary tool teeth and number of workpiece corners commanded with G51.2/G251.

$$Sw = Sh * \frac{Q}{P}$$

Sw: Rotary tool axis rotation speed (r/min)
Sh: Workpiece axis rotation speed (r/min)
P: Workpiece axis rotation ratio (Number of workpiece corners)
Q: Rotary tool axis rotation ratio (Number of rotary tool gear teeth)

(3) The rotary tool axis rotation direction is determined by the sign of the rotary tool axis selection Q commanded with G51.2/G251.
In other words, if the Q sign is "+", the rotary tool axis will rotate in the same direction as the workpiece axis. If the Q sign is "-", the rotary tool axis will rotate in the reverse direction of workpiece axis.

(4) After tool spindle synchronization IB (G51.2/G251) is commanded, the relationship between the workpiece axis and rotary tool axis rotation is held until tool spindle synchronization IB cancel (G50.2/G250) is commanded, the "spindle-spindle polygon machining cancel" signal is input, or until the "reset or emergency stop" signal is input. Even at feed hold, the workpiece axis and rotary tool axis synchronization states are held.

Operation for polygon machining with rotary tool axis

(1) When the tool spindle synchronization IB mode is commanded, even if neither the forward run nor reverse run command is input for the rotary tool axis, the rotary tool axis will start rotating.

(2) If spindle stop is commanded to a rotary tool axis during the tool spindle synchronization IB mode (when the "spindle stop" signal is ON), the rotary tool axis will stop rotating even if the workpiece axis is rotating.

(3) The rotation command (S command) and constant surface speed control are invalid in respect to the rotary tool axis during the tool spindle synchronization IB mode. Note that the modal is updated, so these will be validated when the spindle-spindle polygon machining is canceled.

(4) If the rotation speed commanded to the workpiece axis exceeds the maximum rotation speed of the rotary tool axis or the maximum clamp speed designated by address S following G92, the workpiece axis rotation speed is clamped to prevent the rotation speed from exceeding those maximum speeds.

Acceleration/deceleration control

(1) Acceleration/deceleration of the workpiece axis will be carried out linearly according to the spindle synchronization acceleration/deceleration time constant (parameter "#3049 spt") of the spindle selected as the workpiece axis.

(2) By setting the spindle synchronization multi-step acceleration/deceleration time constant changeover speed levels 1 to 7 (parameters "#3054 sptc1" to "#3060 sptc7") and the scale for the time constant changeover speed (parameters "#3061 spdiv1" to "#3067 spdiv7"), the acceleration/deceleration time can be changed in up to eight steps.

(3) If the workpiece axis command rotation speed is changed during spindle synchronization state, the commanded speed will be reached by accelerating or decelerating according to the spindle acceleration/deceleration set in the parameters while maintaining the synchronized state.

Phase alignment control

(1) If the tool spindle synchronization IB command (R is treated as "0" when there is no R command) is commanded with G51.2/G251, the workpiece axis spindle rotating at an arbitrary rotation speed will accelerate/decelerate to the rotation speed following the rotation ratio command of the workpiece axis spindle and the spindle synchronization state will be entered. The spindles will then enter the spindle synchronization state. After that, the phases will be aligned to realize the rotation phase commanded with the R address.

(2) The spindle synchronization phase shift amount is commanded the shift amount from the rotary tool axis spindle's reference position ("one rotation" signal). This is not the shift amount for the workpiece axis.



Relationship with other functions

Spindle-mode rotary axis control

Tool spindle synchronization IB cannot be executed using the spindle-mode rotary axis. An operation error (M01 1024) will occur.

Tool center point control

Tool spindle synchronization IB is not available during tool center point control. A program error (P942) occurs. However, the tool center point control command can be issued during tool spindle synchronization IB.

Inclined surface machining

Tool spindle synchronization IB is not available during execution of the inclined surface machining command. A program error (P951) occurs. However, the inclined surface machining command can be issued during tool spindle synchronization IB.



Precautions

Restrictions regarding phase alignment control

(1) Make sure that the spindle (and workpiece axis spindle) actual rotation speed and encoder rotation speed's rotation ratio has the following relation.
Spindle rotation speed/encoder rotation speed = "n" ("n" is an integer of 1 or more)
If this relationship is not established, the encoder's reference position will not stay at a constant position on the spindle, and thus the phase (position) will deviate with each phase alignment command.
Note that even in this case, as shown below, if the number of workpiece corners (number of rotary tool teeth) corresponds to the rotation ratio, the phase (position) of the blade and workpiece will not deviate.
(Workpiece axis spindle rotation speed * Number of workpiece teeth) /encoder rotation speed = "n" ("n" is an integer of "1" or more)

(2) During phase alignment control, the phase are aligned to the reference position of each spindle's encoder. Thus, if the position relation of the workpiece and reference position (workpiece and reference position) deviates when the power is turned ON/OFF or the tool is replaced, etc., the phase will deviate.

Precautions for programming

(1) G51.2 (G251) must be commanded alone in a block, which also applies to G50.2 (G250).

(2) The R command can be omitted when entering the tool spindle synchronization IB mode, but the P and Q commands must always be issued. A program error (P33) will occur if there are not the P and Q commands.

(3) To change the P, Q or R modal value in the tool spindle synchronization IB mode, command G51.2/G251 again. In this case, R can be commanded alone. However, if either P or Q is also changed, always command P and Q again.

(4) Commands can be issued to each part system, but two or more part systems cannot be used simultaneously. The part system commanded first will be valid, and an operation error (M01 1005) will occur for that commanded last.

(5) The spindle No. designated in the parameters will be used if D_H_ is omitted from the G51.2/G251 command.

(6) A program error (P610) will occur if the workpiece axis No. (#1518) and rotary tool axis No. (#1519) are the same as the value set in the parameters. A program error (P33) will occur if the spindle is connected in analog mode. (These parameters depend on the MTB specifications.)

(7) After G51.2/G251 was commanded, the cutting feed block will not start until synchronization is established. (An operation error (M01 1033) will occur, and the program stops.)

(8) Tool spindle synchronization IB (G51.2/G251) machining cannot be executed by designating the spindle which is used in the synchronous tapping. An operation error (M01 1007) will occur. Also, the synchronous tapping cannot be commanded using a spindle which is used in G51.2/G251 command. An operation error (M01 1139) will occur.

(9) If the rotary tool axis number or workpiece axis number is changed in the tool spindle synchronization IB mode, a program error (P33) will occur.

18.2.3 Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon) ; G51.2/G50.2 or G251/G250

Function and purpose

This function controls so that the workpiece (spindle) and tool (rotary tool axis) are synchronously rotating with the commanded ratio to conduct polygon machining. This function is available for machining of square-head bolts, bolt heads of hexagon-head bolts, or hexagon nuts.
Tool spindle synchronization IB and tool spindle synchronization IC are switched depending on the setting of the parameter (#1501).

This function cannot be combined with the following spindle synchronization functions while it is active. An operation error (M01 1005) occurs.

- Spindle synchronization I
- Spindle synchronization II
- Tool spindle synchronization IA
- Tool spindle synchronization IB (IC)
- Tool Spindle Synchronization II

Command format

Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon mode) ON (or G251)

```
G51.2 P__ Q__ ;
```

P,Q	Spindle and rotary tool axis rotation ratio (P__:Q__) P : Spindle Q : Rotary tool axis Command range : Integer value between 1 and 200, -1 and -200 Rotation direction : Designate with a sign ("+" for forward rotation, whereas "-" for reverse rotation).

Tool spindle synchronization IC (spindle-NC axis, polygon) cancel (or G250)

```
G50.2 ;
```

In addition to the G50.2 command, the tool spindle synchronization IC mode is also canceled in the following cases.

- Power OFF
- Emergency stop
- Reset (reset 1, reset 2, reset & rewind)



Detailed description

Details of operation

S1000 ;	The spindle rotation speed (workpiece rotation speed) is commanded.
G51.2 P1 Q2 ; : (Cutting into workpiece) :	The polygon machining mode is entered with the G51.2 command. The spindle and rotary tool axis start rotating, and control is applied so that the spindle rotation speed and tool axis rotation speed are the commanded ratio (P:Q).
G50.2 ;	The polygon machining mode between the spindle and rotary tool axis is canceled by the G50.2 command, and the spindle and rotary tool axis rotation stop.

The rotary tool axis depends on the setting of the base specification parameter "#1501 polyax".

Rotation direction

(1) The spindle rotation direction during the polygon machining mode is determined by the P command sign and the spindle parameter "#3106 zrn_typ/bit7" (synchronous tap command polarity).

P command sign	#3106 zrn_typ/bit7	Rotation direction
(+)	0	CW
(+)	1	CCW
(-)	0	CCW
(-)	1	CW

(2) The rotation direction of the rotary tool axis during the polygon machining mode is determined by the Q command sign and the base specifications parameters "#1018 CCW".

Q command sign	#1018 CCW	Rotation direction
(+)	0	CW
(+)	1	CCW
(-)	0	CCW
(-)	1	CW

Relationship with other functions

Spindle-mode rotary axis control

If the spindle-mode rotary axis is changed to the servo axis mode while tool spindle synchronization IC is executed using the spindle-mode rotary axis as the polygon spindle, an operation error (M01 1024) occurs. Also, if tool spindle synchronization IC is commanded using the spindle-mode rotary axis of servo axis mode as the polygon axis, an operation error (M01 1024) occurs.

Spindle position control (Spindle/C axis control)

If the spindle/C axis is changed to the C axis mode while tool spindle synchronization IC is executed using the spindle/C axis as the polygon-related spindle, an operation error (M01 1026) occurs. Also, if tool spindle synchronization IC is commanded using the spindle/C axis in the C axis mode as the polygon-related spindle, an operation error (M01 1026) occurs.

High-speed high-accuracy Control

When tool spindle synchronization IC is commanded while the high-speed high-accuracy control is valid or when high-speed high-accuracy control is commanded during tool spindle synchronization IC, high-speed high-accuracy control is temporarily canceled, and tool spindle synchronization IC is executed preferentially.

Arbitrary axis exchange control

The arbitrary axis exchange command (G140), arbitrary axis exchange return command (G141), or reference axis arrange return command (G142) cannot be issued in the part system for tool spindle synchronization IC. A program error (P501) occurs. However, tool spindle synchronization IC using the arbitrary axis exchange axis can be commanded.

The rotary tool axis of tool spindle synchronization IC cannot be commanded as the axis to be replaced. Doing so triggers the arbitrary axis exchange disable state.

3-dimensional coordinate conversion

The 3-dimensional coordinate conversion cannot be commanded in the part system for tool spindle synchronization IC. A program error (P922) occurs.
Also, tool spindle synchronization IC cannot be commanded during 3-dimensional coordinate conversion. A program error (P921) occurs.

Tool center point control

Tool spindle synchronization IC is not available during tool center point control. A program error (P942) occurs. The tool center point control command can be issued during tool spindle synchronization IC.

<Note>

- Do not designate the rotary tool axis in the rotary axis configuration parameters. Doing so may cause an unexpected operation by the tool center point control command during tool spindle synchronization IC.

Inclined surface machining command

Tool spindle synchronization IC is not available during execution of the inclined surface machining command. A program error (P951) occurs. Also, the inclined surface machining command cannot be issued during tool spindle synchronization IC. A program error (P952) occurs.



Precautions

(1) The tool spindle synchronization IC specifications must be provided to use this function.
If "G51.2/G251" or "G50.2/G250" is commanded without the specifications, a program error (P39) will occur.

(2) G51.2 (G251) must be commanded alone in a block, which also applies to G50.2 (G250).
- If the G51.2/G251 (G50.2/G250) command and G code of group 0 are commanded in the same block, the G code commanded last in the block will have the priority.
- If the G51.2/G251 (G50.2/G250) command and G code other than a group 0 code are commanded in the same block, a program error (P33) will occur.

(3) While in the polygon machining mode, a movement command cannot be issued in the machining program for an NC axis set as the rotary tool axis.
If a movement command is issued to the rotary tool axis during the polygon machining mode, the program error (P32) will occur.

(4) The NC axis set as the rotary tool axis can be used as a feed axis in modes other than the polygon machining mode.

(5) The following functions are invalid for the rotary tool axis during the polygon machining mode.
- Override
- Feed hold
- Stored stroke limit

(6) The spindle rotation speed can be changed with the S command even during the polygon machining mode.
The spindle override and spindle rotation speed clamp are also valid.
If the spindle rotation speed is changed, the rotary tool axis rotation speed will also change so that the spindle and rotary tool axis established the P:Q ratio.

(7) The forward run/reverse run commands are invalid for the spindle when the polygon machining mode is in effect.

(8) If the feedrate for the rotary tool axis exceeds the rapid traverse rate (axis specifications parameters "#2001 rapid") when the polygon machining mode is in effect, the speed will be clamped at the rapid traverse rate. If the rotary tool axis is clamped at the rapid traverse rate, the spindle speed will also be set to lower than the command speed so that the spindle and rotary tool axis establish the P:Q ratio.

(9) The position loop gain for the rotary tool axis will be the value set in the axis specifications parameters "#2017 tap_g" during the polygon machining mode. The position loop gain for the spindle will be the spindle parameters "#13002 PGN" setting value.

(10) To perform polygon machining, set the parameter "#8213 rotary axis type" to "0" or "1".

(11) The following functions cannot be used simultaneously with polygon machining.
- Synchronous tapping
- Thread cutting

(12) If an axis other than the rotary tool axis reaches the stroke end during the polygon machining mode, the axis other than the rotary tool axis will stop moving, but the rotary tool axis and spindle rotation will not stop.

(13) If the rotary tool axis reaches the stroke end during the polygon machining mode, the rotary tool axis and spindle rotation will stop, and the movement of axes other than the rotary tool axis will also stop.

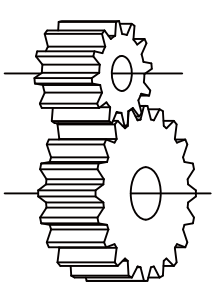
(14) If the spindle specifications parameter "#3106 zrn_typ/bit4" is set to "0", the polygon machining will start after the spindle returns to the zero point. (This parameter setting depends on the MTB specifications.)

18.3 Tool Spindle Synchronization II

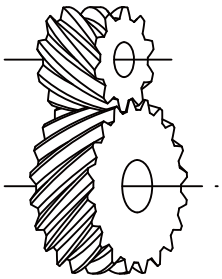
18.3.1 Tool Spindle Synchronization II (Hobbing) ; G114.3/G113

Function and purpose

This function is to cut the gear with hob (hob cutter).
A spur gear can be machined by synchronizing and rotating the hob axis and the workpiece axis in a constant ratio.
A helical gear can be machined by compensating the workpiece axis according to the gear torsion angle for the Z axis movement.

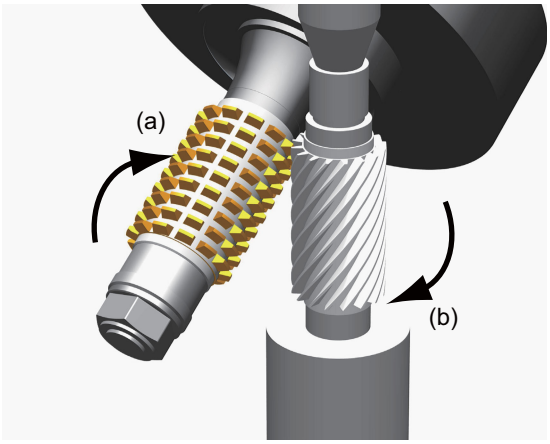


Spur gears



Helical gears

By synchronizing and rotating the hob axis and the workpiece axis in a constant rotation ratio, a gear is machined so that the cutter is engaged with gear.



(a) Hob
(b) Gear

In this manual, the hob axis and the workpiece axis are defined as follows:

Hob axis : Rotary tool axis on which a hob is mounted
Workpiece axis : Rotary axis on which a workpiece is mounted
Hob threads : Number of the screw paths created by cutter part on hob. Usually this is 1 row.

This function cannot be combined with the following spindle synchronization functions while it is active. The operation error (M01 1005) occurs.

- Spindle synchronization I
- Spindle synchronization II
- Tool spindle synchronization IA
- Tool spindle synchronization IB (IC)
- Tool spindle synchronization II

Command format

Tool spindle synchronization II ON is set as the hobbing mode for the spur gears that synchronously rotates two axes at different speeds by designating the hob axis and workpiece axis and designating the rotation ratio (number of hob threads and number of gear teeth) for the two designated axes.

It is set as the hobbing mode for the helical gears by designating the gear torsion angle and module or diametral pitch as well as the setting of spur gears.

Tool spindle synchronization II OFF cancels the synchronous state of the hob axis and workpiece axis rotating in synchronization with the tool spindle synchronization II (hobbing) command.

Tool spindle synchronization II (hobbing) ON (for spur gears)

```
G114.3 H__ D__ E__ L__ R__ ;
```

H	Hob axis selection
D	Workpiece axis selection
E	Hob axis rotation ratio designation
L	Workpiece axis rotation ratio designation
R	Workpiece axis phase shift amount

Tool spindle synchronization II (hobbing) ON (for helical gears)

```
G114.3 H__ D__ E__ L__ P__ Q__ R__ ;
```

H	Hob axis selection
D	Workpiece axis selection
E	Hob axis rotation ratio designation
L	Workpiece axis rotation ratio designation
P	Gear torsion angle designation
Q	Module or diametral pitch designation
R	Workpiece axis phase shift amount

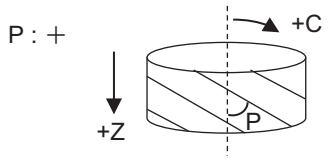
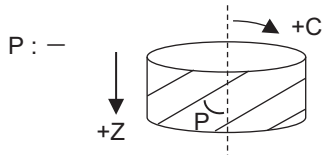
Tool spindle synchronization II (hobbing) OFF

```
G113.1 ;
```

Note

(1) An axis address that involves any travel cannot be put in the same block as the tool spindle synchronization II cancel command. Doing so causes the program error (P33) when the cancel command is issued, which causes automatic operation to pause.

Description of address

Address	Meaning	Command range (unit)	Remarks
H	Hob axis selection Command the spindle number of the hob axis. (*2)	Spindle No.: 1 to n (n: Maximum number of available spindles) Spindle name: 1 to 9	•If there is no command, the program error (P33) occurs. •If an analog-connected spindle is selected, the program error (P33) occurs. •If disconnected spindle No. is designated, the program error (P35) occurs.
D	Workpiece axis selection Command the rotary axis No. of the workpiece axis.	-9 to -1, 1 to 9 ±1 to 8 Axis No. (in part system) ±9: C axis	•If there is no command, the program error (P33) occurs. •The rotation direction of the workpiece axis in respect to the hob axis is commanded with the D sign. •If the D sign is "+", the workpiece axis will rotate in the forward direction when the hob axis rotates in the forward direction. If the D sign is "-", the workpiece axis will rotate in the reverse direction when the hob axis rotates in the forward direction. •If the axis specified as the workpiece axis is not a rotary axis, the program error (P33) occurs. •If C axis is selected when there is no C axis, the program error (P33) occurs.
E	Hob axis rotation ratio designation Command the hob axis rotation ratio (hob threads).	0 to 999	•If there is no command, the rotation ratio will be interpreted as "1". •If E0 is commanded, the workpiece axis will stop (synchronized with the Z axis for a helical gear). (*3)
L	Workpiece axis rotation ratio designation Command the workpiece axis rotation ratio (number of gear teeth).	1 to 999	• If there is no command, the rotation ratio will be interpreted as "1".
R	Workpiece axis phase shift amount Command the amount to shift from the workpiece axis reference position to synchronize with the hob axis reference position.	-359999 to 359999 Decimal point input possible (*4)	•The commanded shift amount will be applied in the workpiece axis counter's positive direction. •The phase alignment is performed according to the shift amount specified by address R. •If address R is omitted, the operation is performed in the same way as when "R0" is commanded if the hob axis parameter "#3130 syn_spec/bit3" is "1". (This setting depends on the MTB specifications.) •When the input setting unit is "0.001°", the range will be "-359.999°" to "359.999°".
P	Gear torsion angle designation Command the torsion angle for the helical gear.	-89000 to 89000 Decimal point input possible (*5)	•If there is no P command, or if "P0" is commanded, a spur gear will be machined. •To move the Z axis in the plus direction after entering the hobbing mode, command the direction that the workpiece axis is twisted. P sign: when it is "+", + direction P sign: when it is "-", - direction   •When the input setting unit is "0.001°", the range will be "-89.000" to "89.000".

Address	Meaning	Command range (unit)	Remarks
Q	Module designation Command the normal module for helical gear. When inch input, command the diametral pitch.	Metric input Module designation 10 to 25000 Inch input Diametral pitch designation 100 to 2540000 Decimal point input possible (*6)	♦If there is no Q command for helical gear (when P is designated), the program error (P33) occurs. ♦For spur gear (when P is not designated, or P0 is commanded), the Q command is ignored. ♦If the input setting unit is 0.001 (mm) (or 0.0001 (inch^{-1})), the range will be as follows. [Module designation] 0.01 to 25. (0.01 to 25 (mm) [Diametral pitch designation] 0.01 to 254. (0.01 to 254 (inch^{-1}))

(*1) If a value exceeding the command range is commanded, the program error (P35) occurs.

(*2) There are two types of spindle designation 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.

(*3) When address E = 0 is commanded for spur gears, the workpiece axis will not rotate. Do not use this except for special cutting (cutting of only part of the gears, etc.).

(*4) The range that can be set depends on the input setting unit (parameter "#1003 iunit"). (Example) When the input setting unit is 0.000001°, the range is -359.999999° to 359.999999°.

(*5) If the decimal point input is OFF, the available setting range varies according to the input setting unit (parameter "#1003 iunit"). When the input setting unit is 0.000001°, the range is -89000000 to 89000000.

(*6) If the decimal point input is OFF, the available setting range varies according to the input setting unit (parameter "#1003 iunit"). If the input setting unit is "0.000001 (mm)" ("0.0000001 (inch^{-1})" in inch system), the range will be as follows.

	Available range
Metric system	10000 to 25000000
Inch system	100000 to 2540000000



Detailed description

Rotation ratio change during tool spindle synchronization II (hobbing) mode

The rotation ratio and the number of hob threads can be changed without stopping the hob axis or the workpiece axis during tool spindle synchronization II (hobbing) mode.

G114.3 E__ L__ P__ Q__ ; Rotation ratio change

E	Hob axis rotation ratio designation
L	Workpiece axis rotation ratio designation
P	Gear torsion angle designation
Q	Module or diametral pitch designation

(1) Each address can be omitted in the G114.3 command in the tool spindle synchronization II (hobbing) mode. If the address is omitted, modal value of the last command is used.

(Example) When changing only the workpiece axis rotation ratio (the modal value of the previous command is used for items other than the workpiece axis rotation ratio.)
G114.3 L50;

(2) If the following commands are issued, the program error (P33) occurs.

(a) When the hob axis number and workpiece axis number are changed

(b) When other than "0" is commanded by the E command in the E = 0 state, or "0" is commanded by the E command in the E ≠ 0 state.

(3) The workpiece axis rotation speed may be changed by rotation ratio. At this time, the acceleration/deceleration time constant follows the hobbing workpiece axis time constant (parameter "#2195 hob_tL").

(4) The "Spindle rotation speed synchronization completion" signal (X18A9) is turned OFF by changing the rotation ratio. This signal is turned ON when the workpiece axis rotation speed reaches the prescribed range for hob axis rotation speed after completing the rotation ratio change.

(5) The hob axis rotation speed cannot be changed while the rotation ratio is changed (during workpiece axis acceleration/deceleration). If the rotation command is issued for hob axis during the rotation ratio change, the commanded rotation speed is applied after completing the rotation ratio speed change.

(6) The helical gear machining by Z axis movement does not be executed while the rotation ratio is changed (during workpiece axis acceleration/deceleration).
The helical gears machining is executed after completing the rotation ratio change.

(7) The phase of hob axis and workpiece axis during rotation ratio changing (during workpiece axis acceleration/deceleration) or after changing is not warrantable. A phase cannot be aligned with gears machining of the last command.

(8) The "Hob axis delay (advance) monitoring", "Compensation control by workpiece axis" and "The workpiece axis feed forward control" are invalid while the rotation ratio is changed (during workpiece axis acceleration/deceleration).
These functions are valid after finishing the rotation ratio change.

(9) If the rotation ratio is changed during the hobbing mode in the phase alignment control, the phase alignment is performed after the rotation ratio was changed. The phase alignment is performed with the changed rotation ratio. For details, refer to the section "Phase change during tool spindle synchronization II (hobbing) mode".

Phase change during tool spindle synchronization II (hobbing) mode

The workpiece axis phase can be changed without stopping the hob axis or the workpiece axis during the tool spindle synchronization II (hobbing) mode. After the phase was changed, it is also possible to change the phase again.

The G114.3 command during the tool spindle synchronization control II (hobbing) mode changes the workpiece axis phase shift amount with address R.

G114.3 R__ ;	**Phase change**

R	Workpiece axis phase shift amount

Address R can be omitted.
When address R is omitted, the workpiece axis phase shift amount before this command is maintained regardless of the parameter "#3130/bit3".

(Example 1) When changing the workpiece axis phase shift amount:

G114.3 H4 D9 E1 L1 R30. ;	Hobbing starts. The phase alignment is performed with "R30.".
:	
G114.3 R40. ;	The phase is changed to "R40.".

(Example 2) When the same phase shift amount as when hobbing start is commanded:

G114.3 H4 D9 E1 L1 R30. ;	Hobbing starts. The phase alignment is performed with "R30.".
:	
G114.3 R30. ;	If the same phase as the current phase is commanded, the phase is not changed.

(Example 3) When address R is omitted:

G114.3 H4 D9 E1 L1 R30. ;	Hobbing starts. The phase alignment is performed with "R30.".
:	
G114.3 ;	If address R is omitted, the phase is not changed. The workpiece axis phase shift amount before this command is maintained.

Even if address R is not commanded in the hobbing mode command (G114.3) when hobbing starts, a phase change command can be performed. However, in this case, no modal value is specified for address R, so no phase change is performed if address R is omitted.

(Example 4) When changing the workpiece axis phase shift amount (no phase alignment at the start of hobbing):

G114.3 H4 D9 E1 L1 ;	Hobbing starts. The phase alignment is not performed.
:	
G114.3 R40. ;	The phase is changed to "R40.".

(Example 5) When address R is omitted (no phase alignment at the start of hobbing):

G114.3 H4 D9 E1 L1 ;	Hobbing starts. The phase alignment is not performed.
:	
G114.3 ;	The phase change is not performed. The phase alignment is not performed.

Combination of phase change and rotation ratio change during tool spindle synchronization II (hobbing) mode

The phase change and the rotation ratio change can be commanded at the same time by commanding any of addresses L, E, P, and Q together with the phase change command. In this case, the phase change is performed after the rotation ratio change is completed. When the rotation ratio is changed, the phase change is performed even if there is no change in the value of address R. Combination operations of the phase change and rotation ratio change are as shown in the table below.

For details on the phase change, refer to the section "Phase change during tool spindle synchronization II (hobbing) mode".
For details on the rotation ratio change, refer to the section "Rotation ratio change during tool spindle synchronization II (hobbing) mode".

	Rotation ratio change (G114.3) With address (L, E, P, or Q)	**Rotation ratio change (G114.3) Without address**
Phase change (G114.3) With address R	After the rotation ratio is changed, the phase change is performed.	Only the phase change is performed.
Phase change (G114.3) Address R omitted	After the rotation ratio is changed, the phase change is performed based on the modal value of address R. If no modal value is specified for address R, the phase alignment is not performed.	Neither rotation ratio change nor phase change is performed.

(1) The phase change is performed by aligning the reference position of the hob axis and the reference position of the workpiece axis with the absolute positions, as with the normal phase alignment.

(2) If the phase change is performed with a G114.3 command while the hob axis is stopped, the phase change is performed at the timing of the next S command. If the hob axis is rotating, the phase change is performed with a G114.3 command.

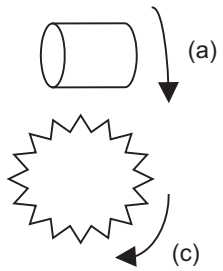
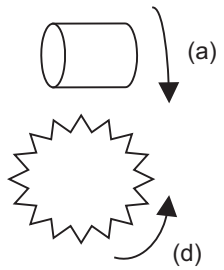
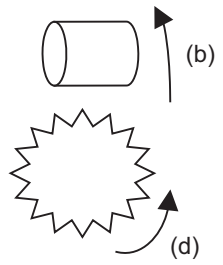
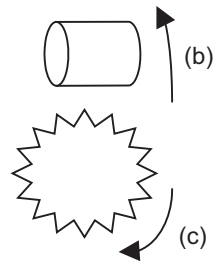
(3) If the phase change is performed with a G114.3 command while the hob axis is rotating, the phase change is performed by the "Phase alignment during hob axis rotation" control. For details, refer to the section "Tool spindle synchronization II (hobbing) command during hob axis rotation".

(4) If a cutting feed block is operated while the speed synchronization or phase synchronization is in progress, the operation error (M01 1056) occurs, and the cutting feed block does not start. After synchronization is completed, the cutting feed block starts.

Rotation speed and rotation direction

The rotation speed and rotation direction of the hob axis and workpiece axis during tool spindle synchronization II (hobbing) are as follows.

(1) The rotation speed and rotation direction of hob axis are the rotation speed commanded with the S command and the rotation direction commanded with the M command, etc., for the spindle selected as the hob axis.

When the sign of D command is "+"	When the sign of D command is "-"
	
	

(a) Hob axis: Forward rotation
(b) Hob axis: Reverse rotation
(c) Workpiece axis: + direction
(d) Workpiece axis: - direction

<Note>

- The rotation directions (+/- directions) of the workpiece axis shown above refer to the forward and reverse directions of the machine coordinate system of the workpiece axis.

(2) The workpiece axis rotation speed is determined by the hob threads specified using the hobbing mode command and the number of gear teeth.

$$Sw = Sh * \frac{E}{L}$$

Sw : Workpiece axis rotation speed (r/min)
Sh: Hob axis rotation speed (r/min)
E: Hob axis rotation ratio (number of hob threads)
L: Workpiece axis rotation ratio (number of gear teeth)

(3) The workpiece axis rotation direction is as follows.

The workpiece axis rotation direction is determined by the sign of the workpiece axis selection "D" commanded with the hobbing mode command.
In other words, when the "D" sign is "+", the workpiece axis will rotate in the same direction as the hob axis, and when the "D" sign is "-", the workpiece axis will rotate in the direction opposite to the hob axis.

(4) After tool spindle synchronization II (hobbing) was commanded, the relationship between the hob axis and workpiece axis rotation is held in all operation modes of automatic and manual modes until spindle synchronization cancel (G113.1) is commanded or until the "spindle synchronization cancel" signal is input.
Even during reset or feed hold, the hob axis and workpiece axis synchronization state is held.

Workpiece axis control

(1) When the hobbing mode is commanded during hob axis rotation, the workpiece axis accelerates up to the speed required in synchronization with the hob axis according to the hobbing workpiece axis time constant (#2195 hob_tL) using the constant-gradient acceleration/deceleration control. Then rotates in synchronization with hob axis.

(2) The "Axis selection" signal and "In axis motion" signal of workpiece axis are not output during the tool spindle synchronization II (hobbing) mode.

(3) If a manual movement command is issued to the workpiece axis during the tool spindle synchronization II (hobbing) mode, the manual movement will be superimposed on the workpiece axis movement with tool spindle synchronization. In this case, the "Axis selection" signal and "In axis motion" signal of the workpiece axis will be output.
Note that, if the movement command is issued in the manual reference position return mode, the operation error (0005) occurs.
An automatic movement command can be issued to the workpiece axis during the tool spindle synchronization II (hobbing) mode. Refer to "(2) Command compensation" in "Compensation control by workpiece axis" for details of the command to the workpiece axis.

(4) During the tool spindle synchronization II (hobbing) mode, the operations in respect to the "input" signals of external deceleration, interlock and machine lock for workpiece axis are as follows.

	Interlock	Machine lock	External deceleration
Movement by the hobbing function	Invalid	Invalid	Invalid
Movement by manual command	Valid for manual interlock	Valid for manual machine lock	Valid
Automatic compensation by incremental command	Valid for automatic interlock	Valid for automatic machine lock	Valid

(5) If the "Servo OFF" signal is input for the workpiece axis during the tool spindle synchronization II (hobbing) mode, the tool spindle synchronization II (hobbing) is canceled because synchronization cannot be maintained.

(6) The workpiece axis rotation speed is determined according to the hob axis rotation speed, so designate the hob axis rotation speed so that the workpiece axis cutting clamp speed is not exceeded.

(7) The C axis counter on each screen will be updated as shown below during the tool spindle synchronization II (hobbing) mode.

(a) When the workpiece axis is a rotary-type rotary axis

The axis will rotate in the 0.000 to 359.999 range in the normal manner.

(b) When the workpiece axis is a linear-type rotary axis (all coordinate positions linear type)

The axis will rotate in the 360° range including the machine coordinate position and workpiece coordinate position when hobbing starts.

(c) When the workpiece axis is a linear-type rotary axis (workpiece coordinate positions linear type)

The axis will rotate in the 360° range including the workpiece coordinate position when hobbing starts.

(Example)

Coordinate position when the hobbing starts		Rotation range	
125.000	(°)	0.000 to 359.999	(°)
750.500	(°)	720.000 to 1079.999	(°)
-252.200	(°)	-360.000 to -0.001	(°)

(8) If the hobbing command is issued before the workpiece axis completes reference position return, the program error (P430) occurs.

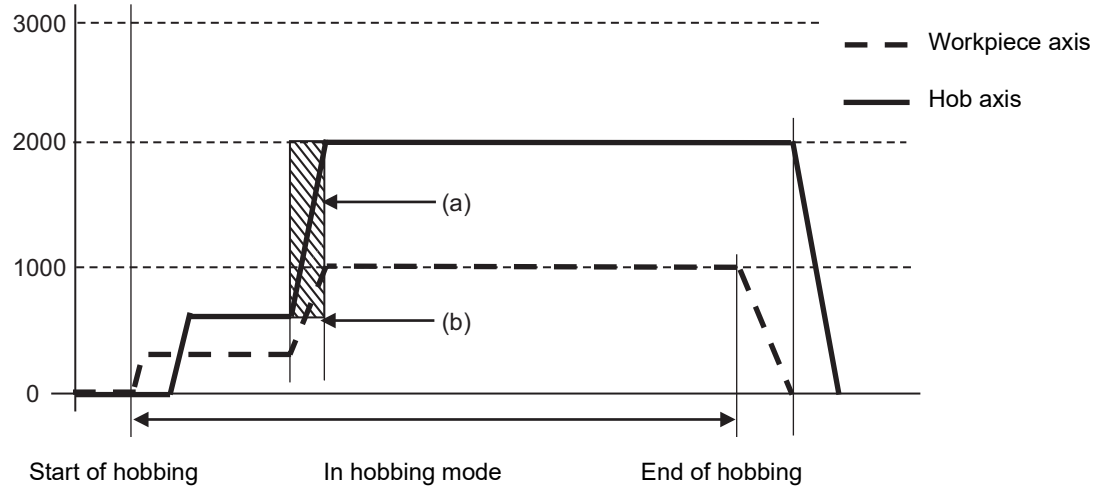
Acceleration/deceleration control

Normally, the workpiece axis that holds the workpiece has a greater load in acceleration/deceleration than the rotary tool spindle. By performing multi-step acceleration/deceleration control using the spindle synchronization acceleration/deceleration time constant of the workpiece axis with a larger load, the optimal acceleration/deceleration is possible under various machining conditions. These settings depend on the MTB specifications (parameter "#3130 syn_spec/bit7").

(1) Multi-step acceleration/deceleration based on the hob axis (#3130 syn_spec/bit7 = 0)

The hob axis and workpiece axis will carry out multi-step acceleration/deceleration with the spindle synchronization acceleration/deceleration time constant (spt) set for the spindle selected as the hob axis. However, the workpiece axis rotates at a speed determined by the tool spindle rotation ratio and the workpiece axis rotation ratio, so the acceleration rate is greater than that of the hob axis.

<Multi-step acceleration/deceleration example based on hob axis>



(a) Multi-step acceleration/deceleration example based on the hob axis

(b) The workpiece axis operates synchronously with the acceleration/deceleration of the hob axis.

(2) Multi-step acceleration/deceleration based on the workpiece axis (#3130 syn_spec/bit7 = 1)

When the parameter "#3130 syn_spec/bit7" is "1", multi-step acceleration/deceleration is performed using the spindle synchronization acceleration/deceleration time constant (spt) of the spindle selected as the workpiece axis. When this setting is used, the acceleration/deceleration is performed with a time constant according to the rotation speed ratio of each axis, as shown in the following formula.

The multi-step acceleration/deceleration time constant based on the workpiece axis is changed according to the hob axis rotation ratio specification and workpiece axis rotation ratio specification. This allows the acceleration/deceleration time to be maintained close to the setting of the spindle synchronization acceleration/deceleration time constant (spt) of the spindle selected as the workpiece axis, which enables the optimal acceleration/deceleration under various machining conditions.

[Multi-step acceleration/deceleration time constant based on workpiece axis]

$Wt = Ws \times (L/R)$

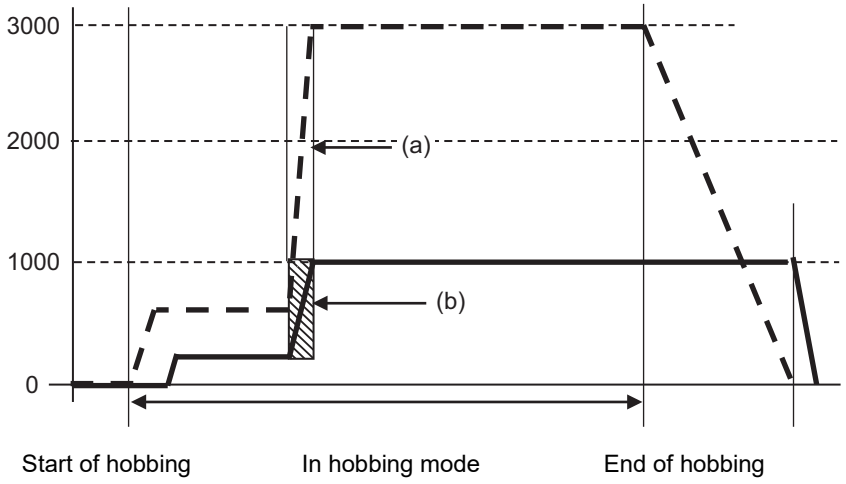
Wt: Multi-step acceleration/deceleration time constant based on workpiece axis
Ws: Spindle synchronization acceleration/deceleration time constant of workpiece axis (spt)
L: Tool spindle (hob spindle) rotation ratio (number of hob threads)
R: Workpiece axis rotation ratio (number of gear teeth)

<Note>

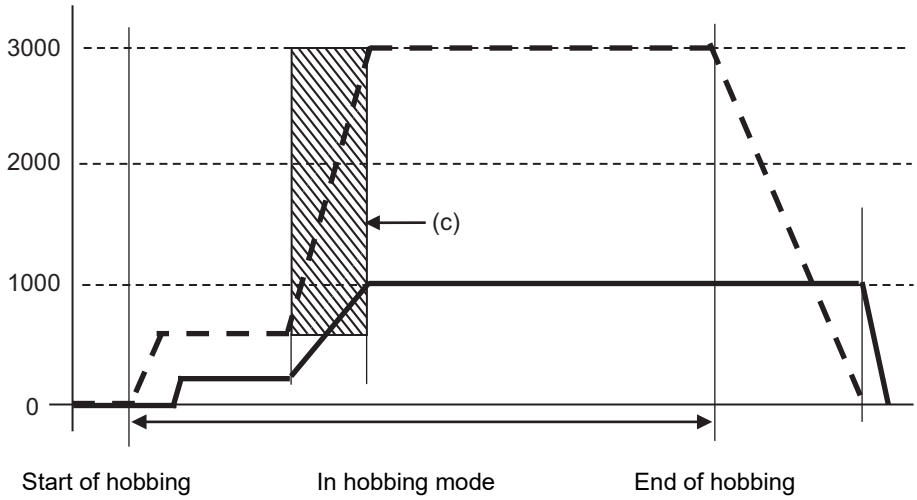
- The multi-step acceleration/deceleration based on the workpiece axis is effective when the workpiece axis is targeted for the spindle position control (spindle/C axis). When the workpiece axis is rotated with a servo axis, the multi-step acceleration/deceleration is performed based on the hob axis, regardless of the setting of parameter "#3130 syn_spec/bit7".

<Comparison example of multi-step acceleration/deceleration based on the hob axis and multi-step acceleration/deceleration based on the workpiece axis>

#3130/bit7 = 0
(Based on the hob axis)



#3130/bit7 = 1
(Based on the work-piece axis)



Hob axis

Workpiece axis

(a) This is the multi-step acceleration/deceleration based on the hob axis, so the acceleration/deceleration of the workpiece axis becomes severe.

(b) Multi-step acceleration/deceleration based on the hob axis

(c) The multi-step acceleration/deceleration is performed using the multi-step acceleration/deceleration time constant based on the workpiece axis.

Phase alignment control

The phase alignment control can be performed to keep the positional relationship between the hob axis and the workpiece axis the same every time.
When the phase alignment is commanded, the phase is aligned with the gear closest to the origin of the hob axis, taking into account the simplest integer ratio between the number of hob threads and the number of gear teeth.

To carry out phase alignment during hobbing, the spindle encoder commanded to the hob axis must have a Z phase and satisfy the following conditions.

Control system	Gear ratio conditions
Semi-closed control	Spindle side gear : Motor side gear = 1:1
Full-closed control	Spindle end : Encoder end = 1:1

Phase alignment control (Phase alignment shift amount)

The phase is aligned with the closest gear, taking into account the simplest integer ratio between the number of hob threads and the number of gear teeth. The phase alignment shift amount is determined by the number of hob threads, number of gear teeth, workpiece axis phase shift amount, and hob axis phase shift amount commanded in the hobbing mode command.

(1) Relationship between number of hob threads, number of gear teeth, and phase

The phase of one motor revolution is 360 (°); however, the hob axis and workpiece axis individually have a different phase weight depending on the hob thread number and number of teeth. In other words, considering the respective rotation speed ratios, the weight of one thread and one tooth is the same, and the phase can be calculated by combining the weights of the hob axis and workpiece axis.

(Example 1) L = 2, R = 3, Pw = 200°, Ph = 200°

(Per thread: 180°, Per tooth: 120°)

Pw × R = 200°× 3 = 600 The remainder of "600/360°" is 240°.	When one tooth is 360°, the phase is one tooth + 80°
Ph × L = 200°× 2 = 400 The remainder of "400/360°" is 40°.	When one thread is 360°, the phase is one thread + 20°.

Pw: Workpiece axis phase (°)

Ph: Hob axis phase (°)

L: Hob axis rotation ratio (number of hob threads)

R: Workpiece axis rotation ratio (number of gear teeth)

(2) Phase difference between hob axis and workpiece axis

If the phase difference is the difference between the hob axis phase and the workpiece axis phase as viewed from the workpiece axis, it can be expressed as follows, and the phase difference in (Example 1) is -200 (°). (Phase converted from one tooth or one thread to 360 (°))

Pd = (Ph × L) - (Pw × R) = 40 - 240 = -200°

Pd: Phase difference as viewed from the workpiece axis (°)

When the phase difference is calculated from the workpiece axis phase shift amount (*1) and the hob axis phase shift amount (*2), the difference is also obtained by combining the weights. The signs of the phase shift amount and the phase (equivalent to the phase shift amount) are reversed, so it can be expressed as follows.

Ps = - (Pe × L) + (Pa × R)

Ps: Phase difference due to the phase shift command as viewed from the workpiece axis (°)

Pa: Workpiece axis phase shift amount (°)

Pe: Hob axis phase shift amount (°)

(*1) Address A for G81.4 command, address R for G114.3 command

(*2) Address E for G81.4 command, none for G114.3 command

(3) Conversion of phase difference to each axis reference phase

By considering the weight of the rotation speed ratio in the phase difference, the phase difference can be converted to the phase of each axis.

(Example 1) Phase difference converted to the weight of the workpiece axis

P = Ps/R = Pa - (Pe × L/R)

(Example 2) Phase difference converted to the weight of the hob axis

P = Ps/L = (Pa × R/L) - Pe

Phase alignment control (Phase alignment operation)

(1) When tool spindle synchronization II (workpiece axis phase shift amount) is commanded using the hobbing mode command, the rotary axis commanded to the workpiece axis enters the spindle synchronization II (hobbing) control state.

(2) The hob axis rotation speed follows the Z phase detection speed (parameter "#3109 zdetspd") set in the parameters with the first S command issued for the hob axis after entering the hobbing control state.
If this command rotation speed is 0 (r/min), the workpiece axis will not start rotating, and instead will wait for the next S command.

(3) Phase alignment is carried out when the hob axis is stopped and the workpiece axis is rotating. If "Command position" is selected for the phase alignment position selection, the phase alignment accuracy will improve. This setting depends on the MTB specifications (parameter "#3130 syn_spec/bit6").

(4) The phase alignment is completed when the phase difference between the hob axis and the workpiece axis, excluding the "phase difference by the phase shift command", falls below the spindle phase synchronization achievement level (parameter "#3051 spplv"). If the hob axis rotation ratio (number of hob threads) is greater than the workpiece axis rotation ratio (number of gear teeth), set the weight of the setting value of the spindle phase synchronization achievement level (#3051 spplv) to "the weight of either the hobbing axis or the workpiece axis, whichever has the larger rotation ratio". However, this setting depends on the MTB specifications (parameter "#1703 cfg03/bit3").

(5) After the phases are aligned, the hob axis will accelerate/decelerate to the rotation speed commanded with the S command. The workpiece axis will accelerate/decelerate to the rotation speed obtained based on the hob axis rotation speed allowing for the hob axis and workpiece axis rotation ratio, and will enter the synchronized state.

Compensation control by workpiece axis

(1) Automatic compensation

The workpiece axis is controlled while constantly allowing for hob axis delay (advance) caused by disturbance, etc. This is especially effective in increasing the workpiece accuracy during heavy cutting. Automatic compensation is validated with parameters.
When the amount of the compensation added to the workpiece axis by hobbing conditions etc. changes greatly and rapidly, a servo alarm might occur for the workpiece axis. In that case, with the compensation amount through the primary delay filter, this enables the compensation amount fluctuation to further smoothen. However, the more widely the primary delay time constant is set, the more the effect of the compensation decreases, so the effect of the workpiece accuracy might not improve.

[Spindle NC parameter] (Machine parameter)

#3130 syn_spec/bit0: Tool spindle synchronization II (hobbing) automatic compensation selection
0 (OFF): No compensation
1 (ON): Hob axis delay (advance) is compensated with workpiece axis.

#3134 sphtc: Tool spindle synchronization II (hobbing) automatic compensation primary delay time constant
0: Primary delay filter control invalid
1 to 32768: Primary delay filter time constant setting unit (ms)

(2) Command compensation

Errors in the cutting workpiece shape caused by insufficient machine rigidity, etc., are compensated for with the workpiece axis command in the machining program.

(a) Command the workpiece axis compensation amount as an incremental position.

(b) Command the workpiece axis compensation amount direction in the workpiece axis rotation direction using a "+" command, and in the direction opposite the workpiece axis rotation using a "-" command.

(c) When the movement command is issued with an absolute position for the workpiece axis during the tool spindle synchronization II (hobbing) mode, the program error (P32) occurs.

Feedforward control during tool spindle synchronization II (hobbing) mode

A feedforward control can be issued for the hob axis and the workpiece axis during the tool / spindle synchronization control II (hobbing) mode.

(1) The hob axis feedforward control is controlled according to hob axis feedforward gain (parameter "#3135 sfwd_g").

(2) The workpiece axis feedforward control is controlled according to hob axis feedforward gain (parameter "#3135 sfwd_g") for the workpiece axis rotation contents of the hob axis rotation. The feedforward control is controlled according to workpiece axis feedforward gain (#2155 hob_fwd_g) for the helical compensation of the Z axis movement.

Tool spindle synchronization II (hobbing) command during hob axis rotation

The hobbing mode command and hobbing cancel mode command can be issued during hob axis rotation.

(1) When the hobbing mode command is issued during hob axis rotation, the rotary axis designated as workpiece axis accelerates up to the speed according to the hob and workpiece axes' rotation ratio command. This acceleration follows the hobbing workpiece axis time constant (parameter "#2195 hob_tL") and uses the constant-gradient acceleration/deceleration control. If the setting of hobbing workpiece axis time constant is outside the setting range, set the maximum value in the range.

(2) When the hobbing mode is commanded with the workpiece axis phase shift amount specified, the phase alignment between the hob axis and the workpiece axis is performed after the workpiece axis acceleration is completed.

(3) If "Command position" is selected for the phase alignment position selection, the phase alignment accuracy will improve. This setting depends on the MTB specifications (parameter "#3130 syn_spec/bit6").

(4) The phase alignment is completed when the phase difference between the hob axis and the workpiece axis, excluding the "phase difference by the phase shift command", falls below the spindle phase synchronization achievement level (parameter "#3051 spplv"). If the hob axis rotation ratio (number of hob threads) is greater than the workpiece axis rotation ratio (number of gear teeth), set the weight of the setting value of the spindle phase synchronization achievement level (#3051 spplv) to "the weight of either the hobbing axis or the workpiece axis, whichever has the larger rotation ratio". However, this setting depends on the MTB specifications (parameter "#1703 cfg03/bit3").

(5) Synchronization is established after phase alignment is completed.

(6) If the hobbing cancel mode command is issued during hob axis rotation, the workpiece axis decelerates and stops. This deceleration follows the hobbing workpiece axis time constant (parameter "#2195 hob_tL"), and uses the constant-gradient acceleration/deceleration. If the setting of hobbing workpiece axis time constant is outside the setting range, set the maximum value in the range.

Speed clamp during hobbing mode

If the rotation speed exceeds the maximum rotation speed (parameter "#3005 smax1") during hobbing mode, the rotation speed is clamped.

(1) If the rotation speed of the tool spindle (hob spindle) exceeds the maximum rotation speed (parameter "#3005 smax1") of the tool spindle (hob spindle), the rotation speed is clamped at the maximum rotation speed of the tool spindle (hob spindle).

(2) If the rotation speed of the workpiece spindle (spindle/C axis) exceeds the maximum rotation speed (parameter "#3005 smax1") of the workpiece spindle, the rotation speed of the tool spindle (hob spindle) is clamped so that the rotation speed of the workpiece spindle does not exceed the maximum rotation speed.

[At start of hobbing)

If the rotation speed of the workpiece spindle exceeds the maximum rotation speed at the start of hobbing, the rotation speed of the tool spindle (hob spindle) is clamped from the start of hobbing.

[In hobbing mode]

If the S command is changed during hobbing, or if the workpiece spindle speed exceeds the maximum rotation speed when the phase is changed, the rotation speed of the tool spindle (hob spindle) is clamped.

[Tool spindle (hob spindle): Spindle rotation speed clamp speed]

Sh = Smax × (R/L)

Smax : Workpiece axis rotation speed (r/min)
Sh: Tool spindle (hob spindle) rotation speed (r/min)
L: Hob axis rotation ratio (number of hob threads)
R: Workpiece axis rotation ratio (number of gear teeth)

(3) The rotation speed of the workpiece spindle (servo axis) is not clamped. Be careful of the rotation speed when using it.

[Hobbing with spindle stopped]

If the rotation speed of the workpiece spindle exceeds the maximum rotation speed (parameter "#3005 smax1") of the workpiece spindle, the rotation speed of the tool spindle (hob spindle) is clamped.

[Hobbing during spindle rotation]

If the rotation speed of the workpiece spindle exceeds the maximum rotation speed (parameter "#3005 smax1") of the workpiece spindle with the commanded rotation speed at the start of hobbing, the rotation speed of the tool spindle (hob spindle) is clamped from the start of hobbing.



Relationship with other functions

Synchronous tapping cycle

(1) The synchronous tapping spindle cannot be commanded as the hob spindle of tool spindle synchronization II (hobbing).
The operation error (M01 1007) occurs, which causes automatic operation to pause.

(2) The synchronous tapping cannot be commanded using the hob axis in tool spindle synchronization II (hobbing). The operation error (M01 1139) occurs, which causes automatic operation to pause.

Spindle synchronization I, Spindle synchronization II, Tool spindle synchronization IA/IB (spindle-spindle, polygon)

(1) The tool spindle synchronization II (hobbing) mode cannot be commanded during spindle synchronization I, spindle synchronization II, or tool spindle synchronization IA/IB (spindle-spindle, polygon) ON. The operation error (M01 1005) occurs.

(2) Spindle synchronization I, spindle synchronization II, tool spindle synchronization IA/IB (spindle-spindle, polygon) cannot be commanded during the tool spindle synchronization II (hobbing) mode. The operation error (M01 1005) occurs.

NC reset, Emergency stop

(1) Even if the "reset" signal is input, the tool spindle synchronization II (hobbing) maintains synchronization. However, the synchronization is canceled at emergency stop.

Arbitrary axis exchange command (G140), arbitrary axis exchange return command (G141), reference axis arrange return command (G142)

(1) The arbitrary axis exchange command (G140), arbitrary axis exchange return command (G141) or reference axis arrange return command (G142) cannot be issued in the part system where the tool spindle synchronization II (hobbing) is being performed. The program error (P501) occurs. However, hobbing that uses the axis in arbitrary axis exchange mode is possible.

(2) The workpiece axis of tool spindle synchronization II (hobbing) cannot be commanded as the axis exchange target. Doing so triggers the arbitrary axis exchange disable state.

Door interlock I / Door interlock II

(1) If door interlock I or door interlock II is turned ON during the tool spindle synchronization II (hobbing) mode, tool spindle synchronization II (hobbing) will be canceled because synchronization cannot be maintained.

Spindle-mode rotary axis control

Tool spindle synchronization II (hobbing) cannot be executed using the spindle-mode rotary axis as the hob axis. The operation error (M01 1024) occurs.
However, tool spindle synchronization II (hobbing) can be executed using the spindle-mode rotary axis as the workpiece axis.

3-dimensional coordinate conversion

(1) The 3-dimensional coordinate conversion cannot be commanded in the part system for tool spindle synchronization II (hobbing). The program error (P922) occurs.
(2) Tool spindle synchronization II (hobbing) cannot be commanded during 3-dimensional coordinate conversion. The program error (P921) occurs.

High-speed high-accuracy control

When tool spindle synchronization II (hobbing) is commanded while the high-speed high-accuracy control is valid or when high-speed high-accuracy control is commanded during tool spindle synchronization II (hobbing), high-speed high-accuracy control is temporarily canceled, and tool spindle synchronization II (hobbing) is executed preferentially.

Tool center point control

(1) Tool spindle synchronization II (hobbing) is not available during tool center point control. The program error (P942) occurs.
(2) The tool center point control command can be issued during tool spindle synchronization II (hobbing).

Inclined surface machining command

(1) Tool spindle synchronization II (hobbing) is not available during execution of the inclined surface machining command. The program error (P951) occurs.
(2) The inclined surface machining command cannot be issued during tool spindle synchronization II (hobbing). The program error (P952) occurs.



Precautions and restrictions

(1) When cutting helical gear, correct cutting feed will not be possible in the synchronous feed mode, so always cut in the asynchronous feed mode.

(2) To carry out phase alignment when machining a helical gear, phase alignment will not be carried out correctly if the Z axis is moving, so always carry out phase alignment control when the Z axis is stopped.

(3) The linear-type rotary axis for the absolute position system cannot be used as the hobbing workpiece axis. If used, the absolute position detection alarm (Z70 0002) occurs after the power was turned OFF and ON.

(4) If hobbing control is carried out using the linear-type rotary axis as the hob axis, the current value will be illegal when the hobbing is canceled. In this case, preset the counter after canceling hob machining.

(5) If the hob axis rotation ratio is set to "0", phase alignment will not be carried out. Even if the workpiece axis phase shift amount is commanded, it is ignored.

(6) When the parameter "#3133 spherr" is "0", "Hob axis delay excess" signal (X18B3) is not output. #3133 is a parameter that sets the allowable angle for the hob axis lag or lead, and depends on the MTB specifications.

(7) The following R registers are not updated while the hob axis is accelerating or decelerating.

- "Spindle synchronization: Phase error /Hob axis delay angle" signal (R6516)
- "Spindle synchronization: Maximum phase error/Maximum hob axis delay angle" signal (R6517)

(8) When the R6516 value exceeds the value of the parameter "#3133 spherr", CNC only outputs the "Hob axis delay excess" signal. For information about how to take an action, contact the MTB representative.

(9) G00 or G01 command for the workpiece axis from the machining program should be an incremental command. If it is an absolute command, the program error (P32) occurs.

(10) Always set the position loop gain of the hob axis and the workpiece axis to the same value. If different values are set, the machining accuracy is not warrantable.

- Hob axis : #13003 SP003, #13036 SP036/bit4
- Workpiece axis (NC axis) : #2203 SV003, #2204 SV004, #2257 SV057
- Workpiece axis (Spindle/C axis) : #13002 SP002, #13035 SP035/bitC

(11) While the program format is switched, the H address is not handled as an axis address. When the parameter "#1565 helgear" is set to "H", if a helical gear machining is performed while the program format is switched, the program error (P33) occurs.

(12) Do not command hobbing (tool spindle synchronization II) during synchronous control of the workpiece axis (C1 axis) and rear workpiece axis (C2 axis). When hobbing is commanded during synchronous control, rear workpiece axis (C2 axis) and workpiece axis (C1 axis) will not operate in synchronization (because this does not trigger an alarm), and this may twist the workpiece.

<Note>

- If hobbing is commanded during synchronous control, only operations for the spindle (hob axis) and workpiece axis (C1 axis) can be carried out.

(13) Command the hobbing mode with the workpiece axis stopped. When hobbing is commanded while the workpiece axis is rotating, hobbing mode turns ON after the stop of the workpiece axis was confirmed.

(14) While the speed synchronization or phase synchronization is being performed, the cutting feed block results in the operation error (M01 1056). After synchronization is completed, the cutting feed block starts.

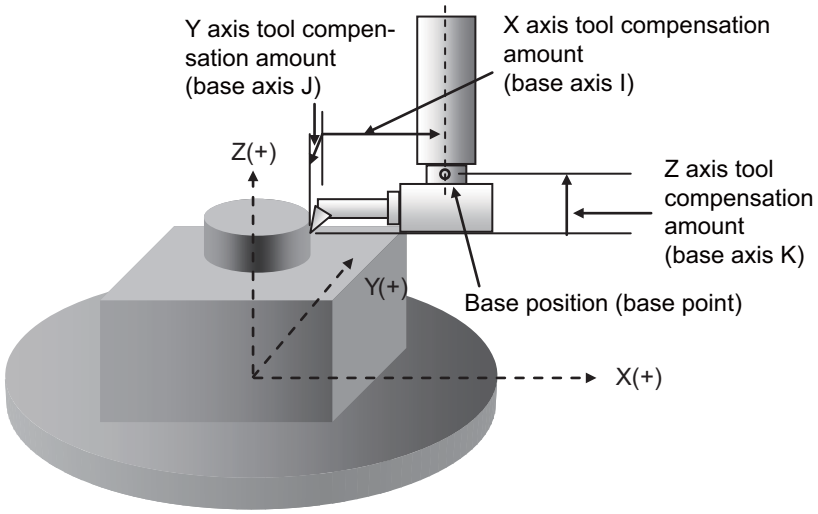
19

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 basic axes (X, Y and Z axes) to be compensated from the tool base position (base point).
To set the compensation amount of the three basic 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 number depends on the specifications (number of tool compensation sets).
If the commanded compensation No. exceeds the specification range, the program error (P170) occurs.
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) If the H command is issued alone, the operation depends on the MTB specifications (parameter "#1274 ext10/bit3").

#1274/bit3 = 0	The compensation amount corresponding to the compensation number does not become valid. The compensation amount designated by the previous command is retained.
#1274/bit3 = 1	The compensation amount corresponding to the commanded compensation number becomes valid.

(3) If G43.7 is commanded with a type other than tool compensation type II of the machining center, the program error (P39) occurs.

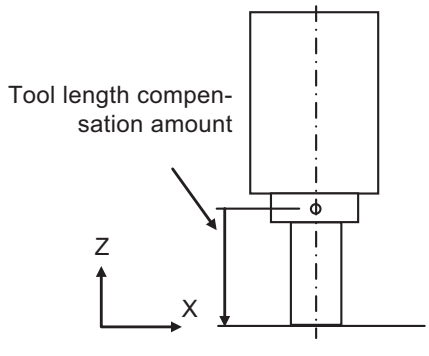
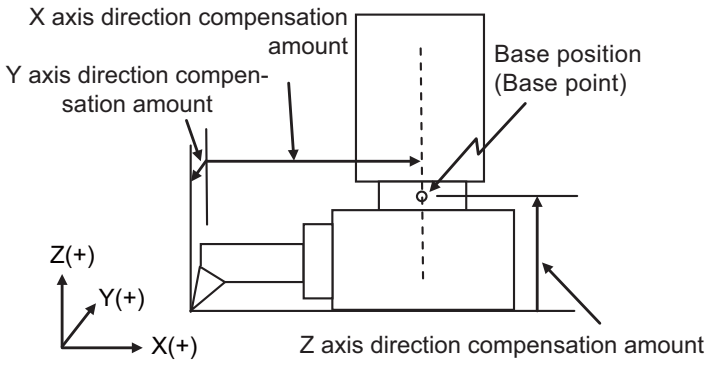
Detailed description

Three basic 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 basic 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)]
	
The H address is used to compensate only one axis.	The H address is used to compensate three axis directions.

Start-up and cancel operations

(1) When G43.7 or G49 is commanded, the compensation operation depends on the MTB specifications (parameter "#1268 ext04/bit6").

#1268 ext04/bit6	Tool length offset operation
0	Compensation operation by axis movement (axis movement type)
1	Compensation operation by coordinate shift (coordinate shift type)

[Case to perform tool position compensation by shift movement (#1268 ext04/bit6 = 0)]

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 command or incremental 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.

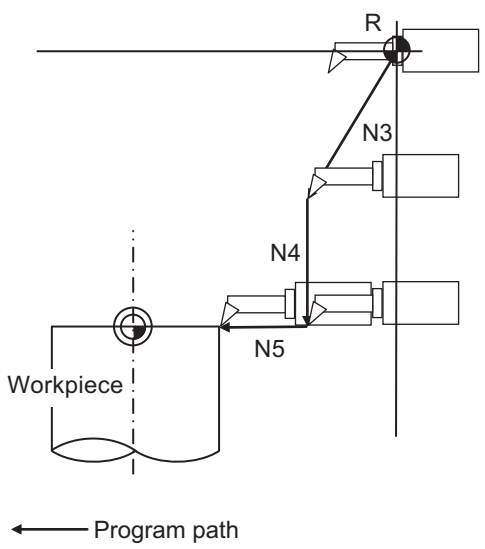
When no movement command is included in the same block as for G43.7 or G49, the operation depends on the MTB specifications (parameter "#1247 set19/bit0" (Movement by tool length compensation command)). For details, refer to "Movement by tool length compensation".

For absolute command

```
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 ;
```

For incremental command

```
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 ;
```



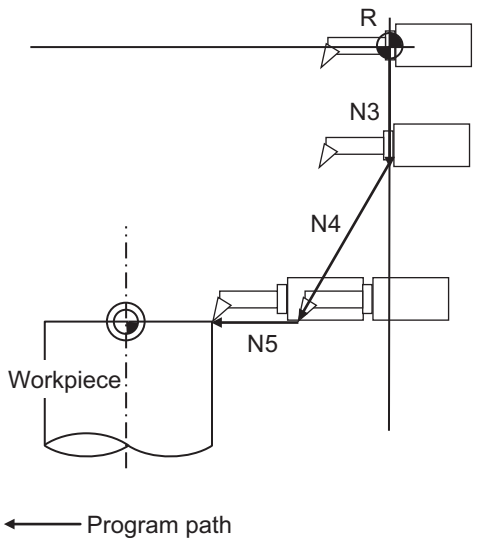
[Case to perform tool position compensation by coordinate shift (#1268 ext04/bit6 = 1)]

The coordinate shift operation in tool position compensation differs depending on the combination of the absolute command or the incremental command and the axis to be compensated that is commanded or not commanded in block G43.7.

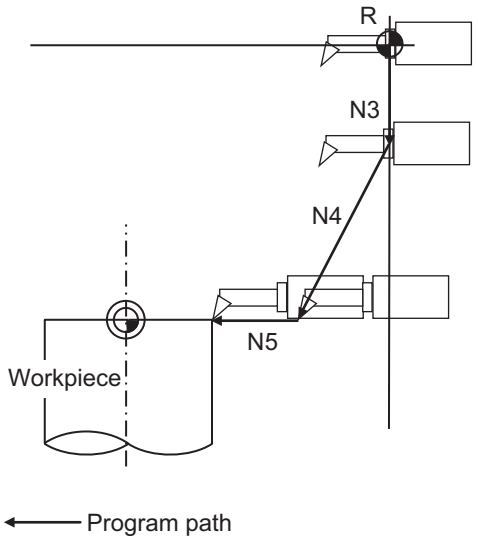
	Absolute command	Incremental command
Axis commanded in block G43.7	The actual end point is set to the coordinate position for which the specified compensation amount is compensated for the coordinates at the end point.	The increment is calculated in the coordinate system after the coordinate shift, thereby, causing no movement by the compensation amount.
Axis not commanded in block G43.7	Only the coordinate system is shifted, and compensation is performed with the subsequent absolute commands.	

<table>
<tr>
<td>For absolute command

N1 G91 G28 X0 Y0 Z0 ;
N2 G00 G90 ;
N3 G43.7 Z80. H01 ;
N4 X90. Y0. Z0. ;
N5 G01 X50. F500 ;

The absolute command is not specified for the X axis in block N3, so no compensation is performed for the X axis in block N3, and only the coordinate system is shifted.
The X axis compensation is performed with the absolute command in block N4.</td>
<td>
</td>
</tr>
<tr>
<td>For incremental command

N1 G91 G28 X0 Y0 Z0 ;
N2 G00 G91 ;
N3 G43.7 Z-50. H01 ;
N4 G90 X90. Y0. Z0. ;
N5 G01 X50. F500 ;

This is an incremental command in block N3, so the axis moves by the commanded incremental position without compensation.
Compensation is performed with the absolute command in block N4.</td>
<td>
</td>
</tr>
</table>

(2) When the compensation amount is changed during compensation, whether or not to move the axis by the changed compensation amount by the axis movement or the coordinate system shift depends on the MTB specifications (parameter "#1268 ext04/bit5").
However, if the parameter "#1292 ext28/bit3" is "1", it is applied in the next block.

#1268 ext04/bit5	Tool length offset operation of changed amount
0	Axis movement type
1	Coordinate shift type

Machining program	#1268/bit5 = 0 Axis movement type		#1268/bit5 = 1 Coordinate shift type		Description
	Program position	Machine position	Program position	Machine position	
N10G10L10P1R-50. ;	0	0	0	0	Z axis compensation amount : "-50.000"
N20G91G28X0Y0Z0 ;	0	0	0	0	Incremental command
N30G43.7H1 ;	50	0	50	0	Tool position compensation (Coordinate shift)
N40G01Z-10. ;	40	-10	40	-10	Incremental command
N50G90G10L10P1R-200. ;	40	-10	40	-10	Change of Z axis compensation amount : "-200.000"
N60G01Z-10. ;	30	-170	180	-20	Coordinate shift or axis movement (Depends on "#1268/bit5".)

Compensation No.

When address H is in the same block as G43.7, the offset number commanded with address H is set as the H modal.

[Case to command address H in the same block as G43.7]

(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 "h1".
G49; :	The tool length compensation is canceled.
G43.7; :	Used again as the tool compensation amount of "h1".

(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 "h1".
G43.7 Hh2 ; :	Used as the tool compensation amount of "h2".

(3) When the H command is issued alone in a block during G43.7 modal, the compensation amount in modal mode is applied continuously.
However, only when the "H0" command is issued, the tool position compensation mode is canceled.

G43.7 Hh1 ; :	Used as the tool compensation amount of "h1".
G43.7 Hh2 ; :	Used as the tool compensation amount of "h2".
Hh3 ; :	The compensation amount designated in "h2" is applied continuously.

[When the address H is commanded alone]

When the parameter "#1274 ext10/bit3" is "1", the H modal can be updated even if the address H of the tool compensation function is commanded alone.
Refer to "12.7.1 H Stand-alone Command" for details.

[Tool offset command check]

It is possible to make a program error occur when the tool offset No. is "0" or when the offset amount specified with the tool offset No. is "0", to prevent commands from being forgotten.
When offset amount check is valid and all the tool offset amounts of three basic axes are "0" at the tool position compensation (G43.7) command, the program error (P169) occurs.
Refer to "12.7.2 Tool Offset Command Check" for details.

Tool compensation cancel at reference position return operation

If the reference position return is performed by G28 or manually, the program changes to G49 mode (tool compensation cancel mode) when the reference position return is completed.
However, when commanding G28, G30, or G30.1 using parameters, or when manually operating the reference position return, it is possible to temporarily cancel tool position compensation and then apply tool position compensation when commanding the movement of the axis to be compensated. If no movement command is issued for the axis to be compensated in the next block, tool position compensation is not applied in that block. Tool position compensation is applied when the movement command is issued.
However, if the parameter "#1292 ext28/bit3" is "1", tool position compensation is applied in the next block. (This setting depends on the MTB specifications.)
When it is temporarily canceled, the modal is retained, thereby, causing G49 not to be commanded.
When the dog-type reference position return is performed, tool position compensation command is canceled.

[Operation at automatic reference position return]

	#1268 ext04/bit7	Automatic reference position return
When the axis reached the reference position:	0	Cancel
	1	Temporary cancel
When the axis moves after the above:	0	Cancel
	1	Applies the tool length offset before the axis reaches the reference position.

(1) Dog-type and high-speed type operation (parameter #1268 ext04/bit7 = 0)

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.

(2) High-speed type operation (parameter #1268 ext04/bit7 = 1)

G43.7 Xx1 Zz1 Hh1 ; :	
G28 Xx2 Zz2 ;	When the axis reaches the reference position, the operation is temporarily canceled.
G01 Xx2 Zz2 Ff2 ; :	The end point is set to the coordinate position that is offset in the positive direction by the compensation amount of offset number "h1" from the end point "z2".

[Operation at manual reference position return]

	#8122	Manual reference position return
When the axis reached the reference position:	0	Cancel
	1	Temporary cancel
When the axis moves after the above:	0	Cancel
	1	Reactivates tool length compensation amount that is applied before the axis reaches the reference position.

(1) Manual dog-type reference position return operation (The same operation is also performed when #8122 is set to "0" and the manual high-speed reference position return is valid.)

G43.7 Xx1 Zz1 Hh1 ; :	
:	If an interruption occurs in the manual dog-type reference position return, the operation is canceled when the axis reaches the reference position.
G01 Xx2 Zz2 Ff2 ; :	Performs the same operation as that in G49 mode.

(2) When #8122 is set to "1" and the manual high-speed reference position return is valid:

G43.7 Xx1 Zz1 Hh1 ; :	
:	If an interruption occurs in the manual dog-type reference position return, the operation is canceled when the axis reaches the reference position.
G01 Xx2 Zz2 Ff2 ; :	The end point is set to the coordinate position that is 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.

Note

- When the compensation amount is temporarily canceled, if "G05 P10000" (high-speed high-accuracy control II/III command) or "G07.1" (cylindrical interpolation command) is executed before a movement command, the program error (P29) occurs.

Operation when the compensation amount is changed during tool position compensation (G43.7)

(1) The compensation start operation for the compensation amount changed during tool position compensation (G43.7) depends on the setting of parameter "#1292 ext28/bit3".

#1292 ext28/bit3	Compensation start
0	Starts compensation from the time the movement command for the axis to be compensated is issued.
1	Starts compensation from the next block.

"Compensation amount" shown here refers to the compensation amount changed by the following operation.

- Compensation amount that is temporarily canceled with the G28/G30/G30.n, etc. command when the parameter "#1268 ext04/bit7" is "1".
- Compensation amount that is temporarily canceled with the G53 command.
- Compensation amount changed when the offset amount is changed during G43.7 mode.

As an example, below shows the compensation operation after G53 temporary cancellation in the case of coordinate shift type (#1268/bit6 = 1).

(Example) At the time of G53 temporary cancellation, compensation starts from the time the movement command for the axis to be compensated is issued. (#1292 ext28/bit3 = 0)
When X is set to "10.", Y to "0." and Z to "-50." for the compensation amount of compensation No. 1:

Machining program	X axis		Z axis	
	Program position	Machine position	Program position	Machine position
N10 G91G28X0.Y0.Z0. ;	0	0	0	0
N20 G90 G43.7X100.Y0.Z-120.H1 ;	100	110	-120	-170
N30 G01X50.F1000 ;	50	60	-120	-170
N40 G53X20.Z-10. ;	20	20	-10	-10
N50G00X0 ;	0	10	-10	-10
N60G00Z0 ;	0	10	0	-50

The Z axis does not move in block N50, so the compensation operation is not performed.

(Example) At the time of G53 temporary cancellation, compensation starts from the next block. (#1292/bit3 = 1)
When X is set to "10.", Y to "0." and Z to "-50." for the compensation amount of compensation No. 1:

Machining program	X axis		Z axis	
	Program position	Machine position	Program position	Machine position
N10 G91G28X0.Y0.Z0. ;	0	0	0	0
N20 G90 G43.7X100.Y0.Z-120.H1 ;	100	110	-120	-170
N30 G01X50.F1000 ;	50	60	-120	-170
N40 G53X20.Z-10. ;	20	20	-10	-10
N50G00X0 ;	0	10	40	-10
N60G00Z0 ;	0	10	0	-50

Compensation is also performed for the Z axis without a movement command in block N50.

(2) If the next block is a macro statement, when the parameter "#8101 MACRO SINGLE" is "0", compensation does not start in the macro statement, but when it is "1", compensation starts.

(3) For the axis movement type (#1268/bit6 = 0), compensation restarts with the axis movement. However, even in the case of the axis movement type, compensation starts with the coordinate shift operation if the G code in the table below is used.

G code	Function	G code	Function
G04	Dwell	G116	Start point timing synchronization
G05.1	Spline interpolation / high-speed high-accuracy control I	G120.1	Machining condition selection I
G07	Hypothetical axis interpolation	G121	Machining condition selection I cancel
G08	High-accuracy control	G122	Activate sub part system I
G11	Programmable data input cancel	G130	Selection of axis (axes) for feedrate command
G12	Circular cutting CW	G140	Arbitrary axis exchange command
G13	Circular cutting CCW	G141	Arbitrary axis exchange return command
G34	Special fixed cycle (bolt hole circle)	G142	Arbitrary axis exchange/reference axis arrange return command
G35	Special fixed cycle (line at angle)	G145	Cancel sub part systems
G36	Special fixed cycle (arc)	G160	Torque limitation skip
G37.1	Special fixed cycle (grid)	G162	Spindle speed fluctuation detection
G53	Basic machine coordinate system selection (*1)	G162.1	Cutting load control
G113.1	Spindle synchronization I cancel	G163	Spindle speed fluctuation detection cancel
G114.1	Spindle synchronization I	G180	Interactive cycle insertion
G115	Start point timing synchronization	G186	Interference check III setting data valid

(*1) When the parameter "#1293 ext29/bit7" is "1", compensation starts. When "0" is set, compensation does not start.

(4) If the G code in the following table is commanded in the next block, compensation starts from the next block.

G code	Function	G code	Function
G05P0	High-speed high-accuracy control II/III OFF	G92.1	Workpiece coordinate system preset
G10	Compensation data input by program		

(5) When the parameter "#1268 ext04/bit6" is set to "1" (compensation operation by coordinate shift) and the coordinate system setting (G92) is commanded in the following cases, the program error (P294) occurs.

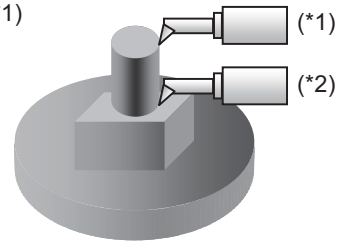
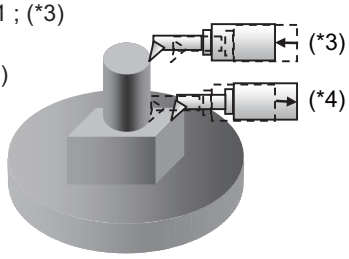
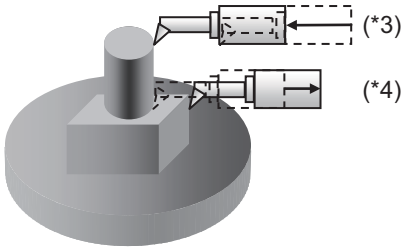
- The coordinate system setting (G92) has been commanded when the tool position compensation amount was temporarily canceled by a command such as G53, G28, G30, or G30.1.
  However, if G92 and G53 are commanded in the same block (G92 G53 X0 Y0 Z0;), no error occurs.
- The tool position compensation amount was changed during tool position compensation (G43.7), and then the coordinate system setting (G92) was commanded without the compensation amount being applied.

(6) If the following parameters are both set to "1", the MCP alarm (Y51 108) occurs, thereby, disabling the automatic operation from being started. (These settings depend on the MTB specifications.)

- "#1247 set19/bit0" (Movement by tool length compensation command)
- "#1292 ext28/bit3" (Operation at changing tool length compensation amount)

Movement by tool length compensation command

When there is no movement command 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").

<table>
<tr><th>G43.7/G49</th><th>Not move by the compensation amounts
(#1247 set19/bit0 = 1)</th><th>Moves by the offset amounts
(#1247 set19/bit0 = 0)</th></tr>
<tr><td rowspan="2">Without movement commands</td><td>:
G00 Xx Yy Zz ;
G43.7 H1 ; (*1)
:
G49 ; (*2)
:

</td><td>:
G00 Xx Yy Zz ;
G43.7 H1 ; (*3)
:
G49 ; (*4)
:

</td></tr>
<tr><td>(*1) Not moved.
(*2) Not moved.
If tool position compensation is commanded alone in a block, the axis does not move, but the tool compensation amount is applied to the program position counter.</td><td>(*3) Movement by (+) compensation amount
(*4) Movement by (-) compensation amount
If the tool position compensation is commanded alone in a block, the axis moves by the tool length compensation amount.</td></tr>
<tr><td rowspan="2">With movement commands</td><td colspan="2">:
G00 Xx Yy Zz ;
G43.7 H1 X10.; (*3)
:
G49 X5. ; (*4)
:

</td></tr>
<tr><td colspan="2">(*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.</td></tr>
</table>

Note

(1) When the parameter "#1268 ext04/bit6" is "1" (compensation operation by coordinate shift), the compensation operation is performed by the coordinate shift regardless of the setting of the parameter "#1247 set19/bit0". When using the movement by tool length compensation command, set the parameter "#1268 ext04/bit6" to "0" (compensation operation by axis movement).

(2) If the parameter "#1247 set19/bit0" (Movement by tool length compensation command) is "1" and any of the following parameters is set to "1", the MCP alarm (Y51 108) occurs, thereby, disabling the automatic operation from being started. (These settings depend on the MTB specifications.)

- "#1274 ext10/bit3" (Tool offset number H stand-alone command (for M system only))
- "#1292 ext28/bit3" (Operation at changing tool length compensation amount)

(3) For the G49 command, command it at a safe position where the tool does not interfere with the machine in consideration of the compensation canceling operation.
When the parameter "#1247 set19/bit0" is set to "0", the axis moves to the position where the compensation was canceled even though there is no axis command in the G49 command block.

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 occurs.

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 cutting	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)	○	○

Modal group	G code	Function	A	B	C
9	G73 - G76 G81 - G89	Fixed cycle for drilling	P801	○	P801
14	G66	User macro modal call	- (*4)	○	○
16	G68	3-dimensional coordinate conversion mode ON	P923	○	○
	G68.2/G68.3	Inclined surface machining command	P954	○	○
19	G50.1	Mirror image by G code cancel	P801	○	○
	G51.1	Mirror image by G code 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) occurs. If it is called after G43.7, the program error (P32) occurs.

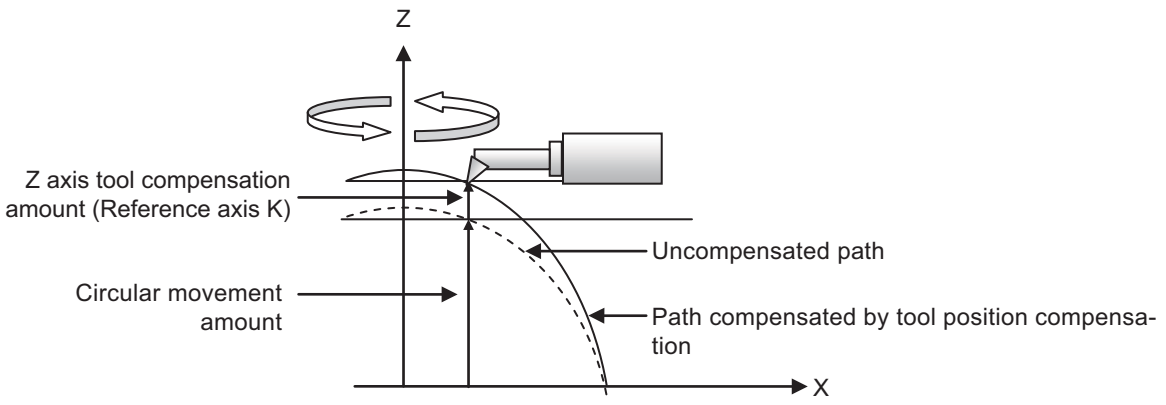
(*3) When "G43.7 G43 H1;" is commanded, the G43 commanded later is enabled.

(*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. However, when the parameter "#1268 ext04/bit7" is "1", the G43.7 modal can be retained when the reference position 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

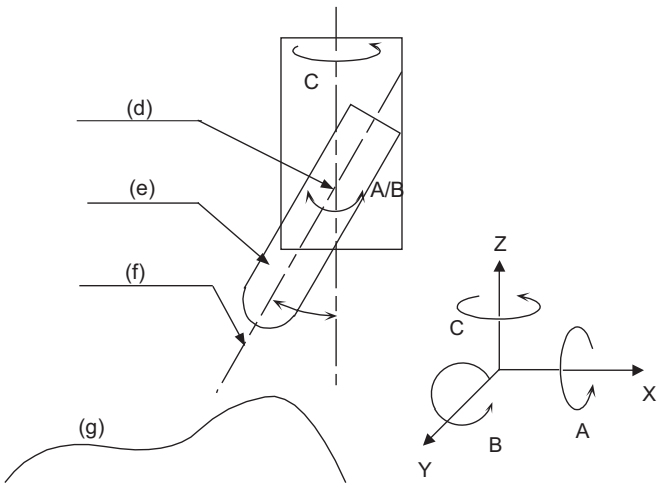
The tool length can be compensated for in the tool axis direction even when the rotary axis rotates and the tool axis direction becomes other than the Z axis direction. By using this function, and setting the deviation between the tool length amount set in the program and the actual tool length as the 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.

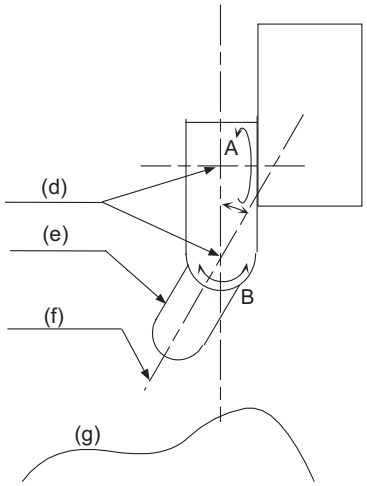
(2) Machine configuration

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.



A axis and C axis, or B axis and C axis



A axis and B axis

(d) Rotation center
(e) Tool
(f) Axis direction (compensation direction)
(g) Workpiece

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 "#2031 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

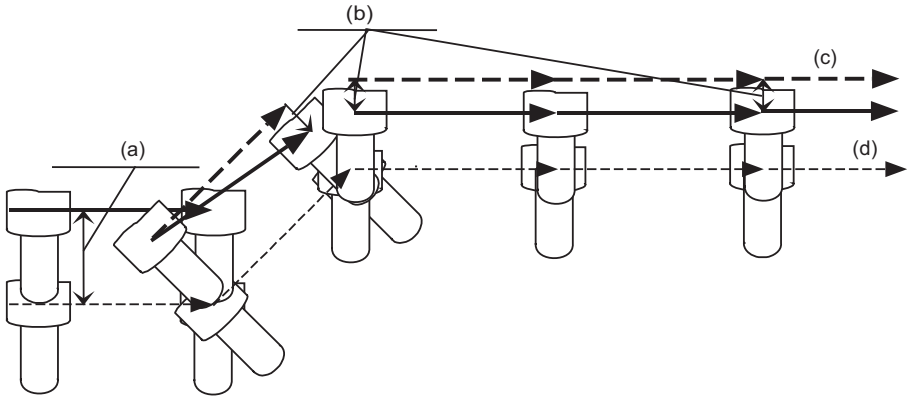
(1) When the following conditions have been met, the handle movement amount is added to the tool length compensation amount in the tool axis direction by rotating the manual pulse generator.
- ♦When the operation mode is MDI, memory or tape operation mode and the state is "during single block stop", "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 is ON).
- ♦In the tool handle feed & interruption mode (YC5E is ON).
- ♦The 3rd axis (tool axis) is selected for the handle selection axis.

(2) The change amount is canceled when the compensation No. is changed.

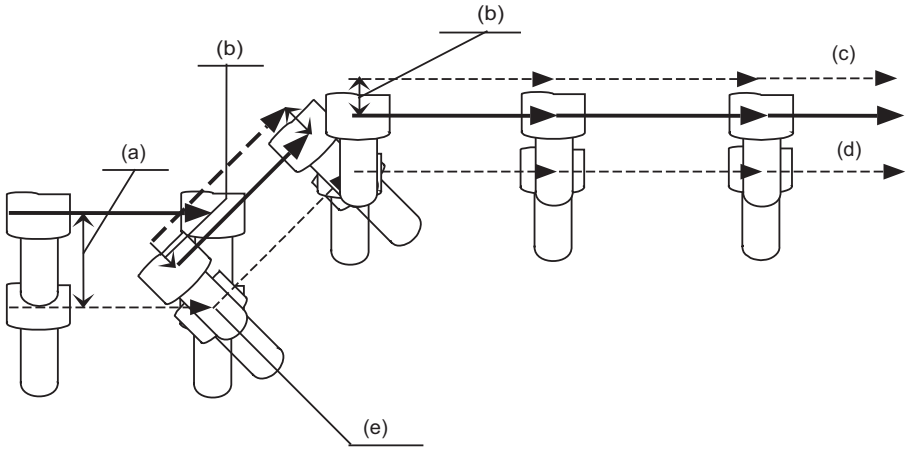
Note

- ♦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 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
(b) Changed compensation amount
(c) Path after compensation
(d) Program path
(e) Single block stop

- ♦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.

