



Changes for the Better

MITSUBISHI CNC

Programming Manual (Lathe System)
M700V/M70V Series








Introduction

This manual is a guide for using the MITSUBISHI CNC M700V/M70V Series.

Programming is described in this manual, so read this manual thoroughly before starting programming. Thoroughly study the "Precautions for Safety" on the following page to ensure safe use of this NC unit.

Details described in this manual

CAUTION




-  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 option functions are added.
Refer to the specifications issued by the machine tool builder before starting 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.


General precautions

- (1) Refer to the following documents for details on handling
MITSUBISHI CNC M700V/M70V Series Instruction Manual IB-1500922





Precautions for Safety

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










 DANGER When the user may be subject to imminent fatalities or major injuries if handling is mistaken.
 WARNING When the user may be subject to fatalities or major injuries if handling is mistaken.
 CAUTION When the user may be subject to injuries or when physical damage may occur if handling is mistaken.

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

The following signs indicate prohibition and compulsory.

	This sign indicates prohibited behavior (must not do). For example,  indicates "Keep fire away".
	This sign indicates a thing that is compulsory (must do). For example,  indicates "it must be grounded".

The meaning of each pictorial sign is as follows.

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

For Safe Use

Mitsubishi CNC is designed and manufactured solely for applications to machine tools to be used for industrial purposes.



Do not use this product in any applications other than those specified above, especially those which are substantially influential on the public interest or which are expected to have significant influence on human lives or properties.

 DANGER

Not applicable in this manual.






 WARNING
--

1. Items related to operation






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

CAUTION






1. Items related to product and manual

-  For items described as "Restrictions" or "Usable State" in this manual, the instruction manual issued by the machine tool builder takes precedence over this manual.
-  Items not described in this manual must be interpreted as "not possible".
-  This manual is written on the assumption that all option functions are added. Refer to the specifications issued by the machine tool builder before starting 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.

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 synchronous spindle rotation command OFF with one workpiece chucked by the basic spindle and synchronous spindle during the spindle synchronization.
Failure to observe this may cause the synchronous spindle stop, and hazardous situation.

3. Items related to programming

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

Disposal



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

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

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

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

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

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

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

Trademarks

MELDAS, MELSEC, EZSocket, EZMotion, iQ Platform, MELSOFT, GOT, CC-Link, CC-Link/LT and CC-Link IE 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® and Windows® are either trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.

CompactFlash and CF are either trademarks or registered trademarks of SanDisk Corporation in the United States and/or other countries.

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.

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

本製品の取扱いについて

(日本語/Japanese)

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

Handling of our product

(English)

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

본 제품의 취급에 대해서

(한국어/Korean)

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

CONTENTS

1 Control Axes	1
1.1 Coordinate Words and Control Axes	2
1.2 Coordinate Systems and Coordinate Zero Point Symbols	4
2 Least Command Increments	7
2.1 Input Setting Unit	8
2.2 Indexing Increment	9
3 Program Formats	11
3.1 Program Format	12
3.2 File Format	16
3.3 Optional Block Skip	18
3.3.1 Optional Block Skip; /	18
3.3.2 Optional Block Skip Addition ; /n	20
3.4 G code	22
3.4.1 Modal, unmodal	22
3.4.2 G code Lists	22
3.4.3 Table of G Code Lists	23
3.5 Precautions Before Starting Machining	28
4 Pre-read Buffers	29
4.1 Pre-read Buffers	30
5 Position Commands	31
5.1 Absolute/Incremental Value Commands ; G90,G91	32
5.2 Radius/Diameter Designation	34
5.3 Inch/Metric Conversion ; G20,G21	35
5.4 Decimal Point Input	37
6 Interpolation Functions	43
6.1 Positioning (Rapid Traverse) ; G00	44
6.2 Linear Interpolation ; G01	51
6.3 Circular Interpolation ; G02,G03	53
6.4 R Specification Circular Interpolation ; G02,G03	57
6.5 Plane Selection ; G17,G18,G19	59
6.6 Thread Cutting	61
6.6.1 Constant Lead Thread Cutting ; G33	61
6.6.2 Inch Thread Cutting ; G33	65
6.6.3 Continuous Thread Cutting ; G33	67
6.6.4 Variable Lead Thread Cutting ; G34	68
6.6.5 Circular Thread Cutting ; G35,G36	71
6.7 Helical Interpolation ; G17,G18,G19 and G02,G03	75
6.8 Milling Interpolation ; G12.1	79
6.8.1 Selecting Milling Mode	81
6.8.2 Milling Interpolation Control and Command Axes	82
6.8.3 Selecting a Plane during the Milling Mode ; G17,G19,G16	84
6.8.4 Setting Milling Coordinate System	86
6.8.5 Preparatory Function	88
6.8.6 Switching from Milling Mode to Turning Mode; G13.1	93
6.8.7 Feed Functions	93
6.8.8 Program Support Functions	93
6.8.9 Miscellaneous Functions	94
6.8.10 Tool Length Compensation	95
6.8.11 Tool Radius Compensation	98
6.8.11.1 Tool Radius Compensation Operation	99
6.8.11.2 Interference Check	116

6.9 Cylindrical Interpolation ; G07.1 (only 6 and 7 in G code list)	126
6.10 Polar Coordinate Interpolation ; G12.1,G13.1/G112,G113 (Only 6, 7 in G code list)	134
6.11 Exponential Interpolation ; G02.3,G03.3	141
7 Feed Functions	147
7.1 Rapid Traverse Rate	148
7.2 Cutting Feedrate	149
7.3 F1-digit Feed	150
7.4 Feed Per Minute/Feed Per Revolution (Asynchronous Feed/Synchronous Feed) ; G94,G95.....	153
7.5 Feedrate Designation and Effects on Control Axes	156
7.6 Thread Cutting Mode	161
7.7 Automatic Acceleration/Deceleration	163
7.8 Rapid Traverse Constant Inclination Acceleration/Deceleration	167
7.9 Cutting Feed Constant Inclination Acceleration/Deceleration	172
7.10 Speed Clamp	179
7.11 Exact Stop Check ; G09.....	180
7.12 Exact Stop Check Mode ; G61	184
7.13 Deceleration Check.....	185
7.13.1 G1 -> G0 Deceleration Check.....	187
7.13.2 G1 -> G1 Deceleration Check.....	188
7.14 Automatic Corner Override ; G62.....	189
7.15 Tapping Mode ; G63	194
7.16 Cutting Mode ; G64.....	195
8 Dwell.....	197
8.1 Dwell (Time Designation) ; G04	198
9 Miscellaneous Functions	201
9.1 Miscellaneous Functions (M8-digits)	202
9.2 Secondary Miscellaneous Functions (A8-digits, B8-digits or C8-digits)	204
9.3 Index Table Indexing	205
9.4 M Code Output during Axis Traveling ; G117	207
10 Spindle Functions.....	209
10.1 Spindle Functions.....	210
10.2 Constant Surface Speed Control ; G96,G97.....	211
10.3 Spindle Clamp Speed Setting ; G92	213
10.4 Spindle/C Axis Control	215
10.5 Spindle Synchronization.....	218
10.5.1 Spindle Synchronization Control I ; G114.1	219
10.5.2 Spindle Synchronization Control II	229
10.5.3 Precautions for Using Spindle Synchronization Control	234
10.6 Tool Spindle Synchronization IA (Spindle-Spindle, Polygon) ; G114.2	236
10.7 Tool Spindle Synchronization IB (Spindle-Spindle, Polygon) ; G51.2 (only 6 and 7 in G code list)	245
10.8 Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon) ; G51.2 (only 6 and 7 in G code list)	252
10.9 Tool Spindle Synchronization II (Hobbing) ; G114.3.....	255
10.10 Multiple-spindle Control.....	273
10.10.1 Multiple-spindle Control I (spindle control command) ; S ○ =	274
10.10.2 Multiple-spindle Control I (spindle selection command) ; G43.1,G44.1,G47.1	275
10.10.3 Multiple-spindle Control II	279
11 Tool Functions	283
11.1 Tool Functions (T8-digit BCD)	284
12 Tool Compensation Functions	285
12.1 Tool Compensation.....	286
12.1.1 Tool Compensation Start	287
12.1.2 Expanded Method at Starting Tool Compensation	288

12.2 Tool Length Compensation	290
12.3 Tool Nose Wear Compensation	292
12.4 Tool Nose R Compensation ; G40,G41,G42,G46.....	293
12.4.1 Tool Nose Point and Compensation Directions	295
12.4.2 Tool Nose Radius Compensation Operations	298
12.4.3 Other Operations during Tool Nose Radius Compensation	316
12.4.4 G41/G42 Commands and I, J, K Designation	324
12.4.5 Interrupts during Tool Nose Radius Compensation	329
12.4.6 General Precautions for Tool Nose Radius Compensation	332
12.4.7 Interference Check	333
12.5 Compensation Data Input by Program ; G10 L2/L10/L11, G11	339
12.6 Tool Life Management II ; G10 L3, G11.....	342
12.6.1 Counting the Tool Life	345

13 Program Support Functions 349

13.1 Fixed Cycles for Turning Machining	350
13.1.1 Longitudinal Cutting Cycle ; G77	351
13.1.2 Thread Cutting Cycle ; G78	354
13.1.3 Face Cutting Cycle ; G79.....	357
13.2 Fixed Cycles for Turning Machining (MITSUBISHI CNC special format) ; G77,G78,G79	360
13.3 Compound Type Fixed Cycle for Turning Machining	362
13.3.1 Longitudinal Rough Cutting Cycle ; G71	363
13.3.2 Face Rough Cutting Cycle ; G72	377
13.3.3 Formed Material Rough Cutting Cycle ; G73.....	379
13.3.4 Finishing Cycle ; G70.....	383
13.3.5 Face Cut-Off Cycle ; G74.....	385
13.3.6 Longitudinal Cut-off Cycle ; G75	387
13.3.7 Compound Thread Cutting Cycle ; G76.....	389
13.3.8 Precautions for Compound Type Fixed Cycle for Turning Machining; G70 to G76	393
13.4 Compound Type Fixed Cycle for Turning Machining (MITSUBISHI CNC special format) ; G71,G73,G74,G76.....	395
13.5 Fixed Cycle for Drilling	400
13.5.1 Face Deep Hole Drilling Cycle 1 (Longitudinal deep hole drilling cycle 1) ; G83 (G87).....	403
13.5.2 Face Tapping Cycle (Longitudinal tapping cycle) / Face Reverse Tapping Cycle (Longitudinal reverse tapping cycle) ; G84 (G88) / G84.1 (G88.1)	405
13.5.3 Face Boring Cycle (Longitudinal boring cycle) ; G85 (G89).....	416
13.5.4 Deep Hole Drilling Cycle 2 ; G83.2	418
13.5.5 Fixed Cycle for Drilling Cancel; G80	420
13.5.6 Precautions When Using a Fixed Cycle for Drilling	420
13.5.7 Initial Point and R Point Level Return ; G98,G99.....	421
13.5.8 Setting of Workpiece Coordinates in Fixed Cycle Mode	422
13.5.9 Drilling Cycle High-Speed Retract	423
13.5.10 Acceleration/Deceleration Mode Change in Hole Drilling Cycle	424
13.6 Fixed Cycle for Drilling (MITSUBISHI CNC special format)	426
13.6.1 Drilling Cycle, Spot Drilling Cycle ; G81	429
13.6.2 Drilling Cycle, Counter Boring Cycle ; G82	430
13.6.3 Deep Hole Drilling Cycle ; G83	431
13.6.4 Stepping Cycle ; G83.1	433
13.6.5 Tapping Cycle ; G84	435
13.6.6 Synchronous Tapping Cycle ; G84.2	439
13.6.7 Boring Cycle ; G85.....	444
13.6.8 Boring Cycle ; G89.....	445
13.6.9 Precautions for Using Fixed Cycle for Drilling (MITSUBISHI CNC special format)	446
13.7 Subprogram Control; M98, M99, M198	448
13.7.1 Subprogram Call ; M98,M99.....	448
13.7.2 Subprogram Call ; M198.....	454
13.8 Variable Commands	455
13.9 User Macro	460
13.9.1 User Macro	460
13.9.2 Macro Call Instruction	461
13.9.2.1 Simple Macro Calls ; G65	461

13.9.2.2 Modal Call A (Movement Command Call) ; G66.....	465
13.9.2.3 Modal Call B (for each block) ; G66.1	467
13.9.2.4 G Code Macro Call	469
13.9.2.5 Miscellaneous Command Macro Call (for M, S, T, B Code Macro Call)	470
13.9.2.6 Detailed Description for Macro Call Instruction	471
13.9.3 ASCII Code Macro	473
13.9.4 Variable	478
13.9.5 Types of Variables	480
13.9.5.1 Common Variables	480
13.9.5.2 Local Variables (#1 to #33)	481
13.9.5.3 Macro Interface Inputs/Outputs (#1000 to #1035, #1100 to #1135, #1200 to #1295, #1300 to #1395)	484
13.9.5.4 Tool Compensation	491
13.9.5.5 Workpiece Coordinate System Compensation (#5201 - #532n).....	492
13.9.5.6 NC Alarm (#3000)	493
13.9.5.7 Integrating Time (#3001, #3002)	494
13.9.5.8 Suppression of Single Block Stop and Miscellaneous Function Finish Signal Waiting (#3003)	494
13.9.5.9 Feed Hold, Feedrate Override, G09 Valid/Invalid (#3004)	495
13.9.5.10 Message Display and Stop (#3006)	495
13.9.5.11 Mirror Image (#3007)	496
13.9.5.12 G Command Modals (#4001-#4021, #4201-#4221)	497
13.9.5.13 Other Modals (#4101 - #4120, #4301 - #4320)	498
13.9.5.14 Position Information (#5001 - #5140 + n)	499
13.9.5.15 External Workpiece Coordinate System Compensation (#2501, #2601)	501
13.9.5.16 Number of Workpiece Machining Times (#3901, #3902)	501
13.9.5.17 Tool Life Management (#60000 - #63016)	502
13.9.5.18 Reading The Parameters (#100000-#100002, #100010)	506
13.9.5.19 Reading PLC data (#100100-#100103,#100110)	510
13.9.5.20 Time Reading Variables (#3001, #3002, #3011, #3012)	513
13.9.5.21 R Device Access Variables (#50000 - #50749, #51000 - #51749, #52000 - #52749) ..	515
13.9.6 Operation Commands	521
13.9.7 Control Commands	526
13.9.8 External Output Commands ; POPEN, PCLOS, DPRNT	529
13.9.9 Precautions	533
13.10 Mirror Image for Facing Tool Posts ; G68,G69	535
13.11 Corner Chamfering I/Corner Rounding I	545
13.11.1 Corner Chamfering I ; G01 X_ Z_ ,C_/I_/K_/C_	545
13.11.2 Corner Rounding I ; G01 X_ Z_ ,R_/R_	547
13.11.3 Corner Chamfering Expansion/Corner Rounding Expansion	549
13.11.4 Interrupt during Corner Chamfering/Interrupt during Corner Rounding	551
13.12 Corner Chamfering II / Corner Rounding II	552
13.12.1 Corner Chamfering II ; G01/G02/G03 X_ Z_ ,C_/I_/K_/C_	552
13.12.2 Corner Rounding II ; G01/G02/G03 X_ Z_ ,R_/R_	554
13.12.3 Corner Chamfering Expansion/Corner Rounding Expansion	556
13.12.4 Interrupt during Corner Chamfering/Interrupt during Corner Rounding	556
13.13 Linear Angle Command ; G01 X_/Z_ A_/A_	557
13.14 Geometric	558
13.14.1 Geometric I ; G01 A_	558
13.14.2 Geometric IB	560
13.14.2.1 Geometric IB (Automatic calculation of two-arc contact) ; G02/G03 P_Q_ /R_	561
13.14.2.2 Geometric IB (Automatic calculation of linear - arc intersection) ; G01 A_ , G02/G03 P_Q_ H_	565
13.14.2.3 Geometric IB (Automatic calculation of linear - arc intersection) ; G01 A_ , G02/G03 R_H_	569
13.15 Parameter Input by Program ; G10 L70/L100, G11	573
13.16 Macro Interruption ; M96,M97	575
13.17 Tool Change Position Return ; G30.1 - G30.5	583
13.18 Coordinate Rotation by Program ; G68.1/G69.1	586
13.19 Balance Cut ; G15,G14.....	600
13.20 Timing Synchronization Operation.....	604

13.20.1 Timing Synchronization Operation (! code) ; !L.....	604
13.20.2 Timing Synchronization Operation with Start Point Designated (Type 1) ; G115	607
13.20.3 Timing Synchronization Operation with Start Point Designated (Type 2) ; G116	609
13.20.4 Timing Synchronization Operation Function Using M codes ; M***	611
13.21 Mixed Control (Cross Axis Control) I ; G110.....	615
13.22 Control Axis Superimposition I ; G126	622
13.23 Control Axis Synchronization Across Part System I ; G125	637
13.24 2-part System Simultaneous Thread Cutting Cycle.....	643
13.24.1 2-part System Simultaneous Thread Cutting Cycle Parameter Setting Command ; G76.....	643
13.24.2 2-part System Simultaneous Thread Cutting Cycle I ; G76.1	644
13.24.3 2-part System Simultaneous Thread Cutting Cycle II ; G76.2.....	646
13.25 2-part System Simultaneous Thread Cutting Cycle (MITSUBISHI special format) ; G76.1,G76.2	650
13.26 Axis Name Switch ; G111	652
14 Coordinate System Setting Functions.....	659
14.1 Coordinate Words and Control Axes	660
14.2 Basic Machine, Workpiece and Local Coordinate Systems.....	662
14.3 Machine Zero Point and 2nd Reference Position (Zero point)	663
14.4 Automatic Coordinate System Setting	664
14.5 Basic Machine Coordinate System Selection ; G53	665
14.6 Coordinate System Setting ; G92	666
14.7 Reference Position (Zero point) Return ; G28,G29.....	668
14.8 2nd, 3rd, and 4th Reference Position (Zero point) Return ; G30	672
14.9 Reference Position Check ; G27.....	675
14.10 Workpiece Coordinate System Setting and Offset ; G54 to G59 (G54.1).....	676
14.11 Local Coordinate System Setting ; G52.....	682
14.12 Workpiece Coordinate System Preset ; G92.1	683
14.13 Coordinate System for Rotary Axis.....	688
15 Protection Function	691
15.1 Chuck Barrier/Tailstock Barrier ; G22,G23	692
15.2 Stored Stroke Limit ; G22,G23.....	696
16 Measurement Support Functions	697
16.1 Automatic Tool Length Measurement ; G37	698
16.2 Skip Function ; G31	702
16.3 Multi-step Skip Function 1 ; G31.n, G04.....	707
16.4 Multi-step Skip Function 2 ; G31 P	709
16.5 Speed Change Skip ; G31 Fn	711
16.6 Programmable Current Limitation ; G10 L14 ;	715
Appendix 1 Program Errors	717
Appendix 2 Order of G Function Command Priority	751
Appendix 3 Supplementary Explanation for Incomplete Thread Area of Thread Control	755
Appendix 4 Parameter Input by Program N No. (G10 L50, G11)	761



Control Axes

1.1 Coordinate Words and Control Axes



Function and purpose

In the case of a lathe, axis names (coordinate words) and directions are defined as follows.

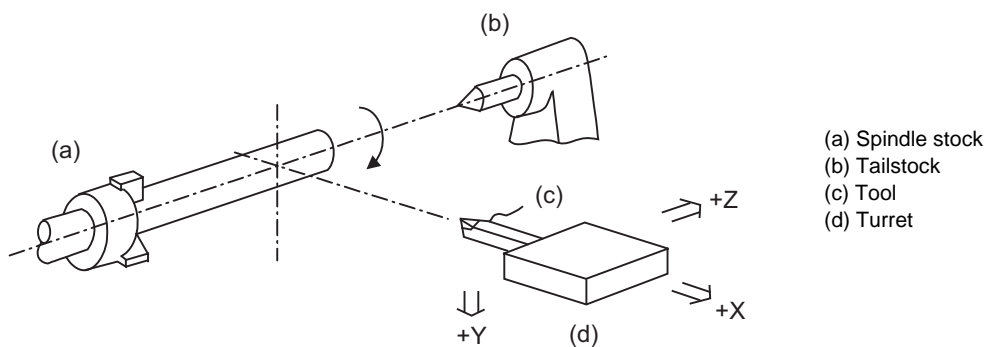
The axis at right angles to the spindle

Axis name: X axis

The axis parallel to the spindle

Axis name: Z axis

Coordinate axes and polarities

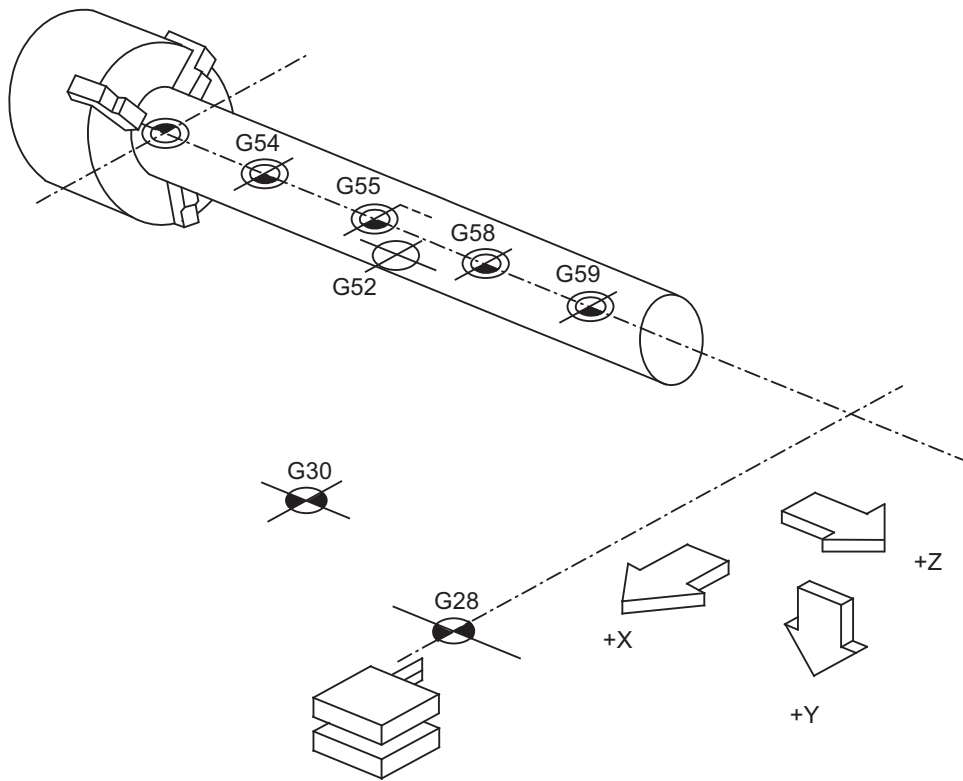






Since coordinates based on the right hand rule are used with a lathe, in the above figure, the positive direction of the Y axis which is at right angles to the X-Z plane is downward.

Note that a circular on the X-Z plane is expressed as clockwise or counterclockwise as seen from the forward direction of the Y axis.

(Refer to the section on circular interpolation.)

Relationship between coordinates



-  Reference position
-  Basic machine coordinate
-  Workpiece coordinate zero points
-  Local coordinate zero point

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

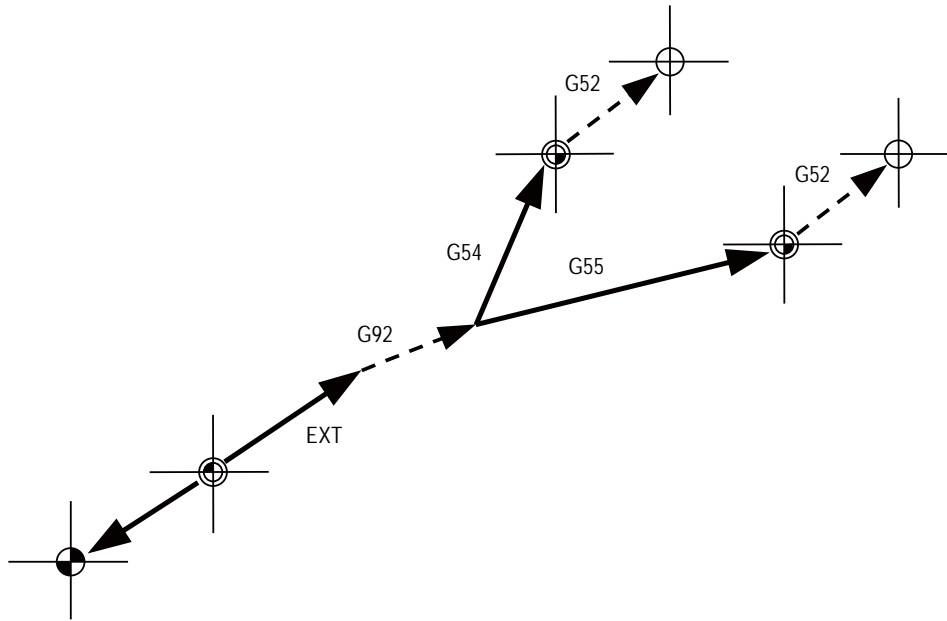
The basic machine coordinate system is the coordinate system that expresses the position (tool change position, stroke end position, etc.) that is specific to the machine.