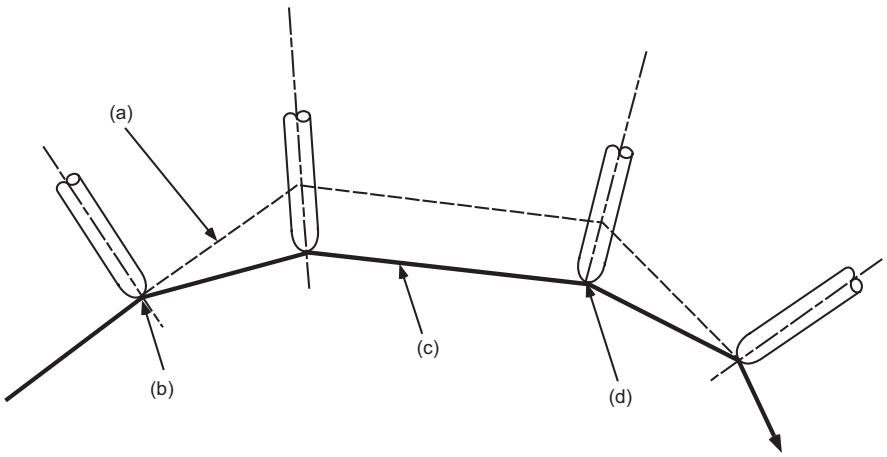
Tool length compensation in the tool axis direction vector

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:
Vx = L * sin(A) * sin(C)
Vy = -L * sin(A) * cos(C)
Vz = L * cos(A)

(2) When the B and C axes are set as the rotary axes:
Vx = L * sin(B) * cos(C)
Vy = L * sin(B) * sin(C)
Vz = L * cos(B)

Vx, Vy, Vz: 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. However, when the parameter "#8122 Keep G43 MDL M-REF" is set to "1" in the high-speed reference position return, it is not cleared.
(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.

Compensation number

[When the address H is commanded alone]

When the parameter "#1274 ext10/bit3" is "1", the H modal can be updated even if the address H of the tool compensation function is commanded alone.
Refer to "12.7.1 H Stand-alone Command" for details.

[Tool offset command check]

It is possible to make a program error occur when the tool offset No. is "0" or when the offset amount specified with the tool offset No. is "0", to prevent commands from being forgotten.
Refer to "12.7.2 Tool Offset Command Check" for details.



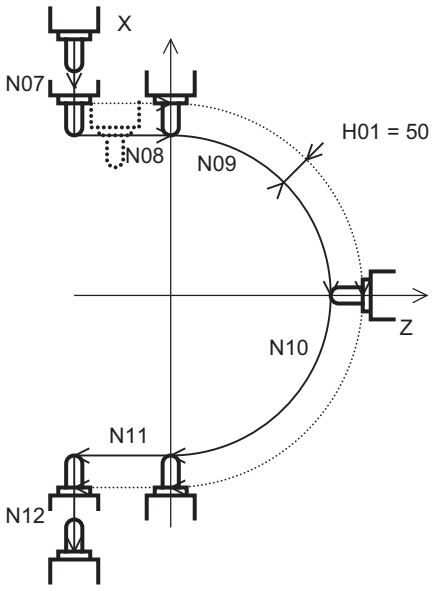
Program example

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 ;          Compensation amount H01 = 50 mm
N02 G28 B0 C0 ;
N03 G90 G54 G00 X400. Y0 ;
N04 Z-150. ;
N05 B90. ;                      B axis: 90°
N06 G18 ;
N07 G43.1 X250. H01 ;           Tool length compensation in the tool axis direction ON
N08 G01 Z0 F200 ;
N09 G02 X0 Z250. I-250. K0 B0 ;  Top right arc, B axis: 0°
N10 G02 X-250. Z0 I0 K-250. B-90. ;  Bottom right arc, B axis: -90°
N11 G01 Z-150. ;
N12 G00 G49  X-400. ;           Tool length compensation in the tool axis direction OFF
N13 G91 G28 B0 C0 ;
N14 G28 X0 Y0 Z0 ;
N15 M02 ;
```

[mm]



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.

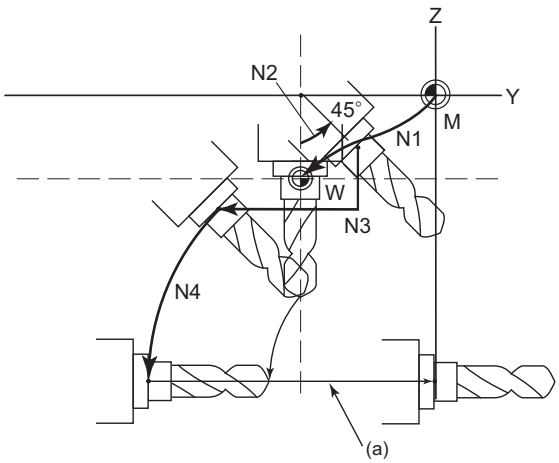
Relationship with automatic reference position return

(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.
However, for high-speed reference position return, the tool length compensation can be reactivated by the parameter "#8122 Keep G43 MDL M-REF" when the axis moves after the axis reached the reference position.

<Y axis Manual reference position return>

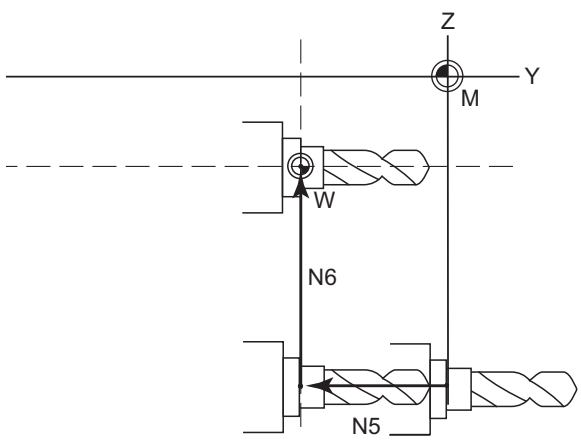


N1 G90 G00 G54 X0 Y0 Z0 ;
Positioning to the workpiece origin
N2 G00 A45. ;
Rotating the rotary axis by 45°
N3 G43.1 H1 ;
Tool length compensation along the tool axis ON
N4 G19 G03 Y-5.858 Z-14.142 J14.142 K-14.142 A90. ;
Circular cutting

Manual dog-type reference position return (a)

N5 G00 Y0. ;
N6 Z0. ;
:
:

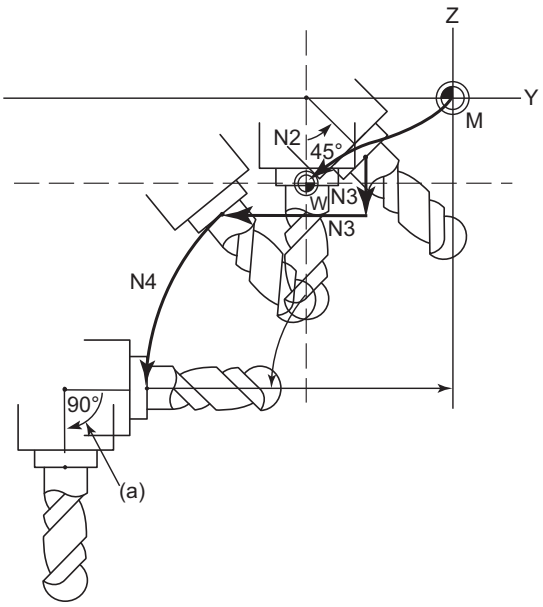
<Movement after Y axis Manual reference position return>



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>

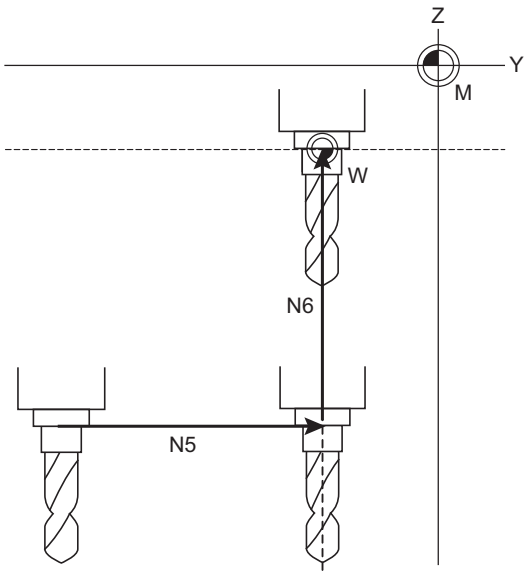


N1 G90 G00 G54 X0 Y0 Z0 ;
Positioning to the workpiece origin
N2 G00 A45. ;
Rotating the rotary axis by 45°
N3 G43.1 H1 ;
Tool length compensation along the tool axis ON
N4 G19 G03 Y-5.858 Z-14.142 J14.142 K-14.142 A90. ;
Circular cutting

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.

Precautions

(1) While the program format is switched, the tool length compensation along the tool axis cannot be performed by using the H axis (axis where the parameter "#1013 axname" is set to "H"). According to the setting of the parameter (#1450 5axis_Spec/bit0), the operation error (M01 0127) occurs when the power is turned ON, or the program error (P932 or P934) occurs when G43.1 is commanded. (This depends on the MTB specifications.)

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 machines as shown below.

(1) A tool tilt type: a machine with two rotary axes set on the head.

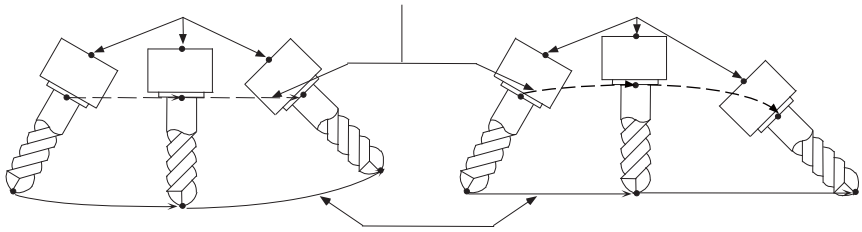
(2) A table tilt type: a machine with two rotary axes set on the table.

(3) A combined type: a machine with one rotary axis set on the tool and another on the table.

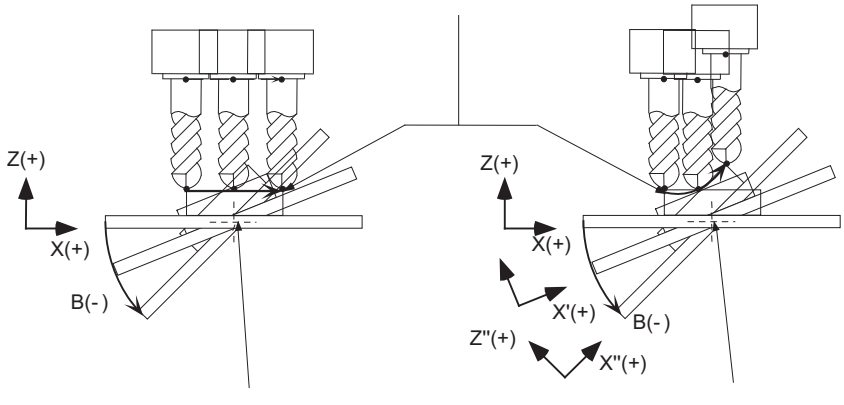
The applicable machine configurations depend on the MTB specifications.
For the applicable machine configurations, refer to "Appendix 1: Functions with Advanced Machining" of "Specifications Manual (Function)".

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).

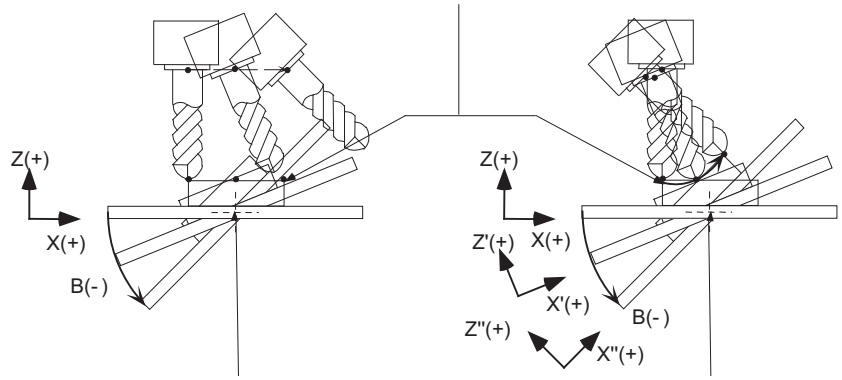
(1) Tool tilt type

Tool center point control OFF and tool length compensation along the tool axis ON	**Tool center point control ON**
Rotation center Program path Rotation center  Path of the tool center point	
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**
Path of the tool center point  Rotation center	Rotation center
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  Rotation center	Rotation center
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 alter.

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 moves in the commanded direction, and the tool center point moves following the movement of the rotary axis, which may cause a machining failure.

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 axes	2 rotary axes	3 orthogonal axes or less + 1 rotary axes	3 orthogonal axes or less + 2 rotary axes
○	○	×	○	×

Command format

There are two command formats: <tool center point control type 1> to command the tool angle with the rotary axis and <tool center point control type 2> to command the tool angle with the angle vector of the workpiece surface or the space angle on the workpiece coordinate system (feature coordinate system).
Type 2 is available only when the number of simultaneous contouring control axes is 5 or more.

Tool center point control type1 ON

G43.4 (X__ Y__ Z__ A__ C__) H__ P__ ;

X, Y, Z	Orthogonal coordinate axis movement command
A, C	Rotary axis movement command
H	Tool length compensation No.
P	G00 temporary cancel designation 0: Follows the setting of the parameter #8139. 1: Does not execute G00 temporary cancel regardless of the parameter setting. 2: Executes G00 temporary cancel regardless of the parameter setting.

Note

(1) When this command is not issued in the same block as the orthogonal coordinate axis movement command or rotary axis movement command, "start-up without movement command" is applied.
If this command is issued in the same block as a movement command, "start-up with movement" is applied.

(2) Commands to I, J, and K will be ignored during the tool center point control type 1.

(3) If address P is omitted, follow the setting of the parameter "#8139 TCP G0 temp.cancel".
If any value other than 0 to 2 is commanded, the program error (P35) will occur.

(4) If the numeric value following each address is omitted, a different operation is performed depending on the setting of the parameter "#1227 aux11/bit4" (Word command check).

When "#1227 aux11/bit4" is set to "0", the omitted numeric value is assumed to be "0".

When "#1227 aux11/bit4" is set to "1", the program error (P33) occurs.

Tool center point control type2 ON

G43.5 (X__ Y__ Z__) I__ J__ K__ H__ (P__) ;	**Designation by angle vector**

G43.5 ,P0 (X__ Y__ Z__) I__ J__ (K__) H__ (P__) ;	**Designation by Euler angles**

G43.5 ,P1 (,Q__) (X__ Y__ Z__) I__ J__ K__ H__ (P__) ;	**Designation by roll-pitch-yaw angles**

X, Y, Z	Orthogonal coordinate axis movement command
I, J, K	♦Angle vector of workpiece surface (Designation by angle vector) I: Component in X axis direction J: Component in Y axis direction K: Component in Z axis direction ♦Euler angles (Designation by Euler angles) (*1) ♦Roll-pitch-yaw angles (Designation by roll-pitch-yaw angles) (*1) (*1) The commanded range of the Euler angles and roll-pitch-yaw angles is -360° to 360°.
H	Tool length compensation No.

P	G00 temporary cancel designation 0: Follows the setting of the parameter #8139. 1: Does not execute G00 temporary cancel regardless of the parameter setting. 2: Executes G00 temporary cancel regardless of the parameter setting.
,Q	Rotation order of roll-pitch-yaw angles (q: Setting value for address ",Q") q 1st 2nd 3rd 123 X Y Z 132 X Z Y 213 Y X Z 231 Y Z X 312 Z X Y 321 Z Y X If address ",Q" is omitted, "q" is handled as "123".

Note

(1) When this command is not issued in the same block as the orthogonal coordinate axis movement command or rotary axis movement command, "start-up without movement command" is applied.
If this command is issued in the same block as a movement command, "start-up with movement" is applied.

(2) Rotary axis movement command cannot be issued during the tool center point control type 2. If commanded, a program error (P33) occurs.

(3) Address ",Q" can be commanded only for "designation by roll-pitch-yaw angles". Address ",Q" cannot be commanded for other designations. If it is commanded, a program error (P33) occurs. Even when address ",Q" is not the numeric value specified above, a program error (P33) occurs.

(4) If I, J, or K is omitted when the tool center point control type 2 is commanded, the omitted address is assumed to "0". However, if all the addresses are omitted in "designation by Euler angles" and "designation by roll-pitch-yaw angles", the current angle is maintained.

(5) For "designation by Euler angles" and "designation by roll-pitch-yaw angles", if I, J, or K is commanded outside the range of -360° to 360, a program error (P35) occurs.

(6) If address P is omitted, follow the setting of the parameter "#8139 TCP G0 temp.cancel".
If any value other than 0 to 2 is commanded, a program error (P35) occurs.

(7) If the numeric value following each address is omitted, a different operation is performed depending on the setting of the parameter "#1227 aux11/bit4" (Word command check).

When "#1227 aux11/bit4" is set to "0", the omitted numeric value is assumed to be "0".

When "#1227 aux11/bit4" is set to "1", the program error (P33) occurs.

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) When this command is not issued in the same block as the orthogonal coordinate axis movement command or rotary axis movement command, "cancel without movement command" is applied.
If this command is issued in the same block as a movement command, "cancel with movement" is applied.

(3) If the numeric value following each address is omitted, a different operation is performed depending on the setting of the parameter "#1227 aux11/bit4" (Word command check).

When "#1227 aux11/bit4" is set to "0", the omitted numeric value is assumed to be "0".

When "#1227 aux11/bit4" is set to "1", the program error (P33) occurs.

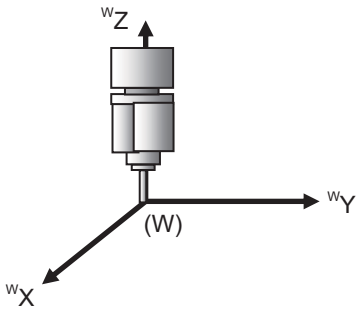
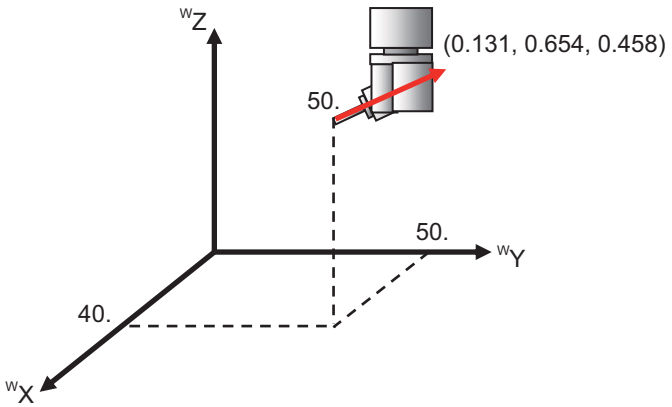
Angle control operation example (Type 2)

[Tool center point control type 2 (Designation by angle vector)]

The angle is determined so that the tool axis direction on the workpiece coordinate system (feature coordinate system) is set to (i, j, k) by the G43.5 command (designation by angle vector).

(Example)

G43.5 X40. Y50. Z50. I0.131 J0.654 K0.458 ;

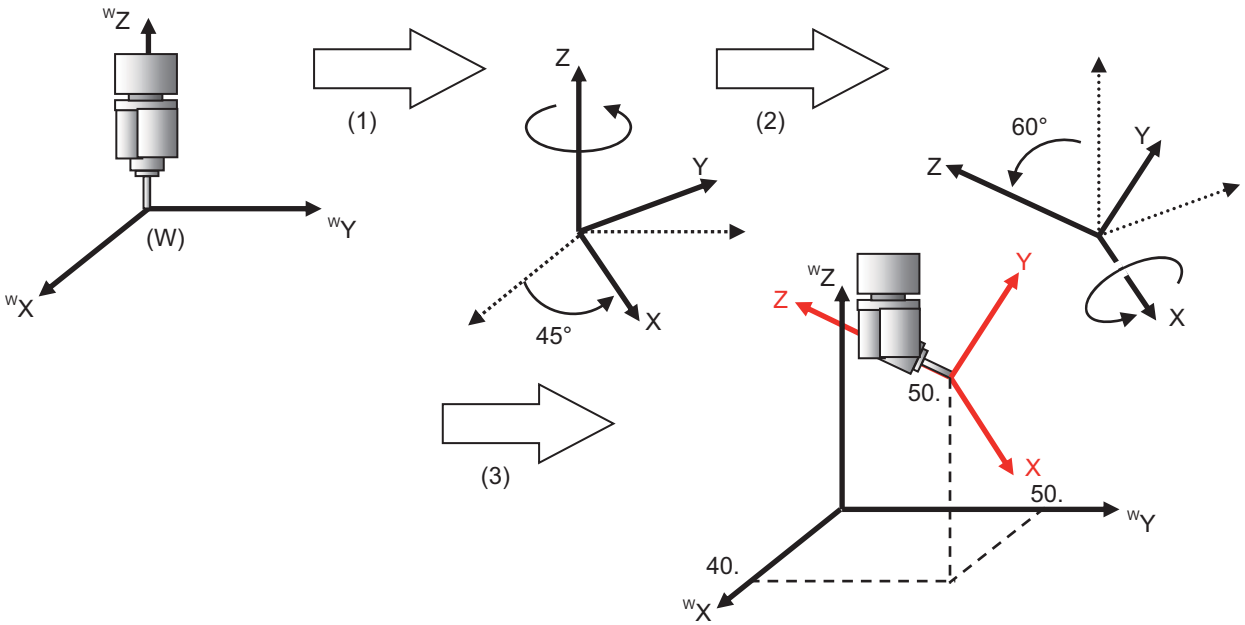
	
Current tool angle	Tool angle after G43.5 command
(W) Workpiece coordinate system (Feature coordinate system)	Component in X axis direction (0.131) Component in Y axis direction (0.654) Component in Z axis direction (0.458)

[Tool center point control type 2 (Designation by Euler angles)]

The angle is determined as shown below by the "G43.5 ,P0" command (designation by Euler angles).

(Example)

G43.5 ,P0 X40. Y50. Z50. I45. J60 ;



(1) Rotate the workpiece coordinate system (feature coordinate system) around the Z axis by 45 degrees.

(2) Rotate the rotated coordinate system around the X axis by 60 degrees.

(3) Control the angle so that the Z axis direction of the coordinate system designated above is handled as the tool axis direction.

Note

(1) The last K command (which rotates the angle by k degrees around a new Z axis) has no meaning, so it need not be issued. This command is ignored even if it is issued.

(2) Looking at the center of rotation from the positive direction of each rotation center axis, the counterclockwise direction is the positive direction of rotation.

(3) The orthogonal coordinate axis (x, y, z) is moved on the currently selected workpiece coordinate system (feature coordinate system). The workpiece coordinate system (feature coordinate system) is not updated by this command.

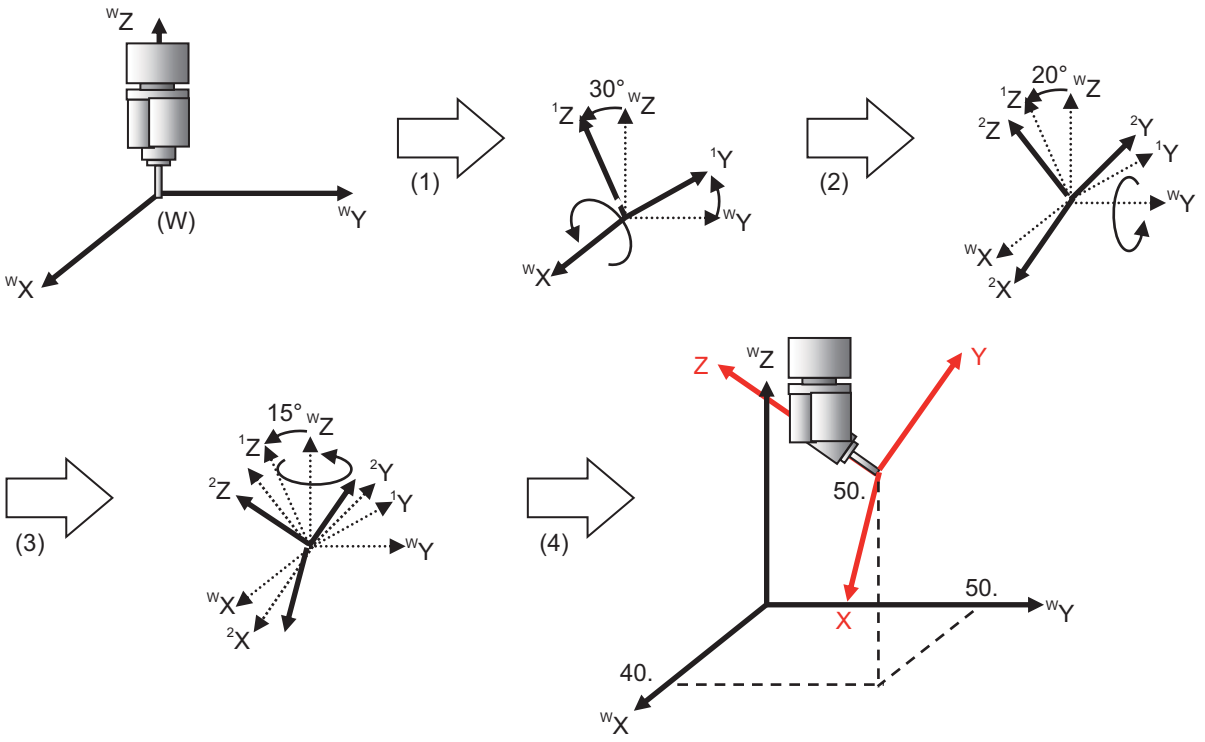
[Tool center point control type 2 (Designation by roll-pitch-yaw angles)]

The angle is determined as shown below by the "G43.5 ,P1" command (designation by roll-pitch-yaw angles).

(Example)

G43.5 ,P1 ,Q123 X40. Y50. Z50. I30. J20. K15. ;

(When q=123 [rotations in the order of WX, WY and WZ])



(1) Rotate the workpiece coordinate system (feature coordinate system) around the X axis by 30 degrees. (Roll angle)

(2) Rotate the unrotated coordinate system around the Y axis by 20 degrees. (Pitch angle)

(3) Rotate the unrotated coordinate system around the Z axis by 15 degrees. (Yaw angle)

(4) Control the angle so that the Z axis direction of the coordinate system designated above is handled as the tool axis direction.

Note

(1) Looking at the center of rotation from the positive direction of each rotation center axis, the counterclockwise direction is the positive direction of rotation.

(2) The orthogonal coordinate axis (x, y, z) is moved on the currently selected workpiece coordinate system (feature coordinate system). The workpiece coordinate system (feature coordinate system) is not updated by this command.



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

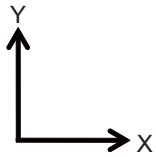
When "0" is set to the programming coordinate system selection parameter, the table coordinate system, which is the valid workpiece coordinate system at that time fixed to the table, is specified as the programming coordinate system. Table coordinate system rotates along the table rotation. And it does not rotate along the tool axis rotation. The coordinate values of X,Y,Z 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.

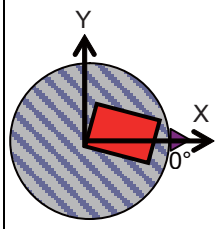
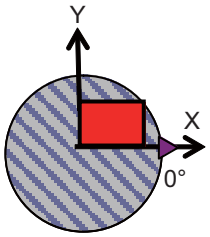
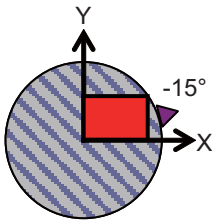
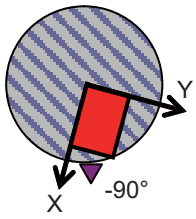
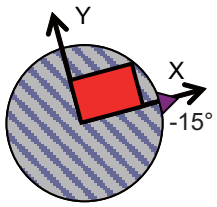
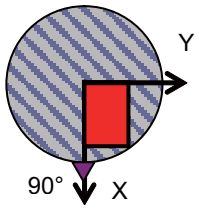
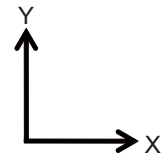
(2) Workpiece coordinate system

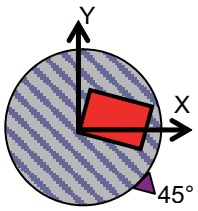
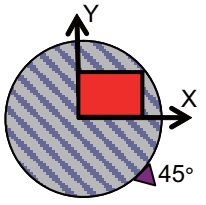
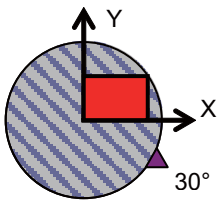
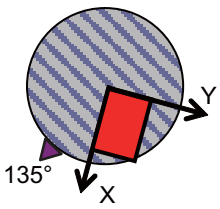
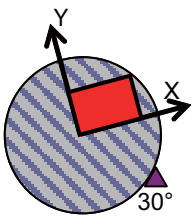
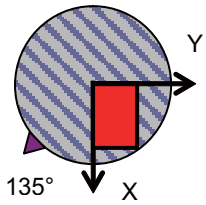
When "1" is set to the programming coordinate system selection parameter, the valid workpiece coordinate system at that time is specified as the programming coordinate system. The coordinate system in this case does not rotate along the table rotation. A linear movement is carried out for the table (workpiece) 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").

<table>
<tr><th>Rotary axis reference position selection</th><th colspan="2">Start position basis (#7911=1)</th><th colspan="2">Zero degree position basis (#7911=0)</th></tr>
<tr><td>The timing to be fixed to the table</td><td colspan="2">The position of rotary axis holding the workpiece at the start of tool center point control</td><td colspan="2">When the position of rotary axis holding the workpiece is at 0 degree on the workpiece coordinate system</td></tr>
<tr><td rowspan="2">Operation example 1
(Workpiece coordinate offset 0°)


Machine coordinate system</td><td>:
G90G54G0C0
C-15. ;
G43.4 Hh ;
:
C90. ;
:</td><td>Workpiece coordinate 0°

</td><td>:
G90G54C0
C-15. ;
G43.4 Hh ;
:
C90. ;
:</td><td>Workpiece coordinate 0°

</td></tr>
<tr><td>Machine coordinate position fixed at -15°

</td><td>Machine coordinate position 90° by C90. command

</td><td>Machine coordinate position -15° by C-15. command

</td><td>Machine coordinate position 90° by C90. command

</td></tr>
<tr><td rowspan="2">Operation example 2
(Workpiece coordinate offset 45°)


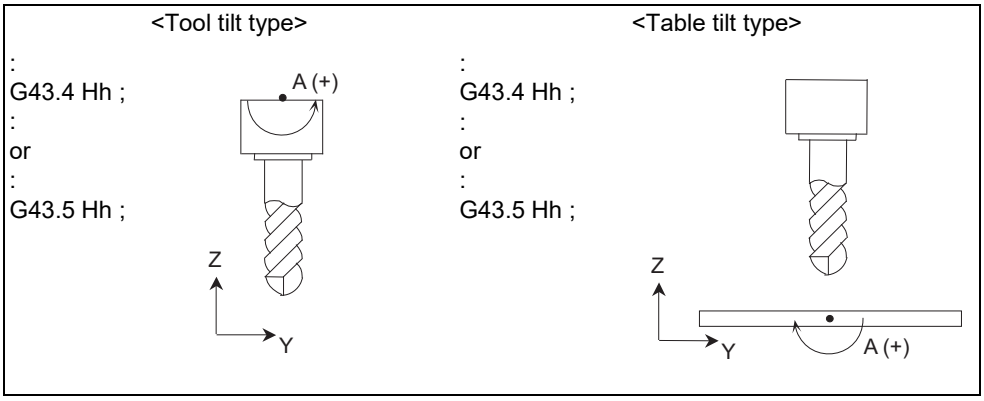
Machine coordinate system</td><td>:
G90G54G0C0
C-15. ;
G43.4 Hh ;
:
C90. ;
:</td><td>Workpiece coordinate 0°

</td><td>:
G90G54C0
C-15. ;
G43.4 Hh ;
:
C90. ;
:</td><td>Workpiece coordinate 0°

</td></tr>
<tr><td>Machine coordinate position fixed at 30°

</td><td>Machine coordinate position 135° by C90. command

</td><td>Machine coordinate position 30° by C-15. command

</td><td>Machine coordinate position 135° by C90. command

</td></tr>
</table>

Start-up

(1) Start-up without movement

(a) Tool center point control type 1, Tool center point control type 2 (without IJK command)

When the tool center point control is ON, no axis movement is performed (including movement for the compensation amount).



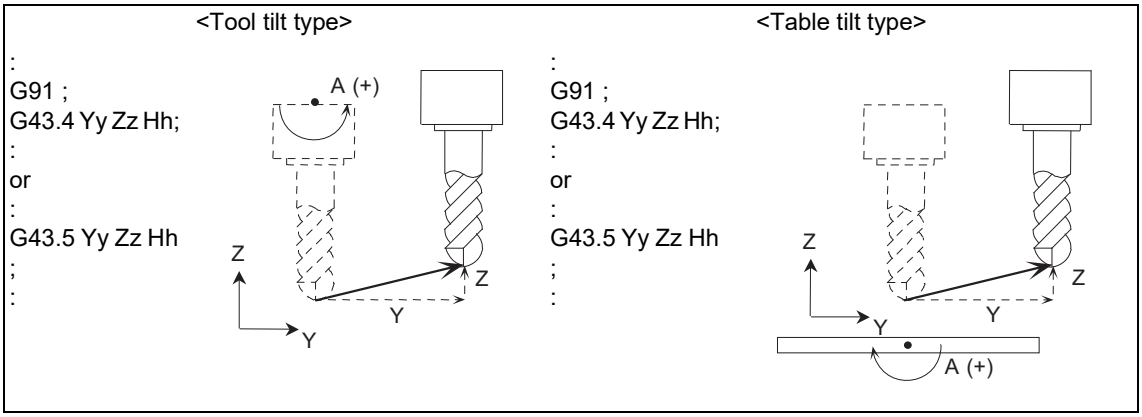
(b) Tool center point control type 2 (with I/J/K command)

"G43.5 Ii Jj Kk Hh ;" performs the same movement as the tool center point control type 1 in (2).

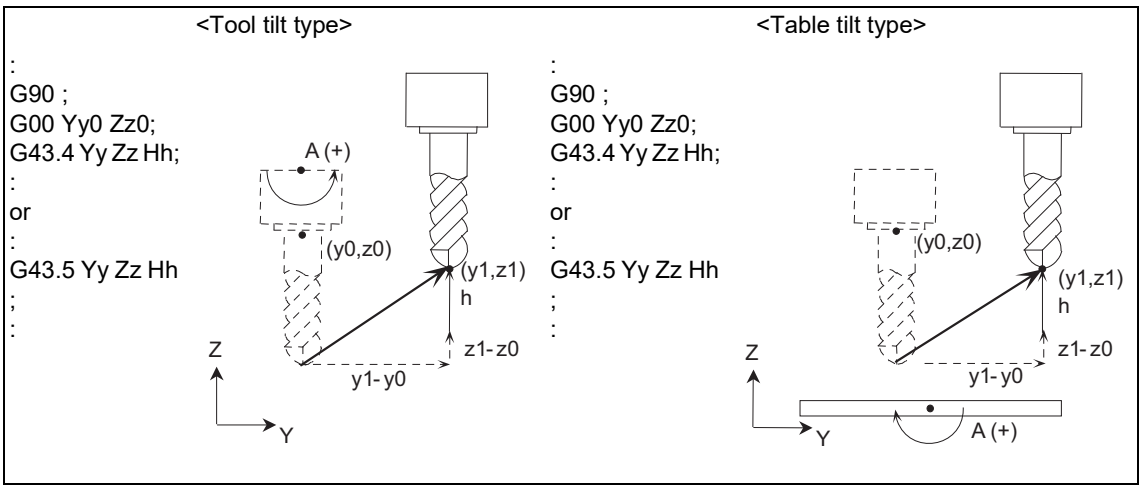
(2) Start-up with movement command (When orthogonal coordinate axis command is issued in the same block)

(a) Tool center point control type 1, Tool center point control type 2 (without IJK command)

When the incremental command (G91) is designated while the tool center point control is ON, the tool center point moves only as much as it is ordered with the movement command.



Under the absolute command (G90), the tool center point moves to y1, z1.

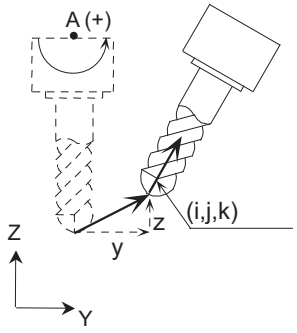


(b) Tool center point control type 2 (with I/J/K command)

The rotary axis moves in accordance with the I, J, or K command issued with the movement command. (With the incremental command (G91) designated)

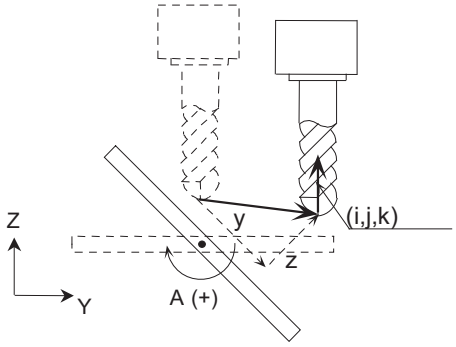
<Tool tilt type>

```
:
G91 ;
G43.5 Yy Zz
Ii Jj Kk Hh;
:
```



<Table tilt type>

```
:
G91 ;
G43.5 Yy Zz
Ii Jj Kk Hh;
:
```



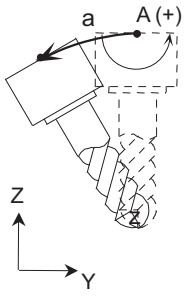
(3) Start-up with movement command (When the rotary axis command is issued in the same block)

(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.

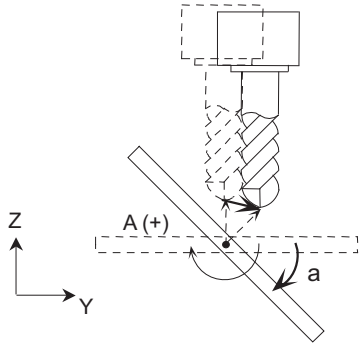
<Tool tilt type>

```
:
G43.4 Aa
Hh;
:
```



<Table tilt type>

```
:
G43.4 Aa
Hh;
:
```



(b) Tool center point control type 2

The program error (P33) occurs.

Cancel

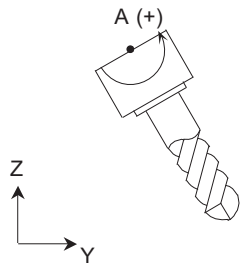
(1) Cancel without movement command

Movement of tool by compensation amount due to the cancel is not performed regardless of absolute/incremental command.

On the other hand, the tool center point control modal is canceled.

<Tool tilt type>

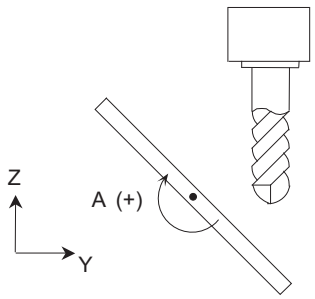
:
G49;
:



The tool will not move.

<Table tilt type>

:
G49;
:



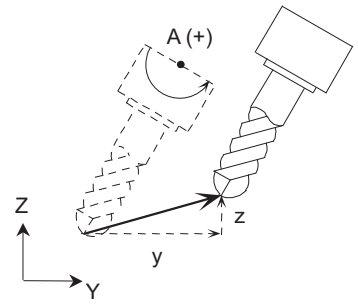
(2) Cancel with movement command (When the orthogonal coordinate axis command is issued in the same block)

Movement of tool by compensation amount due to the cancel is not performed regardless of absolute/incremental command.

Orthogonal coordinate axis movement command is executed upon the cancel of the tool center point control modal.

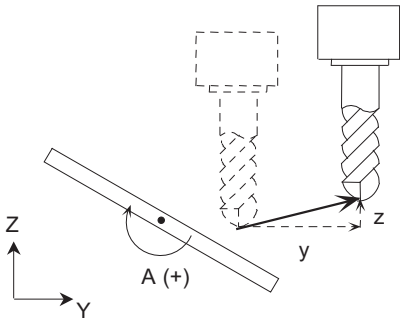
<Tool tilt type>

:
G91;
G49 Yy Zz ;
:



<Table tilt type>

:
G91;
G49 Yy Zz ;
:



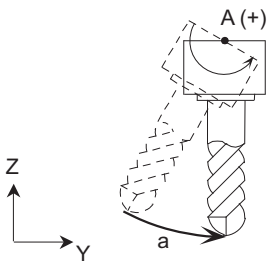
(3) Cancel with movement command (When the rotary axis command is issued in the same block)

Movement of tool by compensation amount due to the cancel is not performed regardless of absolute/incremental command.

Rotary axis movement command is executed upon the cancel of the tool center point control modal.

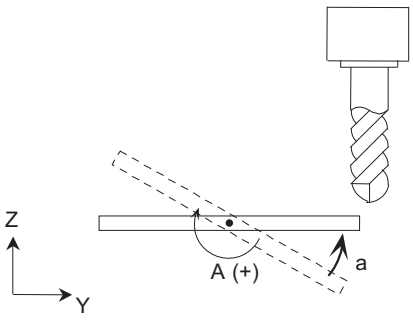
<Tool tilt type>

:
G49 Aa Hh;
:



<Table tilt type>

:
G49 Aa Hh;
:



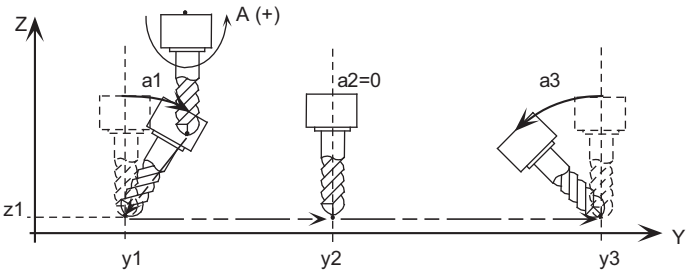
During tool center point control

(1) Tool center point control type 1

(a) When executing the travel command for the orthogonal coordinate axis and rotary axis.

```
:
G90 ;
G43.4 Yy1 Zz1 Aa1 Hh ;
Yy2 Aa2 ;
Yy3 Aa3 ;
:
```

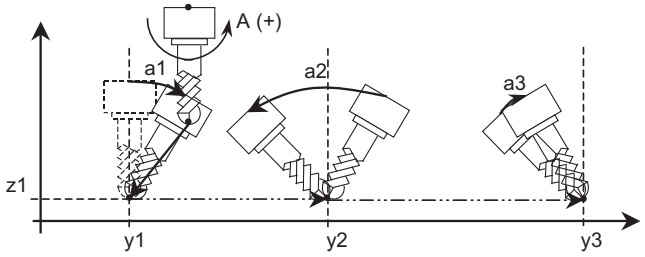
The tool center point moves along the programmed path.



(b) When a travel command is given to a 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.

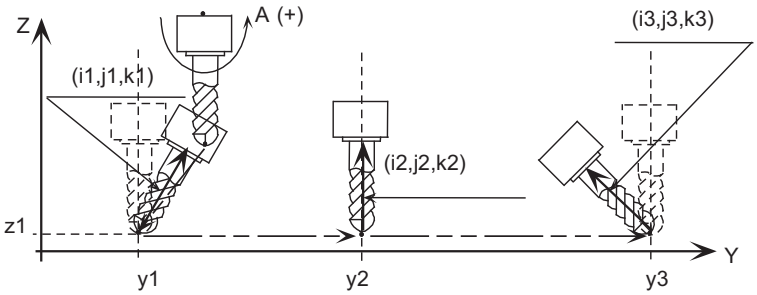


(2) Tool center point control type 2

(a) When executing the movement command to the orthogonal coordinate axis and the I, J, K command.

```
:
G43.5 Yy1 Zz1
Ii1 Jj1 Kk1 Hh ;
Yy2 Ii2 Jj2 Kk2 ;
Yy3 Ii3 Jj3 Kk3 ;
:
```

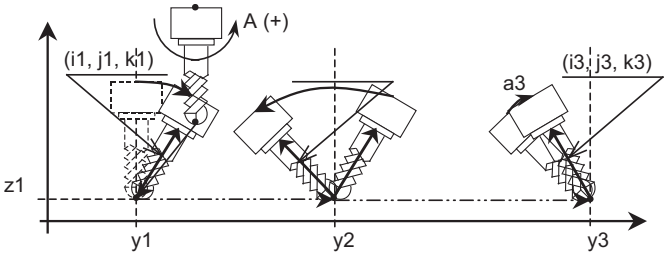
Tool center point moves along the program path.



(b) When executing I, J, K command only.

```
:
G43.5 Yy1 Zz1
Ii1 Jj1 Kk1 Hh ;
Yy2 ;
Ii2 Jj2 Kk2 ;
Yy3 Ii3 Jj3 Kk3 ;
:
```

When executing the I, J, K command only, the orthogonal axis moves without moving the tool center point.



Feedrate during tool center point control

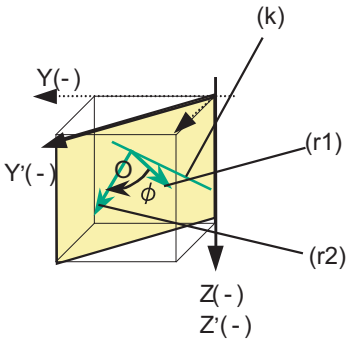
Feedrate during the tool center point control is controlled so that the tool center point moves according to the commanded speed.

Interpolation method

There are two methods of interpolation: single axis rotation interpolation and joint interpolation. Which method to be selected depends on the MTB specifications (parameter "#7910 SLCT_INT_MODE").

(1) Single axis rotation interpolation (#7910 = 1)

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

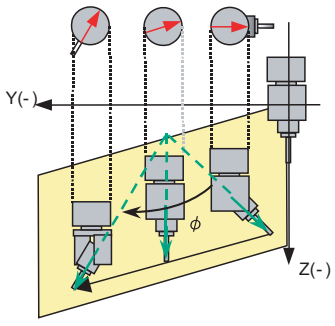
(a) Features
- Tool angle vector always exists on the plane consisting of "O", "r1" and "r2".
- The angular rates of each rotary axis will not be constant.

(b) Operation

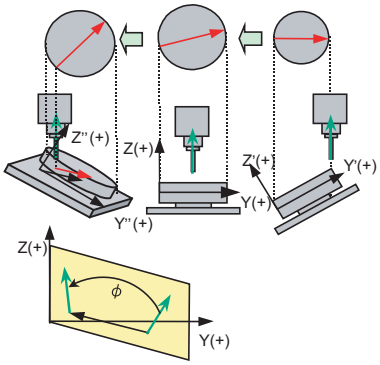
(Example) Current position: Aa°, C0°

When commanding "G90 Yy A-a. C45. ;" or "G90 Yy Ii Jj Kk ;"

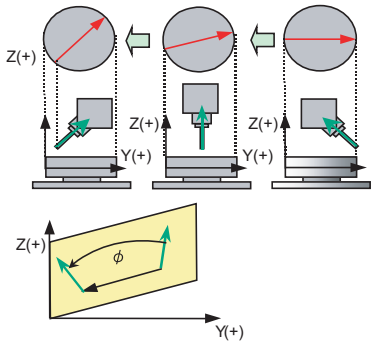
<Tool tilt type>



<Table tilt type>



<Combined type>

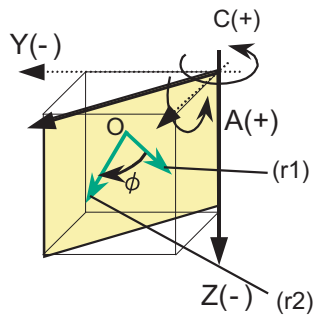


(2) Joint interpolation (#7910 = 0)

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

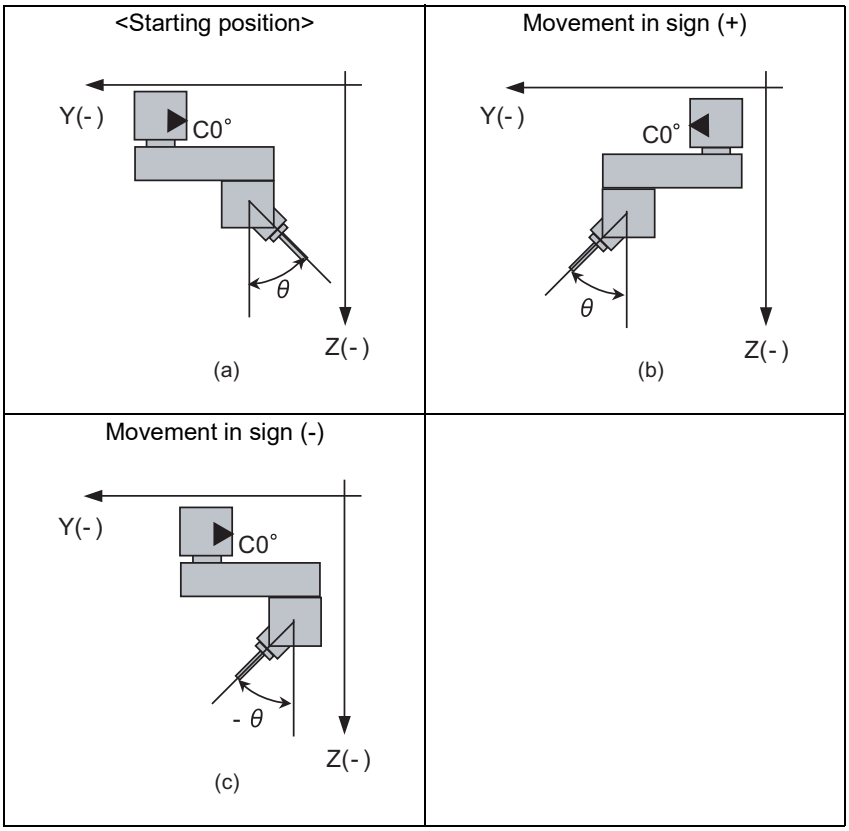
- The angular rates of each rotary axis become constant.
- As this control aims to keep the angular rates of each rotary axis constant, a tool angle vector may not exist on the plane consisting of "O", "r1" and "r2".



(r1) Start-point command vector "r1"
(r2) End-point command vector "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.
Which movement to be selected from the two depends on the MTB specifications (parameter "#7906 PASSTYP").
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".

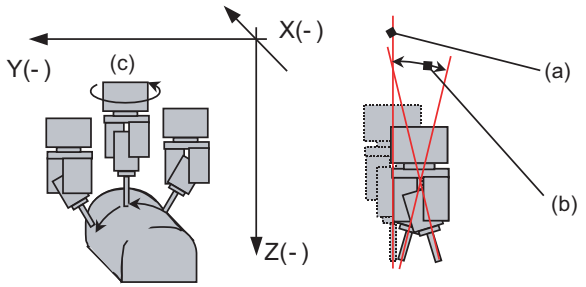


(*1) The position in which the tool-center-point-side rotary axis or the table-base-side rotary axis is 0.

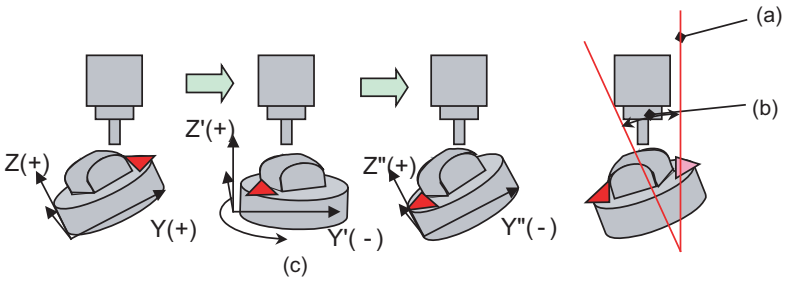
(1) Passing singular point type 1 (#7906 = 0)

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.

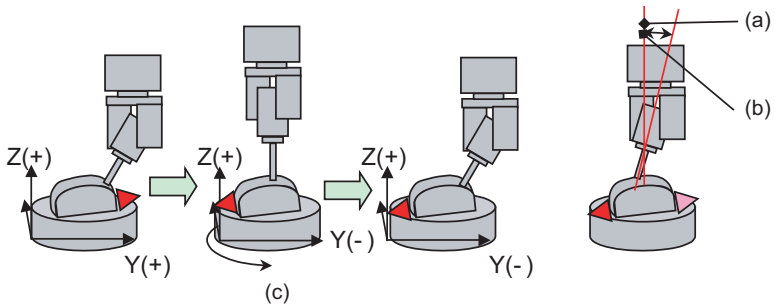
<Tool tilt type>



<Table tilt type>



<Combined type>



(a) Singular point

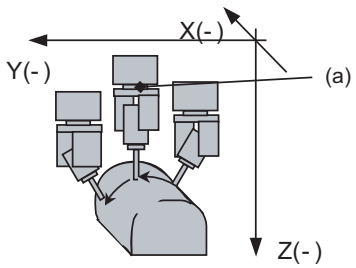
(b) When passing near the singular point, C axis rotates 180° within the parameter "#7907 CHK_ANG" (Near the singular judgment angle).

(c) C axis rotates 180°

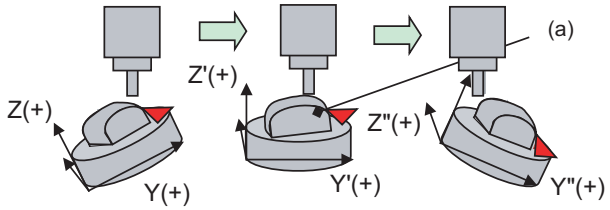
(2) Passing singular point type 2 (#7906 = 1)

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.

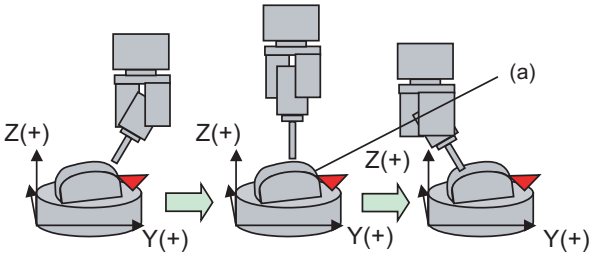
<Tool tilt type>



<Table tilt type>



<Combined type>



(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.	
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. (This depends on the MTB specifications.)

(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.

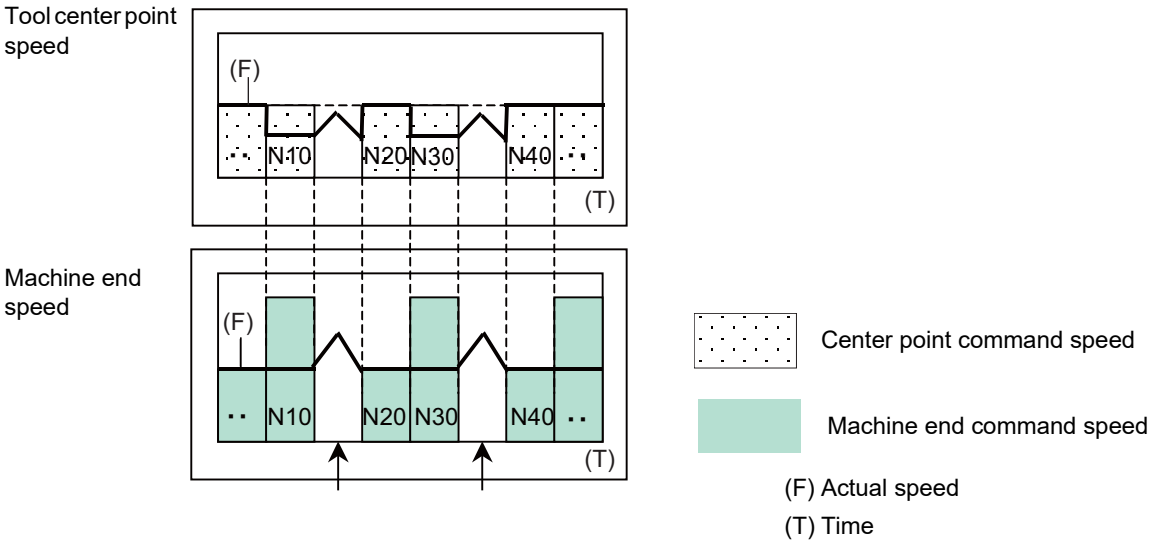
<Machining program example>

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;
 :
```

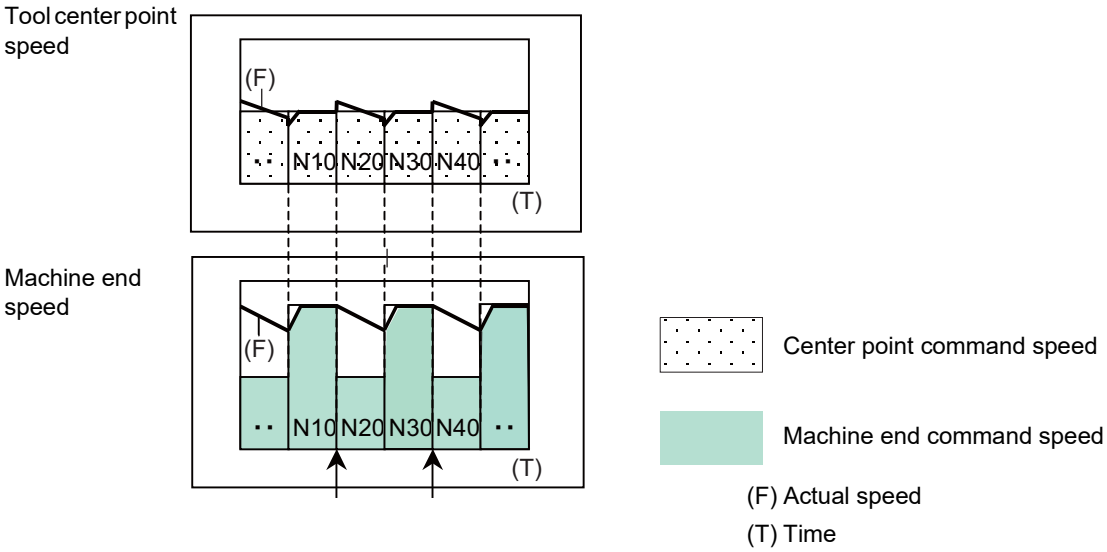
<Note>
- ♦Center point block lengths are even.
- ♦A machine end block length is longer when it has a rotary axis movement command. (In this case, the machine end speed is faster in a block with rotary axis movement than in a block without rotary axis movement.)
- ♦When SSS control is enabled, a machine speed fluctuation suppression is disabled.

(1) Speed when "#7913 = 0"



Awaited to decelerate down to the machine end speed of the next block. Thus, the speed changes sharply.

(2) Speed when "#7913 = 1"



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.

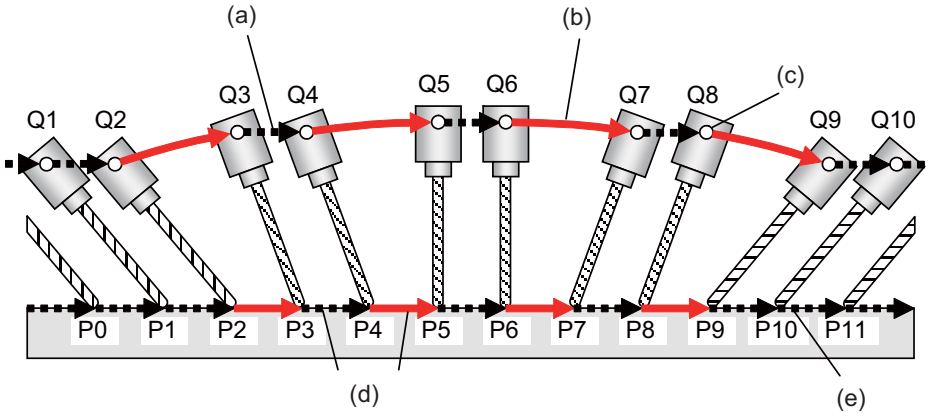
<Note>

- 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 parameter "#7914 ROT_PREFILT" (Rotary axis prefilter time constant). When "0" is set, this function is invalid.

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

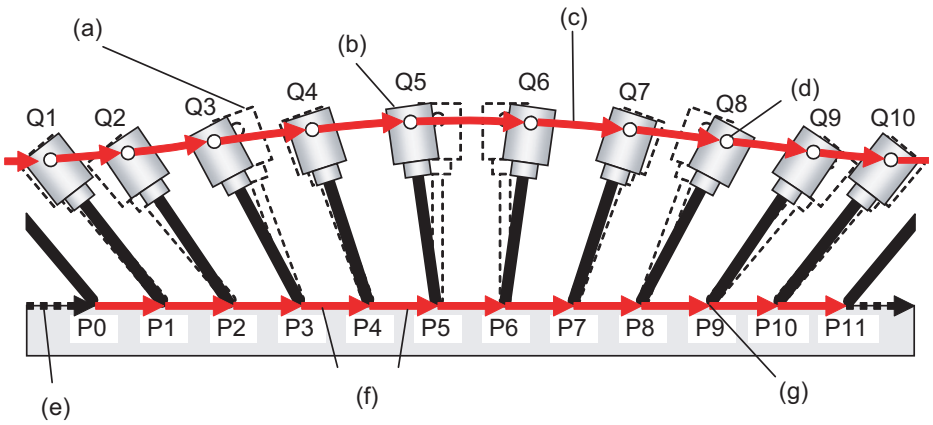
(b) With tool angle shift

(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 position

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 position

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.

Circular command in tool center point control (G43.4/G43.5)

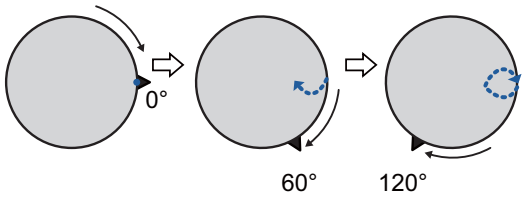
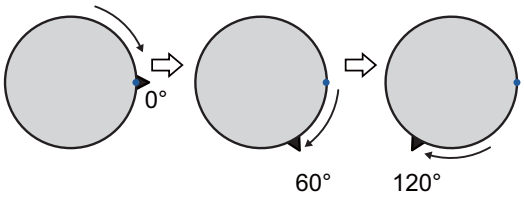
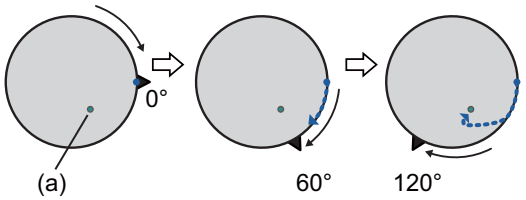
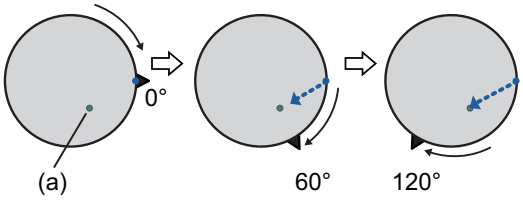
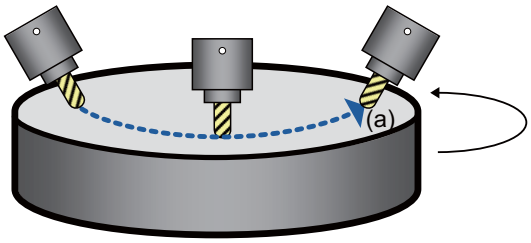
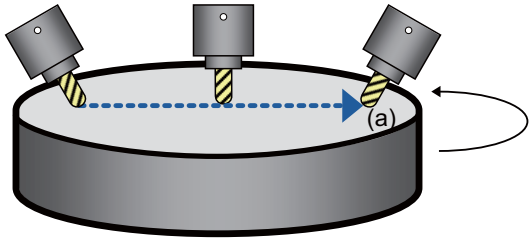
For details, refer to "19.3.1 Circular Command in Tool Center Point Control (G43.4/G43.5)".

G0 temporary cancel during center point control (G43.4/G43.5)

When the programming coordinate system is based on the workpiece coordinate system setting ("#7908 SLCT_PRG_COORD" is set to "1"), the movement of the orthogonal axis synchronous with the table rotation can be canceled temporarily by the G0 command. Therefore, when the system repeats the machining and the next machining position index (table rotation) while rotating the workpiece, no retract operation is required because the tool center point remains unmoved.

(The parameter "#7908 SLCT_PRG_COORD" setting depends on the MTB specifications.)

The following figure shows the movement of the tool center point on the machine coordinate system.

Axis command	When temporary cancel is invalid	When temporary cancel is valid
Table-side rotary axis	 The movement of tool center point occurs.	 The movement of tool center point does not occur.
Orthogonal axis + Table-side rotary axis	 The tool center point does not linearly move to the commanded position.	 The tool center point linearly moves to the commanded position.
Orthogonal axis + Tool-side rotary axis + Table-side rotary axis	 The tool center point does not linearly move to the commanded position.	 The tool center point linearly moves to the commanded position.

Movement of the tool center point on the machine coordinate system

(a) Commanded position

[Function enabling conditions]

This function becomes valid when all of the following conditions are satisfied.

(1) When the tool center point control starts up, one of the following events occurs.
- The "P0" command is issued or the P command is omitted and the parameter "#8139 TCP G0 temp.cancel" is set to "1" (ON).
- "P2" was commanded.

(2) The programming coordinate system is set to the workpiece coordinate system setting ("#7908 SLCT_PRG_CORD" is set to "1").

(3) The interpolation method is set to the joint interpolation system setting ("#7910 SLCT_INT_MODE" is set to "0").

(4) The movement command is set to G00.

(5) The following commands are available.
- During tool center point control type 1 (G43.4) mode: Workpiece-attached rotary axis command
- During tool center point control type 2 (G43.5): I, J, K command

[Precautions]

(1) G0 temporary cancel is available only for a table tilt type or combined type machine. For other machine types, the P command and the parameter #8139 are ignored.

(2) When combining this function with the 3-dimensional tool radius compensation, issue the G0 temporary cancel invalid command (*1) to move to the workpiece surface (cutting position). If G0 temporary cancel is valid, the movement direction differs from that designated in the G0 temporary cancel invalid command. This causes a difference to occur in the tool radius compensation direction, which may lead to an unexpected machine shape.

(*1) This command refers to a movement command other than G00 or a command that does not move the table-side rotary axis.

(3) In the table tilt configuration, the workpiece-attached rotary axis is targeted for G0 temporary cancel. However, if the table base-side rotary axis and workpiece-attached rotary axis are commanded to the same block, both of them are targeted for G0 temporary cancel.

(4) When "G43.4 P_" or "G43.5 P_" is commanded during tool center point control, operation is performed with the last commanded P value.
However, if the P command is omitted, operation is performed with the previously commanded P value. (Refer to the following examples.)

N01 G43.4 P2;	Operated as P2 (G0 temporary cancel valid).
N02 G00 C30.;	
N03 G43.4 P1;	Operated as P1 (G0 temporary cancel invalid).
N04 G00 C60.;	
N05 G43.4;	
N06 G00 C90.;	

(5) The next distance counter is not updated in a block with G0 temporary cancel used.

Compensation number

[When the address H is commanded alone]

When the parameter "#1274 ext10/bit3" is "1", the H modal can be updated even if the address H of the tool compensation function is commanded alone.
Refer to "12.7.1 H Stand-alone Command" for details.

[Tool offset command check]

It is possible to make a program error occur when the tool offset No. is "0" or when the offset amount specified with the tool offset No. is "0", to prevent commands from being forgotten.
Refer to "12.7.2 Tool Offset Command Check" for details.

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

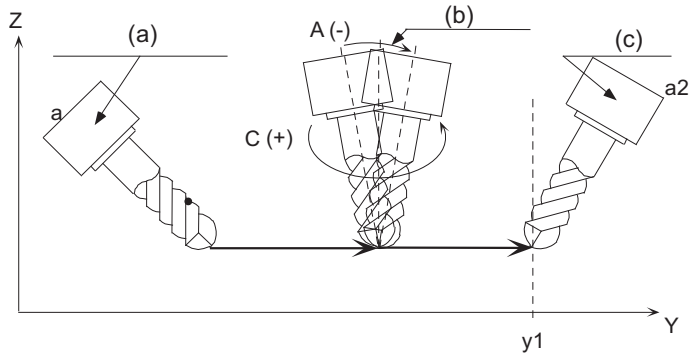
Buffer correction cannot be performed during tool center point control.

(3) Miscellaneous function (MSTB)

Miscellaneous function (MSTB) command can be executed during tool center point control.
(When the tool passes through the singular point, the "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

(a) A macro interruption command can be issued during the tool center point control. However, if a macro interruption is performed in the block in which the tool passes a singular point, the operation may change depending on the setting value of the parameter "#1113 INT_2". Refer to "Interrupt type" in "14.7 Macro Interruption; M96, M97".

(b) When a macro interruption is executed during the tool center point control, disable the reverse run control mode (RVMD). If a macro interruption is executed while the tool center point control is activated and the reverse run control mode (RVMD) is enabled, the operation error (M01 0170) occurs. This error can be cleared by resetting.

(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 resultant 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 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 rate with the rapid traverse (G00) in high-accuracy control mode depends on the MTB specifications (parameter "#1250 set22/bit3").

0	The acceleration rate 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 rate 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.

(39) Coordinate system setting (G92)

To execute the tool tip center control command, G92 shift amount needs to be set to "0" for all the axes in the part system in advance.
When the tool tip center control is commanded, if G92 shift amount is set to an axis other than the ones for which the tool tip center control are commanded, the program error (P942) occurs.
While the tool tip center control is being executed, the coordinate system set with the coordinate setting (G92) are temporarily canceled in all the axes in the part system of the tool tip center control.
When the tool tip center control command is canceled or NC reset is performed, the G92 shift amount which was set before the temporal cancellation is set again.
When the tool tip center control command is executed with the G92 shift amount being set, perform a positioning with the absolute position command at startup and after cancellation.
When a positioning is not performed, the tool may operate in the wrong position.

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). (This depends on the MTB specifications.)
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	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		Max. feedrate is 100 m/min when 1 mm segment G1 block is commanded with 5 axes simultaneously	Max. feedrate is 100 m/min when 1 mm segment G1 block is commanded with 5 axes simultaneously	P33
	P10000	High-speed high-accuracy control II	Max. feedrate is 100 m/min when 1 mm segment G1 block is commanded with 5 axes simultaneously	Max. feedrate is 100 m/min when 1 mm segment G1 block is commanded with 5 axes simultaneously	P33
G05.1		High-speed high-accuracy control I	Max. feedrate is 33.7 m/min when 1 mm segment G1 block is commanded with 5 axes simultaneously	Max. feedrate is 33.7 m/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	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.	P33

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 - 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 - G30.6	Tool exchange position return 1 to 6	P942	-	P941
G31	Skip	P942	-	P941
G31.1 - G31.3	Multi-step skip	P942	-	P941
G33	Thread cutting	P942	P941	P941
G34 - 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 by G code	P942	P941	P941
G50.2/G250	Tool spindle synchronization IB/ IC cancel	P942	Tool center point control is also available while the function at the left is active.	P941
G51.2/G251	Tool spindle synchronization IB/ IC	P942	Tool center point control is also available while the function at the left is active. (*2)	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 - 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

Format	Function	A	B	C
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 - 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.
-	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
G68IiJjKk/ G69	3-dimensional coordinate conversion	P922	P941	P923
G70 - 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 command	The modal is switched to the specified absolute/incremental 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 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.
G113.1	Spindle synchronization I, tool spindle synchronization (IA/IB/II) cancel	P942	Tool center point control is also available while the function at the left is active.	P941

Format	Function	A	B	C
G114.1	Spindle synchronization	P942	Tool center point control is also available while the function at the left is active. (*1)	P941
G114.2	Tool spindle synchronization IA	P942	Tool center point control is also available while the function at the left is active.	P941
G114.3	Tool Spindle Synchronization II	P942	Tool center point control is also available while the function at the left is active.	P941

(*1) If the axis set to the spindle/C axis is used as a rotary axis for tool center point control, the following program errors occurs.

- If spindle synchronization is commanded for the spindle in the C axis mode, the operation error (M01 1026) occurs.
- If tool center point control is commanded in the spindle mode, the program error (P934) occurs.

(*2) Do not designate the rotary tool axis in the rotary axis configuration parameter. If tool center point control command is commanded during tool spindle synchronization IC, it may cause an unexpected operation.



Precautions

(1) If axis movement is commanded for any axis not specified in the rotary axis configuration parameters during tool center point control, the program error (P942) occurs.

(2) When the rotary axis configuration parameter "#7907 CHK_ANG" (Near singular judgment angle) is set to a value smaller than 0.001° (excluding 0.000), the operation is carried out treating the value as 0.001°.

(3) The tool center point control cannot be commanded during the multiple-axis synchronization control. The program error (P740) occurs.

(4) This function is also available in machines with an inclined rotary axis. For details, refer to "Appendix 1: Functions with Advanced Machining" of "Specifications Manual (Function)".

(5) While the program format is switched, the tool center point control cannot be performed by using the H axis (axis where the parameter "#1013 axname" is set to "H"). According to the setting of the parameter (#1450 5axis_Spec/bit0), the operation error (M01 0127) occurs when the power is turned ON, or the program error (P932 or P934) occurs when G43.4/G43.5 is commanded. (This depends on the MTB specifications.)

(6) The acceleration/deceleration method differs depending on the parameter settings (depending on the MTB specifications). Refer to "Relationship with other functions" in "7.8 Rapid Traverse Constant-gradient Acceleration/Deceleration".

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. (This depends on the MTB specifications.)

	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 "#7914 ROT_PREFILT" (Rotary axis prefilter time constant). (*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 position or incremental position can be used for the arc end point coordinate command, but the arc center coordinate value must always be designated with an incremental position 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 / Euler angles (°) / Roll-pitch-yaw angles (°)
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 one of the following commands during tool center point control type 2.

- ♦Workpiece surface angle vector
- ♦Euler angles
- ♦Roll-pitch-yaw angles

(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.

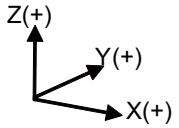
Detailed description

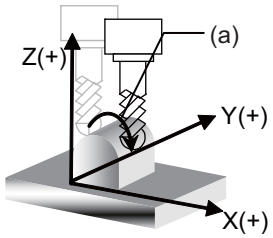
Operation for each type

(1) Tool tilt type (Rotary axis machine coordinate 0°)

<Machining program>

```
 :
G18;
G43.4 H1;
 :
G02 Xx Zz Ii Kk;
 :
```

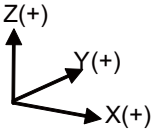


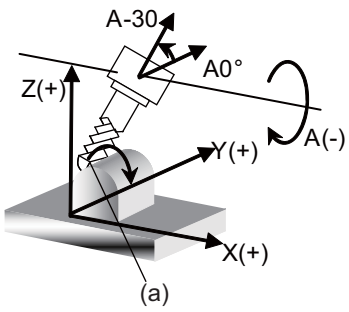


(2) Tool tilt type (Rotary axis machine coordinate - 30°)

<Machining program>

```
 :
G18;
G43.4 H1;
A-30.;
 :
G02 Xx Zz Ii Kk;
 :
```

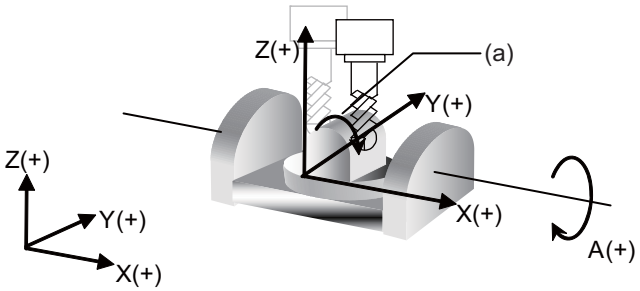




(3) Table tilt type (0° position standard)

<Machining program>

```
 :
G18;
G43.4 H1;
 :
G02 Xx Zz Ii Kk;
 :
```

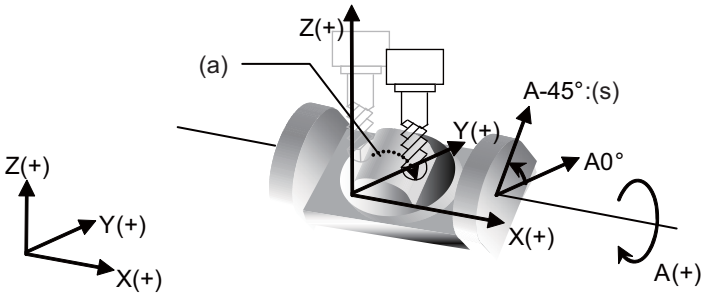


(a) Arc operations

(4) Table tilt type (Start position standard)

<Machining program>

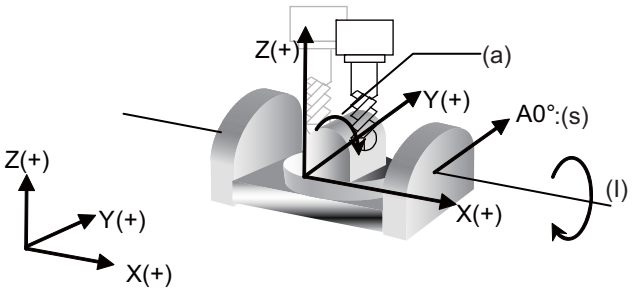
```
 :
G19 A-45.;
G43.4 H1;
 :
G02 Xx Yy Ii Jj;
 :
```



(5) Table tilt type (Programming coordinate system = workpiece coordinate system)

<Machining program>

```
 :
G18;
G43.4 H1;
 :
G02 Xx Zz Ii Kk;
 :
```

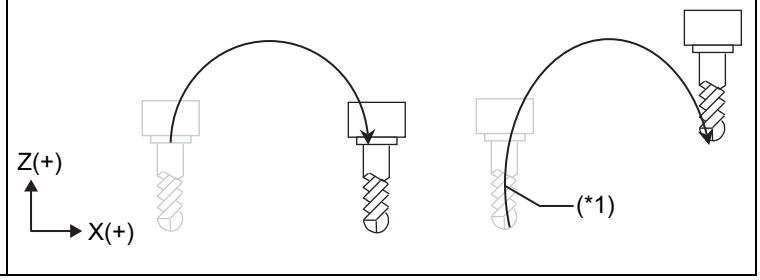


(a) Arc operations (s) Start position (I) About I axis

Precautions

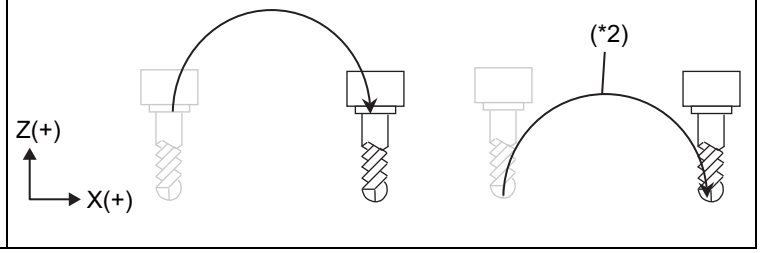
(1) If the circular command is issued without positioning three orthogonal axes after tool center point control has been started without movement command, a program error (P70) or (P71) may occur.

(a) Tool center point control commanded without movement command:

	Tool center point: OFF	Tool center point: ON
: G90G18; G43.4 H1; G02 Xx Zz Ii Kk; :		

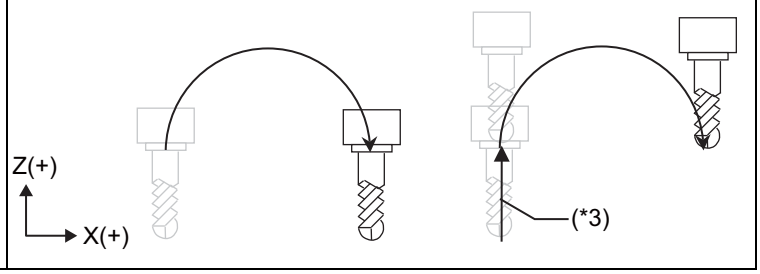
(*1) The current position is used as the tool center point, and the end point of the arc as the tool center point, so the arc radius cannot be obtained.

(b) Tool center point control commanded without movement command:

	Tool center point: OFF	Tool center point: ON
: G90G18; G43.4 H1; G02 Xx Ii K0; : When the start point of the Z axis is equal to the end point, no error will occur.		

(*2) The tool center point is not set as a program path.

(c) Tool center point control commanded with movement:

	Tool center point: OFF	Tool center point: ON
: G90G18; G43.4 H1 Z0; G02 Xx Zz Ii Kk; :		

(*3) The tool center point is used as the Z0 position.

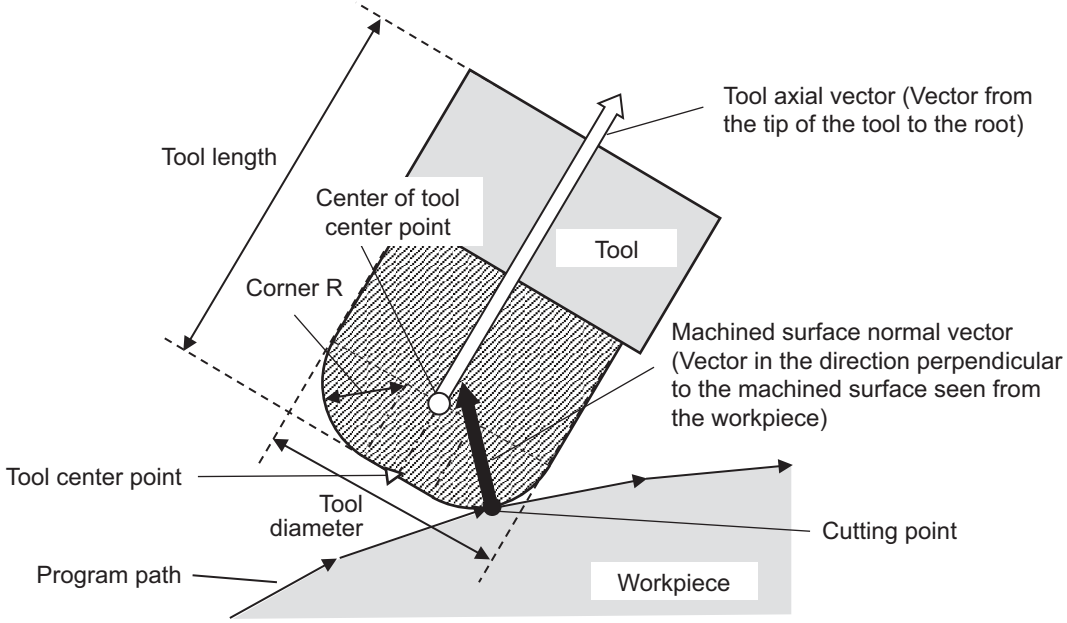
(2) When G49 (tool center point control cancel) is commanded in circular interpolation modal during tool center point control, the circular interpolation modal remains enabled. Therefore, command the linear interpolation (G00 or G01) before canceling tool center point control.

(3) If the spiral interpolation or helical interpolation command is issued during tool center point control, a program error (P942) occurs.

19.4 Tool Cutting Point Control; G43.8/G43.9

Function and purpose

Tool cutting point control is a function to control the point (cutting point) where the tool comes into contact with the workpiece so that it moves at the path and speed specified by the machining program.
NC automatically compensates for the tool length and tool shape (tool radius, corner R). Therefore, machining can be performed without changing the machining program to change tools. This has the advantage of improving the diversion of machining programs.



To use this function, additional specifications of both "tool cutting point control" and "tool center point control" are required. (This depends on the MTB specifications.)

Tool cutting point control	Tool center point control	Operation to be performed when this function is commanded
○	○	The tool cutting point control is performed.
	-	The program error (P940) occurs.
-	○	The program error (P910) occurs.
	-	

○: Available

-: Not available

Note

(1) Set the rotary axis type (parameter "#8213 Rotation axis type") of two rotary axes to "1" (short-cut valid) or "2" or "3" (linear-type rotary axis).
When it is set to "0" (short-cut invalid), it may cause a machining failure.

The applicable machine configurations depend on the MTB specifications.
For the applicable machine configurations, refer to "Appendix 1: Functions with Advanced Machining" of "Specifications Manual (Function)".

Command format

There are two command formats: tool cutting point control type 1 that commands the tool angle with the rotary axis position, and tool cutting point control type 2 that commands the tool angle with the tool axis direction vector.

Tool cutting point control type 1

```
G43.8 (X__ Y__ Z__ A__ B__ C__ H__ D__ P__ ,L2 I__ J__ K__) ;
```

X, Y, Z	Orthogonal axis movement command
A, B, C	Rotary axis movement command
H	Tool length compensation No.
D	Tool radius compensation No.
P	G00 temporary cancel designation 0: Follows the setting of the parameter #8139. 1: Does not execute G00 temporary cancel regardless of the parameter setting. 2: Executes G00 temporary cancel regardless of the parameter setting.
,L2	Machining surface normal line vector command declaration
I, J, K	Machining surface normal line vector

Note

(1) G43.8 is a modal command of G group 8.

(2) When this command is not issued in the same block as the orthogonal coordinate axis movement command or rotary axis movement command, "cancel without movement command" is applied.
If this command is issued in the same block as a movement command, "cancel with movement" is applied.

(3) If "0" is commanded to address H or address D, the tool cutting point control is disabled.

(4) If address H is omitted, the tool length compensation is not applied.

(5) If address D is omitted, the tool radius compensation is not applied.

(6) If address P is omitted, the operation follows the setting of the parameter "#8139 TCP G0 temp.cancel". If any value other than "0" to "2" is commanded, the program error (P35) occurs.

(7) If the value other than "2" is commanded following address ",L", the program error (P33) occurs.

(8) The machining surface normal line vector (address I, J, or K) must be commanded after the machining surface normal line vector command declaration (,L2). If it is commanded before the machining surface normal line vector command declaration, it is ignored.

(9) When either of address "I", "J", or "K" is omitted, the omitted address is assumed to be "0". When all of them are omitted, "I0 J0 K1" is assumed to be commanded.

(10) The program error (P33) occurs when the addresses following the machining surface normal line vector command declaration (,L2) is as follows.

- The addresses "I", "J", and "K" are set to "0".
- An address other than address "I", "J", and "K" is commanded.

Tool cutting point control type 2 (Designation by angle vector)

```
G43.9 (X__ Y__ Z__ I__ J__ K__ H__ D__ P__ ,L2 I__ J__ K__) ;
```

Tool cutting point control type 2 (Designation by Euler angles)

```
G43.9 ,P0 (X__ Y__ Z__ I__ J__ H__ D__ P__ ,L2 I__ J__ K__) ;
```

Tool cutting point control type 2 (Designation by roll-pitch-yaw angles)

G43.9 ,P1 (X__ Y__ Z__ I__ J__ K__ H__ D__ ,Q__ P__ ,L2 I__ J__ K__) ;

X, Y, Z	Orthogonal axis movement command
I, J, K	♦Angle vector of workpiece surface (Designation by angle vector) ♦Euler angles (Designation by Euler angles) ♦Roll-pitch-yaw angles (Designation by roll-pitch-yaw angles)
H	Tool length compensation No.
D	Tool radius compensation No.
,Q	Rotation order of roll-pitch-yaw angles (q: Setting value for address ",Q")

q	1st	2nd	3rd
123	X	Y	Z
132	X	Z	Y
213	Y	X	Z
231	Y	Z	X
312	Z	X	Y
321	Z	Y	X

If address ",Q" is omitted, "q" is handled as "123".

P	G00 temporary cancel designation 0: Follows the setting of the parameter #8139. 1: Does not execute G00 temporary cancel regardless of the parameter setting. 2: Executes G00 temporary cancel regardless of the parameter setting.
,L2	Machining surface normal line vector command declaration
I, J, K	Machining surface normal line vector (Commanded after address ",L2")

Note

(1) G43.9 is a modal command of G group 8.

(2) When this command is not issued in the same block as the orthogonal coordinate axis movement command or rotary axis movement command, "cancel without movement command" is applied.
If this command is issued in the same block as a movement command, "cancel with movement" is applied.

(3) A rotary axis cannot be commanded. If it is commanded, the program error (P33) occurs.

(4) If the value other than "0" or "1" is commanded to address ",P", the program error (P33) occurs.

(5) Address ",Q" can be commanded only for "designation by roll-pitch-yaw angles". Address ",Q" cannot be commanded for other designations. If it is commanded, the program error (P33) occurs. Even when address ",Q" is not the numeric value specified above, the program error (P33) occurs.

(6) If one of the following options is omitted, the omitted address is assumed to be "0". If all are omitted, the current tool angle is maintained.

♦Workpiece surface angle vector
♦Euler angles
♦Roll angle, pitch angle, and yaw angle

(7) The movement of orthogonal axis is made on the currently selected table coordinate system. The coordinate system is not updated by the euler angle, roll angle, pitch angle, or yaw angle command.

(8) If the value outside the range of -360° to 360° is commanded to the euler angle, or to the angles of roll, pitch and yaw, the program error (P35) occurs.

(9) For the euler angle, or the angles of roll, pitch and yaw, looking at the center of rotation from the positive direction of each rotation center axis, the counterclockwise direction is the positive direction of rotation.

(10) Address "K" has no meaning for the euler angle, so it does not need to be issued. This command is ignored even if it is issued.

(11) If "0" is commanded to address H or address D, the tool cutting point control is disabled.

(12) If address H is omitted, the tool length compensation is not applied.

(13) If address D is omitted, the tool radius compensation is not applied.

(14) If address P is omitted, the operation follows the setting of the parameter "#8139 TCP G0 temp.cancel". If any value other than "0" to "2" is commanded, the program error (P35) occurs.

(15) If the value other than "2" is commanded following address ",L", the program error (P33) occurs.

(16) The machining surface normal line vector (address "I", "J", or "K") must be commanded after the machining surface normal line vector command declaration (,L2).

(17) When either of address I, J, or K is omitted, the omitted address is assumed to be "0". When all of them are omitted, "I0 J0 K1" is assumed to be commanded.

(18) The program error (P33) occurs when the addresses following the machining surface normal line vector command declaration (,L2) is as follows.

- The addresses "I", "J", and "K" are set to "0".
- An address other than address "I", "J", and "K" is commanded.

Tool cutting point control cancel

G49 (X__ Y__ Z__ A__ B__ C__) ;

Cancel by another G code command in G group 8

G43 / G44 / G43.1 (X__ Y__ Z__ A__ B__ C__ H__) ;

X, Y, Z	Orthogonal axis movement command
A, B, C	Rotary axis movement command
H	Tool length offset No.

Note

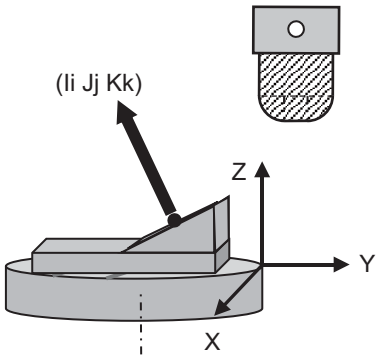
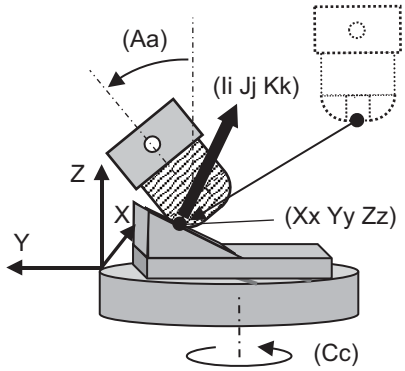
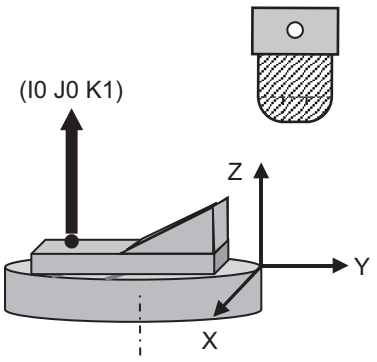
(1) When this command is not issued in the same block as the orthogonal coordinate axis movement command or rotary axis movement command, "cancel without movement command" is applied.
If this command is issued in the same block as a movement command, "cancel with movement" is applied.

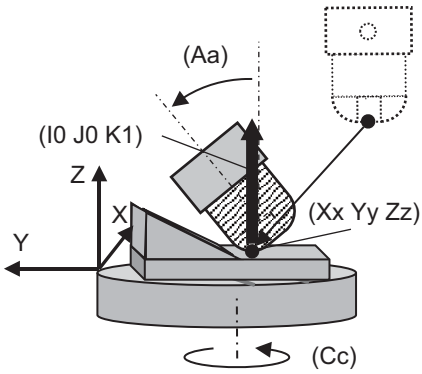
(2) If address H is set to "0" or it is omitted, the tool length compensation is not applied.

Description of operation

Start-up

There are two types: "start-up without movement" that does not accompany axis movement, and "start-up with movement" that accompanies axis movement.
The following shows an example in which the G43.8 command is issued and the rotary axis is the A/C axis; however, this is also applied to the G43.9 command and other machine configurations.

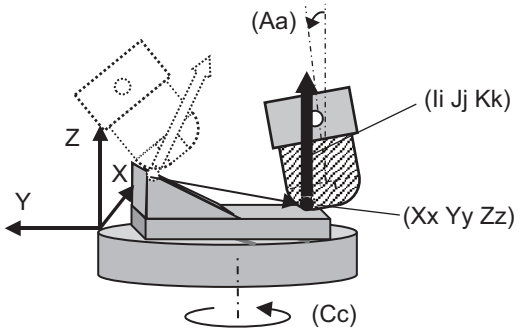
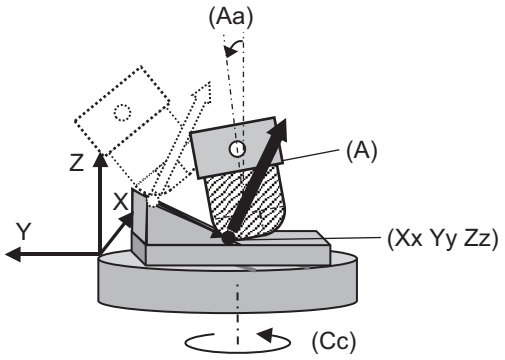
<table>
<tr><th colspan="2"></th><th>Start-up without movement</th><th>Start-up with movement</th></tr>
<tr><td colspan="2">Description of operation</td><td>The tool cutting point control starts.
At this time, both the orthogonal axis and rotary axis do not move.
The tool length and tool shape are compensated for when the first movement command is issued.</td><td>The tool cutting point control starts. After this, the axis moves by the offset amount so that the cutting point is placed at the commanded position.</td></tr>
<tr><td rowspan="2">Machining surface normal line vector command (,L2 Ii Jj Kk)</td><td>Yes</td><td>G43.8 Hh Dd ,L2 Ii Jj Kk ;

</td><td>G43.8 Xx Yy Zz Aa Cc Hh Dd ,L2 Ii Jj Kk ;

</td></tr>
<tr><td>No</td><td>G43.8 Hh Dd ;


If the machining surface normal line vector is not commanded to the start-up block, operation is performed in the same way as ",L2 I0 J0 K1".</td><td>G43.8 Xx Yy Zz Aa Cc Hh Dd ;


If the machining surface normal line vector is not commanded to the start-up block, operation is performed in the same way as ",L2 I0 J0 K1".</td></tr>
</table>

In tool cutting point control mode

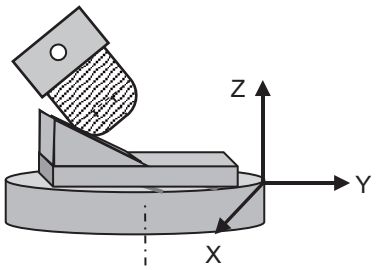
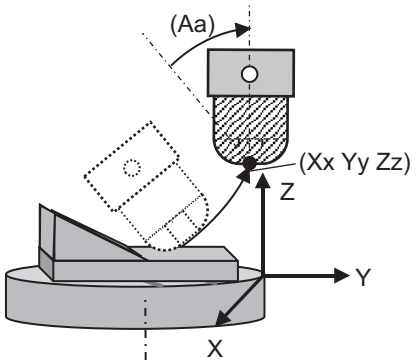
The operation in the tool cutting point control mode is as follows.
The following shows an example in which the G43.8 command is issued and the rotary axis is the A/C axis; however, this is also applied to the G43.9 command and other machine configurations.

<table>
<tr><th colspan="2"></th><th>In tool cutting point control mode</th></tr>
<tr><td colspan="2">Description of operation</td><td>Movement is performed so that the cutting point is placed at the commanded position. At this time, the cutting point moves linearly on the table coordinate system.</td></tr>
<tr><td rowspan="2">Machining surface normal line vector command (,L2 Ii Jj Kk)</td><td>Yes</td><td>G01 Xx Yy Zz Aa Cc F1000 ,L2 Ii Jj Kk ;

</td></tr>
<tr><td>No</td><td>G01 Xx Yy Zz Aa Cc F1000 ;


(A) Previous machining surface normal line vector
If the machining surface normal line vector is not commanded in the tool cutting point control mode, the previous value is used.</td></tr>
</table>

Cancel

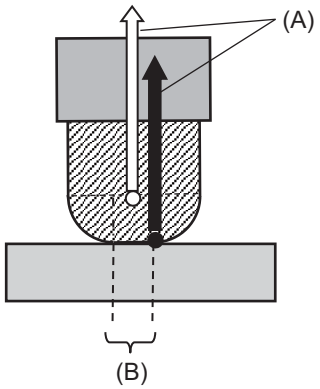
There are two types: "cancel without movement" that does not accompany axis movement, and "cancel with movement" that accompanies axis movement.
The following shows an example in which the G43.8 command is issued and the rotary axis is the A/C axis; however, this is also applied to the G43.9 command and other machine configurations.

	Cancel without movement command	Cancel with movement command
Description of operation	The tool cutting point control is canceled. At this time, both the orthogonal axis and rotary axis do not move.	The tool cutting point control mode is canceled, and then movement is performed. The cutting point may not move linearly because the tool cutting point control is canceled.
	G49 ; 	G49 Xx Yy Zz Aa ; 

Singular angle of tool cutting point control

In end mill tools such as the flat end mill and radius end mill that have a flat surface on the bottom blade part, the state in which the machining surface normal vector and the tool angle vector are parallel is referred to as the "singular angle of tool cutting point control".

In the singular angle of the tool cutting point control, it can be said that the entire bottom surface of the tool is the cutting point, so the cutting point is not uniquely determined, and the operation of the orthogonal axis may become unstable.



(A) The cutting point is not uniquely determined.
(B) The tool axis direction vector and the machining surface normal line vector are parallel.
Tool axis direction vector
Machining surface normal line vector

<Note>
- The ball end mill does not have the plane area on the bottom blade part, so there is no singular angle of tool cutting point control.

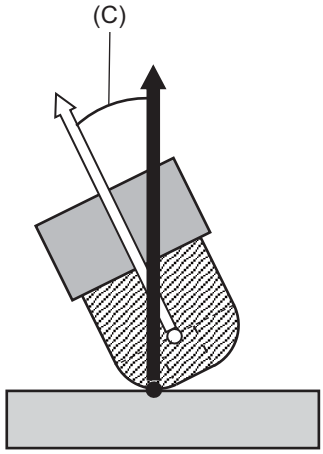
With two parameters, it is possible to avoid an unstable state when the singular angle of tool cutting point control occurs.

- Parameter "#19601 TCPC sing pt agl"
- Parameter "#19602 TCPC sing pt posn"

Singular point neighborhood judgment angle of tool cutting point control

For the parameter "#19601 TCPC sing pt agl", set the threshold of the singular point neighborhood judgment angle of tool cutting point control.

If the angle made by the machining surface normal line vector and tool axis direction vector is within the setting value, it is assumed to be the singular angle of tool cutting point control.



(C) Angle made by the tool axis direction vector and machining surface normal line vector

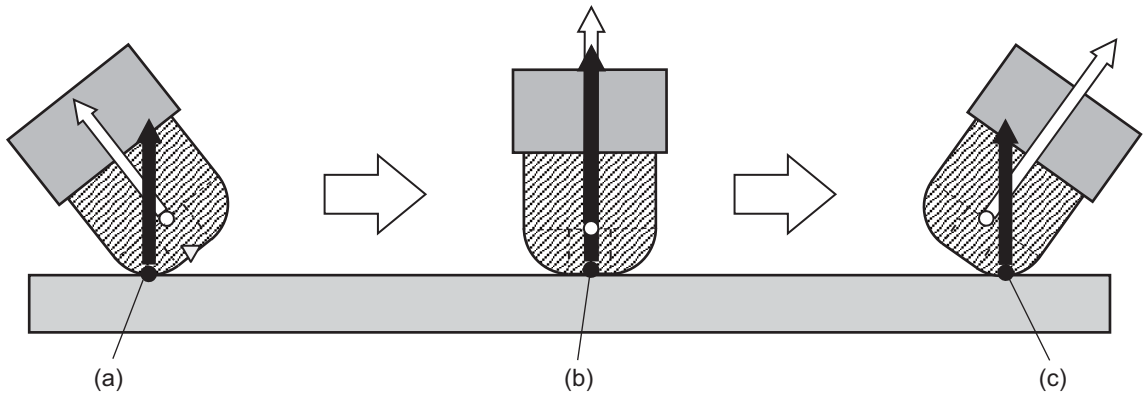
Cutting point position selection at singular angle of tool cutting point control

With the parameter "#19602 TCPC sing pt posn", it is possible to select the cutting point position when the singular angle of tool cutting point control occurs.

(1) Cutting point position selection type 1 ("#19602 TCPC sing pt posn" is set to "0")

The tool center point is set as the cutting point.
This setting is recommended to conduct machining using both sides of the tool.

[Singular angle of tool cutting point control]



(a) Cutting point immediately before the singular angle of tool cutting point control

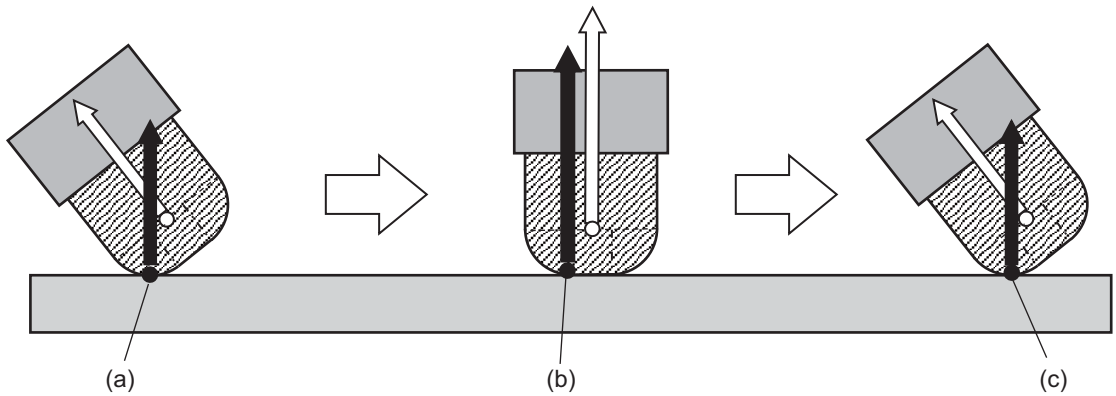
(b) Cutting point at singular angle of tool cutting point control

(c) Cutting point immediately after the singular angle of tool cutting point control

(2) Cutting point position selection type 2 ("#19602 TCPC sing pt posn" is set to "1")

The cutting point is set to the point on the bottom surface of the tool that is closest to the cutting point immediately before the singular angle of tool cutting point control.
This setting is recommended to conduct machining using only one side of the tool.

[Singular angle of tool cutting point control]



(a) Cutting point immediately before the singular angle of tool cutting point control

(b) Cutting point at singular angle of tool cutting point control

(c) Cutting point immediately after the singular angle of tool cutting point control

Note

In the following cases, the operation of the cutting point position selection type 1 (the tool center point is the cutting point) is performed even when the cutting point position selection type 2 is set.

(1) The singular angle of tool cutting point control has occurred in the start-up block.

(2) There is no change in the tool angle vector or the machining surface normal vector in the subsequent blocks after (1) occurred.

Specifications common to tool center point control

This function has specifications common to the tool center point control.

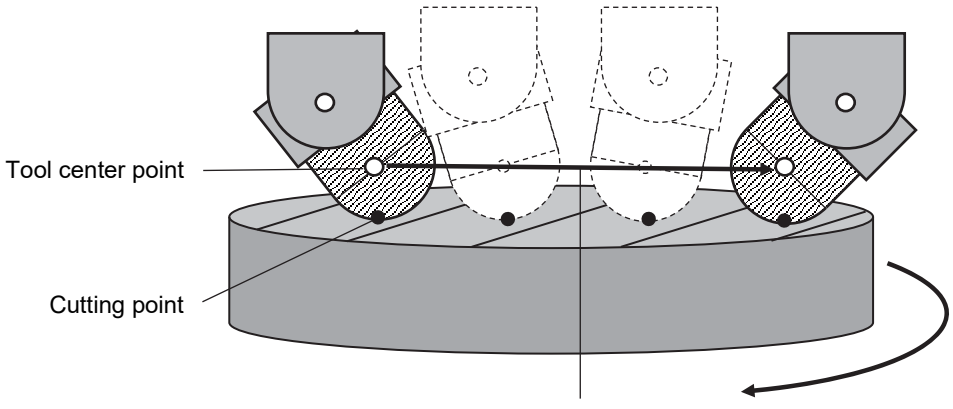
[Specifications common to tool center point control]

(1) Programming coordinate system
(2) Rotary axis reference position selection
(3) Interpolation method
(4) Singular angle (*1)
(5) Machine speed fluctuation suppression
(6) Rotary axis prefiltering
(7) G0 temporary cancel (*2)

(*1) The "singular angle" described in this section and the "singular angle of tool cutting point control" have different meanings.
The tool cutting point control provides the following two types of singular angles.

Item	Meaning	Relation (○: Related, -: Not related)	
		Tool cutting point control	Tool center point control
Singular angle of tool cutting point control	Refers to the state in which the machining surface normal line vector and the tool angle vector are parallel in the end mill tools such as the flat end mill and radius end mill that have a flat surface on the bottom blade part.	○	-
Singular angle	Refers to the state in which the tool axis direction vector and the axis direction of the secondary rotary axis are parallel.	○	○

(*2) In the tool cutting point control mode, the G0 temporary cancel is applied to the path of the tool center point. Therefore, the cutting point may be repositioned when the tool angle is changed.
When issuing the G0 temporary cancel command, do not issue a command that involves a large change in the angle.



The tool center point moves linearly to the commanded position on the machine coordinate system during G0 temporary cancel operation.

Compensation number

[When the address H is commanded alone]

When the parameter "#1274 ext10/bit3" is "1", the H modal can be updated even if the address H of the tool compensation function is commanded alone.
Refer to "12.7.1 H Stand-alone Command" for details.

[Tool offset command check]

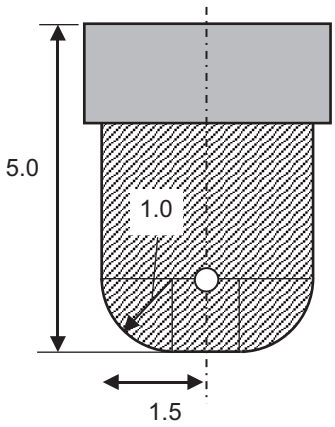
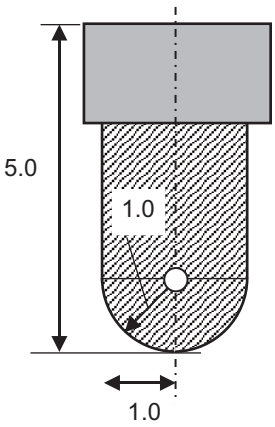
It is possible to make a program error occur when the tool offset No. is "0" or when the offset amount specified with the tool offset No. is "0", to prevent commands from being forgotten.
Refer to "12.7.2 Tool Offset Command Check" for details.



Program example

An example of using this function is shown below.

[Tools used for machining]

Radius end mill (Φ3)		Ball end mill (Φ2)
		
Tool length	5.0 (mm)	5.0 (mm)
Tool radius	1.0 (mm)	1.0 (mm)
Corner rounding	1.5 (mm)	1.5 (mm)

[Parameter setting]

#19601 TCPC sing pt agl	0.001 (°)
#19602 TCPC sing pt posn	1 (Cutting point position selection type 2)
#19603 TCPC corner R	1.0 (mm)
#8117 OFS Diam DESIGN	0 (Radius value designation)

Setting of tool offset screen (for compensation type II)

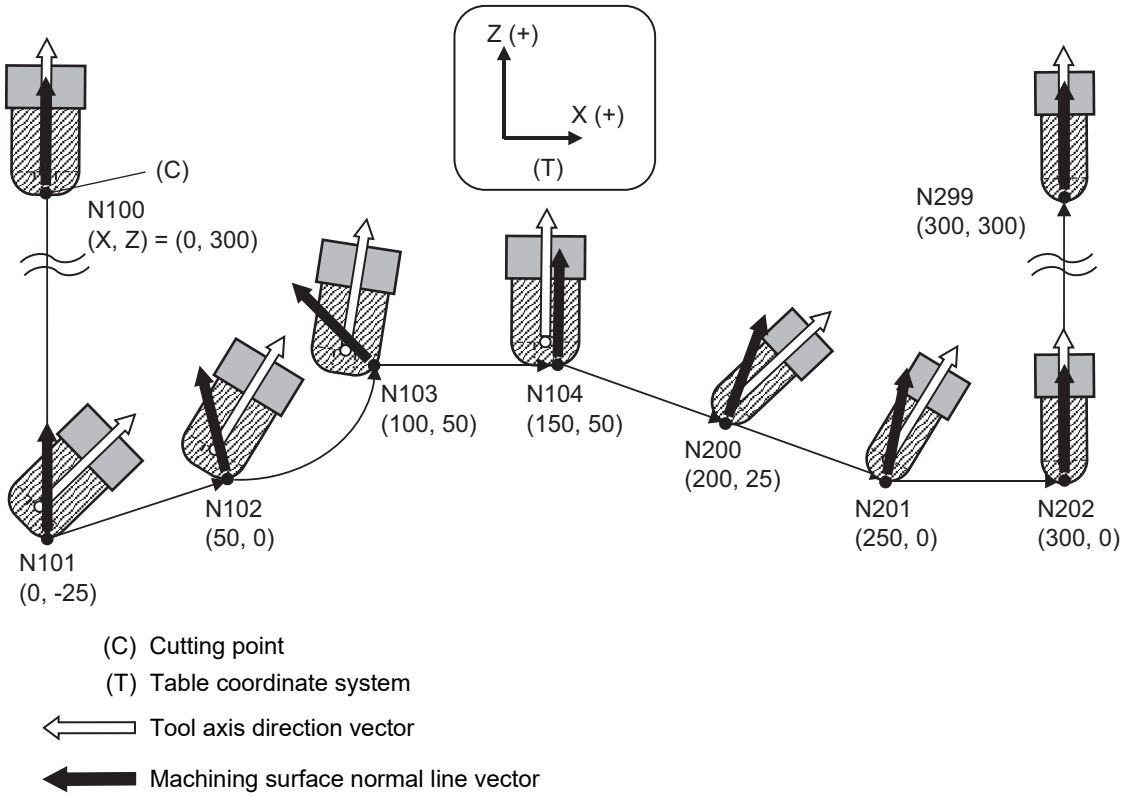
Compensation No.	Length dimension (mm)	Radius dimension (mm)
1	50.000	1.000
2	75.000	1.500
3	100.000	3.000
4	150.000	5.000

In addition, it is assumed that the parameters required to operate the tool cutting point control are set.

[Program example]

N No.	Machining program	Description of operation
:	:	:
99	G90 G54 G94 G18 G00 X0. Z300. B0. ;	
100	G43.8 H1 D2 ;	(1) Tool cutting point control type 1 starts using the radius end mill (φ3). (2) This work is performed by the start-up without movement; therefore, the axis does not move. (3) The machining surface normal line vector was omitted at start-up; therefore, it is assumed that ",L2 I0 J0 K1" was commanded.
101	G00 Z-25. B45. ;	(1) The cutting point moves linearly to the commanded position with rapid traverse. (2) The machining surface normal line vector command was omitted during the tool cutting point control mode; therefore, the previous value (I0 J0 K1) is used as the machining surface normal line vector.
102	G01 X50. Z0. B30. F1000. ,L2 I-0.124 J0. K0.992 ;	The cutting point moves linearly to the commanded position at 1000 (mm / min).
103	G02 X100. Z50. K50. B15. ,L2 I-0.707 J0. K0.707 ;	(1) The cutting point moves to the commanded position by an arc at 1000 (mm / min). (2) It is also possible to command an arc that changes the tool angle.
104	G01 X150. Z50. B0. ,L2 I0. J0. K1. ;	(1) The cutting point moves linearly to the commanded position at 1000 (mm / min). (2) The tool axis direction vector and the machining surface normal line vector are parallel, which causes the singular angle of tool cutting point control. (3) Cutting point position selection type 2 is used; therefore, the cutting point is set to the point on the bottom surface of the tool closest to the immediately preceding cutting point.
:	:	:
199	G49 ;	(1) Tool cutting point control type 1 is canceled. (2) This work is performed by the cancel without movement; therefore, the axis does not move.
200	G43.9 G01 X200. Z25. I0.707 J0. K0.707 H1 D1 F1500 ,L2 I0.124 J0. K0.992 ;	(1) Tool cutting point control type 2 starts using the ball end mill (φ2). (2) This work is performed by the start-up with movement; therefore, the cutting point moves linearly to the command position at 1500 (mm/min).
201	X250. Z0. I0.5 J0. K0.866 ,L2 I0.229 J0. K0.973 ;	The cutting point moves linearly to the commanded position at 1500 (mm/min).
202	G01 X300. Z0. I0. J0. K1. ,L2 I0. J0. K1. ;	(1) The cutting point moves linearly to the commanded position at 1500 (mm/min). (2) The ball end mill does not provide the singular angle of tool cutting point control; therefore, the cutting point moves as commanded.
:	:	:
299	G49 G00 Z300. ;	(1) Tool cutting point control type 2 is canceled. (2) This work is performed by the cancel with movement; therefore, the cutting point moves to the commanded position with rapid traverse.
300	M30 ;	

[Operation examples]



Relationship with other functions

Commands that can be issued to the same block as the tool cutting point control

The following commands can be issued to the same block as the tool cutting point control.

Command	G group	Function
G09	0	Exact stop check
G00 / G01	1	Positioning, Linear interpolation
G17 / G18 / G19	2	Plane selection (X-Y, Z-X, Y-Z)
G90 / G91	3	Absolute command, Incremental command
G93	5	Inverse time feed
G94		Feed per minute (asynchronous feed)
G61	13	Exact stop check mode
G61.1		High-accuracy control
G61.4		Spline interpolation 2
G64		Cutting mode
F	-	Feedrate command
MSTB		Miscellaneous functions, 2nd miscellaneous functions
Macro command		Local variables, Common variables, Arithmetic commands (such as four arithmetic operations, trigonometric functions, square root)

If a command other than the above is issued, an alarm such as the program error (P911) occurs.

Note

(1) Do not use the miscellaneous function macro call command.

(2) If a fixed cycle (G70 to G89) and the tool cutting point control are commanded in the same block, the fixed cycle is executed ignoring the tool cutting point control.

(3) If the user macro simple call (G65/G66/G66.1/G67) and the tool cutting point control are commanded in the same block, the user macro is executed after the tool cutting point control has been completed.

Commands that can be issued during the tool cutting point control modal

The following commands can be issued during the tool cutting point control mode.

Command	G group	Function
G04	0	Dwell (time-based designation)
G05 P0 / P1 / P2 / P10000 / P20000		High-speed machining mode, High-speed high-accuracy control II/III
G05.1 Q0 / Q1		High-speed high-accuracy control I
G08 P0 / P1		High-accuracy control
G09		Exact stop check
G65		User macro simple call
G00 / G01	1	Positioning, Linear interpolation
G02 / G03		Circular interpolation CW, Circular interpolation CCW
G17 / G18 / G19	2	Plane selection (X-Y, Z-X, Y-Z)
G90 / G91	3	Absolute command, Incremental command
G93	5	Inverse time feed
G94		Feed per minute (asynchronous feed)
G20 / G21	6	Inch input, Metric input
G43 / G44	8	Tool length compensation (+), Tool length compensation (-)
G43.1		Tool length compensation along the tool axis
G49		Tool length compensation cancel
G98 / G99	10	Hole drilling cycle I point return, R point return
G54 - G59 / G54.1	12	Workpiece coordinate system selection, Extended workpiece coordinate system selection
G61	13	Exact stop check mode
G61.1		High-accuracy control
G61.4		Spline interpolation 2
G64		Cutting mode
G66 / G66.1 / G67	14	User macro modal call A/B/cancel
G0.5	28	Rapid traverse block overlap
M98 / M99	-	Subprogram call
F		Feedrate command
MSTB		Miscellaneous functions, 2nd miscellaneous functions, Miscellaneous function macro call
Macro command		Local variables, Common variables, Arithmetic commands (such as four arithmetic operations, trigonometric functions, square root), Control commands (IF - GOTO - , WHILE - DO -)

If a command other than the above is issued, an alarm such as the program error (P912) occurs.

Modes where the tool cutting point control command is available

The tool cutting point control can be commanded in the following modes.

Command	G group	Function
G05 P0 / P1 / P2 / P10000 / P20000	0	High-speed machining mode, High-speed high-accuracy control II/III
G05.1 Q0 / Q1		High-speed high-accuracy control I
G08 P0 / P1		High-accuracy control
G00 / G01	1	Positioning, Linear interpolation
G17 / G18 / G19	2	Plane selection (X-Y, Z-X, Y-Z)
G90 / G91	3	Absolute command, Incremental command
G23	4	Stroke check before travel
G93	5	Inverse time feed
G94		Feed per minute (asynchronous feed)
G20 / G21	6	Inch input, Metric input
G40	7	Tool radius compensation cancel
G43 / G44	8	Tool length compensation (+), Tool length compensation (-)
G43.1		Tool length compensation along the tool axis
G49		Tool length compensation cancel
G80	9	Fixed cycle cancel
G98 / G99	10	Hole drilling cycle I point return, R point return
G50	11	Scaling cancel
G54 - G59 / G54.1	12	Workpiece coordinate system selection, Extended workpiece coordinate system selection
G61	13	Exact stop check mode
G61.1		High-accuracy control
G61.4		Spline interpolation 2
G64		Cutting mode
G66 / G66.1 / G67	14	User macro modal call A/B/cancel
G40.1	15	Normal line control cancel
G68.2 / G68.3	16	Inclined surface machining, Inclined surface machining (Tool axis direction command)
G69		Coordinate rotation cancel, 3-dimensional coordinate conversion cancel
G97	17	Constant surface speed control cancel
G15	18	Polar coordinate command cancel
G50.1	19	Mirror image by G code cancel
G13.1	21	Polar coordinate interpolation cancel
G54.2 P0	23	Workpiece position offset for rotary axis cancel
G54.4 P0 / P1 - P7	27	Workpiece installation error compensation ON/OFF
G0.5	28	Rapid traverse block overlap

If a command is issued in a mode other than the above, an alarm such as the program error (P911) occurs.



Relationship with other functions

(1) Program format switch

If the tool cutting point control is commanded when the G code series has been switched from the L system to the M system, the program error (P910) occurs.

(2) Inch/Metric changeover

The tool cutting point control is available in both the inch and metric modes.
However, if inch/metric is switched during the tool cutting point control mode, the program error (P912) occurs.

(3) G00 non-interpolation

Functions as the G00 interpolation operation.

(4) Circular interpolation

Arc-interpolates the cutting point position in the table coordinate system.
The machining surface normal line vector can also be commanded in the arc interpolation (G02/G03) block.
However, if the difference of machining surface normal vector between the start and end points is large, the program error (P70) may occur.
Other specifications and restrictions are the same as for the tool center point control.

(5) Cutting feed override/Rapid traverse override

Override is applied to the feedrate at the cutting point.
If speed clamp is valid, the override is applied to the clamp speed.

(6) F 1-digit feed

Controls so that the cutting point moves at the commanded speed.

(7) Dry run

Dry run is applied to the speed of the cutting point.

(8) External deceleration

External deceleration is applied to the feedrate of the cutting point.

(9) G00 feedrate designation (",F" command)

The G00 feedrate designation by address ",F" is disabled.

(10) Buffer correction

Buffer correction cannot be performed. If an attempted is made to perform it, an error message is displayed.

(11) Program restart

Restart search cannot be performed. If an attempted is made to perform it, the program error (P49) occurs.

(12) Spindle position control (Spindle/C axis control)

The axis set to the spindle/C axis can be used as a rotary axis for the tool cutting point control.
However, if the tool cutting point control is commanded in the spindle mode, the program error (P934) occurs.
Also, when the mode is switched from the C axis mode to the spindle mode during the tool cutting point control mode, the operation error (M01 0186) occurs. The operation error (M01 0186) can be reset by NC reset.

(13) Spindle synchronization I/II

When the reference spindle is in the C axis mode, the axis in spindle synchronization can be used as a rotary axis for the tool cutting point control.
However, if the tool cutting point control is commanded while the reference spindle is in the spindle mode, the program error (P934) occurs.
Also, when the mode is switched from the C axis mode to the spindle mode during the tool cutting point control mode, the operation error (M01 0186) occurs. The operation error (M01 0186) can be reset by NC reset.

(14) Miscellaneous function (MSTB)

When the tool passes through the singular point, the strobe signal is output at the start of the block, and a wait for completion is made when the block is completed.

(15) Coordinate system

The tool cutting point control is performed in the currently selected workpiece coordinate system.
It is not possible to switch the selected coordinate system during the tool cutting point control mode. If an attempt is made to switch it, the program error (P912) occurs.

(16) Manual reference position return

The manual reference position return cannot be performed. If it is performed, the tool moves off the programmed track after that.

(17) Manual interruption

When the manual interruption is made during the feed hold or single block stop, the manual ABS is disabled regardless of whether the absolute/incremental command is issued after the NC is powered ON again.

(18) NC reset

If NC reset is made, the tool stops immediately.
The tool cutting point control mode is canceled even if NC reset 1 and the modal retention are set.

(19) Emergency stop

If an emergency stop is made, the tool stops immediately.

(20) Machine lock

The machine lock for each axis becomes valid for the motor axis.

(21) Stored stroke limit

The stored stroke limit for each axis becomes valid for the motor axis.
If the stored stroke limit is applied to even one axis, all axes stop immediately.

(22) Interlock

The interlock for each axis becomes valid for the motor axis.
If even one axis is interlocked, all axes stop immediately.

(23) Graphic check / 3D solid program check

The path is drawn for the basic three axes; therefore, the correct path cannot be drawn.

(24) Graphic trace

Traces the path of the cutting point.

(25) Machining time computation

The machining time cannot be calculated correctly.

(26) Manual arbitrary reverse run

The reverse run cannot be performed by the manual arbitrary reverse run function.

(27) Arbitrary reverse run

The reverse run cannot be performed by the arbitrary reverse run function.

(28) Chopping

The chopping operation cannot be performed for the target axis of the cutting point command.

(29) MDI interruption

MDI interruption is not available. If an attempted is made to perform it, the operation error (M01 0172) occurs.
The operation error (M01 0172) can be reset by returning the operation mode to the original.

(30) PLC interruption

PLC interruption is not available. If an attempted is made to perform it, the operation error (M01 0172) occurs.
The operation error (M01 0172) can be reset by NC reset.

(31) Macro interruption

Macro interruption is not available. If it is commanded, the program error (P912) occurs.

(32) Figure rotation

The tool cutting point control is available for the path after the figure rotation.

(33) Corner chamfering/corner rounding

Corner chamfering/Corner R is not available. If it is commanded, the program error (P912) occurs.

(34) Linear angle command

The tool cutting point control is available for the path after the linear angle command has been issued.

(35) Geometric command

The tool cutting point control is available for the path after the geometric command has been issued.

(36) Mirror image by parameter setting/Mirror image by external input

The tool cutting point control cannot be combined with the mirror image.
If the tool cutting point control is commanded during mirror image, the program error (P911) occurs.
Do not enable the mirror image during the tool cutting point control mode.

(37) Arbitrary axis exchange control

The tool cutting point control can be used in the axis configuration after the arbitrary axis exchange control (G140) has been completed.
Other specifications and restrictions are the same as for the tool center point control.

(38) Rapid traverse block overlap

The rapid traverse block overlap is disabled.

(39) Arc command overlap

The arc command overlap is disabled.

(40) High-speed high-accuracy control

The micro segment processing capability in the high-speed high-accuracy control mode is the same as for the tool center point control.

(41) Inclined axis control

The tool cutting point control cannot be combined with the inclined axis control.
If the rotary axis configuration parameter is set simultaneously with the inclined axis control parameter, the MCP error (Y05) occurs.

(42) Tool center display

The following positions and speeds are displayed in each counter during the tool cutting point control mode.

Item displayed on the screen	Description	Supplement
Tip Mach posn	Position of the cutting point in the machine coordinate system	
Tip wk G54	Position of the cutting point in the programming coordinate system	This position matches the program-commanded position.
Incln G54	Position of the cutting point in the feature coordinate system	This position matches the program-commanded position when the inclined surface is valid.
WkComp G54	Position of the cutting point in the workpiece installation coordinate system	This position matches the program-commanded position when the workpiece installation error compensation is valid.
FH	Speed of the cutting point in the table coordinate system	This speed matches the program-commanded speed (F).

(43) Inclined surface machining command

The tool cutting point control is performed on the feature coordinate system defined with the inclined surface machining command.

(44) Workpiece installation error compensation

The tool cutting point control is performed on the workpiece installation coordinate system defined with the workpiece installation error compensation command.

(45) 3-dimensional manual feed

For the tool center point rotation of the 3-dimensional manual feed, the manual feed is performed on the rotary axis and three linear axes to which the movement command is issued, while maintaining the positional relationship between the workpiece and the tool center point. (The positional relationship between the cutting point and the workpiece is not maintained.)

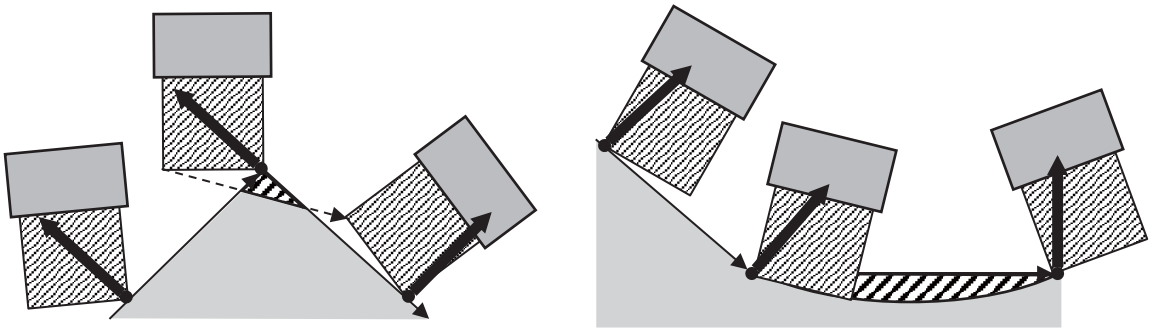
Precautions

(1) It is not possible to command an axis other than the target axis of the tool cutting point control during the tool cutting point control mode.
If it is commanded, the program error (P912) occurs.

(2) The excessive cuttings shown as below may be generated during the tool cutting point control mode.

(a) Excessive cutting at the corner due to the relationship between tool angle and cutting point position

Corners may be cut too much due to the relationship between the commanded tool angle and cutting point position.



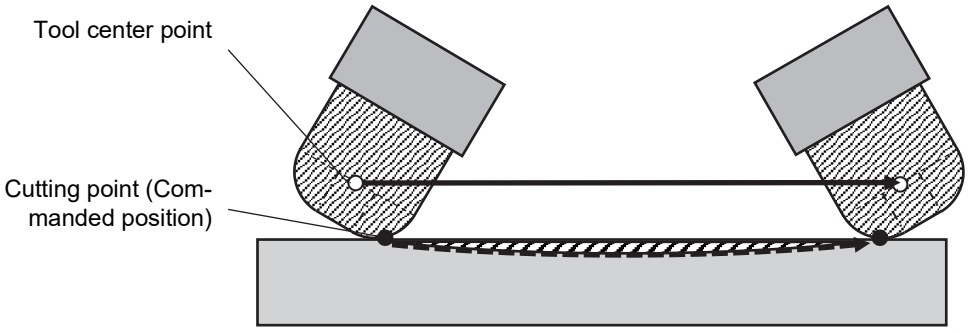
Outer corner　　Inner corner

Machining surface normal line vector

Excessive cutting

(b) Excessive cutting due to the tool angle which changed significantly

In the tool cutting point control mode, the tool center point is interpolated with a straight line; thus, the path of the cutting point is not a straight line. Therefore, if the tool angle is changed significantly, excessive cutting may be generated.



(3) If the numeric value following the program address is omitted while the parameter "#1227 aux11/bit4" (word command value check) is "1" (valid), the program error (P33) occurs.

<Example>

G43.8 H1 D1 ,L2 IJK ;	There is no number following the addresses "I", "J" and "K" an alarm occurs.

(4) If the program address is duplicated while the parameter "#1241 set13/bit2" (address duplication check) is "1" (valid), the program error (P33) occurs.

<Example>

G43.9 X100. Y200. X300. H1 D1 ;	Address "X" is duplicated, and therefore an alarm occurs.

However, in tool cutting point control type 2 (G43.9), the addresses "I", "J" and "K" to specify the tool axis direction and the addresses "I", "J" and "K" to specify the machining surface normal line vector have different meanings. Therefore, no alarm occurs even if the address duplication check is enabled.

<Example>

G43.9 H1 D1 I0.707 J0. K0.707 ,L2 I0. J0. K1.;	No alarm occurs because the addresses "I", "J", and "K" overlap with another set of addresses "I", "J", and "K", but the address ",L2" is inserted between them.

G43.9 H1 D1 I0.707 J0. K0.707 I0. J0. K1. ,L2;	An alarm occurs because the addresses "I", "J", and "K" overlap with another set of addresses "I", "J", and "K", and the address ",L2" is not inserted between them.

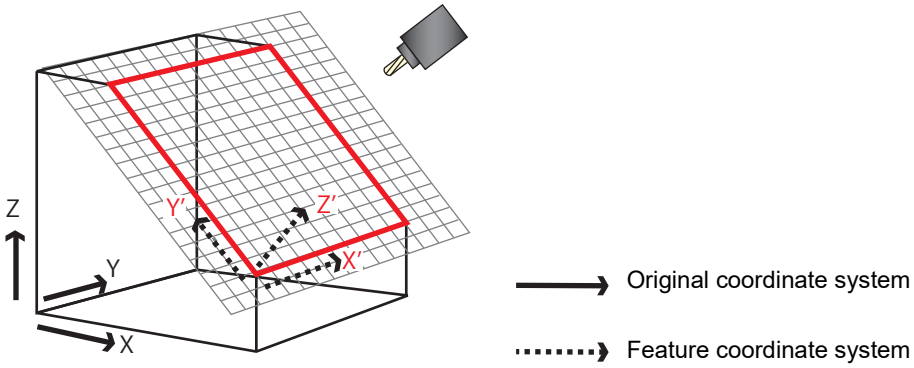
(5) The acceleration/deceleration method differs depending on the parameter settings (depending on the MTB specifications). Refer to "Relationship with other functions" in "7.8 Rapid Traverse Constant-gradient Acceleration/ Deceleration".

19.5 Inclined Surface Machining; G68.2, G68.3, G68.4/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 applicable machine configurations depend on the MTB specifications.

For the applicable machine configurations, refer to "Appendix 1: Functions with Advanced Machining" of "Specifications Manual (Function)".

Feature coordinate system command method

The feature coordinate system is defined using the following method.

G code	Command method	Section
G68.2 P0	Define using Euler angles	19.5.1
G68.2 P1	Define using roll-pitch-yaw angles	19.5.2
G68.2 P2	Define using three points in a plane	19.5.3
G68.2 P3	Define using two vectors	19.5.4
G68.2 P4	Define using projection angles	19.5.5
G68.2 P10	Define by selecting the registered machining surface	19.5.6
G68.3	Define using tool axis direction	19.5.7
G69	Cancel inclined surface machining command	

- G68.2 and G69 belong to modal of the G code group 16.
- 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.
- G68.2 must be commanded alone in a block, which also applies to G68.3 and G69. When it is commanded in the same block with other G codes or travel commands, etc., a program error (P954) occurs.
- The G69 command cannot be issued during circular interpolation or fixed cycle mode. If issued, a program error (P952) will occur.

19.5.1 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 zero point Command by the absolute positions 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 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, 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.

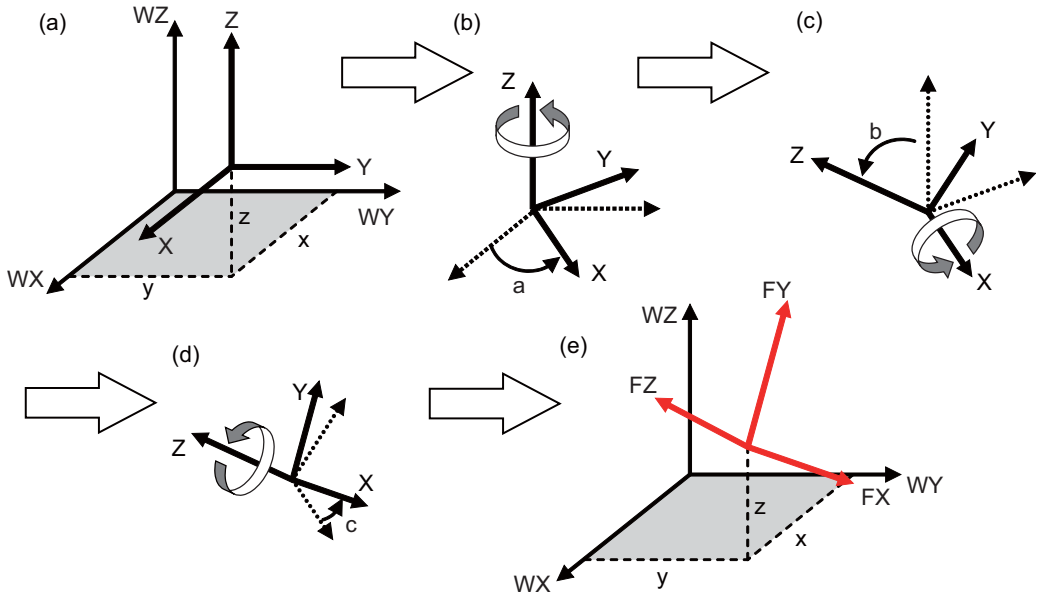
(b) Rotate the coordinate system, which was defined by shifting the origin in (a), by angle a about its Z axis.

(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.



19.5.2 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 positions with respect to the coordinate system before issuing the inclined surface machining command.
Q	Rotation order (q: Setting value for address "Q") q: 123 (First: X, Second: Y, Third: Z) 132 (First: X, Second: Z, Third: Y) 213 (First: Y, Second: X, Third: Z) 231 (First: Y, Second: Z, Third: X) 312 (First: Z, Second: X, Third: Y) 321 (First: Z, Second: Y, Third: X) If address Q is omitted, "q" will be handled as "123".
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])

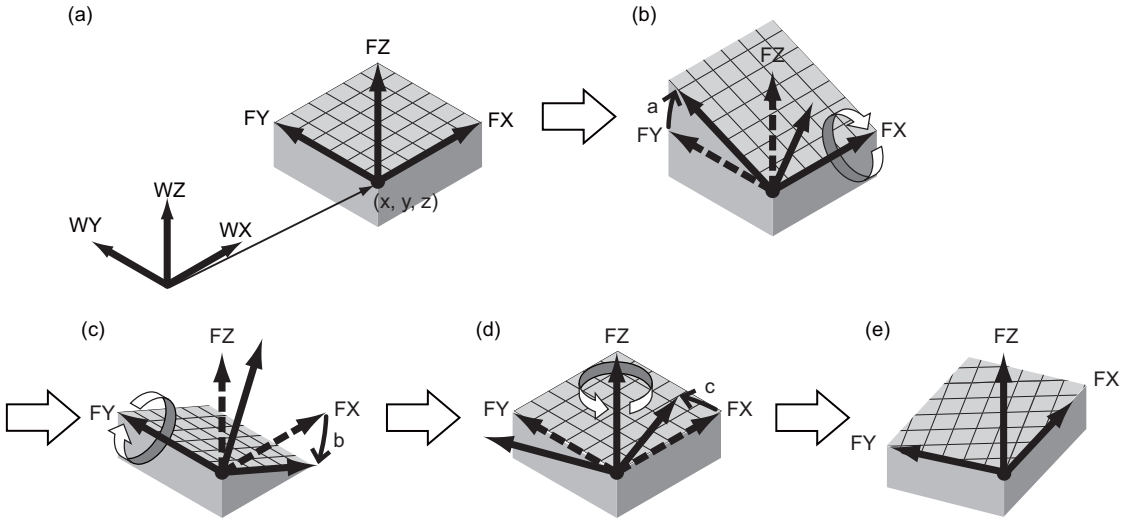
(a) Define a point (x, y, z) in the coordinate system before issuing the inclined surface machining command, as the feature coordinate system's origin.

(b) Rotate the 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.



19.5.3 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 position 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 positions 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.

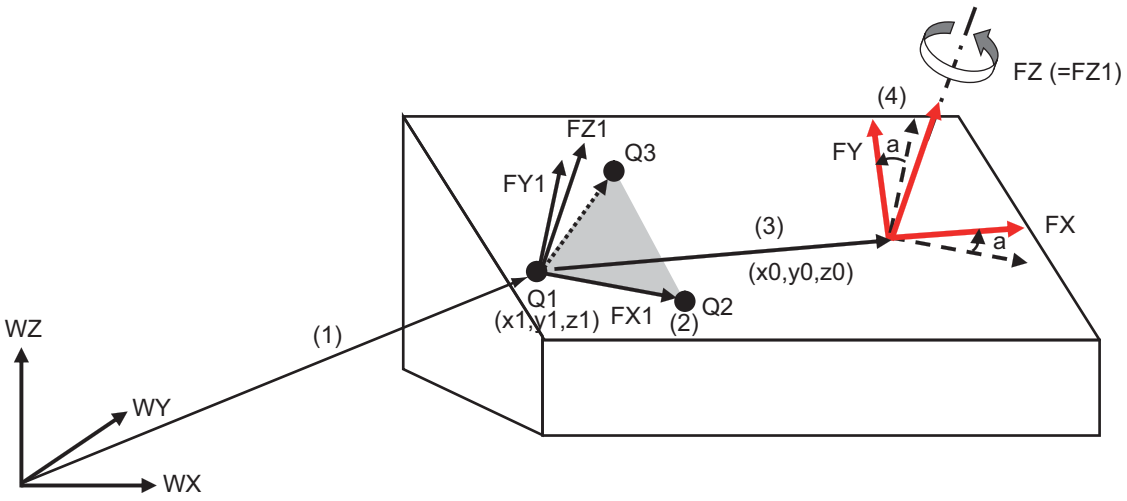
- Feature coordinate system's X axis is in the direction from the 1st point (Q1) to the 2nd point (Q2). Normally, designate a point on Y axis (+ direction) as Q3. (If the commanded X axis and Y axis are not at perfect right angles, the Y axis will be automatically compensated to be at right angles to the X axis.)
- Feature coordinate system's Z axis is in the direction of the cross product of (Q2-Q1)×(Q3-Q1).
- Feature coordinate system's Y axis is determined with respect to the right-handed system.

(3) When shift distance (x0, y0, z0) of the feature coordinate system's origin is commanded, the feature coordinate system's origin is further parallel translated by (x0, y0, z0). Command the parallel translation distance with respect to the feature coordinate system before parallel translation. Always specify x0, y0 and z0 by incremental position.

(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)

19.5.4 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 zero point Command by the absolute positions 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 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 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)×(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.

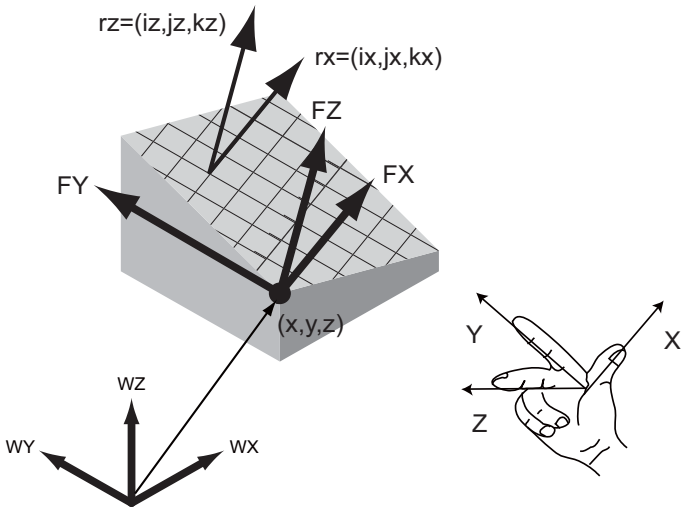
Normally, the direction of rz=(iz, jz, kz) is the Z axis (positive direction) of the feature coordinate system.

(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)

19.5.5 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 zero point Command by the absolute positions 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 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 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).

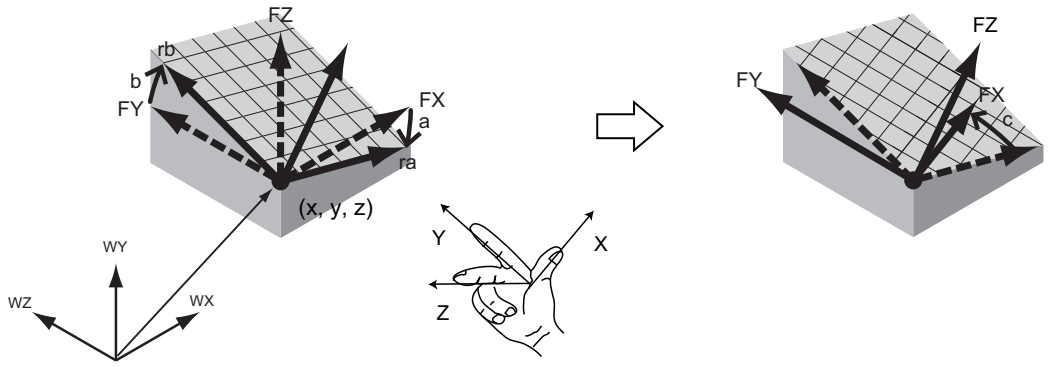
(2) Define the X, Y and Z axes' directions of the feature coordinate system in the following procedure.
- The direction in which the X axis of the coordinate system before issuing the inclined surface machining command is rotated by angle a about the Y axis is defined as "ra".
- The direction in which 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 Zy Ia Jb Kc ;



Coordinate system before issuing the inclined surface machining command
(Workpiece coordinate system)

19.5.6 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, or when "Q0" or "Q1" is commanded, the workpiece No. 1 is selected.

(2) When address D is omitted, or when "D0" or "D1" is commanded, the machining surface No. 1 (BASE-SURFACE) is selected.

(3) If any address other than P, Q, and D is designated when "G68.2 P10" is commanded, a program error (P954) occurs.

(4) A program error (P954) occurs 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) occurs.

(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 being selected" signal (XD28), "R-Navi: Selected workpiece No." signal (R660), "R-Navi: Selecting machine surface number" 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.

(10) If "G68.2 P10" is commanded when R-Navi is not applied, the program error (P956) occurs.

19.5.7 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 zero point Command by the absolute positions 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 zero point 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.

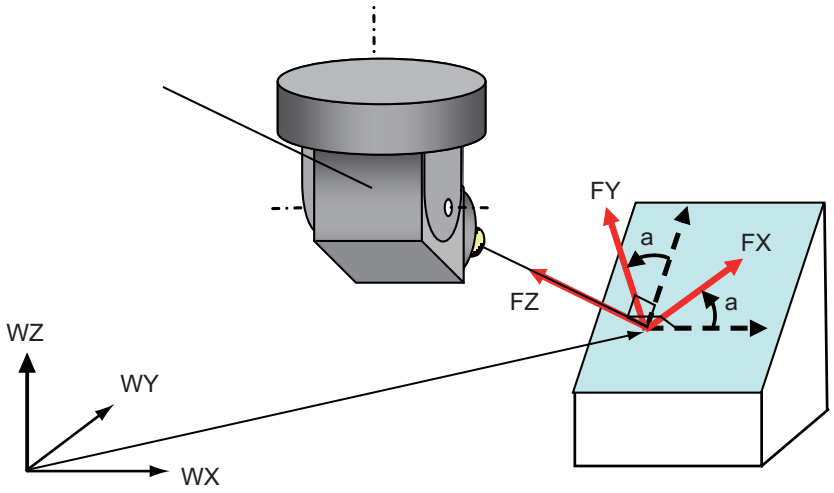
Detailed description

(1) Designate the feature coordinate system's zero point 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)

19.5.8 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) G53.1 must be commanded alone in a block, which also applies to G53.6. When it is commanded in the same block with other G codes or travel command etc., a program error (P953) occurs.

(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 commaned to address H

```
  :
G53.6 ← Error (P957)
  :
```

(12) If the inclined surface which cannot be calculated is defined when the tool axis direction control (G53.1/G53.6) is commanded, the program error (P29) occurs.

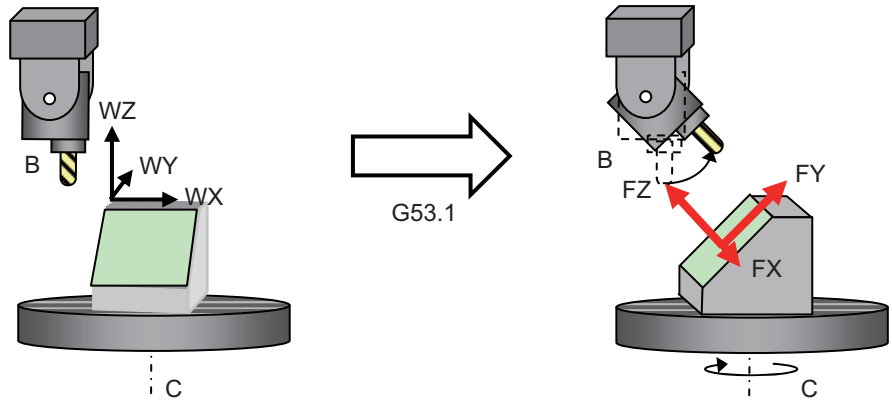
Detailed description

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.

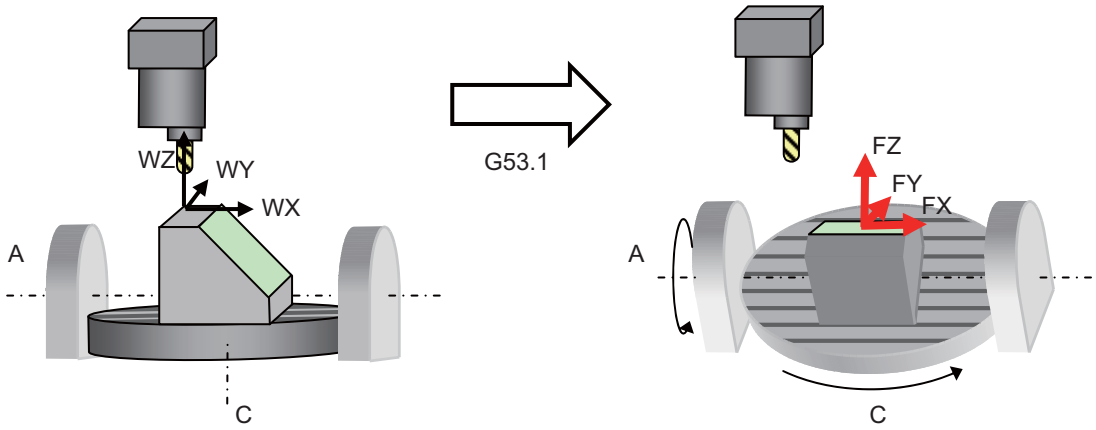
(1) For combined type B-C axes

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.



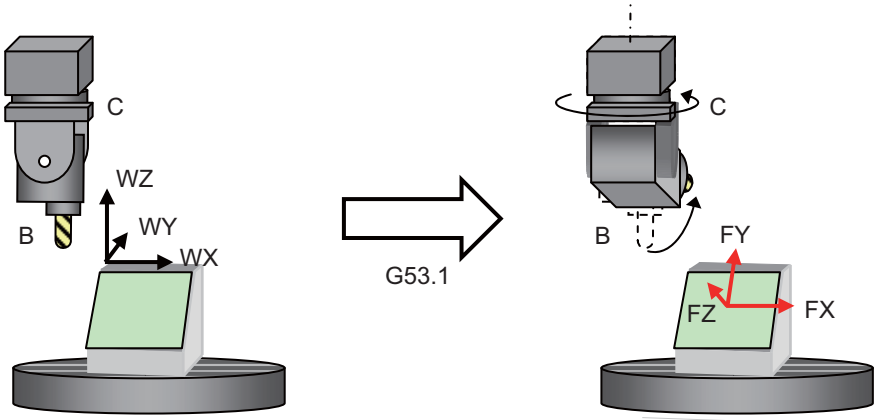
(2) For table tilt type A-C axes

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

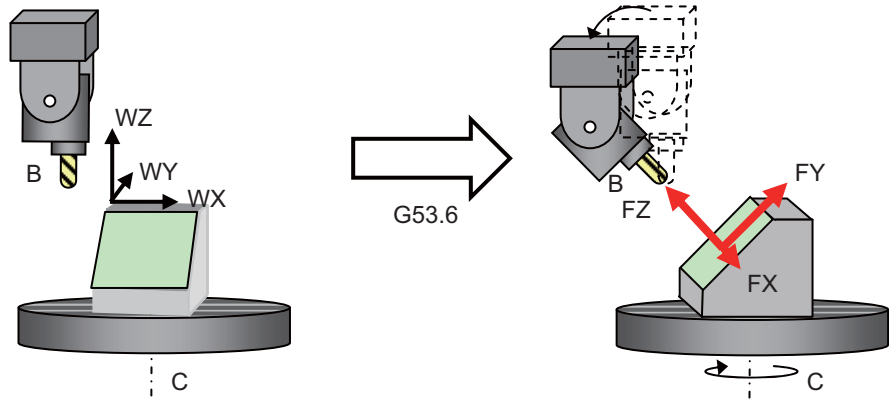
For G53.6 command, the tool axis end position of the tool axis direction seen from workpiece is fixed to be in +Z direction in the feature coordinate system, and up to 3 orthogonal axes (x axis, Y axis, and Z axis) and 2 rotary axes moves simultaneously.
The number of axes to be moved simultaneously 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.

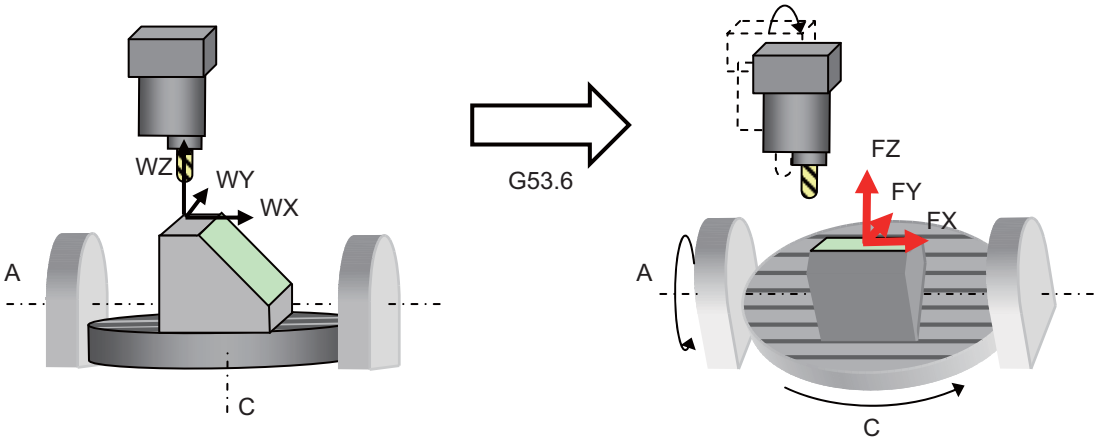
(1) For combined type B-C axes

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.



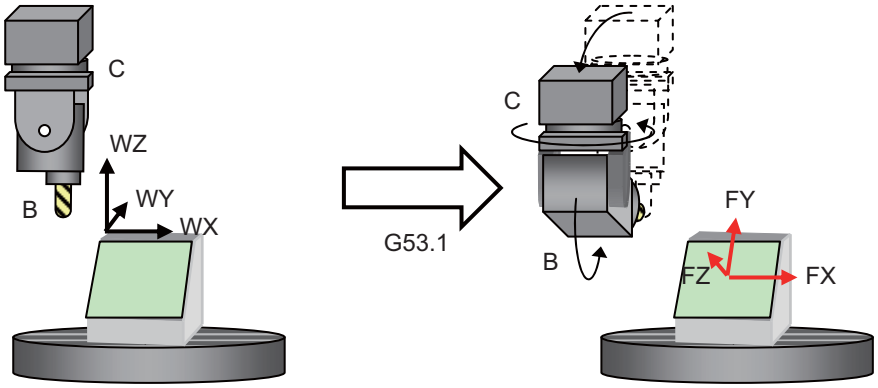
(2) For table tilt type A-C axes

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.



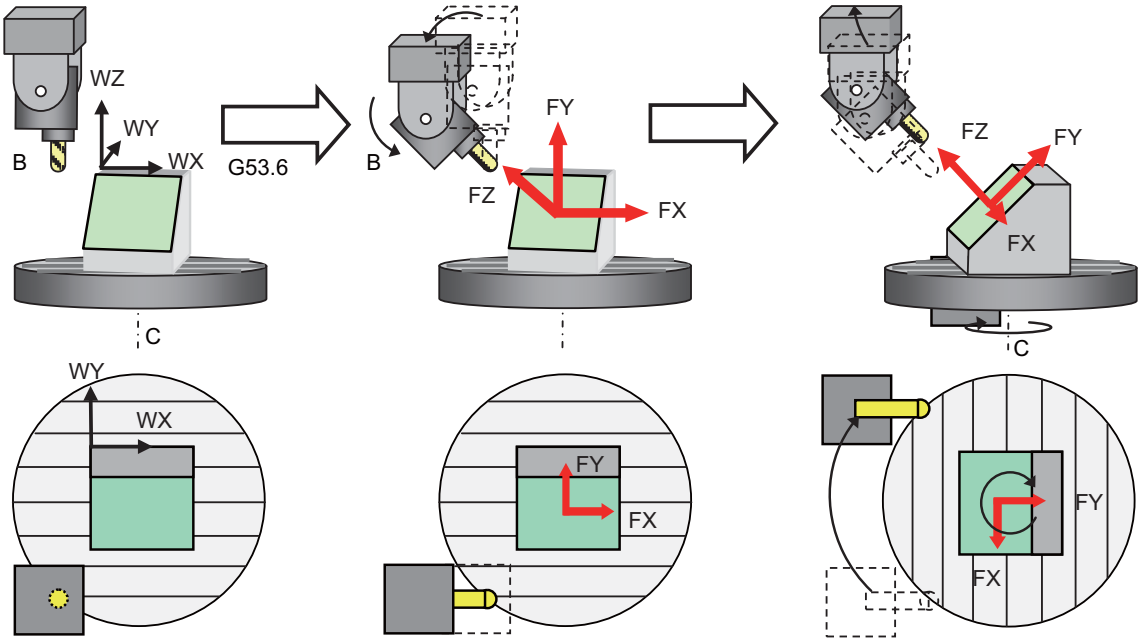
[When the number of simultaneous contouring control axes is 4 axes or fewer]

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

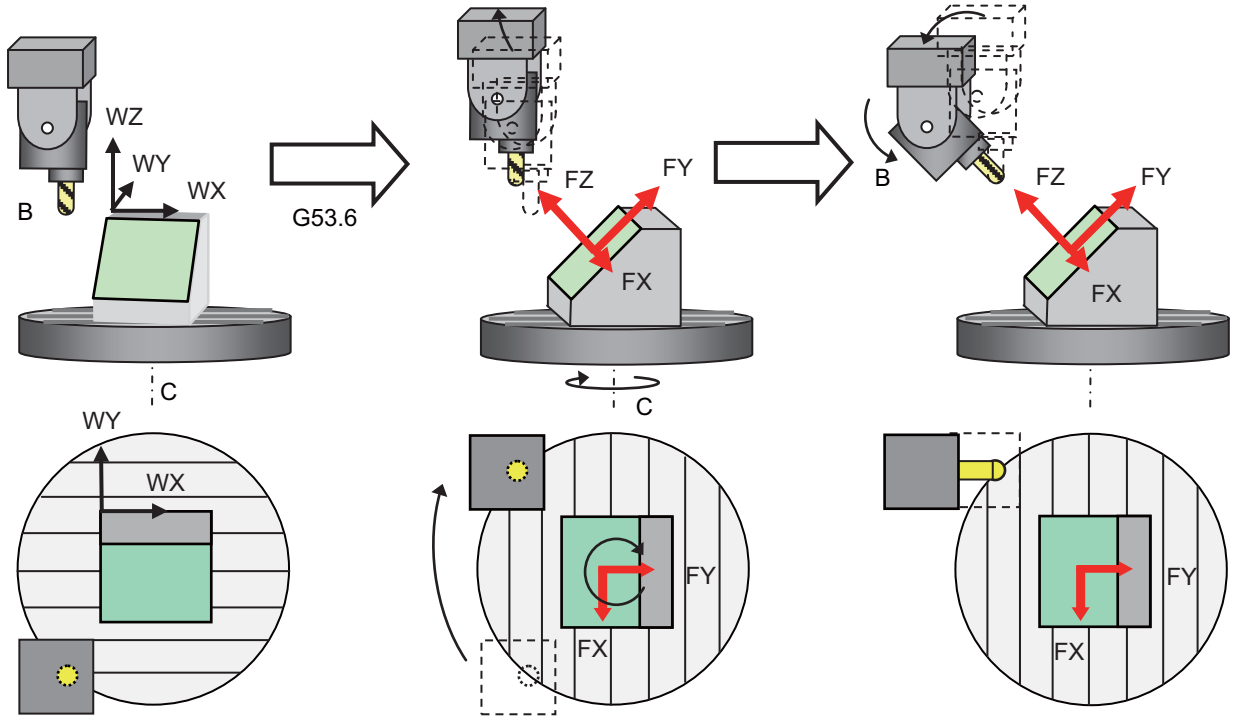
(a) When moving in the order of primary rotary axis and secondary rotary axis

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.



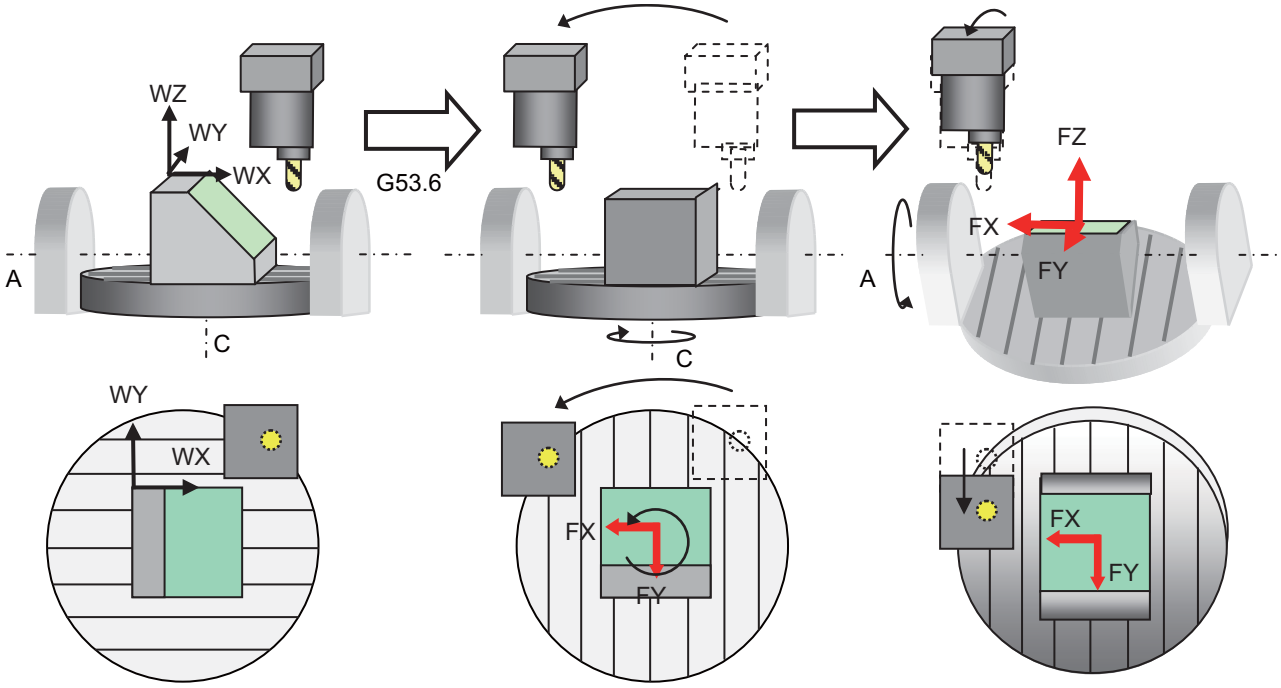
(b) When moving in the order of secondary rotary axis and primary rotary axis

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.



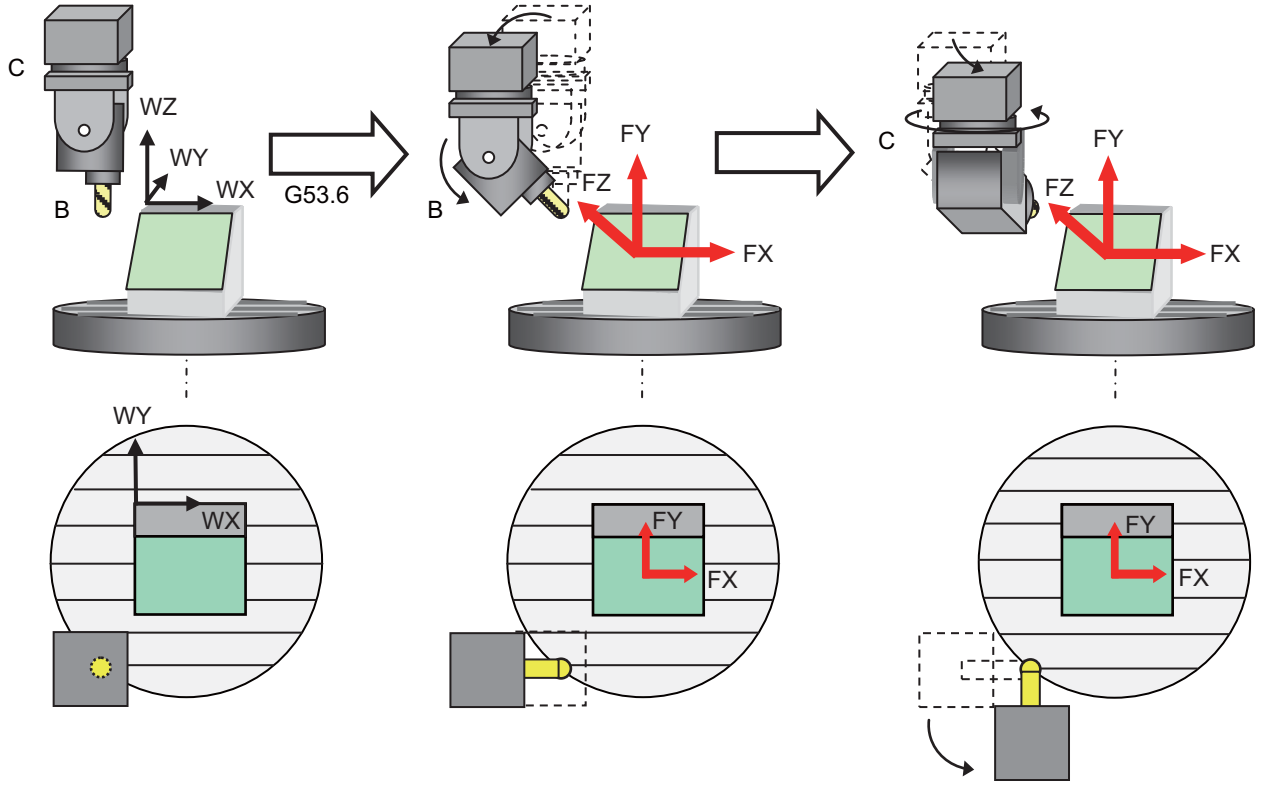
(2) For table tilt type A-C axes (When moving in the order of secondary rotary axis and primary rotary axis)

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" is any of 0, 1 or 2) in G53.1 command to select either one of the solutions.

The meaning of the address P differs depending on the polarity of the primary rotary axis.

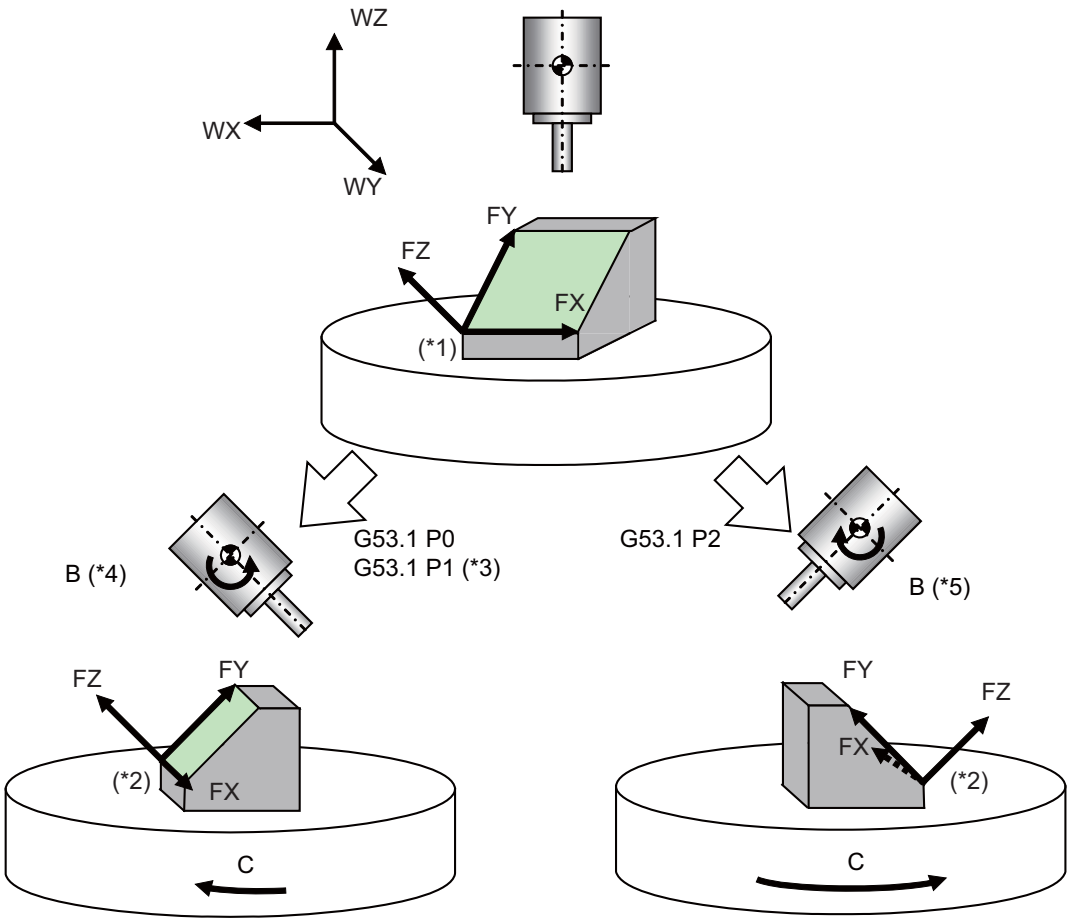
Value of "P"	Solution to be selected (ISO polarity)	Solution to be selected (non-ISO polarity)
P = 0	Default solution for each machine type	Default solution for each machine type
P = 1	The solution that makes the primary rotary axis rotation be "positive"	The solution that makes the primary rotary axis rotation be "negative"
P = 2	The solution that makes the primary rotary axis rotation be "negative"	The solution that makes the primary rotary axis rotation be "positive"

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	
		ISO polarity	Non-ISO polarity
Tool tilt type	Tool-side 2nd rotary axis	The solution that makes the primary rotary axis rotation be "positive" (same operation as when P is "1")	The solution that makes the primary rotary axis rotation be "negative" (same operation as when P is "1")
Table tilt type	Table-side 2nd rotary axis	The solution that makes the primary rotary axis rotation be "negative" (same operation as when P is "2")	The solution that makes the primary rotary axis rotation be "positive" (same operation as when P is "2")
Combined type	Tool-side rotary axis	The solution that makes the primary rotary axis rotation be "positive" (same operation as when P is "1")	The solution that makes the primary rotary axis rotation be "negative" (same operation as when P is "1")

The following figure shows the operation of combined type BC.



(*1) Indicates the 1st feature coordinate system.

(*2) Indicates the 2nd feature coordinate system.

(*3) For combined type machines, the solution that is selected when "G53.1P0" is commanded differs depending on the polarity of the primary rotary axis. When the axis has the ISO polarity, the solution that makes the primary axis rotation be "positive" will be selected. When the axis has the non-ISO polarity, the solution that makes the primary axis rotation be "negative" will be selected.

(*4) For ISO polarity, "B > 0". For non-ISO polarity "B < 0".

(*5) For ISO polarity, "B < 0". For non-ISO polarity "B > 0".