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 machine manufacturers)

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



-  Reference position
-  Basic machine coordinate zero point
-  Workpiece coordinate zero points
-  Local coordinate zero point
-  Offset set by a parameter
-  Offset set by a program
("0" is set when turning the power ON)

- G52 Local coordinate system offset (*1)
- G54 Workpiece coordinate (G54) system offset (*1)
- G55 Workpiece coordinate (G55) system offset
- G92 G92 Coordinate system shift
- EXT External workpiece coordinate offset

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

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

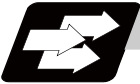
Using the G92 command, the basic machine coordinate system can be shifted and made into a hypothetical machine coordinate system. At the same time, workpiece coordinate systems 1 to 6 are also shifted.



2

Least Command Increments

2.1 Input Setting Unit



Function and purpose

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

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

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



Detailed description

Program command units for each axis and input setting units, common for all axes, are determined by the setting of parameters as follows.

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



Precautions

- (1) Inch/metric changeover can be handled by either a parameter screen (#1041 I_inch: valid only when the power is turned ON) or G commands (G20 or G21).
However, the changeover by a G command applies only to the program command units, and not to the input setting units. Consequently, the tool offset amounts and other compensation amounts as well as the variable data should be preset in order to correspond to input setting units.
- (2) The millimeter and inch systems cannot be used together.
- (3) When performing a circular interpolation between the axes whose program command units are different, the center command (I, J, K) and the radius command (R) are designated by the input setting units. (Use a decimal point to avoid confusion.)

2.2 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.

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



Precautions

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



Program Formats

3.1 Program Format

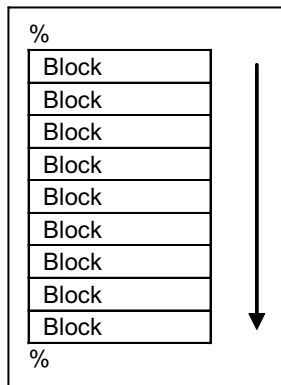
A collection of commands assigned to an NC to move a machine is called "program".

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

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

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

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

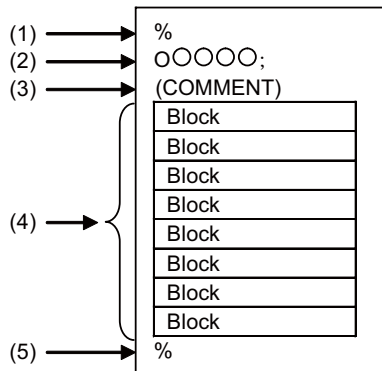




Detailed description

Program

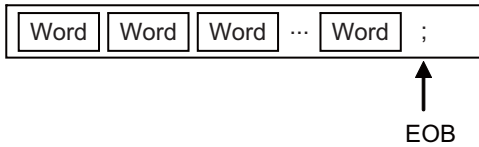
A program format looks as follows.



- (1) Program start
Input an End Of Record (EOR, %) at the head of a program.
It is automatically added when writing a program on an NC. When using an external device, do not forget to input it at the head of a program. For details, refer to the description of the file format.
- (2) Program No.
Program Nos. are used to classify programs by main program unit or subprogram unit. They are designated by the address "O" followed by numbers of up to 8 digits. Program Nos. must be written at the head of programs. A setting is available to prohibit O8000s and O9000s from editing (edit lock). Refer to the instruction manual for the edit lock.
- (3) Comment
Data between control out "(" and control in ")" is ignored.
Information including program names and comments can be written in.
- (4) Program section
A program is a collection of several blocks.
- (5) Program end
Input an end of record (EOR, %) at the end of a program.
It is automatically added when writing a program on an NC.

Block and word

[Block]

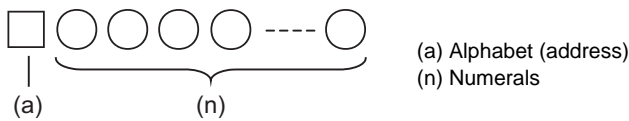


A block is a least command increment, consisting of words.

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

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

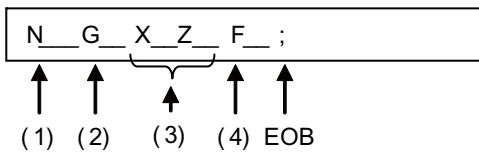
[Word]



A word consists of a set of an alphabet, which is called an address, and numerals (numerical information). Meanings of the numerical information and the number of significant digits of words differ according to an address.

(Note) Leading zeros can be omitted from numerals.

The major contents of a word are described below.



(1) Sequence No.

A "sequence No." consists of the address "N" followed by numbers of up to 6 digits (Normally 3 or 4 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 functions (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.

There are 6 types of G code systems, 2, 3, 4, 5, 6 and 7. Refer to the description of G code system for available G codes.

(3) Coordinate words

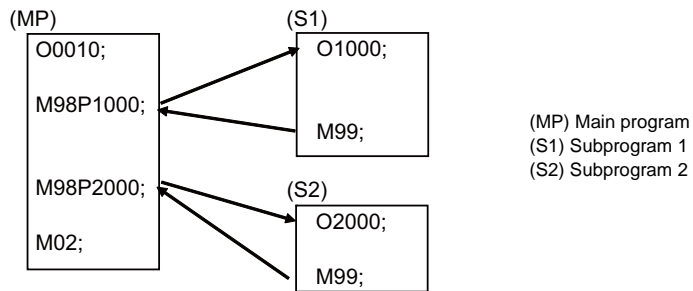
"Coordinate words" specify the coordinate position and movement amounts of tool machine axes. They consist of an address which indicates each axis of a tool machine followed by numerical information (+ or - signs and numerals).

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

(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 the description of subprogram control for the details of the execution of subprograms.

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 I/O device	M700VW	M700VS	M70V
NC memory	○	○	○
HD (internal hard disc)	○	-	-
Serial	○	○	○
Memory card (front IC card)	○	○	○
DS (NC control unit side compact flash)	○	-	-
FD	○	-	-
USB memory	-	○	-
Ethernet	○	○	○

Program file format

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

- (1) NC memory (Creates program on NC)

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

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

(2) External device (except for serials, such as memory card, DS, FD, USB memory)

[Single program]	[Multiple programs]
<pre> CRLF (COMMENT) CRLF G28 XYZ CRLF : : M02 CRLF % ^Z </pre>	<pre> CRLF O100(COMMENT) CRLF G28 XYZ CRLF : : M02 CRLF O101(COMMENT1) CRLF : M02 CRLF % ^Z </pre>

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

(3) External device (serial)

<pre> % LF O100(COMMENT) LF G28 XYZ LF : : M02 LF % </pre>
--

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

3.3 Optional Block Skip

3.3.1 Optional Block Skip; /



Function and purpose

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



Detailed description

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

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

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



Program example

- 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 the optional block skip signal state being 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 ;	

- 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 = not defined at vrble set" 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 G1 X25. /Z25. ;NG (User macro, a division instruction; a program error results.)

/N20 G1 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 No. search.
- (5) All blocks with the "/" code are also input and output during tape storing and tape output, regardless of the position of the optional block skip switch.