19.5.9 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

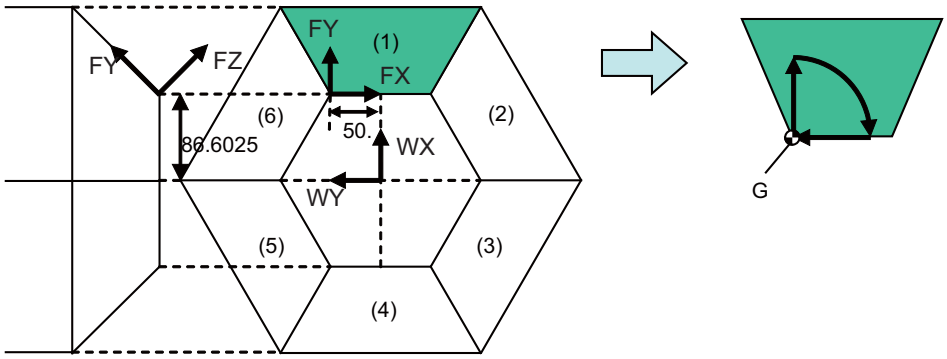
The machining program #10 is for machining an identical shape on each face of the hexagonal column using a combined type machine. Feature coordinate systems on each face are defined in the blocks N1 to N6, and then same machining is performed using the subprogram (Machining program #100). The workpiece 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

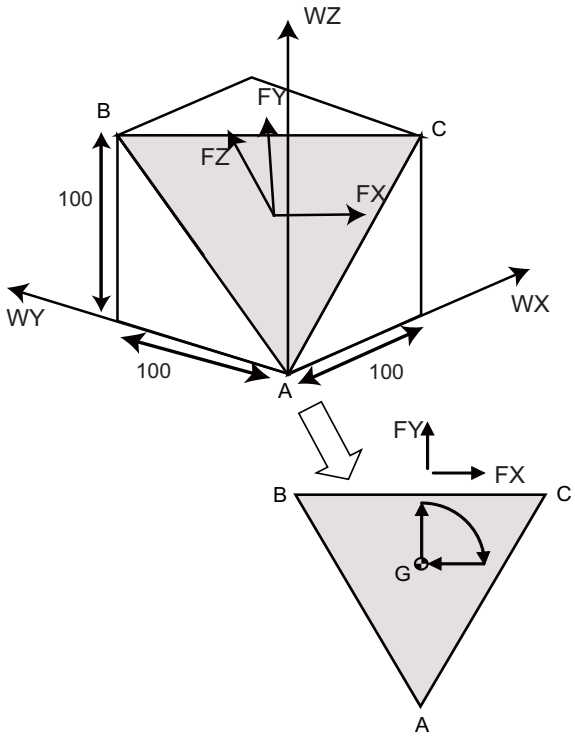
```
N5 G28XYZBC;
   G54X0Y0Z0;
   M200;
   G68.2 P4 X33.3333 Y 33.3333 Z66.6666 I-45 J45 K-60;
   M98 P100;
   G69;
   M30;
```

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;
```



(X0, Y0, Z0) = (33.3333, 33.3333, 66.6667)

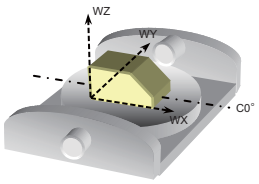
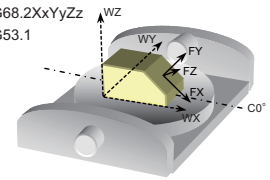
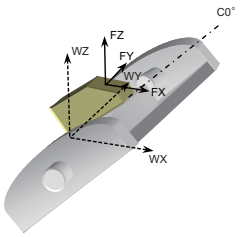
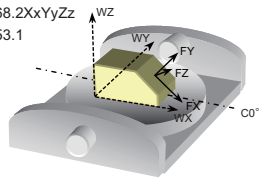
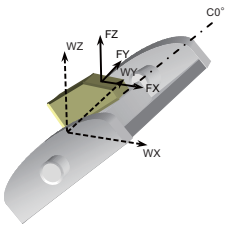
G: Feature coordinate system's origin

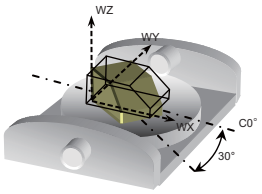
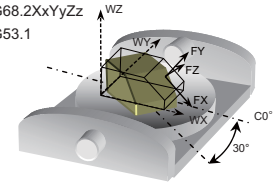
19.5.10 Rotary Axis Basic Position Selection

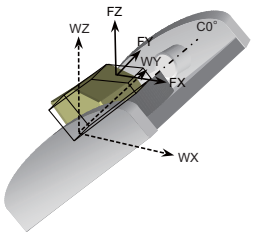
Detailed description

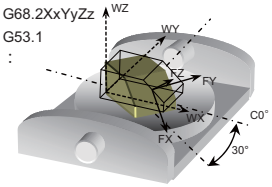
When inclined surface machining is commanded, the basic position for establishing the feature coordinate system's origin can be set with the parameter ("#7915 SLCT_SLOPE_CRD_MOD" (Rotary axis basic position in inclined surface machining)).

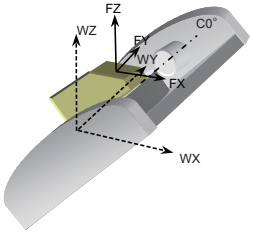
There are two types of basic position selection. One is to set the feature coordinate system in view from the workpiece coordinate system independently of the rotary axis' position when inclined surface machining is commanded (Start position basis), and the other is to set the feature coordinate system 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).

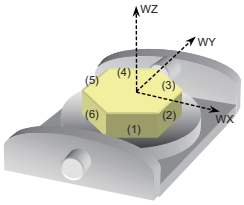
	Rotary axis basic position in inclined surface machining	
	Start position basis (#7915=1)	Zero degree position basis (#7915=0)
When workpiece is placed in the workpiece coordinate system direction: Workpiece coordinate offset A0. C0. 	: G90 G54 A0. C0. G68.2XxYyZz G53.1 :  When G68.2 is commanded: Feature coordinate system is defined at a position in view from a workpiece coordinate system.  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.	: G90 G54 A0. C0. G68.2XxYyZz G53.1 :  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.  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.

<table>
<tr><th></th><th colspan="2">Rotary axis basic position in inclined surface machining</th></tr>
<tr><th></th><th>Start position basis (#7915=1)</th><th>Zero degree position basis (#7915=0)</th></tr>
<tr><td>When workpiece is placed deviated from the workpiece coordinate system:
Workpiece coordinate offset A0. C30.

</td><td>:
G90 G54 A0. C30.
G68.2XxYyZz
G53.1
:


When G68.2 is commanded:
Feature coordinate system is defined at a position in view from a workpiece coordinate system.


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.</td><td>:
G90 G54 A0. C30.
G68.2XxYyZz
G53.1
:


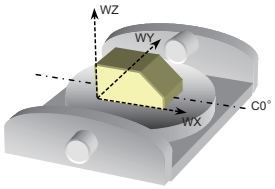
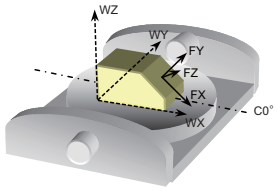
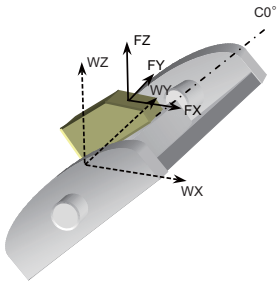
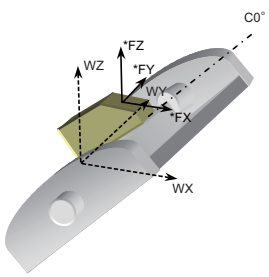
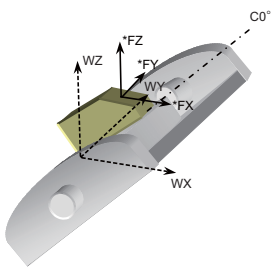
When G68.2 is commanded:
Feature coordinate system is defined at a position in view from a workpiece coordinate system regardless of rotary axis position.


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.</td></tr>
<tr><td>Example) Polygon machining:
Machining an identical shape on each of the six surfaces.

</td><td>(Subprogram)
:
G68.2 Xx Yy Zz Ii 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.</td><td>(Main program)
:
G68.2 Xx Yy Zz Ii 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.</td></tr>
</table>

Combination with tool center point (TCP) control

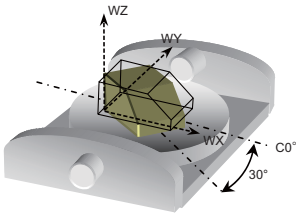
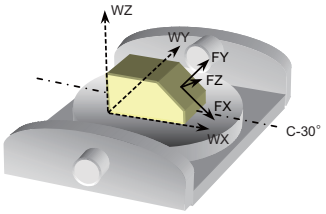
When inclined surface machining control is commanded together with the TCP control (G43.4), define the table-interlocked feature coordinate system by setting the parameter (#7911 Rotary axis basic position selection) for TCP control. When zero degree position basis (#7911=0) is selected, it is possible 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/G68.3/G68.4) or tool axis direction control command (G53.1).

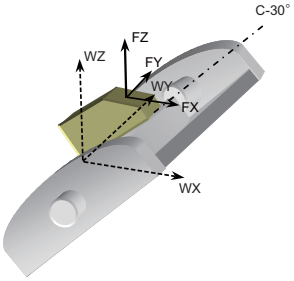
When workpiece is placed in the workpiece coordinate system direction:

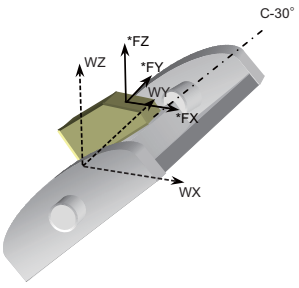
<table>
<tr><td rowspan="2"></td><td colspan="2">Rotary axis basic position selection</td></tr>
<tr><td>Workpiece coordinate system zero point for a basis (#7911 = 0)</td><td>The position when the tool center point is commanded for a basis (#7911 = 1)</td></tr>
<tr><td rowspan="3">Inclined surface machining command
Zero degree position basis (#7915 =0) Workpiece coordinate offset A 0. C0.
</td><td colspan="2">:
G90 G54 A0. C0.
G68.2XxYyZz
G53.1
G43.4
:

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.

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.</td></tr>
<tr><td>
When G43.4 is commanded:
Feature coordinate system is fixed to the table at 0 degree of the coordinate system.</td><td>
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.</td></tr>
</table>

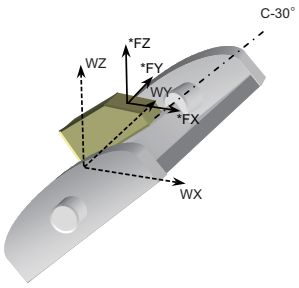
When workpiece is placed deviated from workpiece coordinate system

(1) When TCP control (G43.4) is commanded at the same angle as of tool axis direction control (G53.1)

<table>
<tr><td rowspan="2"></td><td colspan="2">Rotary axis basic position selection</td></tr>
<tr><td>Workpiece coordinate system zero point for a basis (#7911 = 0)</td><td>The position when the tool center point is commanded for a basis (#7911 = 1)</td></tr>
<tr><td rowspan="3">Inclined surface machining command
Zero degree position basis (#7915 =0) Workpiece coordinate offset A 0. C30.

</td><td colspan="2">:
G90 G54 A0. C0.
G68.2XxYyZz
G53.1
G43.4
:


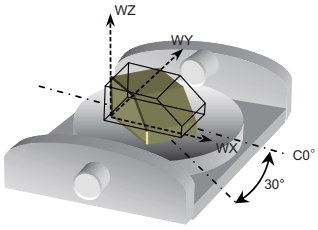
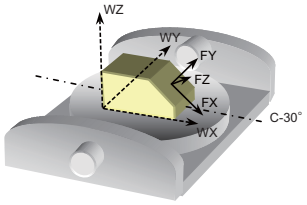
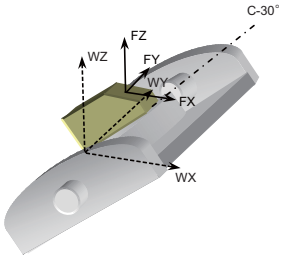
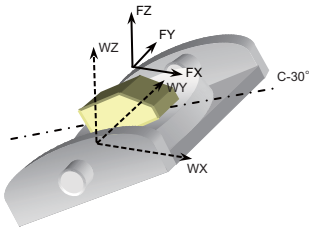
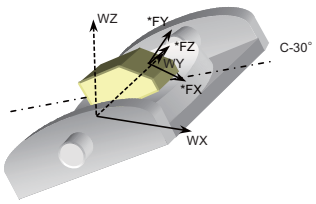
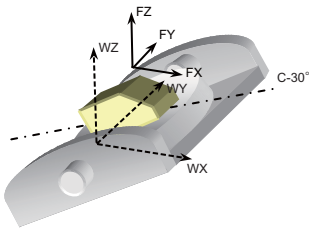
When G68.2 is commanded:
Feature coordinate system is defined at a position in view from a workpiece coordinate system regardless of rotary axis position.


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.</td></tr>
<tr><td>

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.</td><td>

When G43.4 is commanded:
Feature coordinate system is fixed to the table at the position of the present feature coordinate system.</td></tr>
</table>

(2) When TCP control (G43.4) is commanded at a different angle from tool axis direction control (G53.1).

<table>
<tr><td rowspan="2"></td><td colspan="2">Rotary axis basic position selection</td></tr>
<tr><td>Workpiece coordinate system zero point for a basis (#7911 = 0)</td><td>The position when the tool center point is commanded for a basis (#7911 = 1)</td></tr>
<tr><td rowspan="3">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
</td><td colspan="2">:
G90 G54 A0. C0.
G68.2XxYyZz
G53.1
G00 A30.
G43.4
:

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.

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.

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.</td></tr>
<tr><td>
When G43.4 is commanded:
Feature coordinate system is fixed to the table at a degree determined based on the rotary axis position.</td><td>
When G43.4 is commanded:
Feature coordinate system is fixed to the table at the position of the present feature coordinate system.</td></tr>
</table>

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/G68.3/G68.4) or tool axis direction control command (G53.1).

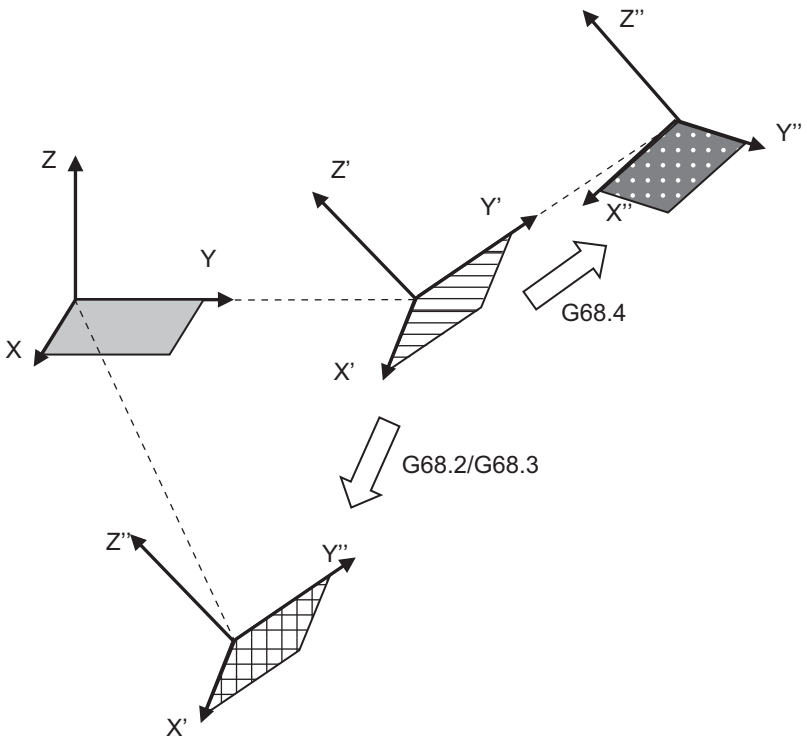
19.5.11 Inclined Surface Machining Multi-command

Function and purpose

A new coordinate system (feature coordinate system) can be defined by issuing the inclined surface machining commands repeatedly during the inclined surface machining.

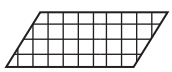
There are two types of inclined surface machining multi-commands: the absolute multi-command (G68.2/G68.3) that commands the workpiece coordinate system as the base point, and the incremental multi-command (G68.4) that commands the previous feature coordinate system as the base point.

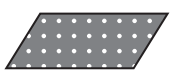
Command method	G code	Designation of feature coordinate system zero point	Designation of feature coordinate system rotation angle
Absolute multi-command	G68.2 G68.3	Designate the shift amount for the workpiece coordinate system.	Designate the rotation angle for the workpiece coordinate system.
Incremental multi-command	G68.4	Designate the shift amount for the previous feature coordinate system.	Designate the rotation angle for the previous feature coordinate system.




Workpiece coordinate system


Feature coordinate system defined with the first inclined surface machining command (G68.2/G68.3)


Feature coordinate system defined with the absolute multi-command (G68.2/G68.3)


Feature coordinate system newly defined with the incremental multi-command (G68.4)

Furthermore, the inclined surface machining multi-command can also be issued to the feature coordinate system defined with the inclined surface machining multi-command. There are no restrictions regarding the nesting level.

Command format

Inclined surface machining absolute multi-command

Follow the command format of the inclined surface machining command (G68.2/G68.3). (Refer to "Feature coordinate system command method" in "19.5 Inclined Surface Machining; G68.2, G68.3, G68.4/G69".)

Define a new coordinate system (feature coordinate system) for the X, Y, and Z axes of the currently set workpiece coordinate system.

Inclined surface machining incremental multi-command

Follow the command format of the inclined surface machining command (G68.2).

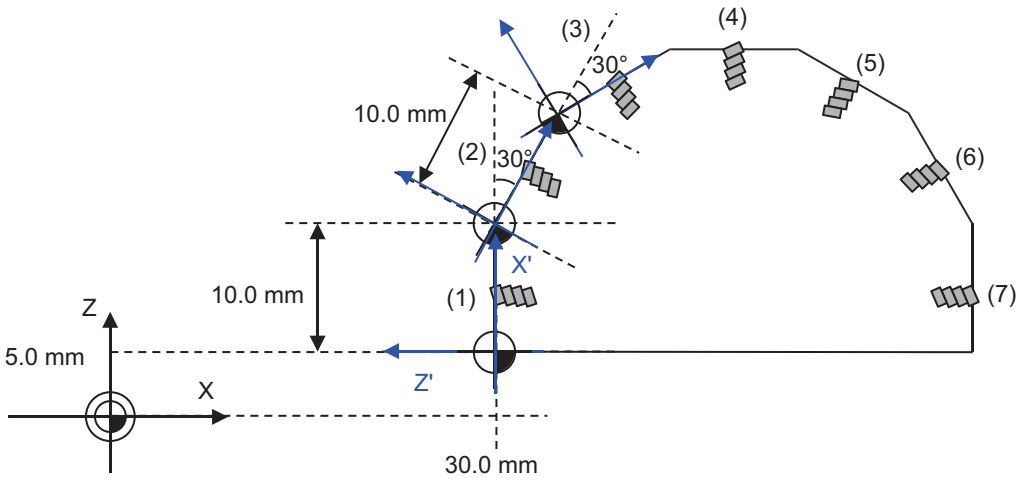
Define a new coordinate system (feature coordinate system) for the X, Y, and Z axes of the currently set feature coordinate system.

Command	Command method	Section
G68.4 [P0]	Define using Euler angles	19.5.1
G68.4 P1	Define using roll-pitch-yaw angles	19.5.2
G68.4 P2	Define using three points in a plane	19.5.3
G68.4 P3	Define using two vectors	19.5.4
G68.4 P4	Define using projection angles	19.5.5
G69	Inclined surface machining cancel	

- G68.4 and G69 belong to G code group 16 modal.
- If address P is omitted when G68.4 is commanded, it is assumed that G68.4 P0 is designated (define using Euler angles).
- If address P is not set to "0" to "4" when G68.4 is commanded, the program error (P954) occurs.
- If address P or Q includes a decimal point when G68.4 is commanded, it is rounded to an integer.
- G68.4 or G69 must be commanded alone in a block.
  When it is commanded in the same block with other G codes or travel commands, etc., the program error (P954) occurs.
- If G68.4 is commanded while the feature coordinate system is not set because G68.2/G68.3 has not been commanded, a new feature coordinate system is defined for the X, Y, and Z axes of the currently selected workpiece coordinate system.
- If G69 is commanded, the feature coordinate system is canceled, and the workpiece coordinate system is returned.

Example of use

When drilling holes on the machining surfaces (1) to (7) with the following regularity, the command can be simplified by the inclined surface machining incremental multi-command (G68.4).



Workpiece coordinate system Feature coordinate system

(1) to (7) Machining surface

[Main program]

```
 :
G90 G43 H1 G0 Z0.                 Tool length compensation
G68.2 P1 X30. Z5. J-90.           Inclined surface machining command
G53.6 P2                          Tool axis direction control
G00 X5. Z2.
G83 Z-3. Q2. F500.                Deep hole drilling cycle
G0 Z50.
M98 P100 L6                       Machining of machining surfaces (2) to (7)
 :
```

[Subprogram O100]

```
N101 G68.4 P1 X10. J30.           Inclined surface machining incremental multi-command
N102 G53.6 P2                     Tool axis direction control
N103 G00 X5. Z2.
N104 G83 Z-3. Q2. F500.           Deep hole drilling cycle
N105 G00 Z50.
M99
```

19.5.12 Relationship between Inclined Surface Machining and Other Functions



Relationship with other functions

Commands Available in Inclined Surface Machining Mode

If a command other than those listed below is issued during inclined surface machining (G68.2/G68.3/G68.4), the 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 - G30.6	Tool exchange position return
G31	Skip (*1)
G31.1 - 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	Mirror image by G code 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
G68.2, G68.3	Inclined surface machining command
G68.4	Inclined surface machining incremental multiple command
G69	Coordinate rotation cancel, Inclined surface machining cancel
G70 - G76, G80 - G89	Fixed cycle for drilling (Including synchronous tapping)
G90, G91	Absolute command, Incremental command

Command	Function
G93	Inverse time feed
G94	Feed per minute
G95	Feed per revolution
G98, G99	Fixed cycle initial level return, R point level return
G127	Prohibits the reverse run in all part systems
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/G68.3/G68.4) 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
G51.2 (G251)	Tool spindle synchronization IB/IC
G50.2 (G250)	Tool spindle synchronization IB/IC cancel
G54 - 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
G68.2, G68.3	Inclined surface machining command
G68.4	Inclined surface machining incremental multiple command
G69	Coordinate rotation cancel, 3-dimensional coordinate conversion cancel
G80	Fixed cycle cancel
G90, G91	Absolute command, Incremental 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)
G114.2	Tool spindle synchronization IA
G113.1	Spindle synchronization cancel (*1), Tool spindle synchronization IA cancel
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

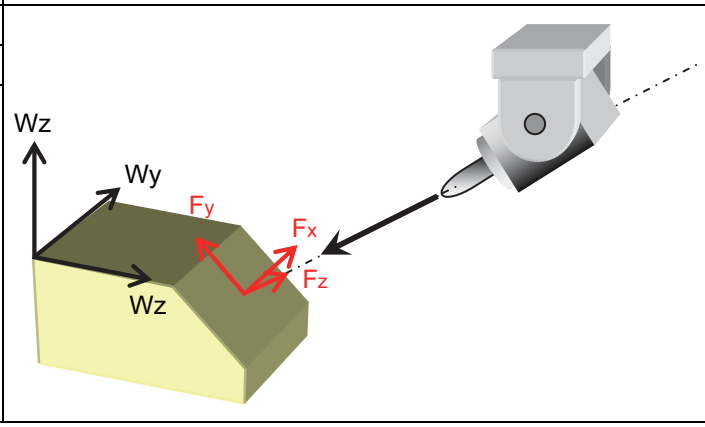
If tool axis direction control is commanded in a mode other than those listed below, a program error (P953) occurs.

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
G51.2 (G251)	Tool spindle synchronization IB/IC
G50.2 (G250)	Tool spindle synchronization IB/IC cancel
G54 - 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 - G68.9	In inclined surface machining
G80	Fixed cycle cancel
G90, G91	Absolute command, Incremental 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)
G114.2	Tool spindle synchronization IA
G113.1	Spindle synchronization cancel (*1), Tool spindle synchronization IA cancel
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 Ii 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.

19.5.13 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.

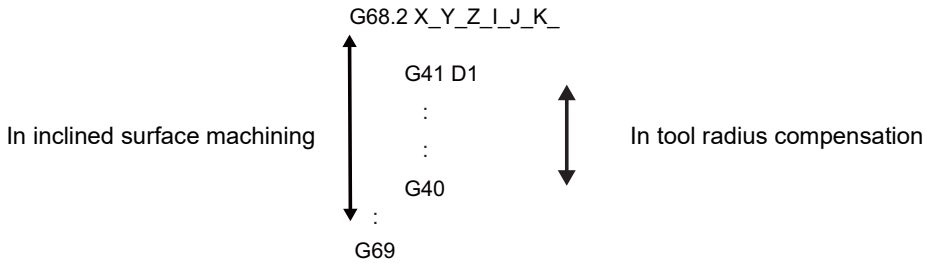
n: Varies depending on the number of control axes.

(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 rstint" or "#1210 RstGmd/bitF" is set to "0", however, the inclined surface machining mode is kept even when Reset is input.)

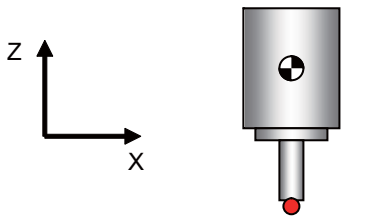
(4) When the external deceleration signal is input, the signal is not input to the axes of the feature coordinate system, but to the axes of the machine coordinate system that is actually operating.

(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.

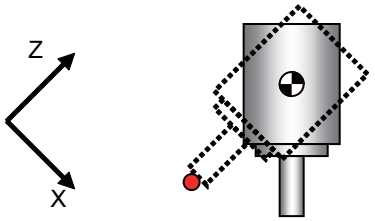
(6) Tool radius compensation, mirror image by G code, fixed cycle command, tool center point control, scaling, and tool length compensation along the tool axis should be nested in the inclined surface machining command. Thus, these commands need to be commanded between the inclined surface machining command (G68.2, etc.) and G69.



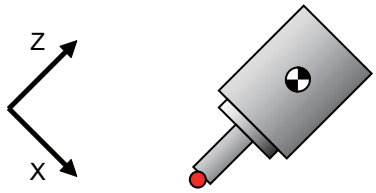
(7) If inclined surface machining (G68.2/G68.4) is commanded while tool length compensation is active, the actual tool tip position does not match the current position. In such case, command G53.1 to align the tool axis direction with the Z axis of the feature coordinate system, which will make the tool 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 and PLC 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.

(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;
      :
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 N10 or N11 is possible.

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) When the high-accuracy control is executed, the acceleration at rapid traverse (G00) during the inclined surface machining command is the same as that at cutting feed (G01) (*1).

(*1) Acceleration rate determined by "#1206 G1bF" (maximum speed) and "#1207 G1btL" (time constant).

(37) 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.

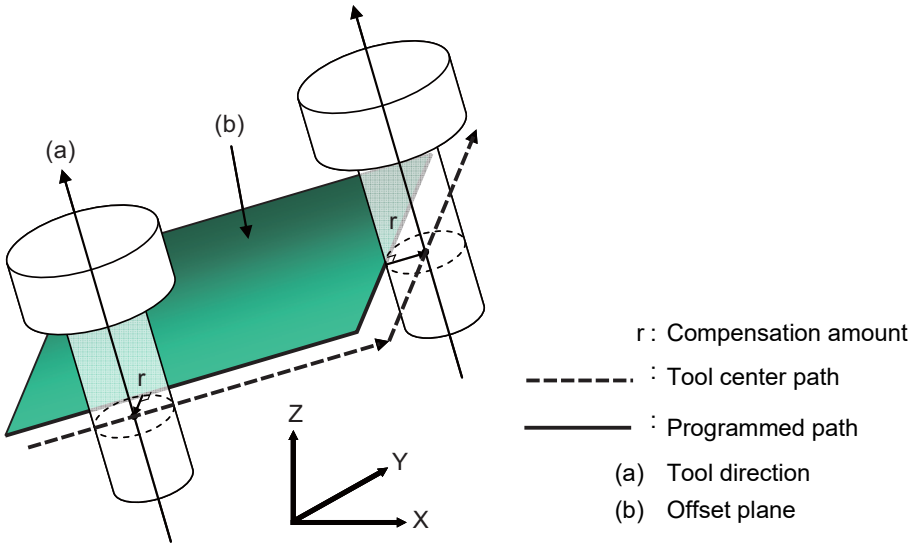
(38) The inclined surface machining command cannot be commanded during the multiple-axis synchronization control. The program error (P740) occurs.

(39) The acceleration/deceleration method differs depending on the parameter settings (depending on the MTB specifications). Refer to "Relationship with other functions" in "7.8 Rapid Traverse Constant-gradient Acceleration/Deceleration".

19.6 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.

The applicable machine configurations depend on the MTB specifications.
For the applicable machine configurations, refer to "Appendix 1: Functions with Advanced Machining" of "Specifications Manual (Function)".

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.

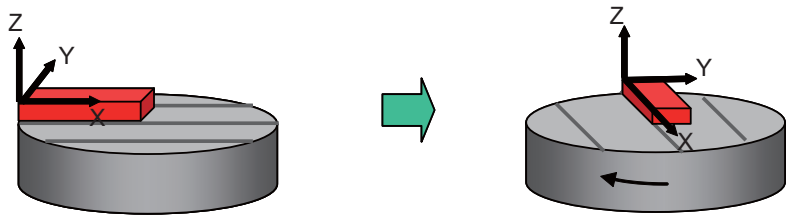
How to calculate the compensation vector

The compensation vectors for tool radius compensation are obtained as shown below.

(1) Table coordinate system

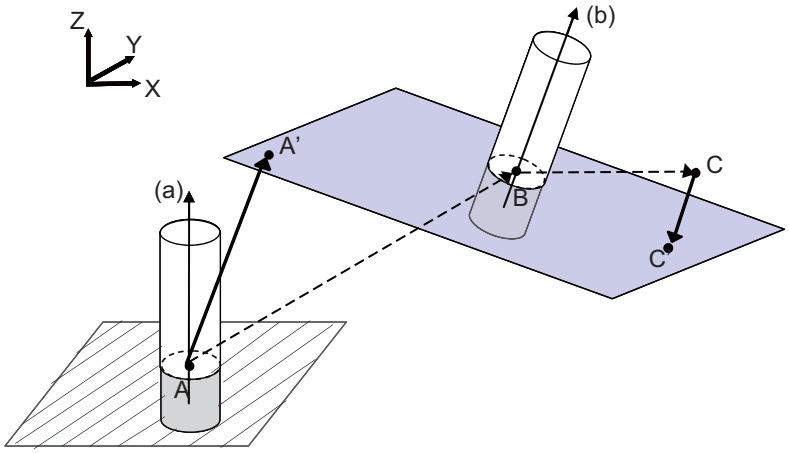
Convert the program command into the path on the surface of the table coordinate system. Table coordinate system rotates with the workpiece (the figure below) as the table rotates. The command path on this coordinate system is the relative command path of the tool against the workpiece.

<Default state> **<When the table rotates>**



(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

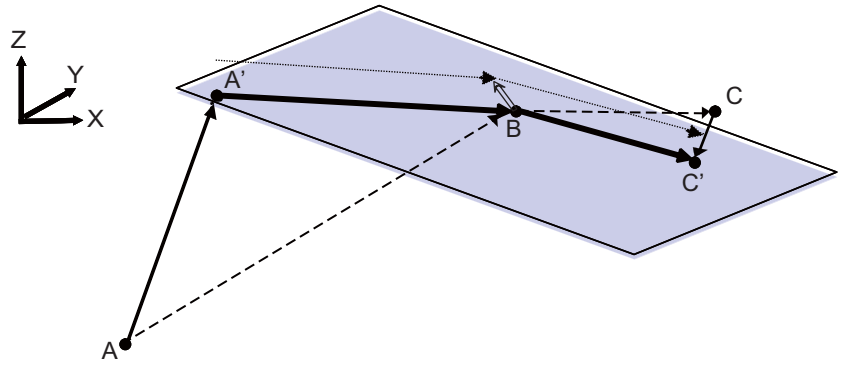
Offset plane at point A

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.

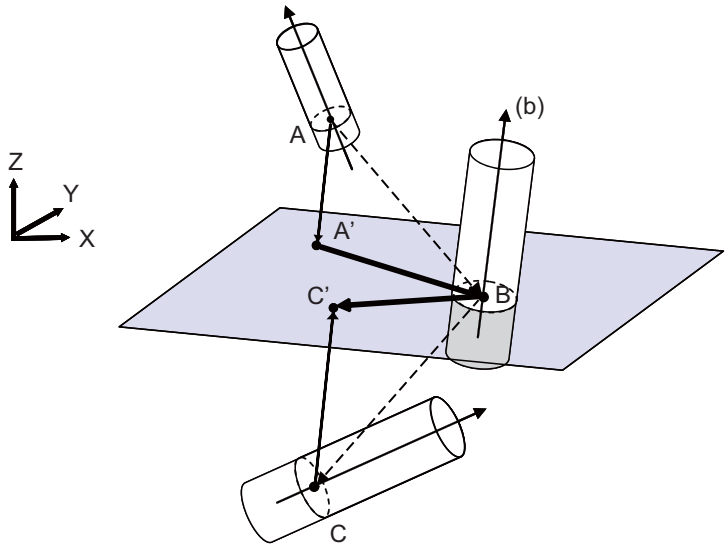


Path on the table coordinate system on the offset plane

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.)

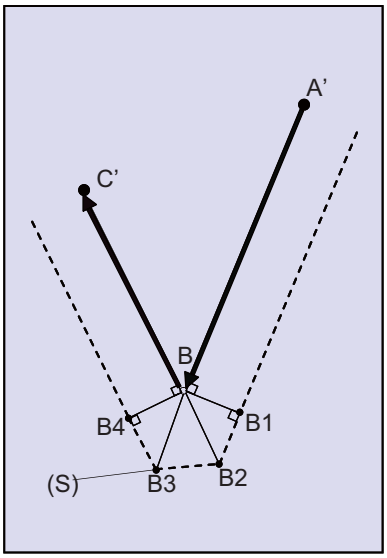


(b) Tool direction at point B

Path on the table coordinate system

Offset plane at point B

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.



Path on the table coordinate system on the offset plane

(S) Single block stop point

Tool offset command check

It is possible to make a program error occur when the tool offset No. is "0" or when the offset amount specified with the tool offset No. is "0", to prevent commands from being forgotten.

Note

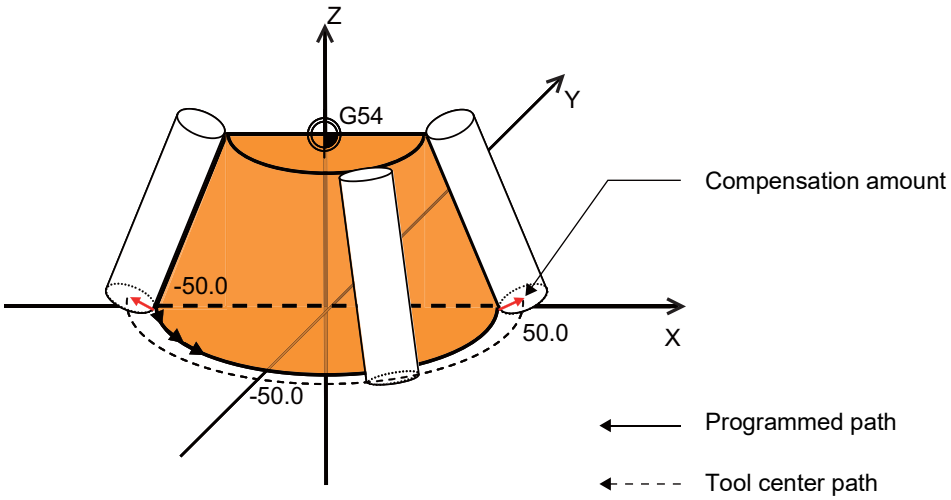
(1) The D address of circle cutting (G12/G13) is not checked by the tool offset command check.

Refer to "12.7.2 Tool Offset Command Check" for details.

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 command
G91	Incremental 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
G50.2 (G250)	Tool spindle synchronization IB/IC cancel (*3)
G51.2 (G251)	Tool spindle synchronization IB/IC (*3)
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 command, Incremental command
G93	Inverse time feed
G94	Feed per minute
G95	Feed per revolution
G96, G97	Constant surface speed control ON/OFF
G113.1	Spindle synchronization I (*2), Tool spindle synchronization IA/Tool spindle synchronization II cancel
G114.1	Spindle synchronization I (*2)
G114.2	Tool spindle synchronization IA
G114.3	Tool Spindle synchronization II
M98, M99	Subprogram call, main program return
F	Feedrate command

	Function
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-)
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.

(*2) The 3-dimensional tool radius compensation (tool's vertical-direction compensation) can be commanded using the C axis in spindle synchronization.

(*3) Do not designate the rotary tool axis in the rotary axis configuration parameter. If tool spindle synchronization IC is commanded during 3-dimensional tool radius compensation (tool's vertical-direction compensation), it may cause an unexpected operation.

Modes in which G41.2/G42.2 command can be issued

	Function
G00, G01	Positioning, Linear interpolation
G13.1, G113	Polar coordinate interpolation cancel
G15	Polar coordinate command cancel
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
G50.2 (G250)	Tool spindle synchronization IB/IC cancel (*2)
G51.2 (G251)	Tool spindle synchronization IB/IC (*2)
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 command, Incremental command
G93	Inverse time feed
G94	Feed per minute
G95	Feed per revolution
G96, G97	Constant surface speed control ON/OFF

	Function
G98, G99	Fixed cycle initial level return, R point level return
G113.1	Spindle synchronization I (*1), Tool spindle synchronization IA/Tool spindle synchronization II cancel
G114.1	Spindle synchronization I (*1)
G114.2	Tool spindle synchronization IA
G114.3	Tool Spindle Synchronization II

(*1) The 3-dimensional tool radius compensation (tool's vertical-direction compensation) can be commanded using the C axis in spindle synchronization.

(*2) Do not designate the rotary tool axis in the rotary axis configuration parameter. If tool spindle synchronization IC is commanded during 3-dimensional tool radius compensation (tool's vertical-direction compensation), it may cause an unexpected operation.

Combination with arbitrary axis exchange control

(1) 3-dimensional tool radius compensation (Tool's vertical-direction compensation) in arbitrary axis exchange

When the 3-dimensional tool radius compensation (tool's vertical-direction compensation) is commanded after axis exchange, if the axis status matches the axis configuration of the rotary axis configuration parameter (#7900 and after), the 3-dimensional tool radius compensation (tool's vertical-direction compensation) is enabled. However, after axis exchange, the I axis, J axis, and K axis configurations must match the horizontal axis (#7900), vertical axis (#7901), and height axis (#7902) configurations designated in the rotary axis configuration parameter.

By performing the arbitrary axis exchange control to obtain the axis configuration designated in the rotary axis configuration parameter, the NC selects the rotary axis configuration parameter of which the axis status after axis exchange matches, and performs the 3-dimensional tool radius compensation (tool's vertical-direction compensation).

(2) Restrictions for combination with arbitrary axis exchange control

(a) If the arbitrary axis exchange (G140, G141, or G142) command is issued during 3-dimensional tool radius compensation (tool's vertical-direction compensation) modal, a program error (P169) will occur.

(b) If an axis in the part system in 3-dimensional tool radius compensation (tool's vertical-direction compensation) is exchanged by another part system, an operation error (M01 1101) will occur.

(c) If the 3-dimensional tool radius compensation (tool's vertical-direction compensation) command is issued while the part system does not contain any axis that is designated in the rotary axis configuration parameters (#7900 and after), a program error (P934) occurs.

(d) However, while all the axes designated in the rotary axis configuration parameter exist in the part system with the G140 commanded, if the I axis, J axis, and K axis configurations after axis exchange do not match the horizontal axis (#7900), vertical axis (#7901), and height axis (#7902) configurations designated in the rotary axis configuration parameter, a program error (P934) occurs.

(3) Axis operation after axis exchange by combination of reset setting (modal retention, cross retention)

The part system in the 3-dimensional tool radius compensation (tool's vertical-direction compensation) does not return the axis status regardless of the mixed control cancel setting ("#1280 ext16/bit1" = "0") even if the reset command with no modal canceled (parameter "#1151 rstint" = "0") and NC reset 1 are executed.

(4) Graphic check operation with arbitrary axis exchange combined

If the 3-dimensional tool radius compensation command (tool's vertical-direction compensation) after arbitrary axis exchange is to be drawn with the graphic check, the graphic check is performed while axis exchange is not performed by ignoring the arbitrary axis exchange command.

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).

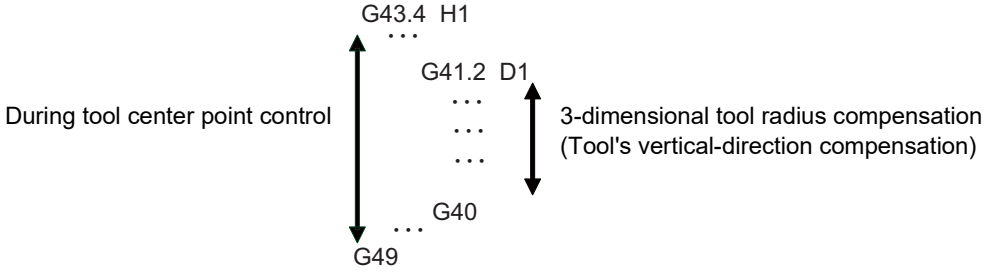
(8) Turning ON the "PLC interruption" signal in 3-dimensional tool radius compensation (tool's vertical-direction compensation) will cause an operation error (M01 0232).

(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.

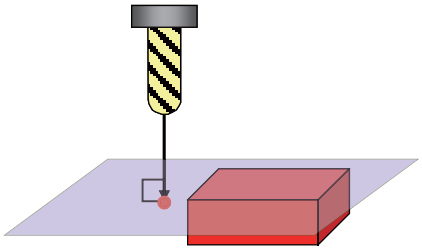


(12) When used with the tool center point control, the compensation is applied to the tool center point path.

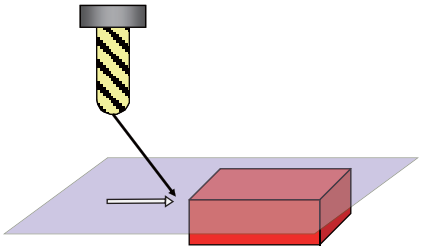
(13) The restart search from the block in 3-dimensional tool radius compensation (tool's vertical-direction compensation) is possible while the restart search from the block concurrently using the tool center point control is impossible.

(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) If the 3-dimensional tool radius compensation (tool's vertical-direction compensation) is commanded in spindle mode when the axis set to the spindle/C axis is used as a rotary axis of 3-dimensional tool radius compensation (tool's vertical-direction compensation), a program error (P934) occurs.
If the mode is changed to the spindle mode during the 3-dimensional tool radius compensation (tool's vertical-direction compensation), an operation error (M01 0186) occurs.
When the 3-dimensional tool radius compensation (tool's vertical-direction compensation) modal is retained by resetting the modal retention (#1151 rstint = 0), an operation error (M01 0186) occurs at the time of the next automatic operation start.
To cancel the 3-dimensional tool radius compensation (tool's vertical-direction compensation) modal, command the 3-dimensional tool radius compensation (tool's vertical-direction compensation) cancel (G40) to the head (first block) of the machining program.

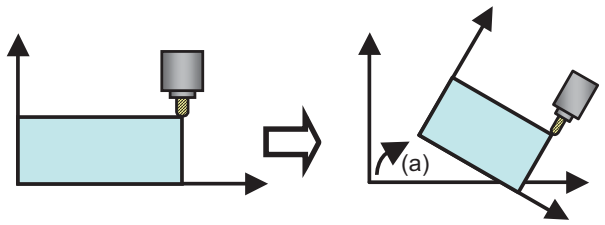
(19) The 3-dimensional tool radius compensation cannot be commanded during the multiple-axis synchronization control. The program error (P740) occurs.

(20) While the program format is switched, the 3-dimensional tool radius compensation (tool's vertical-direction compensation) cannot be performed by using the H axis (axis where the parameter "#1013 axname" is set to "H"). According to the setting of the parameter (#1450 5axis_Spec/bit0), the operation error (M01 0127) occurs when the power is turned ON, or the program error (P932 or P934) occurs when G41.2/G42.2 is commanded. (This depends on the MTB specifications.)

19.7 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.

The applicable machine configurations depend on the MTB specifications.
For the applicable machine configurations, refer to "Appendix 1: Functions with Advanced Machining" of "Specifications Manual (Function)".

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

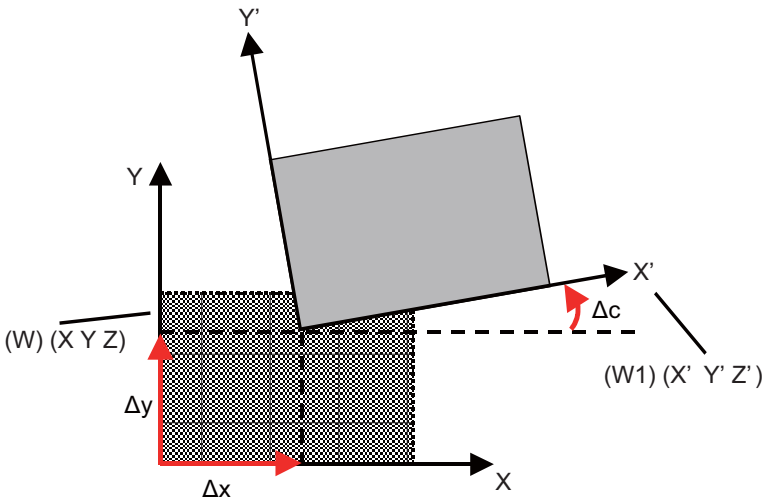
(1) G54.4 must be commanded alone in a block. When it is commanded in the same block with other G codes, travel commands, or miscellaneous commands, a program error (P33) occurs.

(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 “23.27 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.

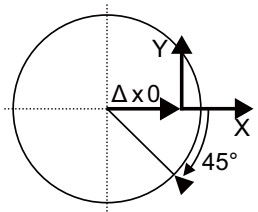
Common (A)

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

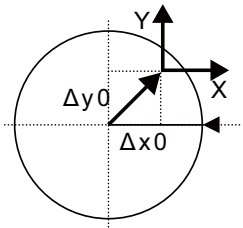
Error at C=45° of common



Δx0	10.0
Δy0	0.0
Δz0	0.0
Δa0	
Δb0	
Δc0	

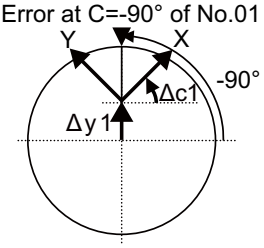
(a)

Error at C=0° of common



Δx0	7.071
Δy0	7.071
Δz0	0.0
Δa0	
Δb0	
Δc0	

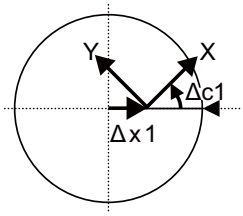
Error at C=-90° of No.01



Δx1	0.0
Δy1	5.0
Δz1	0.0
Δa1	0.0
Δb1	0.0
Δc1	45.0

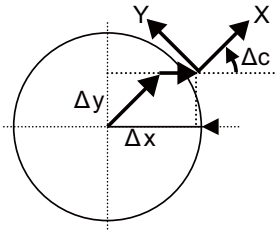
(c)

Error at C=0° of No.01



Δx1	5.0
Δy1	0.0
Δz1	0.0
Δa1	0.0
Δb1	0.0
Δc1	45.0

(b)



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.

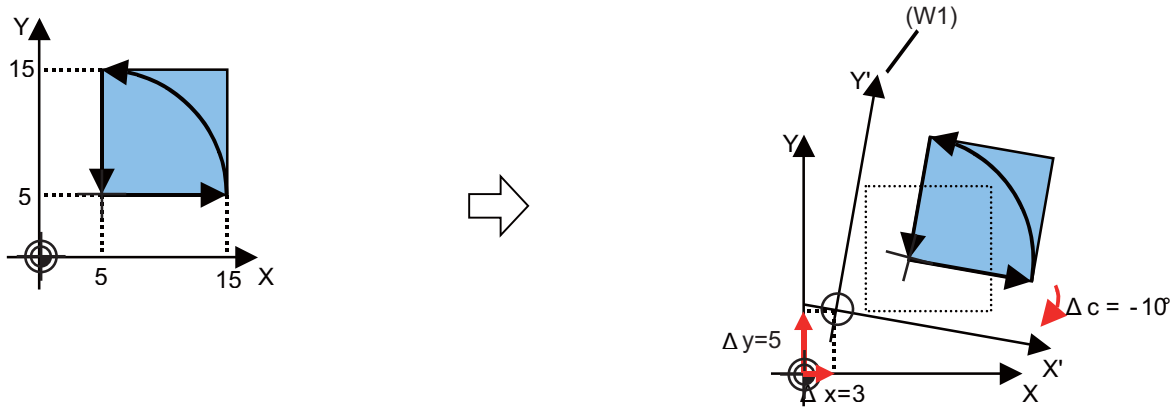
Details of operation

(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.

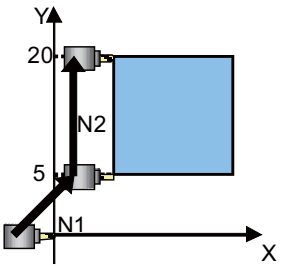
Program (A)	Workpiece installation error compensation No.01	G54.4P1
G90 G0 X5. Y5. G1 X15. Y5. F500 G3 X5. Y15. R10. F500 G1 X5. Y5. F500	Δx = 3. Δy = 5. Δz = 0. Δa = 0. Δb = 0. Δc = -10. Rotary axis position 1 = 0° Rotary axis position 2 = 0°	G90 G0 X5. Y5. G1 X15.Y5. F500 G3 X5. Y15. R10. F500 G1 X5. Y5. F500 G54.4P0



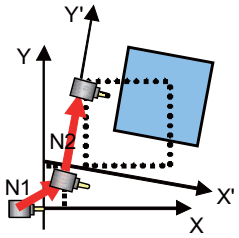
(W1) Workpiece installation coordinate system

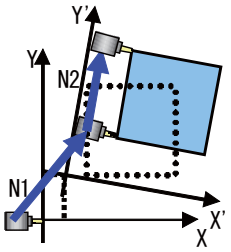
[Precautions in commanding and canceling G54.4]

In the first travel command after commanding G54.4, make sure to issue an absolute command. In the G54.4 command block, the machine does not make any movement. Therefore, if an incremental command is issued immediately after having commanded G54.4, the operation may result in what is not intended by the program. (Once an absolute command is issued, the operations of blocks thereafter will be as intended by the program even with incremental commands.)
This precaution also applies to a cancel operation, therefore make sure to issue an absolute command immediately after commanding G54.4 P0.

<table>
<tr><td>Operation with no installation error</td><td colspan="2">If an incremental command is issued immediately after having commanded G54.4</td></tr>
<tr><td rowspan="3">

(Incremental command)
N1 G91 G0 X5. Y5.
N2 G91 G1 Y15.

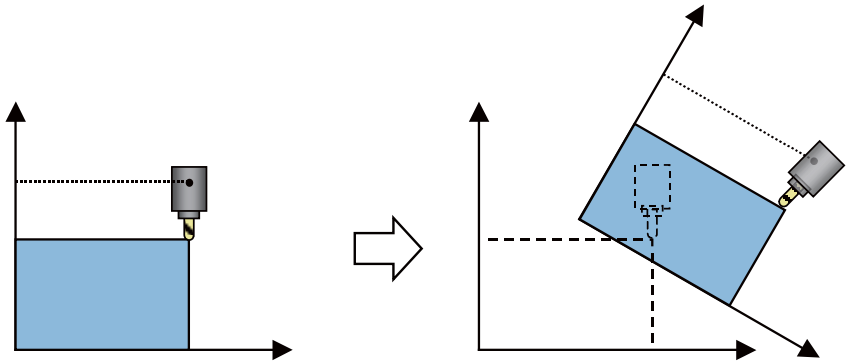
(Absolute command)
N1 G90 G0 X5. Y5.
N2 G90 G1 Y20.</td><td>

G54.4 P1
N1 G91 G0 X5. Y5.
N2 G91 G1 Y15. F500</td><td>If an incremental position 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.</td></tr>
<tr><td colspan="2">If an absolute command is issued immediately after having commanded G54.4</td></tr>
<tr><td>

G54.4 P1
N1 G90 G0 X5. Y5.
N2 G90 G1 Y20. F500</td><td>If an absolute position is commanded in the next block of G54.4, the operation results in what is intended by the program.</td></tr>
</table>

(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.

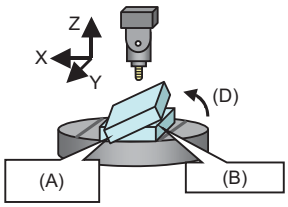


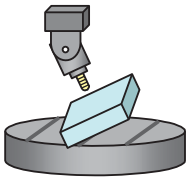
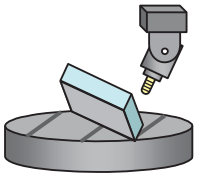
Workpiece installation error compensation ON

In addition to XYZ, the rotary axis position is also compensated.

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)



	Position after tool direction compensation	
	Sign of B=Positive	Sign of B=Negative
		
(Program command)	(Program command)	(Program command)
B = 0°	B = 30°	B = -30°
C = 0°	C = 0°	C = 180°

(A) Actual workpiece installation position	(Current position)	(Machine position)	(Current position)	(Machine position)
(B) Ideal position	B 0.	B 30.	B 0.	B -30.
(D) Error of Y axis angle Δb = 30°	C 0.	C 0.	C 0.	C 180.

<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
G17, G18, G19	Plane selection
G20, G21	Inch/metric command (*1)
G28	Automatic 1st reference position return
G29	Start position return
G30	2nd to 4th reference position return
G30.1 - G30.6	Tool exchange position return
G31	Skip (*2)
G31.1 - 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 - 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 - G76, G80 - G89	Fixed cycle for drilling Fixed cycle for drilling (Including synchronous tapping)
G90, G91	Absolute command, Incremental command
G93	Inverse time feed
G94	Feed per minute

Command	Function
G95	Feed per revolution
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 - 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 command, Incremental 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.

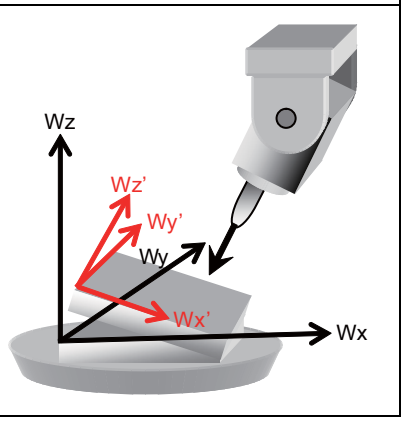
Parameters and PLC signal status to which the workpiece installation error compensation command can not be issued

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 - #8627	-
Mirror image by parameter setting	#8211	-
Mirror image by external input	-	Y7C0 - 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".

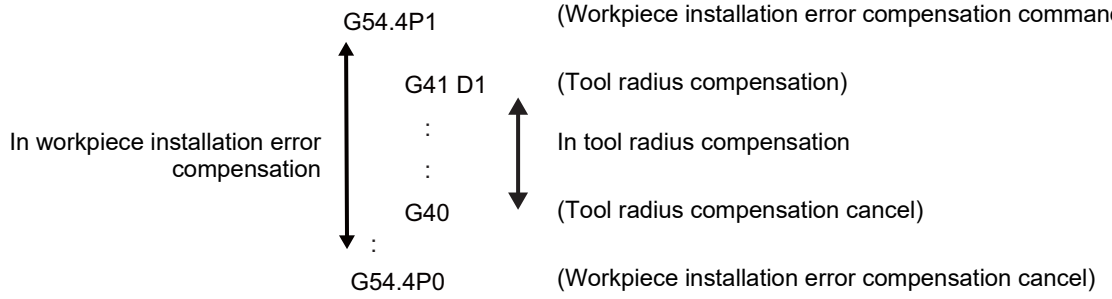
Precautions

(1) During workpiece installation error compensation command, the coordinates on the workpiece installation coordinates 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 or PLC interruption during workpiece installation error compensation. If any of these is executed during workpiece installation error compensation, an operation error (M01 0170) occurs.

(7) When workpiece installation error compensation is commanded during MDI interruption or PLC interruption, a program error (P546) occurs.

(8) In graphic check, the path of the original workpiece coordinate system is displayed. In 2D check and 3D check (check method I), circular interpolation is drawn by straight lines.

(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 rate 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 rate for rapid traverse (G0) in high-accuracy control).

- When "#1250 set22/bit3" = "0"
  Same as the acceleration rate for cutting feed (G01) (the acceleration rate determined with "#1206 G1bF" (Maximum speed) and "#1207 G1btL" (Time constant))
- When "#1250 set22/bit3" = "1"
  Same as the acceleration rate for rapid traverse (G00) (the acceleration rate 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.

(29) The workpiece installation error compensation cannot be commanded during the multiple-axis synchronization control. The program error (740) occurs.

(30) While the program format is switched, the workpiece installation error compensation cannot be performed by using the H axis (axis where the parameter "#1013 axname" is set to "H"). According to the setting of the parameter (#1450 5axis_Spec/bit0), the operation error (M01 0127) occurs when the power is turned ON, or the program error (P932 or P934) occurs when G54.4 is commanded. (This depends on the MTB specifications.)

(31) The acceleration/deceleration method differs depending on the parameter settings (depending on the MTB specifications). Refer to "Relationship with other functions" in "7.8 Rapid Traverse Constant-gradient Acceleration/Deceleration".

(32) When a macro interruption is executed during the workpiece installation error compensation, disable the "reverse run control mode" signal (RVMD). If a macro interruption is executed while the workpiece installation error compensation is activated and the "reverse run control mode" (RVMD) is enabled, the operation error (M01 0070) occurs. This error can be cleared by resetting.

20

Coordinate System Setting Functions

20.1 Coordinate Words and Control Axes

Function and purpose

The NC axis, spindle, PLC axis and auxiliary axis are generically called the control axis.

NC axis	The axis is used to perform automatic operation by coordinate value command from machining program and to perform manual operation such as jog feed, handle feed etc. X, Y, Z, U, V, W, A, B and C can be used as the axis name. An axis where no motor is connected (hypothetical axis) can be set.
Spindle	The axis is used to rotate the part where the tool/workpiece is attached. The rotation is commanded by rotation speed and spindle forward run signal/spindle reverse run signal.
PLC axis	The axis is used to command the position and speed from user PLC. Two or more axes are not used together for interpolation; instead, each axis is used individually. Manual operation such as jog feed, handle feed, etc. can be performed. An axis where no motor is connected (hypothetical axis) can be set.

The following functions can be added depending on the parameter setting.

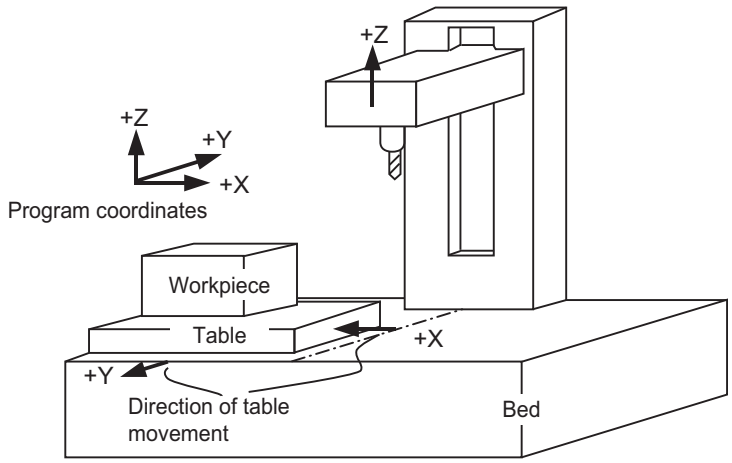
- Spindle position control (Spindle/C axis control)
  A spindle can be controlled as the rotary axis. A coordinate value (angle) can be commanded from machining program in the same way as NC axis.
- Spindle-mode rotary axis control
  NC axis can be controlled by the command of rotation speed and spindle forward run signal/spindle reverse run signal in the same way as spindle.

Note

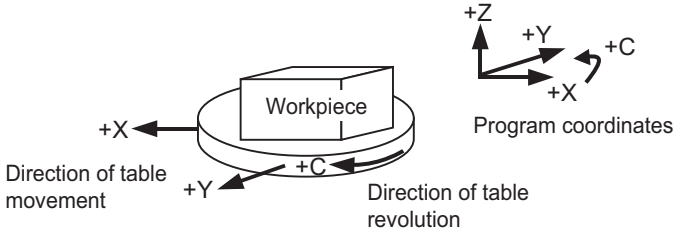
(1) The number of control axes that can be controlled by one CNC depends on the model.

To specify each machining direction, use alphabetical coordinate words that are predefined appropriately.

X-Y table



X axis, Y axis and rotating table

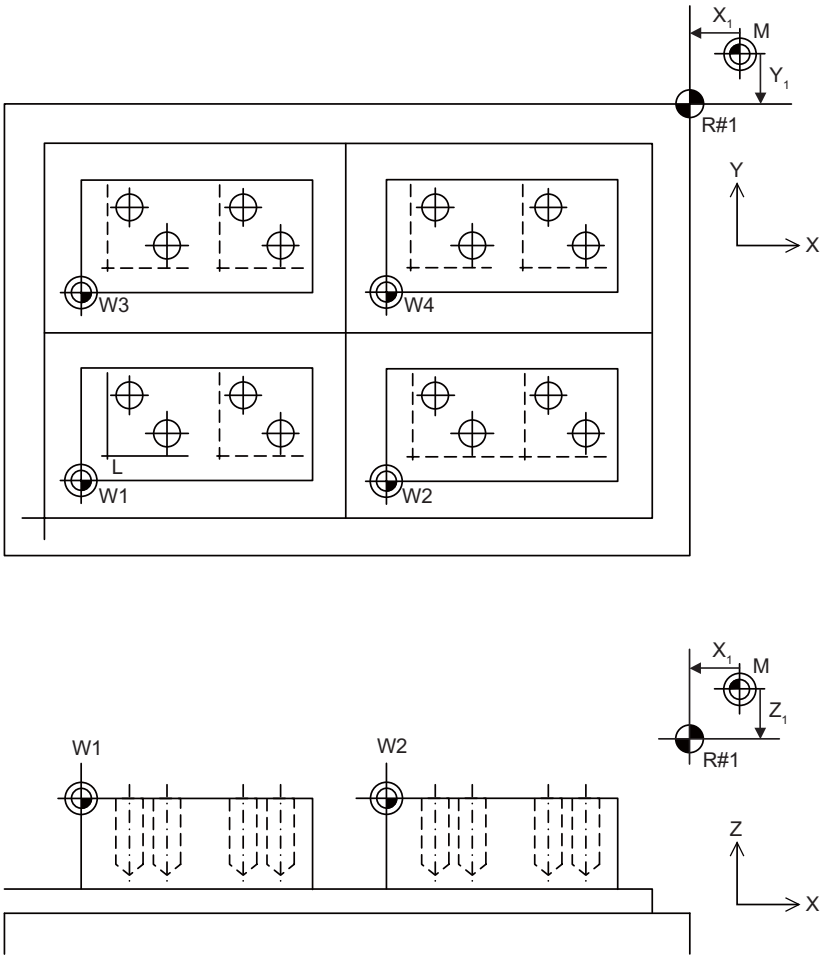


20.2 Types of Coordinate Systems

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).



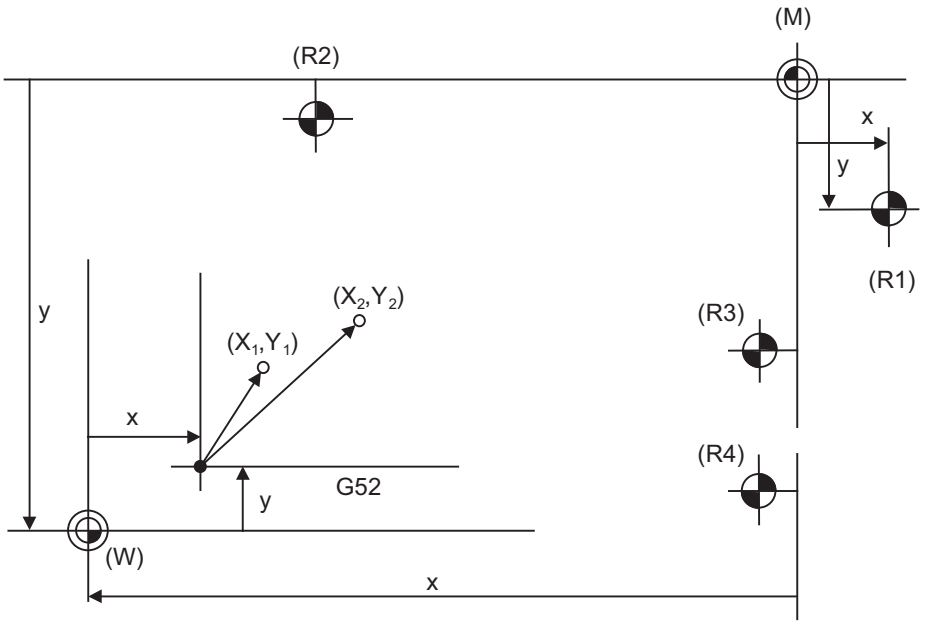
M: Basic machine coordinate system
W: Workpiece coordinate system
L: Local coordinate system

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.

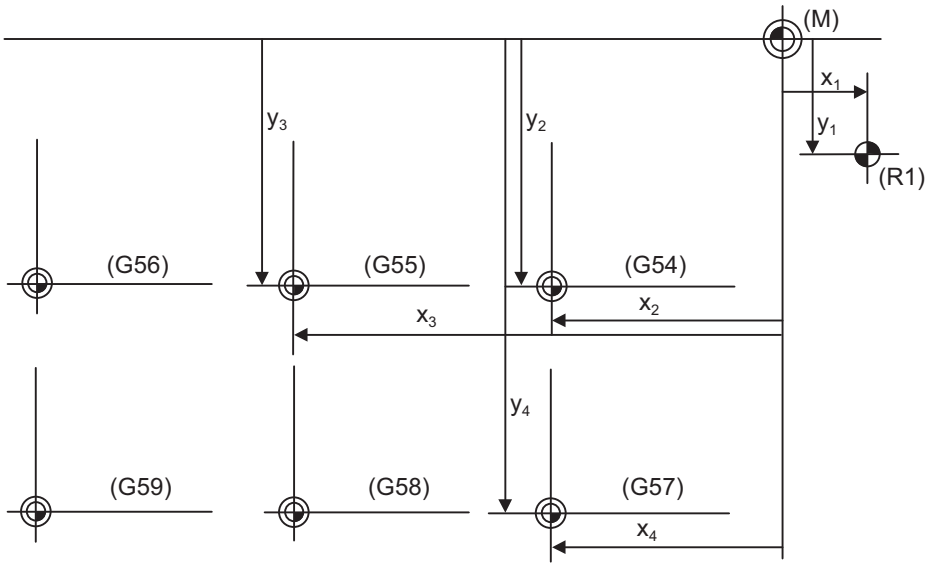


(M) Basic machine coordinate system
(G52) Local coordinate system
(W) Workpiece coordinate systems (G54 to G59)
(R1) 1st reference position
(R2) 2nd reference position
(R3) 3rd reference position
(R4) 4th reference position

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
(R1) 1st reference position
(G54) Workpiece coordinate system 1
(G55) Workpiece coordinate system 2
(G56) Workpiece coordinate system 3
(G57) Workpiece coordinate system 4
(G58) Workpiece coordinate system 5
(G59) Workpiece coordinate system 6

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 where the parameter "#1017 rot" is set to "1" is controlled with the rotary axis' coordinate system, as a rotary axis. (This depends on the MTB specifications.)
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 -99999.999° to +99999.999° for the linear type.
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.
The machine coordinate value and relative position differ according to the parameters.

<table>
<tr><th rowspan="3"></th><th colspan="4">Rotary axis</th><th rowspan="3">Linear axis</th></tr>
<tr><th colspan="2">Rotating type rotary axis</th><th colspan="2">Linear type rotary axis</th></tr>
<tr><th>Short-cut invalid</th><th>Short-cut valid</th><th>Workpiece coordinate position Linear type</th><th>All-coordinate position Linear type</th></tr>
<tr><td>#1017 setting value</td><td colspan="4">1</td><td>0</td></tr>
<tr><td>#8213 setting value</td><td>0</td><td>1</td><td>2</td><td>3</td><td>-</td></tr>
<tr><td>Workpiece coordinate position</td><td colspan="2">Displayed in the range of 0° to 359.999°.</td><td colspan="3">Displayed in the range of -99999.999° to +99999.999°.</td></tr>
<tr><td>Machine coordinate position/ Current position</td><td colspan="3">Displayed in the range of 0° to 359.999°.</td><td colspan="2">Displayed in the range of -99999.999° to +99999.999°.</td></tr>
<tr><td>Absolute command</td><td>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.</td><td>Moves with a short-cut to the end point.</td><td colspan="3">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).</td></tr>
<tr><td>Incremental command</td><td colspan="5">Moves in the direction of the commanded sign by the commanded incremental amount starting at the current position.</td></tr>
<tr><td rowspan="2">Reference position return</td><td colspan="5">Depends on the absolute command or the incremental command during the movement to the intermediate point.</td></tr>
<tr><td colspan="3">Returns with movement within 360 degrees.</td><td colspan="2">Moves and returns in the R point direction for the difference from the current position to the R point.</td></tr>
</table>

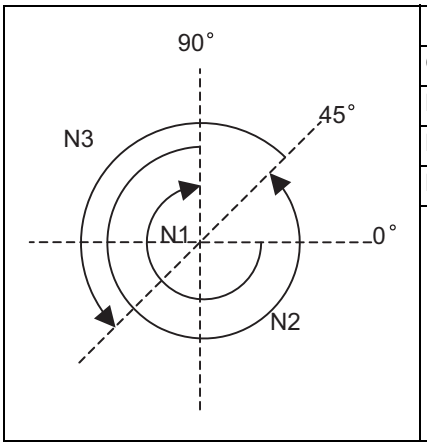
Operation example

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 current position are displayed in the range of 0° to 359.999°.

(2) For the absolute command, the axis moves according to the sign by the remainder amount obtained by dividing by 360°.

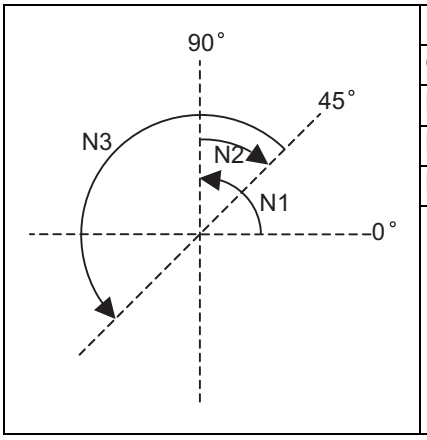


Program	Workpiece	Machine
G28 C0. ;		
N1 G90 C-270. ;	90.000	90.000
N2 C405. ;	45.000	45.000
N3 G91 C180. ;	225.000	225.000

Rotary type (short-cut valid)

(1) The machine coordinate position, workpiece coordinate position and current position are displayed in the range of 0° to 359.999°.

(2) For the absolute command, the axis rotates to the direction having less amount of movement to the end point.



Program	Workpiece	Machine
G28 C0. ;		
N1 G90 C-270. ;	90.000	90.000
N2 C405. ;	45.000	45.000
N3 G91 C180. ;	225.000	225.000

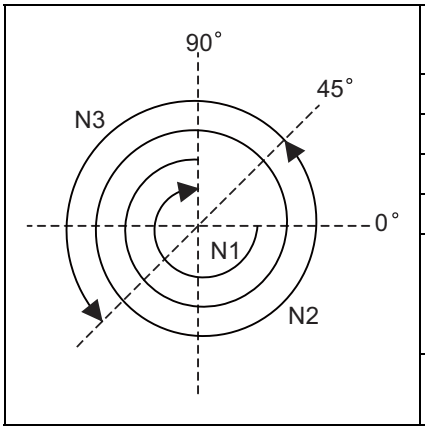
Linear type (workpiece coordinate position linear type)

(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 -99999.999° 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	Current 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	

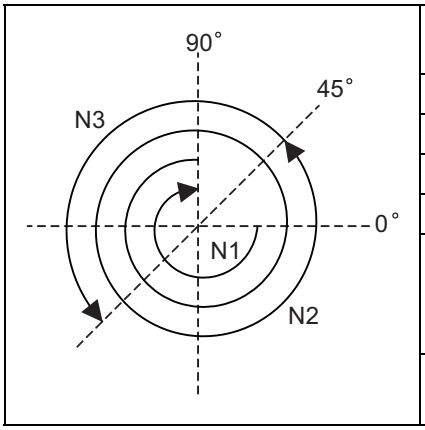
Linear type (all coordinate position linear type)

(1) The workpiece coordinate position counter is displayed in the range of -99999.999° 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	Current 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 indicates the position (tool exchange 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 turned ON, the basic machine coordinate system is automatically set as referenced to the reference position (zero point) return position, which is determined by the automatic or manual reference position (zero point) return.

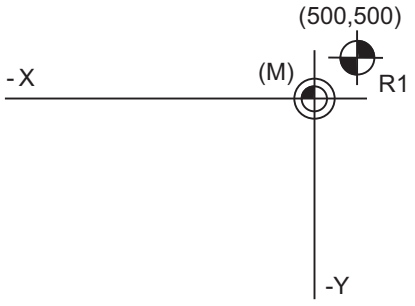
(2) The basic machine coordinate system is not changed by the G92 command.

(3) The G53 command is valid only in the designated block.

(4) In the incremental command mode (G91), the G53 command provides movement with the incremental position 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.



(M) Basic machine coordinate system
(R1) 1st reference position

1st reference position coordinate value: X=+500 and Y=+500

(7) The G53 command will move with cutting feed 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.

(9) 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) Nose R compensation
When the compensation direction is reversed during nose R compensation, a program error (P157) will occur except when it is commanded in G00 block. Even if directions differ between before and after the G53 block, an error will not occur because compensation is temporarily canceled. A parameter can be set to move the tool in the same compensation direction.

(3) Machine coordinate system selection, Feedrate designation
If an ",F" command is issued when there are no specifications of the feedrate command for G53, a program error (P39) occurs.

(4) Inclined surface machining
When the G53 command is issued during inclined surface machining, a program error (P951) occurs.

(5) Polar coordinate interpolation
Do not issue the G53 command during the polar coordinate interpolation mode.

(6) Polar coordinate command
The axis command with the G53 command is not interpreted as the polar coordinate command during the polar coordinate command mode.

(7) Tool length compensation along the tool axis
When the G53 command is issued while the compensation status is still established, the compensation is temporarily canceled, and the tool moves to the machine position designated by G53.

(8) Mirror image by G code
The mirror image will not be applied to the G53 command.

(9) High-speed high-accuracy control
If the G53 command is issued during the high-speed high-accuracy control II mode, a program error occurs.

(10) 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.

(11) Tool center point control
If the G53 command is issued during the tool center point modal, a program error (P942) occurs.

(12) Workpiece installation error compensation
If the G53 command is issued during workpiece installation error compensation, a program error (P545) occurs.



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. Therefore, the update of the coordinate counter such as the "next command" or "remaining command" may not follow the actual movement.

20.4 Coordinate System Setting; G92

Function and purpose

The zero point position of the program coordinate system can be shifted to an arbitrary position by G92 command.

Command format

G92 X__ Y__ Z__ α__ ;

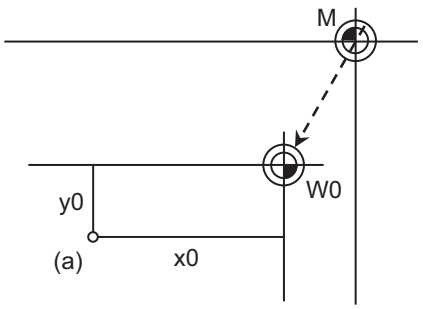
α	Additional axis

Detailed description

As shown in the figure below, when a coordinate system setting is specified by G92, G92 shift amount is applied so that the tool position in the current workpiece coordinate system (*1) is set to the coordinate position specified by G92, and the workpiece coordinate system is shifted accordingly. All the workpiece coordinate systems (G54 to G59 and G54.1 Pn) are also shifted simultaneously at this time, but the machine does not move.

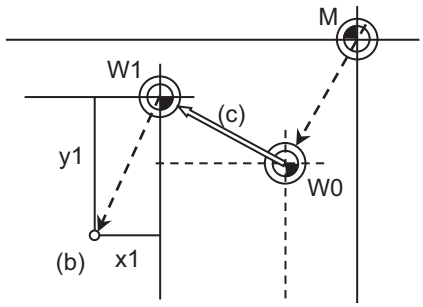
(*1) The "tool position in the current workpiece coordinate system" includes the compensation amounts for tool radius, tool length and tool position compensation.

Coordinate system created by automatic coordinate system setting



G92 Xx1 Yy1 ;

Coordinate system after coordinate system setting by G92



A workpiece coordinate system in which the current tool position is set at (x1, y1) will be newly created.

(a) Tool position
(b) G92 commanded position
(c) G92 shift amount
(W0) Workpiece coordinate system
(W1) Workpiece coordinate system after G92 command
(M) Basic machine coordinate system

The following values are set as G92 shift amounts.
X: x0 - x1
Y: y0 - y1

There are two methods to return a shifted coordinate system to its original position.

(1) Carry out dog-type reference position return.

(2) Move the coordinate system to the machine coordinate system zero point, then command G92 and G53 in the same block to set the machine coordinate system.

G90 G53 G00 X0 Y0 ;	Positioning at machine coordinate system zero point
G92 G53 X0 Y0 ;	Coordinate system zero setting in machine coordinate system This returns all the workpiece coordinates (G54 to G59 and G54.1 Pn) to their original positions



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.

(2) For the "G92 G53" command, do not specify a value other than "0" for the axis address. When a command other than "0" is specified, the operation depends on the setting of "#1751 cfgPR01/bit4".

When "#1751 cfgPR01/bit4" is set to "0", the program error (P35) occurs.

When "#1751 cfgPR01/bit4" is set to "1", the operation is performed assuming that "0" is commanded to the axis address.

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 or on the G54.1 extended workpiece coordinate system 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 (or G55 to G59, G54.1P__) G52 X__ Y__ Z__ α__ ;

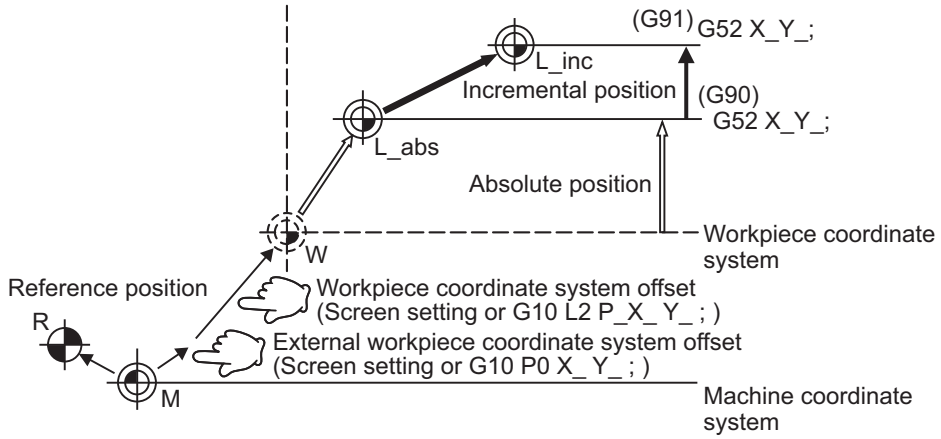
α	Additional axis

Detailed description

(1) When G52 is commanded with the absolute command, command position on the workpiece coordinate system being selected is determined as the zero point of local workpiece coordinate system.
When G52 is commanded with the incremental command, the zero point of local workpiece coordinate system depends on the MTB specification (parameter "#1296 ext32/bit1").

[#1296 ext32/bit1 = 0]

When G52 is commanded with the incremental command, the zero point of the local workpiece coordinate system shifts by the commanded amount.



L_abs: Zero point of local workpiece coordinate system commanded with the absolute command

L_inc: Zero point of local workpiece coordinate system commanded with the incremental command

[#1296 ext32/bit1 = 1]

Regardless of the absolute command or incremental command, command position on the workpiece coordinate system being selected is determined as the zero point of local workpiece coordinate system. (Same as when commanded with the absolute command)

(2) The local coordinate system can be canceled by commanding "G52 X0 Z0 Y0 ;".

When the parameter "#1296 ext32/bit1" is "1", it can be canceled by commanding "0" with the incremental command.

(3) The G52 command is valid until a new G52 command is issued. In the G52 command block, no movement is performed. 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).

(4) The local coordinate system offset is cleared at resetting.
The local coordinate system offset is not cleared by reset 1 when the parameter "#1151 rstint" is set to "0".

(5) The local coordinate system offset will be cleared when the dog-type reference position return is carried out or the power is turned ON again.

(6) The local coordinate system for the G54 to G59 commands is independently set on each workpiece coordinate system. Only a single local coordinate system is set to the extended workpiece coordinate system G54.1 (G54) P command. Even if the P command value is changed, the local coordinate system offset amount is inherited.

(7) The workpiece coordinate system where the local coordinate system is set when G52 is commanded depends on the MTB specifications (parameter "#1296 ext32/bit3").

[#1296 ext32/bit3 = 1]

The local coordinate system is set for all the workpiece coordinates (G54 to G59, G54.1).

[#1296 ext32/bit3 = 0]

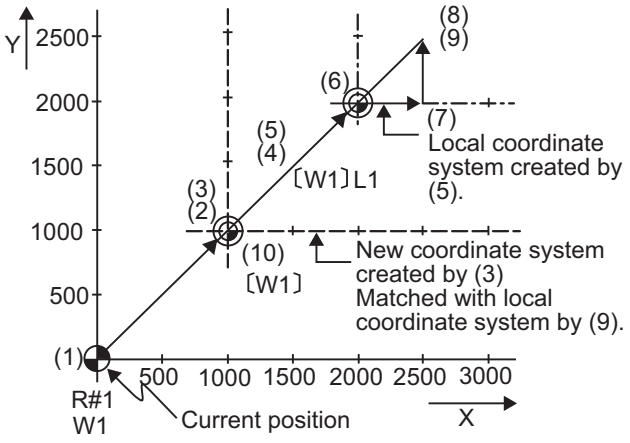
The local coordinate system is set for only the selected workpiece coordinate system (G54 to G59, G54.1).

<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 1) Local coordinates for absolute mode (The local coordinate system offset is not cumulated)

(1) G28 X0 Y0 ;
(2) G00 G90 X1000 Y1000 ;
(3) G92 X0 Y0 ;
(4) G00 X500 Y500 ;
(5) G52 X1000 Y1000 ;
(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 2-1) Local coordinates for incremental command mode (#1296 ext32/bit1 = 0)

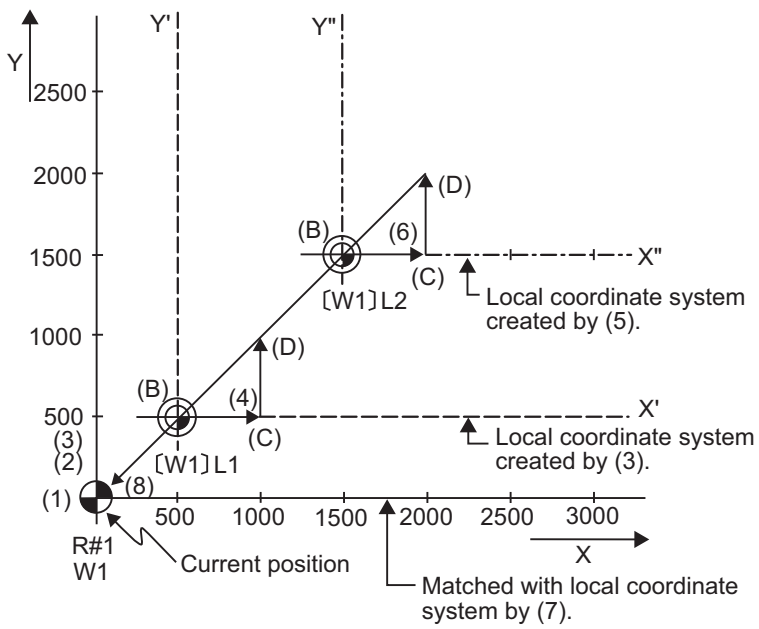
The local coordinate system offset is cumulated.

<Main program>

(1) G28 X0 Y0 ;
(2) G92 X0 Y0 ;
(3) G91 G52 X500 Y500 ;
(4) M98 P100 ;
(5) G52 X1000 Y1000 ;
(6) M98 P100 ;
(7) G52 X-1500 Y-1500 ;
(8) G00 G90 X0 Y0 ;
M02 ;

<Subprogram>

(A) O100 ;
(B) G90 G00 X0 Y0 ;
(C) G01 X500 F100 ;
(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 XY coordinate system (1000, 1000) position by (5).
The local coordinate system is created at the XY 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.

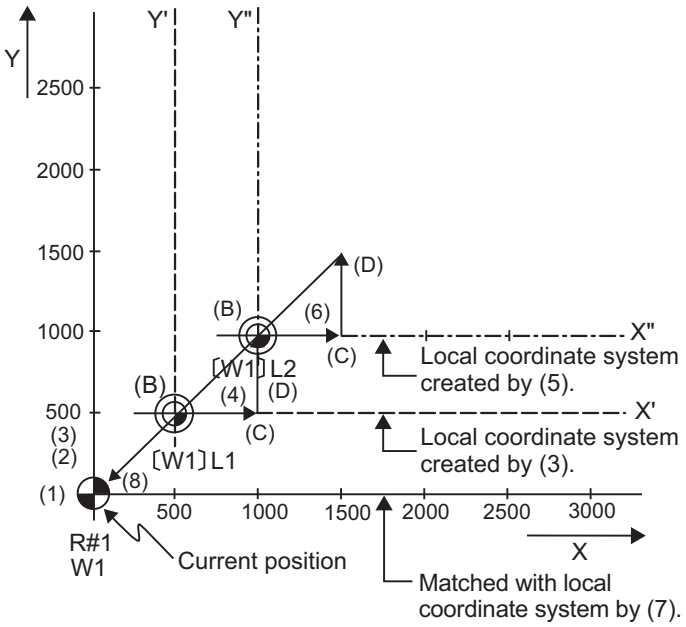
(Example 2-2) Local coordinates for incremental command mode (#1296 ext32/bit1 = 1)
The local coordinate system offset is not cumulated.

<Main program>

(1) G28 X0 Y0 ;
(2) G92 X0 Y0 ;
(3) G91 G52 X500 Y500 ;
(4) M98 P100 ;
(5) G52 X1000 Y1000 ;
(6) M98 P100 ;
(7) G52 X0 Y0 ;
(8) G00 G90 X0 Y0 ;
M02 ;

<Subprogram>

(A) O100 ;
(B) G90 G00 X0 Y0 ;
(C) G01 X500 F100 ;
(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 XY coordinate system (1000, 1000) position by (5).
The local coordinate system is created at the XY coordinate system (0, 0) 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.

(Example 3-1) When used together with workpiece coordinate system (#1296 ext32/bit3 = 0)

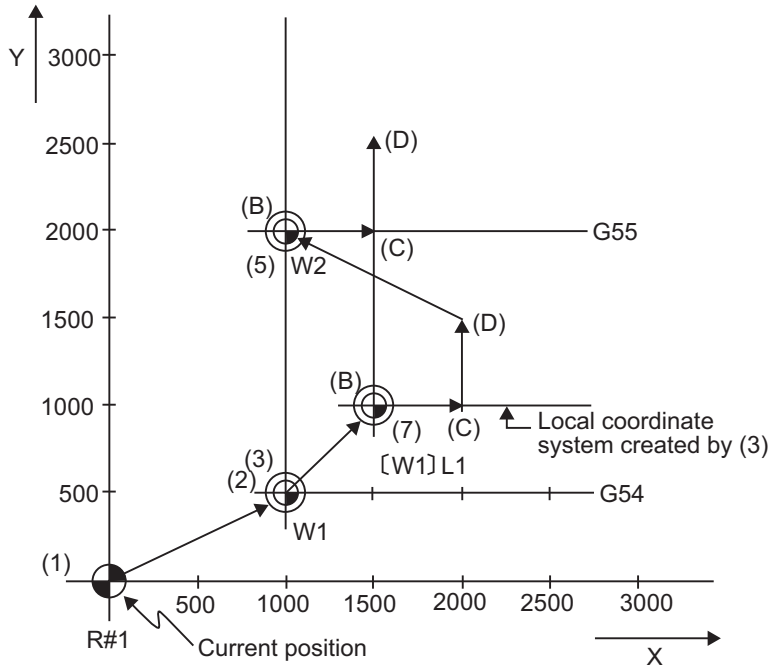
<Main program>

(1) G28 X0 Y0 ;
(2) G00 G90 G54 X0 Y0 ;
(3) G52 X500 Y500 ;
(4) M98 P200 ;
(5) G00 G90 G55 X0 Y0 ;
(6) M98 P200 ;
(7) G00 G90 G54 X0 Y0 ;
:
M02 ;

	G54	G55	
X	1000	1000	Workpiece coordinate system offset (Parameter setting value)
Y	500	2000	

<Subprogram>

(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;". When "#1296 ext32/bit1" is "1", it is also canceled by "G91 G54 G52 X0 Y0;".

(Example 3-2) When used together with workpiece coordinate system (#1296 ext32/bit3 = 1)

<Main program>

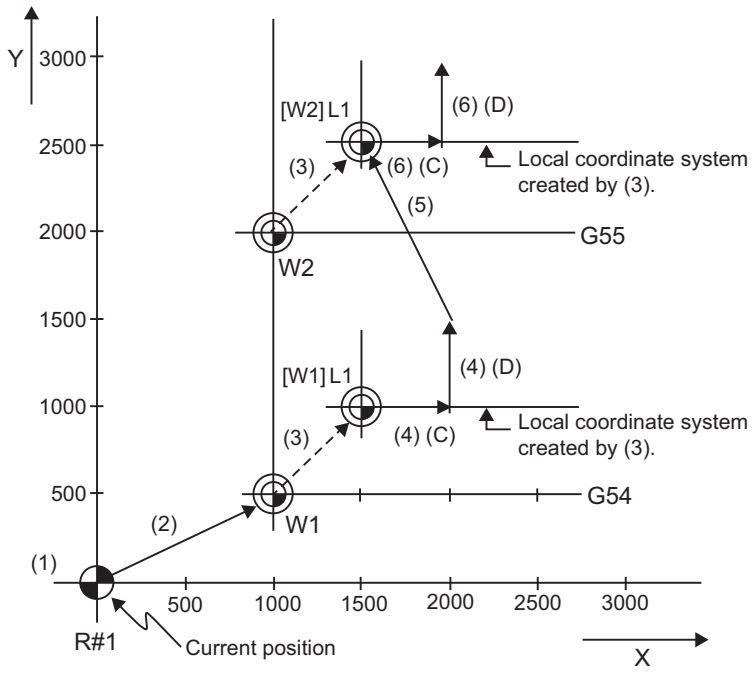
(1) G28 X0 Y0 ;
(2) G00 G90 G54 X0 Y0 ;
(3) G52 X500 Y500 ;
(4) M98 P200 ;
(5) G00 G90 G55 X0 Y0 ;
(6) M98 P200 ;
:
M02 ;

	G54	G55
X	1000	1000
Y	500	2000

Workpiece coordinate system offset (Parameter setting value)

<Subprogram>

(A) O200 ;
(B) G00 X0 Y0 ;
(C) G01 X500 F100 ;
(D) Y500 ;
(E) M99 ;



The local coordinate system is set for all the workpiece coordinate systems (G54.1, G54 to G59), and is also set for G55 by the G52 command in (3). (The part indicated by [W2] L1 in the figure)

(Example 4) Combination of workpiece coordinate system G54 and multiple local coordinate systems

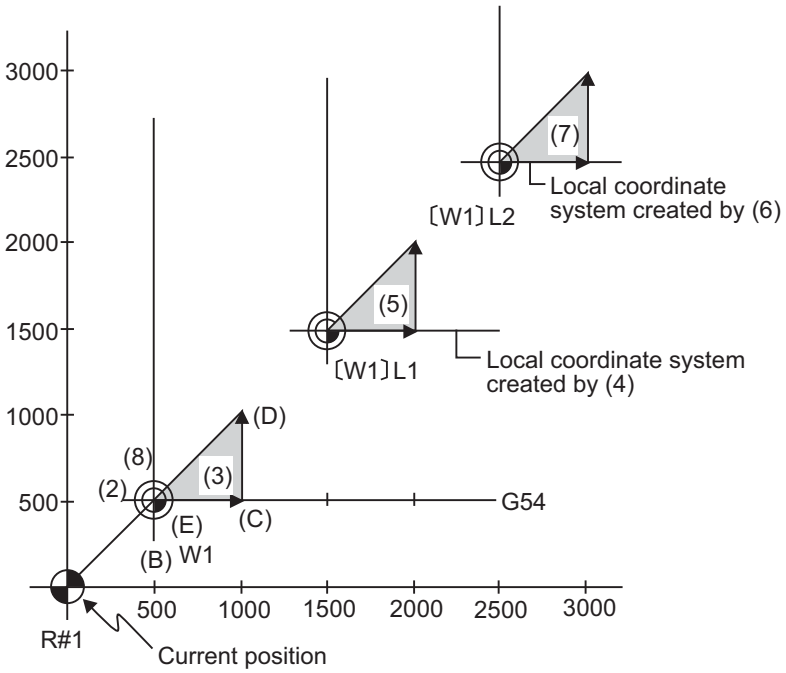
<Main program>

(1) G28 X 0 Y0 ;
(2) G00 G90 G54 X0 Y0 ;
(3) M98 P300;
(4) G52 X1000 Y1000 ;
(5) M98 P300 ;
(6) G52 X2000 Y2000 ;
(7) M98 P300 ;
(8) G52 X0 Y0 ;
:
M02 ;

G54
X 500
Y 500
Workpiece coordinate system offset (Parameter setting value)

<Subprogram>

(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 Selection and Extended Workpiece Coordinate System Selection; G54 to G59, G54.1



Function and purpose

The workpiece coordinate systems facilitate the programming on the workpiece, serving the reference position of the machining workpiece as the zero point.

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 six 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.

Workpiece coordinate system

(G90) G54 - G59 ;	**Workpiece coordinate system selection**

Extended workpiece coordinate system

G54.1 Pn ;	**Extended workpiece coordinate system selection (P1 to P300) (*1)**

G54 Pn ;	**Extended workpiece coordinate system selection (P1 to P300) (*1)** **(It is valid when the parameter "#1274 ext10/bit5" is "1".)**

(*1) The maximum number of coordinate systems depends on the specifications.

Note

(1) Address "P" cannot be omitted in the G54.1 command.

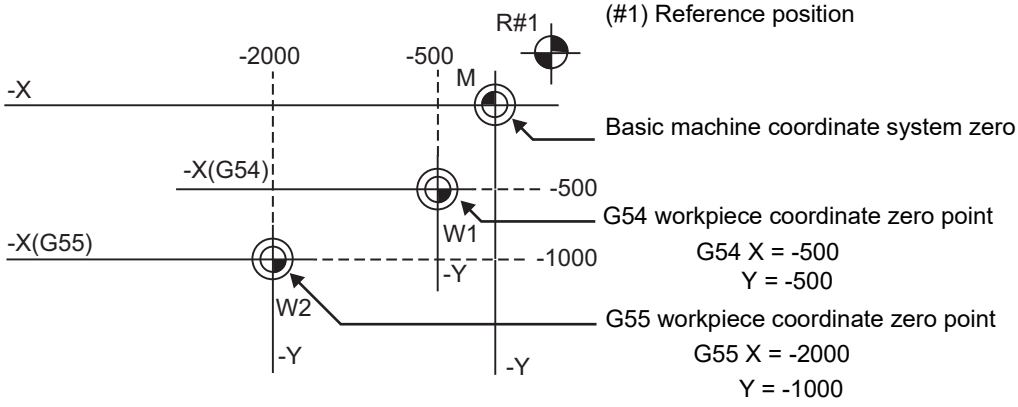
Detailed description

(1) Even if the switching of the workpiece coordinate system is commanded using G54 to G59 or G54.1Pn, the tool radius compensation amount and tool length compensation amount are not canceled.

(2) The G54 workpiece coordinate system is selected when the power is turned ON.

(3) G54 through G59 and G54.1Pn commands are modal commands (group 12).

(4) The position of the workpiece coordinate system can be specified in [Coord] on the setup screen. For the offset amount, specify the distance from the basic machine coordinate system zero point to the workpiece coordinate system zero point.



(5) 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.

(6) Do not command a G code for which a P code is used in the same block as G54.1. If such a G code is commanded, a P code is used for a prior G command or the program error occurs (P33).

(7) Extended workpiece coordinate system selection (G54.1) and the number of workpieces depend on the MTB specifications. When unavailable commands are executed, the program error (P39) occurs.

(8) The local coordinate system for the G54 to G59 commands is independently set on each workpiece coordinate system; however, only a single local coordinate system is set to the "G54.1 (G54) Pn" command.

Even if the extended workpiece coordinate system number 1 to 300 is changed, the local coordinate system offset amount is inherited.

(9) The workpiece coordinate system offset amount and the extended workpiece coordinate system offset amount can be specified by the G10 L2/L20 command and the variable command.
For details, refer to the following chapters.

- 15.9.3 Compensation Data Input by Program (Workpiece Offset Amount) ; G10 L2/L20, G11
- 23.8 System Variables (Workpiece Coordinate Offset)
- 23.10 System Variables (Extended Workpiece Coordinate Offset)

⚠ CAUTION

⚠ If the workpiece coordinate system offset amount is changed during single block stop, the new setting will be valid from the next block.

(10) When the "#1151 rstint" parameter is set to "0", the modal of G54.1 command will be retained even if the reset 1 is carried out.

(11) Address "P" cannot be omitted in the G54.1 command.
Address "P" can be omitted in the G54 command; however, the standard workpiece coordinate system (G54) is selected regardless of the parameter "#1274 ext10/bit5".

(12) 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.

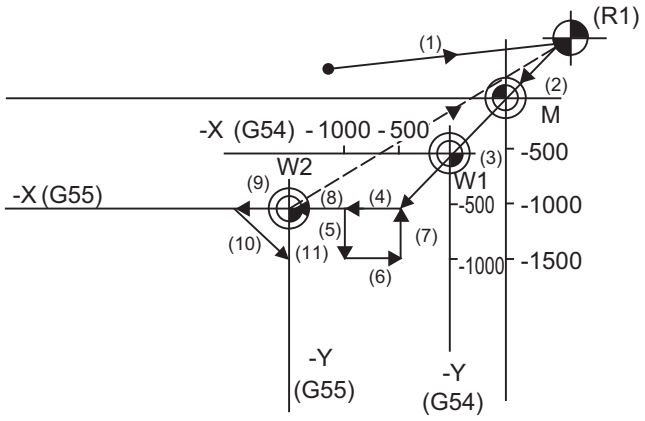
(Example)

P54.1 P5 ;	Changed to P5 workpiece coordinate system.
P3 ;	Ignored.

Program example

(Example 1)

(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 2) The same machining is performed on the six workpieces which are placed on the G54 to G59 coordinate systems respectively.

(1) Setting of workpiece coordinate system offset data

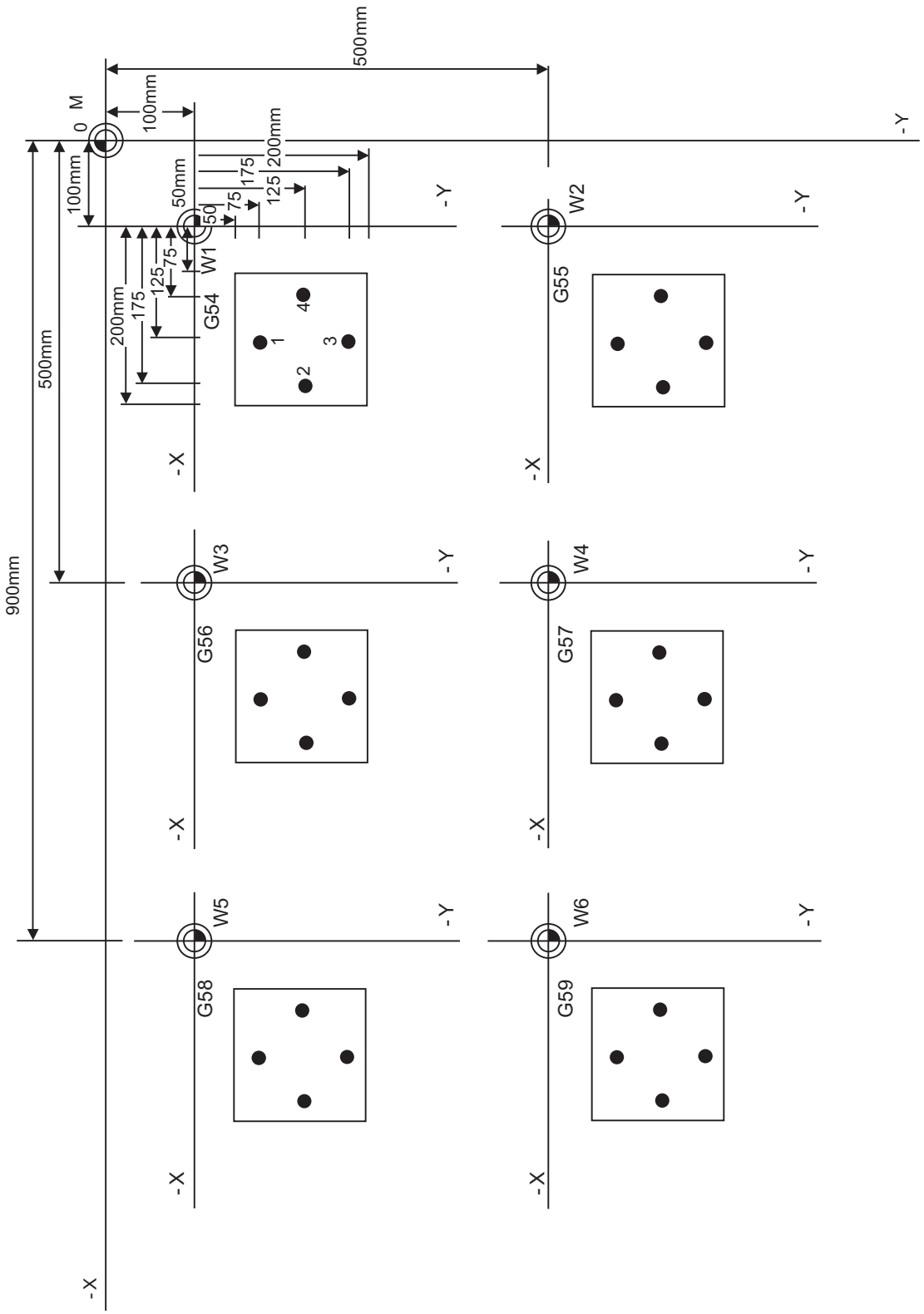
Workpiece 1	X=-100.000, Y=-100.000.....G54
Workpiece 2	X=-100.000, Y=-500.000.....G55
Workpiece 3	X=-500.000, Y=-100.000.....G56
Workpiece 4	X=-500.000, Y=-500.000.....G57
Workpiece 5	X=-900.000, Y=-100.000.....G58
Workpiece 6	X=-900.000, Y=-500.000.....G59

(2) Positioning program (main)

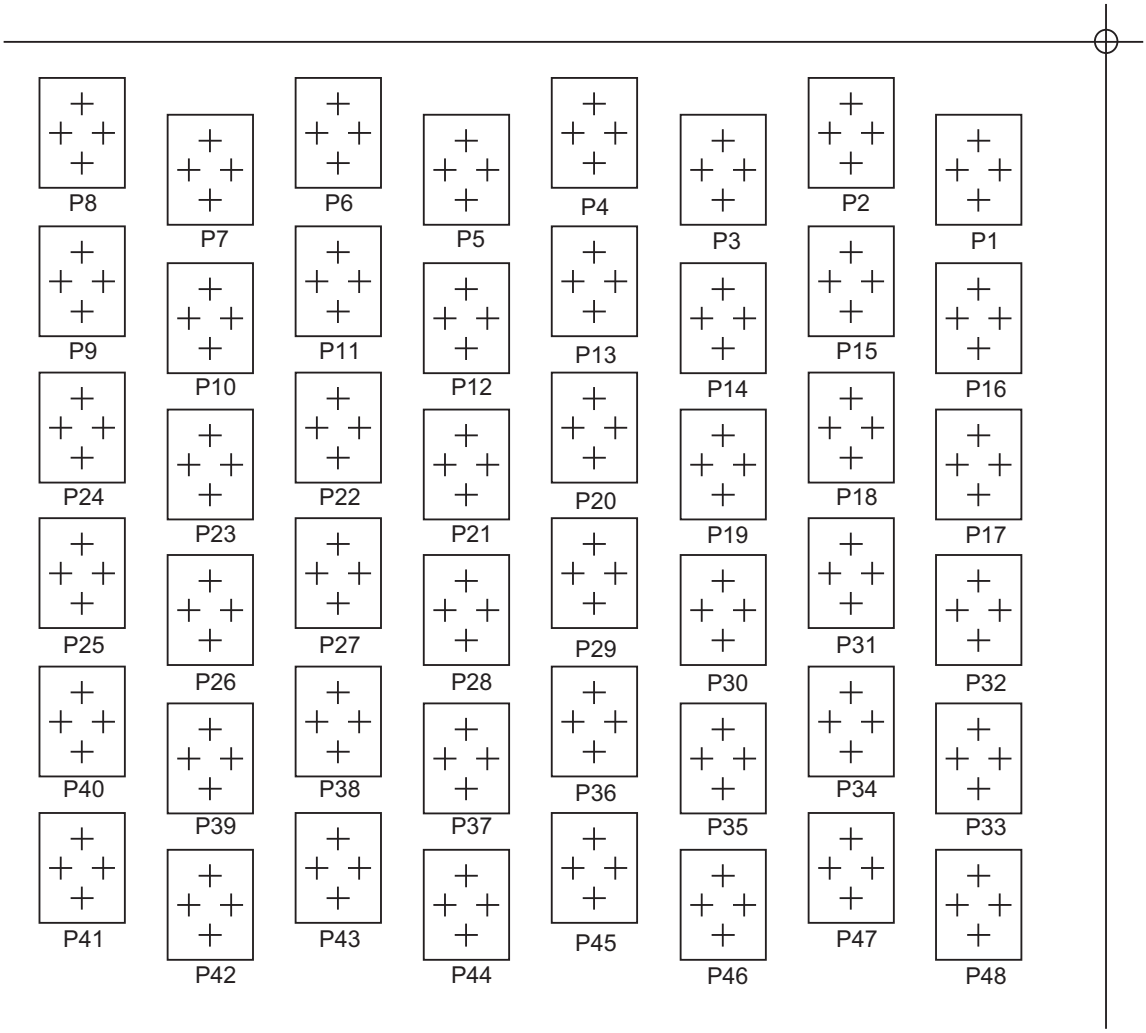
```
G91 G28 X0 Y0 Z0 ;                    At power ON
N1  G90 G54 M98 P100 ;
N2      G55 M98 P100 ;
N3      G57 M98 P100 ;
N4      G56 M98 P100 ;
N5      G58 M98 P100 ;
N6      G59 M98 P100 ;
N7  G91 G28 X0 Y0 Z0 ;
N8  M02 ;
%
```

(3) 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      G91 G28 X0 Y0 Z0 ;
        :
N4      G90 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      G91 G28 X0 Y0 Z0 ;
        :
N6      G90 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 ;
```