3.3.2 Optional Block Skip Addition ; /n



Function and purpose

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



Detailed description

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



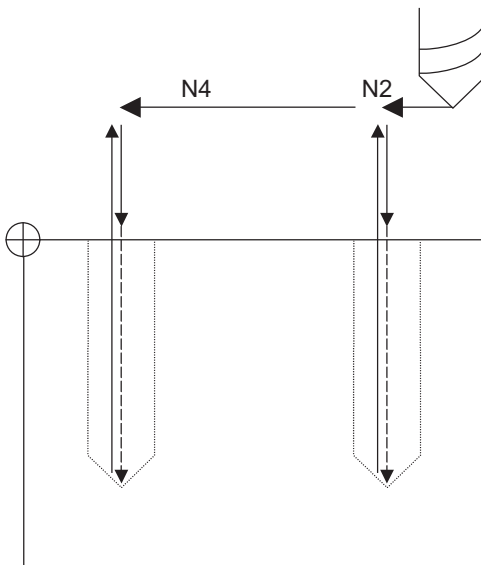
Program example

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

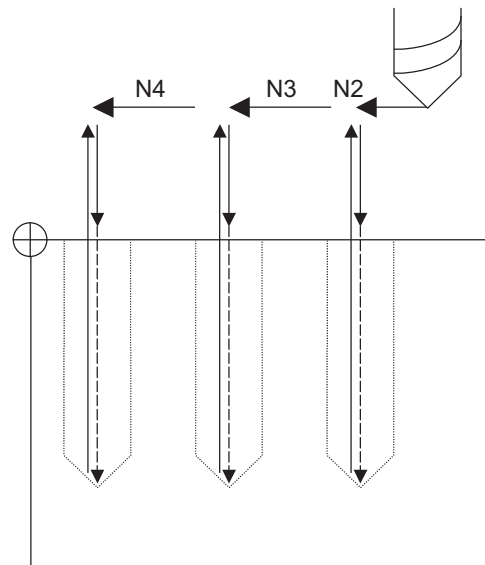
```

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

Part 1
Optional block skip 5 signal ON



Part 2
Optional block skip 5 signal OFF



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

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

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

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

3.4.2 G code Lists

G codes include the six G code lists 2, 3, 4, 5, 6 and 7.

cmdtyp	G code lists
3	2
4	3
5	4
6	5
7	6
8	7

Here, G functions are explained using the G code list 3.

3.4.3 Table of G Code Lists

G code lists						Group	Function	Section
Standard		Special						
2	3	4	5	6	7			
Δ G00	Δ G00	Δ G00	Δ G00	Δ G00	Δ G00	01	Positioning	6.1
Δ G01	Δ G01	Δ G01	Δ G01	Δ G01	Δ G01	01	Linear interpolation	6.2
G02	G02	G02	G02	G02	G02	01	Circular interpolation CW/Helical interpolation CW	6.3 6.4 6.7
G03	G03	G03	G03	G03	G03	01	Circular interpolation CCW/Helical interpolation CCW	6.3 6.4 6.7
G02.3	G02.3	G02.3	G02.3	G02.3	G02.3	01	Exponential interpolation CW	6.11
G03.3	G03.3	G03.3	G03.3	G03.3	G03.3	01	Exponential interpolation CCW	6.11
G04	G04	G04	G04	G04	G04	00	Dwell	8.1
				G07.1 G107	G07.1 G107	19	Cylindrical interpolation	6.9
G09	G09	G09	G09	G09	G09	00	Exact stop check	7.11
G10	G10	G10	G10	G10	G10	00	Programmable compensation data input, Parameter input by program, Tool life management data registration	12.5 13.15 12.6
G11	G11	G11	G11	G11	G11	00	Cancel (Programmable compensation data input, Parameter input by program, Tool life management data registration)	12.5 13.15 12.6
				G12.1 G112	G12.1 G112	19	Polar coordinate interpolation ON	6.10
				G13.1 G113	G13.1 G113	19	Polar coordinate interpolation cancel	6.10
G12.1	G12.1	G12.1	G12.1			19	Milling interpolation ON	6.8
*G13.1	*G13.1	*G13.1	*G13.1			19	Milling interpolation cancel	6.8
*G14	*G14	*G14	*G14			18	● Balance cut OFF	13.18
G15	G15	G15	G15			18	● Balance cut ON	13.18
G16	G16	G16	G16			02	Milling interpolation plane selection Y-Z cylindrical plane	6.8.3
Δ G17	Δ G17	Δ G17	Δ G17	Δ G17	Δ G17	02	Plane selection X-Y	6.5
Δ G18	Δ G18	Δ G18	Δ G18	Δ G18	Δ G18	02	Plane selection Z-X	6.5
Δ G19	Δ G19	Δ G19	Δ G19	Δ G19	Δ G19	02	Plane selection Y-Z	6.5
Δ G20	Δ G20	Δ G20	Δ G20	Δ G20	Δ G20	06	Inch command	5.3
Δ G21	Δ G21	Δ G21	Δ G21	Δ G21	Δ G21	06	Metric command	5.3
G22	G22	G22	G22			04	Barrier check ON	15.1
*G23	*G23	*G23	*G23			04	Barrier check OFF	15.1
				G22	G22	00	Soft limit ON	15.2
				G23	G23	00	Soft limit OFF	15.2
G27	G27	G27	G27	G27	G27	00	Reference position check	14.9
G28	G28	G28	G28	G28	G28	00	Automatic reference position return	14.7
G29	G29	G29	G29	G29	G29	00	Start position return	14.7
G30	G30	G30	G30	G30	G30	00	2nd, 3rd and 4th reference position return	14.8
G30.1	G30.1	G30.1	G30.1	G30.1	G30.1	00	Tool change position return 1	13.17
G30.2	G30.2	G30.2	G30.2			00	Tool change position return 2	13.17
G30.3	G30.3	G30.3	G30.3			00	Tool change position return 3	13.17
G30.4	G30.4	G30.4	G30.4			00	Tool change position return 4	13.17
G30.5	G30.5	G30.5	G30.5			00	Tool change position return 5	13.17

G code lists						Group	Function	Section
Standard		Special						
2	3	4	5	6	7			
G31	G31	G31	G31	G31	G31	00	Skip/Multi-step skip function 2	16.2 16.4
G31.1	G31.1	G31.1	G31.1	G31.1	G31.1	00	Multi-step skip function 1-1	16.3
G31.2	G31.2	G31.2	G31.2	G31.2	G31.2	00	Multi-step skip function 1-2	16.3
G31.3	G31.3	G31.3	G31.3	G31.3	G31.3	00	Multi-step skip function 1-3	16.3
G32	G33	G32	G33	G32	G33	01	Thread cutting	6.6.1 6.6.2
G34	G34	G34	G34	G34	G34	01	Variable lead thread cutting	6.6.4
G35	G35	G35	G35	G35	G35	01	Circular thread cutting CW	6.6.5
G36	G36	G36	G36	G36	G36	01	Circular thread cutting CCW	6.6.5
G37	G37	G36/G37	G36/G37	G36/G37 G37.1 G37.2	G36/G37 G37.1 G37.2	00	Automatic Tool Length Measurement	16.1
*G40	*G40	*G40	*G40	*G40	*G40	07	Tool nose radius compensation cancel	12.4
G41	G41	G41	G41	G41	G41	07	Tool nose radius compensation left	12.4
G42	G42	G42	G42	G42	G42	07	Tool nose radius compensation right	12.4
G46	G46	G46	G46	G46	G46	07	Tool nose radius compensation (automatic direction identification) ON	12.4
G43.1	G43.1	G43.1	G43.1	G43.1	G43.1	20	1st spindle control mode ON	10.10.2
G44.1	G44.1	G44.1	G44.1	G44.1	G44.1	20	Selected spindle control mode	10.10.2
G47.1	G47.1	G47.1	G47.1	G47.1	G47.1	20	All spindles simultaneous control mode ON	10.10.2
G50	G92	G50	G92	G50	G92	00	Spindle Clamp Speed Setting Coordinate system setting	10.3 14.6
*G50.2	*G50.2	*G50.2	*G50.2			11	Scaling cancel	
G51.2	G51.2	G51.2	G51.2			11	Scaling ON	
				G50.2 G250	G50.2 G250	00	Polygon machining mode cancel (spindle-tool axis synchronization)	10.7
				G51.2 G251	G51.2 G251	00	Polygon machining mode ON (spindle-tool axis synchronization)	10.7
G52	G52	G52	G52	G52	G52	00	Local coordinate system setting	14.11
G53	G53	G53	G53	G53	G53	00	Basic Machine Coordinate System Selection	14.5
*G54	*G54	*G54	*G54	*G54	*G54	12	Workpiece coordinate system selection 1	14.10
G55	G55	G55	G55	G55	G55	12	Workpiece coordinate system selection 2	14.10
G56	G56	G56	G56	G56	G56	12	Workpiece coordinate system selection 3	14.10
G57	G57	G57	G57	G57	G57	12	Workpiece coordinate system selection 4	14.10
G58	G58	G58	G58	G58	G58	12	Workpiece coordinate system selection 5	14.10
G59	G59	G59	G59	G59	G59	12	Workpiece coordinate system selection 6	14.10
G54.1	G54.1	G54.1	G54.1	G54.1	G54.1	12	Work coordinate system selection 48 sets expanded	14.10
G61	G61	G61	G61	G61	G61	13	Exact stop check mode	7.12
G62	G62	G62	G62	G62	G62	13	Automatic Corner Override	7.14
G63	G63	G63	G63	G63	G63	13/19	Tapping mode	7.15
*G64	*G64	*G64	*G64	*G64	*G64	13/19	Cutting mode	7.16
G65	G65	G65	G65	G65	G65	00	User macro call	13.9.1
G66	G66	G66	G66	G66	G66	14	User macro modal call A	13.9.1
G66.1	G66.1	G66.1	G66.1	G66.1	G66.1	14	User macro modal call B	13.9.1
*G67	*G67	*G67	*G67	*G67	*G67	14	User macro modal call cancel	13.9.1
G68	G68	G68	G68			15	Mirror image for facing tool posts ON	13.10
G69	G69	G69	G69			15	Mirror image for facing tool posts OFF	13.10

G code lists						Group	Function	Section
Standard		Special						
2	3	4	5	6	7			
				G68	G68	15	Mirror image for facing tool posts ON or balance cut mode ON	13.10
				*G69	*G69	15	Mirror image for facing tool posts OFF or balance cut mode cancel	13.10
G68.1	G68.1	G68.1	G68.1	G68.1	G68.1	16	Coordinate rotation by program ON	13.18
G69.1	G69.1	G69.1	G69.1	G69.1	G69.1	16	Coordinate rotation by program cancel	13.18
G70	G70	G70	G70	G70	G70	09	Finishing cycle	13.3.4
G71	G71	G71	G71	G71	G71	09	Longitudinal rough cutting cycle	13.3.1
G72	G72	G72	G72	G72	G72	09	Face rough cutting cycle	13.3.2
G73	G73	G73	G73	G73	G73	09	Formed material rough cutting cycle	13.3.3
G74	G74	G74	G74	G74	G74	09	Face cut-off cycle	13.3.5
G75	G75	G75	G75	G75	G75	09	Longitudinal cut-off cycle	13.3.6
G76	G76	G76	G76	G76	G76	09	Compound thread cutting cycle ● 2-part system simultaneous thread-cutting cycle parameter setting command	13.3.7 13.24.1
G76.1	G76.1	G76.1	G76.1	G76.1	G76.1	09	● 2-part system synchronous thread-cutting cycle (1)	13.24.2 13.25
G76.2	G76.2	G76.2	G76.2	G76.2	G76.2	09	● 2-part system synchronous thread-cutting cycle (2)	13.24.3 13.25
G90	G77	G90	G77	G90	G77	09	Longitudinal cutting fixed cycle	13.1.1
G92	G78	G92	G78	G92	G78	09	Thread cutting fixed cycle	13.1.2
G94	G79	G94	G79	G94	G79	09	Face cutting fixed cycle	13.1.3
*G80	*G80	*G80	*G80	*G80	*G80	09	Fixed cycle for drilling cancel	13.5 13.5.5 13.6
G81	G81	G81	G81	G81	G81	09	Fixed cycle (drill/spot drill)	13.6 13.6.1
G82	G82	G82	G82	G82	G82	09	Fixed cycle (drill/counter boring)	13.6 13.6.2
G79	G83.2	G79	G83.2	G79	G83.2	09	Deep hole drilling cycle 2	13.5.4
G83	G83	G83	G83	G83	G83	09	Deep hole drilling cycle (Z axis)	13.5 13.5.1
G83.1	G83.1	G83.1	G83.1	G83.1	G83.1	09	Stepping cycle	13.6 13.6.4
G84	G84	G84	G84	G84	G84	09	Tapping cycle (Z axis)	13.5 13.5.2
G85	G85	G85	G85	G85	G85	09	Boring cycle (Z axis)	13.5 13.5.3
G87	G87	G87	G87	G87	G87	09	Deep hole drilling cycle (X axis)	13.5 13.5.1
G88	G88	G88	G88	G88	G88	09	Tapping cycle (X axis)	13.5 13.5.2
G89	G89	G89	G89	G89	G89	09	Boring cycle (X axis)	13.5 13.5.3
G84.1	G84.1	G84.1	G84.1	G84.1	G84.1	09	Reverse tapping cycle (Z axis)	13.5.2
G84.2	G84.2	G84.2	G84.2	G84.2	G84.2	09	Synchronous tapping cycle	13.6 13.6.6
G88.1	G88.1	G88.1	G88.1	G88.1	G88.1	09	Reverse tapping cycle (X axis)	13.5.2
G50.3	G92.1	G50.3	G92.1	G50.3	G92.1	00	Workpiece coordinate preset	14.12
Δ G96	Δ G96	Δ G96	Δ G96	Δ G96	Δ G96	17	Constant surface speed control ON	10.2
Δ G97	Δ G97	Δ G97	Δ G97	Δ G97	Δ G97	17	Constant surface speed control OFF	10.2

G code lists						Group	Function	Section
Standard		Special						
2	3	4	5	6	7			
Δ G98	Δ G94	Δ G98	Δ G94	Δ G98	Δ G94	05	Feed per minute (asynchronous feed)	7.4
Δ G99	Δ G95	Δ G99	Δ G95	Δ G99	Δ G95	05	Feed per revolution (synchronous feed)	7.4
-	Δ G90	-	Δ G90	-	Δ G90	03	Absolute value command	5.1
-	Δ G91	-	Δ G91	-	Δ G91	03	Incremental value command	5.1
-	*G98	-	*G98	-	*G98	10	Fixed cycle initial level return	13.5 13.5.7
-	G99	-	G99	-	G99	10	Fixed cycle R point level return	13.5 13.5.7
G110	G110	G110	G110			00	● Mixed control (cross axis control) I	13.21
G111	G111	G111	G111	G111	G111	00	Axis name switch	13.26
G113	G113	G113	G113			00	Spindle synchronization cancel /Polygon machining (spindle-spindle synchronization) mode cancel	10.5 10.6
G114.1	G114.1	G114.1	G114.1			00	Spindle synchronization	10.5.1
G114.2	G114.2	G114.2	G114.2			00	Polygon machining (spindle-spindle synchronization) mode ON	10.6
G114.3	G114.3	G114.3	G114.3			00	Tool spindle synchronization II (Hobbing) ON	10.9
G115	G115	G115	G115	G115	G115	00	● Start point designation synchronization Type 1	13.20.2
G116	G116	G116	G116	G116	G116	00	● Start point designation synchronization Type 2	13.20.3
G117	G117	G117	G117	G117	G117	00	● M code output during axis traveling	9.4
G125	G125	G125	G125			00	● Control axis synchronization across part systems	13.23
G126	G126	G126	G126			00	● Control axis superimposition	13.22



Precautions

- (1) A program error (P34) will occur if a G code unlisted on the Table of G code lists is commanded.
- (2) An alarm will occur if a G code without additional specifications is commanded.
- (3) A (*) symbol indicates the G code to be selected in each group when the power is turned ON or when a reset is executed to initialize the modal.
- (4) A (Δ) symbol indicates the G code for which parameters selection is possible as an initialization status when the power is turned ON or when a reset is executed to initialize the modal. Note that inch/metric changeover can only be selected when the power is turned ON.
- (5) A (●) symbol indicates a function dedicated for multi-part system.
- (6) If two or more G codes from the same group are commanded in a block, the last G code will be valid.
- (7) 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 tool builder.
- (8) 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.
 - "Reset 2 "and "Reset and 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 point return.
 - Conforms to "Reset & rewind".

Precautions for G code lists 6 and 7

- (1) G68,G69

When both the mirror image for facing tool posts option and balance cut option are valid, G68 and G69 will be handled as the command to turn the mirror image for facing tool posts ON/OFF. (The mirror image for facing tool posts has the priority.)
- (2) G36

G36 is used for the two functions, automatic tool length measurement and circular thread cutting (CCW). Which function to be applied depend on the setting of the parameter "#1238 set10/bit0" (circular thread cutting).

When "#1238 set10/bit0" is set to "0"

G code	Function
G35	Circular thread cutting clockwise rotation (CW)
G36	Automatic tool length measurement X
G37	Automatic tool length measurement Z

When "#1238 set10/bit0" is set to "1"

G code	Function
G35	Circular thread cutting clockwise rotation (CW)
G36	Circular thread cutting counterclockwise rotation (CCW)
G37	Automatic tool length measurement Z

CAUTION

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

3.5 Precautions Before Starting Machining

CAUTION

1. 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.
2. 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 offset amount and etc.

4

Pre-read Buffers

4.1 Pre-read Buffers



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 nose 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.)



Precautions

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

Position Commands

5.1 Absolute/Incremental Value Commands ; G90,G91

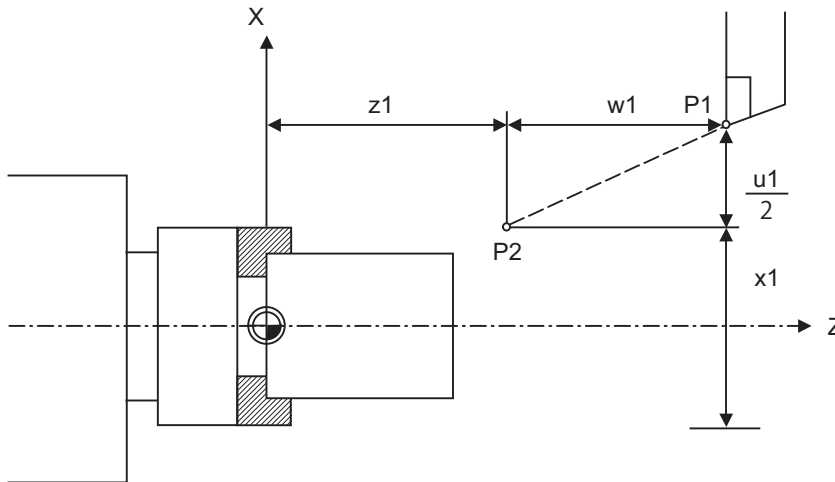


Function and purpose

There are two methods of issuing tool movement amount commands: the absolute value method and the incremental value method.

To designate the coordinates of a point to move, the absolute value method issues a command using the distance from the coordinate zero point; on the other hand, the incremental value method issues a command using the distance from the present point.

Use an axis address or a G command to choose between the absolute or incremental value command. Whether an axis address or a G command is valid can be selected according to the parameter setting. The following figure shows what happens when the tool is moved from point P1 to point P2.



- (1) Movement command by an axis address (when "#1076 AbsInc" is "1")
 Absolute value command: G00 Xx1 Zz1 ;
 Incremental value command: G00 Uu1 Ww1 ;
- (2) Movement command by a G command (when "#1076 AbsInc" is "0")
 Absolute value command: G90 G00 Xx1 Zz1 ;
 Incremental value command: G91 G00 Xu1 Zw1 ;



Command format

G90; ... Absolute value command

G91; ... Incremental value command

When the parameter "#1076 AbsInc" is set to "0", a G command selects either the incremental or absolute value commands.

After commanding G90/ G91, coordinates will be commanded with incremental or absolute values.



Detailed description

Selection between incremental or absolute value commands by an axis address

When the parameter "#1076 AbsInc" is set to "1", an axis address selects either the incremental or absolute value commands.

- (1) Set correspondence between addresses and axes with the following parameters.

#1013 axname

#1014 incax

Following table shows the example of when "X,Z,C,Y" are set to "#1013 axname" and "U,W,H,V" are set to "#1014 incax".

		Command method
Absolute value	X axis	Address X
	Z axis	Address Z
	C/Y axis	Address C/Y
Incremental value	X axis	Address U
	Z axis	Address W
	C/Y axis	Address H/V

(Note 1) The C/Y axis is an example of additional axes.

- (2) Absolute and incremental values can be used together in the same block.

(Example) X__ W__ ; ... An absolute value command for X axis and an incremental value command for Z axis



Precautions

- (1) Designation of a circular radius (R) and center (I,J,K) is always conducted with incremental values.
- (2) When parameter "#1076 AbsInc" is 1, and H is used for the incremental command address, address H of blocks in M98, G114.2 will be handled as the parameter of command, and the axis will not be moved.

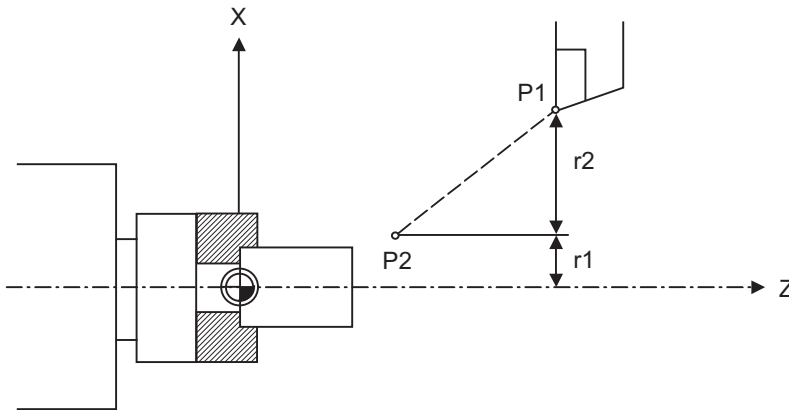
5.2 Radius/Diameter Designation



Function and purpose

On a lathe, a workpiece rotates, so its coordinate positions, dimensions, and commands can be designated by radius/ diameter values. Commands using diameter values are called diameter designation, and commands with radius values are called radius designation.

Either radius or diameter designation can be used depending on the setting of the parameter (#1019 dia). The figure below shows the command procedure when the tool is to be moved from point P1 to point P2.



X command		U command		Remarks
Radius	Diameter	Radius	Diameter	
$X = r1$	$X = 2 * r1$	$U = r2$	$U = 2 * r2$	Even when a diameter designation has been selected, the U command can exclusively be changed into a radius designation by the parameter "#1077 radius". (Note) "U" is an incremental command address.



Precautions and restrictions

- (1) In the above example, the tool moves from P1 to P2 in the minus direction of the X axis. So when this is using incremental value command, the minus sign is given to the numerical value being commanded.
- (2) In this manual, diameter commands are used in descriptions of both the X and U axes for the sake of convenience.

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

G20; ... Inch command

G21; ... Metric command



Detailed description

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

Output unit, command unit and setting unit

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

NC axis

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

PLC axis

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



Precautions

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

5.4 Decimal Point Input



Function and purpose

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

Use the parameter "#1078 Decpt2" to select whether minimum input command increment (type I) or zero point (type II) to apply to the least significant digit of data without a decimal point.



Detailed description

- (1) The decimal point command is valid for the distances, angles, times and speeds in machining programs.
- (2) Refer to the table "Addresses used, validity of decimal point commands" for details on the valid addresses for the decimal point commands.
- (3) In decimal point command, the valid range of command value is as shown below. (for input command increment cunit=10)

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

- (4) The decimal point command is valid even for commands defining the variable data used in subprograms.
- (5) Decimal point commands for decimal point invalid addresses are processed as an integer only data which everything below the decimal point is ignored. Decimal point invalid addresses include the followings; D,H,L,M,N,O,P,S,T.
All variable commands, however, are treated as data with decimal points.

Decimal point input I, II and decimal point command validity

Decimal point input I and II will result as follows when decimal points are not used in an address which a decimal point command is valid. Whether an address is valid or invalid for the decimal point command is shown in the table below.

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

- (1) Decimal point input I
The least significant digit of command data matches the command unit.
(Example) When "X1" is commanded in 1 μ m system, the same result occurs as for an "X0.001" command.
- (2) Decimal point input II
The least significant digit of command data matches the command unit.
(Example) When "X1" is commanded in 1 μ m system, the same result occurs as for an "X1." command.

-Addresses used, validity of decimal point commands-

Address	Decimal Point Command	Application	Remarks
A	Valid	Coordinate position data	
	Invalid	2nd miscellaneous function code	
	Valid	Angle data	
	Invalid	MRC program No.	
	Invalid	Parameter input by program Axis No.	
	Valid	Deep hole drilling cycle (2) Safety distance	
	Valid	Spindle synchronous acceleration/deceleration time constant	
B	Valid	Coordinate position data	
	Invalid	2nd miscellaneous function code	
C	Valid	Coordinate position data	
	Invalid	2nd miscellaneous function code	
	Valid	Corner chamfering amount	,C
	Valid	Tool compensation data input by program Nose R compensation amount (incremental)	
	Valid	Chamfering width (slitting cycle)	
D	Valid	Automatic tool length measurement, deceleration range d	
	Invalid	Parameter input by program byte type data	
	Invalid	Synchronous spindle No. at spindle synchronization	
E	Valid	Inch threads, Precision thread lead	
	Valid	Corner cutting feedrate	
F	Valid	Feedrate	
	Valid	Thread lead	
G	Valid	Preparatory function code	
H	Valid	Coordinate position data	
	Invalid	Sequence Nos. in subprograms	
	Invalid	Parameter input by program bit type data	
	Invalid	Selection of linear - arc intersection (geometric)	
	Invalid	Basic spindle No. at spindle synchronization	
I	Valid	Circular center coordinates	
	Valid	Nose R compensation/ tool radius compensation vector components	
	Valid	Deep hole drilling cycle (2) First cut amount	
	Valid	G0/G1 in-position width Hole drilling cycle G0 in-position width	,I
J	Valid	Circular center coordinates	
	Valid	Nose R compensation/ tool radius compensation vector components	
	Invalid	Deep hole drilling cycle (2) Dwell at return point	
	Valid	Hole drilling cycle G1 in-position width	,J
K	Valid	Circular center coordinates	
	Valid	Nose R compensation/ tool radius compensation vector components	
	Invalid	Hole drilling cycle Number of repetitions	
	Valid	Deep hole drilling cycle (2) Second and subsequent cut amounts	
	Valid	Thread lead increase/decrease amount (variable lead thread cutting)	

Address	Decimal Point Command	Application	Remarks
L	Invalid	Subprogram Number of repetitions	
	Invalid	Tool compensation data input by program Type selection	L2 L10 L11
	Invalid	Parameter input by program selection	L70
	Invalid	Parameter input by program two-word type data	4byte
	Invalid	Timing synchronization	
	Invalid	Tool life data	
M	Invalid	Miscellaneous function codes	
N	Invalid	Sequence Nos.	
	Invalid	Parameter input by program data No.	
O	Invalid	Program No.	
P	Invalid	Dwell time	
	Invalid	Subprogram call Program No.	
	Invalid	2nd, 3rd and 4th reference position No.	
	Invalid	Constant surface speed control, axis No.	
	Invalid	MRC finishing shape start sequence No.	
	Valid	Cut-off cycle shift amount/cut amount	
	Invalid	Compound thread cutting cycle number of cutting passes, chamfering, tool nose angle	
	Valid	Compound thread cutting cycle Thread height	
	Invalid	Tool compensation data input by program input compensation No.	
	Invalid	Parameter input by program section No.	
	Valid	Coordinate position data	
	Invalid	Skip signal command	
	Valid	Arc center coordinates (absolute value) (geometric)	
	Invalid	Subprogram return destination sequence No.	
	Invalid	Extended workpiece coordinate system No.	
Invalid	Tool life data group No.		
Q	Invalid	Minimum spindle clamp rotation speed	
	Invalid	MRC finishing shape end	
	Valid	Cut-off cycle Cut amount/shift amount	
	Valid	Compound thread cutting cycle Minimum cut amount	
	Valid	Compound thread cutting cycle First cut amount	
	Valid	Deep hole drilling cycle 1 Cut amount of each pass	
	Invalid	Tool compensation data input by program Hypothetical tool nose point No.	
	Invalid	Deep hole drilling cycle (2) Dwell at cut point	
	Valid	Arc center coordinates (absolute value) (geometric)	
	Valid	Thread cutting start shift angle	
	Invalid	Tool life data management method	

Address	Decimal Point Command	Application	Remarks
R	Valid	R-designated arc radius	
	Valid	Arc radius of corner rounding	,R
	Valid	Automatic tool length measurement, deceleration range r	
	Valid	MRC longitudinal/face escape amount	
	Invalid	MRC shaping division No.	
	Valid	Cut-off cycle, return amount	
	Valid	Cut-off cycle, escape amount	
	Valid	Compound thread cutting cycle Finishing allowance	
	Valid	Compound thread cutting cycle turning cycle, taper difference	
	Valid	Hole drilling cycle/deep hole drilling cycle (2) Distance to R point	
	Valid	Tool compensation data input by program Nose R compensation	
	Valid	Coordinate position data	
	Valid	Rough cutting cycle (longitudinal) (face) pull amount	
	Invalid	Synchronous tap/ asynchronous tap changeover	,R
	Valid	Synchronous spindle phase shift amount	
	Valid	Rotation angle	
	Invalid	Skip acceleration/deceleration time constant	
S	Invalid	Spindle function codes	
	Invalid	Maximum spindle clamp rotation speed	
	Invalid	Constant surface speed control, surface speed	
	Invalid	Parameter input by program word type data	2byte
T	Invalid	Tool function codes	
U	Valid	Coordinate position data	
	Valid	Tool compensation data input by program	
	Valid	Rough cutting cycle (longitudinal) Cut amount	
	Valid	Dwell	
V	Valid	Coordinate position data	
	Valid	Tool compensation data input by program	
W	Valid	Coordinate position data	
	Valid	Tool compensation data input by program	
	Valid	Rough cutting cycle (longitudinal) Cut amount	
X	Valid	Coordinate position data	
	Valid	Dwell	
	Valid	Tool compensation data input by program	
Y	Valid	Coordinate position data	
	Valid	Tool compensation data input by program	
Z	Valid	Coordinate position data	
	Valid	Tool compensation data input by program	

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



Program example

(1) Program example of decimal point valid address

Program example	Decimal point command 1		Decimal point command 2 When 1 = 1mm
	When 1 = 1 μ m	When 1 = 10 μ m	
G0 X123.45 (decimal points are all mm points)	X123.450 mm	X123.450 mm	X123.450 mm
G0 X12345	X12.345 mm (last digit is 1 μ m 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.

(Example1) G00 X123+0 ;

This is the X axis 123mm command. It will not be 123 μ m.



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

G00 X_/U_ Z_/W_ ,I_ ; ... Positioning (Rapid Traverse)

X_/U_	X axis end point coordinate (X is the absolute value of workpiece coordinate system, U is the incremental value from present position)
Z_/W_	Z axis end point coordinate (Z is the absolute value of workpiece coordinate system, W is the incremental value from present position)
,I	In-position width. This is valid only in the commanded block. A block that does not contain this address will follow the parameter "#1193 inpos" settings. 1 to 999999

The command addresses are valid for all additional axes.



Detailed description

- (1) Positioning will be performed at the rapid traverse rate set in the 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 (G83 to G89) in the 09 group are cancelled (G80) by the G00 command.

CAUTION

1. 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 can be selected by the parameter "#1086 G0Intp". The positioning time does not change according to the path.

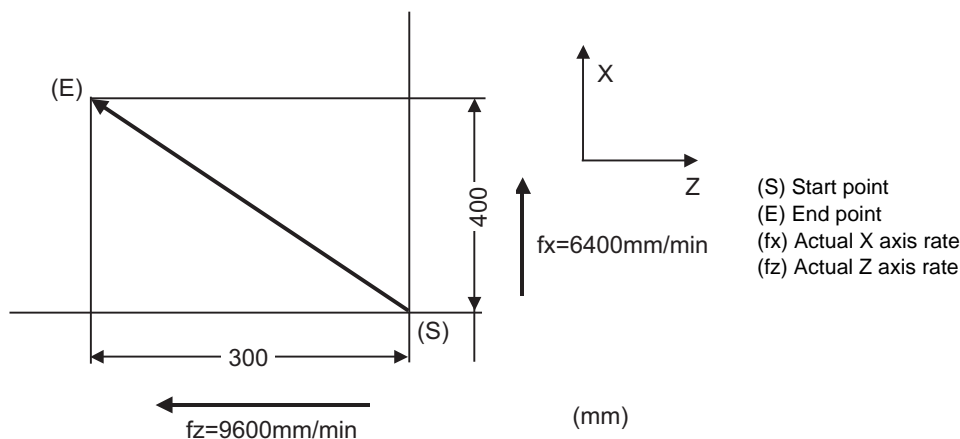
- (1) Linear path: When the parameter "#1086 G0Intp" is set to "0"

In positioning, a tool follows the shortest path which 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 Z-axis rapid traverse rates are both 9600mm/min;

G00 Z-300000 X400000 ; (With an input setting unit of 0.001mm)

The tool will follow the path shown in the figure below.



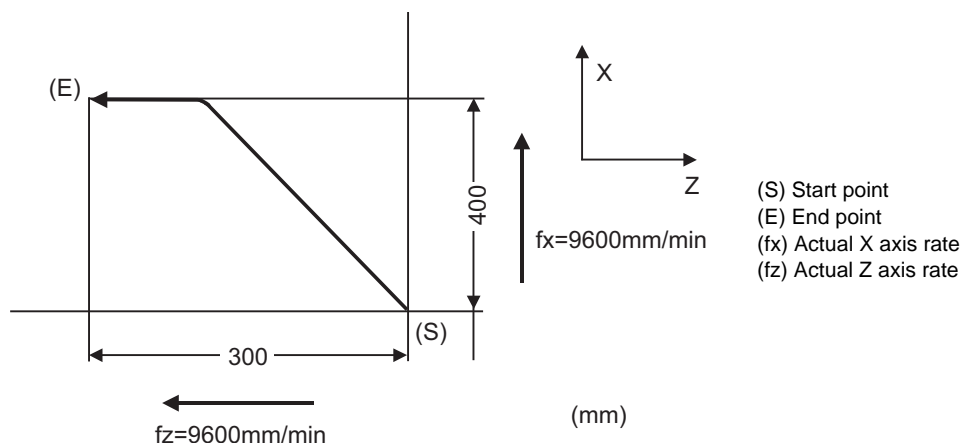
- (2) Non-linear path: When the parameter "#1086 G0Intp" is set to "1"

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 Z-axis rapid traverse rates are both 9600mm/min;

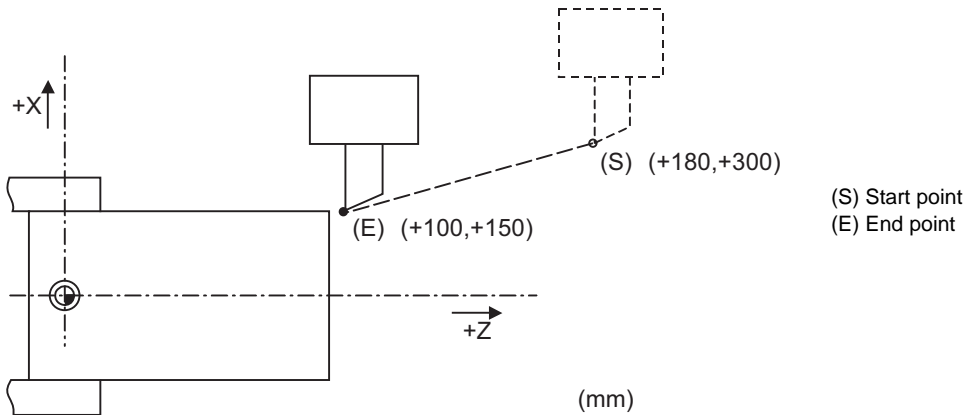
G00 Z-300000 X400000 ; (With an input setting unit of 0.001mm)

The tool will follow the path shown in the figure below.





Program example



G00 X100. Z150. ;	Absolute value command
G00 U-80. W-150. ;	Incremental value command



Precautions for deceleration check

There are two methods for the deceleration check; commanded deceleration method and in-position check method. Select a method with the parameter "#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)

The deceleration check method set in basic specification parameter "#1193 inpos" is used for blocks that do not have the in-position width command.

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 which 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 which 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")	

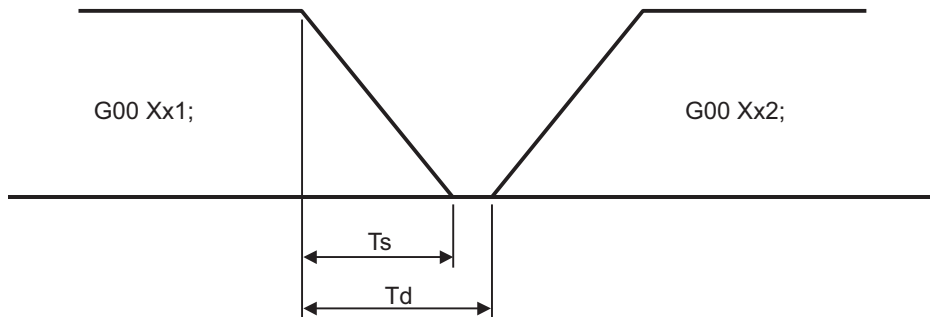
* Following descriptions are for the case of rapid traverse. For G01, interpret the parameters into suitable ones.

Commanded deceleration method when "inpos" = "0"

Upon completion of the rapid traverse (G00), the next block will be executed after the deceleration check time (Td) has elapsed.

The deceleration check time (Td) is as follows, depending on the acceleration/deceleration type set in the parameter "#2003 smgst".

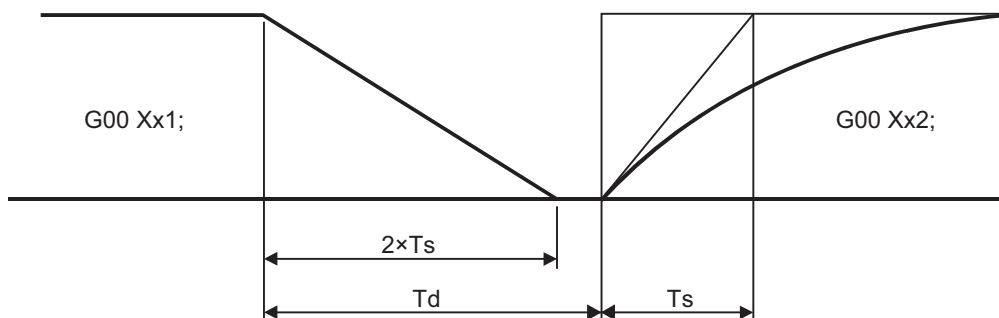
(1) Linear acceleration/linear deceleration



(Ts) Acceleration/deceleration time constant

(Td) Deceleration check time: $T_d = T_s + (0 \text{ to } 7\text{ms})$

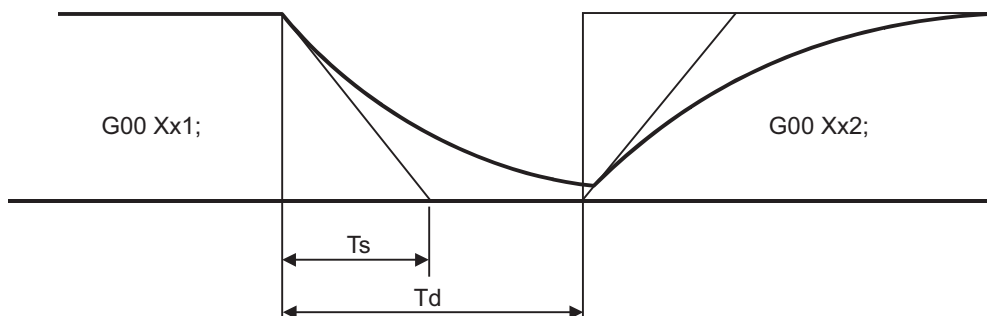
(2) Exponential acceleration/linear deceleration



(Ts) Acceleration/deceleration time constant

(Td) Deceleration check time: $T_d = 2 \times T_s + (0 \text{ to } 7\text{ms})$

(3) Exponential acceleration/exponential deceleration (Primary delay)



(Ts) Acceleration/deceleration time constant

(Td) Deceleration check time: $T_d = 2 \times T_s + (0 \text{ to } 7\text{ms})$

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.

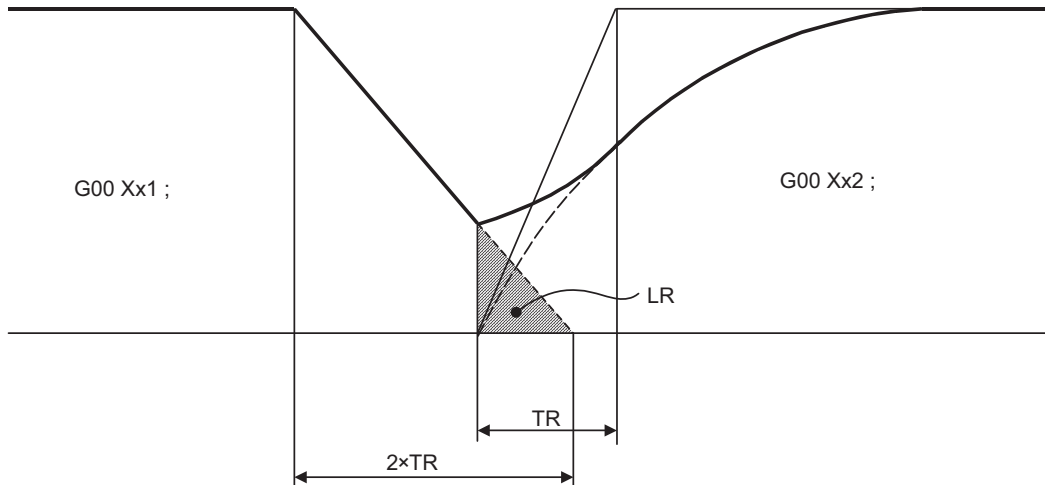
In-position check method when "inpos" = 1

Upon completion of the rapid traverse (G00), the next block will be executed after confirming that the remaining distances for each axis are below the fixed amounts.

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.

G0 deceleration pattern



(TR) Rapid traverse acceleration and deceleration time constant

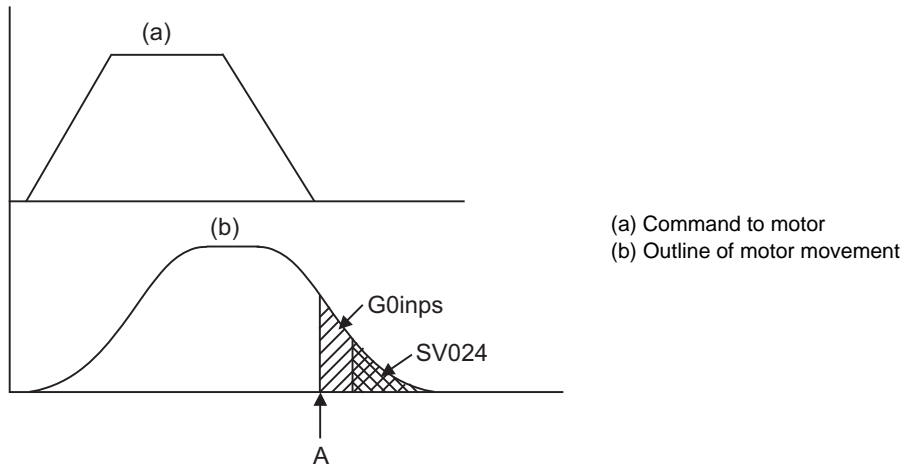
(LR) In-position width

The in-position width LR indicates the remaining distance from the previous block at the start of the next block (shaded area of the figure above).

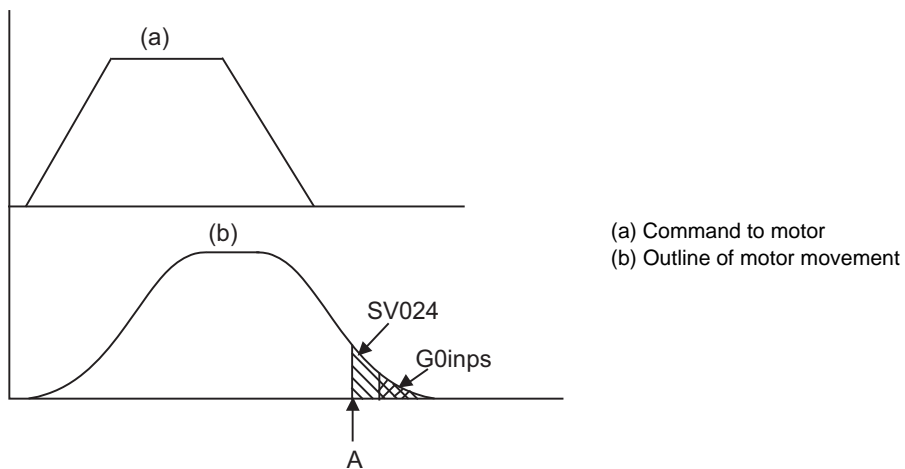
The purpose of the rapid traverse deceleration check is to minimize the positioning time. The bigger the setting value for the in-position width, the shorter the time is, but the remaining distance of the previous block at the start of the next block also becomes larger, and this could become an obstacle 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 using 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_ ,I_ ;	
X,Z	Positioning coordinate value of each axis
,I	In-position width

Execution of the next block starts after confirming that the position error amount of the positioning (rapid traverse: G00) command block is less than the in-position width issued in this command.

The bigger one of in-position width (SV024, G0inps (For G01, G1inps)) with parameter or in-position width specified by program will be adapted as the in-position width.

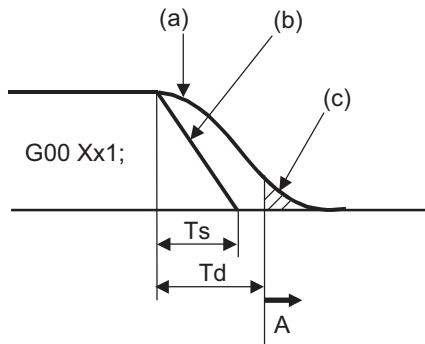
When there are several movement axes, the system confirms that the position error amount of each movement axis in each part system is less than the in-position width issued in this command before executing the next block.

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

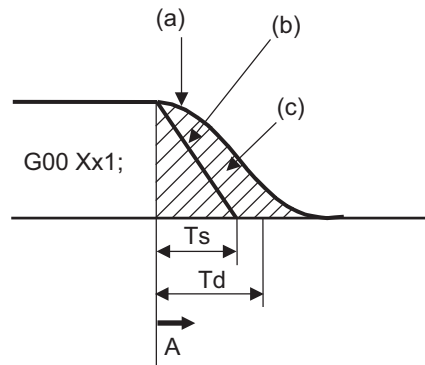
After completing deceleration of the command system (A), the servo system's position error amount and the parameter setting value (in-position width) are compared.



- (a) Servo machine position
- (b) Command
- (c) In-position width (Servo system position error amount)
- (Ts) Acceleration/deceleration time constant
- (Td) Deceleration check time: $T_d = T_s + (0 \text{ to } 7\text{ms})$

(2) In-position check with programmable command ("I" address command)

After starting deceleration of the command system (A), the position error amount and commanded in-position width are compared.



- (a) Servo machine position
- (b) Command
- (c) In-position width (Error amount between command end point and machine position)
- (Ts) Acceleration/deceleration time constant
- (Td) Deceleration check time: $T_d = T_s + (0 \text{ to } 7\text{ms})$

6.2 Linear Interpolation ; G01



Function and purpose

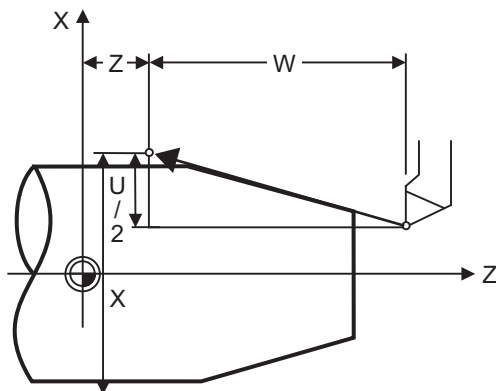
This command is accompanied by coordinate words and a feedrate command. It makes the tool move (interpolate) linearly from its current position to the end point specified by the coordinate words at the speed specified by address F. In this case, the feedrate specified by address F always acts as a linear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



Command format

G01 X_ /U_ Z_ /W_ α _ F_ ,I_ ; ... Linear interpolation

X,U,Z,W, α	Coordinate values (α is the additional axis.)
F	Feedrate (mm/min or ° /min)
,I	In-position width. This is valid only in the commanded block. A block that does not contain this address will follow the parameter "#1193 inpos" settings. 1 to 999999



Detailed description

- (1) G01 command is a modal command in the 01 group. When G01 command is issued in succession, it can only be issued with coordinate words in subsequent blocks.
- (2) The feedrate for a rotary axis is commanded by °/min (decimal point position unit). (F300=300°/min)
- (3) The G functions (G70 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_ Z_ F_ ,I_ ;	
X,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.

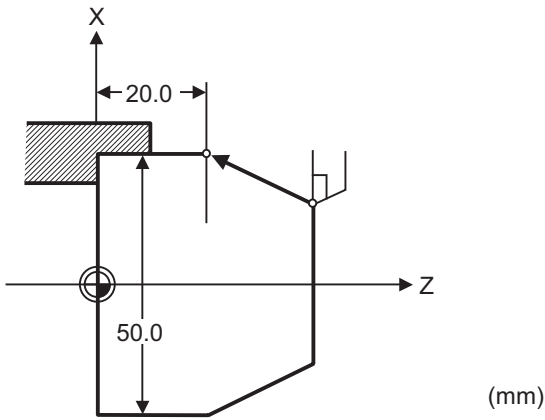
- When the error detection switch is ON.
- When G09 (exact stop check) is commanded in the same block.
- When G61 (exact stop check mode) is selected.

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



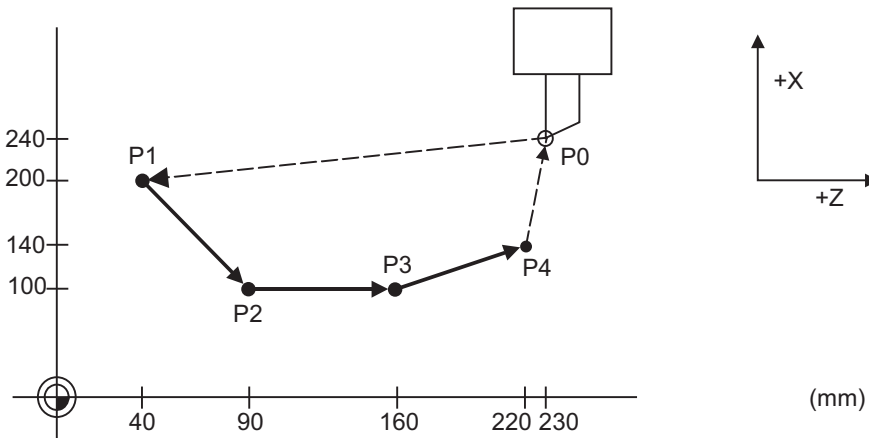
Program example

(Example 1)



G01 X50.0 Z20.0 F300 ;	
------------------------	--

(Example 2) Cutting in the sequence of P1 -> P2 -> P3 -> P4 at 300mm/min feedrate. However, P0 -> P1 and P4 -> P0 are for tool positioning.



G00 X200. Z40. ;	P0 -> P1
G01 X100. Z90. F300 ;	P1 -> P2
Z160. ;	P2 -> P3
X140. Z220. ;	P3 -> P4
G00 X240. Z230. ;	P4 -> P0

6.3 Circular Interpolation ; G02,G03



Function and purpose

These commands serve to move the tool along a circular.



Command format

G02 X__/U__ Z__/W__ I__ K__ F__ ; ... Circular interpolation : Clockwise (CW)

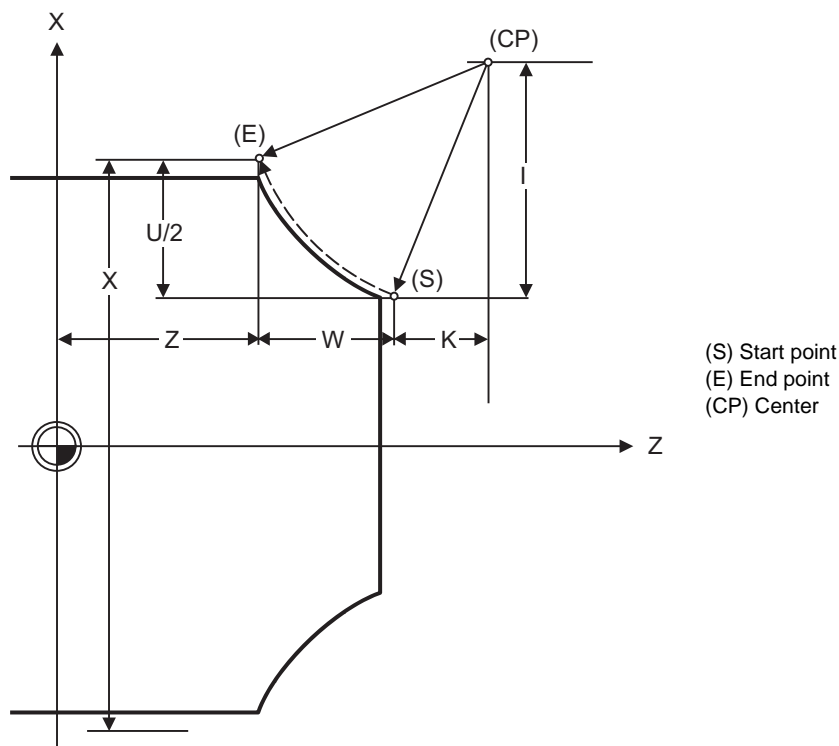
G03 X__/U__ Z__/W__ I__ K__ F__ ; ... Circular interpolation : Counterclockwise (CCW)

X/U	Circular end point coordinates, X axis (X is the absolute value of workpiece coordinate system, U is the incremental value from present position)
Z/W	Circular end point coordinates, Z axis (Z is the absolute value of workpiece coordinate system, W is the incremental value from present position)
I	Circular center, X axis (I is the radius command incremental value of X coordinate at the center as seen from the start point.)
K	Circular center, Z axis (K is the incremental value of Z coordinate at the center as seen from the start point.)
F	Feedrate

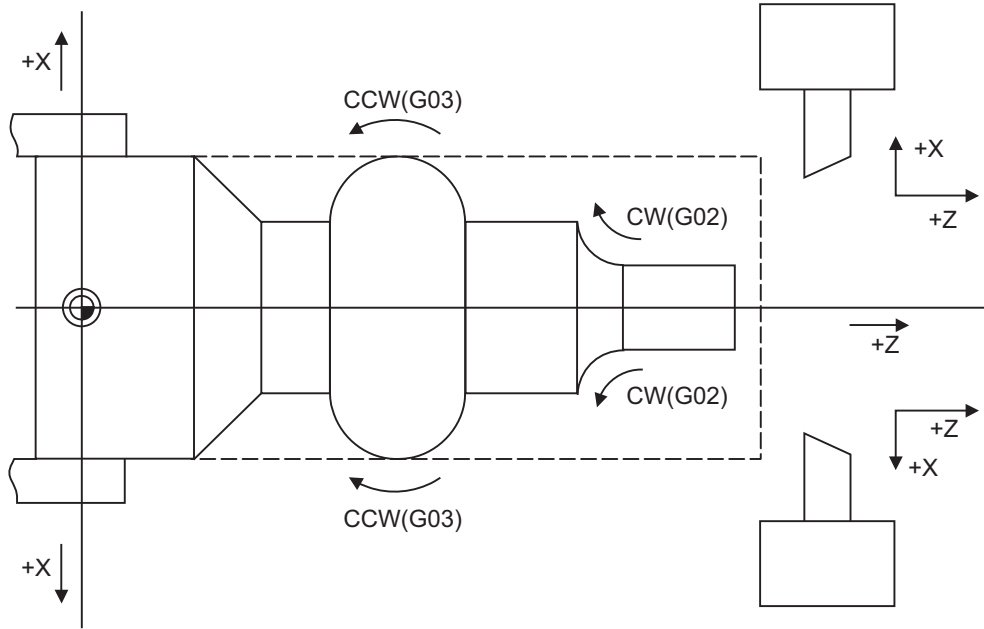


Detailed description

- (1) The arc center coordinate value is commanded with a program command unit. Caution is required for the circular command of an axis for which the program command unit (#1015 cunit) 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 Clockwise (CW)
 G03 Counterclockwise (CCW)



- (3) An arc which extends for more than one quadrant can be executed with a single block command.
- (4) The following information is needed for circular interpolation.
 - (a) Rotation direction : Clockwise (G02) or counterclockwise (G03)
 - (b) Circular end point coordinates : Given by addresses X, Z, U, W
 - (c) Circular center coordinates : Given by addresses I, K (incremental value commands)
 - (d) Feedrate : Given by address F
- (5) A program error results when I, K or R is not commanded. Consideration must be given to the sign for I and K since I is the distance in the X-axis direction to the arc center from the start point and K in the Z-axis direction.
- (6) No T commands can be issued in the G2/G3 modal status. A program error (P151) will occur if a T command is issued in the G2/G3 modal status.

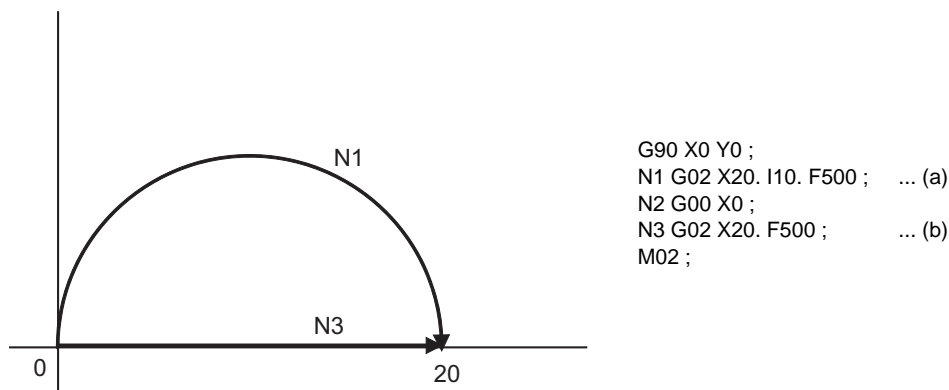
Change into linear interpolation command

Program error (P33) will occur when the center and radius are not designated at circular command.

When the parameter "#11029 Arc to G1 no Cent (Change command from arc to linear when no arc center designation)" is set, the linear interpolation can be operated up to the terminal coordinate value only for that block. However, a modal is the circular modal.

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

(Example) The parameter "#11029 Arc to G1 no Cent (Change command from arc to linear when no arc center designation)" = "1"

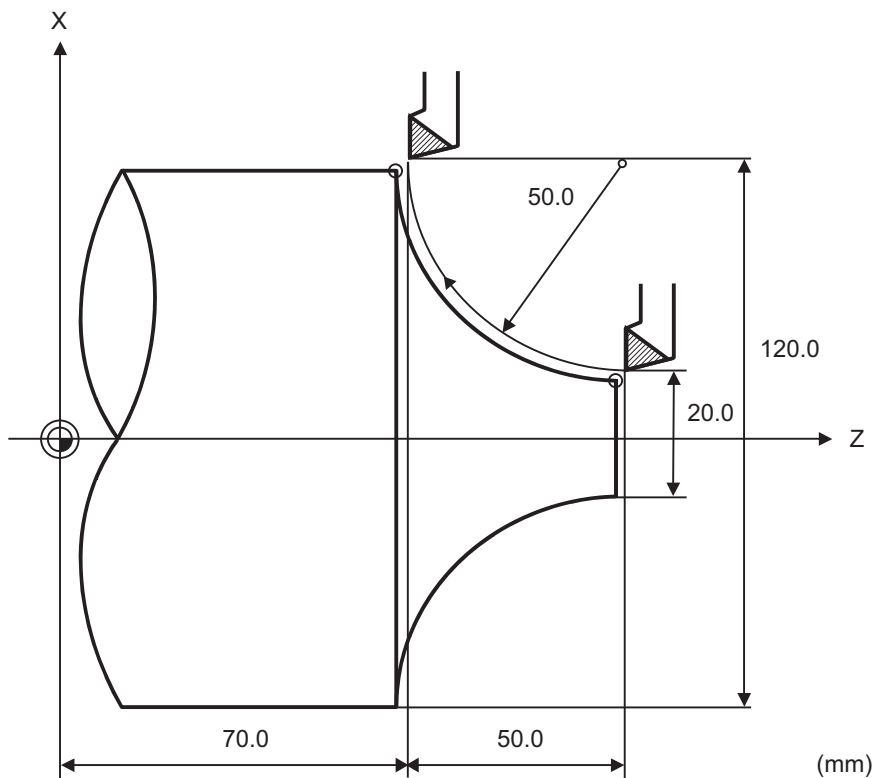


(a) The circular interpolation (G02) is executed because there is a center command.

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



Program example

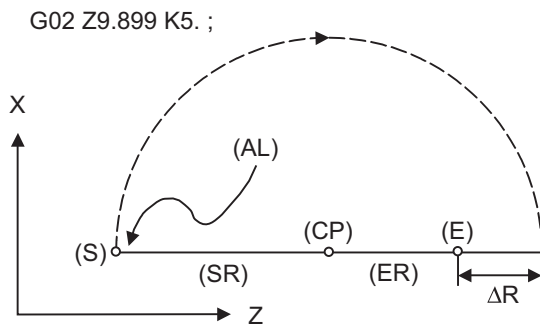


G2 X120.0 Z70.0 I50.0 F200 ;	Absolute value command
G2 U100.0 W-50.0 I50.0 F200 ;	Incremental value command



Precautions

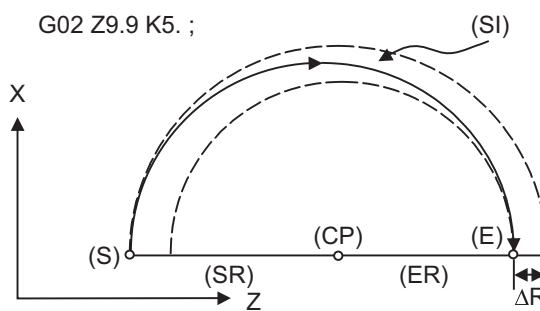
- (1) The terms "clockwise" (G02) and "counterclockwise" (G03) used for circular operations are defined as a case where, in a right-hand coordinate system, the negative direction is viewed from the positive direction of the coordinate axis which is at right angles to the plane in question.
- (2) If all the end point coordinates are omitted or the end point is at the same position as the start point, commanding the center using I 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".



#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
 (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.



#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
 (E) End point
 (SR) Start point radius
 (ER) End point radius
 (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, K) designation, these commands can also be issued by directly designating the circular radius R.



Command format

G02 X/U__ Z/W__ R__ F__ ; ... R specification circular interpolation Clockwise (CW)

G03 X/U__ Z/W__ R__ F__ ; ... R specification circular interpolation Counterclockwise (CCW)

X/U	X axis end point coordinate
Z/W	Z axis end point coordinate
R	Circular radius
F	Feedrate

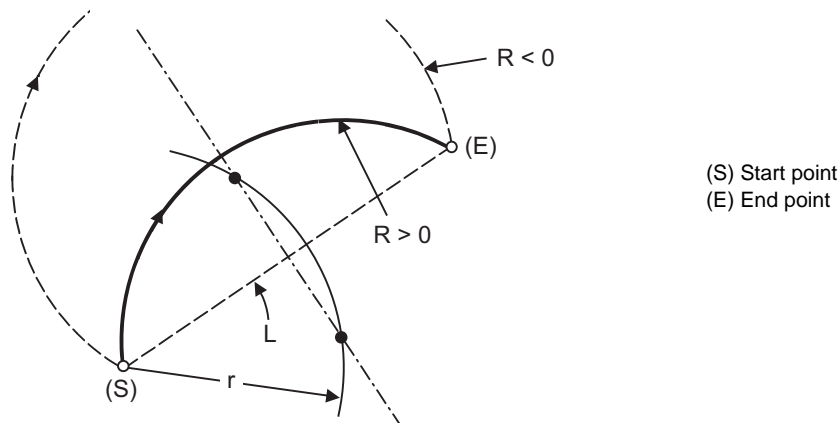
The arc radius is commanded with a program command unit. Caution is required for the arc command of an axis for which the input command unit differs. Command with a decimal point to avoid confusion.



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.



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

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

Where L is the line from the start point to the end point. If an R specification and I, 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, K specification circular command should therefore be used in such a case.

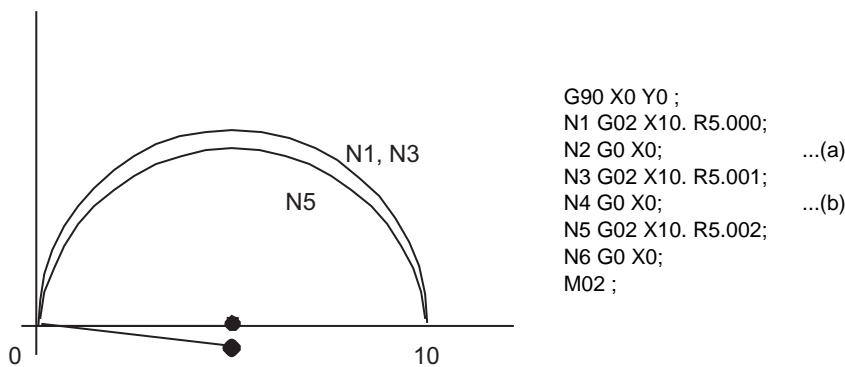
Circular center coordinate compensation

When "the error margin between the segment connecting the start and end points" and "the commanded radius x 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 segment connecting the start and end points" is compensated as the circular center.

Set the setting value to the parameter "#11028 Tolerance Arc Cent (Tolerable correction value of arc center error)".

(Example) "#11028 Tolerance Arc Cent" = "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



- (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 x 2 = 10.002 "Error 0.002 -> Compensate"
 N5: Radius x 2 = 10.004 "Error 0.004 -> Do not compensate"
 Therefore, this example is shown in the above figure.



Program example

(Example 1)

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

(Example 2)

G02 Xx1 Zz1 Ii1 Kk1 Rr1 Ff1 ;	R specification circular on X-Z plane (When the R specification and I, K specification are contained in the same block, the R specification has priority in processing.)
-------------------------------	---

6.5 Plane Selection ; G17,G18,G19



Function and purpose

These commands are used to select the control plane and the plane on which the circular exists.

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 rotation axis is entered as a parallel axis, the commands can select the plane containing the rotation axis.

These commands are used to select:

- The plane for circular interpolation
- The plane for nose R compensation



Command format

G17; ... I-J plane selection

G18 ; ... K-I plane selection

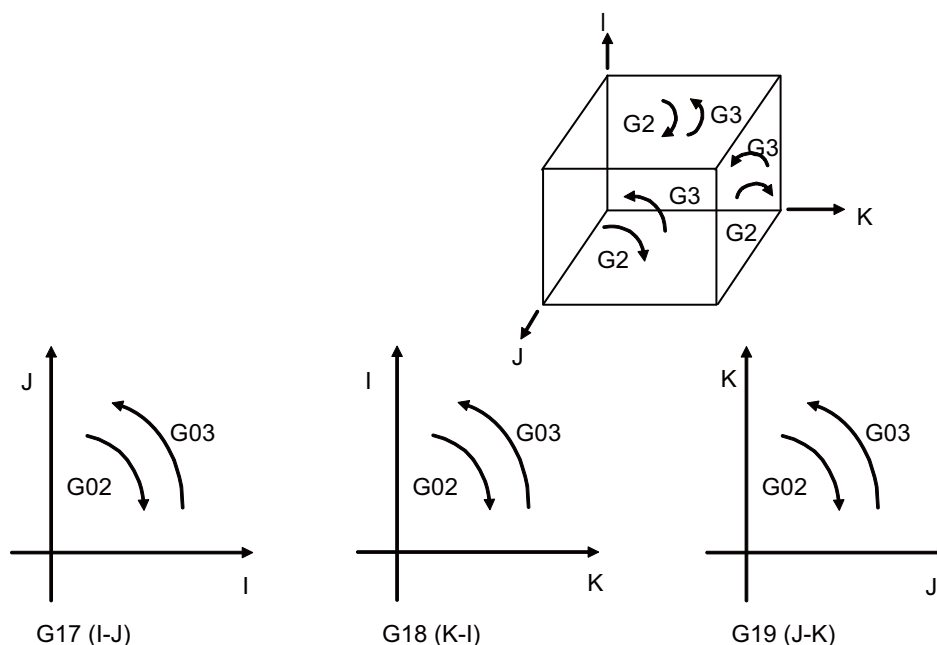
G19; ... J-K plane selection



Detailed description

I, J and K indicate each basic axis or parallel axis.

When the power is turned ON or when the system is reset, the plane set by the parameters "#1025 I_plane" is selected.



Parameter entry

	#1026 to 1028 base_I,J,K	#1029 to 1031 aux_I,J,K	Basic axes and parallel axes can be entered in the parameters. The same axis name can be entered in duplication, but when it is assigned in duplication, the plane is determined by plane selection system (5). The axis which is not registered as the control axis cannot be set.
I	X	Y	
J	Y		
K	Z		

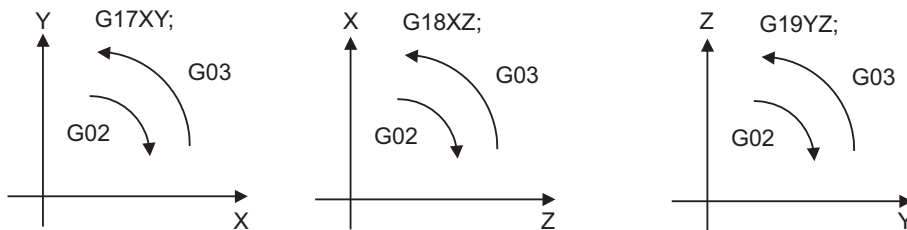
Table 1 Examples of plane selection parameter entry

Plane selection system

This section describes the plane selection shown in the "Table 1 Examples of plane selection parameter entry".

- (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.

(Example)



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

G18 X_ Z_ ; Z-X plane

Y_ Z_ ; Z-X plane (no plane change)

- (3) When the axis addresses are omitted in the block containing the plane selection G codes (G17, G18, G19), it is assumed that the axis addresses of the 3 basic axes have been assigned.

G18 ; (Z-X plane = G18 XZ ;)

- (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) 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.

G18 XYZ ; The Z-X plane is selected. Therefore, the Y movement is unrelated to the selected plane.

- (Note 1) When the "2" in the parameter "#1025 I_plane" is kept ON, the G18 plane is selected when the power is turned ON or when the system is reset.

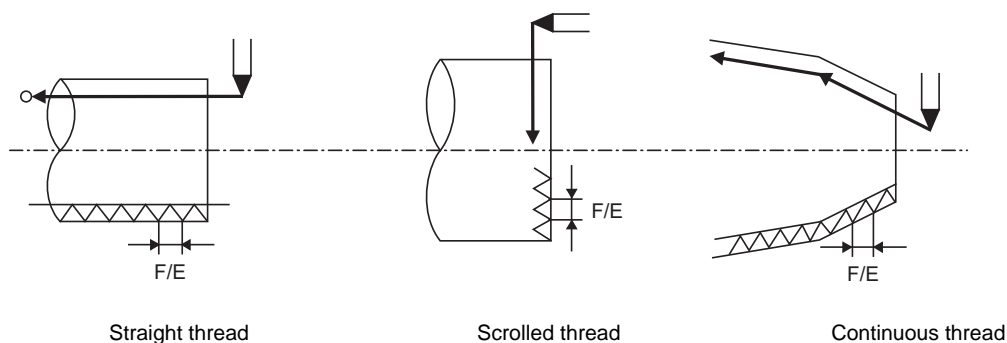
6.6 Thread Cutting

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, tapered thread-cutting, and continuous thread-cutting.



Straight thread

Scrolled thread

Continuous thread



Command format

Normal lead thread cutting

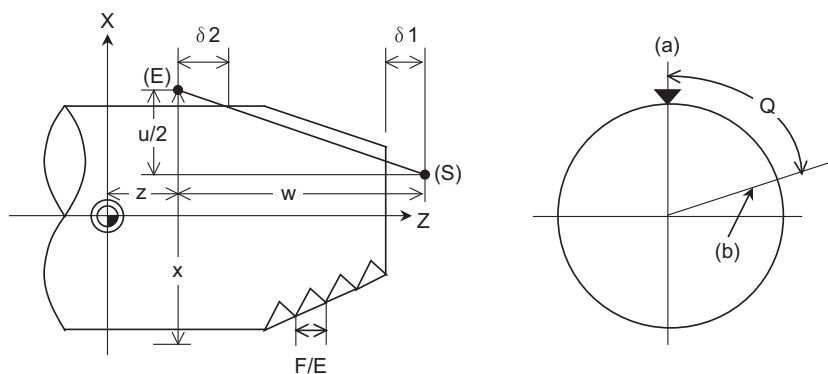
G33 Z/W__ X/U__ F__ Q__ ;

Z,W,X,U	Thread end point
F	Lead of long axis (axis which moves most) direction
Q	Thread cutting start shift angle, 0.001 to 360.000°

Precision lead thread cutting

G33 Z/W__ X/U__ E__ Q__ ;

Z,W,X,U	Thread end point
E	Lead of long axis (axis which moves most) direction
Q	Thread cutting start shift angle, 0.001 to 360.000°



$\delta 1 >$ Illegal lead at start of thread cutting

$\delta 2 >$ Illegal lead at end of thread cutting

(S) Start point

(E) End point

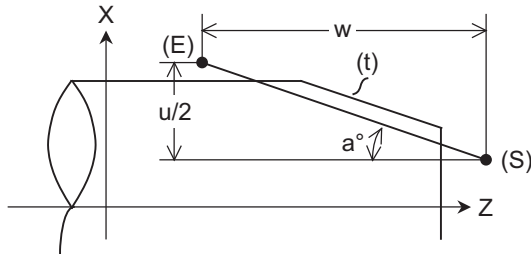
(a) One-rotation synchronization signal

(b) Thread cutting start position



Detailed description

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



(t) Tapered thread section (E) End point (S) Start point

When $a < 45^\circ$, Lead is in Z-axis direction.

When $a < 45^\circ$, Lead is in X-axis direction.

When $a = 45^\circ$, Lead can be in either Z or X-axis direction.

Thread cutting metric input

Input setting unit	B (0.000mm)		
Command address	F (mm/rev)	E (mm/rev)	E (ridges/inch)
Least Command Increments	1(=0.0001) (1.=1.0000)	1(=0.00001) (1.=1.00000)	1(=1.00) (1.=1.00)
Command range	0.0001 - 999.9999	0.00001 - 999.99999	0.03 - 999.99

Input setting unit	C (0.0001mm)		
Command address	F (mm/rev)	E (mm/rev)	E (ridges/inch)
Least Command Increments	1(=0.00001) (1.=1.00000)	1(=0.000001) (1.=1.000000)	1(=1.000) (1.=1.000)
Command range	0.00001 - 999.99999	0.000001 - 999.999999	0.026 - 222807.017

Input setting unit	D (0.00001mm)		
Command address	F (mm/rev)	E (mm/rev)	E (ridges/inch)
Least Command Increments	1(=0.000001) (1.=1.000000)	1(=0.0000001) (1.=1.0000000)	1(=1.0000) (1.=1.0000)
Command range	0.000001 - 999.999999	0.0000001 - 999.9999999	0.0255 - 224580.0000

Input setting unit	E (0.000001mm)		
Command address	F (mm/rev)	E (mm/rev)	E (ridges/inch)
Least Command Increments	1(=0.0000001) (1.=1.0000000)	1(=0.00000001) (1.=1.00000000)	1(=1.00000) (1.=1.00000)
Command range	0.0000001 - 999.9999999	0.00000001 - 999.99999999	0.02540 - 224719.00000

Thread cutting inch input

Input setting unit	B (0.0001inch)		
Command address	F (inch/rev)	E (inch/rev)	E (ridges/inch)
Least Command Increments	1(=0.00001) (1.=1.00000)	1(=0.000001) (1.=1.000000)	1(=1.000) (1.=1.000)
Command range	0.00001 - 39.37007	0.000001 - 39.370078	0.025 - 9999.999

Input setting unit	C (0.00001inch)		
Command address	F (inch/rev)	E (inch/rev)	E (ridges/inch)
Least Command Increments	1(=0.000001) (1.=1.000000)	1(=0.0000001) (1.=1.0000000)	1(=1.0000) (1.=1.0000)
Command range	0.000001 - 39.370078	0.0000001 - 39.3700787	0.0254 - 9999.9999

Input setting unit	D (0.000001inch)		
Command address	F (inch/rev)	E (inch/rev)	E (ridges/inch)
Least Command Increments	1(=0.0000001) (1.=1.0000000)	1(=0.00000001) (1.=1.00000000)	1(=1.00000) (1.=1.00000)
Command range	0.0000001 - 39.3700787	0.00000001 - 39.37007873	0.02540 - 9999.99999

Input setting unit	E (0.0000001inch)		
Command address	F (inch/rev)	E (inch/rev)	E (ridges/inch)
Least Command Increments	1(=0.00000001) (1.=1.00000000)	1(=0.000000001) (1.=1.000000000)	1(=1.000000) (1.=1.000000)
Command range	0.00000001 - 39.37007873	0.000000001 - 39.370078736	0.025400 - 9999.999999

(Note 1) It is not possible to assign a lead where the feedrate as converted into feed per minute exceeds the maximum cutting feedrate.

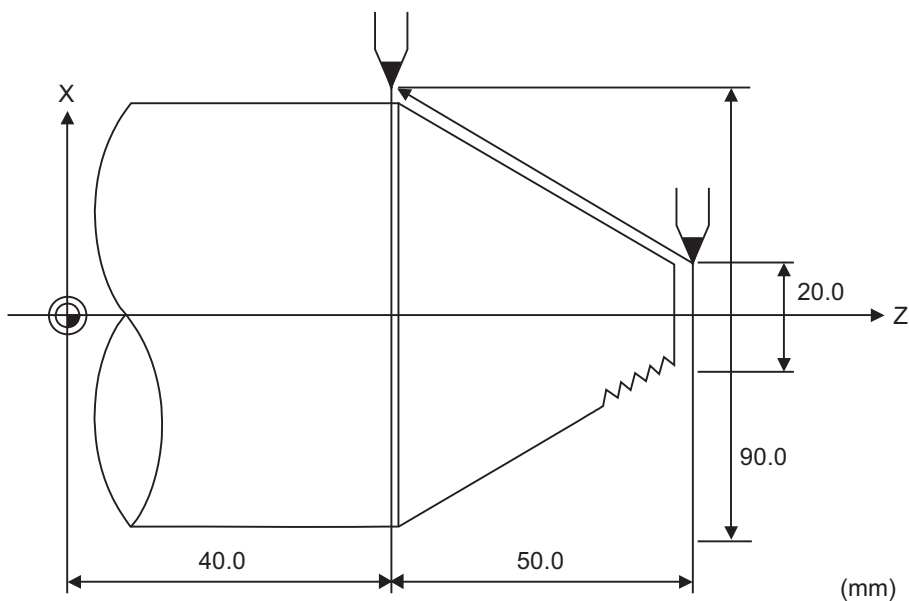
- (3) The constant surface speed control function should not be used for taper thread cutting commands or scrolled thread cutting commands.
- (4) The spindle rotation speed should be kept constant throughout from the rough cutting until the finishing.
- (5) If the feed hold function is employed during 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 $\delta 1$ and $\delta 2$ to the required thread length.
- (9) The spindle rotation speed is subject to the following restriction :

$$1 \leq R \leq \text{Maximum feed rate/Thread lead}$$
 Where $R \leq$ Tolerable speed of encoder (r/min)
 R: Spindle rotation speed (r/min)
 Thread lead = mm or inches
 Maximum feedrate= mm/min or inch/mm (this is subject to the restrictions imposed by the machine specifications.)
- (10) A program error (P97) may occur when the result of the expression (9) is $R < 1$ because the thread lead is very large to the highest cutting feedrate.

- (11) Dry run is valid for thread cutting but the feedrate based on dry run is not synchronized with the spindle rotation.
The dry run signal is checked at the start of thread cutting and any switching during thread cutting is ignored.
- (12) Synchronous feed applies for the thread cutting commands even with an asynchronous feed command (G94).
- (13) Spindle override and cutting feed override are invalid and the speeds are fixed to 100% during thread cutting.
- (14) When a thread cutting command is programmed during nose R 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 cancelled) is first executed and then the automatic operation stops. Note that automatic operation is stopped until the G33 command axis starts moving.
- (17) The thread cutting command waits for the single rotation synchronization signal of the rotary encoder and starts movement.
Note that carry out timing synchronization operation between part systems before issuing a thread cutting command with multiple part systems. For example, when using the 1-spindle specifications with multi-part systems, if one part system issues a thread cutting command during ongoing thread cutting by another part system, the movement will start without waiting for the rotary encoder single rotation synchronization signal.
- (18) The thread cutting start shift angle is not modal. If there is no Q command with G33, this will be handled as "Q0".
- (19) The automatic handle interrupt/interruption is valid during thread cutting.
- (20) If a value exceeding 360.000 is command in G33 Q, a program error (P35) will occur.
- (21) G33 cuts one row with one cycle. To cut two rows, change the Q value, and issue the same command.



Program example



G33 X90.0 Z40.0 E12.34567 ;	Absolute value command
G33 U70.0 W-50.0 E12.34567 ;	Incremental value command

6.6.2 Inch Thread Cutting ; G33



Function and purpose

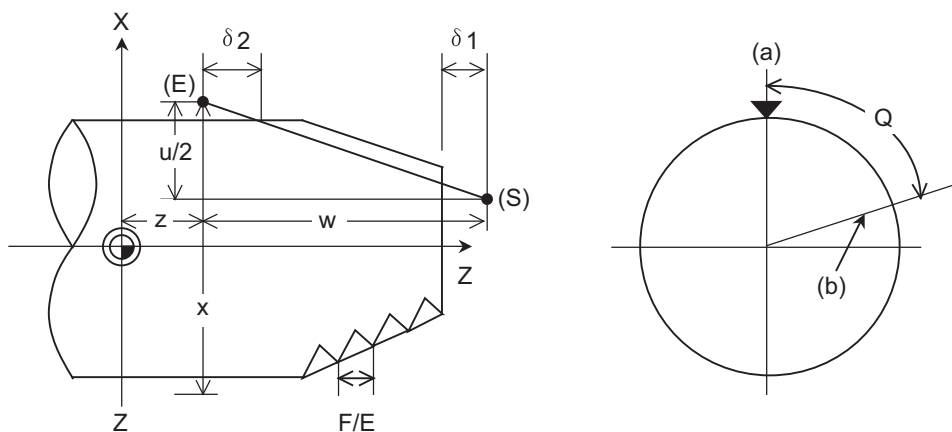
If the number of ridges per inch in the long axis direction is assigned in the G33 command, the feed of the tool synchronized with the spindle rotation will be controlled, which means that constant-lead straight thread-cutting and tapered thread-cutting can be performed.



Command format

G33 Z/W__ X/U__ E__ Q__ ; ... Inch thread cutting

Z,W,X,U	Thread end point
E	Number of ridges per inch in the long axis direction (axis which moves the most) (decimal point command can also be assigned)
Q	Thread cutting start shift angle, 0.001 to 360.000°



$\delta 1 >$ Illegal lead at start of thread cutting

$\delta 2 >$ Illegal lead at end of thread cutting

(S) Start point

(E) End point

(a) One-rotation synchronization signal

(b) Thread cutting start position

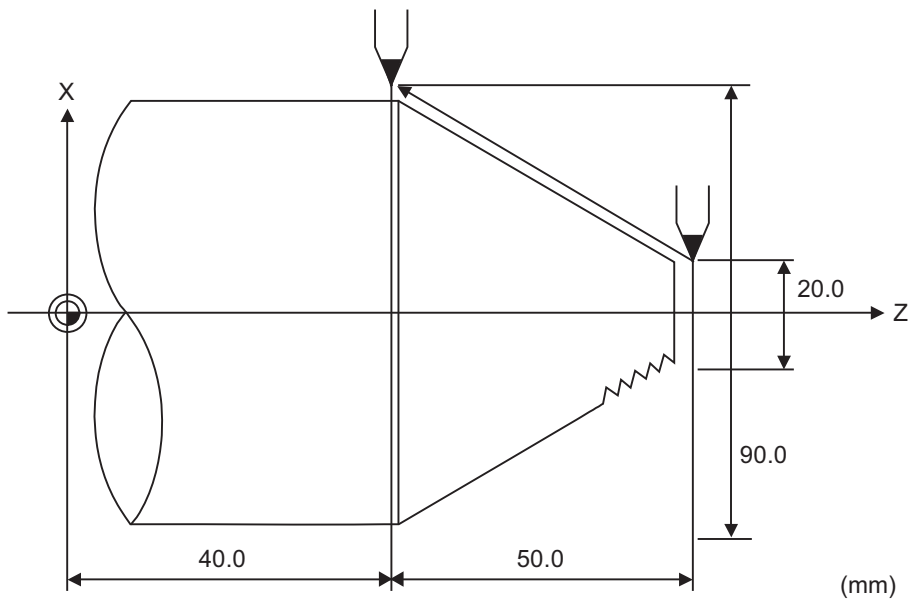


Detailed description

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



Program example



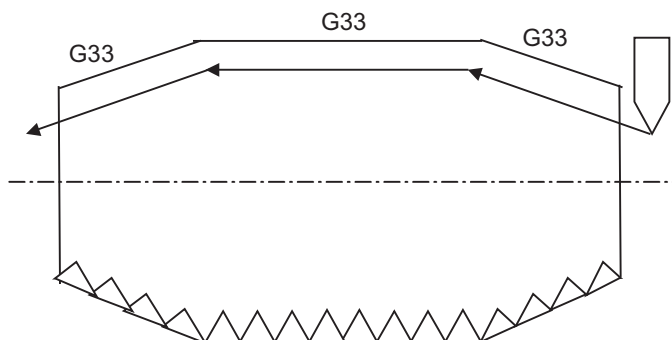
G33 X90.0 Z40.0 E12.0 ;	Absolute value command
G33 U70.0 W-50.0 E12.0 ;	Incremental value command

6.6.3 Continuous Thread Cutting ; G33



Function and purpose

Continuous thread cutting is possible by assigning thread cutting commands continuously. In this way, it is possible to cut special threads whose lead or shape changes.



Command format

G33 Zz1/Ww1 Xx1/Uu1 Ff1/Ee1 Qq1 ; ... Continuous thread cutting

Zzn,Wwn,Xxn,Uun	Thread end point
Ffn/Een	Lead of long axis (axis which moves most) direction
Qqn	Thread cutting start shift angle, 0.001 to 360.000°



Detailed description

- (1) The first thread cutting block in the continuous thread cutting command waits for the spindle's single rotation synchronization signal before starting thread cutting. From the second and following blocks, movement starts without waiting for the spindle's single rotation synchronization command. Thus, the thread cutting start shift angle (Q) can be commanded only in the first block.
- (2) G33 command can be omitted from the second and following blocks.
- (3) When commanding continuous thread cutting, command the thread cutting commands in successive blocks. If a command other than thread cutting is issued, continuous thread cutting will not take place. Note that if a command that does not involve axis movement (G4 dwell command, MST command, etc.) is commanded between the thread cutting command blocks, whether to wait for the spindle's single rotation synchronization signal after the 2nd block can be selected with the parameters.

# No.	Item	Details	Setting range
1270	ext06/bit6	Set the continuous thread cutting Z phase wait operation. 0: If there is no movement command (MST command, etc.) between the thread cutting blocks, the 2nd block thread cutting waits for the spindle's single rotation synchronization signal before starting movement. 1: Even if there is no movement command (MST command, etc.) between the thread cutting blocks, the 2nd block thread cutting starts movement without waiting for the spindle's single rotation synchronization signal.	0 / 1

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

6.6.4 Variable Lead Thread Cutting ; G34



Function and purpose

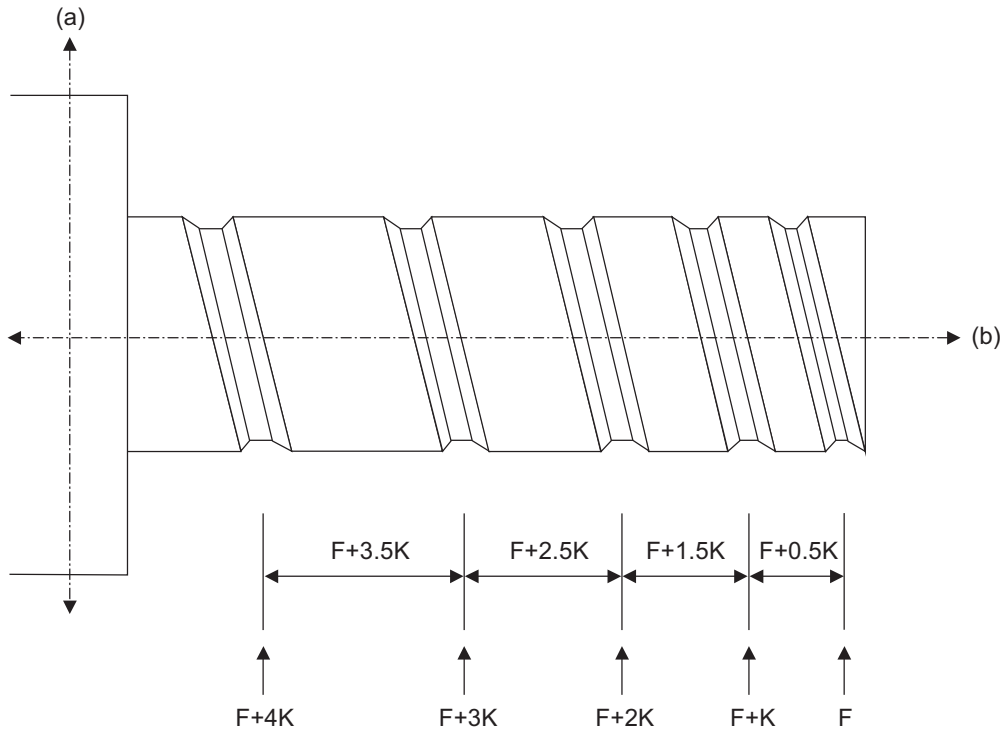
Variable lead thread cutting is enabled by a command specifying a lead increment or decrement amount per turn of the screw.



Command format

G34 X/U__ Z/W__ F/E__ K__ ; ... Variable lead thread cutting

X/U Z/W	Thread end point
F/E	Standard screw lead
K	Lead increment or decrement amount per turn of the screw



- (a) Non-lead axis
- (b) Lead axis
- (F) Lead speed



Detailed description

- (1) The command range is as shown below.
Thread cutting metric input

Input setting unit	B (0.001mm)	
Command address	F (mm/rev)	E (mm/rev)
Least Command Increments	1(=0.0001) (1.=1.0000)	1(=0.00001) (1.=1.00000)
Command range	0.0001 - 999.9999	0.00001 - 999.99999

Input setting unit	C (0.0001mm)	
Command address	F (mm/rev)	E (mm/rev)
Least Command Increments	1(=0.00001) (1.=1.00000)	1(=0.000001) (1.=1.000000)
Command range	0.00001 - 999.99999	0.000001 - 999.999999

Input setting unit	D (0.00001mm)	
Command address	F (mm/rev)	E (mm/rev)
Least Command Increments	1(=0.000001) (1.=1.000000)	1(=0.0000001) (1.=1.0000000)
Command range	0.000001 - 999.999999	0.0000001 - 999.9999999

Input setting unit	E (0.000001mm)	
Command address	F (mm/rev)	E (mm/rev)
Least Command Increments	1(=0.0000001) (1.=1.0000000)	1(=0.00000001) (1.=1.00000000)
Command range	0.0000001 - 999.9999999	0.00000001 - 999.99999999

Input setting unit	B/C/D/E	
Command address	K(n*mm/rev)	
Least Command Increments	n: Number of pitches	
Command range	Same as F or E (signed)	

Thread cutting inch input

Input setting unit	B (0.0001inch)	
Command address	F (inch/rev)	E (inch/rev)
Least Command Increments	1(=0.00001) (1.=1.00000)	1(=0.00001) (1.=1.00000)
Command range	0.00001 - 39.37007	0.00001 - 39.37007

Input setting unit	C (0.00001inch)	
Command address	F (inch/rev)	E (inch/rev)
Least Command Increments	1(=0.000001) (1.=1.000000)	1(=0.000001) (1.=1.000000)
Command range	0.000001 - 39.370078	0.000001 - 39.370078

Input setting unit	D (0.000001inch)	
Command address	F (inch/rev)	E (inch/rev)
Least Command Increments	1(=0.0000001) (1.=1.0000000)	1(=0.0000001) (1.=1.0000000)
Command range	0.0000001 - 39.3700787	0.0000001 - 39.3700787

Input setting unit	E (0.0000001inch)	
Command address	F (inch/rev)	E (inch/rev)
Least Command Increments	1(=0.00000001) (1.=1.00000000)	1(=0.00000001) (1.=1.00000000)
Command range	0.00000001 - 39.37007873	0.00000001 - 39.37007873

Input setting unit	B/C/D/E
Command address	K(n*inch/rev)
Least Command Increments	n: Number of pitches
Command range	Same as F or E (signed)

- (2) A positive value of K indicates incremental pitches.
Movement amount of one block (n pitches) = (F + K) + (F + 2K) + (F + 3K) + ... + (F + nK)
- (3) A negative value of K indicates decremental pitches.
Movement amount of one block (n pitches) = (F - K) + (F - 2K) + (F - 3K) + ... + (F - nK)
- (4) A program error will occur if the thread lead is not set correctly.

Error No.	Details	Remedy
P 93	Illegal pitch value (1) An invalid value is specified for F/E or K in a thread cutting command. (2) The last lead goes outside of the F/E command range.	The last lead goes outside of the F/E command range. LL : Last lead NP : Number of pitches $LL = \sqrt{(F^2 + 2KZ)}$ $NP = (-F + LL) / K$

- (5) The other details are the same as G33.
Refer to "Constant lead thread cutting ; G33".

6.6.5 Circular Thread Cutting ; G35,G36



Function and purpose

Circular thread cutting making the longitudinal direction the lead is possible.

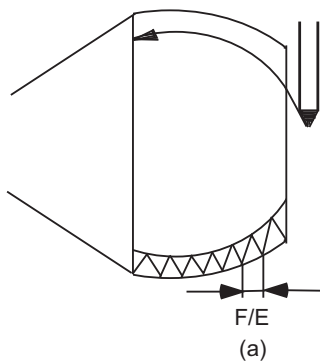
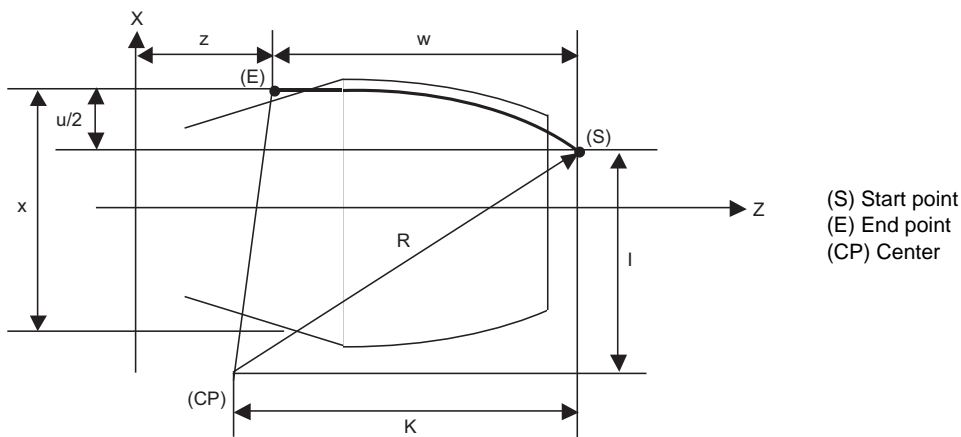


Command format

G35 X/U__ Z/W__ I__ K__ (R__) F/E__ Q__ ; ... Circular thread cutting Clockwise (CW)

G36 X/U__ Z/W__ I__ K__ (R__) F/E__ Q__ ; ... Circular thread cutting Counterclockwise (CCW)

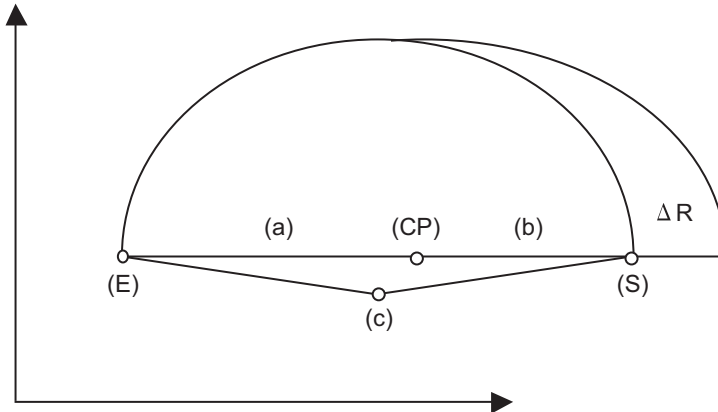
X/U	X axis circular end point coordinate (X is the absolute value of axis workpiece coordinate system, U is the incremental value from current position)
Z/W	Z axis circular end point coordinate (Z is the absolute value of workpiece coordinate system, W is the incremental value from current position)
I	X axis circular center (incremental value of circular center looking from start point)
K	Z axis circular center (incremental value of circular center looking from start point)
R	Circular radius
F/E	Longitudinal (axis with largest movement amount) direction lead (F.. normal lead thread cutting/E .. precision lead thread, inch thread)
Q	Thread cutting start shift angle, 0 to 360.000°





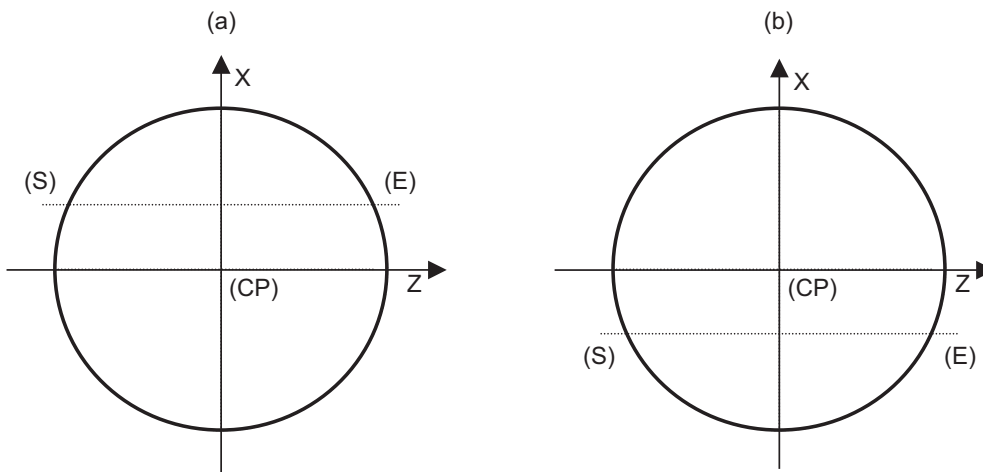
Detailed description

- (1) A program error (P33) will occur if the start point and end point match or if the arc center angle is more than 180°.
- (2) The following will occur if the start point radius and end point radius do not match.
 - A program error (P70) will occur if the error ΔR is more than parameter "#1084 RadErr" (arc error).
 - Interpolation will start from the arc center where the start point radius and end point radius match if the error ΔR is less than parameter "#1084 RadErr".



(a) End point radius (b) Start point radius (c) Obtained center
 (S) Start point (E) End point (CP) Center

- (3) A program error (P33) will occur if the R_ sign is negative.
- (4) A program error (P33) will occur if there is no I_K_ command and R_ command.
- (5) The R_ command will have the priority if the I_K_ command and R_ command are issued in the same block.
- (6) If the arc center is (0,0), the arc command can be issued for two successive quadrants. A program error (P33) will occur if an arc with more than three quadrants is issued.
 [When Z axis is long axis]



(a) 1st and 4th quadrant (b) 2nd and 3rd quadrant
 (S) Start point (E) End point (CP) Center

- (7) When the movement amount is equal, the horizontal direction in the selected plane will be the long axis.

Plane selection	Long axis when movement amount is equal
G17 (XY plane)	I axis
G18 (ZX plane)	K axis
G19 (YZ plane)	J axis

- (8) G36 is used for the two functions, automatic tool length measurement and circular thread cutting (CCW). Which function to be applied depends on the setting of the parameter "#1238 set10/bit0" (circular thread cutting).

When "#1238 set10/bit0" is set to "0"

G code	Function
G35	Circular thread cutting clockwise (CW)
G36	Automatic tool length measurement X

When "#1238 set10/bit0" is set to "1"

G code	Function
G35	Circular thread cutting clockwise (CW)
G36	Circular thread cutting counterclockwise (CCW)
G37	Automatic tool length measurement Z
G37.1	Automatic tool length measurement X
G37.2	Automatic tool length measurement Z

- (9) If the lead axis and non-lead axis cutting feedrate is faster than the clamp speed when thread cutting is started, the "M01 operation error 107" will occur, and thread cutting will not start.
- (10) During thread cutting, the cutting feed rate may exceed the clamp speed to guarantee the lead. In this case, the error "M01 operation error 107" will appear, but thread cutting will continue. However, if the "cutting feedrate > clamp speed" is established during circular thread cutting commanded in the second or following block of continuous thread cutting, automatic operation will be stopped just before the circular thread cutting command in the 2nd block, and the error "M01 operation error 107" will appear.
- (11) Continuous thread cutting is possible by assigning thread cutting commands continuously. In this way, it is possible to cut special threads whose lead or shape changes midway.
The continuous thread cutting command can be issued in the order of arc -> arc, arc -> constant lead, and constant lead -> arc.
- (12) An illegal lead is normally produced at the start of the thread and at the end of the cutting because of servo system delay.
Thus, command the length of the required thread length and also the illegal thread length of the start and end of the thread cutting.
As another method, command the required thread length as a circular thread (G35/G36), and then command the illegal lead length before and after that command (start and end of thread cutting) as a constant lead thread (G33). (Continuous thread cutting in order of constant lead -> arc -> constant lead.)



Relation with other functions

- (1) A program error (P113) will occur if the G35/G36 command is issued to an axis not within the plane.
- (2) The thread cutting speed is not synchronized with the spindle rotation when dry run is valid. (The thread pitch is not guaranteed.)
- (3) If the dry run switch is turned ON during thread cutting, the dry run signal will be ignored.
- (4) 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 (when the thread cutting mode is terminated).
- (5) Circular thread cutting will function normally even during mirror image.
- (6) A program error (P201) will occur if the G35/G36 circular thread cutting command is issued in the finish shape program of the compound type fixed cycle for turning machining.
- (7) A program error (P385) will occur if thread cutting corner rounding or corner chamfering is commanded during circular thread cutting or the next block.
- (8) Geometric and circular thread cutting cannot be commanded simultaneously. If commanded simultaneously, a program error (P395 or P70) will occur.
- (9) If thread cutting is commanded during nose R compensation, nose R compensation will be temporarily canceled, and thread cutting will be executed.
- (10) Do not issue the circular thread cutting command during constant surface speed control. The thread will not be cut correctly because the spindle rotation speed will change during thread cutting.



Precautions

- (1) Spindle override does not function during thread cutting.
- (2) A program error (P39) will occur if G35/G36 is commanded when the additional specifications are not provided.

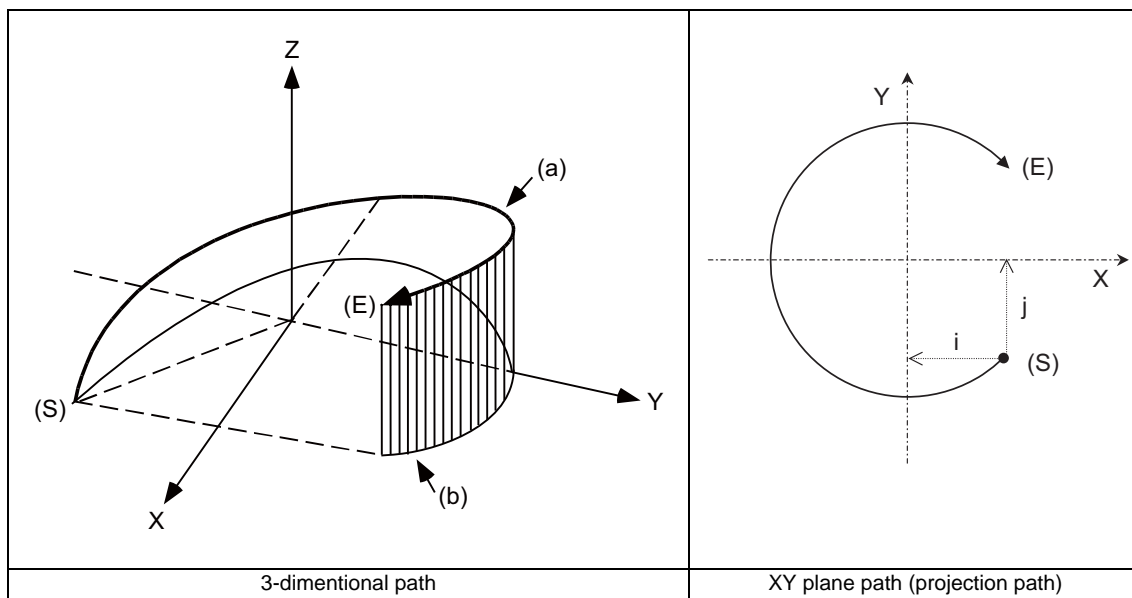
6.7 Helical Interpolation ; G17,G18,G19 and G02,G03



Function and purpose

This function is for circularly interpolating 2 axes on the selected plane and simultaneously interpolating the other axis linearly in synchronization with the circular motion.

When this interpolation is performed with 3 orthogonal axes, the tool will travel helically.



- (a) Program command path
- (b) XY plane projection path in command program
- (S) Start point
- (E) End point



Command format

G17/G18/G19 G02/G03 X/U__ Y/V__ Z/W__ I__ J__ F__(R__ F__); ... Helical interpolation

G17/G18/G19	Arc plane (G17: X-Y plane, G18: Z-X plane, G19: Y-Z plane)
G02/G03	Arc rotation direction (G02: clockwise, G03: counterclockwise)
X/U, Y/V	Arc end point coordinates
Z/W	Linear axis end point coordinates
I, J	Arc center coordinates
R	Arc radius
F	Feedrate

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

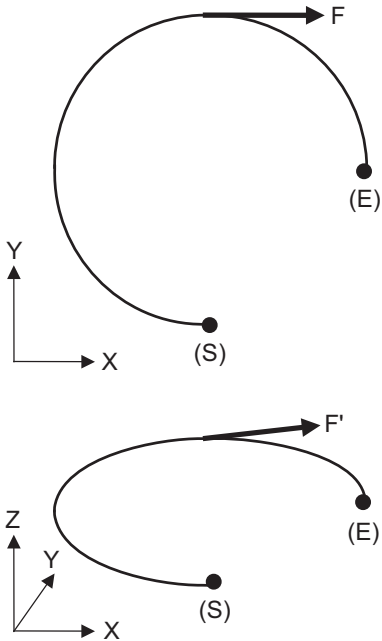


Detailed description

Speed designation during the helical interpolation

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

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



(S) Start point

(E) End point

The arc plane element speed designation and normal speed designation can be selected with the parameter.

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

Arc plane element speed designation selection

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

G17 G91 G02 X10. Y10. Z-4. I10. F100 ;	Helical interpolation at speed at which arc plane element is F100
G01 X20. ;	Linear interpolation at F100
G02 X10. Y-10. Z4. J10. ;	Helical interpolation at speed at which arc plane element is F100
G01 Y-40. F120;	Linear interpolation at F120
G02 X-10. Y-10. Z-4. I-10. ;	Helical interpolation at speed at which arc plane element is F120
G01 X-20. ;	Linear interpolation at F120

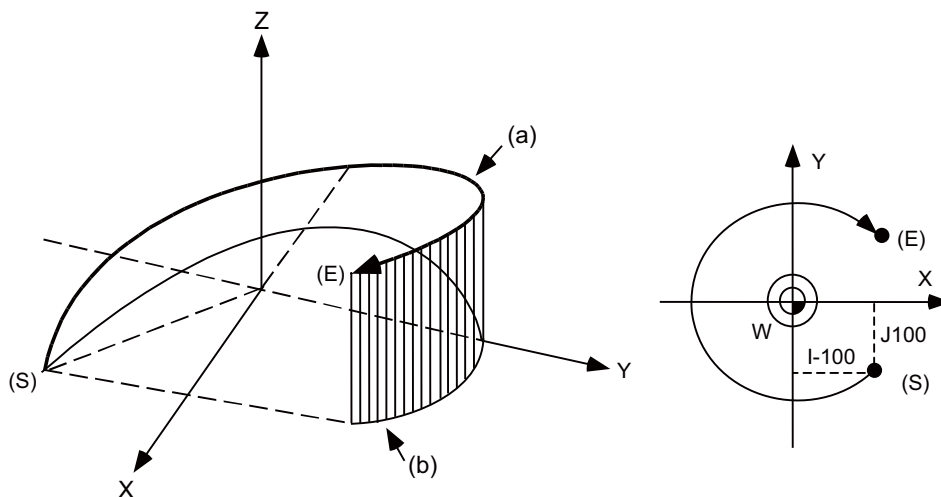
When the arc plane element speed designation is selected, only the helical interpolation speed command is converted to the speed commanded with the arc plane element and operates. The other linear and arc commands operate as normal speed commands.

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



Program example