500mm
100mm
M
0
-Y
100mm
50mm
175
200mm
125
75
50
-Y
W2
-Y
G55
W1
75
125
175
200mm
G54
4
1
3
2
500mm
-X
-X
900mm
W3
-Y
W4
-Y
G56
G57
-X
-X
W5
-Y
W6
-Y
G58
G59
-X
-X
-X

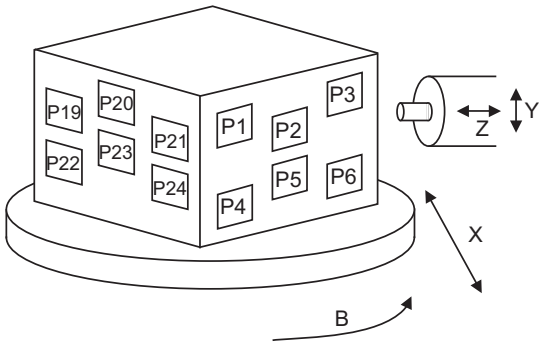
(Example 3) Program example to continuously use 48 sets of extended workpiece coordinate system offsets.
In this example, the offsets for each workpiece are set beforehand in P1 to P48 when 48 workpieces are fixed on a table, as shown in the drawing below.



O1000		O1001	
G91 G28 X0 Y0 Z0 ;	Reference position return	G90 ;	
#100=1 ;	Number initialization in extended workpiece coordinate system P	G43 X-10.Y-10.Z-100.H10.;	Contour
WHILE [#100LE48]D01 ;	Repeat P No. to 48	G01 X-30.;	
G54.1 P#100 ;	Workpiece coordinate system setting	Y-30.;	
M98 P1001 ;	Call subprogram	X-10.;	
#100=#100+1 ;	P No. +1	Y-10.;	
END1 ;		G00 G40 Z10.;	
G91 G28 Z0 ;	Return to reference position	G98 G81 X-20.Y-15.Z-150.R5.F40;	Drilling
G91 G28 X0 Y0 ;		X-25.Y-20.;	
M02 ;		X-20.Y-25.;	
		X-15.Y-20.;	
		G80 ;	
		M99 ;	

(Example 4) Program example to transfer the extended workpiece coordinate system offset to the standard workpiece coordinate system offset

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)

```
G91 G28 X0 Y0 Z0 B0 ;  Reference position return
G90 ;                  Absolute command 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 ;
G91 G28 X0 Y0 B0 ;     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 extended 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
G91 G28 Z0 ;
M99 ;
N100 G90 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 ;
G91 G28 Z0 ;
M99 ;
%
```

Relationship with other functions

Multiple-axis synchronization control

(1) The workpiece offset of the slave axis depends on the MTB specifications.

#1255 set27/bit5	#1712 cfg12/bit3	Workpiece offset of slave axis
0	0	[During synchronous operation] The slave axis operates on the master axis coordinate system. A value with the master axis workpiece offset applied is displayed in the workpiece coordinate counter of the slave axis. [After synchronous operation is canceled] The axis operates and the workpiece coordinate counter is displayed on each axis coordinate system.
1	0	
0	1	
1	1	[During both synchronous operation and tool length compensation] The master axis and the slave axis operate on each coordinate system. A value with the slave axis workpiece offset applied is displayed in the workpiece coordinate counter of the slave axis. [During synchronous operation and after tool length compensation is canceled] The slave axis operates on the master axis coordinate system. [After synchronous operation is canceled] The axis operates and the workpiece coordinate counter is displayed on each axis coordinate system.

When the parameter "#1712 cfg12/bit3" is "1" and the workpiece offset is changed by the system variables, etc. during tool length offset, the changed workpiece offset is enabled by commanding the tool length offset again. When the tool length is not changed, command the tool length offset with the same tool No. as the previous one.

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

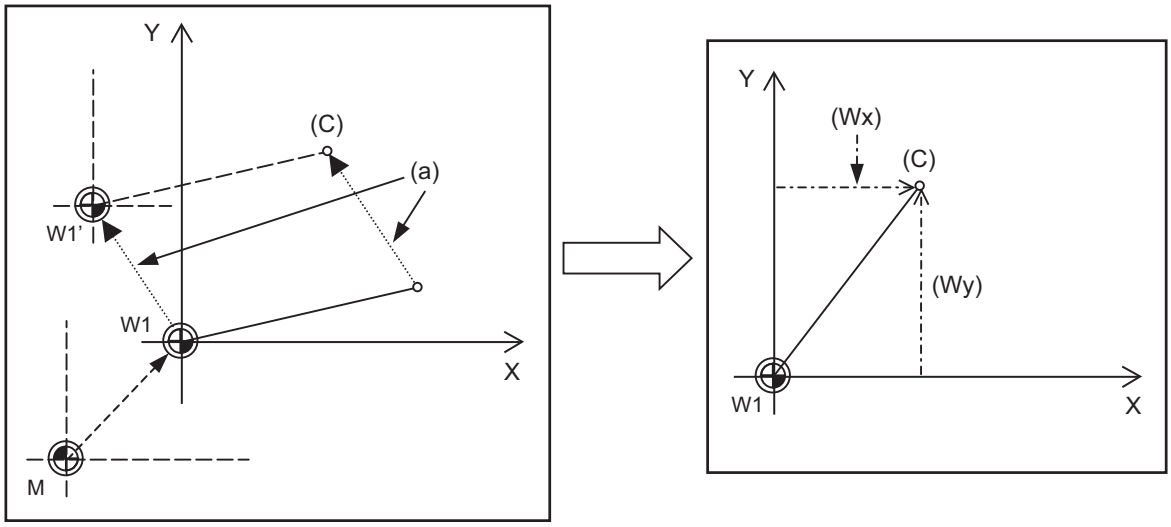
G92.1 X0. Y0. Z0. α0 ;

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) G92.1 must be commanded alone in a block.
(4) Whether to conduct an error check when the coordinate system preset command (G92.1) is issued alone in a block depends on the MTB specifications (parameter "#1242 set14/bit1").

Detailed description

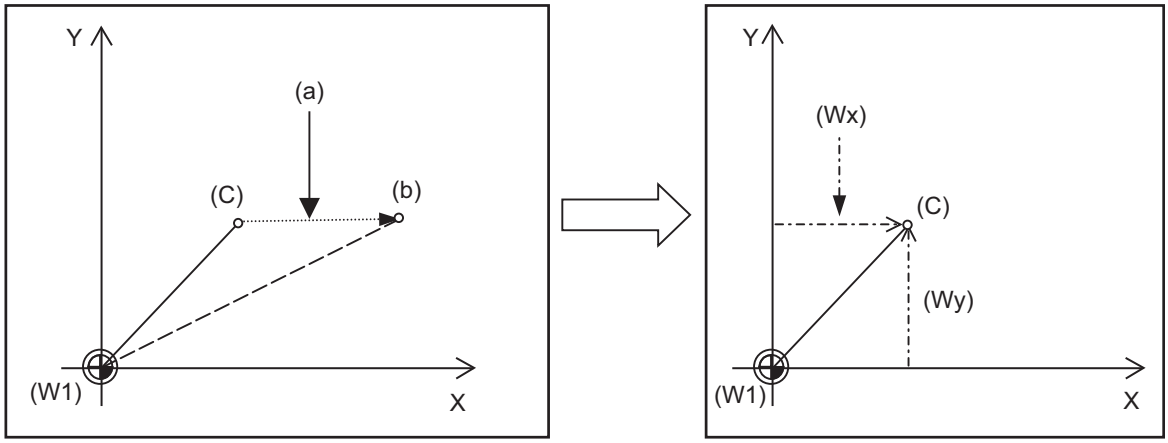
(1) When manual operation is carried out when manual absolute is set to OFF, or if the axis is moved with handle interrupt.



(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.

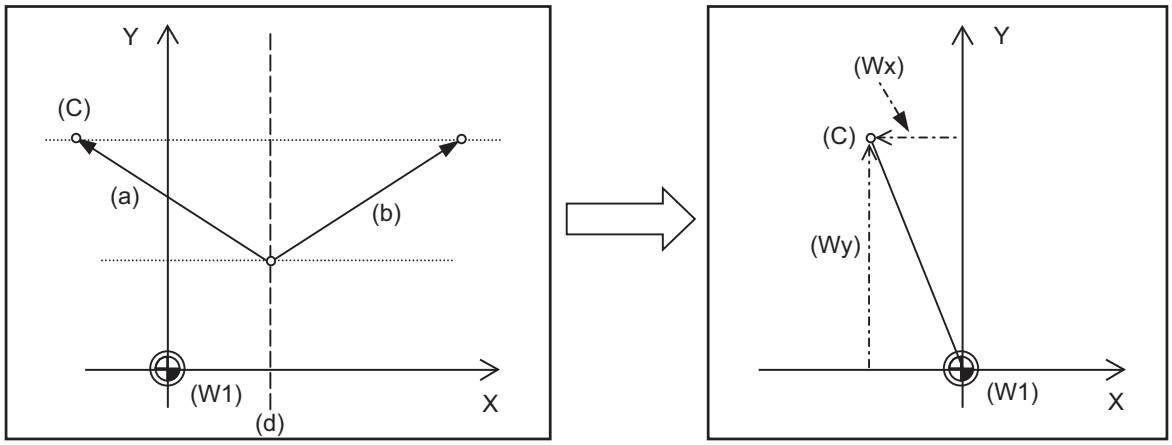
(2) When movement command is issued in machine lock state



(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.

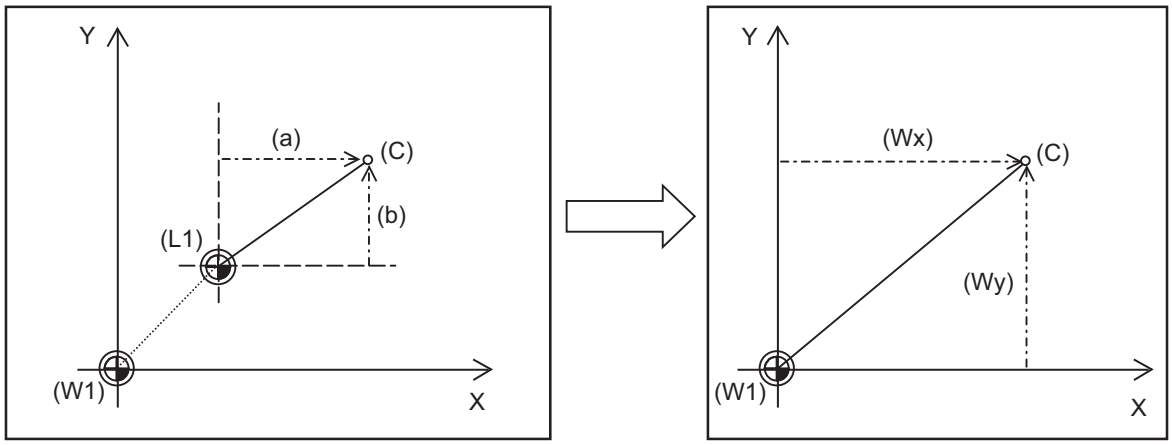
(3) When operation is carried out with mirror image



(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.

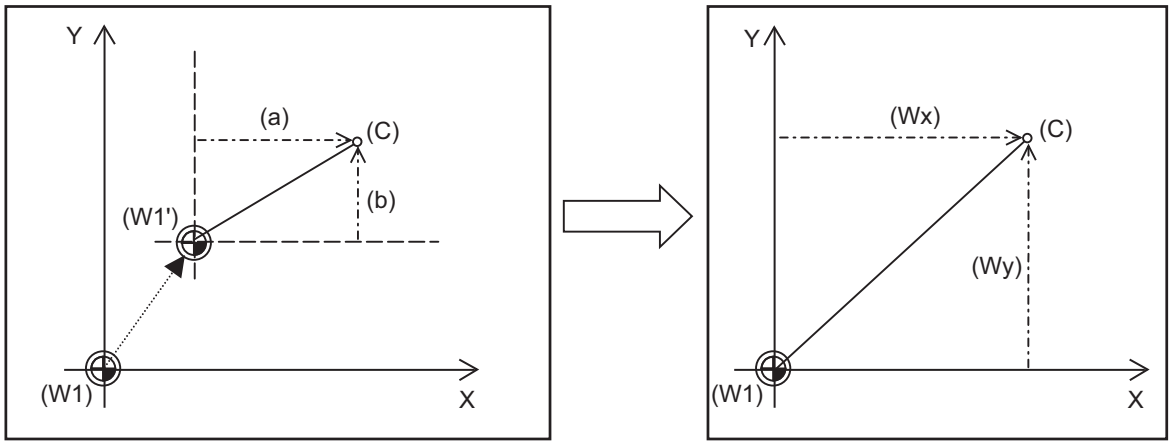
(4) Setting local coordinate system with G52



(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

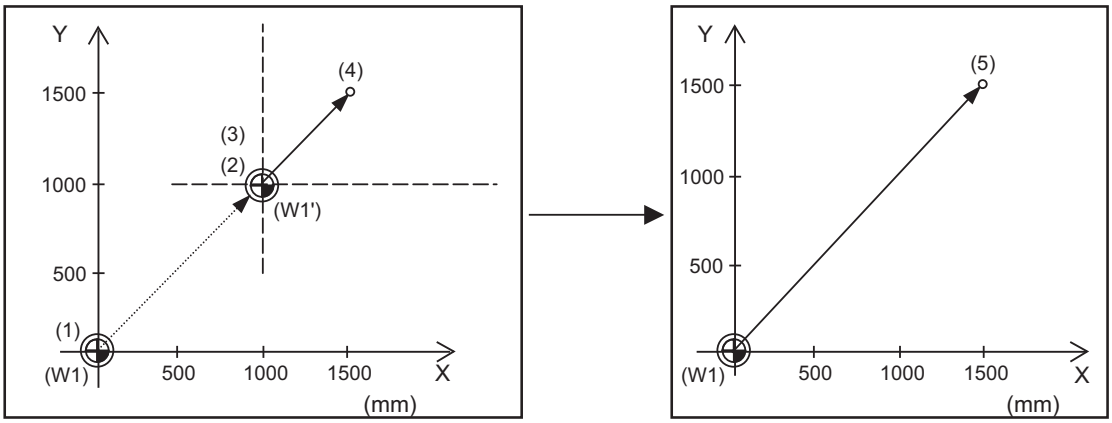
(W1') Workpiece zero point after G92 command

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 commanded alone in a block (*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.

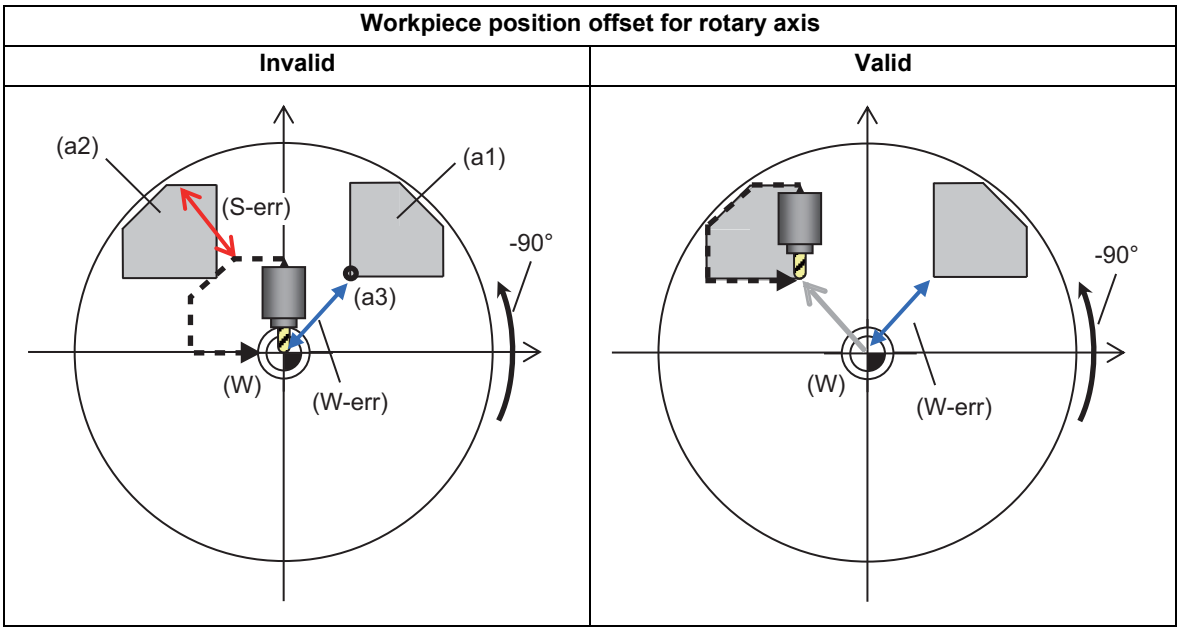
20.8 Workpiece Position Offset for Rotary Axis ; G54.2

Function and purpose

For a machine equipped with a table rotary axis, this function compensates an installation error between workpiece coordinate zero point and workpiece reference position.
When a workpiece set on a rotary table is machined, if there is a difference between the workpiece coordinate zero point and the workpiece reference position, an error occurs in the machined shape. Setting the installation error (workpiece position offset amount for rotary axis) between the workpiece coordinate zero point and the workpiece reference position corrects in accordance with the table rotation angle for each block. This can reduce the shape error when machining is carried out, which provides more accurate machining.

This function is available only in the first part system. When this function is used in other than the first part system, the program error (P34) occurs.
The offset is applied to an axis for which the parameter "#2100 rpcax" (Workpiece position offset for rotary axis valid) is enabled.

Workpiece position offset for rotary axis

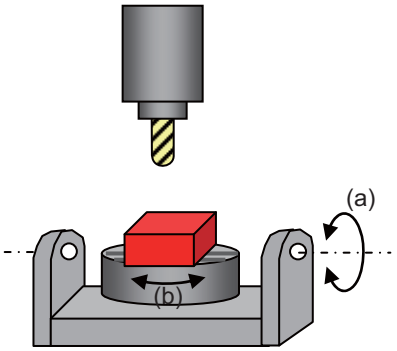
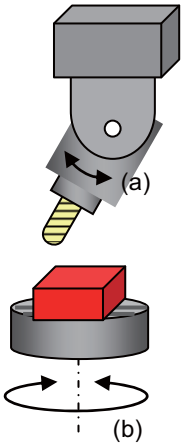
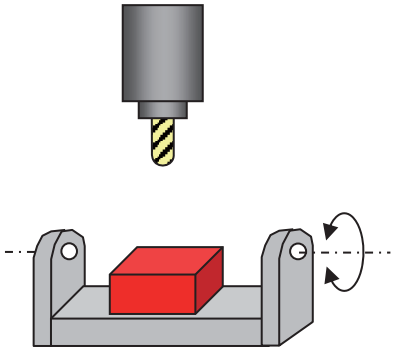


(a1) Workpiece position before rotation
(a2) Workpiece position after rotation of -90°
(a3) Workpiece reference position
(S-err) Shape error
(W-err) Workpiece installation error
Offset amount after rotation
Tool path

Axis configuration of applicable machines

The function is available 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	Table tilt type	Combined type	4-axis machine
Description	Two rotary axes on table side	One rotary axis each on tool head side and table side	Machine equipped with 3 orthogonal axes and 1 rotary axis
Example of machines			
Primary rotary axis (a)	Table-side 1st rotary axis	Tool-side rotary axis	None
Secondary rotary axis (b)	Table-side 2nd rotary axis	Table-side rotary axis	None

Note

(1) This function is not applicable to the machines as below:
- A machine whose direction from the tool tip to the tool base is not parallel to the Z axis (Z axis positive direction) when the machine positions of all the rotary axes are 0°
- A machine whose rotation center axis of the rotary axis is not parallel to any orthogonal coordinate axis

Command format

G54.2 P__ ;

P	Set No. of workpiece position offset for rotary axis (n = 0 to 8) 0: Cancels the workpiece position offset for rotary axis. 1 to 8: Applies offset with the corresponding workpiece position offset amount.

(1) G54.2 must be commanded alone in a block. When it is commanded in the same block with other G codes, travel commands or miscellaneous commands, the program error (P34) occurs.

(2) When address P is not commanded, the program error (P33) occurs. Also, commanding values other than "0" to "8" to the address P causes the program error (P35).

(3) If "P" is given in a block that contains no G54.2 command, the address P is not regarded as a command for specifying the workpiece position offset No. for rotary axis.

(4) G54.2 is a modal command of the G code group 23.

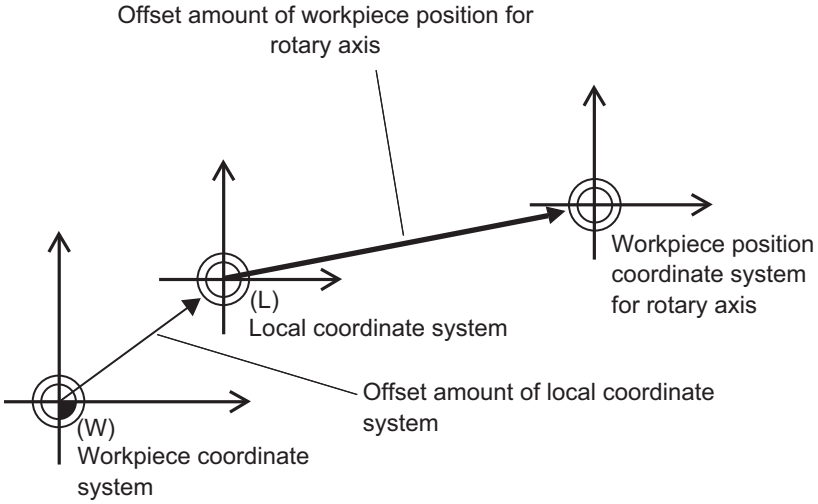
(5) When "G54.2 Pm" is given during "G54.2 Pn", offset is applied with the offset No. "m" that is specified later.

(6) Designate the compensation amount of the set to be specified with address P for the system variable. For details about the system variable for designating the compensation amount, refer to "23.9 System Variables (Workpiece Position Offset Amount for Rotary Axis)".

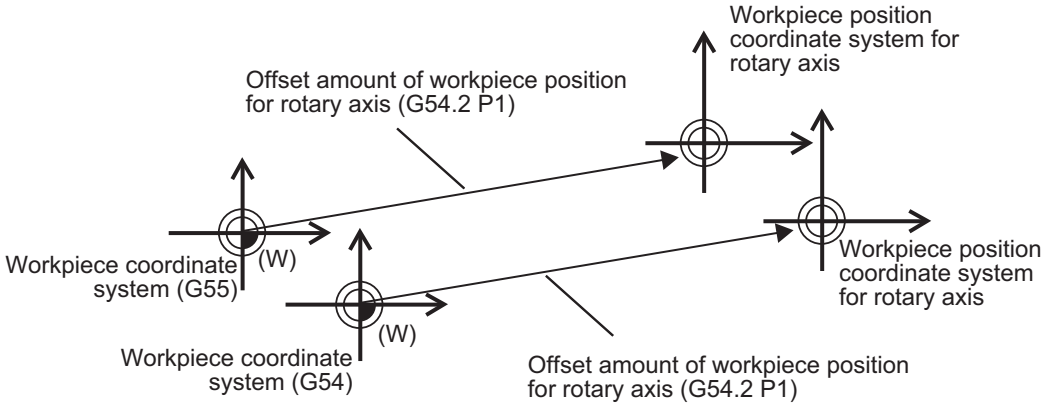
Detailed description

Workpiece position coordinate system for rotary axis

When workpiece position offset for rotary axis is commanded, a coordinate system, to which the workpiece position offset amount for rotary axis is reflected, is to be defined. The coordinate system is called "workpiece position coordinate system for rotary axis".
The workpiece coordinate position counter at the time when this function is enabled indicates the position on the workpiece position coordinate system for rotary axis.



When the workpiece coordinate system is changed while the workpiece position offset for rotary axis is enabled, the workpiece position coordinate system for rotary axis according to the workpiece coordinate system after the change is defined. Thus, the workpiece position offset amount for rotary axis needs to be commanded in accordance with the measured workpiece coordinate system.
The following figure shows the relation between the workpiece position coordinate system for rotary axis on the workpiece coordinate system (G54) and the workpiece position coordinate system for rotary axis on the workpiece coordinate system (G55) when the local coordinate system offset amount is "0". Even if the workpiece position offset amount for rotary axis is the same, the zero points of the workpiece position coordinate systems for rotary axis are different.



Operation example

Start/Cancel of workpiece position offset for rotary axis

The workpiece position offset for rotary axis mode is turned ON by the command of "G54.2 Pn" (n = 1 to 8). The machine moves by the workpiece position offset amount for rotary axis specified with the address "P". When a rotary axis angle at G54.2 command is different from the rotary axis angle at measurement of the workpiece position offset amount for rotary axis specified by P, the workpiece position offset amount for rotary axis according to the current rotary axis angle is calculated and machine moves.

When the workpiece position offset for rotary axis is started or canceled, the operation depends on the MTB specifications (parameter "#1250 set22/bit7").

Items (a) and (b) in the table below show differences depending on this setting.

(a): The machine moves. (bit7 = 0)

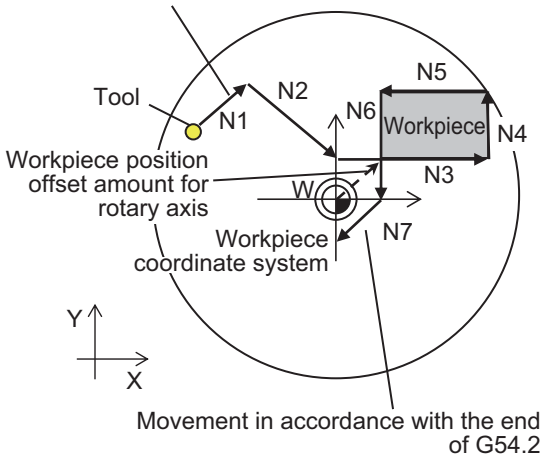
(b): The machine does not move. (bit7 = 1)

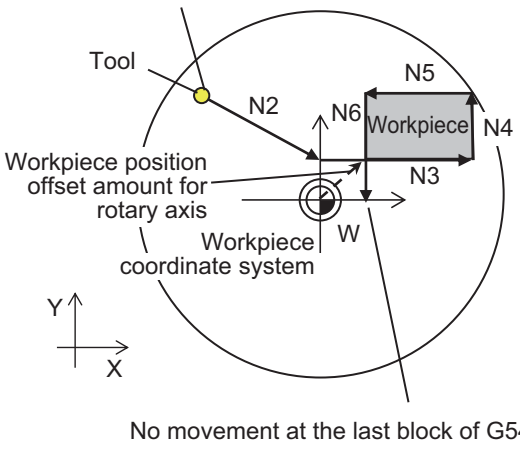
Program	Workpiece coordinate position			Machine position			Details of operation
	X	Y	Z	X	Y	Z	
G54;	-20	10	100	-20	10	0	Initial position
#5521 = 5.0; (*1)	-20	10	100	-20	10	0	Set the workpiece position offset amount for rotary axis. (*2)
#5522 = 5.0;	-20	10	100	-20	10	0	
#5523 = 0.0;	-20	10	100	-20	10	0	
#5524 = 0.0;	-20	10	100	-20	10	0	
#5525 = 0.0;	-20	10	100	-20	10	0	
N1 G54.2 P1;	**(a): -20** **(b): -25**	**(a): 10** **(b): 5**	100	**(a): -15** **(b): -20**	**(a): 15** **(b): 10**	0	Workpiece position offset for rotary axis is enabled. (a) The machine moves by the designated offset amount. (b) The machine does not move but the workpiece coordinate position shifts.
N2 G90 G00 X-5. Y0.;	-5	0	100	0	5	0	The machine moves by the set offset amount in the state that the offset is applied to X and Y axes directions.
G00 Z5.;	-5	0	5	0	5	-95	
G01 Z0. F1000;	-5	0	0	0	5	-100	
N3 G90 G01 X20.;	20	0	0	25	5	-100	
N4 Y10.;	20	10	0	25	15	-100	
N5 X0.;	0	10	0	5	15	-100	
N6 Y-5.;	0	-5	0	5	0	-100	
N7 G54.2 P0;	**(a): 0** **(b): 5**	**(a): -5** **(b): 0**	0	**(a): 0** **(b): 5**	**(a): -5** **(b): 0**	-100	Workpiece position offset for rotary axis is disabled. (a) The machine moves by the canceled offset amount. (b) The machine does not move but the workpiece coordinate position shifts by the canceled offset amount.

(*1) #5521 to #5525 are system variables of the workpiece position offset amount for rotary axis. For details, refer to "23.9 System Variables (Workpiece Position Offset Amount for Rotary Axis)".

(*2) This setting example is available when an error occurs 5.0 mm in the X axis direction and 5.0 mm in the Y axis direction while the angle of the rotary axis is set to 0 degree.

When the machine moves	When the machine does not move
Movement in accordance with the start of G54.2	No movement at the first block of G54.2





Although the above example is described with absolute command (G90), the operation is the same even with incremental command (G91).

When "The machine does not move" (#1250/bit7 = 0) is selected, the machine does not move by the workpiece position offset amount for rotary axis specified by "P" in "G54.2 Pn".
When the absolute command (G90) is issued, the end point of the next movement block will be the position to which the offset is applied.
When the incremental command (G91) is issued, the end point will not be the position to which the offset is applied because the machine moves by the commanded incremental amount from the current position.

Operation during workpiece position offset for rotary axis

In G54.2 mode, the offset amount which is designated with "P" is applied. However, when the rotary axis is rotated, a direction that the offset to be applied is changed in accordance with the rotary axis angle in the block end point. With the change in the direction of the offset to be applied, the linear axes move.
The offset amount is calculated only for the block end point of machining programs. Therefore, the path is not necessarily performed as it is intended.
When the workpiece position offset for rotary axis is started or canceled, the operation depends on the MTB specifications (parameter "#1597 rpcNoMove").

Items (a) and (b) in the table below show differences depending on this setting.

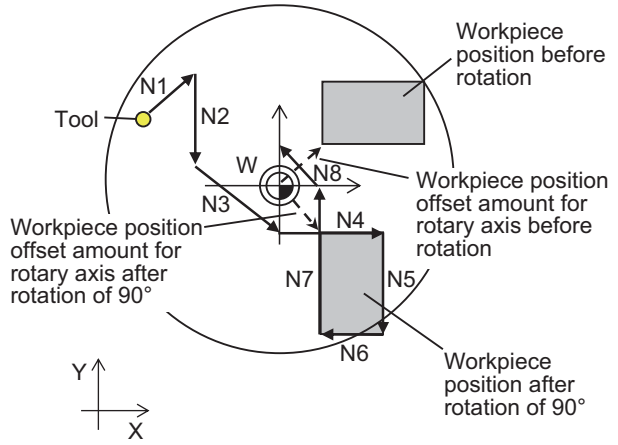
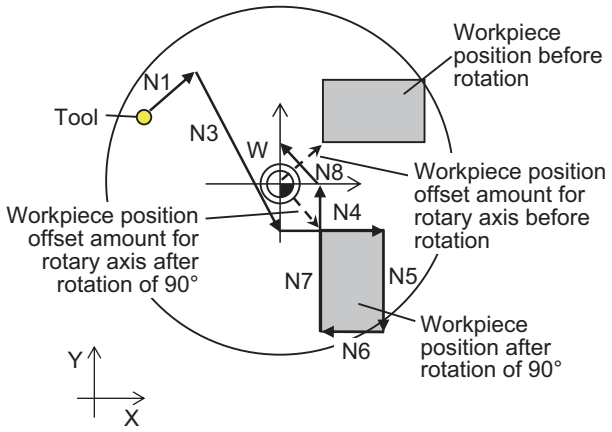
(a): The machine moves. (#1597 = 0)

(b): The workpiece coordinate position moves. (The machine does not move.) (#1597 = 1)

Program	Workpiece coordinate position			Machine position			Details of operation
	X	Y	Z	X	Y	Z	
G54;	-20	10	100	-20	10	0	Initial position
#5521 = 5.0; (*1)	-20	10	100	-20	10	0	Set the workpiece position offset amount for rotary axis. (*2)
#5522 = 5.0;	-20	10	100	-20	10	0	
#5523 = 0.0;	-20	10	100	-20	10	0	
#5524 = 0.0;	-20	10	100	-20	10	0	
#5525 = 0.0;	-20	10	100	-20	10	0	
N1 G54.2 P1;	-20	10	100	-15	15	0	Workpiece position offset for rotary axis is enabled. The machine moves by the designated offset amount.
N2 G90 G00 C90.;	-20	**(a): 10** **(b): 20**	100	-15	**(a): 5** **(b): 15**	0	The offset direction varies by 90 degrees. Offset is applied by 5.0 mm in the X axis direction and -5.0 mm in the Y axis direction. (a) The machine moves by 10 mm in the negative direction of Y axis. (b) The machine does not move but the workpiece coordinate position varies.
N3 G90 G00 X-5. Y0.;	-5	0	100	0	-5	0	The machine moves by the changed offset amount in the X and Y axes directions while offset is applied.
G00 Z5.;	-5	0	5	0	-5	-95	
G01 Z0. F1000;	-5	0	0	0	-5	-100	
N4 G90 G01 X10.;	10	0	0	15	-5	-100	
N5 Y-20.;	10	-20	0	15	-25	-100	
N6 X0.;	0	-20	0	5	-25	-100	
N7 Y5.;	0	5	0	5	0	-100	
N8 G54.2 P0;	0	5	0	0	5	-100	Workpiece position offset for rotary axis is disabled. The machine moves by the canceled offset amount.

(*1) #5521 to #5525 are system variables of the workpiece position offset amount for rotary axis. For details, refer to "23.9 System Variables (Workpiece Position Offset Amount for Rotary Axis)".

(*2) This setting example is available when an error occurs 5.0 mm in the X axis direction and 5.0 mm in the Y axis direction while the angle of the rotary axis is set to 0 degree.

When the machine moves	When the machine does not move
	
The workpiece position offset amount for rotary axis varies.	When the workpiece position offset for rotary axis varies, no axis movement occurs.

When the machine does not move (#1597 is set to "0"), the position on the machine coordinate system does not change, but the position on the workpiece coordinate system changes.
When the absolute command (G90) is issued, the offset is applied to the end point of the next movement block.
When the incremental command (G91) is issued, the offset is not applied to the end point of the next movement block because the machine moves by the commanded incremental amount from the current position.

Operation at NC reset

Whether to cancel the offset amount when NC is reset depends on the MTB specifications (combination of the parameters "#1151 rstint" and "#1210 RstGmd/bit16").

Parameter setting	Operation at reset
#1151 = 1 and "#1210/bit16" = 0	The mode before the reset is canceled. (Same as "G54.2 P0".)
Other than the above	The mode before the reset is held.

The same operation is also performed for emergency stop.
Even if the setting is designed so that the axis moves when the workpiece position offset for rotary axis is changed, the axis does not move by the canceled offset amount when NC is reset.

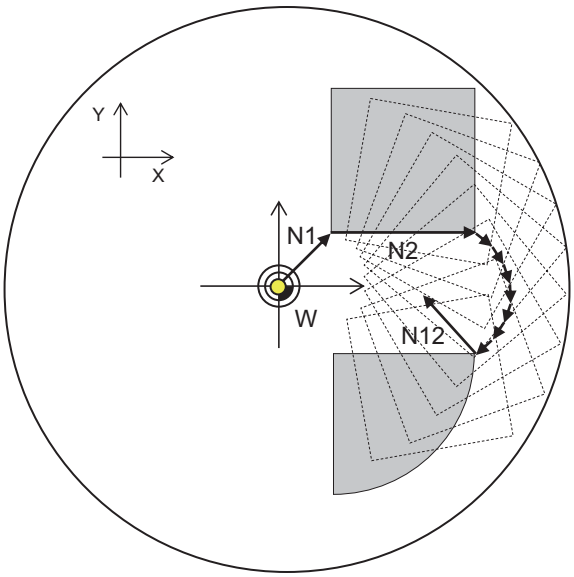
Operation example

The following describes the operation with a machining example of cutting in sector form by rotating a rotary table.
When there is an error between the workpiece coordinate zero point and the workpiece reference position, the workpiece position offset for rotary axis function enables machining in the same way as in the case with no installation error.
However, if C axis command is issued in one block, the correct machining shape may not be obtained because the offset is applied only to the block end point.

Program	Workpiece coordinate position			Machine position			Details of operation
	X	Y	Z	X	Y	Z	
G54;	0	0	100	0	0	0	Initial position
#5521 = 5.0; (*1)	0	0	100	0	0	0	Set the workpiece position offset amount for rotary axis. (*2)
#5522 = 5.0;	0	0	100	0	0	0	
#5523 = 0.0;	0	0	100	0	0	0	
#5524 = 0.0;	0	0	100	0	0	0	
#5525 = 0.0;	0	0	100	0	0	0	
N1 G54.2 P1;	0	0	100	5	5	0	Workpiece position offset for rotary axis is enabled. The machine moves by the designated offset amount.
G00 Z5.;	0	0	5	5	5	-95	The machine moves while the offset is applied by 5.0 mm to the X axis and Y axis respectively.
G01 Z0. F1000;	0	0	0	5	5	-100	
N2 G90 G01 X10.;	10	0	0	15	5	-100	
N3 C10.;	10	0	0	15.79	4.06	-100	The offset amount varies depending on the angle of the C axis. Accordingly, the machine moves in the X axis and Y axis directions.
N4 C20.;	10	0	0	16.41	2.99	-100	
N5 C30.;	10	0	0	16.83	1.83	-100	
N6 C40.;	10	0	0	17.04	0.62	-100	
N7 C50.;	10	0	0	17.04	-0.62	-100	
N8 C60.;	10	0	0	16.83	-1.83	-100	
N9 C70.;	10	0	0	16.41	-2.99	-100	
N10 C80.;	10	0	0	15.79	-4.06	-100	
N11 C90.;	10	0	0	15	-5	-100	
N12 G54.2 P0;	10	0	0	10	0	-100	Workpiece position offset for rotary axis is disabled. The machine moves by the canceled offset amount.

(*1) #5521 to #5525 are system variables of the workpiece position offset amount for rotary axis. For details, refer to "23.9 System Variables (Workpiece Position Offset Amount for Rotary Axis)".

(*2) This setting example is available when an error occurs 5.0 mm in the X axis direction and 5.0 mm in the Y axis direction while the angle of the rotary axis is set to 0 degree.



Relationship with other functions

Column A: Operation to be performed when the combination function is commanded during modal of this function (G54.2 Pn)

Column B: Operation to be performed when this function (G54.2 Pn) is commanded during modal of the combination function

x: Alarm generation (the text in parentheses refers to the number of the program error to be generated)

-: No combination

G code		Function name	(A)	(B)
G00		Positioning	(*1)	(*1)
G0.5		Rapid traverse block overlap	(*2)	(*2)
G01		Linear interpolation	(*3)	(*3)
G02/G03		Circular interpolation	(*4)	(*4)
G02/G03		Helical interpolation	(*4)	(*4)
G02.1/G03.1		Spiral interpolation	(*4)	(*4)
G02.3/G03.3		Exponential interpolation	× (P34)	× (P34)
G02.4/G03.4		3-dimensional circular interpolation	× (P34)	× (P34)
G04		Dwell	(*5)	-
G05	P0	High-speed mode cancel	(*5)	-
	P1	High-speed machining mode	33.7 m/min (*6)	33.7 m/min (*6)
	P2		168 m/min (*6)	168 m/min (*6)
	P10000	High-speed high-accuracy control II	168 m/min (*6)	168 m/min (*6)
	P20000	High-speed high-accuracy control III	270 m/min (*6)	270 m/min (*6)
G05.1	Q0	Cancel	(*5)	(*7)
	Q1	High-speed high-accuracy control I	67.5 m/min (*6)	67.5 m/min (*6)
	Q2	Spline interpolation	× (P34)	× (P34)
G06.2		NURBS interpolation	× (P34)	× (P34)
G07		Hypothetical axis interpolation	(*4)	(*4)
G07.1/G107		Cylindrical interpolation	× (P34)	× (P34)
G08		High-accuracy control	(*5)	(*5)
G09		Exact stop check	(*8)	(*8)

G code	Function name	(A)	(B)
G10	Data input by program	(*9)	× (P34)
G11	Data input by program cancel	(*5)	(*7)
G12/G13	Circular cutting	× (P34)	× (P34)
G12.1/G112	Polar coordinate interpolation ON	× (P34)	× (P34)
G13.1/G113	Polar coordinate interpolation OFF	(*5)	(*7)
G15	Polar coordinate command OFF	(*5)	(*7)
G16	Polar coordinate command ON	× (P34)	× (P34)
G17 - G19	Plane selection	(*9)	(*9)
G20/G21	Inch/metric	(*11)	(*11)
G22	Stroke check before travel ON	(*7)	(*7)
G23	Stroke check before travel cancel	(*5)	(*5)
G27	Reference position check	(*12)	-
G28	Reference position return	(*12)	-
G29	Start position return	(*12)	-
G30	2nd to 4th reference position return	(*12)	-
G30.1 - G30.6	Tool exchange position return	(*12)	-
G31	Skip	(*13)	(*13)
G31.1 - G31.3	Multi-step skip	(*13)	(*13)
G33	Thread cutting	× (P34)	-
G34	Special fixed cycle (bolt hole cycle)	(*7)	(*7)
G35	Special fixed cycle (line at angle)	(*7)	(*7)
G36	Special fixed cycle (arc)	(*7)	(*7)
G37	Automatic tool length measurement	(*12)	(*12)
G37.1	Special fixed cycle (grid)	(*7)	(*7)
G38	Tool radius compensation vector designation	(*14)	(*14)
G39	Tool radius compensation corner arc	(*14)	(*14)
G40	Tool radius compensation cancel	(*5)	(*7)
G41/G42	Tool radius compensation	(*14)	(*14)
G40.1/G150	Normal line control cancel	(*5)	(*7)
G41.1/G151 G42.1/G152	Normal line control	× (P34)	× (P34)
G41.2/G42.2	3-dimensional tool radius compensation	× (P34)	× (P34)
G43/G44	Tool length compensation	(*14)	(*14)
G43.1	Tool length compensation along the tool axis	× (P34)	× (P34)
G43.4/G43.5	Tool center point control	× (P34)	× (P34)
G43.7	Tool position compensation	× (P34)	× (P34)
G45 - G48	Tool position offset	(*14)	(*14)
G49	Tool length compensation cancel Tool length compensation along the tool axis cancel Tool center point control cancel Tool position compensation cancel	(*5)	(*7)
G50	Scaling cancel	(*5)	(*7)
G51	Scaling ON	× (P34)	× (P34)
G50.1	Mirror image by G code cancel	(*5)	(*7)
G51.1	Mirror image by G code ON	× (P34)	× (P34)
G52	Local coordinate system setting	(*15)	(*15)
G53	Basic machine coordinate system selection	(*12)	(*12)
G53.1/G53.6	Tool axis direction control	× (P34)	-

G code	Function name	(A)	(B)
G54 - G59	Workpiece coordinate system selection	(*15)	(*15)
G54.1	Extended workpiece coordinate system selection	(*15)	(*15)
G54.2	Workpiece position offset for rotary axis	(*16)	(*16)
G54.4	Workpiece installation error compensation	× (P34)	× (P34)
G60	Unidirectional positioning	(*4)	-
G61	Exact stop check mode	(*8)	(*8)
G61.1	High-accuracy control ON	(*5)	(*5)
G61.2	High-accuracy spline	× (P34)	× (P34)
G61.4	Spline interpolation II	× (P34)	× (P34)
G62	Automatic corner override	(*5)	(*5)
G63	Tapping mode	(*5)	(*5)
G64	Cutting mode	(*5)	(*5)
G65	User macro simple call	(*4)	(*17)
G66/G66.1	User macro modal call	(*4)	(*17)
G67	User macro modal call cancel	(*5)	(*7)
G68	Coordinate rotation by program ON	× (P34)	× (P34)
G68.2/G68.3	Inclined surface machining command	× (P34)	× (P34)
G69	Coordinate rotation by program OFF Inclined surface machining command OFF	(*5)	(*7)
G70 - G89	Fixed cycle	(*7)	-
G90	Absolute command	(*15)	(*15)
G91	Incremental command	(*15)	(*15)
G92	Coordinate system setting	(*15)	(*15)
G92.1	Workpiece coordinate system preset	(*15)	(*15)
G93	Inverse time feed	(*5)	(*5)
G94	Feed per minute	(*5)	(*5)
G95	Feed per revolution	(*5)	(*5)
G96	Constant surface speed control ON	× (P34)	× (P34)
G97	Constant surface speed control OFF	(*5)	(*7)
G98	Fixed cycle (Initial level return)	(*18)	-
G99	Fixed cycle (R point level return)	(*18)	-
G100 - G225	User macro	(*4)	(*17)
G120.1	Machining condition selection I	(*5)	(*5)
G121	Machining condition selection I cancel	(*5)	-
G122	Sub part system I	× (P34)	× (P34)
G127	Prohibit manual arbitrary reverse run	× (P34)	× (P34)
G140	Arbitrary axis exchange command	× (P34)	× (P34)
G141	Arbitrary axis exchange return command	× (P34)	× (P34)
G142	Arbitrary axis exchange/reference axis arrange return command	× (P34)	× (P34)
G145	Sub part system control cancel	× (P34)	× (P34)
G160	Torque limitation skip	(*5)	(*5)
G186	Interference check III interfering object data enable command	(*5)	(*14)

(*1) Operates at rapid traverse rate.

(*2) Operates at rapid traverse overlap.

(*3) Operates at cutting feedrate.

(*4) The function described at the left is performed in a position to which offset is applied.

(*5) The function described at the left is performed.

(*6) Micro segment processing capability in G01 block of 1 mm line segment is described.

The performance described here applies under the following conditions.

- 6-axis system (including spindle) or less
- 1-part system
- 3 axes or less commanded simultaneously in G01
- Block containing only axis name and movement amount (Does not contain macro or variable command)
- During G61.1 high-accuracy control mode or during cutting mode (G64)
- During tool radius compensation cancel (G40) (only for high-speed machining mode II)
- Parameter "#1259 set31/bit1" is set to "1"
  (The number of process blocks per unit time is configured at low speed mode.)

Depending on the combination with other functions, operation may not performed as shown in the table.

(*7) The function described at the left is performed at the position after offset.

(*8) A deceleration check is performed at the block end point after offset.

(*9) The function is enabled at the next G54.2 command.

(*10) Operates on a selected plane.

(*11) Operates in a commanded unit system. Offset amount is shown in an original unit system.

(*12) Operates while the offset is canceled temporarily.

(*13) Skips at "skip" signals.

(*14) Offset is applied to the position after the operation (or to the state after the operation) of the function described at the left.

(*15) Offset is applied in the commanded coordinate system.

(*16) Offset amount for the sets commanded with P is applied.

(*17) Offset is applied.

(*18) Returns to the initial point or the R point after offset.



Precautions

(1) If any of the parameters related to the workpiece position offset for rotary axis or the workpiece position offset amount for rotary axis is changed during G54.2 mode, the change is applied after G54.2 is commanded next time. However, when "#2100 rpcax" of an axis is disabled during G54.2 mode, the workpiece position offset for rotary axis is disabled for the disabled axis; thus, the operation is performed with the offset amount when the function is enabled until it is canceled. For example, under the condition that the workpiece position offset for rotary axis is enabled for all the X, Y and Z axes, when it is disabled only for the X axis during execution of "G54.2 P1", only the X axis does not move with the offset amount of "G54.2 P2" but with that of "G54.1 P1" even when "G54.2 P2" is commanded. When "G54.2 P0" is commanded, all the X, Y and Z axes return to the workpiece coordinate system.

(2) For M80 and M830 Series, when commands are given to two or more rotary axes simultaneously, the program error (P10) may occur because linear axes also move according to the offset.

(3) When "The machine does not move." (*1) is selected, designate the absolute position with G00 or G01 after the G54.2 command, and perform positioning. No offset is applied with the incremental command. For the circular command, offset is only applied to the end point, so a program error (P70 or P71) may occur.

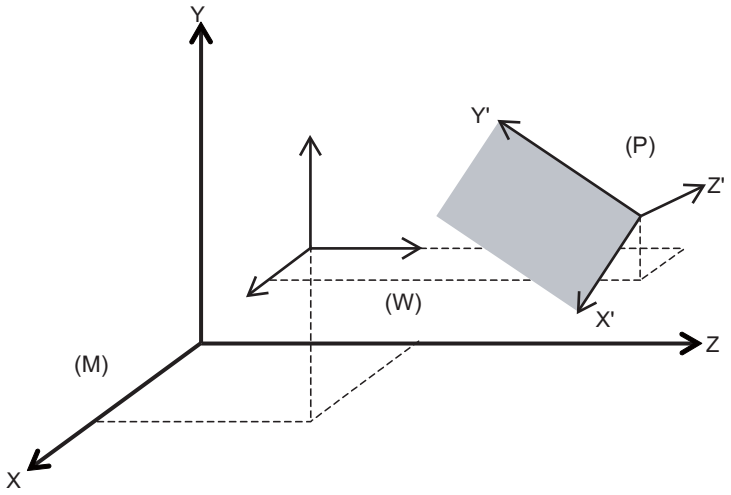
(*1) This setting depends on the MTB specifications (the parameter "#1250 set22/bit7"). Refer to "Operation Description" for details.

(4) When the rotary axis is moved by manual interrupt operation during G54.2 mode, the workpiece position offset amount for rotary axis is unchanged.
The offset amount is calculated when the rotary axis travel command or G54.2 command is given in a program. When the "Manual absolute" signal (ABS) is "1" (ON), and if the rotary axis is moved by an incremental command after being moved by manual interruption, the axis moves to the angle (end point) where the travel amount of the rotary axis by manual interruption is included, and the offset is updated according to the angle.

20.9 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 3-dimensional coordinate conversion is commanded, 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) Set only one of the three axes to "1". Set the other two axes to "0".
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°.

3-dimensional coordinate conversion mode cancel command

```
G69 ;
```



Detailed description

(1) Command the rotation center coordinates with the absolute command.

(2) If the rotation center coordinates are omitted, the zero point of the currently set coordinate system will be the rotation center coordinates.

(3) In the 3-dimensional coordinate conversion, the addresses "I", "J" and "K" cannot be omitted, and all of them need to be specified.
When one or two of "I", "J" and "K" are not commanded, the program error (P926) occurs.
When none of "I", "J" and "K" is commanded, this function operates as coordinate rotation by program.

(4) Set "1" in only one of "I", "J" and "K", and set "0" for the other two.
A program error (P33) will occur if "1" is set in two or more.

(5) A program error (P33) will also occur if "0" is set for all addresses "I", "J" and "K".

(6) 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".

(7) If a G code that cannot be commanded in the 3-dimensional coordinate conversion modal is issued, a program error (P921) will occur. When 3-dimensional coordinate conversion is commanded during the modal where 3-dimensional coordinate conversion cannot be carried out, a program error (P922) will occur. For details, refer to "Relationship with other functions".

(8) Issue this G code alone in a block. If another G code command is issued to the same block as that of the G68 command, a program error (P923) will occur.

(9) The 3-dimensional coordinate conversion command for the rotary axis will result in a program error (P32).

(10) If a 3-dimensional coordinate conversion command is issued when there are no specifications for 3-dimensional coordinate conversion, a program error (P920) will occur.

(11) Addresses "I", "J" and "K" for determining the rotation center axis refer to the axes set to basis axis I (#1026 base_I), basis axis J (#1027 base_J), and basis axis K (#1028 base_K) respectively. Designate the correct basis axes I, J and K. If these settings are incorrect, a program error (P610) 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.
Therefore, 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 after the conversion command is issued. (If G52 is issued, a program error (P921) will occur.)

(5) After the conversion command is issued, the G68 program coordinate system can be canceled either by the G69 command or reset input. (However, when the parameter "#1151 rsint" is set to "0", the 3-dimensional coordinate conversion mode will be kept.)

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 rotation center coordinate of this G code 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 command either the absolute modal or the incremental modal (G90 or G91) is being executed at that point.

(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. (This is equivalent to simply setting "0".)

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.
Therefore, 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.)

(3) Even if multiple commands are issued, all 3-dimensional coordinate conversion commands are canceled by executing the 3-dimensional coordinate conversion cancel command (G69) once.

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.

[1st 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)

[2nd 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

Manual operations (jog feed, incremental feed, and manual handle feed) during the 3-dimensional coordinate conversion modal can be performed in the coordinate system (G68 program coordinate system) after the 3-diensional coordinate conversion command was issued.

When the machining stops during 3-dimensional coordinate conversion, operations such as a pulling operation by manual feed can be performed in this program coordinate system.

However, when turning OFF the power stops the machining, whether or not the manual feed is executed by the G68 program coordinate system before the power was turned OFF depends on the MTB specifications (parameter "#1754 cfgPR04/bit1").
To perform manual operation during oblique hole machining with the G68 program coordinate system command, set the "3-dimensional coordinate conversion or Coordinate rotation by program: Manual feed coordinates switch input" signal to ON to feed the tool in the Z axis direction, which moves in the tool axis direction. This action is called "manual feed for 3-dimensional coordinate conversion".

(1) Coordinate switching enable conditions

A manual operation coordinate change by the "3-dimensional coordinate conversion or Coordinate rotation by program: Manual feed coordinates switch" signal (YD14) is available only when the "3D coordinate conversion: Manual feed valid" signal (XD14) is set to ON. (The operation of the PLC signal depends on the MTB specifications.)
The manual operation coordinate change by the "3-dimensional coordinate conversion or Coordinate rotation by program: Manual feed coordinates switch" signal becomes valid after three basic axes have stopped. When the "3-dimensional coordinate conversion or Coordinate rotation by program: Manual feed coordinates switch" signal is switched while even one of three basic axes is moving, a coordinate change is performed after three basic axes have stopped.
The "3D coordinate conversion: Manual feed valid" signal 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 at least once.

However, if the signal is canceled by the G69 command, the signal does not turn 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

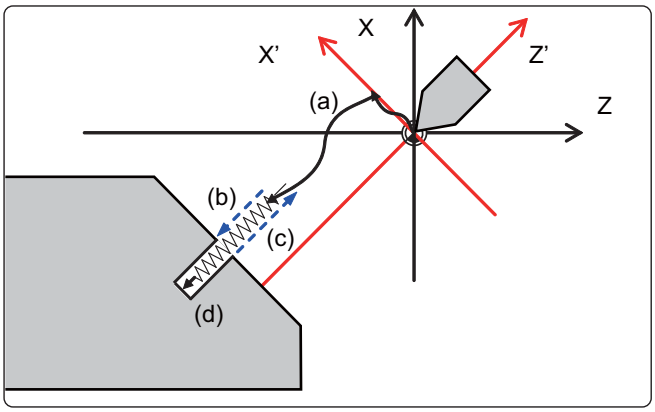
(3) Set the handle mode, then select the Z axis with the 1st handle. Then, check that the "3D coordinate conversion: Manual feed valid" signal is set to ON.

(4) Set the "3-dimensional coordinate conversion or Coordinate rotation by program: Manual feed coordinates switch" signal to ON.

(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" signal (Y920) 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 movement amount by handle feed conforms to that on the G68 program coordinate system. The movement amount on the G68 program coordinate system becomes the integral multiple of the handle magnification.

(e) When the handle feed is selected, if the one-scale movement amount by manual feed for 3-dimensional coordinate conversion exceeds the movement amount for a specified time at clamp speed, the operation error (M01 0060) occurs at the time of pulse occurrence, not at the time of handle axis selection, and thus, the tool cannot be moved. To move the tool, reduce the handle magnification.

(f) In the reference position return mode, operation is made on the machine coordinate system. The operation is not made on the coordinate system after the coordinate rotation.

(g) 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 0140) will occur. To use the tool escape mode, set the "3-dimensional coordinate conversion or Coordinate rotation by program: Manual feed coordinates switch" signal to OFF.

(h) This function is not compatible with the manual tool length measurement function and workpiece position measurement function based on the manual feed for 3-dimensional coordinate conversion. If it is started, an operation error (M01 0140) will occur. While the manual tool length measurement function or workpiece position measurement function is active, turn the "3-dimensional coordinate conversion or Coordinate rotation by program: Manual feed coordinates switch" signal to OFF.

(i) When the "Manual/Automatic simultaneous valid" signal (Y920) is set to ON for any of three basic axes, operation is performed in the same way as when the "Manual/Automatic simultaneous valid" signal (Y920) for three basic axes is set to ON.

(j) When the "Manual machine lock" signal (Y8C0) is set to ON for any of three basic axes, operation is performed in the same way as when the "Manual machine lock" signal (Y8C0) for three basic axes is set to ON.

(k) 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.

(l) 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 the "3-dimensional coordinate conversion or Coordinate rotation by program: Manual feed coordinates switch" signal.

(m) If the G69 command exists up to the block to be restart-searched at program restart, the "3-dimensional coordinate conversion or Coordinate rotation by program: Manual feed valid" signal (XD14) is turned OFF as the G69 command does.

(n) In the 3-dimensional coordinate conversion manual-feed mode, two or more axes of three basic axes cannot be moved simultaneously. If they are moved simultaneously, an operation error (M01 0141) will occur.

(o) Even when the parameter "#1754 cfgPR04/bit1" is set to "1", if any of the following conditions is satisfied, the manual feed after the power was turned OFF operates in the machining coordinate system.

- The parameter "#1753 cfgPR03/bit3" is "0" and the G68 command part system is under arbitrary axis exchange control.

(4) When the axis configuration is different from that at 3-dimensional coordinate conversion command, even if the mode is switched to jog, handle or incremental mode, the manual feed for 3-dimensional coordinate conversion is not available. The manual feed is executed on the machine coordinate system.
The axis configuration is different from that at 3-dimensional coordinate conversion command under the following conditions.

- The arbitrary axis exchange control (*1) is commanded after the 3-dimensional coordinate conversion modal is cleared.
- The 3-dimensional coordinate conversion is commanded while the arbitrary axis exchange control is executed, then reset or emergency stop (*2) is performed.

(*1) In arbitrary axis exchange control, even when the axis configuration is the same as that at 3-dimensional coordinate conversion command, the manual feed is executed on the machine coordinate system.

(*2) When the parameter is set to retain the axis exchange status at reset or emergency stop, the manual feed is available after 3-dimensional coordinate conversion.

[When 3-dimensional coordinate conversion is commanded during arbitrary axis exchange control]

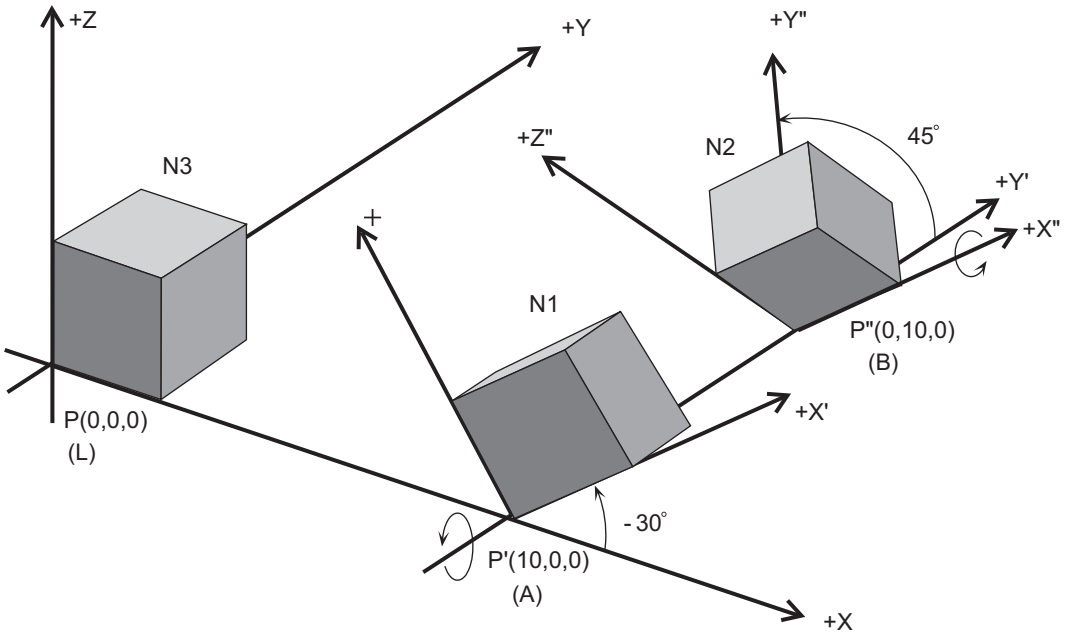
#1280 ext16/bit1	#1753 cfgPR03/bit3	Operation	Coordinate system in manual operation
0 (Restores the reference axis arrange)	0 (Restores the reference axis arrange)	Reset	Machine coordinate system
		Emergency stop	
	1 (Retains the axis exchange status)	Reset	Machine coordinate system
		Emergency stop	G68 program coordinate system
1 (Retains the axis exchange status)	0 (Restores the reference axis arrange)	Reset	G68 program coordinate system
		Emergency stop	Machine coordinate system
	1 (Retains the axis exchange status)	Reset	G68 program coordinate system
		Emergency stop	

The parameters "#1280 ext16/bit1" (Mixed control cancel with reset) and "#1753 cfgPR03/bit3" (Retention of arbitrary axis exchange control after emergency stop/power off) depend on the MTB specifications.

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° 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.

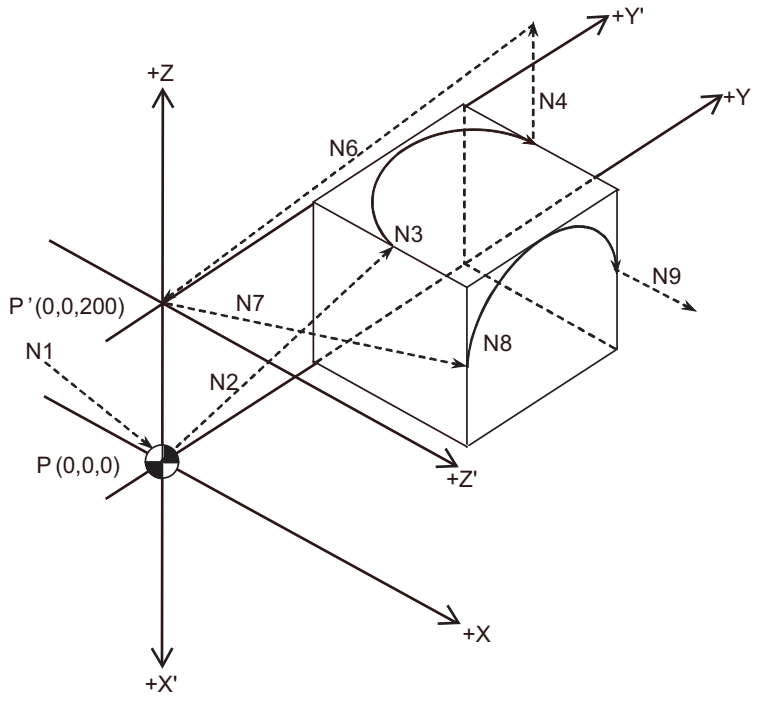
Program example 2

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).

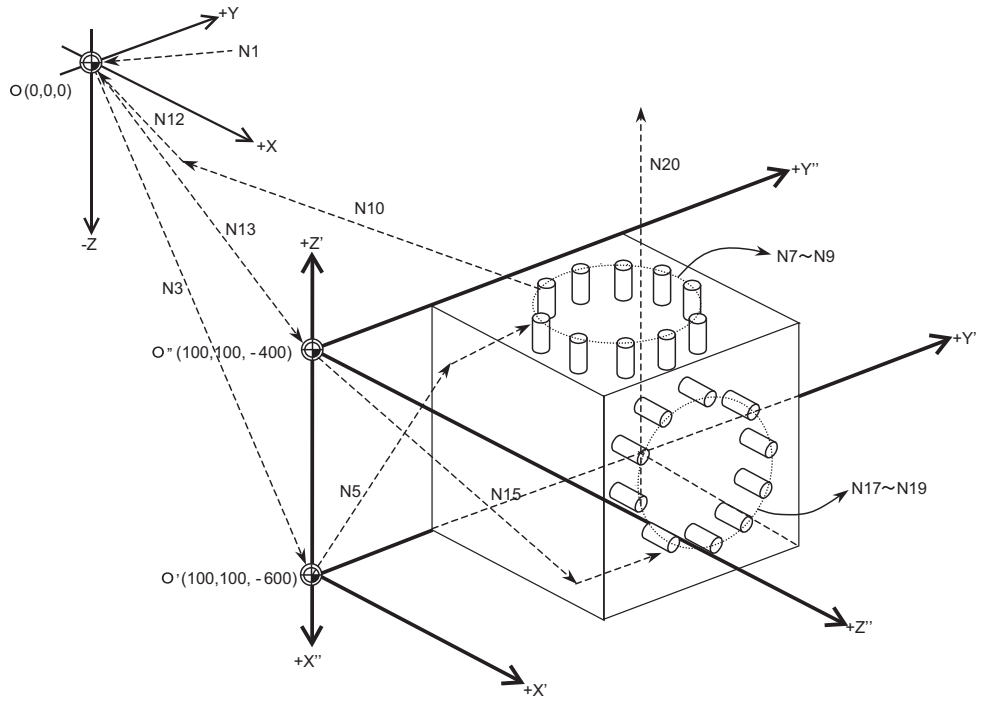
N01 G17 G90 G00 X0 Y0 Z0;	Position to the workpiece zero point P.
N02 G00 X100. Y200. Z200.;	Move to (100, 200, 200) with rapid traverse.
N03 G02 X100. Y400. J100. F1000;	Carry out arc cutting on workpiece top.
N04 G00 Z300.;	Escape +100 in +Z direction with rapid traverse.
N05 G68 X0 Y0 Z200. I0 J1 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) with rapid traverse.
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 with rapid traverse.
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 with rapid traverse.
N06 G91;	Incremental 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 with rapid traverse.
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 with rapid traverse.
N16 G91;	Incremental 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 with rapid traverse.
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) Automatic reference position return

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 exchange position return

3-dimensional coordinate conversion is not carried out for the tool change position even if a G30.n command 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 the tool length compensation, tool radius compensation or tool position compensation is executed 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 the mirror image command is issued 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.

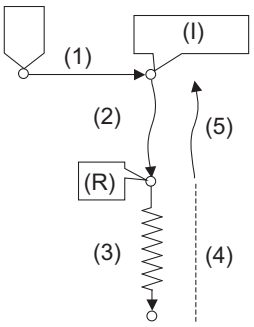
However, the fixed cycle hole drilling rapid traverse rate during the 3-dimensional coordinate conversion modal is switched as shown below by the settings of the parameters "#15663 DselctDrillaxMode" and "#1564 3Dspd". (This depends on the MTB specifications.)

Fixed cycle rapid traverse rate during 3-dimensional coordinate conversion

#1566	0 (Rapid traverse mode)	1 (Cutting mode)	
#1564	-	0	1 to 1000000
Rapid traverse rate	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 at (1) in the figure below 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 rate 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

Each operation shown in the figure is set to compensation operation regardless of the setting of the parameter #1086 (G00 non-interpolation).

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	○	×	○	○
G74	○	×	×	○
G75	○	×	○	○
G76	○	×	○	○
G81	○	×	○	○
G82	○	×	○	○
G83	○	×	○	○
G84	○	×	×	○
G85	○	×	×	○
G86	○	×	○	○
G87	○	○	×	○
G88	○	×	○	○
G89	○	×	×	○

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 3-dimensional 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 improvement) 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 3-dimensional coordinate conversion mode, regardless of the values of #1566 (Switch drill axis's mode from rapid traverse during 3-dimensional coordinate conversion) and #1564 (Hole drilling cycle during 3-dimensional 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), a program error (P32) will occur.

(13) Initial constant surface speed

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.
However, for circular interpolation in the 3-dimensional coordinate conversion modal, when the "Interlock" signal is set to ON for any one of the three basic axes (length, width, and height), the axis is interlocked even if the movement is not commanded for the axis.

(16) Coordinate read variable

To read the workpiece coordinate system/skip coordinate system during the 3-dimensional coordinate conversion modal, the 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". This is also applied to the absolute position display of the special display unit.

<Note>

- 1µm of display deviation may occur during 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>

- A display deviation by 1 um may occur during 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) External deceleration

The external deceleration in the 3-dimensional coordinate conversion modal will be valid for the movement axis for the coordinate value after executing 3-dimensional coordinate conversion.
However, for the external deceleration of each axis in the circular interpolation during the 3-dimensional coordinate conversion modal, when the "External deceleration" signal is set to ON for any one of the three basic axes (length, width, and height), the external deceleration speed is applied to the axis even if the movement is not commanded for the axis.

(21) Manual operation in G68 program coordinate system

Refer to "Manual Operation in G68 Program Coordinate System" in "Detailed description".

(22) Multiple-axis synchronization control

3-dimensional coordinate conversion for master axis and slave axis of the multiple-axis synchronization control is not supported. Do not command the 3-dimensional coordinate conversion.

(23) Others

(a) 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 command) mode has to be issued.

(Example)

```
G68 X50. Y100. Z150. I1 J0 K0 R60. ;
G90 G00 X0 Y0 Z0 ;
G41 D01 ;
      . . . . . .
G40 ;
G69 ;
```

(b) The G00 command during the 3-dimensional coordinate conversion modal is set to the interpolation type regardless of the setting of the basic parameter "#1086 G0Intp" (G00 non-interpolation).

(c) Origin zero cannot be executed during the 3-dimensional coordinate conversion modal.

(d) 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

o: Can be commanded.

Δ: Cannot be commanded. (However, this is not assumed to be a program error.)

-: This is an unmodal command, so not targeted.

G command	Function	A	B	C
G00	Positioning	○	○	P923
G01	Linear interpolation	○	○	P923
G02	Circular interpolation CW	○	○	P923
	Helical interpolation CW	P921	○	P923
G03	Circular interpolation CCW	○	○	P923
	Helical interpolation CCW	P921	○	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	○	-	P45
G05 P0	High-speed machining mode cancel	○	-	P923
G05 P2	High-speed machining mode II	P921	P922	P45
G05 P3	High-speed machining mode III	P34	P34	P923
G05 P10000	High-speed high-accuracy control II	P921	P922	P923
G05 P20000	High-speed high-accuracy control III	P921	P922	P33
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	○	-	P45
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
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	○	○	○

G command	Function	A	B	C
G21	Metric command	○	○	○
G27	Reference position check	○	-	P45
G28	Automatic reference position return	○	-	P45
G29	Start position return	○	-	P45
G30	2nd to 4th reference position return	○	-	P45
G30.1	Tool exchange position return 1	○	-	P33
G30.2	Tool exchange position return 2	○	-	P33
G30.3	Tool exchange position return 3	○	-	P33
G30.4	Tool exchange position return 4	○	-	P33
G30.5	Tool exchange position return 5	○	-	P33
G30.6	Tool exchange position return 6	○	-	P33
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	-	P45
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
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

G command	Function	A	B	C
G50.1	Mirror image cancel	○	-	P923
G51.1	Mirror image ON	○	○	P923
G52	Local coordinate system setting	P921	-	P45
G53	Machine coordinate system setting	○	-	P45
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
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	○	○	○
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

G command	Function	A	B	C
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/P926 (*1)	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 command	○	○	○
G91	Incremental command	○	○	○
G92	Coordinate system setting	P921	-	P923
	Spindle speed clamp	P921	○	P923
G92.1	Workpiece coordinate preset	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)	○	○	○
G115	Start point designation synchronization (Type 1)	○	○	P923
G116	Start point designation synchronization (Type 2)	○	○	P923

(*1) The program error differs depending on the setting value of the parameter "#1271 ext07/bit1".
When "#1271 ext07/bit1" is "0", the program error (P922) occurs.
When "#1271 ext07/bit1" is "1", the program error (P926) occurs.