```
G17 G02 X100. Y100. Z100. I-100. J100. F120 ;
```



- (a) Program command path
- (b) XY plane projection path in command program
- (S) Start point
- (E) End point

The left drawing shows the process as an exploded view, and the right drawing shows the arc plane from directly above.

At the start of the block, the axis centers at the point -100mm in the X axis direction and 100mm in the Y axis direction from the workpiece coordinates (start point), and starts cutting at the feedrate 120mm/min while rotating.



Precautions and restrictions

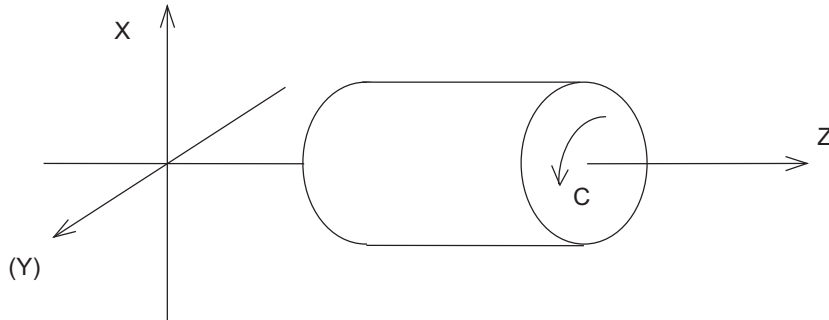
- (1) When executing helical interpolation, command another linear axis (several axis can be commanded) that does not contain the circular interpolation command and arc axis.
- (2) Up to the number of simultaneous contouring control axes can be commanded simultaneously.
- (3) A command exceeding one rotation cannot be issued. (It is complied with the circular interpolation command specifications.)
- (4) Command the feedrate as the composite speed for each axis.
- (5) With helical interpolation, the axis that configures the plane is the circular interpolation axis, and the other axes are the linear interpolation axes.
- (6) The movement of the linear interpolation axis is stopped and only the circular interpolation axis operates during the corner chamfering or corner R commands.
- (7) Refer to the circular interpolation (G02,G03) for other precautions.

6.8 Milling Interpolation ; G12.1



Function and purpose

Milling interpolation is used to perform contouring control by converting commands programmed in an orthogonal coordinate system into movements of a linear axis and rotation axis (workpiece rotation).



(Y) Hypothetical axis

G12.1 command starts a milling and G13.1 command cancels the milling to return to a normal turning.



Command format

G12.1 D__ E=__ ; ... Milling mode ON

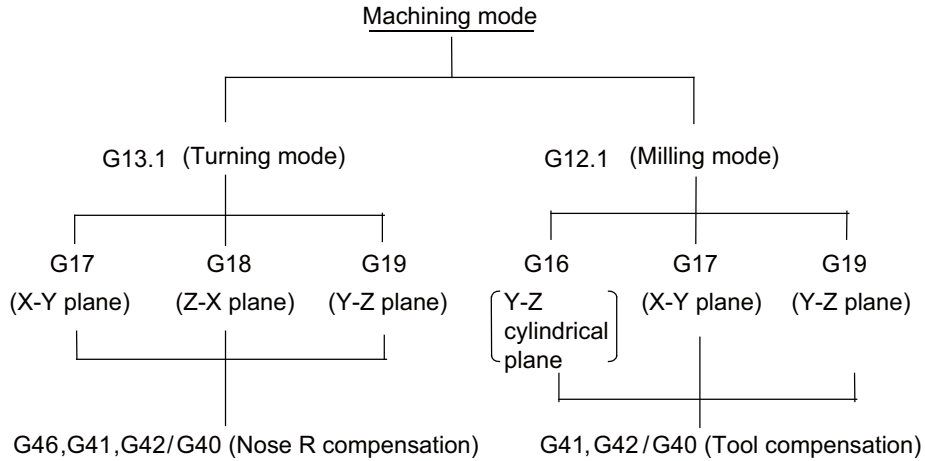
D	Selection of milling hypothetical axis name
E=	Designation of milling interpolation rotation axis

G13.1; ... Milling mode OFF (Turning mode)

Address	Meaning of address	Command range (unit)	Remarks
D	Selection of milling hypothetical axis name	0: Y axis 1: Rotation axis name	<ul style="list-style-type: none"> - If there is no D command, the milling hypothetical axis name will follow parameter (#1517 mill_C). - If only the D command is issued, it will be handled as D0. - A program error (P35) will occur if a value other than 0 or 1 is set to the numerical command following the D command.
E=	Milling Interpolation Designation of rotation axis	G12.1 command system rotation axis command address	<ul style="list-style-type: none"> - If there is no E command, the parameter (#1516 mill_ax) will be followed. - A program error (P33) will occur if only an E command is issued. - A program error (P33) will occur if an axis address is not commanded after "E=". - A program error (P300) will occur if an axis that does not exist in the command system is designated as the rotation axis name. - A program error (P32) will occur if a value is commanded for the rotation axis name. - To issue a program command after the "E= rotation axis name", delimit the "E= rotation axis name" and the other command with a comma (.). A program error (P33) will occur if there is no comma.

The following G codes are used to select milling and to set the conditions.

G code	Function	Remarks
G12.1 G13.1	Milling mode ON Milling mode OFF	Default is G13.1
G16 G17 G19	Selection of Y-Z cylindrical plane Selection of X-Y plane Selection of Y-Z plane	One of G17, G16, and G19 can be designated as the default value (when G12.1 is issued) by the parameter. #8113 Milling initial G16 #8114 Milling initial G19
G41 G42	Tool radius compensation left Tool radius compensation right	Default is G40 (radius compensation cancel).



6.8.1 Selecting Milling Mode



Detailed description

- (1) The G12.1 and G13.1 commands are used to switch between the turning (G13.1) and milling (G12.1) modes.
- (2) These commands are modal and the initial mode effective at power ON is the turning mode.
- (3) The following requirements must be satisfied before a G12.1 command is issued. Otherwise, a program error (P485) will occur.
 - (a) Nose R compensation has been canceled.
 - (b) Constant surface speed control has been canceled.
- (4) If one of the command axes in the milling mode has not completed reference position return, a program error (P484) will occur.
- (5) The G12.1 command automatically cancels an asynchronous mode F command. Therefore, specify an F value in milling mode.



Precautions

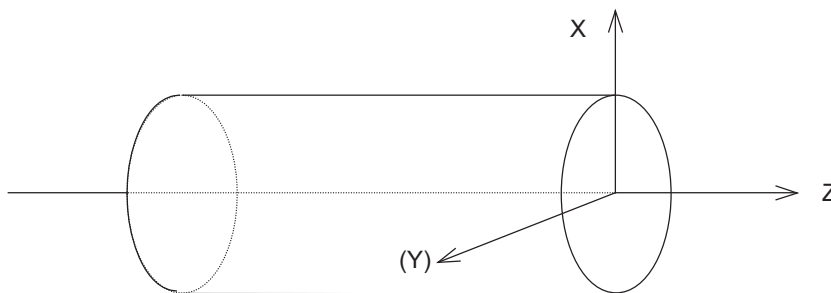
- (1) If G12.1 is executed without movement command after nose R compensation is canceled by an independent G40 command, nose R compensation is canceled in the G12.1 block.
- (2) If the milling interpolation command is issued during the mirror image, a program error (P486) will occur.
- (3) When the G12.1 command is issued, the deceleration check is executed.
- (4) If a command other than a plane selection is issued in the same block as G12.1 command, a program error (P33) will occur.

6.8.2 Milling Interpolation Control and Command Axes



Detailed description

- (1) The two orthogonal linear axes (X and Z axis) and a rotation axis are used as control axes for milling interpolation. The rotation axis is selected with the E command. The axis designated with the parameter (#1516 mill_ax) will be selected if there is no E command.
- (2) The command axes for milling interpolation are three orthogonal linear axes. The names of axes should be X, Z, and a hypothetical axis.
 A hypothetical axis is a virtual axis for interpolation command which intersects the X and Z axes at right angles. The hypothetical axis name will be either of the control rotation axis names which are selected in Y or (1) by a D command. The axis name designated with the parameter (#1517 mill_C) will be selected if there is no D command.



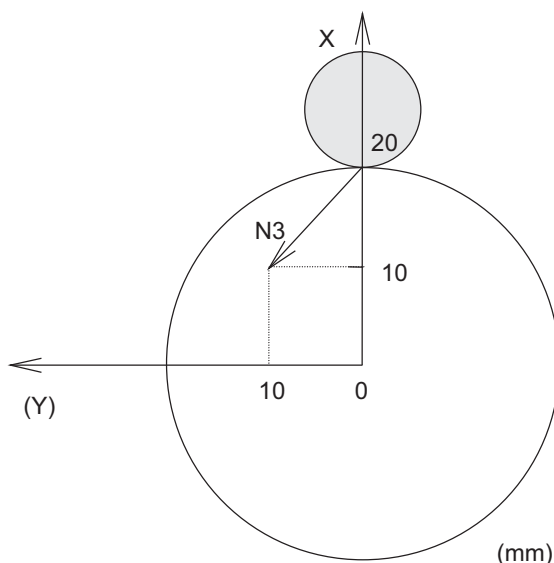
(Y) Hypothetical axis

- (3) Command axis X for milling is not just the interpolated one of control axis X. It is handled as X in the milling coordinate system when a G12.1 command is issued.
- (4) Whether the position in the milling coordinate system is handled by a diameter command or radius command is selected by the following parameter.

Parameter	Details
#8111 Milling Radius	0 : Radius command for all axes
	1 : Follows parameter (#1019 dia) for each axis

(Example 1)

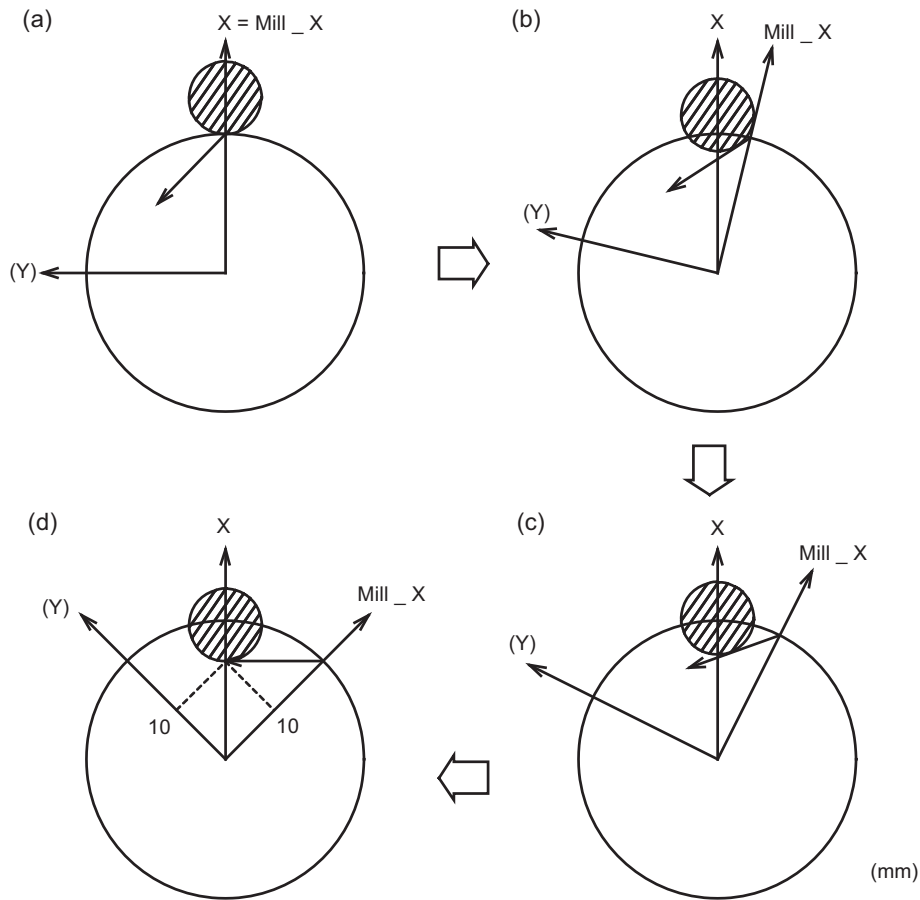
When C axis is used for rotation axis, and "Y" for hypothetical axis name



(Y) Hypothetical axis

```
(Program 1)
:
:
:
:
N1 G0 X40. ;
N2 G12.1 ;
(or G12.1 E=C, D0 ;)
N3 G1 X10. Y10. F10. ;
:
:
:
```

N3 of program 1 is executed as follows:



(Y) Hypothetical axis

Current values of (d)

- X 28.284 (diameter value display)
- C 45.000 (Except tool radius compensation amount)

- (5) Milling interpolation is also available for a two-control-axis system consisting of one linear axis and one rotation axis. The X axis must be used as the linear axis. The rotation and milling hypothetical axes are selected as shown above. In milling mode, the G17 plane must be selected.
- (6) The table below lists the incremental axis names of the hypothetical axis used in milling mode. These axis commands are radius commands only.

Selected hypothetical axis	Absolute axis name	Incremental axis name
Y axis	Y	V
Rotation axis (C)	#1013 axname set axis name (C)	#1014 incax set axis name (H)

(The following description uses Y for the hypothetical axis name and C for the rotation axis name.)

6.8.3 Selecting a Plane during the Milling Mode ; G17,G19,G16



Function and purpose

A plane selection command decides the plane on which a tool moves by circular interpolation or tool radius compensation in milling mode.



Command format

G17; ... X-Y plane

G19; ... Y-Z plane

G16 C; ... Y-Z cylindrical plane

C	Cylinder radius value
---	-----------------------



Detailed description

These G commands for plane selection are modal. A plane is automatically selected according to parameters each time the turning mode is switched to the milling mode by a G12.1 command.

Plane selection default value	G17	G19	G16	
#8113 Milling initial G16	0	0	1	1
#8114 Milling initial G19	0	1	0	1

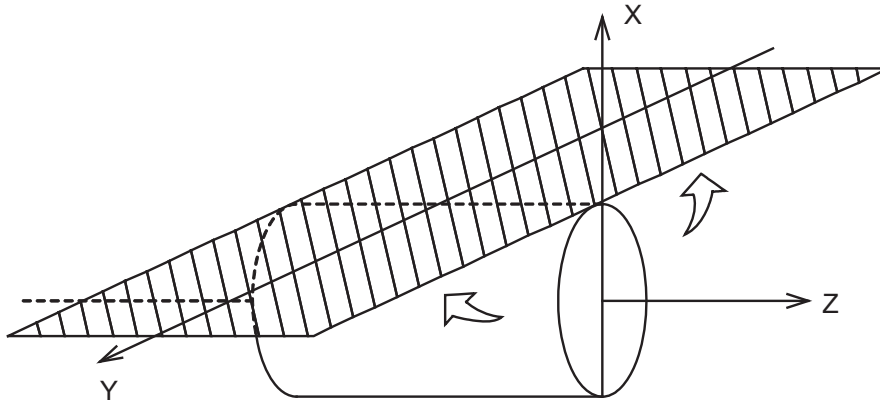
When the milling mode is switched back to the turning mode by a G13.1 command, the plane, that was selected before the milling mode is entered, is restored.

Planes to be selected

The three planes to be selected are explained below.

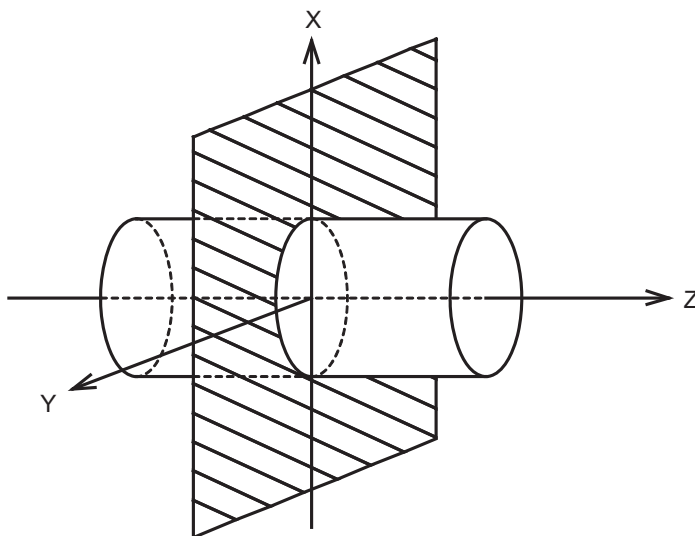
(1) G16 Y-Z cylindrical plane

G16 indicates the plane obtained by developing a cylinder with its bottom radius X . This is useful to process the side face of a workpiece.



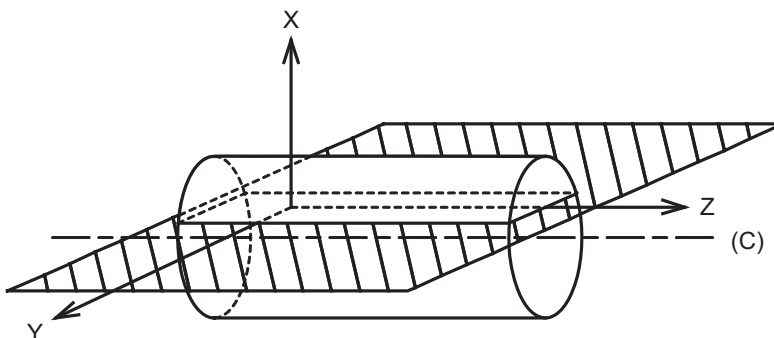
(2) G17 X-Y plane

G17 is an X-Y plane in an XYZ orthogonal coordinate system. This is useful to process the end face of a workpiece.



(3) G19 Y-Z plane

G19 is a Y-Z plane in an XYZ orthogonal coordinate system.



(C) Center of workpiece

6.8.4 Setting Milling Coordinate System



Function and purpose

The coordinate system in a milling mode is set depending on a plane which is selected each time the turning mode (G13.1) is switched to the milling mode by a G12.1 command.

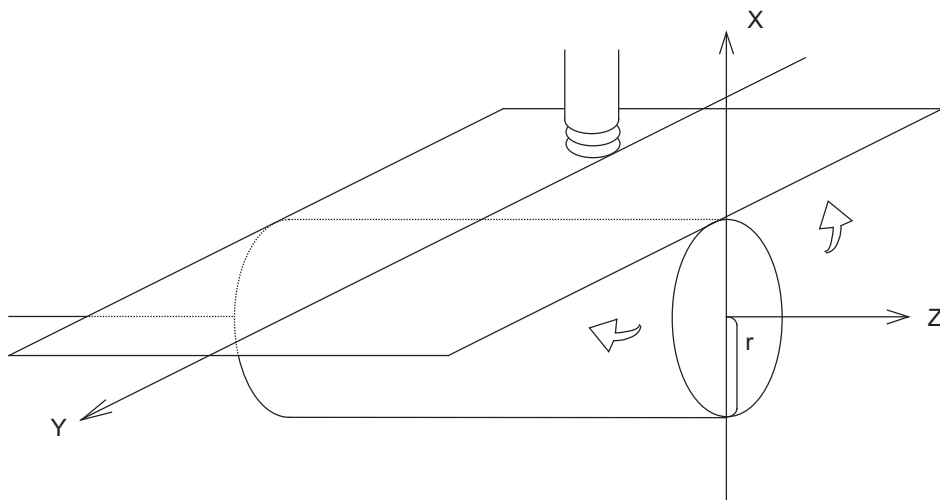


Detailed description

G16 plane

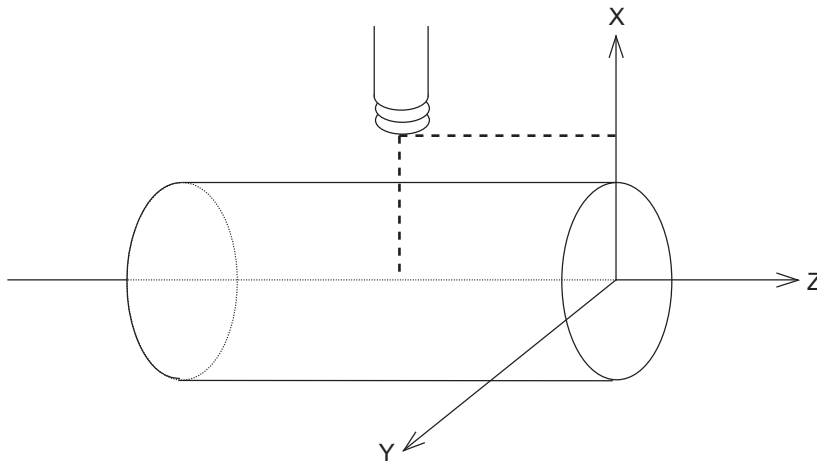
- (1) To select a G16 plane, specify the radius value of a cylinder by "G16 C__";. If no radius value is specified, the current X axis value at the time of G16 command is used as the radius value to define a cylinder. If no radius value can be defined, a program error (P485) will occur.
 - (2) As in normal turning mode, the X axis indicates the distance from the center line of the workpiece.
 - (3) G16 (Y-Z cylindrical plane) is actually the side of a cylinder.
 - (4) The X axis indicates the distance from the center line of the workpiece. The Y axis indicates the circumference with the radius value of the bottom of the cylinder which is defined by a G16 command.
 - (5) The zero point of the Y axis is the position where a G12.1 command is issued.
- (Example)

:		:		:		:
:		:		:		:
G12.1 G16 C50. ;	or	G12.1 ;	or	G12.1 Ee,Dd	or	G12.1 Ee,Dd ;
		G16 C50. ;		G16 C50. ;		G16 C50. ;
:		:		:		:
:		:		:		:



G17 and G19 planes

- (1) For the X and Z axes, the current positions are set as radius value in the coordinate value.
- (2) The Y axis is fixed as an axis which intersects the X and Z axes at right angles. Y=0 is set in a G12.1 command.



- (Note 1) In the milling mode on the G17 plane, the X axis is operated in the area (positive or negative side) that existed before issuing the G12.1 command. To control the X axis in the positive side in the milling mode, move the X axis to the positive area (including 0) before issuing the G12.1 command. To control the X axis in the negative side in the milling mode, move the X axis to the negative area (not including 0) before issuing the G12.1 command.

6.8.5 Preparatory Function



Detailed description

Valid G codes in milling mode

Classification	G code	Function	Classification	G code	Function
*	G00	Positioning		G65	Macro call
*	G01	Linear interpolation		G66	Macro modal call A
*	G02	Circular interpolation (CW)		G66.1	Macro modal call B
*	G03	Circular interpolation (CWW)		G67	Macro modal call Cancel
	G04	Dwell			
	G09	Exact stop check		G80	Hole drilling cycle cancel
				G83	Deep hole drilling cycle (Z axis)
	G13.1	Turning mode		G84	Tap cycle (Z axis)
				G85	Boring cycle (Z axis)
				G87	Deep hole drilling cycle (X axis)
○	G16	Plane selection Y-Z cylindrical plane		G88	Tap cycle (X axis)
	G17	Plane selection X-Y		G89	Boring cycle (X axis)
				G90	Absolute value command
	G19	Plane selection Y-Z		G91	Incremental value command
	G22	Barrier check ON		G94	Asynchronous feed
	G23	Barrier check OFF			
				G98	Hole drilling cycle initial point return
				G99	Hole drilling cycle R point return
				G61	Exact stop mode
	G40	Tool radius compensation cancel			
	G41	Tool radius compensation left		G64	Cutting mode
	G42	Tool radius compensation right			

*: Milling interpolation command

○ : G code effective only in milling mode

- (1) If an invalid G code is issued in milling mode, a program error (P481) will occur.
If the milling interpolation ON (G12.1) is issued in milling mode, a program error (P481) will occur.
- (2) In milling mode, all movement commands are issued by the coordinate system which is already determined by the selected machining plane. The rotation axis thus cannot be moved by a direct command in milling mode. To perform milling to a workpiece at a specific position, therefore, positioning must be done before a milling mode command.

(Example)