Note

(1) All the G codes that are not listed in the table are not available.

Precautions

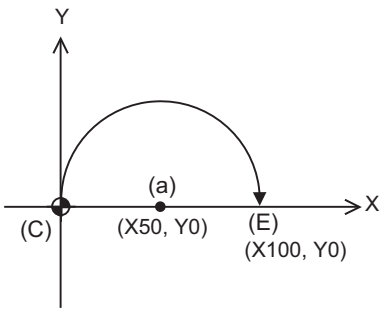
Precautions related to arc command

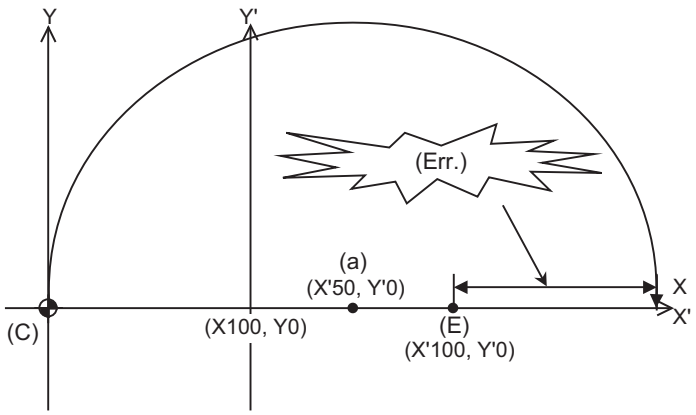
If the first command after the 3-dimensional coordinate conversion command is an arc shape, and the center of the arc did not change before and after the 3-dimensional coordinate conversion, an arc is drawn. However, an error will occur in the following cases:

(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



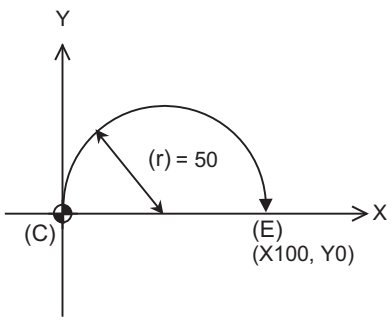


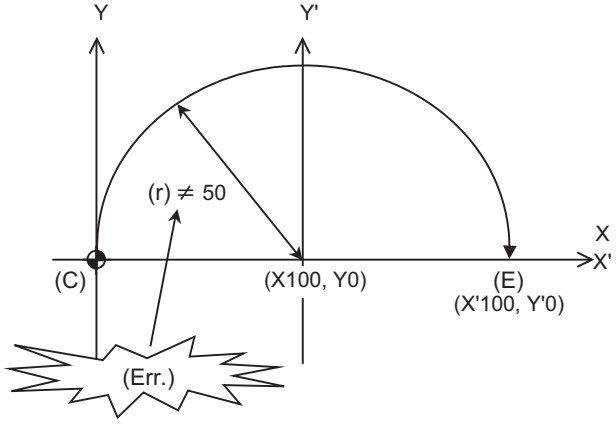
(a) Arc center (C) Current position (E) End point (Err.) Program error

(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

20.10 Coordinate Rotation by Program; G68/G69

Function and purpose

Coordinate rotation by program is used to rotate the programmed shape by the specified angle and to machine it. Even complex shapes can be easily machined in the rotated position.

Command format

Coordinate rotation ON

G68 X__ Y__ R__ ;

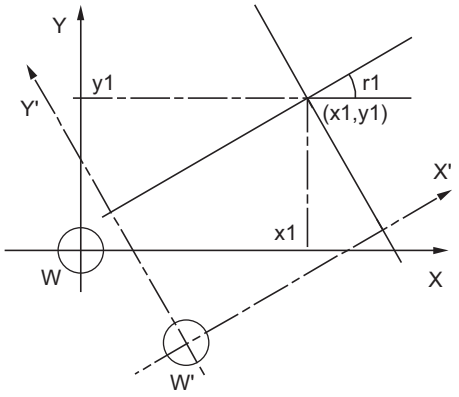
X, Y	Rotation center coordinates Two axes (X, Y or Z) corresponding to the selected plane are designated with absolute positions.
R	Rotation angle Designate "+" for the counterclockwise direction, and "-" for the clockwise direction.

Select the plane to be rotated by G17 to G19 commands before G68 command.

Coordinate rotation cancel

G69 ;

The following figure indicates the operation when the command "G68 Xx1 Yy1 Rr1;" is issued.



W: Local coordinate system before the rotation
W': Local coordinate system after the rotation
r1: Rotation angle
(x1, y1) Rotation center

Detailed description

(1) Always command the rotation center coordinate with an absolute position. Even if commanded with an incremental command address, it will not be handled as an incremental position. The rotation angle command (R command) value depends on the G90/G91 modal.

(2) If the rotation center coordinates are omitted, the position where the G68 command was executed will be the rotation center.

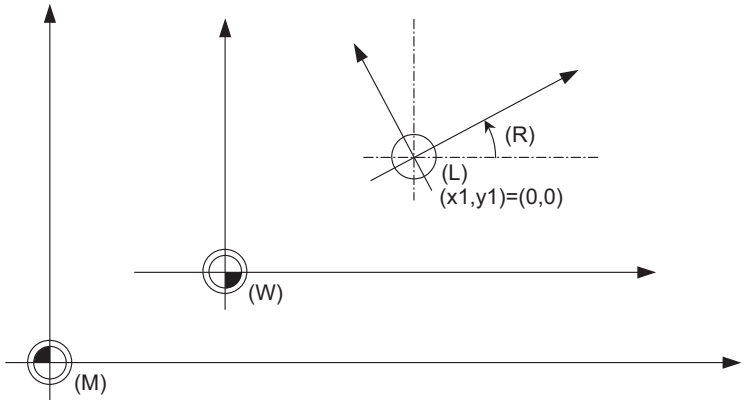
(3) The coordinate rotates counterclockwise by the angle designated in rotation angle command (R command).

(4) The setting range of rotation angle command (R command) is -360.000 to 360.000. When a value out of the setting range is commanded, a remainder of the value divided by 360 degrees (°) is commanded.

(5) Since the rotation angle is modal data, if once commanded, it will not be changed until the new angle is commanded. Thus, the command of rotation angle command (R command) can be omitted.

If the R command is omitted when G68 is commanded for the first time, the rotation angle becomes 0 degrees (°).

(6) Coordinate rotation by program is commanded in the local coordinate system. The relationship between the rotated coordinate system, the workpiece coordinate system, and the basic machine coordinate system is shown in the figure below.
Additionally, if the G52 command (local coordinate system setting) is not issued in the currently selected workpiece coordinate system, the position on the local coordinate system (L) will be the same as that on the workpiece coordinate system (W).



(R) Rotation angle
(L) Local coordinate system
(W) Workpiece coordinate system
(M) Basic machine coordinate system

(7) When the command for coordinate rotation by program is issued during coordinate rotation, it 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.)

(10) Coordinate rotation by program is valid only in the automatic operation mode.

Manual feed on local coordinate system

The coordinate system for manual feed is any of the following:

- Local coordinate system (coordinate system where coordinate rotation is applied)
- Machine coordinate system (coordinate system where coordinate rotation is not applied)

When manual feed is performed in the axis direction of the coordinate system where coordinate rotation is applied since the machining during the coordinate rotation by program stops due to power outage, emergency stop, etc., select the local coordinate system.

When the parameter "#1754 cfgPR04/bit4" (Coordinate rotation by program: manual feed function selection) is "1", the machine coordinate system or local coordinate system can be selected by the "3-dimensional coordinate conversion or coordinate rotation by program: Coordinate system for manual feed" signal (YD14).

When all the following conditions are satisfied, the coordinate system is switched to the one selected by this signal (YD14).
(The parameter and signal switch operation depend on the MTB specifications.)

- The coordinate rotation by program is commanded at least once.
  (Cancellation is not performed by G69 command.)
- The jog, handle or incremental mode is selected.
- All the three basic axes have stopped.
  (If any of the three basic axes is moving, it is switched after the axis stops.)

Precautions for manual feed

(1) The speed limit is applied so that the speed distributed to the movement amount in each axis direction does not exceed the clamp speed of each axis.

(2) 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.

(3) The movement amount by handle feed conforms to that on the local coordinate system. The movement amount on the local coordinate system becomes the integral multiple of the handle magnification.

(4) In handle feed, if the one-scale movement amount by manual feed for coordinate rotation by program exceeds the movement amount for a specified time at clamp speed, the operation error (M01 0060) occurs at the time of pulse occurrence, not at the time of handle axis selection, and thus, the tool cannot be moved. To move the tool, reduce the handle magnification.

(5) In the reference position return mode, operation is made on the machine coordinate system. The operation is not made on the coordinate system after the coordinate rotation.

(6) The manual feed operation for coordinate rotation by program is not available in tool retract and return mode. If it is started, the operation error (M01 0140) occurs. To use the tool retract and return mode, set the "3-dimensional coordinate conversion or Coordinate rotation by program: Manual feed coordinates switch" signal (YD14) to OFF.

(7) This function is not compatible with the manual tool length measurement function and workpiece position measurement function based on the manual feed for coordinate rotation by program. If it is started, the operation error (M01 0140) occurs. While the manual tool length measurement function or workpiece position measurement function is active, set the "3-dimensional coordinate conversion or Coordinate rotation by program: Manual feed coordinates switch" signal (YD14) to OFF.

(8) When the "Manual/Automatic simultaneous valid" signal (Y920) is set to ON for any of three basic axes, operation is performed in the same way as when the "Manual/Automatic simultaneous valid" signal (Y920) for three basic axes is set to ON.

(9) When the "Manual machine lock" signal (Y8C0) is set to ON for any of three basic axes, operation is performed in the same way as when the "Manual machine lock" signal (Y8C0) for three basic axes is set to ON.

(10) 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.

(11) Even if the coordinate rotation by program modal state is canceled by reset, etc., the manual feed is possible in the local coordinate system before the modal is canceled.

(12) If the G69 command exists up to the block to be restart-searched at program restart, it also turns OFF the "3-dimensional coordinate conversion or coordinate rotation by program: Manual feed valid" signal (XD14) as the G69 command usually does.

(13) In the manual feed for coordinate rotation by program, two or more axes of three basic axes cannot be moved simultaneously.
If they are moved simultaneously, the operation error (M01 0141) occurs.

(14) Even when the parameter "#1754 cfgPR04/bit1" is set to "1", if any of the following conditions is satisfied, the manual feed after the power was turned OFF operates in the machine coordinate system.

- The parameter "#1753 cfgPR03/bit3" is "0" and the G68 command part system is under arbitrary axis exchange control.

(15) To initialize zero point because it is lost, turn OFF the "3-dimensional coordinate conversion or Coordinate rotation by program: Manual feed coordinates switch" signal (YD14), and perform the manual feed on the machine coordinate system.

(16) When the axis configuration is different from that at coordinate rotation by program, even if the mode is switched to jog, handle or incremental mode, the manual feed for coordinate rotation by program is not available. The manual feed is executed on the machine coordinate system.
The axis configuration is different from that at coordinate rotation by program under the following conditions.

- The arbitrary axis exchange control (*1) is commanded after the coordinate rotation by program is commanded.
- The coordinate rotation by program is commanded while the arbitrary axis exchange control is executed, then reset or emergency stop (*2) is performed.

(*1) Even when arbitrary axis exchange control is commanded so that the axis configuration is the same as that at coordinate rotation by program, the manual feed is executed on the machine coordinate system.

(*2) When the parameter is set to retain the axis exchange status at reset or emergency stop, the manual feed is available after coordinate rotation by program.

[When coordinate rotation by program is commanded during arbitrary axis exchange control]

#1280 ext16/bit1	#1753 cfgPR03/bit3	Operation	Manual operation after "Operation"
0 (Restores the reference axis arrange)	0 (Restores the reference axis arrange)	Reset	Machine coordinate system
		Emergency stop	
	1 (Retains the axis exchange status)	Reset	Machine coordinate system
		Emergency stop	Local coordinate system
1 (Retains the axis exchange status)	0 (Restores the reference axis arrange)	Reset	Local coordinate system
		Emergency stop	Machine coordinate system
	1 (Retains the axis exchange status)	Reset	Local coordinate system
		Emergency stop	

The parameters "#1280 ext16/bit1" (Mixed control cancel with reset) and "#1753 cfgPR03/bit3" (Retention of arbitrary axis exchange control after emergency stop/power off) depend on the MTB specifications.

(17) Only when the coordinate rotation by program is commanded while the plane is configured by setting the parameters "#1026 Base axis I", "#1027 Base axis J" and "#1028 Base axis K", the manual feed is executed on the local coordinate system.

(Example) When the parameters are set as follows and the coordinate rotation by program is commanded in G18 modal (X-Z plane), the manual feed is executed on the local coordinate system where coordinate rotation is applied. The plane cannot be configured in G17/G19 mode; therefore, even when the coordinate rotation by program is commanded, the manual feed is executed on the machine coordinate system.

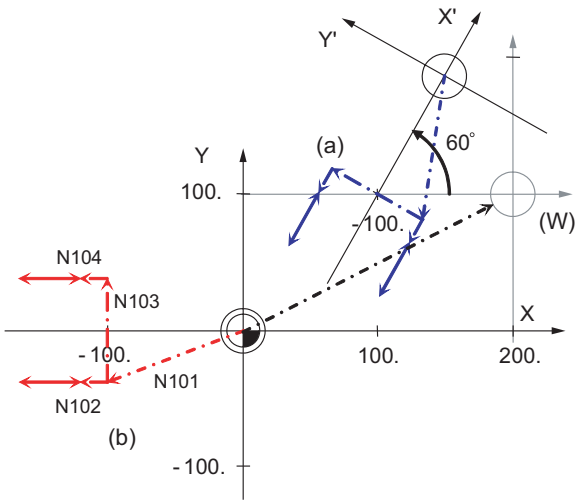
- The parameters "#1026 Base axis I" is set to "X", "#1027 Base axis J" is set to "0", and "#1028 Base axis K" is set to "Z".

Program example

Absolute command for coordinate rotation by program

N01 G28 X0. Y0.;
N02 G54 G52 X200. Y100. ; Local coordinate designation
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
(b) Program coordinate
(W) Local coordinates (before rotation)

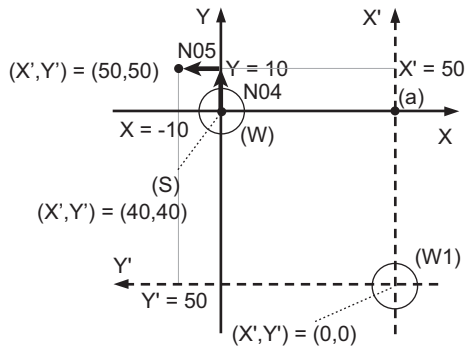
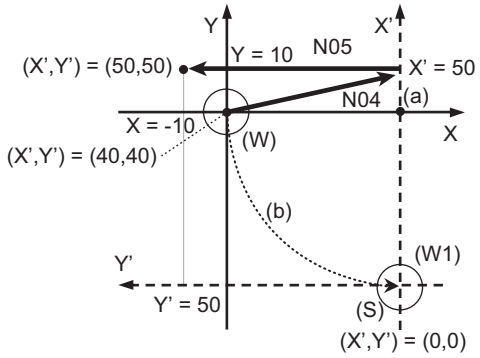
Operation immediately after start of coordinate rotation

Immediately after the coordinate rotation is commanded or the coordinate system offset during coordinate rotation is switched, it is recommended to simultaneously command absolute positions of two axes on the rotation plane as the first destination. The same program can be used in machines where the parameter settings are different.
When one axis is commanded immediately after the coordinate rotation command, the operation is as follows after the coordinate system offset during coordinate rotation is switched. Select the operation by the parameter "#19003 PRG coord rot type".

When the absolute position of only one axis on the rotation plane is commanded for the first movement after the coordinate rotation command, the operation is as follows.

(1) When "#19003 PRG coord rot type" is "0", the start position (S) is the position at the time of coordinate rotation command. The command position (X') on the rotation coordinate system is calculated by N04, and only the command axis is moved to the calculated position.
(2) When "#19003 PRG coord rot type" is "1", the operation is the same as when N04 is "X50.Y0.". The start position (S) is the position where the current position is rotated at the same time as the coordinate rotation command, and the end point position is calculated using it.

```
N01 G17 G28 X0. Y0.;
N02 G90 G92 G53 X0. Y0.;
N03 G68 X40. Y0. R90.;          Coordinate rotation ON
N04 G00 X50.;
N05 Y50.;
N06 G69;                        Coordinate rotation cancel
N07 M02 ;                       End
```

#19003 = 0	#19003 = 1
	

⟶ Machine movement path
(S) Start point
(a) Center of rotation
(b) The start point is rotated virtually
(W) Local coordinate system before rotation
(W1) Local coordinate system after rotation

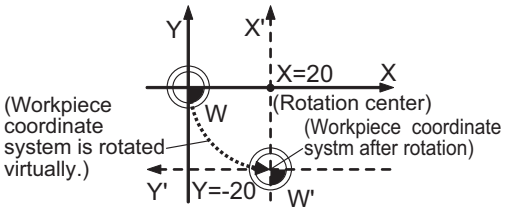
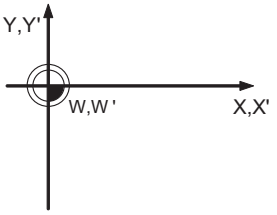
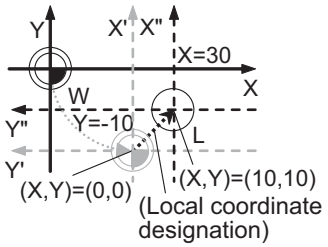
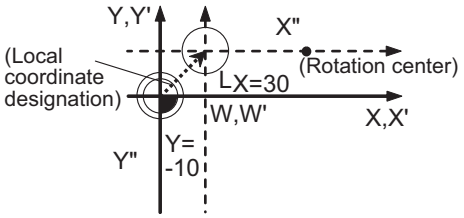
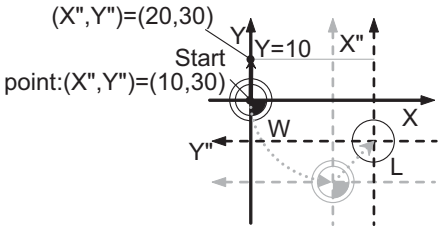
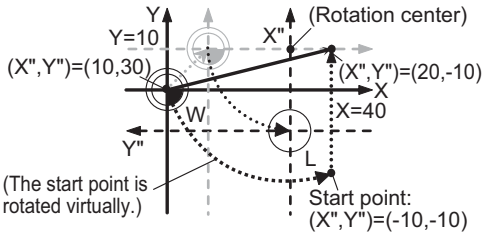
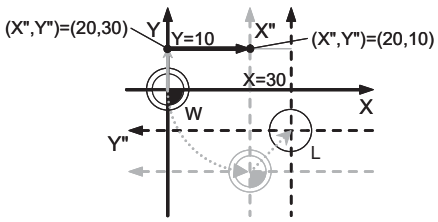
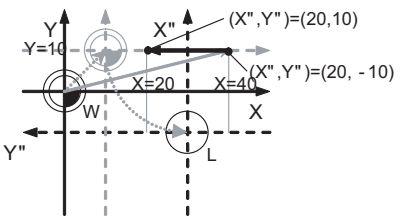
Local coordinate designation during coordinate rotation by program

When the local coordinate system (G52) is set during coordinate rotation, the operation is as follows.

(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      W: Workpiece coordinate system
N08 M02 ;                   End                             L: Local coordinate system
```

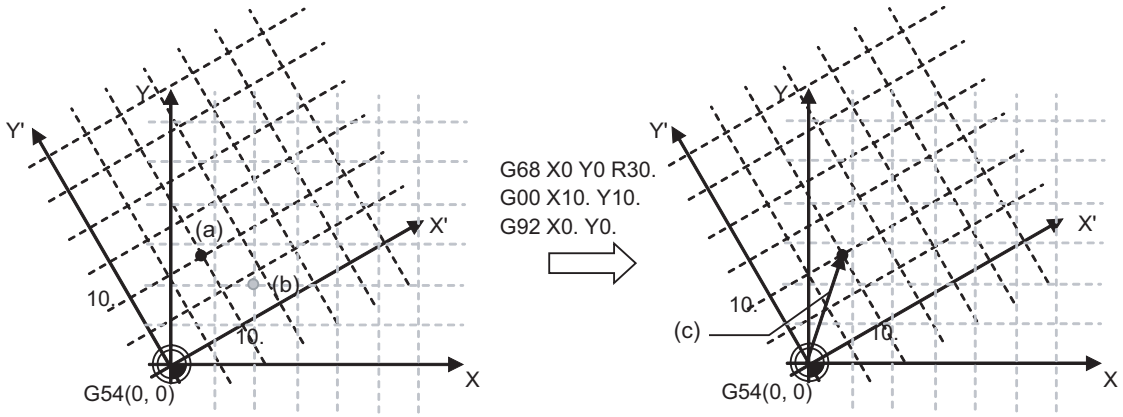
	(1) Operation of #19003 = 0	(2) Operation of #19003 = 1
N03		
	Workpiece coordinate system is rotated virtually.	Workpiece coordinate system is not rotated.
N04		
	The workpiece coordinate zero point after rotation is considered as (X,Y)=(0,0). The position after shifted by "10." each in the X and Y 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		
	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		

Coordinate system designation during coordinate rotation by program

When the coordinate system setting (G92) is executed during coordinate rotation by program (G68), this program operates same as "Local coordinate designation during coordinate rotation by program".

(1) When "#19003 PRG coord rot type" is "0", the position is preset to the current position commanded on the rotated coordinate system.

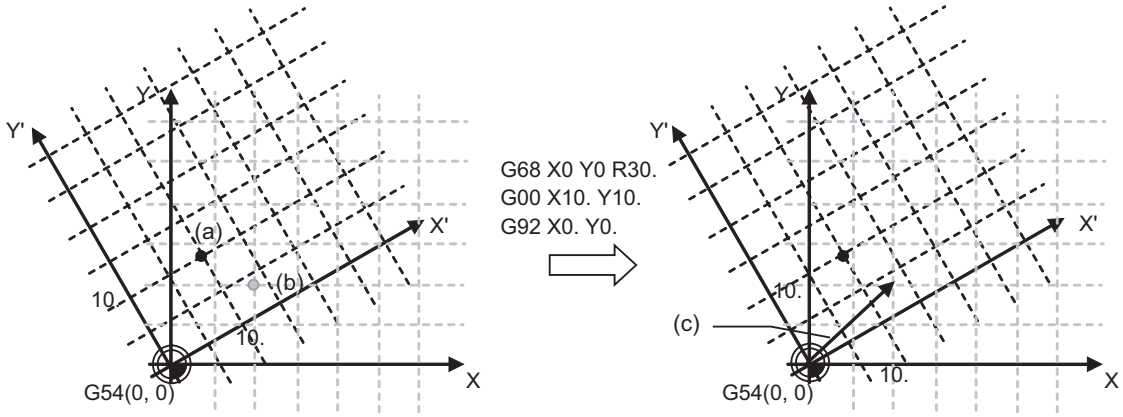
(Example) Setting on the coordinate system (X'-Y') after rotation



(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

(1) When "#19003 PRG coord rot type" is "1"and the coordinate system setting (G92) is executed during coordinate rotation by program, the rotation center of the coordinate is not shifted. (It stays at the same position in respect to the basic machine coordinate system.)

Operation when circular interpolation is commanded after commanding rotation

Basically, command two axes in the rotation plane by positioning/linear interpolation with an absolute position immediately after the coordinate rotation command.

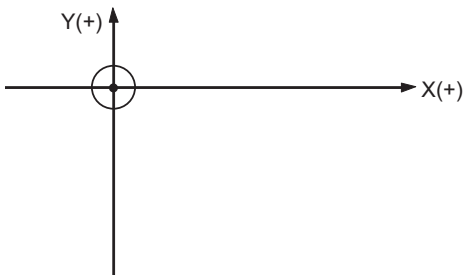
When commanding the circular interpolation, command the movement by positioning/linear interpolation to the start point even if the axis does not move.

When the circular interpolation is commanded straight after the coordinate rotation command, the following operation takes place depending on the setting of the parameter "#19003 PRG coord rot type".

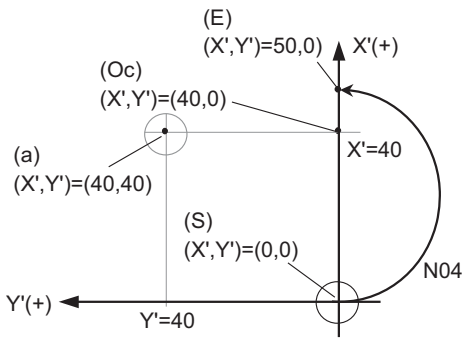
(1) When "#19003 PRG coord rot type" is "1", the start point position which was assumed to be rotated with the coordinate rotation and the actual axis position are positioned in a different position in respect to the basic machine coordinate system. Consequently, the axis cannot move along with the circular path between the start point and the end point, and a program error (P70) will occur.

```
N01 G17 G28 X0. Y0.;
N02 G90 G92 G53 X0. Y0.;
N03 G68 X40. Y0. R90.;        Coordinate rotation ON
N04 G03 X50. R-25. F500;
N05 G00 Y50.;
N06 G69;                      Coordinate rotation cancel
N07 M02;                      End
```

[Local coordinate system before rotation]



[Local coordinate system after rotation]

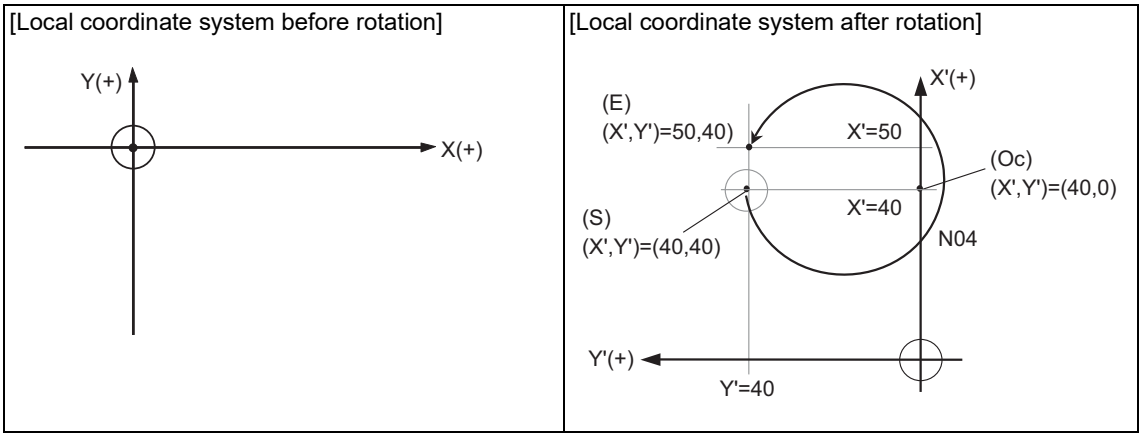


(a) Actual axis position
(Oc) Rotation center
(S) Start point (*1)
(E) End point

(*1) Start point which was assumed to be rotated with the coordinate rotation

(2) When "#19003 PRG coord rot type" is "0", the circular interpolation start point does not rotate with the coordinate rotation. It remains in the same position before the coordinate rotation in respect to the basic machine coordinate system. Consequently, the axis operates the circular interpolation from this start point to the end point.

```
N01 G17 G28 X0. Y0.;
N02 G90 G92 G53 X0. Y0.;
N03 G68 X40. Y0. R90.;        Coordinate rotation ON
N04 G03 X50. R-25. F500;
N05 G00 Y50.;
N06 G69;                      Coordinate rotation cancel
N07 M02;                      End
```



(Oc) Rotation center

(S) Start point (E) End point



Relationship with other functions

(1) A program error (P111) will occur if the plane selection code is commanded during the coordinate rotation mode.

(2) A program error (P485) will occur if polar coordinate interpolation is commanded during the coordinate rotation mode.

(3) A program error (P481) will occur if coordinate rotation is commanded during the polar coordinate interpolation mode.

(4) A program error (P485) will occur if cylindrical interpolation is commanded during the coordinate rotation mode.

(5) A program error (P481) will occur if coordinate rotation is commanded during the cylindrical interpolation mode.

(6) A program error (P34) will occur if the workpiece coordinate system preset (G92.1) is commanded during the coordinate rotation mode.

(7) Coordinate rotation by program 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.

(8) If the tool position offset is commanded during the coordinate rotation mode, a program error (P141) will occur.

(9) Coordinate rotation by program for master axis and slave axis of the multiple-axis synchronization control is not supported. Do not command the coordinate rotation by program.



Precautions

(1) Always use an absolute command for the movement command immediately after G68 and G69.

(2) If the manual absolute is ON and has interrupted the coordinate rotary axis, do not use automatic operation for the following absolute command.

(3) The intermediate point during automatic 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 of the coordinate will be shifted. (The center will follow the coordinate system.)

(5) If the workpiece coordinates are changed (ex. from G54 to G55) during coordinate rotation by program, the rotation center of the coordinate is set to the workpiece coordinate position of the rotation center coordinates designated for the G68 command, on a new workpiece 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" is "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" is "0"). The feedrate in this case is the resultant speed of each axis' rapid traverse rate, thus, 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.

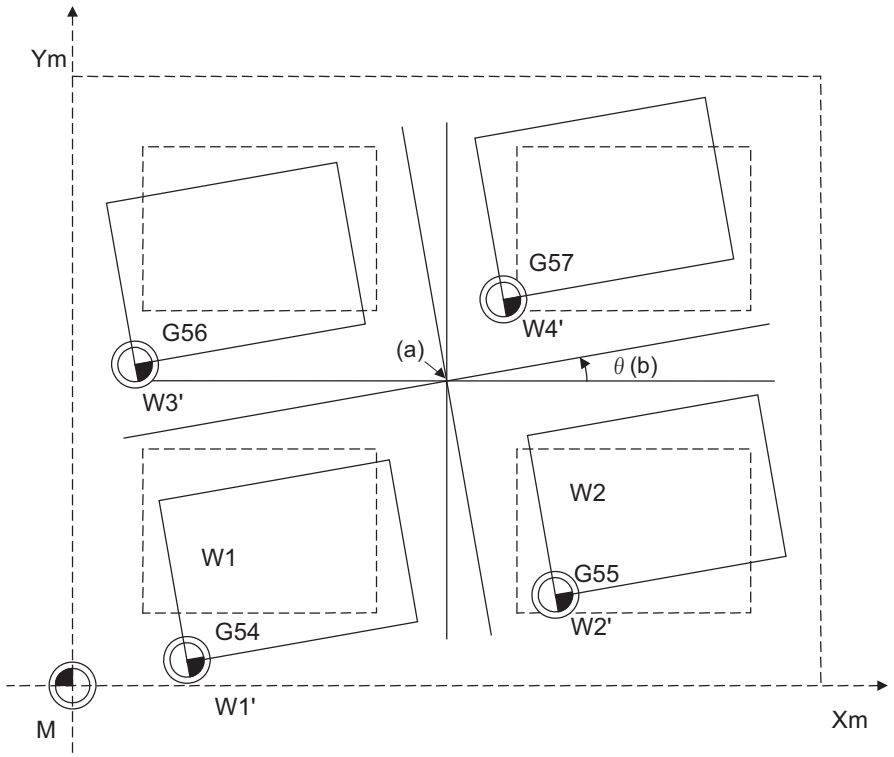
(12) The coordinates can also be rotated for the parallel axis. Select the plane that contains the parallel axis before issuing the G68 command. The plane cannot be selected in the same block as the G68 command.

(13) The coordinates can be rotated for the rotary axis. The angle will be interpreted as the length when rotating.

20.11 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.

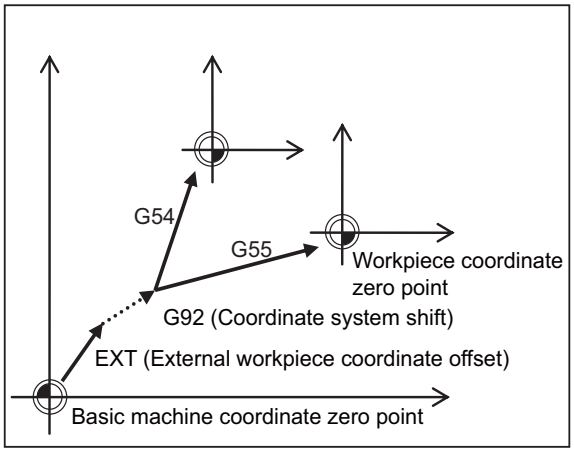
- Automatic operation: The setting values become valid from the next block after parameter settings have been configured.
- Manual operation: The setting values become valid if the PLC signal (manual feed coordinate switching for coordinate rotation by parameter) is set to ON after parameter settings have been configured.

Concept of coordinate system

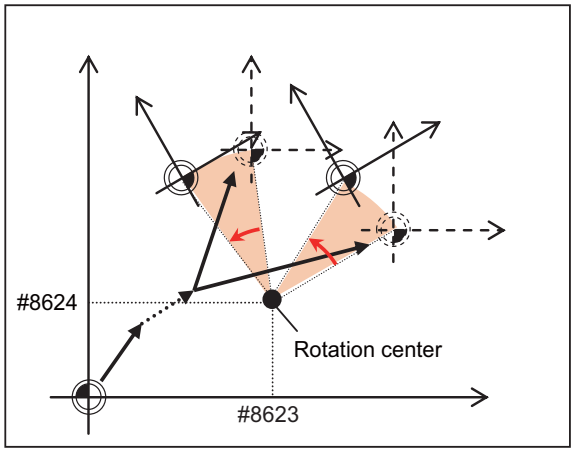
(1) Set the parameters "#8623 Coord rot centr(H)" and "#8624 Coord rot centr(V)" at the position of the machine coordinate system.

(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 positions.

[Parameters]

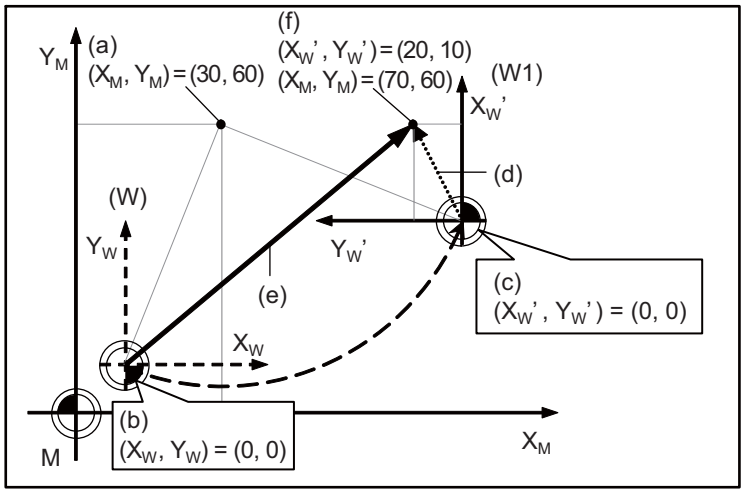
#8621 Coord rot plane (H) = X
#8622 Coord rot plane (V) = Y
#8623 Coord rot centr (H) = 30.0
#8624 Coord rot centr (V) = 60.0
#8627 Coord rot angle = 0.0

[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 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 position

(2) Command by incremental position

(3) Circular interpolation command

Operation when an axis configuring the rotation plane is commanded by the absolute position

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]

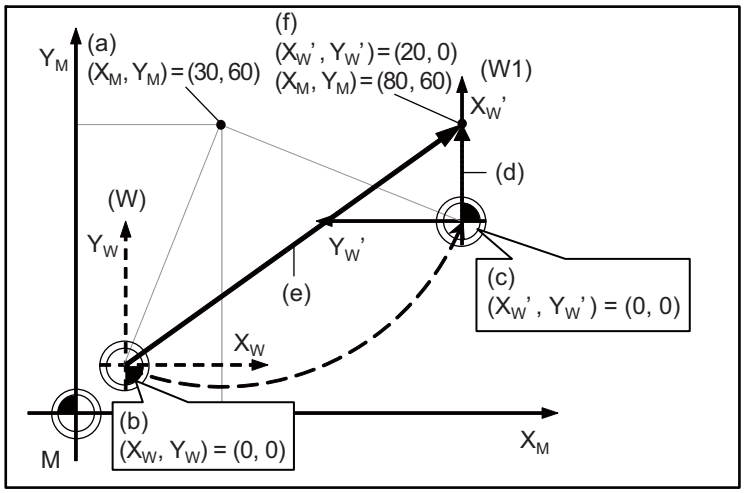
#8621 Coord rot plane (H) = X
#8622 Coord rot plane (V) = Y
#8623 Coord rot centr (H) = 30.0
#8624 Coord rot centr (V) = 60.0
#8627 Coord rot angle = 0.0

[Machining program]

```
N01 G17 G28 X0. Y0.;
N02 G54 G90 X0. Y0.;
N03 G10 K90.;              (Coordinate rotation start)
N04 G54 G90 G00 X20.;      (Absolute 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]

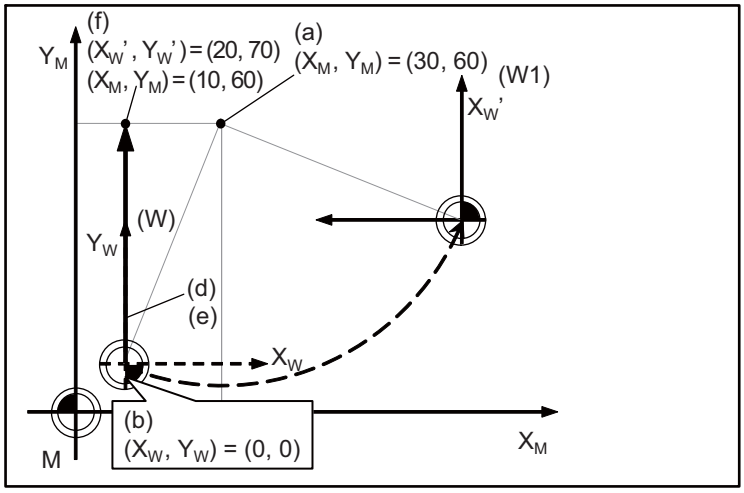
#8621 Coord rot plane (H) = X
#8622 Coord rot plane (V) = Y
#8623 Coord rot centr (H) = 30.0
#8624 Coord rot centr (V) = 60.0
#8627 Coord rot angle = 0.0

[Machining program]

```
N01 G17 G28 X0. Y0.;
N02 G54 G90 X0. Y0.;
N03 G10 K90.;              (Coordinate rotation start)
N04 G54 G90 G00 X20.;      (Absolute 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 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]

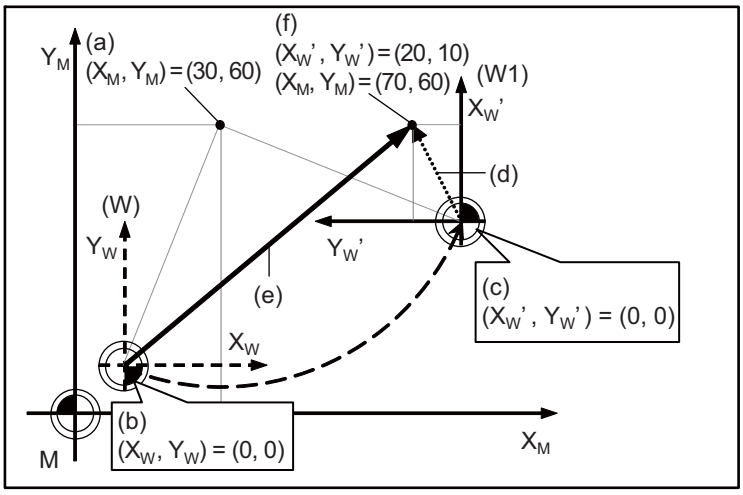
#8621 Coord rot plane (H) = X
#8622 Coord rot plane (V) = Y
#8623 Coord rot centr (H) = 30.0
#8624 Coord rot centr (V) = 60.0
#8627 Coord rot angle = 0.0

[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 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]

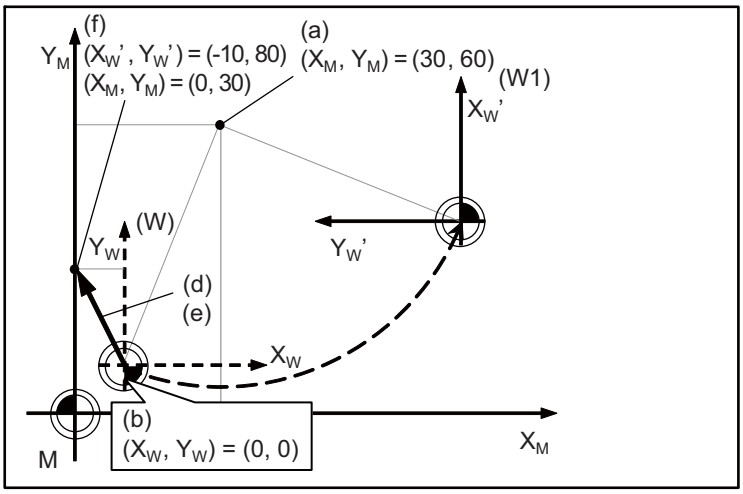
#8621 Coord rot plane (H) = X
#8622 Coord rot plane (V) = Y
#8623 Coord rot centr (H) = 30.0
#8624 Coord rot centr (V) = 60.0
#8627 Coord rot angle = 0.0

[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 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]

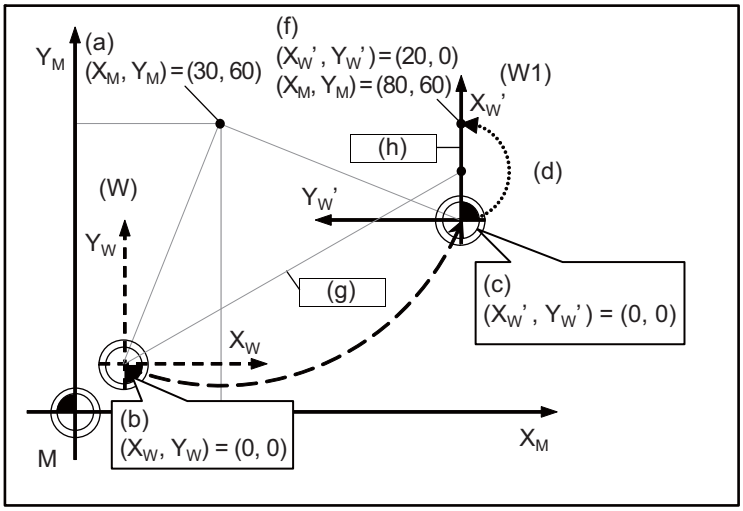
#1084 Arc error = 0.1
#8621 Coord rot plane (H) = X
#8622 Coord rot plane (V) = Y
#8623 Coord rot centr (H) = 30.0
#8624 Coord rot centr (V) = 60.0
#8627 Coord rot angle = 0.0

[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]

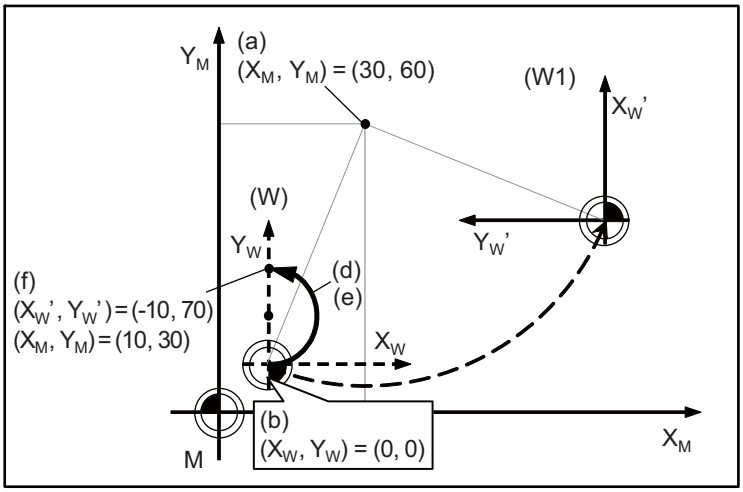
#1084 Arc error = 0.1
#8621 Coord rot plane (H) = X
#8622 Coord rot plane (V) = Y
#8623 Coord rot centr (H) = 30.0
#8624 Coord rot centr (V) = 60.0
#8627 Coord rot angle = 0.0

[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]

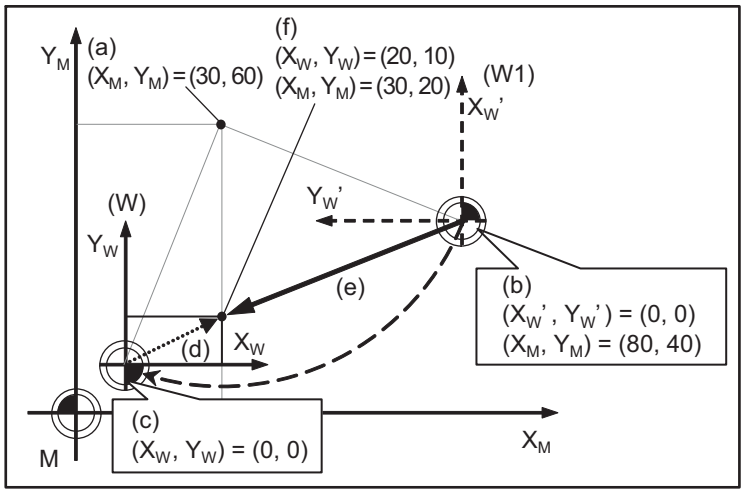
#8621 Coord rot plane (H) = X
#8622 Coord rot plane (V) = Y
#8623 Coord rot centr (H) = 30.0
#8624 Coord rot centr (V) = 60.0
#8627 Coord rot angle = 90.0

[Machining program]

```
N01 G54 G90 X50.Y50.;
N02 G54 G90 X0. Y0.;
N03 G10 K0.;                    (Coordinate rotation cancel)
N04 G91 G00 X20. Y10.; (Incremental 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]

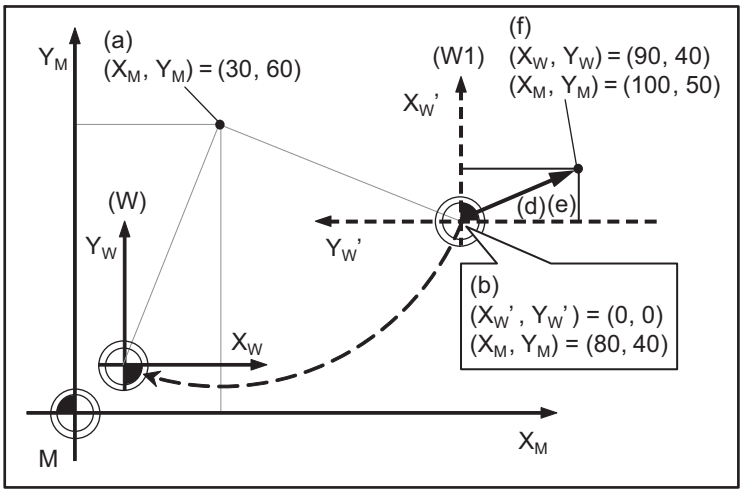
#8621 Coord rot plane (H) = X
#8622 Coord rot plane (V) = Y
#8623 Coord rot centr (H) = 30.0
#8624 Coord rot centr (V) = 60.0
#8627 Coord rot angle = 90.0

[Machining program]

N01 G54 G90 X50.Y50.;
N02 G54 G90 X0. Y0.;
N03 G10 K0.; (Coordinate rotation cancel)
N04 G91 G00 X20. Y10.; (Incremental 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 coordinate rotation by program 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

[G54 workpiece coordinate system offset]

X = 10.0
Y = 10.0

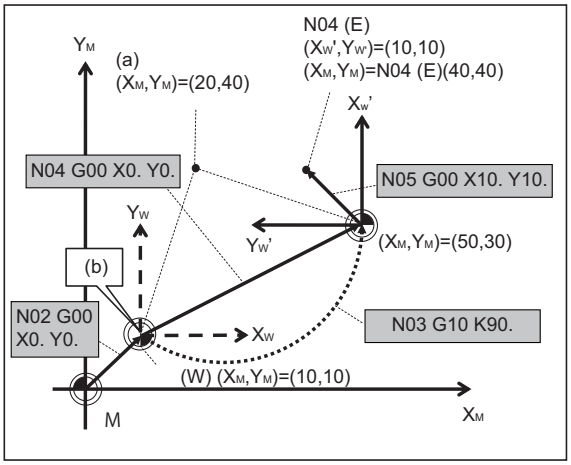
[Machining program]

(1) N01 G54 G17 G28 X Y;
N02 G90 G00 X0. Y0.;
N03 G10 K90. ;
N04 G00 X0. Y0.;
N05 G00 X10. Y10.;

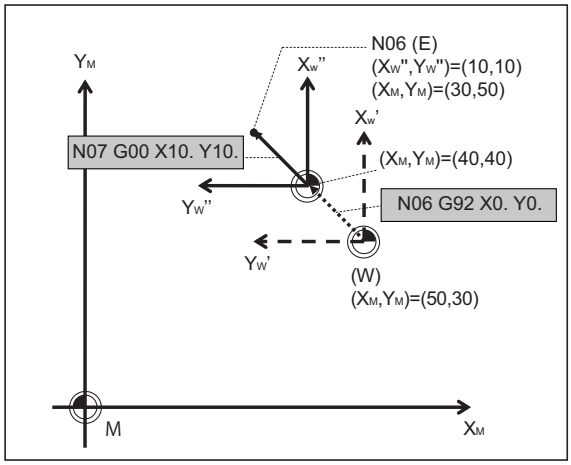
(2) N06 G92 X0. Y0.;
N07 G00 X10. Y10.;

(3) N08 G92.1 X0. Y0.;
N09 G00 X0. Y0.;
N10 G00 X-10. Y10.;

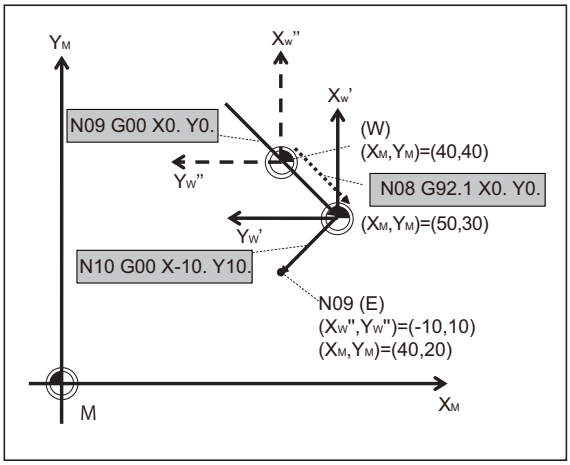
(1)



(2)



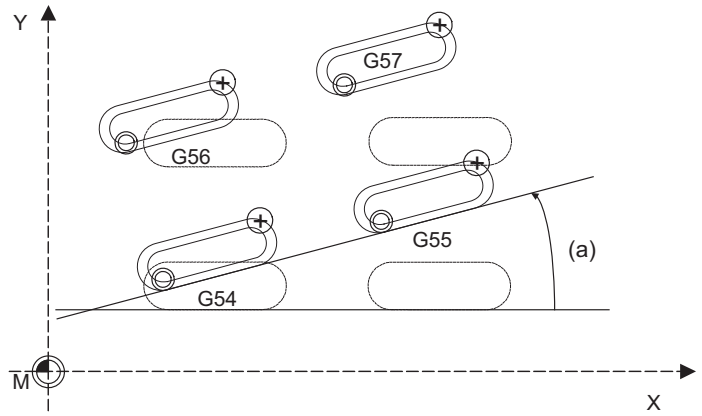
(3)



(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.



(a) Rotation movement (15 degree)

```
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 processing for high-speed machining may temporarily canceled when the following high-speed high-accuracy functions are used together during coordinate rotation by parameter.
No program errors occur; however, the processing capability is the same as when the high-speed machining is set OFF.

- High-speed machining mode I / high-speed machining mode II
  (The function is temporarily canceled during coordinate rotation by parameter.)
- High-speed high-accuracy control I / High-speed high-accuracy control II / High-speed high-accuracy control III
  (While the coordinate rotation by parameter is enabled, the high-speed machining mode is temporarily canceled and only the high-accuracy control is enabled.)

(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.

20.12 Scaling; G50/G51

Function and purpose

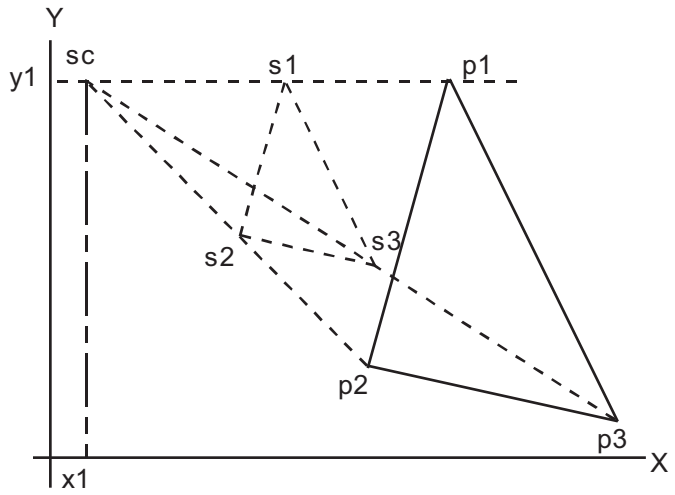
By multiplying the moving axis command values within the range specified under this command by the factor, the shape commanded by the program can be enlarged or reduced to the desired size.

Command format

Scaling ON (When setting 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.

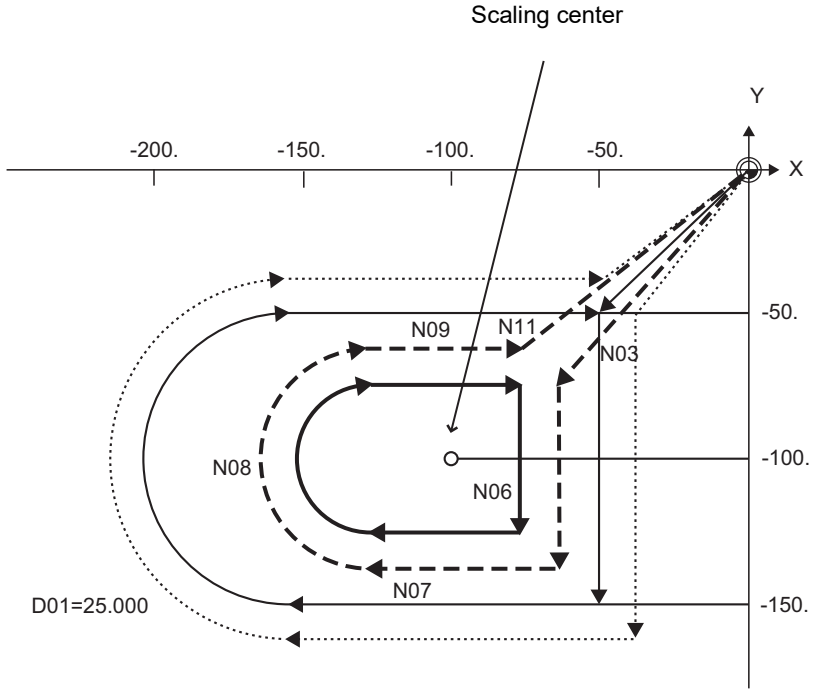
- Scaling was commanded though there was no scaling specification. (P350)
- The upper limit of the factor command range was exceeded in the same block as G51. (P35)
  (When using the machining parameter scaling factor, the factor is calculated as 1, when -0.000001 < factor < 0.000001, or the factor is 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.

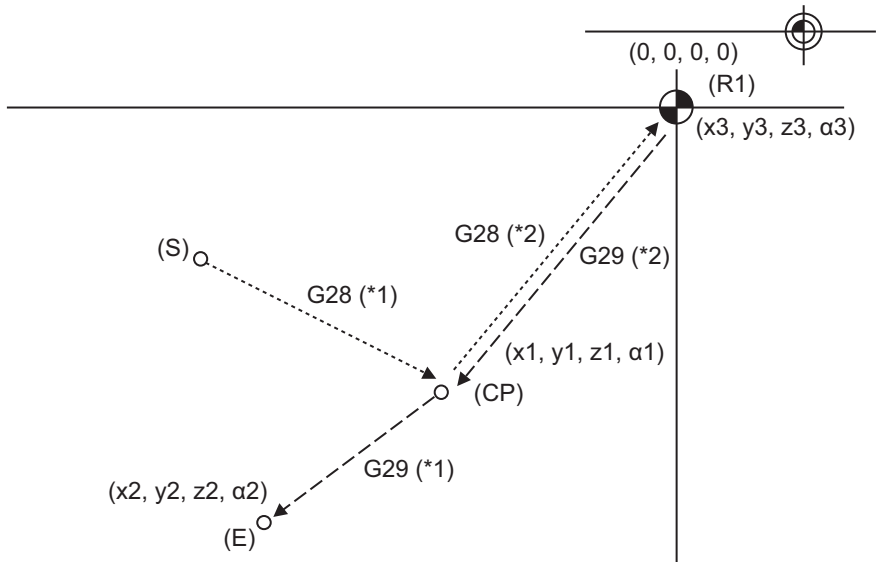
(7) If manual interruption is made during scaling, manual ABS selection is ignored for the movement followed by an incremental command and operation performed is the same as in manual ABS OFF.

20.13 Automatic 1st Reference Position Return and Start Position 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. (For G30, refer to "20.14 2nd, 3rd, 4th Reference Position Return; G30".)

The following shows an operation when "G28 Xx1 Yy1 Zz1 αα1;" and "G29 Xx2 Yy2 Zz2 αα2;" are commanded.



(R1) 1st reference position
(S) G28 start point
(CP) Intermediate point
(E) G29 end point

(*1) Moves in interpolation or non-interpolation. (MTB specifications)

(*2) Moves in non-interpolation.

Command format

G28 X__ Y__ Z__ α__ ;	**Automatic 1st reference position return**

X, Y, Z, α	Coordinate value of the intermediate point ("α" is an additional axis) -99999.999 to 99999.999 (mm) The coordinate value depends on absolute/incremental commands. The unspecified axis is not returned.

G29 X__ Y__ Z__ α__ ;	**Start position return**

X, Y, Z, α	Coordinate value of the return destination (end point of G29) ("α" is an additional axis) -99999.999 to 99999.999 (mm) The coordinate value depends on absolute/incremental commands. The unspecified axis is not returned.

Detailed description

Automatic 1st reference position return (G28)

(1) The G28 command is equivalent to the following:

G00 Xx1 Yy1 Zz1 αα1;	Move to the intermediate point.
G00 Xx3 Yy3 Zz3 αα3;	Move to the reference position in the selected coordinate system. (*1)

(*1) This reference position is set in the parameter "#2037 G53ofs" as 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 subsequence 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 the reference position return is completed, the "1st reference position reached n-th axis" signal (X800 or above) is output and "#1" appears at the axis name line on the setting and display unit screen.

(4) When the parameter "#1091 Mpoint" (Ignore middle pnt) is set to "1", the intermediate point of the program is ignored and the axis is directly returned to the reference position.

Start position return (G29)

(1) The G29 command is equivalent to the following:

G00 Xx1 Yy1 Zz1 αα1;	Move to the intermediate point (x1, y1, z1, α1) of G28 (or G30).
G00 Xx2 Yy2 Zz2 αα2;	Move from the reference position to the end point in the selected coordinate system.

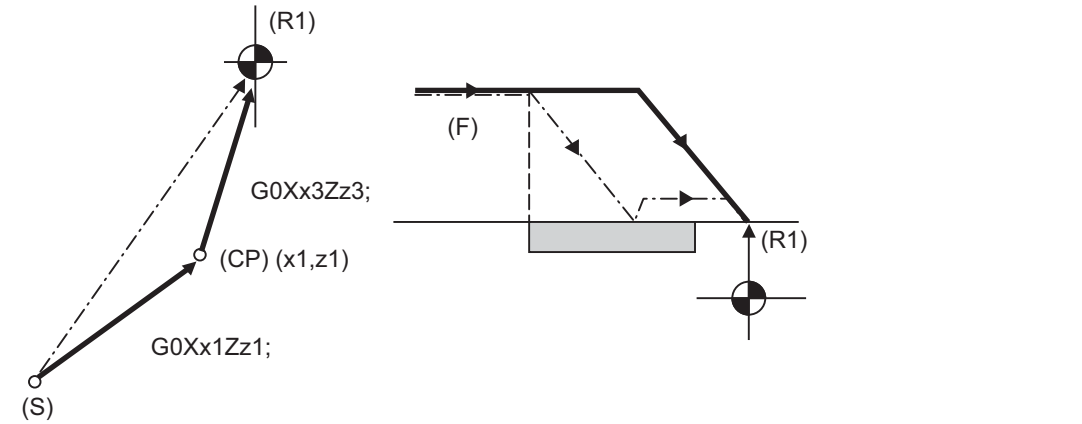
Rapid traverse (non-interpolation type) applies independently to each axis for the positioning from the reference position to the intermediate point.

(2) When the parameter "#1091 Mpoint" (Ignore middle pnt) is set to "1", the intermediate point of the program is ignored and the axis is directly returned to the end point.

(3) G29 is enabled for both G28 and G30, and performs positioning of the specified axis after returning to the newest intermediate point.

Program example

(Example 1) G28 Xx1 Zz1 ;



1st operation after power has been turned ON

2nd and subsequent operations

Near-point dog

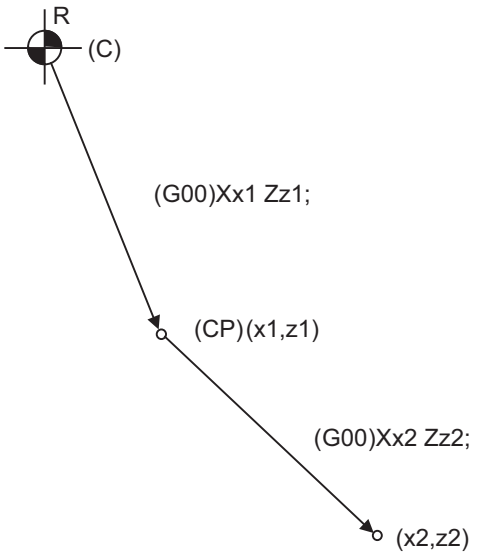
(S) Return start position

(CP) Intermediate point

(R1) Reference position (#1)

(F) Rapid traverse rate

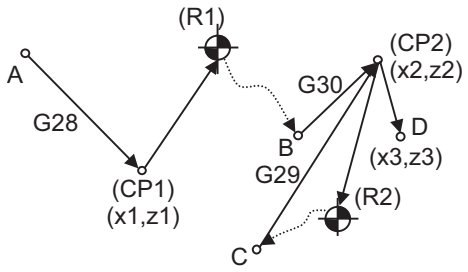
(Example 2) G29 Xx2, Zz2 ;



(C) Current position

(CP) G28, G30 Intermediate point

(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)



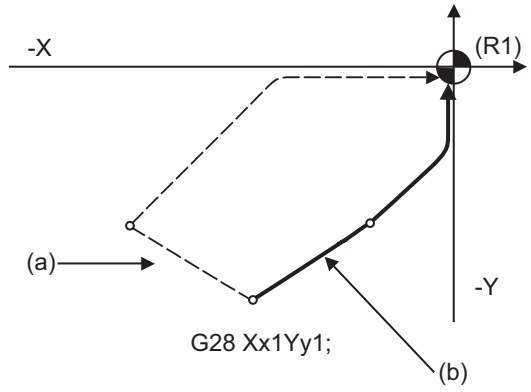
(CP1) Old intermediate point
(CP2) New intermediate point
(R1) Reference position (#1) (R2) 2nd reference position (#2)

Relationship with other functions

Mirror image

When the reference position return is performed during mirror image, the following occurs:

From start point to intermediate point	The mirror image is enabled. The tool moves in the opposite direction to that of the command.
From intermediate point to reference position	The tool moves by ignoring the mirror image.



(a) X-axis mirror image (b) No mirror image
(R1) 1st reference position

Precautions

(1) The program error (P430) occurs if G29 is executed without executing automatic 1st reference position return (G28) after the power has been turned ON.

(2) The operations of G28, G29 and G30 commands with single block operation depend on the MTB specifications (parameter "#1279 ext15/bit6" (Enable single block stop at the intermediate point)).

0: A single block stop is not performed at intermediate point.

1: A single block stop is performed at intermediate point.

(3) Note the following points during single block stop at intermediate point.

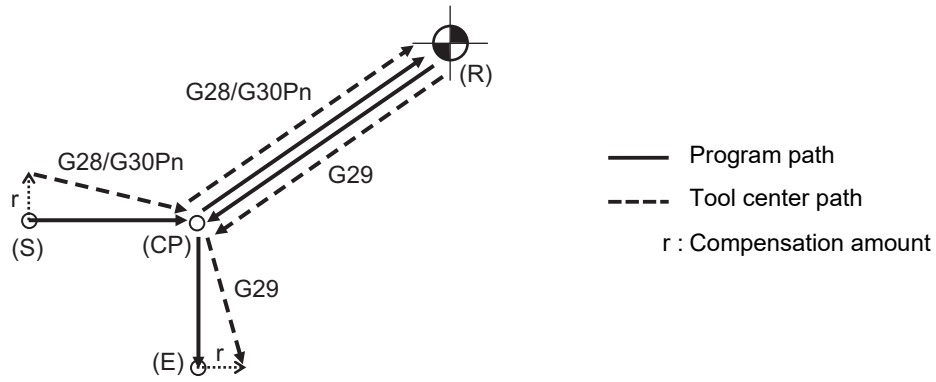
- If the mode is switched to MDI mode or reference position return mode, the operation error (M01 0013) occurs.
- If the PLC interrupt operation is performed, the operation error (M01 0129) occurs.
- If the NC is reset, the intermediate point position at the time of start position return (G29) is not updated.

(4) If a miscellaneous function is commanded in the same block as G28, G29, or G30, completion of the miscellaneous function is awaited at the end point, instead of the intermediate point.

(5) 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.

(6) If the reference position parameter is changed while the reference position return operation is suppressed due to interlock, etc., the operation error (M01 0027) occurs.

(7) With reference position return on a plane during tool radius compensation, the tool moves without tool radius compensation (zero compensation) from the intermediate point as far as the reference position. With a subsequent G29 command, the tool moves 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.



(R) Reference position
(S) G28 or G30 start position
(CP) Intermediate point
(E) G29 end point

(8) The tool compensation will be canceled during reference position return unless it is already canceled, and the compensation amount will be cleared.

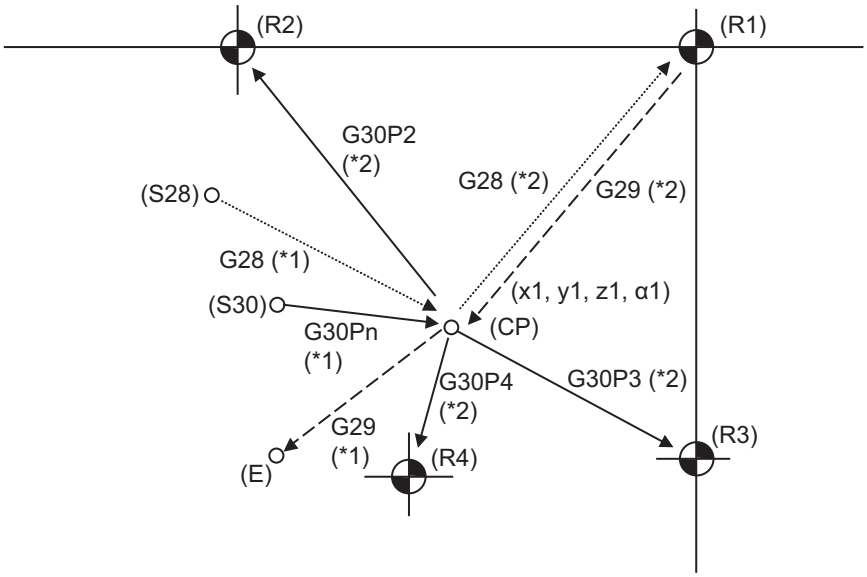
(9) In G28, when the Z axis is canceled, the movement of the Z axis to the intermediate point is ignored, and only the position from subsequent positioning is displayed. (The machine itself will not move.)

(10) When G28/G30 is commanded in the same block as G49, whether or not the program error (P45) occurs depends on the MTB specifications (parameter "#12147 SpecErr_Avd").

20.14 2nd, 3rd, 4th Reference Position Return; G30

Function and purpose

The axis returns to the 2nd reference position (or 3rd or 4th reference position) when "G30 P2" (or "G30 P3" or "G30 P4") is commanded. (For G28 and G29, refer to "20.13 Automatic 1st Reference Position Return and Start Position Return; G28, G29".)



(S28) G28 start point
(S30) G30 start point
(E) G29 end point
(CP) Intermediate point
(R1) 1st reference position
(R2) 2nd reference position
(R3) 3rd reference position
(R4) 4th reference position

(*1) Moves in interpolation or non-interpolation. (MTB specifications)

(*2) Moves in non-interpolation.

Command format

G30 P2 (P3, P4) X__ Y__ Z__ α__; 2nd, 3rd, 4th reference position return

P	Reference position No. P2: 2nd reference position return P3: 3rd reference position return P4: 4th reference position return When the command is omitted or when a value out of the range is specified, the axis returns to the 2nd reference position.
X, Y, Z, α	Coordinate value of the intermediate point ("α" is an additional axis) -99999.999 to 99999.999 (mm) The coordinate value depends on absolute/incremental commands. The unspecified axis is not returned.

Detailed description

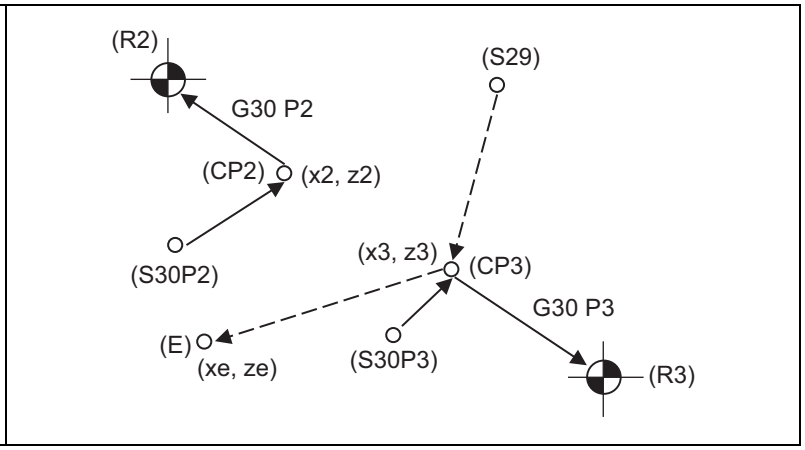
(1) The 2nd, 3rd, or 4th reference position is specified by P2, P3, or P4.

(2) In the "2nd, 3rd, 4th reference position return" mode, as in the "automatic 1st reference position return" mode, the tool returns to the 2nd, 3rd, or 4th reference position via the intermediate point specified by G30. When the reference position return is completed, the "n-th reference position reached m-th axis" signal (X820 or above) is output and "#n" appears at the axis name line on the setting and display unit screen. ("n" indicates the reference position number.)

(3) The 2nd, 3rd, and 4th reference position coordinates refer to the positions specific to the machine, and depend on the MTB specifications. (Setting of the parameters "#2038 #2_rfp", "#2039 #3_rfp" and "#2040 #4_rfp")

(4) If G29 is commanded after completion of "2nd, 3rd, 4th reference position return", the intermediate position used last is used as the intermediate position for returning by G29. ((CP3) in the figure below)

```
G30 P2 Xx2 Zz2 ;
:
G30 P3 Xx3 Zz3 ;
:
G29 Xxe Zze ;
```



(R2) 2nd reference position
(R3) 3rd reference position
(S30P2) G30 P2 start point
(CP2) G30 P2 intermediate point
(S30P3) G30 P3 start point
(CP3) G30 P3 intermediate point

For G29, refer to "20.13 Automatic 1st Reference Position Return and Start Position Return; G28, G29".

(5) When the parameter "#1091 Mpoint" (Ignore middle pnt) is set to "1", the intermediate point of the program is ignored and the axis is directly returned to the reference position.

Relationship with other functions

Mirror image

Refer to "Relationship with other functions" in "20.13 Automatic 1st Reference Position Return and Start Position Return; G28, G29".

Precautions

(1) The operations of G28, G29 and G30 commands with single block operation depend on the MTB specifications (parameter "#1279 ext15/bit6" (Enable single block stop at the intermediate point)).

0: A single block stop is not performed at intermediate point.

1: A single block stop is performed at intermediate point.

(2) Note the following points during single block stop at intermediate point.

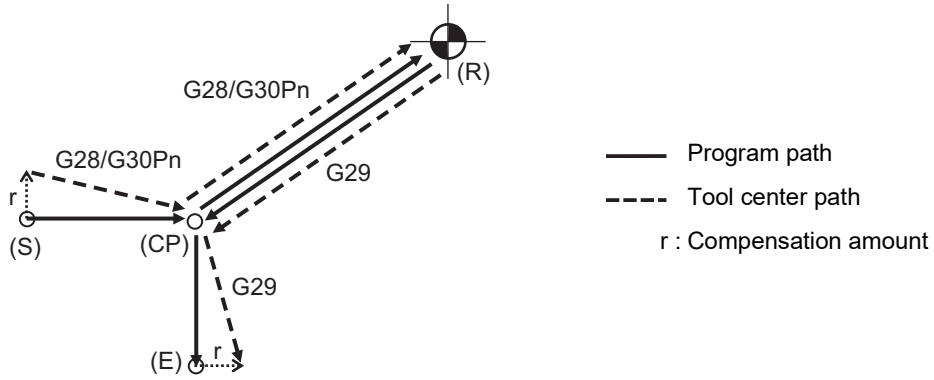
- If the mode is switched to MDI mode or reference position return mode, the operation error (M01 0013) occurs.
- If the PLC interrupt operation is performed, the operation error (M01 0129) occurs.
- If the NC is reset, the intermediate point position at the time of start position return (G29) is not updated.

(3) If a miscellaneous function is commanded in the same block as G28, G29, or G30, completion of the miscellaneous function is awaited at the end point, instead of the intermediate point.

(4) 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.

(5) If the reference position parameter is changed while the reference position return operation is suppressed due to interlock, etc., the operation error (M01 0027) occurs.

(6) With reference position return on a plane during tool radius compensation, the tool moves without tool radius compensation (zero compensation) from the intermediate point as far as the reference position. With a subsequent G29 command, the tool moves 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.



(R) Reference position
(S) G28 or G30 start position
(CP) Intermediate point
(E) G29 end point

(7) The tool length compensation amount for the axis involved is canceled after the 2nd, 3rd and 4th reference position return.

(8) When G28/G30 is commanded in the same block as G49, whether or not the program error (P45) occurs depends on the MTB specifications (parameter "12147 SpecErr_Avd").

20.15 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>

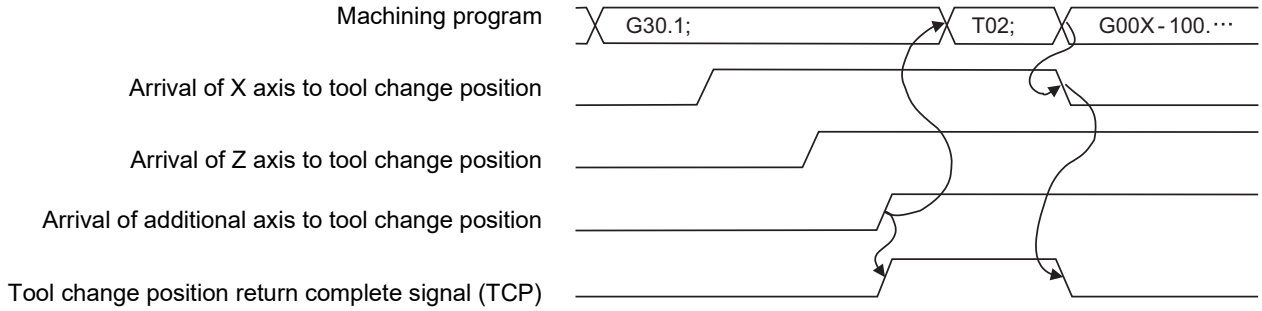
- An arrow (->) indicates the order of axes that begin to return. A hyphen (-) indicates that the axes begin to return simultaneously. (Example: "Z axis -> X axis - Y axis" indicates that the Z axis returns to the tool change position, then the X axis and Y axis do at the same time.)

(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)

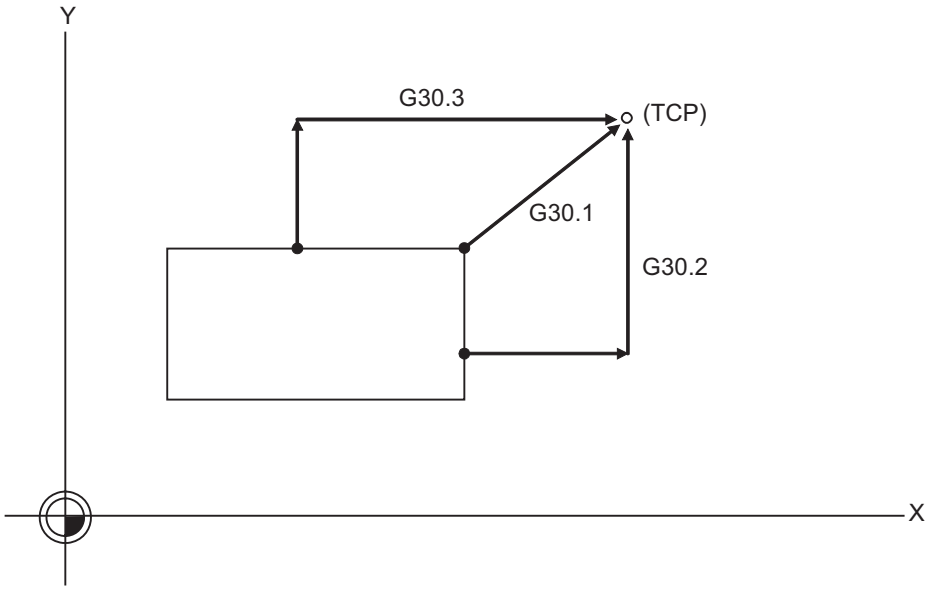


(4) When a tool change position return command is issued, tool offset data such as for tool length offset and tool radius compensation for the axis that moved is canceled.

(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.)

20.16 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

G27 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 simultaneous contouring control axes.

(3) An alarm will occur if the reference position is not reached after the command is completed.

21

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

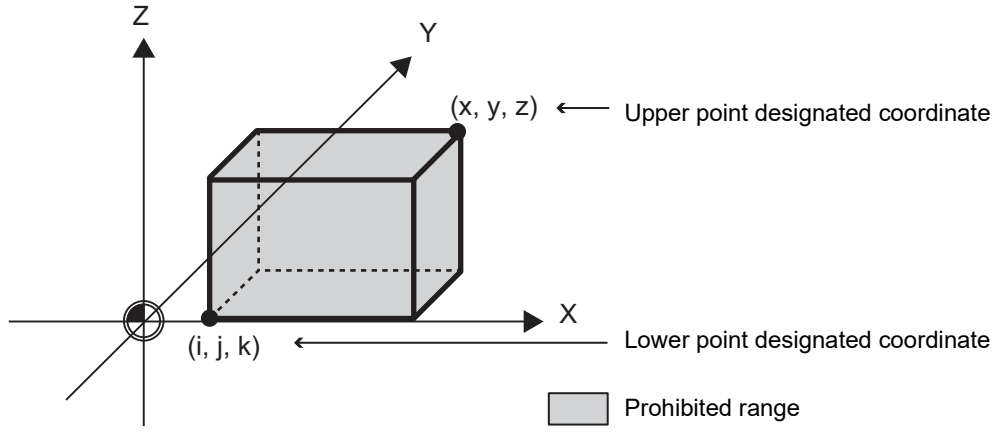
(1) In the following command format, the basic axes are X, Y and Z. Command the base axis designated in the parameters #1026 to #1028.

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

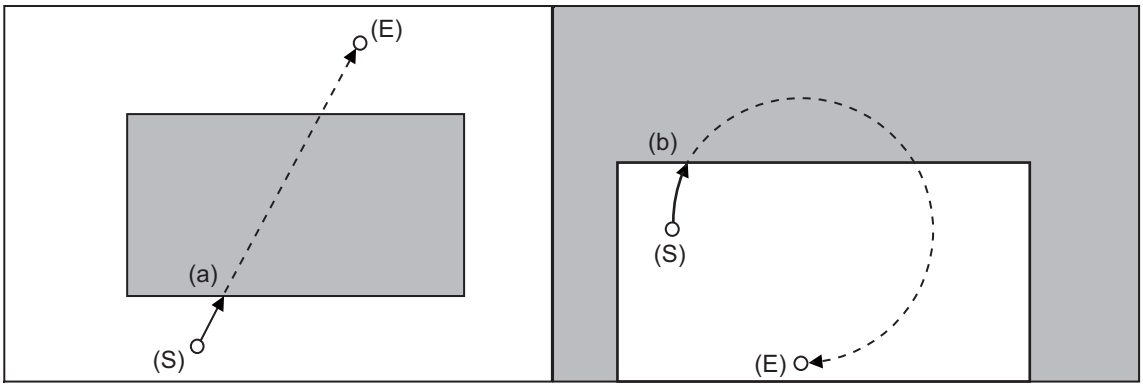
When "#12057 OT_prechkON" is set to "1", stroke check before travel is carried out for the stored stroke limit prohibited area.
The stroke check before travel is available only for the axes for which the stored stroke limit prohibited area is designated. Stroke check before travel will not be carried 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.

Path passing decision

The tool path of the travel command can also be targeted for "Stroke check before travel in stored stroke limit area" by setting the parameter "#8179" to "1". If a prohibited-area passing command is executed as shown below while the enabling conditions for stroke check before travel are satisfied, operation is stopped due to a program error (P452).

This function applies only to G00, G01, G02, G03, G28, G30, G33, and G53. If a non-target command is executed, a decision is made depending on whether or not the end point is included in the prohibited area. Also, when the axis passes through the prohibited area by a non-target command while the end point is not included in the prohibited area, an operation error (M01 0007) occurs just before the end point gets into the prohibited area, and operation is stopped.

This function is available for stored stroke limit I, stored stroke limit IB, stored stroke limit II, and stored stroke limit IIB. If there is even one axis with stored stroke limit IC enabled, this function is disabled, and the stroke check before travel is determined depending on whether or not the end point is included in the prohibited area.



(a) Even if a prohibited-area passing command is executed when the end point is inside the prohibited area, a program error (P452) occurs.

(b) Also, even if a prohibited-area passing command is executed when the end point is outside the prohibited area, a program error (P452) occurs.

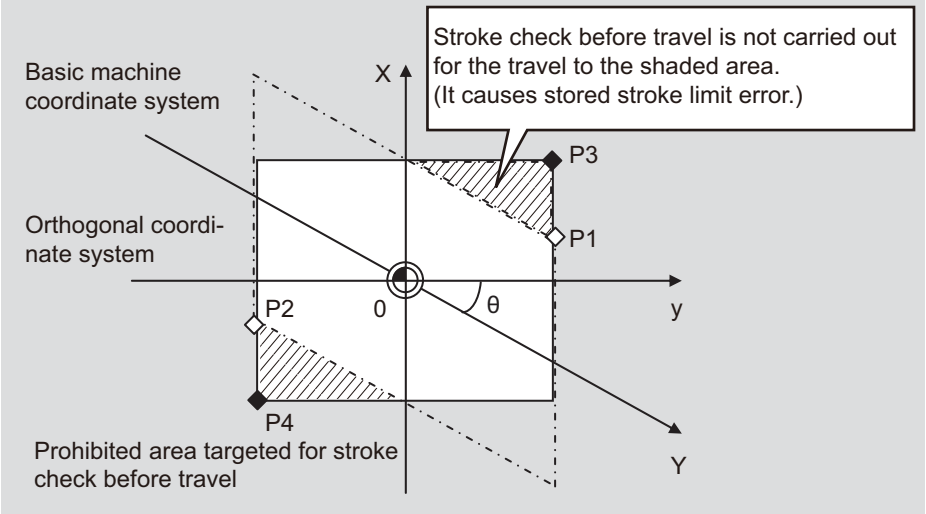
Relationship with other functions

(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
G07.1	Cylindrical interpolation
G12.1/G112	Polar coordinate interpolation
G54.4	Workpiece installation error compensation
G43.4/G43.5	Tool center point control
G68.2/G68.3	Inclined surface machining command
G05 P1	High-speed machining mode I
G05 P2	High-speed machining mode II
G68	3-dimensional coordinate conversion

G code	Function
G05.1 Q1	High-speed high-accuracy control I
G05 P10000	High-speed high-accuracy control II
G05 P20000	High-speed high-accuracy control III
G05.1 Q2	Spline interpolation
G06.2	NURBS interpolation
G30.1 - G30.6	Tool exchange position return
-	PLC axis control

(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.

(4) When the following commands are executed, the stroke check before travel by the path passing decision is not available. In a block including any of the following commands, the stroke check before travel is performed depending on whether or not the end point is included in the prohibited area.

G code	Function
G02, G03	Helical interpolation
G02.1, G03.1	Spiral/conical interpolation
G02.2, G03.2	Involute Interpolation
G02.3, G03.3	Exponential interpolation
G02.4, G03.4	3-dimensional circular interpolation

(5) When the prohibited area is changed by switching the stored stroke limit I area, the changed contents are not reflected in the decision of the stroke check before travel.

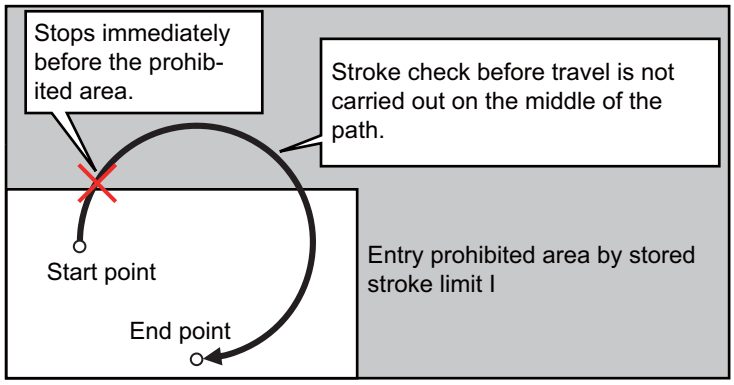
Precautions

(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) When the parameter "#8179" is set to "0", the tool path of the travel command is not targeted for a stroke check before travel. An operation error (M01 0007) occurs and processing stops by the stored stroke limit function just before the prohibited area. To target the following command for the stroke check before travel, set the parameter "#8179" to "1".



(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 position by the automatic reference position return G28/G30 and a travel from the reference position to the intermediate point by 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" using 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" using the parameter input by program (G10L70).

(9) The command path passing decision cannot be combined with stored stroke limit IC. If there is even one axis with stored stroke limit IC enabled, a decision is made depending on whether or not the end point is included in the prohibited area.

(10) Even when G00, G28, G30, or G53 is operated in non-interpolation mode, the command path passing decision is made, assuming that the command is operated on a line connecting the start point and end point.

(11) A command for rotary axis is not targeted for the command path passing decision. In a block including a command for rotary axis, a decision is made depending on whether or not the orthogonal axis passes through the prohibited area.

21.2 Enable Interfering Object Selection Data; G186

Function and purpose

In the interference check III, an interfering object can be defined with a combination of multiple solids.
The maximum numbers of definitions of solids and interfering objects, the maximum number of solids that can be used to define an interfering object, the number of interfering objects, etc. differ depending on the model.

The interfering objects to be checked 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: 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 "23.32 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 (Y769) is turned 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, turn ON the "Interference check III: Enable interfering object selection data" signal (Y769) or issue the "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.

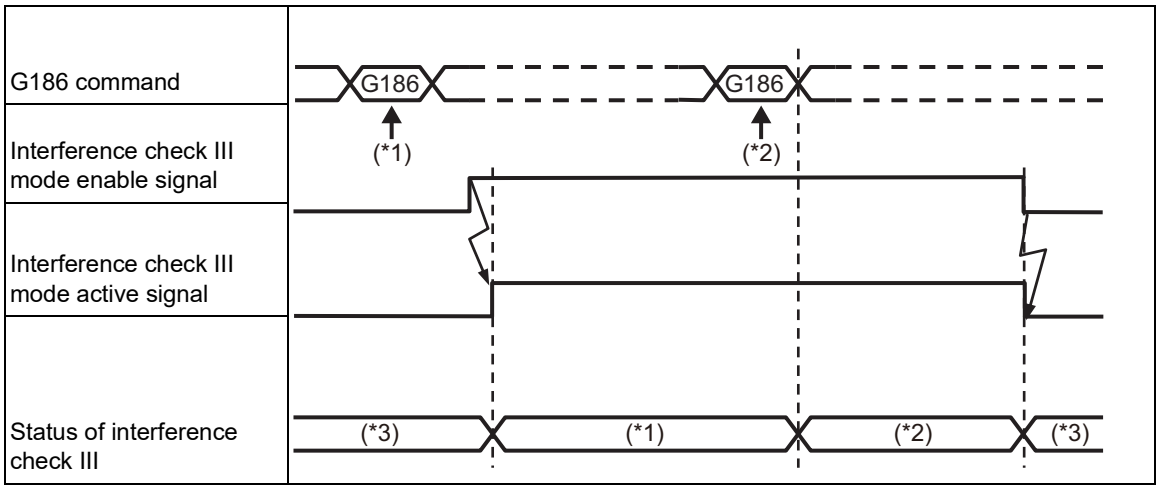
(4) In the case the alarm occurs due to the consistency check, the interfering data will not be updated. For the interference check between interfering objects, the interfering data enabled last time is continuously used.

(5) If the settings are made to cause an alarm due to the consistency check, the 3D model on the interference check III screen may not be displayed correctly.

Interference check III mode enable command

While the "Interference check III mode" signal (Y76A) is ON after the "Enable interfering object selection data" signal was turned ON or the "Enable interfering object selection data" command (G186) was executed, the interference between interfering objects is checked. While the interference check III is being executed, the "Interference check III: In interference check III mode" signal (X772) is turned ON.
After the NC power is turned ON, if the "Interference check III mode" signal (Y76A) is turned ON without executing the "Enable interfering object selection data" signal or the "Enable interfering object selection data" command (G186) even once, the operation error (M03 1001) occurs.

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 micro 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 0.1μm 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.

22

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

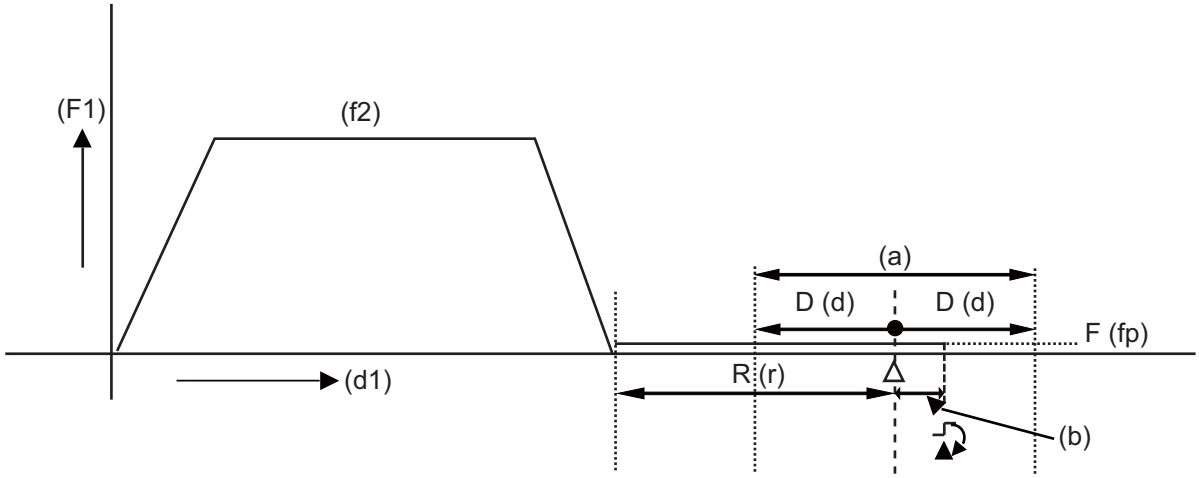
G37 Z__ R__ D__ F__ ;

Z	Measuring axis address and coordinates of the measurement position
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 setting of "R", "D" or "F" is omitted, the value set in the parameter is used instead. <Parameter> ("AUTO TLM" on machining parameter screen) • "#8004 SPEED": 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 is shown below.

- ♦When the tool (measurement axis) reaches the sensor and the sensor signal detects it, movement on the axis stops.
- ♦When the tool (measurement axis) exceeds the measurement allowable range (a), movement on the axis stops.
- ♦When the sensor signal is detected within the measurement allowable range (a), the measurement is performed.
- ♦When the axis stops moving outside of the measurement allowable range (a), the program error (P607) occurs.



(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 1 mm/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 "0.2" (ms).
As a result, the measuring error shown below is caused.

Maximum measuring error [mm] = Measuring speed [mm/min] * 1/60 * 0.2 [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.

Maximum overtravel [mm] = Measuring speed [mm/min] * 1/60 * 1/Position loop gain [1/s]

The standard position loop gain is 33 (1/s).

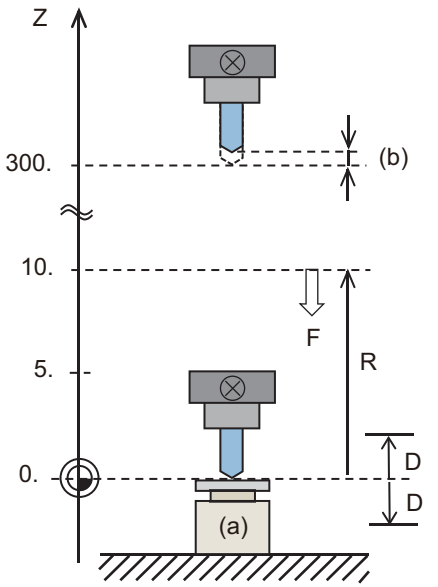
Operation example

This example of an operation is for the tool compensation memory type II.

Before using a new tool, measure the tool length compensation amount using the manual tool length measurement function. Set the measured adjustment amount to the "Length" item on the screen. Set "0" to the "L wear" item.

To measure the adjustment amount that is changed by using a tool, execute the automatic tool length measurement function (G37).
After G37 was executed, the change in the adjustment amount is added to the "L wear" item.

[mm]