```

:
G0 X100. C180. ;      => Positioning before milling
G12.1 ; (or G12.1 E=C,D0 ;)
G0 X50. ;
:
    
```

- (3) If an axis other than X, Z, and Y (rotation axis) is specified in milling mode, a program error (P481) will occur.
- (4) In milling mode, the Y axis can be specified by only four G codes: G00, G01, G02, and G03. These are called the milling interpolation commands.
- (5) The G84, G88 synchronous tapping cycles cannot be used in the milling mode. The asynchronous tapping can be used during the milling mode; however, the synchronous tapping command must not be issued.

Positioning (G00)

If a G00 command is issued in milling mode, positioning is made to the specified point on the selected plane at a rapid traverse rate.

```
G00 X/U__ Y/V__ Z/W__ ;
```

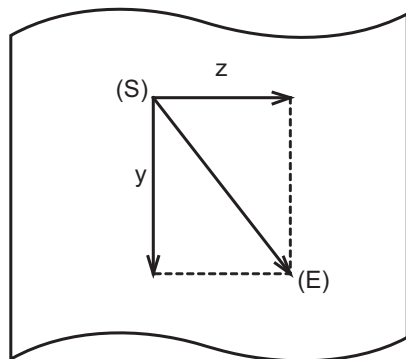
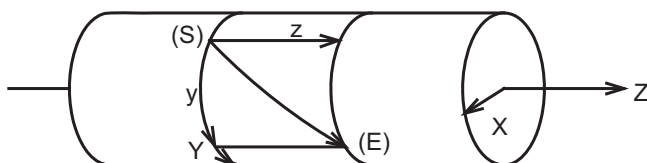
Linear interpolation (G01)

If a G01 command is issued in milling mode, linear interpolation is made to the specified point on the selected plane at the speed specified by an F command modal speed.

- (1) G16 mode (plane selection Y-Z cylindrical plane)

Program format

```
G01 Y/V__ Z/W__ X/U__ F__ ;
```



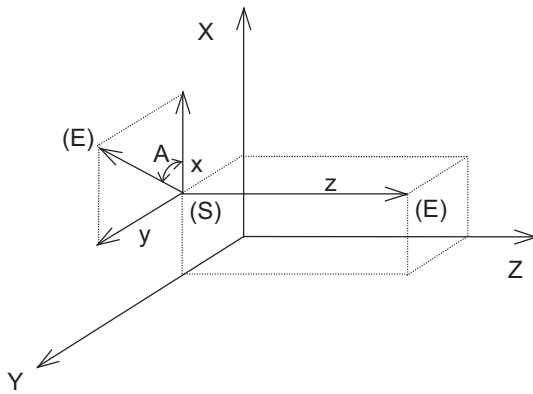
(S) Start point

(E) End point

(2) G17 mode (plane selection X-Y)

Program format

G01 X/U__ Y/V__ Z/W__ F__ ;

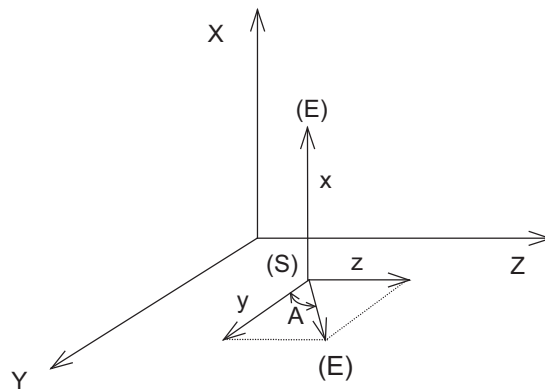


(S) Start point (E) End point

(3) G19 mode (plane selection Y-Z)

Program format

G01 Y/V__ Z/W__ X/U__ F__ ;



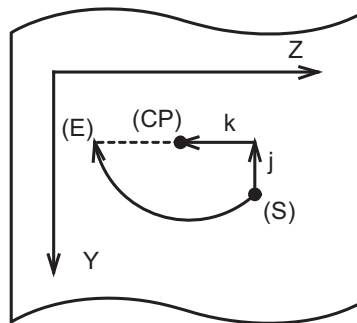
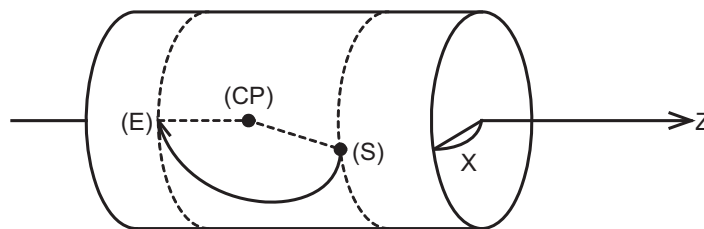
(S) Start point (E) End point

Circular interpolation (G02/G03)

If a G02 or G03 command is issued in milling mode, circular interpolation is performed at the specified speed on the selected plane.

(1) G16 mode

G02/G03 Y/V__ Z/W__ J__ K__ F__ ;	
or	
G02/G03 Y/V__ Z/W__ R__ F__ ;	
G02	Circular interpolation (clockwise)
G03	Circular interpolation (counterclockwise)
Y/V	Circular end point coordinate, Y axis (Y: absolute value, V: incremental value)
Z/W	Circular end point coordinate, Z axis (Z: absolute value, W: incremental value)
J/K	Circular center incremental value (radius command incremental value from the start point to the center)
R	Circular radius
F	Feedrate



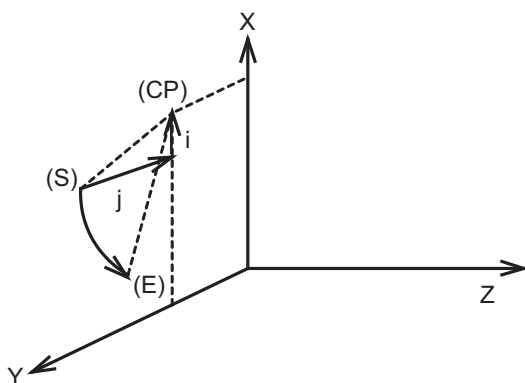
(S) Start point

(E) End point

(CP) Center of circular

(2) G17 mode

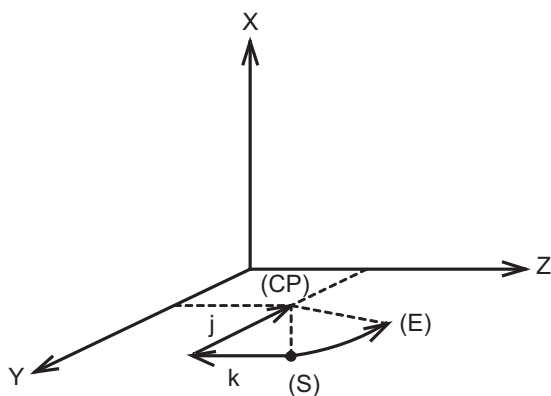
G02/G03 X/U__ Y/V__ I__ J__ F__ ;	
or	
G02/G03 X/U__ Y/V__ R__ F__ ;	
X/U	Circular end point coordinate, X axis (X: absolute value, U: incremental value)
Y/V	Circular end point coordinate, Y axis (Y: absolute value, V: incremental value)
I/J	Circular center incremental value (incremental value from the start point to the center)
R	Circular radius
F	Feedrate



(S) Start point (E) End point (CP) Center of circular

(3) G19 mode

G02/G03 Y/V__ Z/W__ J__ K__ F__ ;	
or	
G02/G03 Y/V__ Z/W__ R__ F__ ;	
Y/V	Circular end point coordinate, Y axis (Y: absolute value, V: incremental value)
Z/W	Circular end point coordinate, Z axis (Z: absolute value, W: incremental value)
J/K	Circular center incremental value (incremental value from the start point to the center)
R	Circular radius
F	Feedrate



(S) Start point (E) End point (CP) Center of circular

6.8.6 Switching from Milling Mode to Turning Mode; G13.1



Detailed description

- (1) A G13.1 command is used to cancel the milling mode and return to the turning mode.
- (2) The G13.1 command is effective if the following requirement is met. If not, a program error (P485) will occur.
 - (a) Tool radius compensation has been canceled.
- (3) The G13.1 command restores the plane selected before the G12.1 command was issued.
- (4) The G13.1 command restores the synchronous/asynchronous mode and the asynchronous F value which was selected before G12.1 command was issued.



Precautions

- (1) If G13.1 is executed after an independent G40 command without a movement command, tool radius compensation is canceled in the G13.1 block.
- (2) When the G13.1 command is issued, the deceleration check is executed.
- (3) If another command is issued during the G13.1 command, a program error (P33) will occur.

6.8.7 Feed Functions



Detailed description

An asynchronous feed mode (G94 command) can use F (6-digit integer + .3-digit decimal point) to specify the feedrate per minute in units of 0.001mm/min. The specifiable range is 0.001 to 999999.999mm/min. If the effective speed exceeds the cutting feed clamp speed, it is clamped by that clamp speed.



Precautions

- (1) Whenever the turning mode is switched to the milling mode by a G12.1 command, the F command modal value is canceled.
After each mode change, therefore, the feedrate must be set by an F command.
- (2) A G12.1 command forces the mode to shift to the asynchronous mode.
- (3) When the milling mode is canceled by a G13.1 command, both the feed mode and F command modal value return to the state before the G12.1 command was issued.

6.8.8 Program Support Functions



Relation with other functions

The following program support functions are valid in milling mode:

- (1) Linear angle command
- (2) Variable commands
- (3) Automatic corner chamfering/corner rounding
- (4) Geometric function
- (5) Hole drilling cycle
- (6) Subprogram function
- (7) User macro

6.8.9 Miscellaneous Functions



Relation with other functions

- (1) M and B commands can be issued in milling mode.
- (2) In milling mode, an S command specifies not the spindle rotation speed but the rotary tool speed.
- (3) If a T command is issued in milling mode, a program error (P485) will occur. Before issuing a G12.1 command, therefore, tool selection must be done.