```
G91 G28 Z0 ;
T01 ;
M06 ;
G90 G43 G00 Z300. H01 ;
G37 Z0. R10. D2. F100 ;
G00 Z10. ;
M02 ;
```

The difference between the measurement position and the commanded position is added to the "Length" item.

(a) Measuring device
(b) Wear amount

Note

(1) For tool compensation memory type II, this value is set to the "L wear" item. For tool compensation memory type I, this value is set as the compensation amount without distinguishing between tool length dimension and tool length wear amount. The tool compensation memory type depends on the MTB specifications (parameter "#1037 cmdtyp").



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_" 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 "#8005 ZONE r" > "D command" or parameter "#8006 ZONE d"

(8) When the "D" address and parameter "#8006 ZONE d" in (7) above are "0", 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 "#8005 ZONE r" and "#8006 ZONE d" in (7) above are all "0", 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 "G43 H_" 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 Dril_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

Skip command

G31 X__ Y__ Z__ α__ R__ F__ ;

X, Y, Z, α	Axis coordinate value They are commanded as absolute or incremental positions according to the G90/G91 modal when commanded. α is the additional axis.
R	Acceleration/deceleration command R0: Acceleration/deceleration time constant is "0". (No automatic acceleration/deceleration after interpolation) R1: Acceleration/deceleration time constant is valid. (Accelerates/decelerates with the time constant set with the parameters "#2102 skip_tL" and "#2103 skip_t1".) When omitted, R0 will be applied.
F	Feedrate (mm/min)

Multiple skip command

G31 D1 X__ Y__ Z__ α__ P__ R__ F__ ;

X, Y, Z, α	Axis coordinate value They are commanded as absolute or incremental positions according to the G90/G91 modal when commanded. α is the additional axis.
D	Skip signal command method When D1 is commanded, multiple skip (skip operation with logical product (AND) of "P__") is performed. When D0 is commanded or D command is omitted, multi-step skip 2 (G31 Pn method) is performed. For detail of the multi-step skip function 2, refer to "22.4 Multi-step Skip Function 2; G31 P".
P	Skip signal command (1 to 255) (*1) When there is no P command, it is handled as "P1". When a number with a decimal point is commanded, the decimal places are ignored.
R	Acceleration/deceleration command R0: Acceleration/deceleration time constant is "0". (No automatic acceleration/deceleration after interpolation.) R1: Acceleration/deceleration time constant is valid. (Accelerates/decelerates with the time constant set with the parameters "#2102 skip_tL" and "#2103 skip_t1".) When omitted, R0 will be applied.
F	Feedrate (mm/min)

(*1) The skip signals 1 to 8 can be used. Specify which signal of these to be used, with address P in the program. When D1 is commanded, if all the combination signals specified with P are turned ON, the skip operation is performed.
For detail of the specification method of P command, refer to "22.4 Multi-step Skip Function 2; G31 P".

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").

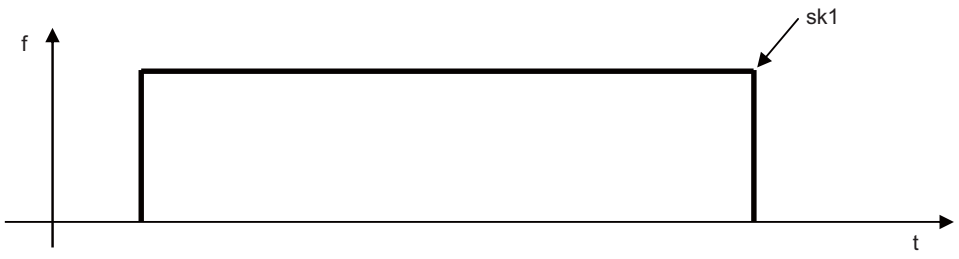
	#12022/bit2 = 0	#12022/bit2 = 1
Skip speed if F command is not programmed in the G31 block	The value of parameter "#1174 skip_F" is used as the skip speed.	The skip speed is determined based on the modality of F when G31 is executed.
	The program error (P603) occurs if the value of parameter "#1174 skip_F" is "0".	The program error (P62) occurs 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	F modal is not updated even if the G31 block contains an F command.	The F modal that is updated by 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.

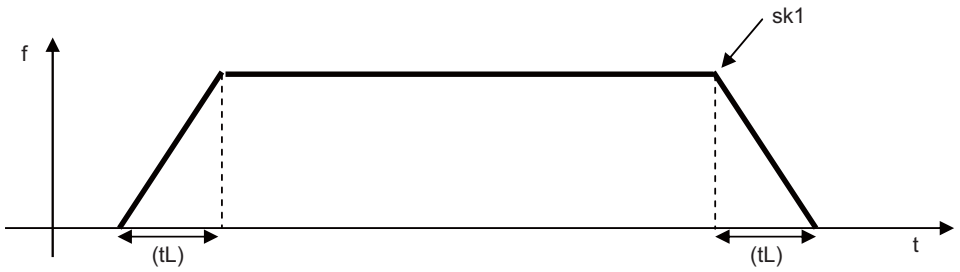
(3) When R0 is commanded or the R command is omitted, the step acceleration/deceleration will be applied to G31 block after the interpolation without performing the automatic acceleration/deceleration.
When R1 is commanded, the automatic acceleration/deceleration will be performed according to the cutting 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-gradient acceleration/deceleration ("#1201 G1_acc" is set to "1") is valid, the acceleration/deceleration with fixed time constant 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 there is no R command]



[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 operation is the same as when the acceleration/deceleration time constant is "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 functions, confirm 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 signal specified with address P 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) When the G31 command is issued during nose radius compensation or tool radius compensation, the program error (P608) occurs.

(10) When there is no F command in the G31 command and the parameter speed is also zero, the program error (P603) occurs.

(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.

(12) Command the acceleration/deceleration (R0/R1) for each G31 command. When there is no R command or when a command other than R0/R1 is issued, the operation is the same as that of when R0 is commanded (acceleration/deceleration time constant is "0") and automatic acceleration/deceleration after interpolation is not performed.

(13) Command the skip signal command method switch (D0/D1) whenever a G31 command is commanded. If a command other than D0/D1 is commanded, the program error (P35) occurs.

(14) For multiple skip (D1), once the skip signal specified with address P is turned ON, the state (ON) is kept. (Even when the skip signal is turned OFF after it is turned ON, the ON state is kept.)
However, if feed hold stop is made in this state, the skip signal is turned OFF. To perform skip stop after feed hold, the skip signal specified with the address P needs to be entered again.

(15) Specify an appropriate value for feedrate according to the touch probe specifications, etc.

(16) When a skip signal specified with address P is input, the skip coordinate values are stored in the system variables (#35201 or later, #35401 or later) corresponding to the skip signal.

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.
:
```

G31 coasting

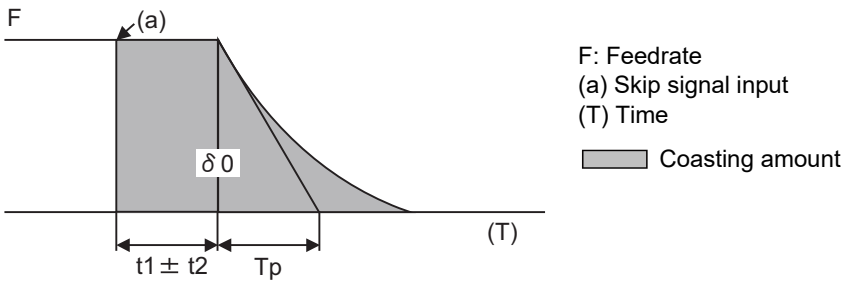
The amount of coasting from when the skip signal is input during the G31 command until the machine stops differs according to the parameter "#1174 skip_F" or F command in G31.
The time to start deceleration to stop after responding to the skip signal is short, so the machine can be stopped precisely with a small coasting amount. The coasting amount can be calculated from the following formula.

$$\delta 0 = \frac{F}{60} \times Tp + \frac{F}{60} \times (t1 \pm t2)$$

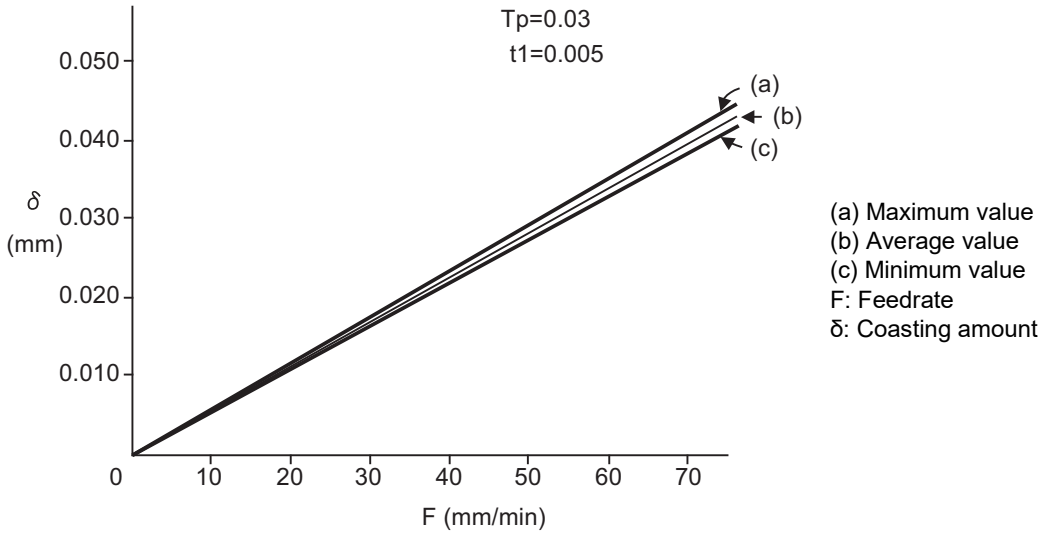
$$= \underbrace{\frac{F}{60} \times (Tp+t1)}_{\delta 1} \underbrace{\pm \frac{F}{60} \times t2}_{\delta 2}$$

δ0 : Coasting amount (mm)
F : G31 skip speed (mm/min)
Tp : Position loop time constant (s) = (Position loop gain)$^{-1}$
t1 : Response delay time (s) = (Time taken from the detection to the arrival of the "Skip" signal at the controller)
t2 : Response error time 0.001 (s)
When G31 is used for calculation, the value calculated from the section indicated by δ1 in the above equation can be compensated for, however, δ2 results in calculation error.

Stop pattern with skip signal input is shown below.



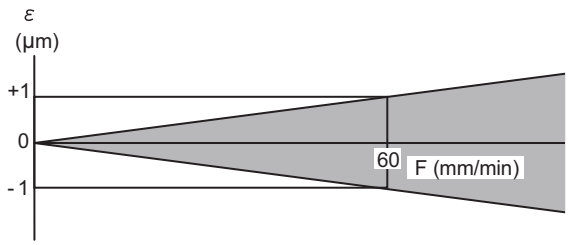
The relationship between the coasting amount and speed when "Tp" is 30 ms and "t1" is 5 ms is shown in the following figure.



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.



ε : Readout error
ε=±(F/60)×t 2
F : Feedrate
t2 : Response error time 0.001 (s)

Measurement value

Readout error of skip signal input coordinates

Readout error with a 60 mm/min feedrate is as shown below and the measurement value is within readout error range of ± 1 μm:

ε = ±(60/60) × 0.001 = ±0.001 (mm)

(2) Readout of coordinates other than skip signal input coordinate

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

```
#110 = 100; (Skip feedrate)
#111 = Response delay time t1;
:
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
:
```

(2) Compensating for workpiece coordinates

```
#110 = 100; (Skip feedrate)
#111 = Response delay time t1;
#112 = Position loop time constant Tp;
:
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
:
```

Operation 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").

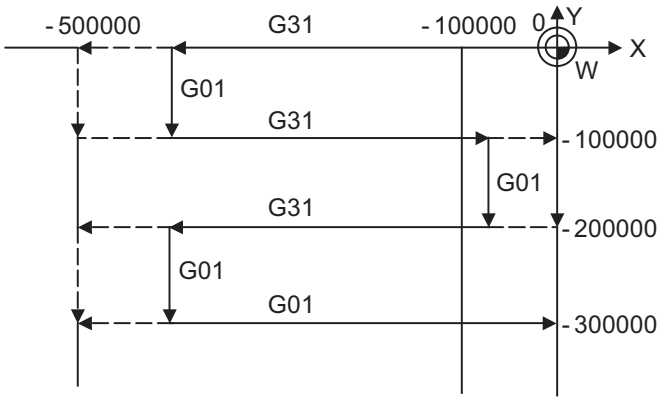
#1366	Operation
0	When any part system is executing the G31 command, the G31 command issued for other part systems is subjected to a block interlock state, and such G31 command will be executed after the current G31 command execution is completed. (No error is displayed.) In a single block operation, 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.

Also refer to the section of "Skip command on multiple part systems at the same time" in the operation example.

Operation example

(Example 1) Execution example of G31 command

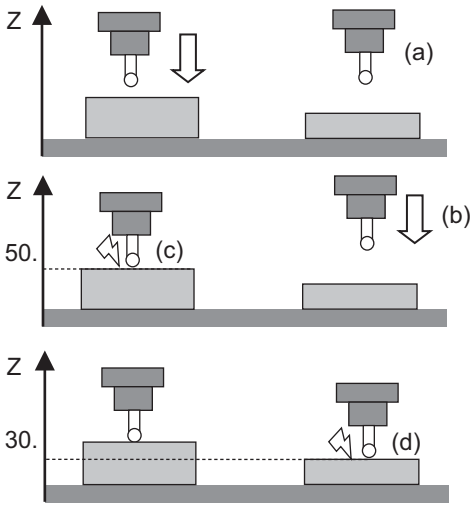
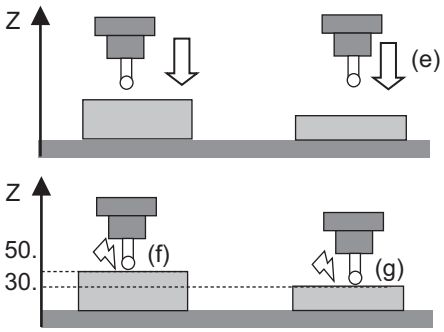
```
G90  G00  X-100000  Y0;
       G31  X-500000  F100;
       G01  Y-100000;
       G31  X-0  F100;
            Y-200000;
       G31  X-500000  F100;
            Y-300000;
            X0;
```



(Example 2) Skip command on multiple part systems at the same time

When the following machining program is executed, the operation varies depending on the setting of the parameter "#1366 skipExTyp".

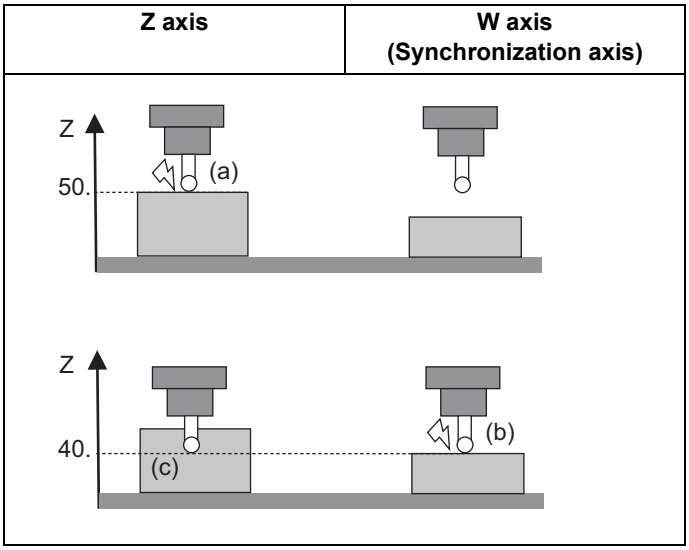
$1	G31 P1 Z40. F10 ; #501 = #5063;	Z1 axis when the skip signal 1 is ON
$2	G31 P2 Z20. F10 ; #502 = #5063;	Z2 axis when the skip signal 2 is ON

#1366 skipExTyp = 0		**#1366 skipExTyp = 1**	
Z1 axis ($1)	**Z2 axis ($2)**	**Z1 axis ($1)**	**Z2 axis ($2)**
			
(a) While G31 is commanded in the other part system ($1), the G31 command is subjected to a block interlock state. (b) After G31 ends in the other part system ($1), G31 is performed. (c) "50.000" is set in the common variable #501. (d) "30.000" is set in the common variable #502.		(e) G31 command is performed in the multiple part systems. (f) "50.000" is set in the common variable #501. (g) "30.000" is set in the common variable #502.	

(Example 3) Usage example of D1 (To stop by entering skip signals 1 and 2)

This is an example when the following machining program is executed.

G31 D1 P3 Z30. F10 ;	
#101 = #35203;	(Z axis when the skip signal 1 is ON)
#102 = #35224;	(W axis when the skip signal 2 is ON)

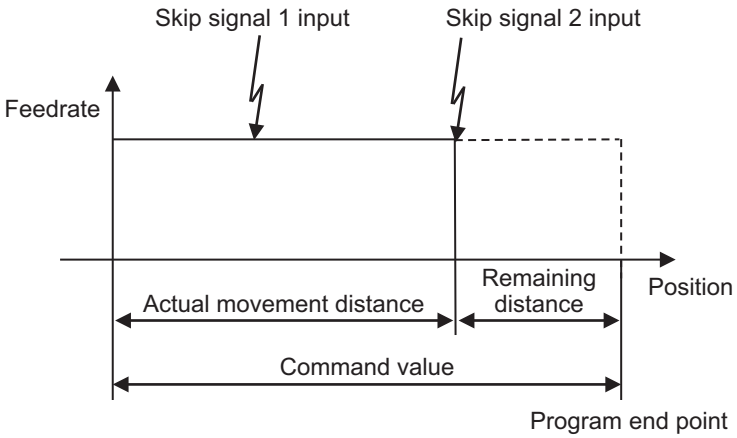


(a) The skip signal 1 is turned ON.

(b) The skip signal 2 is turned ON

Note

- In (c) of the figure, do not issue a movement command to move more than play of the touch probe. There is a risk that the touch probe is broken.



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).

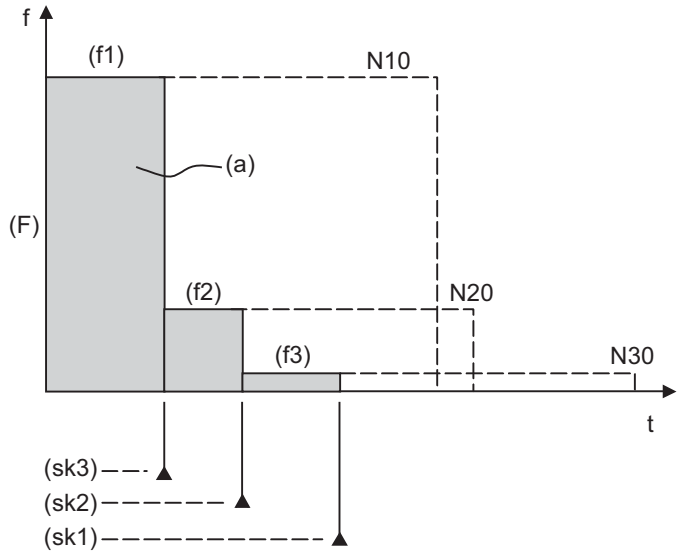
Operation example

(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)

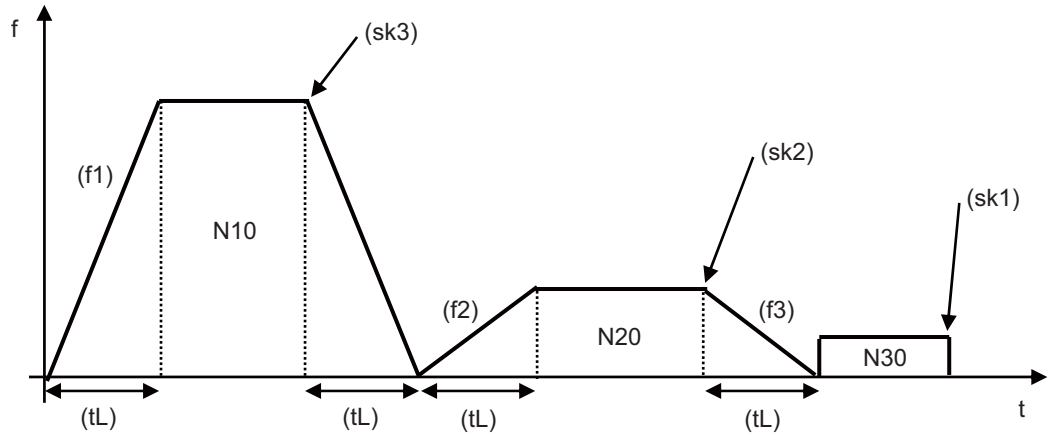
[Program example]
N10 G31.1 X200.0 ;
N20 G31.2 X40.0 ;
N30 G31.3 X1.0 ;



(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.



(sk1) Skip signal
(tL) Skip time constant

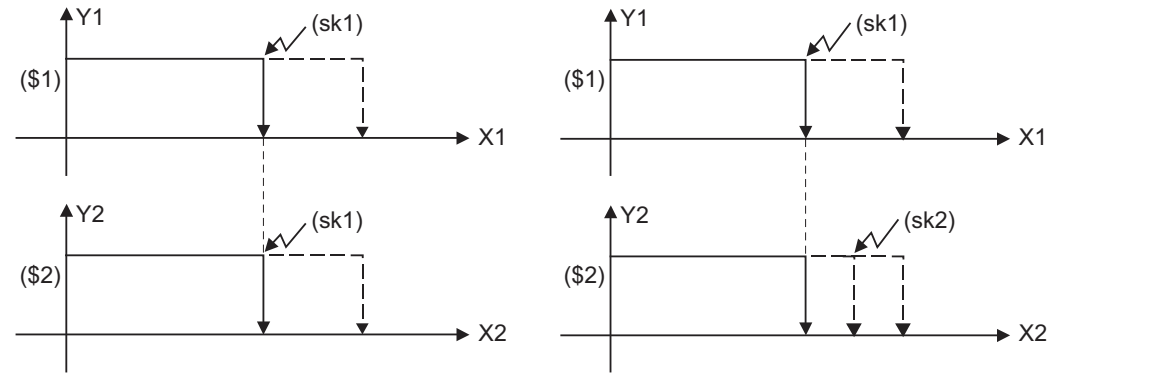
(2) If a skip signal with the condition set during G04 (dwell) is input, the remaining dwell time is canceled and the following block is executed.

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).



[Same skip signals input in both 1st and 2nd part systems]	[Different skip signals input in 1st and 2nd part systems]

($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 is "0". (No automatic acceleration/deceleration after interpolation.) R1: Acceleration/deceleration time constant is 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_". When there is no P command, it is handled as "P1". The command range of "P_" is from 1 to 255. If a value exceeding the range is commanded, the program error (P35) occurs.
When a number with a decimal point is commanded, the decimal places are ignored.

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 "P_" is not specified, it works as a skip function (G31), not as a multi-step skip function. If speed parameter "F_" 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 "22.2 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≠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.

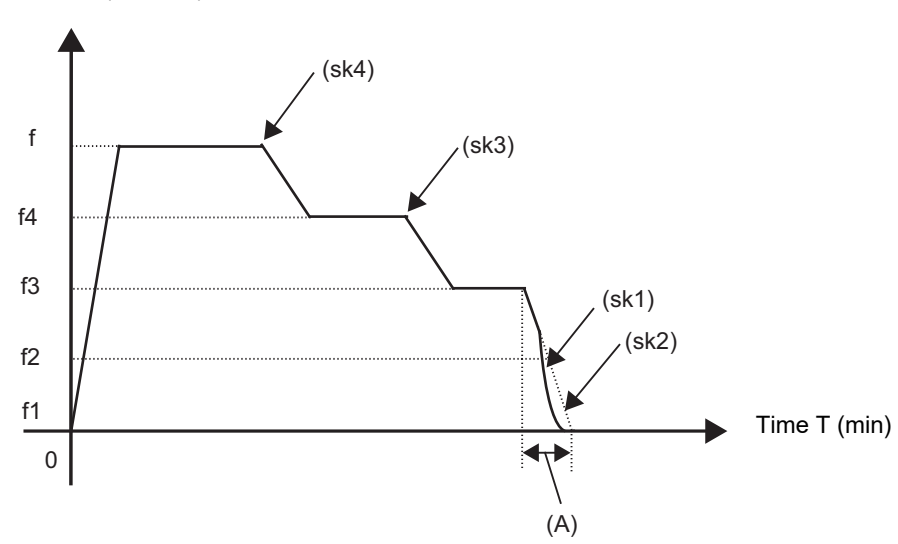
(5) Even if G1 constant-gradient acceleration/deceleration (#1201 G1_acc) is valid, the speed change skip will be the operation of the acceleration/deceleration with fixed time constant.

(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:

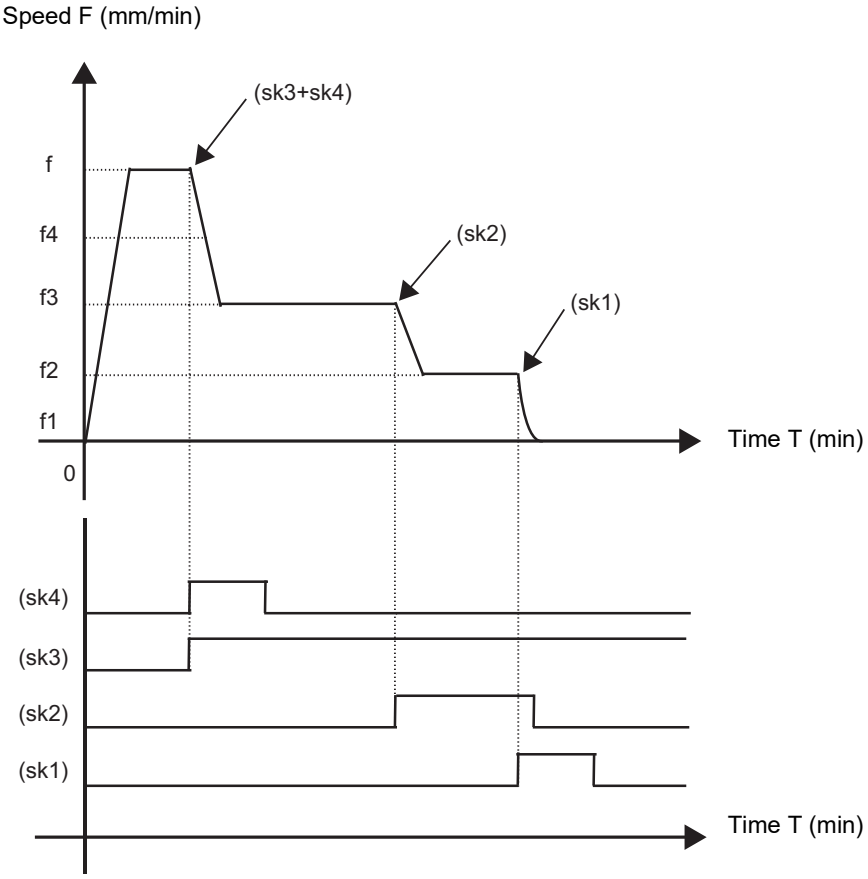
(a) A skip signal (sk2 in the figure) for changing speed is ignored.

(b) A skip signal (sk1 in the figure) for stopping the movement is executed and the speed is set to "0".



(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

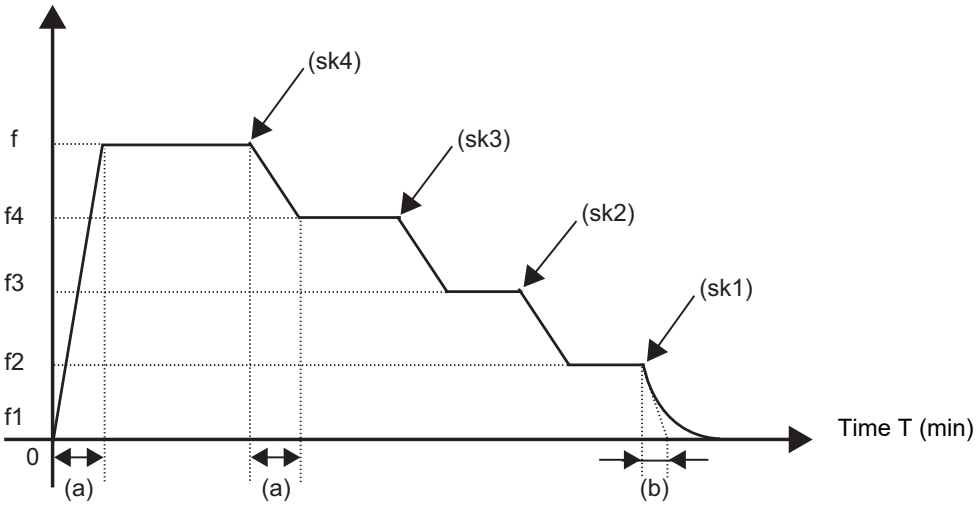
The following shows the operations when a skip time constant and skip signals, 1 (sk1) to 4 (sk4), are input.

(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)

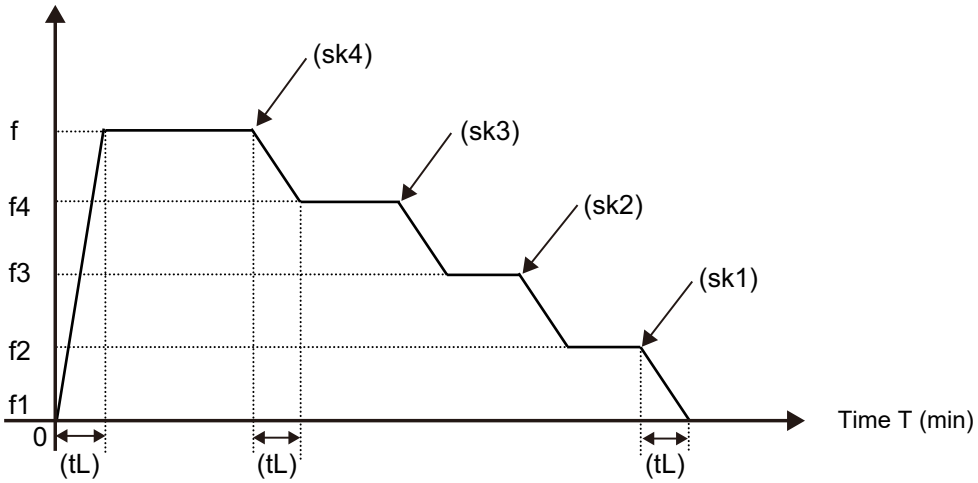


(2) Example of when R1 is commanded

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 exits 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-gradient 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.

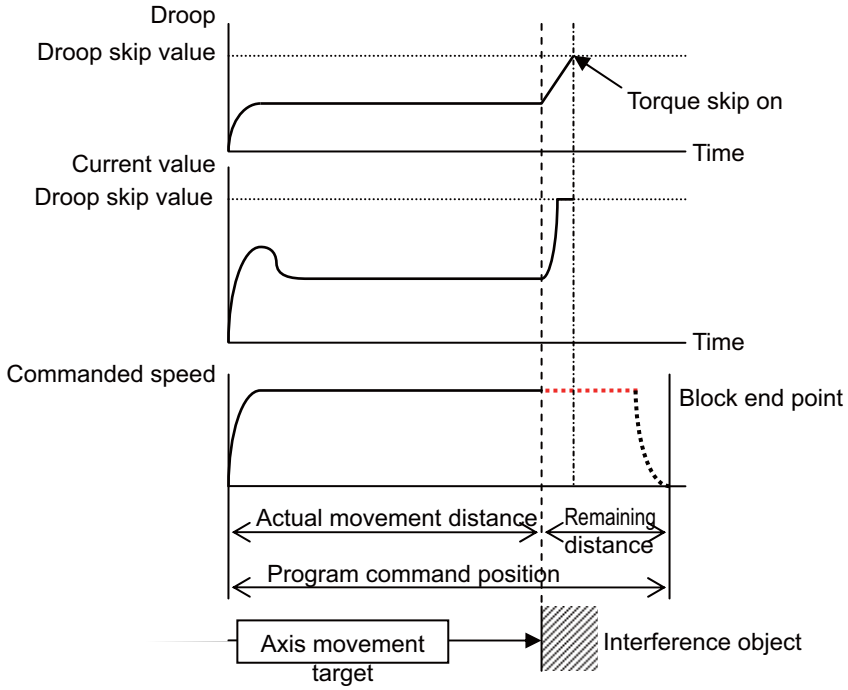
(2) The current position when the torque skip turns on is regarded as the block end point and the remaining distance (command value - actual movement distance) is discarded.

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.

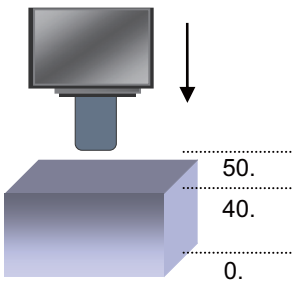
(3) Set the skip coordinate values (workpiece coordinate values) to system variables (#5061 and onwards). When the tool has moved to the end point, set the end point position.



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".

(7) For the programmable current limitation command (G10 L14) while the multiple-axis synchronization control is being executed, current limitation value specified for the master axis is applied to both of the master and slave axes. When the programmable current limitation command is issued to the slave axis while the multiple-axis synchronization control is being executed, the program error (P32) occurs.

23

System Variables

23.1 System Variables List

Mitsubishi Electric CNC provides the following system variables.
Note that the available types and numbers vary depending on the models, 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)	-	23.28
#1100 - #1135 #1300 - #1395	Signal output from NC to PLC	○ (*1)	○ (*1)	23.29
#1900, #1901	Used for normal line control.	○	○	23.25
#2001 - #2000+n #2201 - #2200+n #2401 - #2400+n #2601 - #2600+n	Tool offset data Also refer to the variables starting at #10001, #23001, and #135001.	○	○	23.6
#3000	Used to forcibly set to the alarm mode. Designate the number and message.	-	○	23.13
#3001, #3002	Cumulative time (integrating time)	○	-	23.15
#3001, #3002 #3011, #3012	Time read variables	○	○	23.16
#3003	♦Inhibition of single block stop ♦Inhibition of miscellaneous function finish signal waiting ♦Prohibition of program check reverse run	○	○	23.17
#3004	♦Automatic operation pause OFF ♦Cutting override OFF ♦G09 check OFF ♦Dry run invalid	○	○	
#3006	Used to display and stop a message.	-	○	23.14
#3007	Mirror image	○	-	23.20
#3019	Acquiring the program execution part system No.	○	-	23.21
#3020	Display axis name selection	○	○	23.22
#3901, #3902	Number of machining processes/Maximum number of machining processes	○	○	23.19
#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 #35201 - #35540+n #10005000 - #80005080+n	Position information by part system: #5001 - #5140+n, #35201 - #35540+n Position information common to part system: #1005000 - #80005080+n ♦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 23.12
#5201 - #5320+n	Workpiece coordinate offset data	○	○	23.8

No.	Data type or use	Reading	Setting	Section
#5500 #5501 - #5500+m	•Workpiece position offset for rotary axis: Offset No. •Workpiece position offset for rotary axis: Offset amounts (included in the group specified with "G54.2 Pn").	○	-	23.9
#5521 - #5661+m	Workpiece position offset for rotary axis: Offset amounts	○	○	
#7001 - #8900+n	Extended workpiece coordinate offset data (48- or 96-set specification)	○	○	23.10
#9000	Extended common variable III load file No.	○	○	23.34
#10001 - #10000+n #11001 - #11000+n #16001 - #16000+n #17001 - #17000+n #23001 - #23000+n	Tool offset data Also refer to the variables starting at #2001 and #135001.	○	○	23.6
#26000 - #26077	Workpiece installation error compensation amount	○	○	23.27
#30060 - #30068	Coordinate Rotation Parameter	○	-	23.23
#31001 - #31023	Rotary axis configuration parameter	○	-	23.24
#31100, #31101	Number of reverse run enable blocks, reverse run enable counter	○	-	23.18
#40000 - #40097	Specification of the selected interfering object and interfering model coordinate system offset	○	○	23.32
#50000 - #50749 #51000 - #51749 #52000 - #52749	Data of user backup area for R device	○ (*1)	○ (*1)	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	○	○	
#68121 - #68158				
#68081 - #68088	Tool life	○	○	
#68101 - #68113	Compensation amount	○	○	
#100000	Parameter No. designation	-	○	23.26
#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	○	○	23.33
#100100	Device type designation	-	○	23.31
#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)	○	○	23.10
#135001 - #135000+n #136001 - #136000+n #137001 - #137000+n #138001 - #138000+n	Tool offset data Also refer to the variables starting at #2001, #10001, and #23001.	○	○	23.6
#10003201 - #80003216	Axis exchange information acquisition	○	×	23.35
#10003221 - #80003236	Reference axis arrangement information acquisition	○	×	23.36

(*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	Stroke check before travel	G22:22, G23:23
#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	Polar coordinate interpolation	G07.1:7.1, G12.1:12.1, G13.1:13.1

(Example) The following shows a program example using the variables (#4003, #4203) to read the G command modal of the group 03.
In automatic operation, the program pre-read processing (analysis) is performed up to several blocks ahead. Regarding the modal acquisition of the block being executed (#42**), the acquired value may differ if the number of pre-read processing blocks changes in the high-accuracy control mode or similar.

G28 X0 Y0 Z0 ;	
G90 G1 X100. F1000 ;	
G91 G65 P300 X100. Y100. ;	
M02 ;	
O300 ;	
#1=#4003 ;	G code modal of Group 03 (pre-read) #1 = 91.0
#2=#4203 ;	G code modal of Group 03 (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 block	Execution block	
#4101	#4301		#4111	#4311	Tool length compensationNo. H (*5)
#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 (*2)(*4)	#4119	#4319	Spindle function S
#4110	#4310	Feedrate F (Feed per revolution) (*3)(*4)	#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 of "0x31", "0x32", and "0x33". Thus, the value is as follows:

(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".

(*2) For the variable Nos. #4109 and #4309, the read value changes as follows according to the combination of the parameter "#1752 cfgPR02/bit5" setting and the modal (G94/G95). (This depends on the MTB specifications.)

#1752 cfgPR02/bit5	Read value	
	During G94 modal	During G95 modal
0	F command value of feed per minute	
1	F command value of feed per minute	F command value of feed per revolution

The F command value issued last during the G94 (Feed per minute) or G95 (Feed per revolution) modal can be read.
When the parameter "#1752 cfgPR02/bit5" is "0", even when the F command value is changed after switch to the G95 (Feed per revolution) modal, the value to be acquired by this variable does not change.

(*3) The F command value issued last during the G95 (Feed per revolution) modal can be read.
Even when the F command value is changed after switch to the G94 (Feed per minute) modal, the value to be acquired by this variable does not change.

(*4) They are for readout only. When writing is performed, the program error (P241) occurs.

(*5) The value acquired with #4111 or #4311 differs depending on the MTB specifications (parameter "#1274 ext10/ bit3").

#1274 ext10/ bit3	Read value	
	During G43/G44 modal	Other than G43/G44 modal
0	Tool length compensation No.	0 (fixed)
1	Tool length compensation No.	

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 : #4421	G code (group 01) : G code (group 21)	Some groups are not used.
#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".

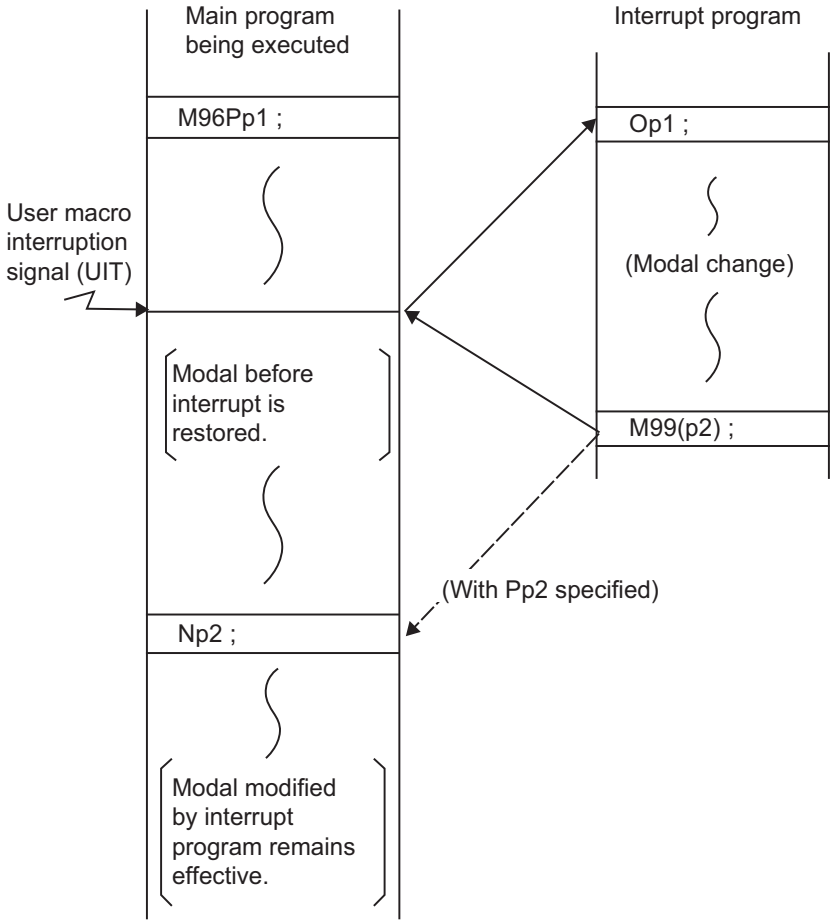
(Example 2)

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)

<table>
<tr><th>Variable No.</th><th colspan="2">Item / Description</th><th>Data range</th><th>Attribute</th></tr>
<tr><td>#68000</td><td>Tool designation method</td><td>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</td><td>1 to 3</td><td>-/W</td></tr>
<tr><td>#68001</td><td>Tool selection No.</td><td>Designate the tool selection number that matches the setting of "#68000".
<table>
<tr><th>#68000</th><th>Details of "#68001"</th><th>Data range</th></tr>
<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>
</table></td><td>Refer to the "Description" column.</td><td>-/W</td></tr>
<tr><td>#68003</td><td>Top vacant registration number on tool management screen</td><td>The tool number indicates a vacant line number.
0: No vacant registration number
1 to 999: Vacant registration number</td><td>0 to 999</td><td>R/-</td></tr>
</table>

- 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) occurs.

(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 "#68158".
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 "#68158" 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 "#68158" without commanding "#68001";

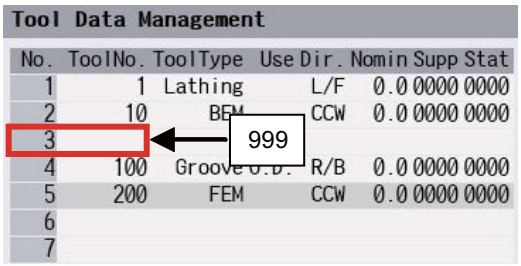
"#68001=0 ;" is commanded.

(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.



[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 J		±9999.999 (mm) ±999.9999 (inch)	R/W

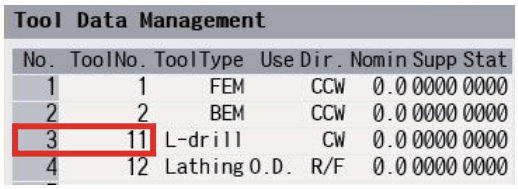
♦If a value exceeding the allowable range is issued, a program error (P35) occurs.

(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.



#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 - #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) occurs.

Cutting conditions ("#68051" to "#68054")

Variable No.	Item / Description	Data range	Attribute
#68051	Spindle rotation speed S	0 to 99999999	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 99999999	R/W

♦If a value exceeding the allowable range is issued, a program error (P35) occurs.

Additional information ("#68061" to "#68072", "#68121" to "#68158")

Variable No.	Item / Description	Data range	Attribute
#68061 - #68066	Customized data 1 to 6	±999999999 (*1)	R/W
#68067 - #68072	Customized data 7 to 12	±9999.999 (*1)	R/W
#68121 - #68158	Customized data 13 to 50	±999999999 (*1)	R/W

(*1) For customized data 1 to 50, the data range varies depending on the data format.

♦If a value exceeding the allowable range is issued, a program error (P35) occurs.
♦Customized data 13 to 50 (#68121 to #68158) can be commanded when the parameter "#1766 cfgPR16/bit4" is "1". If the customized data 13 to 50 are commanded when the parameter is "0", the program error (P241) occurs.

Tool life (#68081 to #68088)

Variable No.	Item / Description			Attribute
	Tool life management I	Tool life management II	Tool 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	Estimated tool life (hours)/Estimated number of cuttings (0 to 4000 min. / 0 to 65000 sets)			R/W
#68086	Cumulative cutting hours/Cumulative number of cuttings (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) occurs.
- If a value exceeding the allowable range is issued, a program error (P35) occurs.

Compensation amount ("#68101" to "#68113")

Variable No.	Item / Description			Attri-bute
	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) ±999.9999999 (inch))	Length dimension (±9999.999999 (mm) ±999.9999999 (inch))	Tool length X (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68104	(Not used)	(Not used)	Tool length Z (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68105	(Not used)	(Not used)	Additional axis tool length (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68106	(Not used)	Length wear (±9999.999999 (mm) ±999.9999999 (inch))	Wear X (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68107	(Not used)	(Not used)	Wear Z (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68108	(Not used)	(Not used)	Additional axis wear (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68109	(Not used)	Radius dimension (±9999.999999 (mm) ±999.9999999 (inch))	Tool nose radius (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68110	(Not used)	Radius wear (±9999.999999 (mm) ±999.9999999 (inch))	Radius wear (±9999.999999 (mm) ±999.9999999 (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 [M8] (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68113	(Not used)	(Not used)	2nd additional axis wear [M8] (±9999.999999 (mm) ±999.9999999 (inch))	R/W

- If an unused variable is commanded, a program error (P241) occurs.
- If a value exceeding the allowable range is issued, a program error (P35) occurs.



Precautions

(1) When the tool management data write command is executed during graphic check for M80V Series, the write is actually performed and the original data is not restored.

(2) The system variables #68000 to #68158 are pre-read prohibition commands.

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 - #10000+n	#2001 - #2000+n	○	○ (*1)	○ Z axis shape (*1)
#11001 - #11000+n	#2201 - #2200+n	×	○ (*2)	○ Z axis wear (*2)
#16001 - #16000+n	#2401 - #2400+n	×	○ (*3)	○ Tool nose radius shape (*3)
#17001 - #17000+n	#2601 - #2600+n	×	○ (*4)	○ Tool nose radius wear (*4)
#135001 - #135000+n	-	×	×	○ X axis wear
#136001 - #136000+n	-	×	×	○ Y axis wear
#137001 - #137000+n	-	×	×	○ X axis shape
#138001 - #138000+n	-	×	×	○ Y axis shape
#23001 - #23000+n	-	×	×	○ Tool nose point

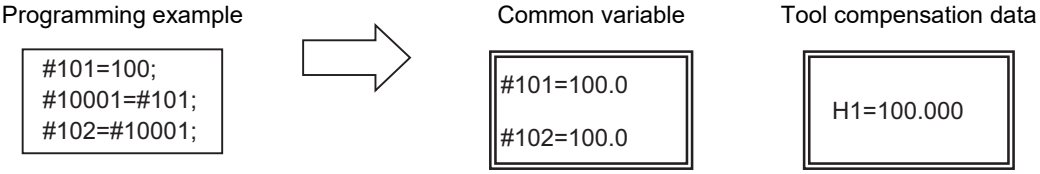
(*1) Length dimension

(*2) Length wear

(*3) Radius dimension

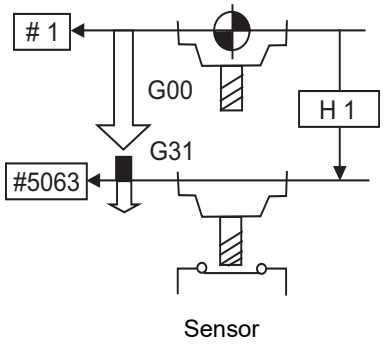
(*4) Radius wear

"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.



(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.

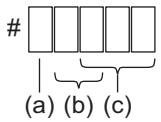
(1) Group number designation

#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) Fixed to "6" (Tool life management)

(b) 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	Estimated tool life data	Refer by registration No.
30	Cumulative 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 estimated tool life data are common to all groups.

(c) Registration No. or data type

Registration No.

1 to 200

Data type

Type	Details
1	Number of registered tools
2	Tool life data (current value)
3	Tool selection No.
4	Number of remaining registered tools
5	Execution signal
6	Cumulated cutting time (min)
7	Signal to indicate tool life over
8	Signal to indicate tool life prediction

List of variables

Variable No.	Item	Type	Details	Data range
60001	Number of registered tools	Common to part systems	"Cumulative cutting hours" and "Cumulative number of cuttings" of a tool in use Cumulative usage data of the tool in use (total of usage data for each compensation number when multiple compensation numbers are used for the tool)	0 to 200
60002	Tool life data (current value)	For each group (*1)	"Cumulative cutting hours" and "Cumulative number of cuttings" of a tool in use Cumulative usage data of the spindle tool or of the tool in use (#60003)	0 to 4000 min 0 to 9999 sets
60003	Tool selection No.		Tool registration number of the tool in use This number is a registration number of the spindle tool. Supplimentary explanation for the selected tool is provided as follows: ♦If spindle tool does not belong to the designated tool, the first tool of the tools with "ST" set to "1" is selected. ♦If there is no tool with "ST" set to "1", the first tool of the tools with "ST" set to "0" is selected. ♦If all tools have reached the end of their tool lives, the last tool is selected.	0 to 1000
60004	Number of remaining registered tools		This is the registration number of first tool which does not reach the end of tool life.	0 to 1000
60005	Execution signal		The value is set to "1" when this group is used in the program being executed. The value is set to "1" when the group number of the spindle tool data matches the group number of the specified group.	0/1
60006	Cumulated cutting time (min)		Indicates the accumulated time that this group was used in the program being executed.	(Not used)
60007	Signal to indicate that tool life is over		The value is set to "1" when all tools in this group have reached the end of their tool lives. The value is set to "1" when all registered tools in the designated group have reached the end of their tool lives.	0/1
60008	Signal to indicate tool life prediction		The value is set to "1" when a new tool is to be selected in the next command of this group. The value is set to "1" when there is an unused tool (ST" is set to "0") and no tool in use ("ST" is set to "1") among the registered tools in the designated group.	0/1

(*1) Designate group number "#60000".

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 for the data such as usage data count method, tool length compensation format and tool radius compensation format bit0,1: Tool length compensation data format bit2,3: Tool radius compensation data format 0: Compensation number designation method 1: Incremental compensation amount method 2: Absolute compensation amount method bit4,5: Tool life management method 0: Cumulative cutting hours 1: Number of times that tool was mounted 2: Number of times that tool was used	0 to FF (H)
62000 +***	Tool status		The state of tool usage 0: Unused tool 1: Tool in use 2: Tool which is consuming its tool life as planned 3: Tool error 1 4: Tool error 2	0 to 4
62500 +***	Estimated tool life data		Estimated tool life (hours) or estimated number of cuttings for each tool (*3)	0 to 4000 min 0 to 9999 sets
63000 +***	Cumulative usage data		Cumulative cutting hours or cumulative number of cuttings for each tool (*3)	0 to 4000 min 0 to 9999 sets
63500 +***	Tool length compensation data		Tool length compensation data set as either compensation No. or length dimension. (*4)	Compensation No.: "0" to the number of tool compensation sets Length dimension: ±99999.999
64000 +***	Tool radius compensation data		Tool radius compensation data set as either compensation No. or radius dimension. (*5)	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.

(*3) It depends on the setting of the tool life management method to manage the tool life by either cumulative cutting hours or cumulative number of cuttings.

(*4) It depends on the setting of tool length compensation data format to compensate the tool length by either compensation No. or the tool length data.

(*5) It depends on the setting of tool radius compensation data format to compensate the tool radius by either compensation No. or the tool radius data.



Program example

(1) Normal commands

#101 = #60001 ;	Reads the number of registered tools.
#102 = #60002 ;	Reads the tool life data (current value).
#103 = #60003 ;	Reads the tool selection No.
#60000 = 10 ;	Designates the group No. of the tool of which the tool life data is to be read. Designated group No. is valid until reset.
#104 = #60004 ;	Reads the remaining number of registered tools of group 10.
#105 = #60005 ;	Reads the execution signal of group 10.
#111 = #61001 ;	Reads the tool No. allocated to #1 of group 10.
#112 = #62001 ;	Reads the status displayed in #1 of group 10.
#113 = #61002 ;	Reads the tool No. allocated to #2 of group 10.
%	

(2) When the group number is not designated:

#104 = #60004 ;	Reads the remaining number of registered tools of the first registered group.
#111 = #61001 ;	Reads the tool No. of #1 of 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 ;	The value of #104 will be "-1".

(4) When an unused registration number is designated (15 tools for group 10):

#60000 = 10 ;	Designates the group No.
#111 = #61016 ;	The value of #111 will be "-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 registration of the tool life management data. The tool life data of group 10 is registered through the commands between G10 and G11.
P10 LLn NNn ;	"10" is the group No., "Ln" is the tool life for a 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 tool No. allocated to #1 of group 10.
G10 L3;	Starts registration of the tool life management data. The tool life data of the group other than group 10 is registered through the commands between G10 and G11.
P1 LLn NNn ;	"1" is the group No., "Ln" is the tool life for a tool, "Nn" is the method.
TTn ;	"Tn" is the tool No.
:	
G11 ;	Registers the tool life data with the G10 command. (The registered data is deleted.)
#111 = #61001 ;	Group 10 does not exist. The value of #111 will be "-1".



Precautions

(1) If a system variable for tool life management is commanded without designating a group No., the data of the group which comes at the head of all the registered groups will be read.

(2) If a system variable for tool life management is commanded with designating non-registered group No., "-1" will be read as the data.

(3) If a system variable for tool life management is commanded with a registered No. which is not used, "-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 is used, 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.

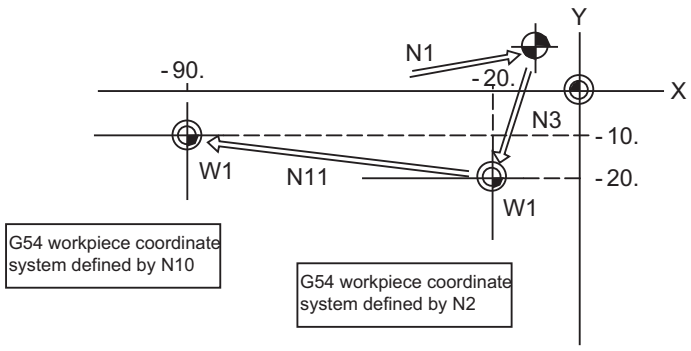
Note

(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)

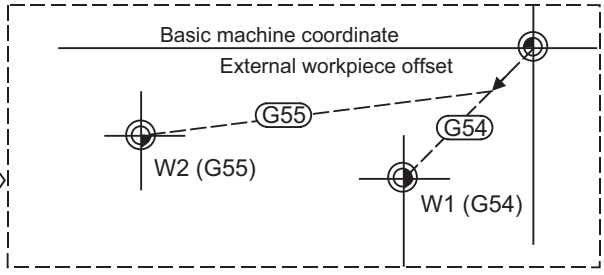
```
N1 G28 X0 Y0 Z0 ;
N2 #5221=-20. #5222=-20. ;
N3 G90 G00 G54 X0 Y0 ;
      ≀
N10 #5221=-90. #5222=-10. ;
N11 G90 G00 G54 X0Y0 ;
      ≀
M02 ;
```



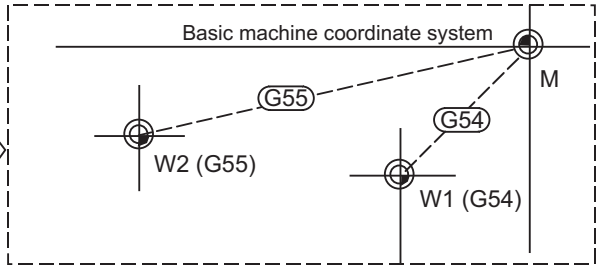
(Example 2)

```
      ≀
N100 #5221=#5221+#5201 ;
     #5222=#5222+#5202 ;
     #5241=#5241+#5201 ;
     #5242=#5242+#5202 ;
     #5201=0 #5202=0;
      ≀
```

Coordinate system before change



Coordinate system after change



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 (Workpiece Position Offset Amount for Rotary Axis)



Function and purpose

The workpiece position offset amount for rotary axis can be read and written with system variables.
The workpiece position offset amount for rotary axis can be set for 8 sets.
The workpiece position offset amount for rotary axis which was specified with the address P of "G54.2 Pn;" command can be read with read-only system variables.

[Setting the workpiece position offset amount for rotary axis]

Compensation No. (Set No.)	1st axis	2nd axis	3rd axis	...	m-th Axis (m: Number of NC axes in a part system)	Attribute
1	#5521	#5522	#5523	...	#5520 + m	read/write
2	#5541	#5542	#5543	...	#5540 + m	read/write
3	#5561	#5562	#5563	...	#5560 + m	read/write
4	#5581	#5582	#5583	...	#5580 + m	read/write
5	#5601	#5602	#5603	...	#5600 + m	read/write
6	#5621	#5622	#5623	...	#5620 + m	read/write
7	#5641	#5642	#5643	...	#5640 + m	read/write
8	#5661	#5662	#5663	...	#5660 + m	read/write

[Read of the workpiece position offset amount for rotary axis which was specified with the address "P" of "G54.2 Pn;" command]

Variable No.	Data	Attribute
#5500	Workpiece position offset for rotary axis No. (offset set No. "n" which was specified with the address "P")	read
#5501	Workpiece position offset amount for rotary axis (1st axis of the n-th set)	read
#5502	Workpiece position offset amount for rotary axis (2nd axis of the n-th set).	read
#5503	Workpiece position offset amount for rotary axis (3rd axis of the n-th set)	read
:	:	
#5500 + m (m: 1 to number of NC axes in a part system)	Workpiece position offset amount for rotary axis (m-th axis of the n-th set)	read

(1) The position of the decimal point is determined based on the setting of parameter "#1003 iunit" (input setting unit).

(2) The values set with the system variables are acquired for the workpiece position offset amount for rotary axis of the specified set. It is not the offset amount after the conversion in accordance with the position (angle) of the rotary axis.

[Setting example]

Machine configuration	Table tilt type
Primary rotary axis	A axis
Secondary rotary axis	C axis
Axis configuration	The X axis, Y axis, Z axis, A axis, and C axis are arranged in order from the first axis.

When C axis is 10°, if an error of 5.0 mm is found in the machine above, give a command as "#5521=5.0" and "#5525=10.0".

23.10 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] System 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 - #700n	P25	#7481 - #748n	P49	#7961 - #796n	P73	#8441 - #844n
P2	#7021 - #702n	P26	#7501 - #750n	P50	#7981 - #798n	P74	#8461 - #846n
P3	#7041 - #704n	P27	#7521 - #752n	P51	#8001 - #800n	P75	#8481 - #848n
P4	#7061 - #706n	P28	#7541 - #754n	P52	#8021 - #802n	P76	#8501 - #850n
P5	#7081 - #708n	P29	#7561 - #756n	P53	#8041 - #804n	P77	#8521 - #852n
P6	#7101 - #710n	P30	#7581 - #758n	P54	#8061 - #806n	P78	#8541 - #854n
P7	#7121 - #712n	P31	#7601 - #760n	P55	#8081 - #808n	P79	#8561 - #856n
P8	#7141 - #714n	P32	#7621 - #762n	P56	#8101 - #810n	P80	#8581 - #858n
P9	#7161 - #716n	P33	#7641 - #764n	P57	#8121 - #812n	P81	#8601 - #860n
P10	#7181 - #718n	P34	#7661 - #766n	P58	#8141 - #814n	P82	#8621 - #862n
P11	#7201 - #720n	P35	#7681 - #768n	P59	#8161 - #816n	P83	#8641 - #864n
P12	#7221 - #722n	P36	#7701 - #770n	P60	#8181 - #818n	P84	#8661 - #866n
P13	#7241 - #724n	P37	#7721 - #772n	P61	#8201 - #820n	P85	#8681 - #868n
P14	#7261 - #726n	P38	#7741 - #774n	P62	#8221 - #822n	P86	#8701 - #870n
P15	#7281 - #728n	P39	#7761 - #776n	P63	#8241 - #824n	P87	#8721 - #872n
P16	#7301 - #730n	P40	#7781 - #778n	P64	#8261 - #826n	P88	#8741 - #874n
P17	#7321 - #732n	P41	#7801 - #780n	P65	#8281 - #828n	P89	#8761 - #876n
P18	#7341 - #734n	P42	#7821 - #782n	P66	#8301 - #830n	P90	#8781 - #878n
P19	#7361 - #736n	P43	#7841 - #784n	P67	#8321 - #832n	P91	#8801 - #880n
P20	#7381 - #738n	P44	#7861 - #786n	P68	#8341 - #834n	P92	#8821 - #882n
P21	#7401 - #740n	P45	#7881 - #788n	P69	#8361 - #836n	P93	#8841 - #884n
P22	#7421 - #742n	P46	#7901 - #790n	P70	#8381 - #838n	P94	#8861 - #886n
P23	#7441 - #744n	P47	#7921 - #792n	P71	#8401 - #840n	P95	#8881 - #888n
P24	#7461 - #746n	P48	#7941 - #794n	P72	#8421 - #842n	P96	#8901 - #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] System variables for extended workpiece coordinate system compensation (n=1 to 8)

	1st axis to nth axis
P1	#101001 - #10100n
P2	#101051 - #10105n
P3	#101101 - #10110n
P4	#101151 - #10115n
P5	#101201 - #10120n
P6	#101251 - #10125n
P7	#101301 - #10130n
P8	#101351 - #10135n
:	:
:	:
P298	#115851 - #11585n
P299	#115901 - #11590n
P300	#115951 - #11595n

23.11 System Variables (Position Information by Part System)



Detailed description

Using system variables #5001 to #5160+n, it is possible to read the position information shown below.

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	Parameter #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 (*1)				#5121	#5122	#5123	...	#5120+n	Enabled
End point coordinates of the block that was interrupted by macro interruption (*1)				#5141	#5142	#5143	...	#5140+n	Enabled

(*1) When the arbitrary axis exchange is performed during macro interruption, the start point and end point coordinates in the block interrupted by the macro interruption cannot be correcly obtained.

When address P is commanded to the G31 (skip) command, the system variables corresponding to the specified skip signal are as follows:

Position information (Skip coordinate)			Axis No. (in part system)					Reading during movement
			1	2	3	...	n	
Parameter #8713	0: Workpiece coordinate system 1: Feature coordinate/Workpiece installation coordinate	Skip signal 1	#35201	#35202	#35203	...	#35200+n	Enabled
		Skip signal 2	#35221	#35222	#35223	...	#35220+n	
		Skip signal 3	#35241	#35242	#35243	...	#35240+n	
		Skip signal 4	#35261	#35262	#35263	...	#35260+n	
		Skip signal 5	#35281	#35282	#35283	...	#35280+n	
		Skip signal 6	#35301	#35302	#35303	...	#35300+n	
		Skip signal 7	#35321	#35322	#35323	...	#35320+n	
		Skip signal 8	#35341	#35342	#35343	...	#35340+n	
Feature coordinate/Workpiece installation coordinate		Skip signal 1	#35401	#35402	#35403	...	#35400+n	
		Skip signal 2	#35421	#35422	#35423	...	#35420+n	
		Skip signal 3	#35441	#35442	#35443	...	#35440+n	
		Skip signal 4	#35461	#35462	#35463	...	#35460+n	
		Skip signal 5	#35481	#35482	#35483	...	#35480+n	
		Skip signal 6	#35501	#35502	#35503	...	#35500+n	
		Skip signal 7	#35521	#35522	#35523	...	#35520+n	
		Skip signal 8	#35541	#35542	#35543	...	#35540+n	

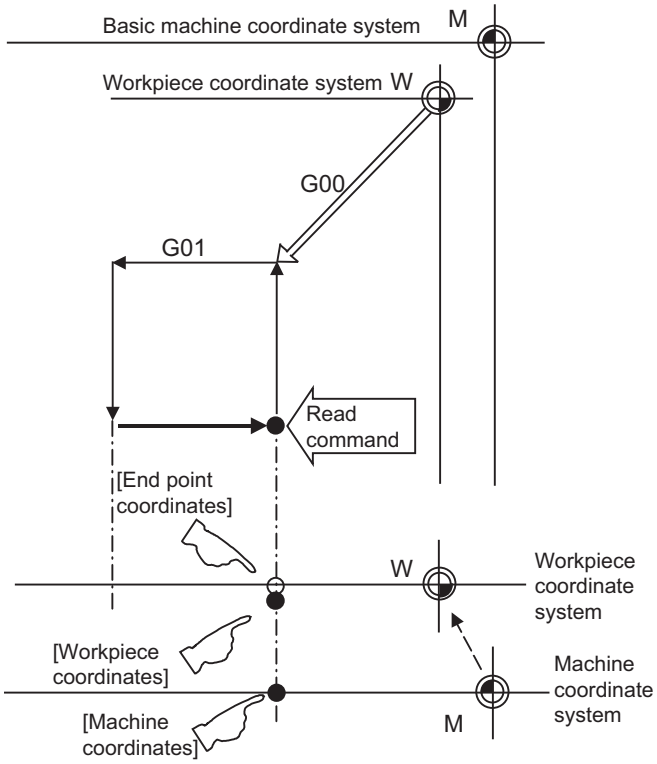
Note

- 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.

Data units and unit systems (Inch/metric)

The position information acquisition variables are acquired according to the setting of "#1004 ctrl_unit" (control unit). The unit system follows the inch/metric command (G20/G21) if "#1041 I_inch" is "0" and "#1226 aux10/bit6" is "1". Otherwise, it follows the setting of "#1041 I_inch".

Detailed explanation of each position information



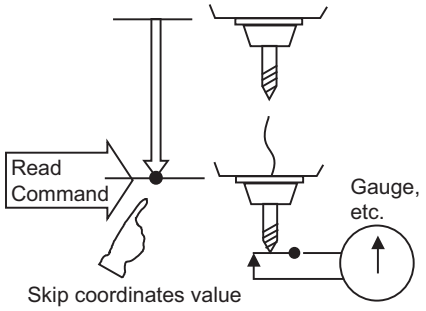
Note

- For system variables #5001 to #5100+n (excluding #5021 to #5020+n) that read the position information, the coordinate positions in the feature coordinate system are set if the inclined surface machining command is in progress.
  For #5021 to #5020+n (machine coordinate positions), the coordinate positions in the machine coordinate system, not the feature coordinate system, are set even while the inclined surface machining command is in progress.

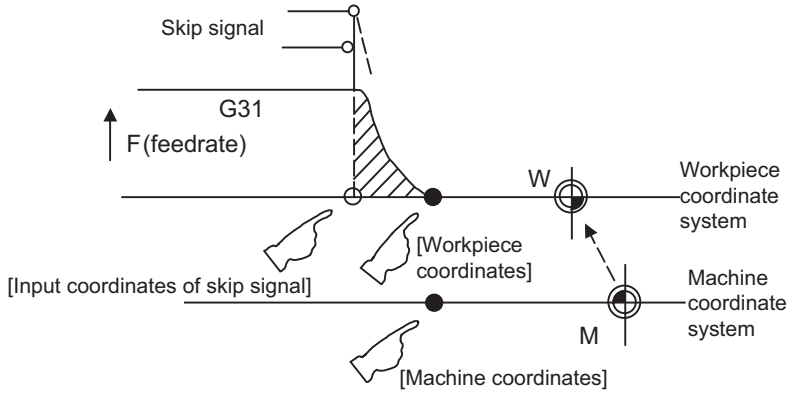
(1) The position of the end point coordinates is position on the workpiece coordinate system.

(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.



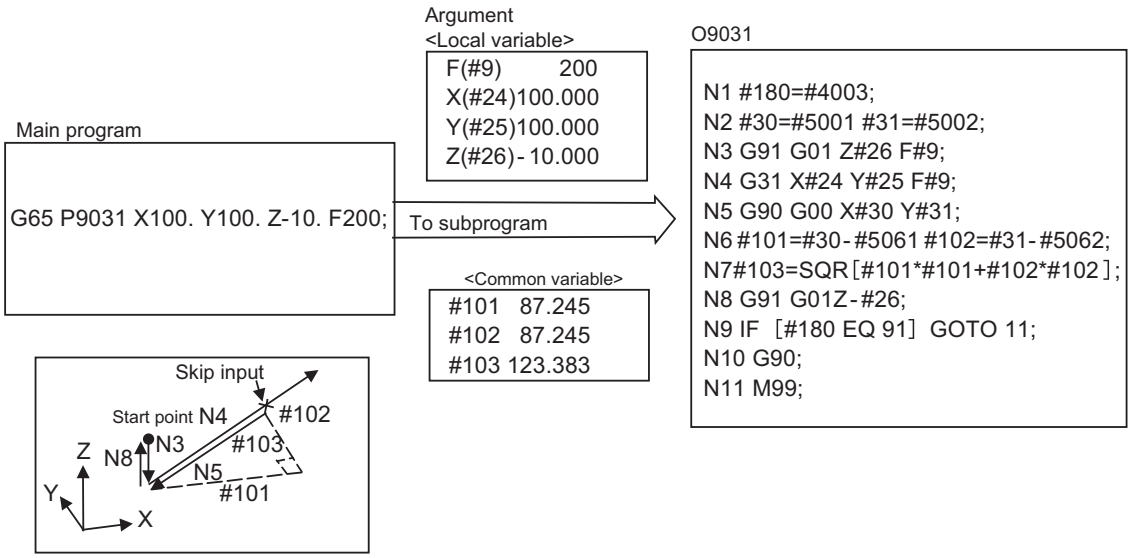
For "●", check stop and then proceed to read.
For "○", reading is possible during movement.

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 the orthogonal axis for the axis of the part system that is 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 position of the skip coordinates is overwritten.
- Skip coordinate position 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 position 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 position 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 selection) is set to "1", the skip coordinate position will be "0", even if G31 command is given in the one-part system or G31 command is given in only one of the multiple part systems.
- The variable to acquire the position information by part system (#5001 to #5160+n) is a read-only variable. If a write command is issued to these system variables, no error occurs, and the block will be ignored.

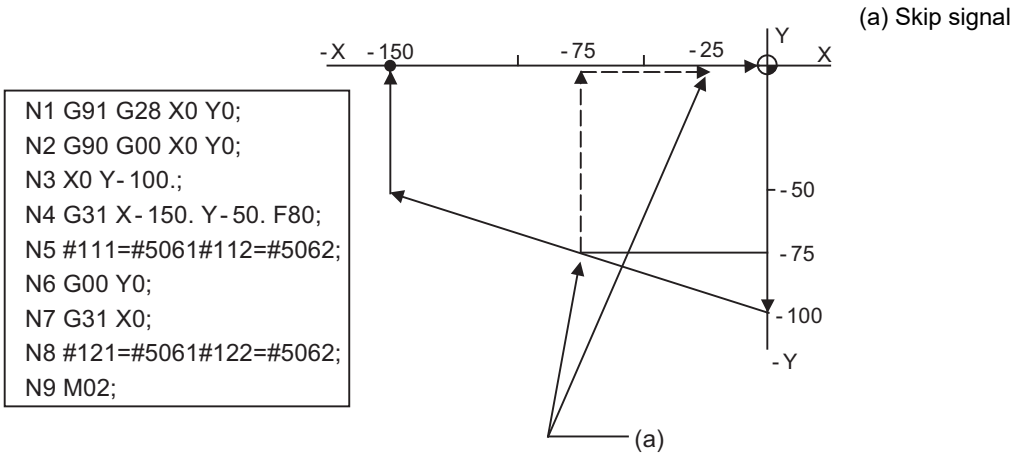
(Example 1) Example of workpiece position measurement

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 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



#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.)
The skip signal input coordinates of N4 block will be read to #122 because there is no Y command in the N7 block

Relationship with other functions

(1) Axis name extension

When a maximum of 16 axes are set for one part system by axis name extension, up to the 16th axis can be acquired as the position information acquisition variable.

(2) Polar coordinate interpolation

The position information acquisition variable that acquires the coordinate position of the machine coordinate system acquires the actual machine position moved by the polar coordinate interpolation.

(3) Tool compensation

When the position information acquisition variable for the tool compensation amount is commanded, the tool compensation amount specified by the tool position offset (G45 to G48) is acquired.

The tool compensation amount specified by the tool length compensation command (G43/G44) is acquired as "0".

(4) High-speed simple program check

When the coordinate position of the machine coordinate system is acquired during the high-speed simple program check operation, the commanded machine coordinate position is acquired.

(5) Arbitrary reverse run, Manual arbitrary reverse run

At the time of the reverse run, the position information is not acquired by the position information acquisition variable.
The operation is performed with the value of the position information acquisition variable acquired at the time of the forward run.

(6) Mirror image

The position information acquisition variable that acquires the coordinate position of the machine coordinate system acquires the coordinate position moved by the mirror image.

(7) Arbitrary axis exchange control

The axis number that specifies the axis with the variable to acquire the position information by part system is the axis number in the part system. Therefore, if the axis configuration in the part system changes due to arbitrary axis exchange control, the coordinate position of the axis after the axis exchange is acquired.

(8) High-speed machining mode

If the position information acquisition variable is commanded during the high-speed machining mode, the high-speed machining mode is temporarily canceled, and the position information at the completion of the block immediately before commanding the position information acquisition variable is acquired.

(9) Inclined axis control

If the position information acquisition variable of the machine coordinate system is commanded during the inclined axis control, the coordinate position on the machine coordinate system (actual axis position) is acquired.
If the position information acquisition variable of the workpiece coordinate system is commanded during the inclined axis control, the coordinate position on the orthogonal coordinate system is acquired.

(10) Diameter/Radius designation switch

The coordinate position acquired by the position information acquisition variable does not change depending on the selection of radius designation/diameter designation by the diameter/radius designation switch (G10.9).
In the M system, it is handled as a radius value, and in the L system, it follows the setting of "#1019 dia".

(11) PLC axis control

The position information acquisition variable cannot acquire the coordinate position of the PLC axis.
If the position information acquisition variable is commanded for the PLC axis, the coordinate position is not acquired, and the program error (P241) occurs.

23.12 System Variables (Position Information Common to Part System)

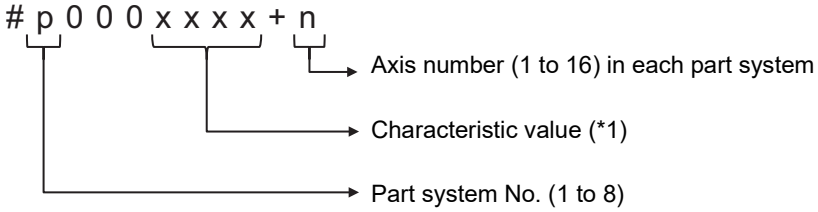
Detailed description

The variables to obtain the position information common to part systems are used to read the machine coordinate position, workpiece coordinate, and tool compensation amount of the specified axis from any part system.

System variable (*1)	Details
#p0005021	Reads the coordinate position of the machine coordinate system for the specified axis.
#p0005041	Reads the coordinate position of the workpiece coordinate system for the specified axis.
#p0005081	Reads the tool compensation amount for the specified axis.

(*1) In the #p000xxxx format, "p" indicates the part system No.

Each variable number is determined as follows.



(*1) System variable number of the variable to acquire the position information by part system. Refer to "23.11 System Variables (Position Information by Part System)".

The part system number and axis number follow the basic axis configuration (the configuration of axes that can be controlled by the part system when the power is turned ON).
This is used by adding the axis number to be acquired to the characteristic value.
The most significant digit corresponds to the part system number.

[Machine coordinate position] (Characteristic value: 5020)

Axis No.	Part system							
	Part system 1	Part system 2	Part system 3	Part system 4	Part system 5	Part system 6	Part system 7	Part system 8
1	#10005021	#20005021	#30005021	#40005021	#50005021	#60005021	#70005021	#80005021
2	#10005022	#20005022	#30005022	#40005022	#50005022	#60005022	#70005022	#80005022
3	#10005023	#20005023	#30005023	#40005023	#50005023	#60005023	#70005023	#80005023
:	:	:	:	:	:	:	:	:
16	#10005036	#20005036	#30005036	#40005036	#50005036	#60005036	#70005036	#80005036

[Workpiece coordinate position] (Characteristic value: 5040)

Axis No.	Part system							
	Part system 1	Part system 2	Part system 3	Part system 4	Part system 5	Part system 6	Part system 7	Part system 8
1	#10005041	#20005041	#30005041	#40005041	#50005041	#60005041	#70005041	#80005041
2	#10005042	#20005042	#30005042	#40005042	#50005042	#60005042	#70005042	#80005042
3	#10005043	#20005043	#30005043	#40005043	#50005043	#60005043	#70005043	#80005043
:	:	:	:	:	:	:	:	:
16	#10005056	#20005056	#30005056	#40005056	#50005056	#60005056	#70005056	#80005056

[Tool position offset amount] (Characteristic value: 5080)

Axis No.	Part system							
	Part system 1	Part system 2	Part system 3	Part system 4	Part system 5	Part system 6	Part system 7	Part system 8
1	#10005081	#20005081	#30005081	#40005081	#50005081	#60005081	#70005081	#80005081
2	#10005082	#20005082	#30005082	#40005082	#50005082	#60005082	#70005082	#80005082
3	#10005083	#20005083	#30005083	#40005083	#50005083	#60005083	#70005083	#80005083
:	:	:	:	:	:	:	:	:
16	#10005096	#20005096	#30005096	#40005096	#50005096	#60005096	#70005096	#80005096



Relationship with other functions

(1) Arbitrary axis exchange control

The axis number that specifies the axis with the variables to obtain the position information common to part systems is the axis number of the basic axis configuration (*1). Therefore, even if the axis configuration in the part system changes due to arbitrary axis exchange control, the axis number specified before and after the axis exchange command is issued remains unchanged.

(*1) Refers to the configuration of axes that can be controlled in the part system when the power is turned ON.

For other details, refer to "23.11 System Variables (Position Information by Part System)".



Precautions

(1) The variables to obtain the position information common to part systems read each data according to the basic axis arrangement (the arrangement of axes that can be controlled in the part system when the power is turned on).

(2) If an undefined axis is designated, the program error (P241) occurs.

(3) The coordinate position is read at the commanded timing.
While the axis is in motion, the program reads the machine coordinate position or workpiece coordinate position in the process of being moved.
Even in the same machining program, the coordinate position read for each cycle may vary depending on conditions such as override and the system load state.

(4) No data can be written to the variables to obtain the position information common to part systems.
When a write command is issued, no operation is executed and the program proceeds to the next block.

23.13 System Variables (Alarm)



Detailed description

The NC unit can be forcibly set to the alarm state by using variable No. #3000.

#3000 = n (ALARM.MESSAGE) ;	
n	Alarm No. (1 to 9999)
ALARM.MESSAGE	Alarm message

Any alarm number from 1 to 9999 can be specified.
The alarm message must be written in not more than 31 half-width characters or 15 full-width characters.
NC alarm 3 signal (program error) is output.
Program execution stops, then "P277 Macro alarm message" and the specified alarm (message and No.) are displayed alternately in the message display area. The above stated items also appears in the [NC message] and [Operator message] fields on the diagnosis screen.

Example of program (alarm when #1 = 0)

:	
IF[#1 NE 0]GOTO 100 ;	
#3000=70 (CALL #PROGRAMMER #TEL #530);	Program error (P277) (*1)
N100	
:	

(*1) "P277 Macro alarm message" and "CALL#PROGRAMMER#TEL#530 70" (specified message and No.) are displayed alternately.

Note

(1) If "0" 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. When the alarm No. is omitted, the program error (P33) occurs.

(3) When 32 or more half-width characters are specified for the alarm message, characters after the 32nd character will not display.
When the commanded string has 16 or more full-width characters, the 16th and subsequent characters are not displayed.

(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).

(5) Do not use half-width kana characters for the alarm message. A part of half-width kana character string will be missing (for Japanese).

(6) To create a machining program using a general text editor, set the character code according to the language used. For details, refer to "3.2 File Format".

23.14 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 = n (MESSAGE) ;	
n	Stop No. (1 to 9999)
MESSAGE	Message

The message should be written in not more than 31 half-width characters or 15 full-width characters and should be enclosed by round parentheses.

Note

(1) If "0" or any number greater than "9999" is specified for the stop No., the command is invalid and the program does not stop.

(2) Specify an alarm message by enclosing it in round parenthesis immediately after the stop No. If there is any character string between the number and the message enclosed in round parenthesis, the program error (P33) occurs. When the stop No. is omitted, the program error (P33) occurs as well.

(3) When only the stop No. is specified and no message is specified, nothing is displayed. The program stops without the stop number being displayed.

(4) When 32 or more half-width characters are specified for the message, the 32nd and subsequent characters are not displayed.
When 16 or more full-width characters are specified, the 16th and subsequent characters are not displayed.

(5) Spaces in message strings are ignored and are not displayed on the screen. To separate strings, insert a character such as "." (period).

(6) Do not use half-width kana characters for the message. A part of half-width kana character string will be missing (for Japanese).

(7) To create a machining program using a general text editor, set the character code according to the language used. For details, refer to "3.2 File Format".

Switching #3006 operation during inhibition of single block stop

The operation when the variable No. #3006 is commanded during inhibition of single block stop by system variable (the variable No. "#3003/bit0" is set to "1") can be switched by the setting of the parameter "#11482 ValBlkStopSelect" (System variable #3006 block stop switch).

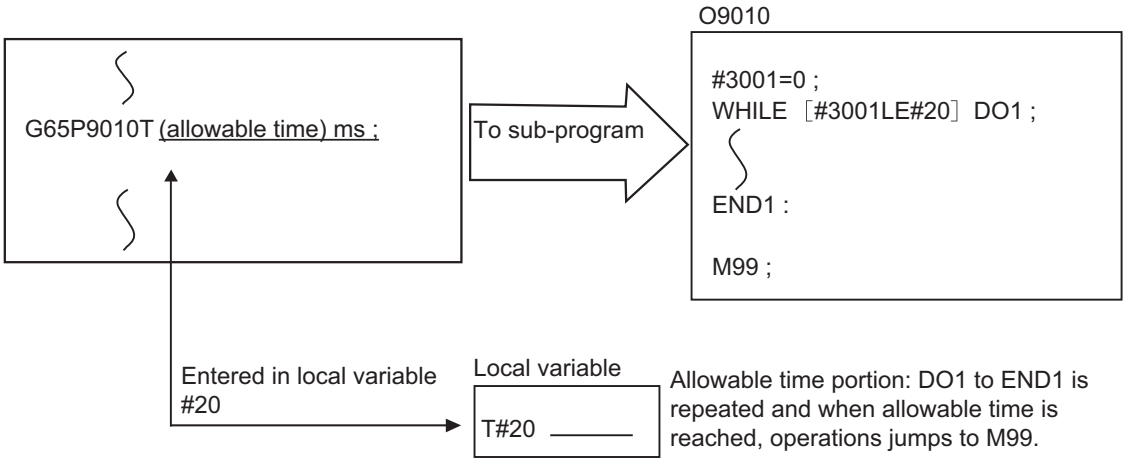
System variable #3003/bit0	Parameter #11482	System variable #3006 operation
0	0	Block stop is executed with the system variable #3006 and the message is displayed.
0	1	
1	0	Block stop is not executed with the system variable #3006. (The message does not remain.)
1	1	Block stop is executed with the system variable #3006 and the message is displayed.

23.15 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	1 ms	Same as when power is switched off	Substitute values to variables	At all times while power is ON
Automatic start	3002	1 ms or 1 h (1 hour)			In-automatic start



(1) The cumulative time is reset to "0" at approximately 2.44×10^{11}ms (approximately 7.7 years).

(2) When a negative value or a value exceeding 244335917226 ms (approximately 7.7 years) (*1) is specified for the cumulative time, the program error (P35) occurs.

(*1) When the unit of the system variable #3002 is "1 h", the threshold is 67871.08811851 hours.
The unit of the system variable #3002 depends on the setting of the parameter "#1273 ext09/bit1" (MTB specifications).

(3) When the parameter "#1237 set09/bit7" is "1", more detailed cumulative time can be measured with higher reading accuracy of the system variables #3001 and #3002.

23.16 System Variables (Time Read Variables)



Detailed description

The following operations can be carried out using the system variable extension for the user macro time.

(1) By adding time information system variable #3011 and #3012, the current date (#3011) and current time (#3012) can be read and written.

(2) By adding parameter #1273/bit1, the unit (millisecond unit/hour unit) of the system variable "#3002" (cumulative time during automatic start) can be changed.

Variable No.	Details
#3001	The cumulative time during power ON can be read and the value can be substituted. The unit is millisecond.
#3002	The cumulative time during automatic start can be read and the value can be substituted. The unit can be changed between millisecond and hour with parameter #1273/bit1.
#3011	The current date can be read and written. YYYY/MM/DD is read as a YYYYMMDD value. If a value "YYYYMMDD" is written, it is set to YY/MM/DD (the year is indicated by the last two digits). Command range for year/ month/day setting: Year (YYYY): 2000 to 2099 Month (MM): 1 to 12 Day (DD): 1 to maximum number of days in one month
#3012	The current time can be read and written. HH/MM/SS is read as a value "HHMMSS". When a value "HHMMSS" is written in, it will be set as HH/MM/SS. Command range for time setting: Hour (HH): 0 to 23 (24-hour system) Minute (MM): 0 to 59 Second (SS): 0 to 59

(3) The cumulative time is reset to "0" at approximately 2.44×10^{11}ms (approximately 7.7 years).

(4) If a negative value or a value exceeding 244335917226 milliseconds (67871.08811851 hours for #3002 time designation) is set for the cumulative time, a program error (P35) will occur.

(5) If a value exceeding the command range is set for the date or time, a program error (P35) will occur.

(6) Always set the month/date/hour/minute/second as a two-digit value when setting the date and time.
If the value only has one digit, always add 0.
(February 14, 2001 => #3011= 20010214 ;, etc.)

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.

23.17 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.

23.18 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

23.19 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.

23.20 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

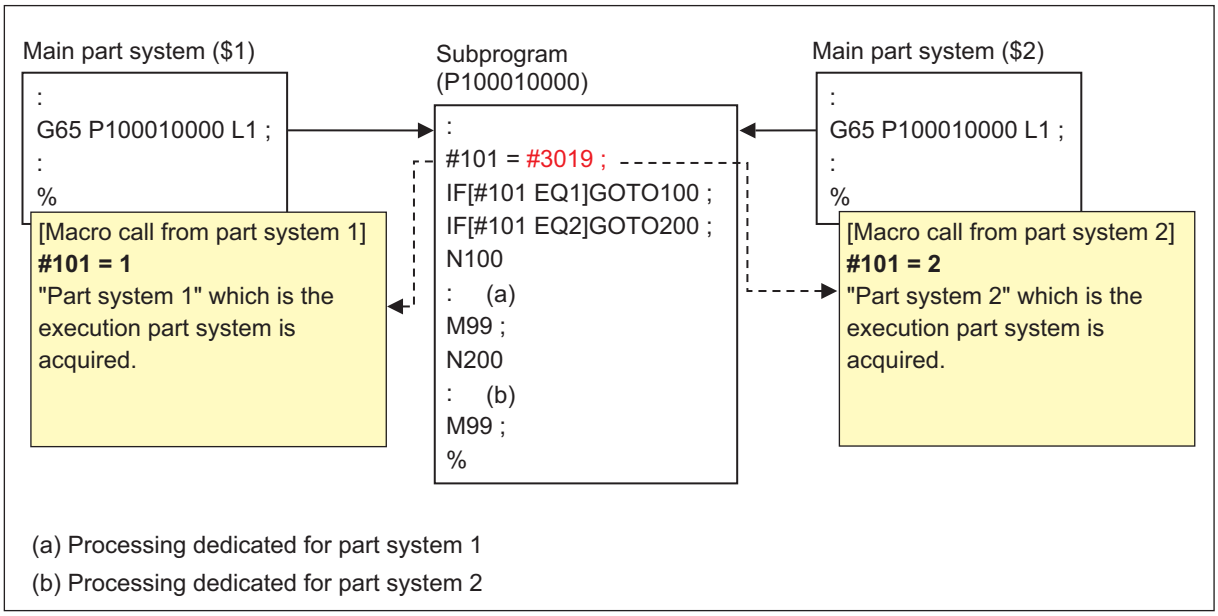
Bit	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
nth axis									8	7	6	5	4	3	2	1

23.21 System Variables (Acquiring the Program Execution Part System No.)

Detailed description

The number of the part system to execute a program can be acquired with the system variable #3019.
For example, if this variable (#3019) is commanded in part system 1, "1" is acquired, and if it is commanded in part system 2, "2" is acquired. If the system variable #3019 is commanded in a subprogram, the part system number of the main part system of the calling source is acquired.
If a machining program is called in another part system (sub part system) that is not in operation by the sub part system control and the system variable #3019 is commanded, the part system number of the sub part system that is the part system to execute the machining program is acquired.
The system variable #3019 is a read-only variable.
If the system variable #3019 is written, the program error (P243) occurs.

(Example 1) Example of combined operation of macro call (G65/G66/G66.1) and #3019



(a) Processing dedicated for part system 1
(b) Processing dedicated for part system 2

23.22 System Variables (Display Axis Name Selection)



Detailed description

This system variable is used to switch the axis name to be displayed on the counter according to change of the machine axis configuration.
The display axis name is switched by setting the system variable #3020 to a value corresponding to the axis name parameter.
The parameter applied to axis name display can be checked by reading the variable value.

Perform axis name switch and reading for each part system.
Only NC axis is the target of axis name switch. (PLC axis is not the target.)

Note

•This variable (#3020) can be used only in the machine tool builder macro program (O100010000 to O199999998).

Variable No.	Details		Data range	Attribute
	Setting value / read value	Corresponding parameter		
#3020	0	#1022 axname2	0 to 5	R/W
	1	#1611 Ax_Dsp_Name[1]		
	2	#1613 Ax_Dsp_Name[2]		
	3	#1615 Ax_Dsp_Name[3]		
	4	#1617 Ax_Dsp_Name[4]		
	5	#1619 Ax_Dsp_Name[5]		

A program error occurs in the following cases.

•The variable #3020 is used in a program other than the machine tool builder macro program. (P241)
•The parameter "#1753/bit5" is "0". (P241)
•The variable #3020 is used in a sub part system of sub part system control II. (P241)
•A value out of the range is written. (P35)
•A parameter where no axis is set is specified. (P35)

The name axis switch operation at power ON, NC reset and emergency stop depends on the MTB specifications. (Parameters "#1022 axname2", "#1611 Ax_Dsp_Name[1]" to "#1619 Ax_Dsp_Name[5]", "#1753 cfgPR03/bit5, bit6, bit7", and "#12146 I_Ax_Dsp_name")

23.23 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)

23.24 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

23.25 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)

23.26 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).

23.27 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.

23.28 System Variables (Macro Interface Input (PLC -> NC))

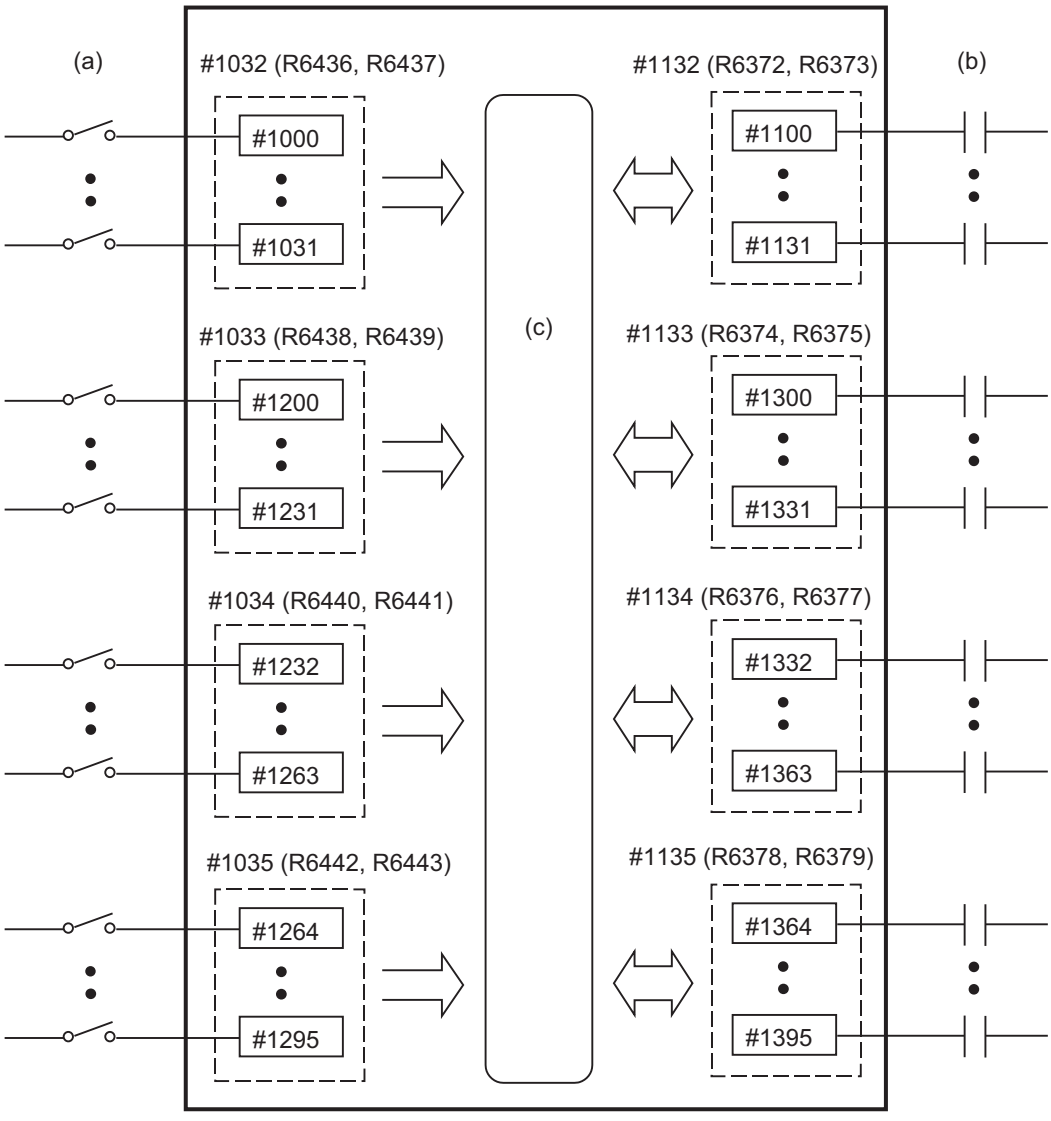
Function and purpose

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 "23.29 System Variables (Macro Interface Output (NC -> PLC))".)

Example of 1st part system



(a) Input signal (b) Output signal (c) Macro instructions



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	R6448/ bit0	R6456/ bit0	R6464/ 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	R6449/ bit0	R6457/ bit0	R6465/ 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

23.29 System Variables (Macro Interface Output (NC -> PLC))

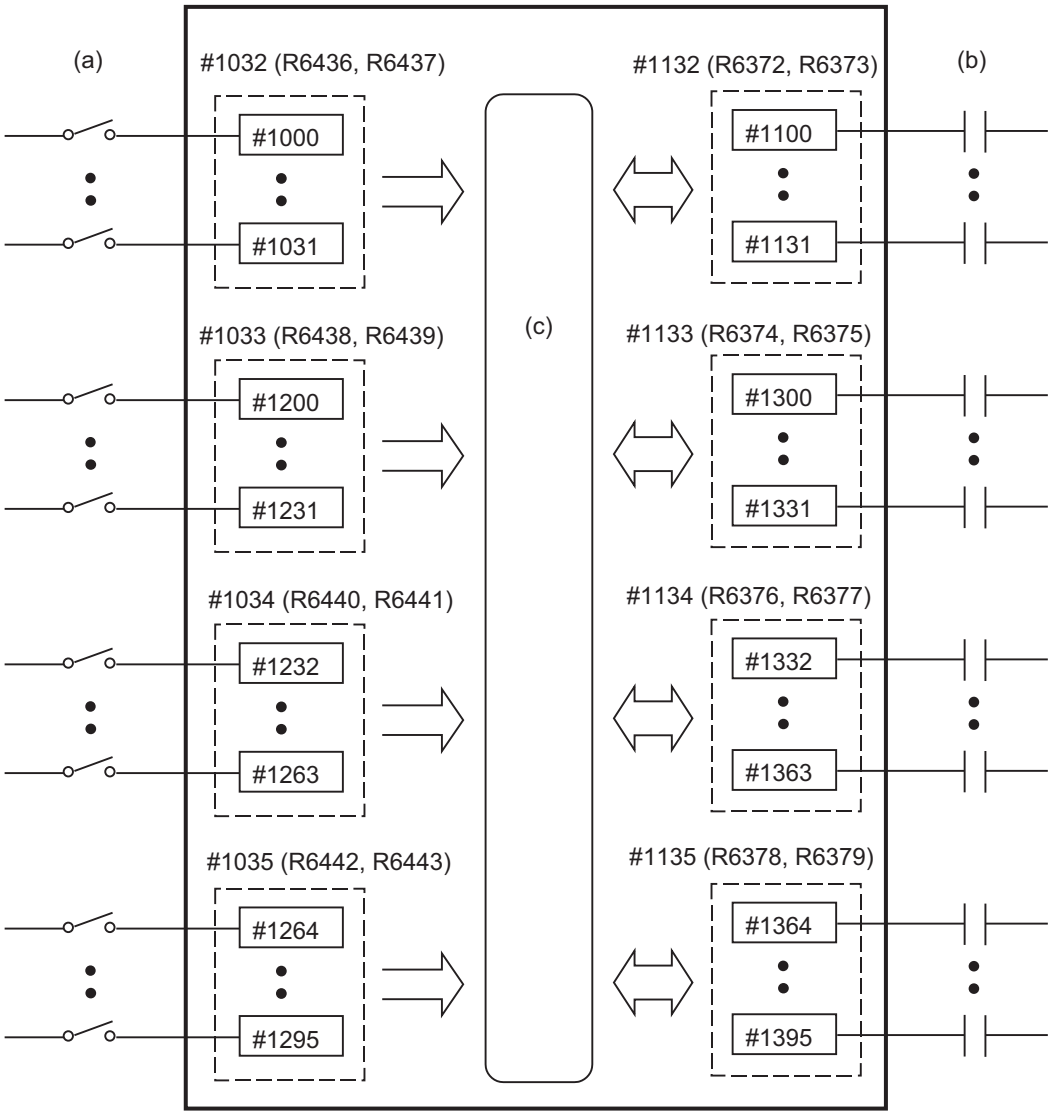
Function and purpose

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 "23.28 System Variables (Macro Interface Input (PLC -> NC))".)

Example of 1st part system



(a) Input signal (b) Output signal (c) Macro instructions



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

23.30 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.
Also, they are linked with the data of ZR devices (ZR150000 to ZR151499, ZR152000 to ZR153499, ZR154000 to ZR155499).

Variable No.	R device		ZR device
#50000	R8300, R8301	User backup area (1500 points)	ZR150000, ZR150001
#50001	R8302, R8303		ZR150002, ZR150003
:	:		:
#50749	R9798, R9799		ZR151498, ZR151499

Variable No.	R device		ZR device
#51000	R18300, R18301	User backup area (1500 points)	ZR152000, ZR152001
#51001	R18302, R18303		ZR152002, ZR152003
:	:		:
#51749	R19798, R19799		ZR153498, ZR153499

Variable No.	R device		ZR device
#52000	R28300, R28301	User backup area (1500 points)	ZR154000, ZR154001
#52001	R28302, R28303		ZR154002, ZR154003
:	:		:
#52749	R29798, R29799		ZR155498, ZR155499



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.



Device	Value	#50000
R8301	0x0001	0x1e240 (Hex) = 123456 (Dec)
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.".

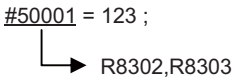
(2) When decimal point valid is selected:

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" (inupt 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.



(1) When decimal point invalid is selected:

Regardless of the setting of the parameter "#1003 iunit" (inupt setting unit) and "#1041 I_inch" (initial inch), substituted value will be set in the R device.

#50001	Device	Value
123 (Dec) = 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 (Dec) = 0x1e078 (Hex)	1230000 (Dec) = 0x12c4b0 (Hex)	12300000 (Dec) = 0xbbaee0 (Hex)	123000000 (Dec) = 0x754d4c0 (Hex)
Device	R8303	0x0001	0x0012	0x00bb	0x0754
	R8302	0xe078	0xc4b0	0xaee0	0xd4c0

#1041 I_inch		1: Inch			
#1003 iunit		B	C	D	E
#50001		1230000 (Dec) = 0x12c4b0 (Hex)	12300000 (Dec) = 0xbbaee0 (Hex)	123000000 (Dec) = 0x754d4c0 (Hex)	1230000000 (Dec) = 0x49504f80 (Hex)
Device	R8303	0x0012	0x00bb	0x0754	0x4950
	R8302	0xc4b0	0xaee0	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 (Dec) = 0x1e240 (Hex)	1234560 (Dec) = 0x12d680 (Hex)	12345600 (Dec) = 0xbc6100 (Hex)	123456000 (Dec) = 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 (Dec) = 0x12d680 (Hex)	12345600 (Dec) = 0xbc6100 (Hex)	123456000 (Dec) = 0x75bca00 (Hex)	1234560000 (Dec) = 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 (Dec) = 0x1e241 (Hex)	1234568 (Dec) = 0x12d688 (Hex)	12345679 (Dec) = 0xbc614f (Hex)	123456790 (Dec) = 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 (Dec) = 0x12d688 (Hex)	12345679 (Dec) = 0xbc614f (Hex)	123456790 (Dec) = 0x75bcd16 (Hex)	1234567899 (Dec) = 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 (Dec) = 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" (inupt setting unit) and "#1041 I_inch" (initial inch).

#1041 I_inch		**0: Metric**			
#1003 iunit		**B**	**C**	**D**	**E**
#50003		1000 (Dec) = 0x3e8 (Hex)	10000 (Dec) = 0x2710 (Hex)	100000 (Dec) = 0x186a0 (Hex)	1000000 (Dec) = 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 (Dec) = 0x2710 (Hex)	100000 (Dec) = 0x186a0 (Hex)	1000000 (Dec) = 0xf4240 (Hex)	10000000 (Dec) = 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" (inupt 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 (Dec) = 0xffffff85 (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" (inupt setting unit) and "#1041 I_inch" (initial inch).

#1041 I_inch		**0: Metric**			
#1003 iunit		**B**	**C**	**D**	**E**
#50004		-123456 (Dec) = 0xfffe1dc0 (Hex)	-1234560 (Dec) = 0xffed2980 (Hex)	-12345600 (Dec) = 0xff439f00 (Hex)	-123456000 (Dec) = 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 (Dec) = 0xffed2980 (Hex)	-12345600 (Dec) = 0xff439f00 (Hex)	-123456000 (Dec) = 0xf8a43600 (Hex)	-1234560000 (Dec) = 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" (inupt 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" (inupt 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.

23.31 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) 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

Device designation value	Device	Unit	Device No.
0	M	Bit	M0 - M61439
1	D	Word	D0 - D4095
2	C	Bit	C0 - C511
4	X (*1)	Bit	X0 - X1FFF
5	Y (*1)	Bit	Y0 - Y1FFF
6	R	Word	R0 - R32767
7	T	Bit	T0 - T2047
9	SM	Bit	SM0 - SM2047

Device designation value	Device	Unit	Device No.
10	F	Bit	F0 - F2047
13	L	Bit	L0 - L1023
18	V	Bit	V0 - V511
19	ST	Bit	ST0 - ST127
20	SD	Word	SD0 - SD2047
21	SB (*1)	Bit	SB0 - SB3FF
22	SW (*1)	Word	SW0 - SW3FF
23	B (*1)	Bit	B0 - BDFFF
24	W (*1)	Word	W0 - 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 #101 = 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, #100 = 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 = 0xFFFE 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.

23.32 System Variables (Interfering Object Selection)



Detailed description

Select the 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).
The maximum numbers of definitions of solids and interfering objects, the maximum number of solids that can be used to define an interfering object, the number of interfering objects, etc. differ depending on the model.

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)	0 to 65535 (decimal)
			bit0: Disable 1st interfering object : bitF: Disable 16th interfering object	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 to 3
			0, 1: Alarm area 2: Warning area 3: Solid setting invalid	0 to 3
#40004	R20308 (L) R20309 (H)	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)
#40005	R20310 (L) R20311 (H)	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)
#40006	R20312 (L) R20313 (H)	1st interfering model coordinate system K axis offset 1	Set the interfering model coordinate system offset with a radius value. (K axis direction) (*1)	
:	:			
#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) R20429 (H)	16th interfering model coordinate system I axis offset 1	Same as above	Same as above

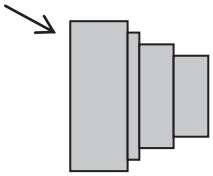
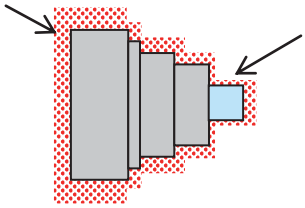
System variable	R register	Item	Details	Setting range (unit) Upper: System variable Lower: R register
#40080	R20430 (L) R20431 (H)	16th interfering model coordinate system J axis offset 1	Same as above	Same as above
#40081	R20432 (L) R20433 (H)	16th interfering model coordinate system K axis offset 1	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.

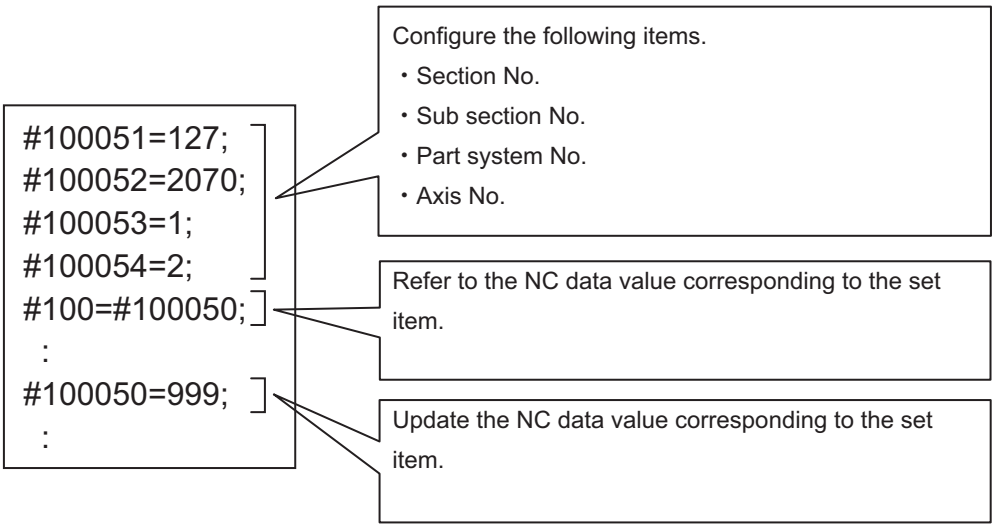
Back side of spindle part (without a workpiece)	Back side of spindle part (with a workpiece)
1st interfering object (back side of spindle part) 	1st interfering object (back side of spindle part) 2nd interfering object (workpiece part)  By specifying the interference III disabled between the back side of spindle part (1st interfering object) and workpiece part (2nd interfering object), these 2 parts are treated as one interfering object.

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.

23.33 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 "500" in the range of 0 to 999.
#100054=3 ;	Axis No. designation	3rd axis
#100050=10 ;	NC data update	Designate "10" 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.

(6) When data other than numerical data (such as character string data) is read to the variables, the program error (P35) may occur.

(7) When the system variable #100051 is set to any of the section Nos. 53 to 56, 108 and 109 (*1) with a graphic check (background), even if the sub-section No. set in #100052 is invalid, the program error (P35) does not occur at reading.

(*1) The section Nos. 53 to 56 are respectively bit, char, word and long data of PLC device.
The section No. 108 is PLC timer data and 109 is PLC counter data.

23.34 System Variables (Extended Common Variables)



Detailed description

Load file number (Extended common variable III)

No.	Data type or use	Data range
#9000	Extended common variable III load file No.	0 to 1000

(1) Copy (load) the extended common variable data file to the extended common variable III in the work memory.
#9000 = 1 to 1000 (*1)
(*1) The setting value of parameter #11829 is the maximum.

(2) Copy (store) the extended common variable III in the work memory to the extended common variable data file.
#9000 = 0

(3) The system variable #9000 is a pre-read prohibition command.

For the usage method, refer to "14.5.2.2 Extended Common Variable III".

Extended common variable/system variable switching

No.	Data type or use	Data range
#9001	Extended common variable/system variable switching	0/1

(1) Switches whether to use #10000 to #89999 as system variables or as extended common variables I or II in the user program.
If "0" is set, #10000 to #89999 are used as system variables.
If "1" is set, #10000 to #89999 are used as extended common variables I or II.

(2) The system variable #9001 is set to "0" when the power is turned ON or reset.

(3) If a value outside the setting range is set in system variable #9001, the program error (P35) occurs. If <null> is set, it is handled as "0".

(4) The system variable #9001 is a command specific for part system.

(5) The system variable #9001 is a pre-read prohibition command.

For the usage method, refer to "14.5.2.1 Extended Common Variable I/II".

Extended common variable/machine tool builder macro variable switching

No.	Data type or use	Data range
#9002	Extended common variable/machine tool builder macro variable switching	0/1

(1) Switches whether to use #10000 to #89999 as system variables/machine tool builder macro variables or as extended common variables I or II in the machine tool builder macro.
If "0" is set, #10000 to #89999 are used as system variables/machine tool builder macro variables.
If "1" is set, #10000 to #89999 are used as extended common variables I or II.

(2) The system variable #9002 is set to "0" when the power is turned ON or reset.

(3) If a value outside the setting range is set in system variable #9002, the program error (P35) occurs. If <null> is set, it is handled as "0".

(4) The system variable #9002 is a command specific for part system.

(5) The system variable #9002 is a pre-read prohibition command.

For the usage method, refer to "14.5.2.1 Extended Common Variable I/II".

23.35 System Variables (Axis Exchange Information Acquisition)

Detailed description

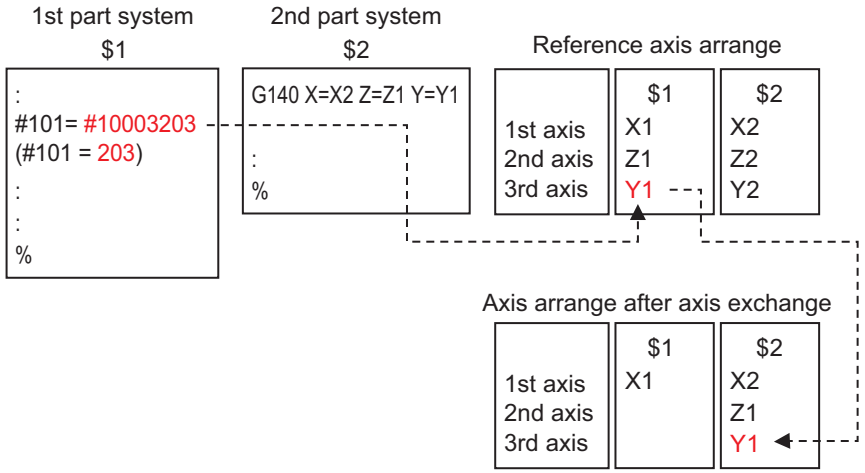
Term

Term	Meaning
Axis arrangement	Refers to the arrangement of axes that can be controlled in the part system
Reference axis arrange	Refers to the arrangement of axes when the power is turned ON.

Details of function

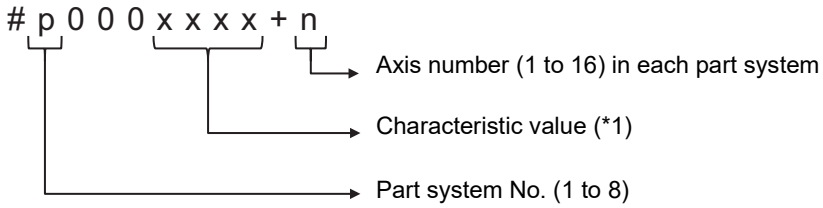
Axis exchange information acquisition variables (#p0003201 to #p0003216) are used to know the axis of reference axis arrangement turns to what number axis in which part system after axis exchange. The part system No. and the axis No. in the part system after axis exchange can be acquired. "p" indicates the part system number (1 to 8).

In the figure below, information of Y1 axis (3rd axis in 1st part system) is acquired by using the variable #10003203. The acquired value "203" indicates that the Y1 axis turns to the 3rd axis in the 2nd part system after the axis exchange.



[Variable No.]

The axis exchange information acquisition variables are configured as shown below.

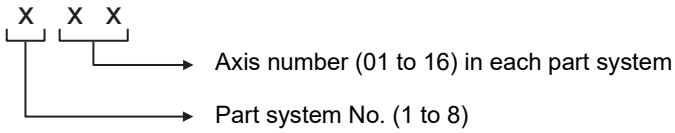


(*1) The fixed value for axis exchange information acquisition variables is "3200".

Axis No.	Part system							
	$1	$2	$3	$4	$5	$6	$7	$8
1	#10003201	#20003201	#30003201	#40003201	#50003201	#60003201	#70003201	#80003201
2	#10003202	#20003202	#30003202	#40003202	#50003202	#60003202	#70003202	#80003202
3	#10003203	#20003203	#30003203	#40003203	#50003203	#60003203	#70003203	#80003203
:	:	:	:	:	:	:	:	:
16	#10003216	#20003216	#30003216	#40003216	#50003216	#60003216	#70003216	#80003216

[Acquired value]

3-digit numbers, "0" or "-1" can be acquired.



The combination of axis type and variable value is shown below.

Acquired value	Meaning of value
3-digit number	It indicates that the axis of reference axis arrangement is controlled as what number of axis in which part system after the axis exchange. (Axis where the part system has the control right)
0	An attempt is made to acquire information of uncontrol axis. (Axis where no part system has the control right)
-1	An attempt is made to acquire an axis which does not exist in the reference axis arrangement.

The axis exchange information acquisition variables are read-only variables.
If the axis exchange information acquisition variables are written, the program error (P243) occurs.



Operation example

[Reference axis arrange]

	1st axis	2nd axis	3rd axis
1st part system ($1)	X1	Z1	Y1
2nd part system ($2)	X2	Z2	C2

	$1	Variable value
	G140 X=X1 Z=Z1 Y=Y1;	
(1)	**#101 = #10003202;**	102
(2)	**#102 = #10003204;**	-1
	G00 X10.;	
	:	
	G140 X=X2 Z=Z2;	
(4)	**#104 = #10003202;**	0
(5)	**#105 = #10003204;**	-1
	G00 Z25.;	
	:	
	G140 X=X1 Z=Z1 Y=Y1 C=C2;	
	G00 Z10.;	
	:	

	$2	Variable value
	G140 X=X2 Z=Z2 C=C2;	
(3)	**#103 = #20003202;**	202
	G00 X20.;	
	:	
(6)	**#106 = #20003202;**	102

Control axes $1 X	Z	Y	C	$2 X	Z	C	Uncontrol axes
X1	Z1	Y1	-	X2	Z2	C2	-
X2	Z2	-	-	C2	-	-	X1 Y1 Z1
X1	Z1	Y1	C2	-	-	-	X2 Z2

Part system		Reference axis arrange	Axis arrangement at acquisition	Common variable (#101 or later) value	System variable value
$1	(1)	$1 2nd axis	$1 2nd axis	#101 = 102	#10003202 = 102
	(2)	$1 4th axis	(Non-existing axis)	#102 = -1	#10003204 = -1
	(4)	$1 2nd axis	(Uncontrol axis)	#104 = 0	#10003202 = 0
	(5)	$1 4th axis	(Non-existing axis)	#105 = -1	#10003204 = -1
$2	(3)	$2 2nd axis	$2 2nd axis	#103 = 202	#20003202 = 202
	(6)	$2 2nd axis	$1 2nd axis	#106 = 102	#20003202 = 102



Precautions

(1) If this system variable is used before the axis exchange is completed, the data might not be acquired correctly.

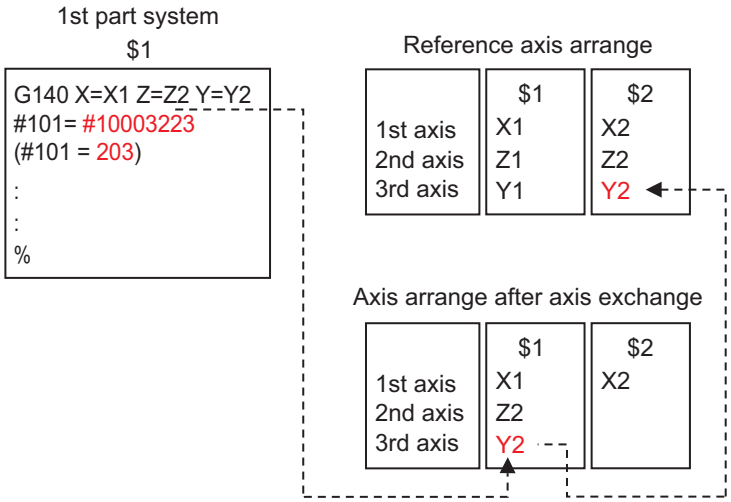
(2) The pre-reading of system variables #p0003201 to #p0003216 is prohibited.

23.36 System Variables (Reference Axis Arrangement Information Acquisition)

Details of function

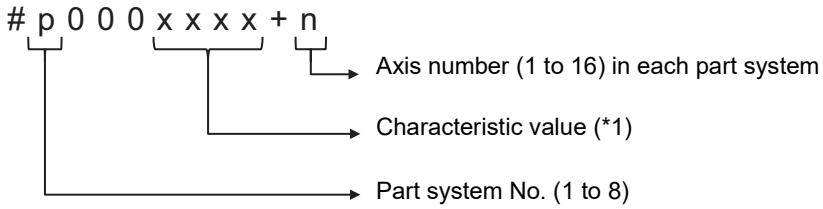
Reference axis arrangement information acquisition variables (#p0003221 to #p0003236) are used to know in which part system and what axis arrangement the current axis was before the axis exchange. The part system No. and the axis No. in the part system in the reference axis arrangement can be acquired. "p" indicates the part system number (1 to 8).

For terms, etc., also refer to "23.35 System Variables (Axis Exchange Information Acquisition)".
In the figure below, where the Y1 axis (3rd axis in 1st part system) was in the reference axis arrangement is acquired by using the variable #10003223.
The acquired value "203" indicates that the Y1 axis was the 3rd axis in the 2nd part system before the axis exchange.



[Variable No.]

The reference axis arrangement information acquisition variables are configured as shown below.

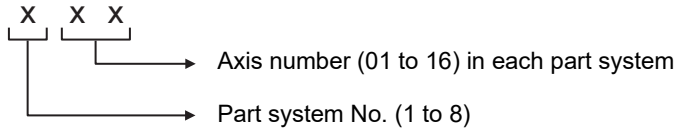


(*1) The fixed value for reference axis arrangement information acquisition variables is "3220".

Axis No.	Part system							
	$1	$2	$3	$4	$5	$6	$7	$8
1	#10003221	#20003221	#30003221	#40003221	#50003221	#60003221	#70003221	#80003221
2	#10003222	#20003222	#30003222	#40003222	#50003222	#60003222	#70003222	#80003222
3	#10003223	#20003223	#30003223	#40003223	#50003223	#60003223	#70003223	#80003223
:	:	:	:	:	:	:	:	:
16	#10003236	#20003236	#30003236	#40003236	#50003236	#60003236	#70003236	#80003236

[Acquired value]

3-digit numbers or "-1" can be acquired.



The combination of axis type and variable value is shown below.

Acquired value	Meaning of value
3-digit number	It indicates that the current axis (control axis) was controlled as what number axis in which part system in the reference axis arrangement. (Axis where the part system has the control right)
-1	An attempt is made to acquire an axis which does not exist in the reference axis arrangement.

The reference axis arrangement information acquisition variables are read-only variables.
If the reference axis arrangement information acquisition variables are written, the program error (P243) occurs.

Operation example

[Reference axis arrange]

	1st axis	2nd axis	3rd axis
1st part system ($1)	X1	Z1	Y1
2nd part system ($2)	X2	Z2	C2

$1		Variable value	$2		Variable value	Control axes $1 X	Z	Y	C	$2 X	Z	C	Uncontrol axes
	G140 X=X1 Z=Z1 Y=Y1;			G140 X=X2 Z=Z2 C=C2;									
(1)	**#101 = #10003222;**	102	(3)	**#103 = #20003222;**	202								
(2)	**#102 = #10003224;**	-1		G00 X20.;		X1	Z1	Y1	-	X2	Z2	C2	-
	G00 X10.;			:									
	:												
	G140 X=X2 Z=Z2;		(6)	**#106 = #20003222;**	-1								
(4)	**#104 = #10003222;**	202											
(5)	**#105 = #10003224;**	-1				X2	Z2	-	-	C2	-	-	X1 Y1 Z1
	G00 Z25.;												
	:												
	G140 X=X1 Z=Z1 Y=Y1 C=C2;												
	G00 Z10.;					X1	Z1	Y1	C2	-	-	-	X2 Z2
	:												

Part system		Current axis arrangement	Reference axis arrange	Common variable (#101 or later) value	System variable value
$1	(1)	$1 2nd axis	$1 2nd axis	#101 = 102	#10003222 = 102
	(2)	$1 4th axis	(Non-existing axis)	#102 = -1	#10003224 = -1
	(4)	$1 2nd axis	$2 2nd axis	#104 = 202	#10003222 = 202
	(5)	$1 4th axis	(Non-existing axis)	#105 = -1	#10003224 = -1
$2	(3)	$2 2nd axis	$2 2nd axis	#103 = 202	#20003222 = 202
	(6)	$2 2nd axis	(Uncontrol axis)	#106 = -1	#20003222 = -1

Precautions

(1) If this system variable is used before the axis exchange is completed, the data might not be acquired correctly.
(2) The pre-reading of system variables #p0003221 to #p0003236 is prohibited.

24

Appx.1: Fixed Cycles

[G81 (O100000810) 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 (O100000820) 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 (O100000830) 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 (O100000840) 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 (O100000850) 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 ;
```

[G86 (O100000860) 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 ;
```

[G87 (O100000870) Back boring]

```
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 ;
```

[G88 (O100000880) 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 ;
```

[G89 (O100000890) 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 ;
```

[G73 (O100000831) 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]

```
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 ;
```

[G84.5 (O745) Punchtap cycle (PT1.0)]

```
G.1 ;
IF[#30] GOTO9 ;
Z#2 G#6 H#7 ;
#2=##5 ;
#3003=#8OR1 ;
#3004=#9OR3 ;
G1 Z#3 ;
IF[[#4EQ0] OR [#4EQ#0]] GOTO1 ;
G4 P#4 ;
N1  G1 Z#29 ;
G1 Z#28 ;
#3900=1 ;
#27=#29+#28 ;
#27=#3+#27 ;
G1 Z-#27 ;
#3004=#9 ;
G4 P#4 ;
#3003=#8 ;
G0 Z-#2, I#23 ;
N9  M99 ;
```

[G84.6 (O746) Punchtap cycle (PT1.5)]

```
G.1 ;
IF[#30] GOTO9 ;
Z#2 G#6 H#7 ;
#2=##5 ;
#3003=#8OR1 ;
#3004=#9OR3 ;
G1 Z#3 ;
IF[[#4EQ0] OR [#4EQ#0]] GOTO1 ;
G4 P#4 ;
N1  G1 Z#29 ;
G1 Z#28 ;
#27=#29+#28-#26 ;
G1 Z-#27 ;
#3900=1 ;
#27=#3+#26 ;
G1 Z-#27 ;
#3004=#9 ;
G4 P#4 ;
#3003=#8 ;
G0 Z-#2, I#23 ;
N9  M99 ;
```

[G84.8 (O748) Punchtap cycle (PT2.0)]

```
G.1 ;
IF[#30] GOTO9 ;
Z#2 G#6 H#7 ;
#2=##5 ;
#3003=#8OR1 ;
#3004=#9OR3 ;
G1 Z#3 ;
IF[[#4EQ0] OR [#4EQ#0]] GOTO1 ;
G4 P#4 ;
N1  G1 Z#29 ;
G1 Z#28 ;
#27=#29+#28-#26 ;
G1 Z-#27 ;
G1 Z#27 ;
G1 Z-#28 ;
#3900=1 ;
#27=#3+#29 ;
G1 Z-#27 ;
#3004=#9 ;
G4 P#4 ;
#3003=#8 ;
G0 Z-#2, I#23 ;
N9  M99 ;
```

[Precautions for MTB]

- The G.1 command is used to realize the fixed cycle operation. Therefore, do not edit the blocks before the G.1 command such as deleting or inserting another command before the G.1 command.
- The fixed cycle program uses modal information of group 01 dedicated for the fixed cycle program so that the modal of the main program that called the fixed cycle program is not affected. Therefore, there is no need to edit the fixed cycle program based on the modal when returning to the main program. In the fixed cycle program, the modal of group 01 is set to G00 after the G.1 command is executed.

25

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 570 and before).

Refer to Programming Manual (Machining Center System) (2/2) for Chapter 15 and succeeding chapters (page 571 and later).

Symbols

!n (!m ...) L 732
/ 17
/n 19

Numerics

2nd, 3rd, 4th Reference Position Return 1181
3-dimensional Circular Interpolation 117
3-dimensional Coordinate Conversion 1125
3-dimensional Tool Radius Compensation 359
3-dimensional Tool Radius Compensation (Tool's Vertical-direction Compensation) 1055

A

Acceleration/Deceleration Mode Change in The Fixed Cycle for Drilling 453
Actual Examples of Using User Macros 552
Arbitrary Axis Exchange 749
Arc 466
Arc Intersection Point Calculation Switch 355
ASCII Code Macro 519
Automatic 1st Reference Position Return and Start Position Return 1176
Automatic Coordinate System Setting 1083
Automatic Corner Override 223, 230
Automatic Tool Length Measurement 1200
Axis Name Extension 656
Axis-based Unidirectional Positioning 79

B

Back Boring 414
Basic Machine Coordinate System Selection 1087
Basic Machine, Workpiece and Local Coordinate Systems 1081
Bolt Hole Cycle 464
Boring 412, 413, 418, 419

C

Changing of Compensation No. during Compensation Mode 337
Chip Removal 455
Circular Command in Tool Center Point Control (G43.4/G43.5) 986
Circular Cutting 88, 424
Circular Interpolation 55
Command Value Range Lists 1316
Common Variables 525
Compensation Data Input by Program (Tool Compensation Amount) 626
Compensation Data Input by Program (Turning Tool) ... 633
Compensation Data Input by Program (Workpiece Offset Amount) 629
Constant Lead Thread Cutting 66
Constant Surface Speed Control 255
Control Axis Synchronization between Part Systems 752
Control Commands 540
Coordinate Rotation by Program 1144
Coordinate Rotation Input by Parameter 1155
Coordinate System for Rotary Axis 1084
Coordinate System Setting 1090
Coordinate Systems and Coordinate Zero Point Symbols ... 3
Coordinate Words and Control Axes 2, 1080
Corner Chamfering Expansion/Corner Rounding Expansion 576, 582
Corner Chamfering I 572
Corner Chamfering I / Corner Rounding I 572
Corner Chamfering II 579
Corner Chamfering II / Corner Rounding II 579
Corner Rounding I 574
Corner Rounding II 581
Cutting Feed Constant-gradient Acceleration/ Deceleration 184
Cutting Feedrate 151
Cutting Load Control 707
Cutting Mode 233
Cutting Reduction Amount Specification Method 460
Cutting Speed Clamp with Acceleration Rate Judgment 853
Cylindrical Interpolation 80

D

Deceleration Check 197
Deceleration Check when Movement in The Opposite Direction Is Reversed 205
Decimal Point Command Valid/Invalid Address Changeover 48
Decimal Point Input 40
Deep Hole Drilling Cycle 391
Define by Selecting the Registered Machining Surface 1022
Detailed Description for Macro Call Instruction 513
Details of Inclined Surface Machining Operation 1033
Diameter Designation of Compensation Amount 351
Diameter/Radius designation switch 34
Drilling Cycle High-Speed Retract 449
Drilling, Counter Boring 390
Drilling, Spot Drilling 389
Dwell (Revolution-based designation) 239
Dwell (Time-based designation) 236

E

Enable Interfering Object Selection Data 1195
Exact Stop Check 192
Exact Stop Check Mode 196
Exponential Interpolation 99
Extended Common Variable I/II 526
Extended Common Variable III 528
External Output Commands 546

F

F1-Digit Feed 152
Face Cutting Cycle 477
Fairing 843
Feed Per Minute/Feed Per Revolution (Asynchronous Feed/Synchronous Feed) 154
Feedrate Designation and Effects on Control Axes 161
Figure Rotation 490
File Format 14
Fine Boring 426
Fixed Cycle for Turning Machining 469
Fixed Cycles 384, 1310

G

G Code Lists 21
G Cole Macro Call 509
G Command Mirror Image 594
G0.5 210
G0.5 P1 208
G00 50
G00 Feedrate Command (,F command) 147
G01 53
G01 A_ 584
G01 A_ , G02/G03 P_Q_H_ 589
G01 A_ , G02/G03 R_H_ 592
G01 X_ Y_ ,C 572
G01 X_ Y_ ,R_ 574
G01 X_/Y_ A_/,A_ 583
G01/G02/G03 X_ Y_ ,C_ 579
G01/G02/G03 X_ Y_ ,R_ 581
G02, G03 ....55, 61, 72
G02.1/G03.1 (Type 1), G02/G03 (Type 2) 113
G02.2/G03.2 131
G02.3,G03.3 99
G02.4, G03.4 117
G02/G03 P_Q_ /R_ 587
G04 ....236, 239
G05 P1, G05 P2 782
G05.1 Q1/Q0, G05 P10000/P0, G05 P20000/P0 825
G05.1 Q2/Q0 858
G06.2 123
G07 129
G07.1 80
G08.5 662
G08.6 716
G09 192
G10 I_ J_/K_ 1155
G10 L10/L11/L12/L13, G11 626
G10 L100, G11 635
G10 L110/L111, G11, G68.2, G69 638
G10 L12/L13, G11 633
G10 L14 1224
G10 L2/L20, G11 629
G10 L3, G11 642
G10 L30, G11 645
G10 L70, G11 624
G10.9 34
G114.1 885
G114.2, G113.1 901
G114.3/G113 916
G115 736
G116 739
G12,G13 88
G12.1, G13.1/G112, G113 90
G120.1, G121 878
G122 762
G125 752
G127 618
G130 166
G135 694
G136 679
G140, G141, G142 749
G16 106
G160 1220
G162.1 707
G162/G163 273
G17, G18, G19 64
G174 470
G175 473
G176 477
G180 652
G186 1195
G187 428
G20, G21 37
G22/G23 1190
G27 1187
G28, G29 1176
G30 1181
G30.1 - G30.6 1184
G31 1204
G31 Fn 1216
G31 P 1214
G31.n, G04 1212
G33 ....66, 70
G34 464
G35 465
G36 466
G37 1200
G37.1 467
G38, G39/G40/G41, G42 308
G40.1/G41.1/G42.1(G150/G151/G152) 598
G40/G41, G42 359
G40/G41.2, G42.2 1055
G41/G42 Commands and I, J, K Designation 328
G43, G44/G49 290
G43.1/G49 946
G43.4, G43.5/G49 953
G43.7/G49 934
G43.8/G43.9 990
G45 to G48 371
G50.1, G51.1 594
G50/G51 1172
G51.2/G50.2 or G251/G250 ....907, 912
G52 1092
G53 1087
G53.1/G53.6 1025
G54 to G59, G54.1 1099
G54.2 1113
G54.4 1066
G60 77
G61 196
G61.1,G08 791
G61.2 876
G61.4 866
G62 230
G63 232
G64 233
G65 501
G66 505
G66.1 507
G68.2, G68.3, G68.4/G69 1010
G68/G69 ....1125, 1144
G73 420
G74 422
G75 424
G76 426
G81 389
G82 390
G83 391
G84 396
G85 412
G86 413
G87 414
G88 418
G89 419
G90,G91 32
G92 ....262, 1090

G92.1 1108
G93 156
G94,G95 154
G96,G97 255
G98, G99 447
General Precautions for Tool Radius Compensation ... 336
Geometric I 584
Geometric IB 586
Geometric IB (Automatic calculation of contact point between line and circular arc) 592
Geometric IB (Automatic calculation of contact point of two circular arcs) 587
Geometric IB (Automatic calculation of intersection point between line and circular arc) 589
Grid 467

H

H Stand-alone Command 379
Handling the Macro Call Command 515
Helical Interpolation 72
High-accuracy Control 791
High-accuracy Spline Interpolation 876
High-speed High-accuracy Control 825
High-speed High-accuracy Control I, II, III 825
High-speed Machining Mode and High-accuracy Controll in Multi-part System 823
High-speed Machining Mode I, II 782
High-speed Mode Corner Deceleration 854
How to Define Feature Coordinate System Using Euler Angles 1012
How to Define Feature Coordinate System Using Projection Angles 1020
How to Define Feature Coordinate System Using Roll-Pitch-Yaw Angles 1014
How to Define Feature Coordinate System Using Three Points in a Plane 1016
How to Define Feature Coordinate System Using Tool Axis Direction 1023
How to Define Feature Coordinate System Using Two Vectors 1018
Hypothetical Axis Interpolation 129

I

Inch Thread Cutting 70
Inch/Metric Conversion 37
Inclined Surface Machining 1010
Inclined Surface Machining Multi-command 1044
Index Table Indexing 247
Indexing Increment 8
Initial High-accuracy Control 822
Initial Point and R Point Level Return 447
Inner Arc Override 231
Input Command Increment Tenfold 7
Input Setting Unit and Program Command Unit 6
Inputting the Tool Life Management Data by G10 L3 Command 642
Inputting the Tool Life Management Data by G10 L30 Command 645
Interactive Cycle Insertion 652
Interactive Macro 655
Interference Check 341
Interrupt during Corner Chamfering/ Interrupt during Corner Rounding 578, 582
Interrupts during Tool Radius Compensation 334
Inverse Time Feed 156
Involute Interpolation 131

L

Line at Angle 465
Linear Angle Command 583
Linear Interpolation 53
Loader Control Part System 778
Local Coordinate System Setting 1092
Local Variables (#1 to #33) 531
Longitudinal Cutting Cycle 470

M

M*** 742
M198 488
M96, M97 556
M98 I_J_K_ 490
M98, M99 482
Machine Zero Point and 2nd, 3rd, 4th Reference Position (Zero Point) 1082
Machining Condition Selection I 878
Macro Call Instructions 500
Macro Interruption 556
Manual Arbitrary Reverse Run Prohibition 618
Miscellaneous Command Macro Call (for M, S, T, B Code Macro Call) 511
Miscellaneous Functions (M8-digits) 244
Modal Call A (Movement Command Call) 505
Modal Call B (for Each Block) 507
Modal, Unmodal 21
Multi-step Skip Function 1 1212
Multi-step Skip Function 2 1214

N

Normal Line Control 598
Number of Tool Offset Sets Allocation to Part Systems 288
NURBS Interpolation 123

O

Operation Commands 535
Optional Block Skip 17
Optional Block Skip Addition 19
Orbit Machining 716
Other Commands and Operations during Tool Radius Compensation 318

P

Parameter Input by Program 624
Plane Selection 64
Polar Coordinate Command 106
Polar Coordinate Interpolation 90
POPEN, PCLOS, DPRNT 546
Position Command Methods 32
Positioning (Rapid Traverse) 50
Precautions 550
Precautions before Starting Machining 27
Precautions for Extended Common Variable 530
Precautions for Inclined Surface Machining 1052
Precautions for Inputting the Tool Life Management Data 649
Precautions for Using a Fixed Cycle 445
Precautions on High-speed High-accuracy Control 855
Pre-read Buffer 30
Program Format 10
Programmable Current Limitation 1224
Punchtap Cycle 432

R
R Specification Circular Interpolation 61
Rapid Traverse Block Overlap 208
Rapid Traverse Block Overlap for G00 210
Rapid Traverse Block Overlap for G28 221
Rapid Traverse Constant-gradient Acceleration/ Deceleration 170
Rapid Traverse Constant-gradient Multi-step Acceleration/Deceleration 176
Rapid Traverse Rate 146
Reference Position Check 1187
Relationship Between Inclined Surface Machining and Other Functions 1047
Reverse Tapping Cycle 422
R-Navi Data Input by Program 638
Rotary Axis Basic Position Selection 1038
S
Scaling 1172
Second Miscellaneous Functions (A8-digits, B8-digits or C8-digits) 246
Selection of Axis (Axes) for Feedrate Command 166
Setting of Workpiece Coordinates in Fixed Cycle Mode 448
Simple Macro Calls 501
Skip Function 1204
Small Diameter Deep Hole Drilling Cycle 393
Smooth Fairing 844
Special Fixed Cycle 463
Speed Change Skip 1216
Spindle Clamp Speed Setting 262
Spindle Functions 254
Spindle Position Control (Spindle/C Axis Control) 264
Spindle Position Control (Spindle/C Axis Control) under Spindle Synchronization Control 896
Spindle Speed Fluctuation Detection 273
Spindle Synchronization 884
Spindle Synchronization I 885
Spiral/Conical Interpolation 113
Spline Interpolation 858
Spline Interpolation 2 866
SSS Control 811
Start of Tool Radius Compensation and Z Axis Cut in Operation 339
Stepping Cycle 420
Stroke Check before Travel 1190
Stroke Check before Travel in Stored Stroke Limit Area 1192
Sub Part System Control I 762
Subprogram Call 482, 488
System Variables 534
System Variables (Acquiring the Program Execution Part System No.) 1264
System Variables (Alarm) 1257
System Variables (Axis Exchange Information Acquisition) 1302
System Variables (Coordinate Rotation Parameter) ...1266
System Variables (Cumulative Time) 1259
System Variables (Display Axis Name Selection) 1265
System Variables (Extended Common Variables) 1301
System Variables (Extended Workpiece Coordinate Offset) 1248
System Variables (G Command Modal) 1228
System Variables (Interfering Object Selection) 1295
System Variables (Machining Information) 1262
System Variables (Macro Interface Input (PLC -> NC)) 1274
System Variables (Macro Interface Output (NC -> PLC)) 1280
System Variables (Message Display and Stop) 1258
System Variables (Mirror Image) 1263
System Variables (Modal Information at Macro Interruption) 1230
System Variables (NC Data Reading/Writing with API Section and Sub-section Nos. Input/Output by Program) 1298
System Variables (Non-G Command Modal) 1229
System Variables (Normal Line Control Parameter) ... 1268
System Variables (Number of Workpiece Machining Times) 1263
System Variables (Parameter Reading) 1269
System Variables (PLC Data Reading) 1292
System Variables (Position Information by Part System) 1250
System Variables (Position Information Common to Part System) 1255
System Variables (R Device Access Variables) 1286
System Variables (Reference Axis Arrangement Information Acquisition) 1305
System Variables (Reverse Run Information) 1263
System Variables (Rotary Axis Configuration Parameter) 1267
System Variables (Time Read Variables) 1260
System Variables (Tool Compensation) 1240
System Variables (Tool Information) 1232
System Variables (Tool Life Management) 1241
System Variables (Workpiece Coordinate Offset) 1246
System Variables (Workpiece Installation Error Compensation Amount) 1273
System Variables (Workpiece Position Offset Amount for Rotary Axis) 1247
System Variables List 1226
T
Tapping Cycle 396
Tapping Mode 232
Text Engraving Cycle 694
Thread Cutting 66
Thread Cutting Cycle 473
Thread Milling Cycle 428
Timing Synchronization Operation 732
Timing Synchronization Operation (! code) 732
Timing Synchronization Operation Function Using M codes 742
Timing Synchronization Operation with Start Point Designated (Type 1) 736
Timing Synchronization Operation with Start Point Designated (Type 2) 739
Timing Synchronization When Timing Synchronization Ignore Is Set 746
Tolerance Control 815
Tool Axis Direction Control 1025
Tool Center Point Control 953
Tool Change Position Return 1184
Tool Compensation 284
Tool Cutting Point Control 990
Tool Functions (T8-digit BCD) 282
Tool Length Compensation Along the Tool Axis 946
Tool Length Offset/Cancel 290
Tool Life Management Set Allocation to Part Systems 650
Tool Nose Radius Compensation (for Machining Center System) 356
Tool Offset Command Check 355, 381

Tool Position Compensation 934
Tool Position Offset 371
Tool Radius Compensation 308
Tool Radius Compensation Operation 309
Tool Shape Input by Program 635
Tool Spindle Synchronization IA (Spindle-Spindle, Polygon) 901
Tool Spindle Synchronization IB (Spindle-Spindle, Polygon) 907
Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon) 912
Tool Spindle Synchronization II (Hobbing) 916
Torque Limitation Skip 1220
Two-dimensional Barcode Engraving Cycle 679

U

Unidirectional Positioning 77
User Macro 499
User Macro Commands 535

V

Variable Commands 493
Variable Value Display in Running Program 567
Variable-acceleration Pre-interpolation Acceleration/ Deceleration 819
Variables Used in User Macros 523
Vibration Cutting Control (VCC) 662

W

Workpiece Coordinate Changing during Radius Compensation 353
Workpiece Coordinate System Preset 1108
Workpiece Coordinate System Selection and Extended Workpiece Coordinate System Selection ... 1099
Workpiece installation error compensation 1066
Workpiece position offset for rotary axis 1113

Revision History

Revision date	Manual No.	Changes
Jul. 2021	IB(NA)1501621-A IB(NA)1501622-A	First edition created.
Mar. 2022	IB(NA)1501621-B IB(NA)1501622-B	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version A3. The following chapters were changed. •6.6.1 Constant Lead Thread Cutting; G33 •6.9 Cylindrical Interpolation; G07.1 •7.4 Feed Per Minute/Feed Per Revolution (Asynchronous Feed/ Synchronous Feed); G94, G95 •8.1 Dwell (Time-based Designation); G04 •8.2 Dwell (Revolution-based Designation); G04 •12.2 Tool Length Compensation/Cancel; G43, G44 / G49 •14.6.2 Control Commands •15.9.6 R-Navi Data Input by Program; G10 L110/L111, G11, G68.2, G69 •17.1.1 High-speed Machining Mode I, II; G05 P1, G05 P2 •17.3.6 Precautions on High-speed High-accuracy Control •18.1.1 Spindle Synchronization I; G114.1 •19.3 Tool Center Point Control; G43.4, G43.5/G49 •19.5.12 Precautions for Inclined Surface Machining •19.6 3-dimensional Tool Radius Compensation (Tool's Vertical-direction Compensation); G40/G41.2, G42.2 •19.7 Workpiece Installation Error Compensation; G54.4 •20.10 Coordinate Rotation by Program; G68/G69 •23.3 System Variables (Non-G Command Modal) Other mistakes were corrected.
Mar. 2023	IB(NA)1501621-C IB(NA)1501622-C	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version A7. The following chapters were added. •14.5.2 Extended Common Variable III •16.4 Loader Control Part System •19.5.11 Inclined Surface Machining Multi-command •23.32 System Variables (Extended Common Variables) The following chapters were changed. •3.4.2 G Code Lists •6.6.1 Constant Lead Thread Cutting; G33 •6.7 Helical Interpolation; G02, G03 •6.9 Cylindrical Interpolation; G07.1 •6.11 Polar Coordinate Interpolation; G12.1, G13.1/G112, G113 •6.12 Exponential Interpolation; G02.3, G03.3 •7.14.1 Rapid Traverse Block Overlap for G00; G0.5 •7.14.2 Rapid Traverse Block Overlap for G28 •10.4 Spindle Position Control (Spindle/C Axis Control) •12.2 Tool Length Compensation/Cancel; G43, G44 / G49 •14.2 Variable Commands •15.12 Axis Name Extension •18.3.1 Tool Spindle Synchronization II (Hobbing) ; G114.3/G113 (Continue to the next page)

Revision date	Manual No.	Changes
		(Continued from the previous page) •19.2 Tool Length Compensation Along the Tool Axis; G43.1/G49 •19.3 Tool Center Point Control; G43.4, G43.5/G49 •19.5 Inclined Surface Machining; G68.2, G68.3, G68.4/G69 •19.5.12 Relationship between Inclined Surface Machining and Other Function •19.5.13 Precautions for Inclined Surface Machining •19.6 3-dimensional Tool Radius Compensation (Tool's Vertical-direction Compensation); G40/G41.2, G42.2 •19.7 Workpiece Installation Error Compensation; G54.4 •23.1 System Variables List •23.14 System Variables (Message Display and Stop) Other mistakes were corrected.
Jul. 2023	IB(NA)1501621-D IB(NA)1501622-D	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version A8. The following chapters were added. •14.5.2 Extended Common Variable •14.5.2.1 Extended common variable I •14.5.2.3 Precautions of extended common variable •14.4.7 Handling the Macro Call Command The following chapters were changed. •3.3.1 Optional Block Skip; / •6.11 Polar Coordinate Interpolation; G12.1, G13.1/G112, G113 •6.18 Involute Interpolation; G02.2/G03.2 •7.8 Rapid Traverse Constant-gradient Acceleration/Deceleration •10.4 Spindle Position Control (Spindle/C Axis Control) •14.2 Variable Commands •14.5.2.2 Extended common variable III •14.7 Macro Interruption; M96, M97 •17.5 Spline Interpolation 2; G61.4 •19.3 Tool Center Point Control; G43.4, G43.5/G49 •19.5 Inclined Surface Machining; G68.2, G68.3, G68.4/G69 •19.5.12 Relationship between Inclined Surface Machining and Other Functions •19.5.13 Precautions for Inclined Surface Machining •19.7 Workpiece Installation Error Compensation; G54.4 •19.9 Applicable Machines •23.1 System Variables List •23.2 System Variables (G Command Modal) •23.31 System Variables (Extended Common Variables) The following chapter was deleted. •23.11 System Variables (External Workpiece Coordinate Offset) Other mistakes were corrected.

Revision date	Manual No.	Changes
Dec. 2023	IB(NA)1501621-E IB(NA)1501622-E	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version A9. The following chapters were added. •13.1.21 Chip Removal •13.1.22 Cutting Reduction Amount Specification Method •14.8 Variable Value Display in Running Program •15.13 Vibration Cutting Control (VCC); G08.5 •15.14 Text Engraving Cycle; G135 The following chapters were changed. •3.4.2 G Code Lists •7.8 Rapid Traverse Constant-gradient Acceleration/Deceleration •12.3.1 Tool Radius Compensation Operation •12.3.2 Other Commands and Operations during Tool Radius Compensation •12.3.6 Changing of Compensation No. during Compensation Mode •12.3.7 Start of Tool Radius Compensation and Z Axis Cut in Operation •13.1 Fixed Cycles •13.1.1 Drilling, Spot Drilling; G81 •13.1.2 Drilling, Counter Boring; G82 •13.1.3 Deep Hole Drilling Cycle; G83 •13.1.4 Tapping Cycle; G84 •13.1.10 Stepping Cycle; G73 •15.10.3 Precautions for Inputting the Tool Life Management Data •17.3.1 High-speed High-accuracy Control I, II, III ; G05.1 Q1/Q0, G05 P10000/P0, G05 P20000/P0 •17.5 Spline Interpolation 2; G61.4 •17.7 Machining Condition Selection I ; G120.1, G121 •19.2 Tool Length Compensation Along the Tool Axis; G43.1/G49 •19.3 Tool Center Point Control; G43.4, G43.5/G49 •19.4 Tool Cutting Point Control; G43.8/G43.9 •19.6 3-dimensional Tool Radius Compensation (Tool's Vertical-direction Compensation); G40/G41.2, G42.2 •19.7 Workpiece Installation Error Compensation; G54.4 •19.9 Applicable Machines •23.2 System Variables (G Command Modal) •23.3 System Variables (Non-G Command Modal) Other mistakes were corrected.
Apr. 2024	IB(NA)1501621-F IB(NA)1501622-F	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version AA. The following chapter was added. •16.3 Control Axis Synchronization between Part Systems ; G125 The following chapters were changed. •3.4.2 G Code Lists •7.13.1 Deceleration Check •14.4.4 G Code Macro Call •14.4.5 Miscellaneous Command Macro Call (for M, S, T, B Code Macro Call) •15.13 Vibration Cutting Control (VCC); G08.5 •17.1.1 High-speed Machining Mode I, II; G05 P1, G05 P2 •17.3.1 High-speed High-accuracy Control I, II, III ; G05.1 Q1/Q0, G05 P10000/P0, G05 P20000/P0 •20.9 3-dimensional Coordinate Conversion; G68/G69 Other mistakes were corrected.

Revision date	Manual No.	Changes
Oct. 2024	IB(NA)1501621-G IB(NA)1501622-G	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version B0. The following chapters were added. •5.4.1 Decimal Point Command Valid/Invalid Address Changeover •12.3.11 Tool Offset Command Check •12.7 Tool Offset Number •23.12 System Variables (Position Information Common to Part System) •23.21 System Variables (Acquiring the Program Execution Part System No.) •23.22 System Variables (Display Axis Name Selection) The following chapters were changed. •5.4 Decimal Point Input •6.11 Polar Coordinate Interpolation; G12.1, G13.1/G112, G113 •12.2 Tool Length Compensation/Cancel; G43, G44 / G49 •12.3 Tool Radius Compensation; G38,G39/G40/G41,G42 •12.6 Tool Position Offset; G45 to G48 •13.1.10 Stepping Cycle; G73 •13.1.11 Reverse Tapping Cycle; G74 •13.1.21 Chip Removal •14.1.2 Subprogram Call; M198 •14.4 Macro Call Instructions •14.4.4 G Code Macro Call •14.4.5 Miscellaneous Command Macro Call (for M, S, T, B Code Macro Call) •14.8 Variable Value Display in Program Being Executed •15.13 Vibration Cutting Control (VCC); G08.5 •15.16 Cutting Load Control; G162.1 •16.3 Control Axis Synchronization between Part Systems ; G125 •16.4.1 Sub Part System Control I; G122 •17.1.1 High-speed Machining Mode I, II; G05 P1, G05 P2 •17.3.1 High-speed High-accuracy Control I, II, III ; G05.1 Q1/Q0, G05 P10000/P0, G05 P20000/P0 •18.3.1 Tool Spindle Synchronization II (Hobbing) ; G114.3/G113 •19.1 Tool Position Compensation; G43.7/G49 •19.2 Tool Length Compensation Along the Tool Axis; G43.1/G49 •19.3 Tool Center Point Control; G43.4, G43.5/G49 •19.4 Tool Cutting Point Control; G43.8/G43.9 •19.5 Inclined Surface Machining; G68.2, G68.3, G68.4/G69 •19.5.8 Tool Axis Direction Control; G53.1/G53.6 •19.6 3-dimensional Tool Radius Compensation (Tool's Vertical-direction Compensation); G40/G41.2, G42.2 •19.8 Rotation Center Error Compensation (Precautions for Creating a Machining Program) •19.9 Applicable Machines •20.9 3-dimensional Coordinate Conversion; G68/G69 •22.2 Skip Function; G31 •23.1 System Variables List •23.11 System Variables (Position Information by Part System) •23.32 System Variables (Interfering Object Selection) Other mistakes were corrected.

Revision date	Manual No.	Changes
Apr. 2025	IB(NA)1501621-H IB(NA)1501622-H	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version B2. The following chapters were added. •23.35 System Variables (Axis Exchange Information Acquisition) •23.36 System Variables (Reference Axis Arrangement Information Acquisition) The following chapters were changed. •5.3 Inch/Metric Conversion; G20, G21 •5.4 Decimal Point Input •6.6.1 Constant Lead Thread Cutting; G33 •6.6.2 Inch Thread Cutting; G33 •7.14.1 Rapid Traverse Block Overlap for G00; G0.5 •12.2 Tool Length Offset/Cancel; G43, G44 / G49 •14.7 Macro Interruption; M96, M97 •15.13 Vibration Cutting Control (VCC); G08.5 •16.1.1 Timing Synchronization Operation (! code) !n (!m ...) L •16.1.4 Timing Synchronization Operation Function Using M codes ; M*** •16.1.5 Timing Synchronization When Timing Synchronization Ignore Is Set •16.4.1 Sub Part System Control I; G122 •17.1.1 High-speed Machining Mode I, II; G05 P1, G05 P2 •17.2.1 High-accuracy Control ; G61.1, G08 •17.2.3 Tolerance Control •17.2.6 High-speed High-accuracy Control in Multi-part System •17.3.1 High-speed High-accuracy Control I, II, III ; G05.1 Q1/Q0, G05 P10000/P0, G05 P20000/P0 •17.3.6 Precautions on High-speed High-accuracy Control •17.4 Spline Interpolation ; G05.1 Q2/Q0 •20.9 3-dimensional Coordinate Conversion; G68/G69 •20.10 Coordinate Rotation by Program; G68/G69 •22.2 Skip Function; G31 •22.4 Multi-step Skip Function 2; G31 P •23.1 System Variables List •23.5 System Variables (Tool Information) •23.11 System Variables (Position Information by Part System) •23.12 System Variables (Position Information Common to Part System) •23.30 System Variables (R Device Access Variables) •23.33 System Variables (NC Data Reading/Writing with API Section and Sub-section Nos. Input/Output by Program) Other mistakes were corrected.

Revision date	Manual No.	Changes
Aug. 2025	IB(NA)1501621-J IB(NA)1501622-J	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version B4. The following chapter was added. •12.3.12 Arc Intersection Point Calculation Switch The following chapters were changed. •1.1 Coordinate Words and Control Axes •7.14.1 Rapid Traverse Block Overlap for G00; G0.5 •10.2 Constant Surface Speed Control; G96, G97 •12.1.2 Number of Tool Offset Sets Allocation to Part Systems •13 Fixed Cycle •13.1.4 Tapping Cycle; G84 •13.1.7 Back Boring; G87 •13.1.13 Fine Boring; G76 •13.1.16 Precautions for Using a Fixed Cycle •14.6.2 Control Commands •15.10.4 Tool Life Management Set Allocation to Part Systems •15.13 Vibration Cutting Control (VCC); G08.5 •15.14 Two-dimensional Barcode Engraving Cycle; G136 •15.15 Text Engraving Cycle; G135 •20.1 Coordinate Words and Control Axes •20.2.4 Coordinate System for Rotary Axis •20.5 Local Coordinate System Setting; G52 •20.6 Workpiece Coordinate System Selection and Extended Workpiece Coordinate System Selection; G54 to G59, G54.1 •23.6 System Variables (Tool Compensation) •23.15 System Variables (Cumulative Time) Other mistakes were corrected.
Jan. 2026	IB(NA)1501621-K IB(NA)1501622-K	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version B8. Chapters 3, 6, 9, 12, 14, 17, 20, and 21 were modified. The main changes are as follows: The following chapter titles were changed: •17.2.6 High-speed Machining Mode and High-accuracy Control in Multi-part System •20.13 Automatic 1st Reference Position Return and Start Position Return; G28, G29 •20.14 2nd, 3rd, 4th Reference Position Return; G30 The contents were revised as follows. Chapter numbers are indicated in parentheses. •Improved explanations for control out and control in. (3.1) •Added the explanation to use involute interpolation and tool radius compensation together. (6.18) •Added the explanation on setting range for spindle position control (spindle/C axis control). (7.1, 7.2) •Corrected mistakes in the command range of cutting feed clamp speed. (7.2) •Corrected description of the relationship between the index table indexing and G27. (9.3) •Added explanation regarding the G40 command issued alone during arc modal. (12.5.2) •Revised explanation on reducing the processing time for user macro control command blocks. (14.6.2) (Continue to the next page)

Revision date	Manual No.	Changes
		(Continued from the previous page) •Added explanation regarding parameter group retention after resetting machining condition selection. (17.7) •Deleted mistakes regarding the parameters "#1148 I_G611" and "#1151 rstini" in high-accuracy control. (17.2.1) •Revised explanation on the coordinate rotation by program and corrected the program example for "Operation immediately after start of coordinate rotation". (20.10) •Corrected the figure of the first movement command after coordinate rotation cancel of parameter coordinate rotation. (20.11) •Added parameters for single block stop at intermediate point and improved overall explanations related to reference position return functions. (20.13, 20.14) •Added explanation on consistency check in interfering object selection. (21.2) Other mistakes were corrected.
May 2026	IB(NA)1501621-L IB(NA)1501622-L	Contents were revised to correspond to Mitsubishi Electric CNC M800V/M80V Series software version BB. Chapters 3, 5, 12, 15, 16, 17, 19, and 23 were modified. The main changes are as follows. Chapter numbers are indicated in parentheses. The following chapter was added. •15.17 Orbit Machining; G08.6 The following chapters were deleted. •19.8 Rotation Center Error Compensation (Precautions for Creating a Machining Program) •19.9 Applicable Machines The contents were revised as follows. •Added explanation on unmodal commands. (3.4.1) •Added G codes and addresses due to addition of orbit machining (G08.6). (3.4.2, 5.4) •Deleted duplicate description. (12.3.1) •Changed reference of rotation center error compensation contents to the Instruction Manual. (16.3) •Deleted unnecessary description about axis addresses. (17.1.1, 17.2.1, 17.3) •Added explanation on the parameter #12053. (17.2.1) •Modified explanation on modal when the high-accuracy control is enabled/disabled. (17.2.1) •Modified description about applicable machine configurations. (19.3, 19.4, 19.5, 19.6, 19.7) •Corrected the numbering of the #23000 series in the system variable list (23.1). Other mistakes were corrected.

Global Service Network

AMERICA

MITSUBISHI ELECTRIC AUTOMATION INC. (AMERICA FA CENTER)
HQ and 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
Cleveland, OH Service Satellite
St. Louis, MO Service Satellite
Indianapolis, IN Service Saltellite

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
Huntsville, AL Satellite
Pittsburg, PA Service Satellite
Tampa, FL Service Satellite
Miami, FL Satelite
Lafayette, LA Service Satellite
Allentown, PA Satelite

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 Satelite
Seattle, WA Service Satellite
Denver, CO Service Satellite

Canada Region Service Center (Toronto)
4299 14TH AVENUE MARKHAM, ONTARIO L3R OJ2, CANADA
TEL: +1-905-475-7728 / FAX: +1-905-475-7935
Edmonton, AB Service Satellite
Montreal, QC Service Satellite

Mexico Region Service Center (Queretaro)
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: +52-442-153-6050
Monterrey, NL Service Satellite
Mexico City, DF 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
Blumenau, Santa Catarina Office

EUROPE

MITSUBISHI ELECTRIC EUROPE B.V.
European Service Headquarters (Dusseldorf, GERMANY)
Mitsubishi-Electric-Platz 1 40882 RATINGEN, GERMANY
TEL: +49-2102-486-5000 / FAX: +49-2102-486-5910

South Germany Service Center (Stuttgart)
SCHELMENWASENSTRASSE 16-20, 70567 STUTTGART, GERMANY
TEL: + 49-711-770598-123 / FAX: +49-711-770598-141

France Service Center (Paris)
2 RUE DE L'UNION, 92565 RUEIL-MALMAISON CEDEX, FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

France Service Satellite (Lyon)
240, 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)
VIA ENERGY PARK 14, VIMERCATE 20871 (MB) 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: +44-1707-288-780 / FAX: +44-1707-278-695

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
MITSUBISHI ELECTRIC EUROPE B.V. HUNGARIAN BRANCH
BUDAÖRS OFFICE PARK, SZABADSÁG ÚT 117., 2040 BUDAÖRS, HUNGARY
TEL: +36-70-433-2263

Turkey Service Center
MITSUBISHI ELECTRIC TURKEY ELEKTRİK ÜRÜNLERİ A.Ş
SERIFALI MAHALLESI KALE SOKAK. NO.41 34775
UMRANIYE, ISTANBUL, TURKEY
TEL: +90-216-969-2500 / FAX: +90-216-661-44-47

Czech Republic Service Center
MITSUBISHI ELECTRIC EUROPE B.V. PRAGUE OFFICE
Pekařská 621/7, 155 00 PRAHA 5, CZECH REPUBLIC
TEL: +420-734-402-587

Sweden Service Center
HAMMARBACKEN 14, P.O.BOX 750 SE-19127, SOLLENTUNA, SWEDEN
TEL: +46-8-6251200 / FAX: +46-8-6251014

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 (Kiev)
CSC Automation Ltd. (Service Partner)
4 B, YEVHENA SVERSTYUKA STR., 02002 KIEV, UKRAINE
TEL: +380-44-494-3344 / FAX: +380-44-494-3366

South Africa Service Center
Adroit Technologies (Service Partner)
20 WATERFORD OFFICE PARK, WATERFORD DRIVE, CNR OF WITKOPPEN ROAD,
FOURWAYS JOHANNESBURG SOUTH AFRICA
TEL: +27-11-658-8100 / FAX: +27-11-658-8101

ASEAN

MITSUBISHI ELECTRIC ASIA PTE. LTD. (ASEAN FA CENTER)
Singapore Service Center
307 ALEXANDRA ROAD MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 / FAX: +65-6476-7439

PHILIPPINES

MELCO FACTORY AUTOMATION PHILIPPINES INC.
Head Office
128 LOPEZ RIZAL STREET, BRGY., HIGHWAY HILLS, MANDALUYONG CITY , MM PHILIPPINES 1550
TEL: +63-2-8256-8042 / FAX: +632-8637-2294

Philippines Service Center
KM.23 WEST SERVICE ROAD SSH, CUPANG ,MUNTINLUPA CITY, PHILIPPINES
TEL: +63-2-8807-0420 / FAX: +63-2-8842-5202

VIETNAM

MITSUBISHI ELECTRIC VIETNAM CO., LTD.
Vietnam Ho Chi Minh Service Center
11TH & 12TH FLOOR, VIETTEL TOWER B, 285 CACH MANG THANG 8 STREET, WARD 12, DISTRICT 10,
HO CHI MINH CITY, VIETNAM
TEL: +84-28-3910-5945 / FAX: +84-28-3910-5947

Vietnam Hanoi Service Center
14TH FLOOR, CAPITAL TOWER, 109 TRAN HUNG DAO STREET, CUA NAM WARD,
HOAN KIEM 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: +60-3-7626-5032

Johor Bahru Service Center
9, Jalan Perjiranan 4/6, Bandar Dato Onn, 81100 Johor.
TEL: 07-3642250 / 2256

Pulau Pinang Service Center
25-G, Pusat Perniagaan Perdana Jaya, Jalan Permatang Rawa, 14000 Bukit Mertajam, Pulau Pinang
TEL: +60-4-510-1838 / Fax: +60-4-510-1835

THAILAND

MITSUBISHI ELECTRIC FACTORY AUTOMATION (THAILAND) CO., LTD.
Thailand Service Center (Bangkok)
101, TRUE DIGITAL PARK OFFICE, 5TH FLOOR, SUKHUMVIT ROAD, BANGCHAK, PHRA KHANONG,
BANGKOK, 10260 THAILAND
TEL: +66-2-092-8600 / FAX: +66-2-043-1231-33

INDIA

MITSUBISHI ELECTRIC INDIA PVT., LTD.
Service Head office and South India Service Center (Bangalore)
CNC eXPerience Park, PLOT NO. 57/A, PEENYA PHASE 3,
PEENYA INDUSTRIAL AREA, BANGALORE 560058, KARNATAKA, INDIA
TEL : +91-80-4655-2121
Chennai Service Satellite
Coimbatore Service Satellite
Hyderabad Service Satellite

North India Service Center (Gurgaon)
PLOT 517, GROUND FLOOR, UDYOG VIHAR PHASE-III, GURUGRAM 122008, HARYANA, INDIA
TEL : +91-124-463-0300
Ludhiana Service Satellite
Panthnagar Service Satellite
Delhi Service Satellite
Jamshedpur Service Satellite
Manesar Service Satellite

West India Service Center (Pune)
ICC-Devi GAURAV TECHNOLOGY PARK, UNIT NO.402, FOURTH FLOOR, NORTH WING,
SURVEY NUMBER 191-192 (P), NEXT to INDIAN CARD CLOTHING COMPANY Ltd,
OPP. VALLABH NAGAR, PIMPRI, PUNE- 411 018, MAHARASHTRA, INDIA
TEL : +91-20-6819-2274
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 : + 91-79-6777-7888
Rajkot Service Satellite

CHINA

MITSUBISHI ELECTRIC AUTOMATION (CHINA) LTD. (CHINA FA CENTER)
CNC Call Center
TEL: +86-400-921-5130

Shanghai Service Center
NO.1386 HONG QIAO ROAD, CHANG NING QU, SHANGHAI 200336, CHINA
TEL: +86-21-2322-3030 / FAX: +86-21-2322-3000*8422
Qingdao Service Center
Suzhou Service Center
Wuhan Service Center
Ningbo Service Center
Hefei Service Center
Beijing Service Center
Tianjin Service Center
Xian Service Center
Dalian Service Center
Chengdu Service Center

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
Dongguan Service Center
Xiamen Service Center

KOREA

MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD. (KOREA FA CENTER)
Korea Seoul Service Center
8F GANGSEO HANGANG XI-TOWER A, 401 YANGCHEON-RO, GANGSEO-GU,
SEOUL 07528 KOREA
TEL: +82-2-3660-9631 / FAX: +82-2-3664-8668

Korea Daegu Service Center
2F KT BLDG. 8, HOGUK-RO, BUK-GU, DAEGU 41518, KOREA
TEL: +82-53-382-7401 / FAX: +82-53-382-7412

Korea Changwon Technical Center
#205, 888, CHANGWON-DAERO, SEONGSAN-GU, CHANGWON-SI,
GYEONGSANGNAM-DO, 51532, KOREA
TEL: +82-55-261-5559 / FAX: +82-55-261-5558

TAIWAN

MITSUBISHI ELECTRIC TAIWAN CO., LTD. (TAIWAN FA CENTER)
Taiwan Taichung Service Center
NO. 8-1, GONGYEQU 16TH RD., XITUN DIST., TAICHUNG CITY 40768 , TAIWAN
TEL: +886-4-2359-0688 / FAX: +886-4-2359-0689

Taiwan Taipei Service Center
11F, NO.88, SEC.6, ZHONGSHAN N. RD., SHILIN 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. RD., YONGKANG DIST., 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 2021-2026 MITSUBISHI ELECTRIC CORPORATION ALL RIGHTS RESERVED

MITSUBISHI ELECTRIC CORPORATION
HEAD OFFICE : TOKYO BLDG.,2-7-3 MARUNOUCHI,CHIYODA-KU,TOKYO 100-8310,JAPAN

MODEL	M800V/M80V Series
MODEL CODE	100-752 100-753
Manual No.	IB-1501621

Made in JAPAN

Specifications are subject to change without notice.