```

:
:
T1212 ;      =>Specify a T command before a G12.1 command.
G0 X100. Z0. ;
G12.1 ; (or G12.1 E=C,D0 ;)
:
T1200 ;      =>In milling mode, a T command causes a program error (P485).
:
G13.1;
    
```

- (4) Complete the tool compensation operation (movement of tool length and wear compensation amount) before executing the milling interpolation.
 If the tool compensation operation is not completed when the milling interpolation start command has been issued, the followings will occur:
 - Machine coordinate will not change even if G12.1 is executed.
 - When G12.1 is executed, the workpiece coordinate will change to that of the post tool length compensation.
 (Even if the cylindrical interpolation is canceled, this workpiece coordinate will not be canceled.)

(Example)

Workpiece coordinate offset (X axis)=20.

Tool compensation amount of T0101 (X axis)=100.

Setting compensation operation with movement command after T command

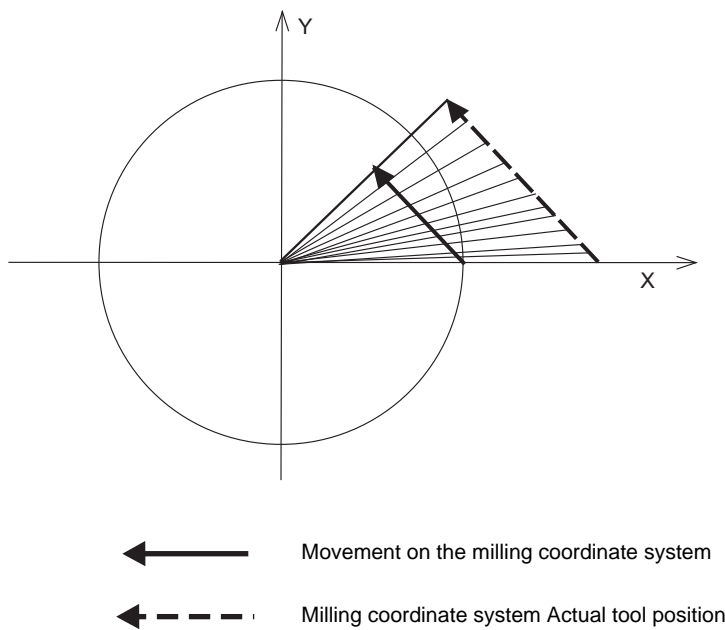
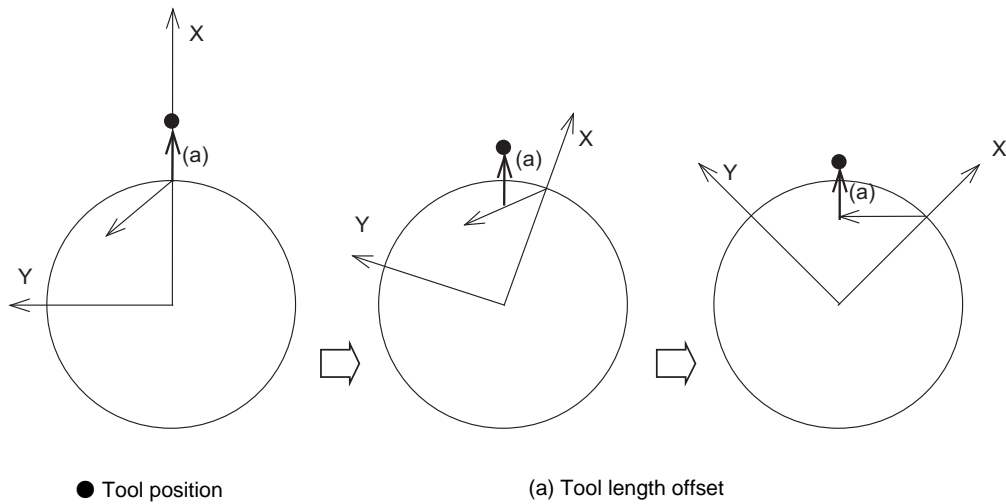
Machining program	Workpiece coordinate		Machine coordinate		
	[X axis]	[C axis]	[X axis]	[C axis]	
:					
G00 X200. C0.;	200.	0.	220.	0.	
T0101;	200.	0.	220.	0.	
G12.1;	100.	0.	220.	0.	<-The workpiece coordinate system is shifted (without moving the axis)
G01 X50. F1000;	50.	0.	170.	0.	
:					

6.8.10 Tool Length Compensation

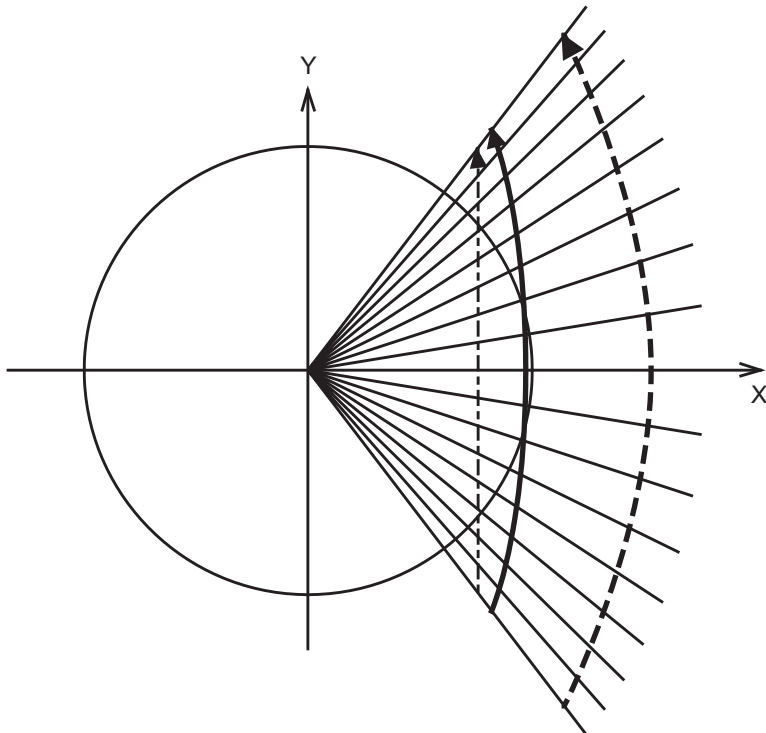


Detailed description

- (1) In milling mode, tool compensation is performed by adding the specified tool length offset amount to the cutting coordinates converted from the milling coordinate system.



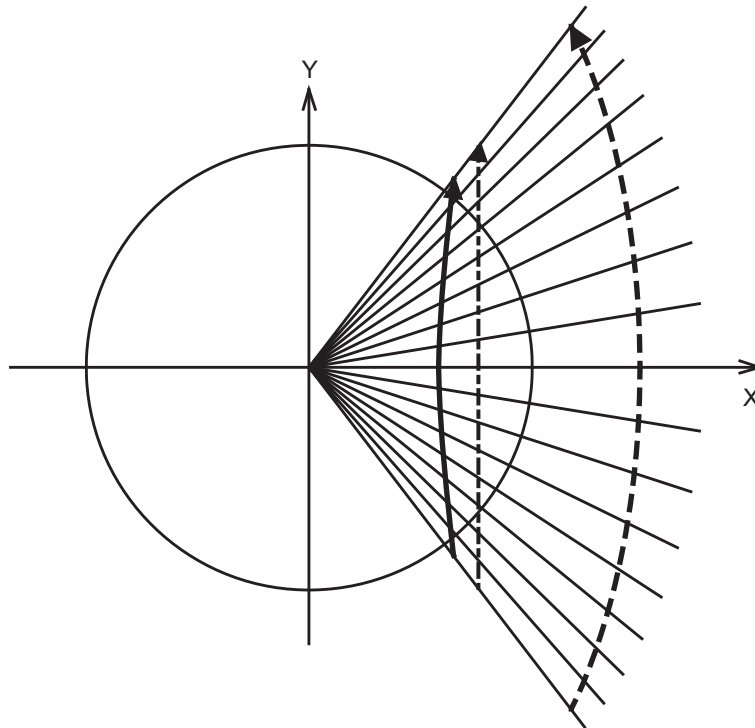
- (2) As can be seen from (1) in previous page, if the specified compensation amount is different from the actual tool length, desired shape will not be obtained.
- (a) If the compensation amount is larger than tool length
 Example: The actual tool length is 15.0 when tool length X = 20.0



- ← - - - - - Milling command
- ← ———— Movement on the milling coordinate system
- ← - - - - - Milling coordinate system Actual tool position

(b) If the compensation amount is smaller than tool length

Example: The actual tool length is 25.0 when tool length X = 20.0



- ← - - - - - Milling command
- ← Movement on the milling coordinate system
- ← - - - - - Milling coordinate system Actual tool position

6.8.11 Tool Radius Compensation



Function and purpose

The workpiece shape can be compensated in the direction of the vector by the radius amount of the tool specified by a G command (G40 to G42) and selected compensation No.



Command format

G40 X__ Y__ ; ... Tool radius compensation cancel

G41 X__ Y__ ; ... Tool radius compensation (left)

G42 X__ Y__ ; ... Tool radius compensation (right)

- (1) A tool radius compensation command must be issued after entering a milling mode.
The tool radius compensation command must be canceled before entering a turning mode.
- (2) A tool compensation No. must be specified before entering a milling mode (before issuing a G12. 1 command).
In a milling mode, a T command causes a program error (P485).
- (3) Tool radius compensation is performed on the selected plane.

G17 plane	XY axes
G19 plane	YZ axes
G16 plane	

6.8.11.1 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) While a G12.1 command is effective
- (2) After a compensation cancel command (G40) is issued

In the compensation cancel mode, the compensation vector is 0 and the tool center path matches the programmed path.

Programs including tool radius compensation must be terminated in the compensation cancel mode.

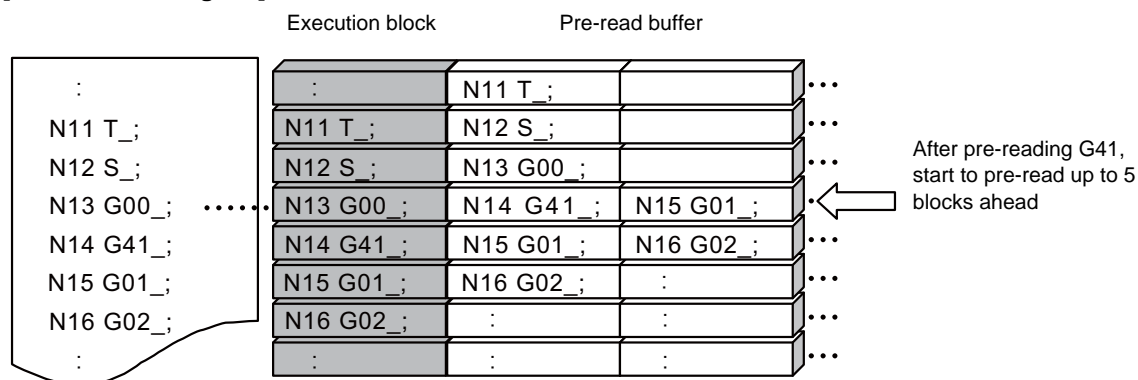
Tool radius compensation start (startup)

Tool radius compensation starts when all the following conditions are met in the compensation cancel mode.

- (1) A G41 or G42 command is issued.
- (2) The tool radius compensation No. is greater than 0 and equal to or less than the maximum compensation No.
- (3) The movement command is not a circular command.

Whether in continuous or single block operation, compensation always starts after reading three blocks, or if the three blocks do not contain any movement command, up to five continuous blocks will be pre-read. In compensation mode, too, up to 5 blocks are pre-read and the compensation is arithmetically processed.

[Control state diagram]



There are two ways of starting the compensation operation: type A and type B.

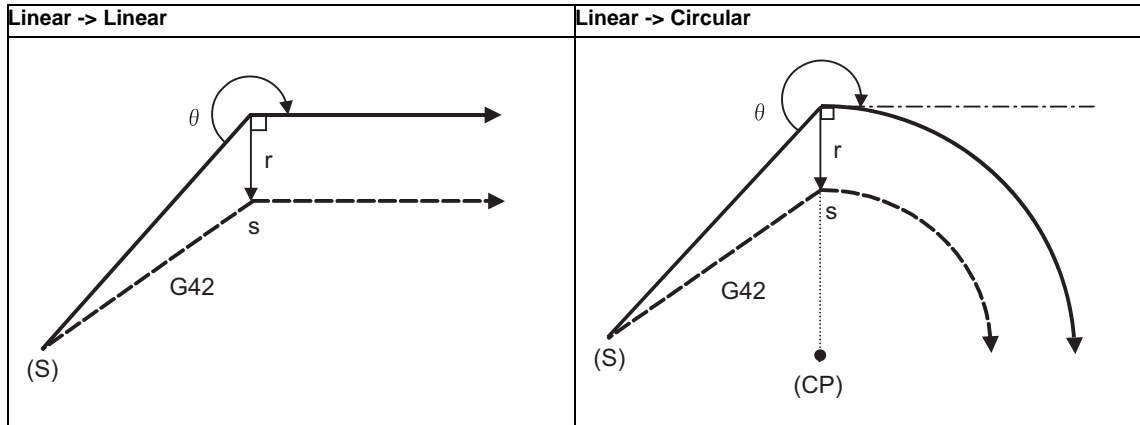
Choose the type by the parameter "#8157 tool nose compensation type B".

This type is used in common with the compensation cancel type.

#8157 tool nose compensation type B	Type	Explanation
0	Type A	When starting up/canceling the tool nose radius compensation and radius compensation, type A will not conduct intersection operation processing to the starting up/canceling command blocks and, instead, convert them to an compensation vector which is vertical to the command vector.
1	Type B	When starting up/canceling the tool nose radius compensation and 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



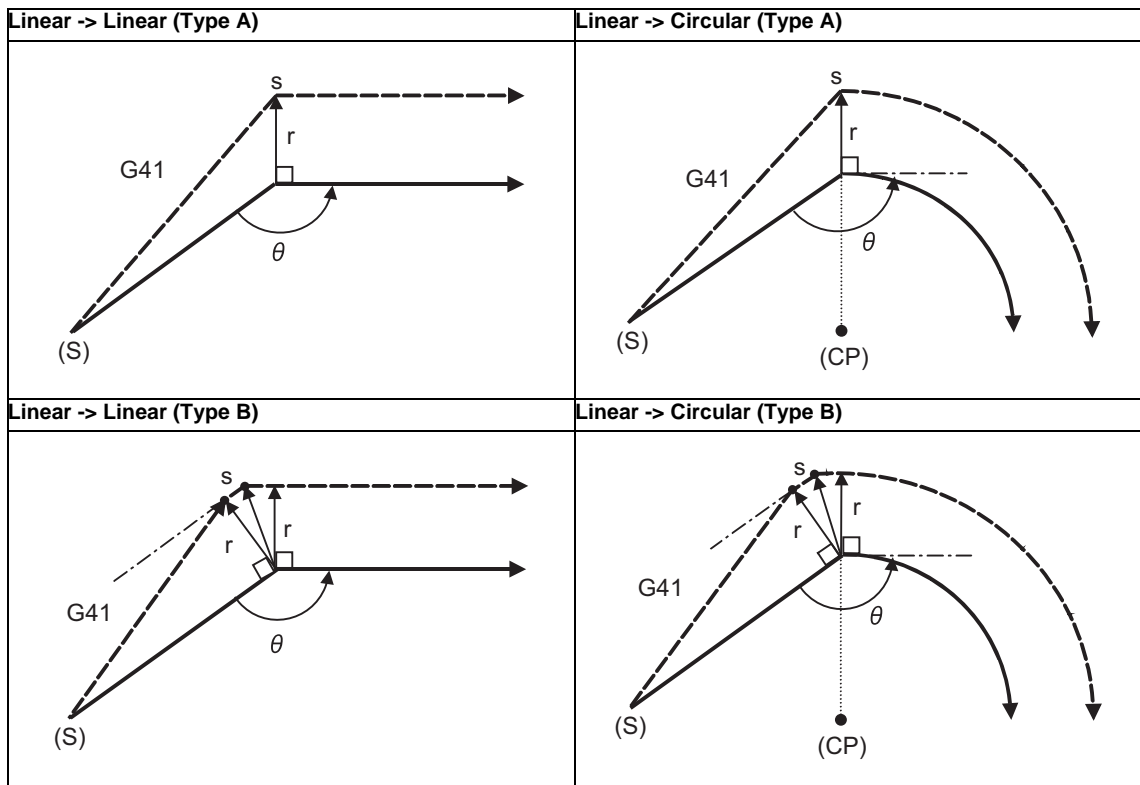
(S) Start point

(CP) Center of circular

r : Compensation amount

s : Stop point with single block

(2) Machining an outside corner (obtuse angle) [$90^\circ \leq \theta < 180^\circ$]

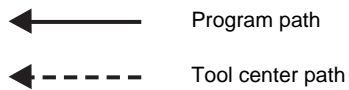


(S) Start point

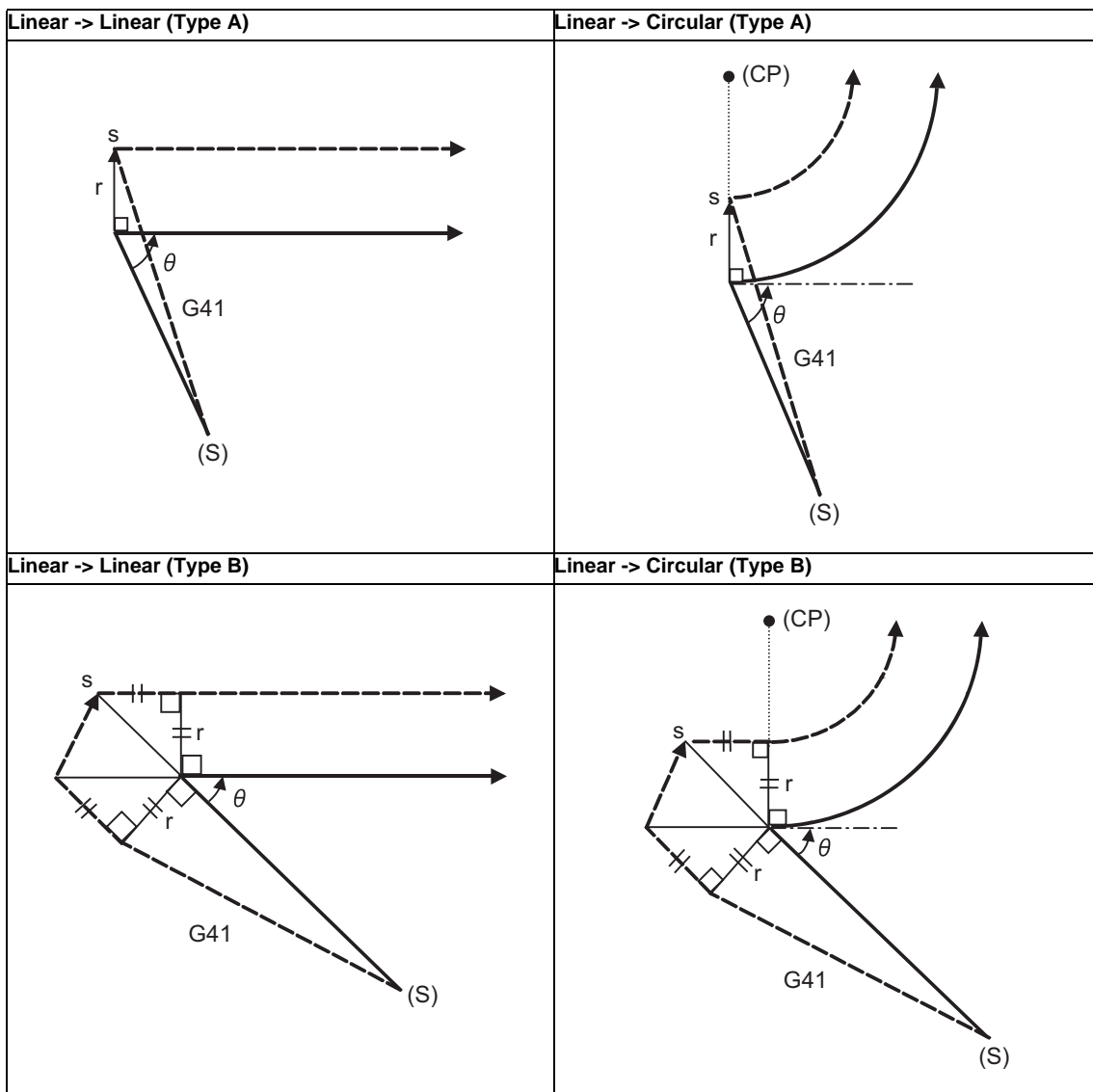
(CP) Center of circular

r : Compensation amount

s : Stop point with single block



(3) Machining an outside corner (acute angle) [$\theta < 90^\circ$]

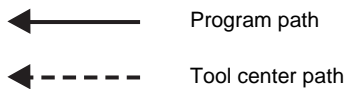


(S) Start point

(CP) Center of circular

r : Compensation amount

s : Stop point with single block



(Note 1) If there is no axis movement command in the same block as G41 or G42, compensation is performed perpendicularly to the next block's direction.

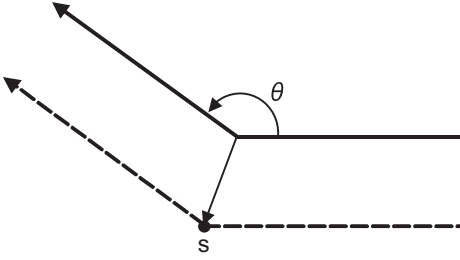
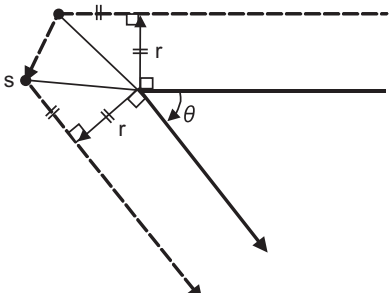
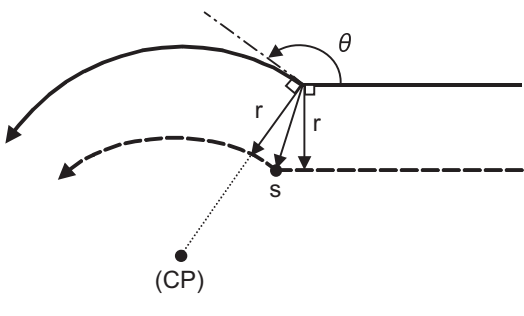
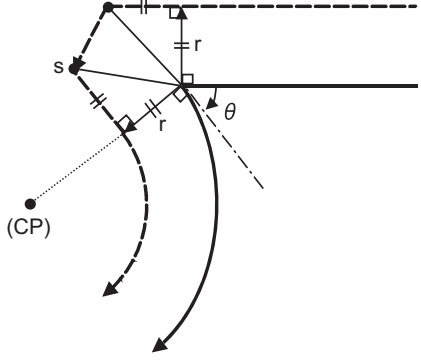
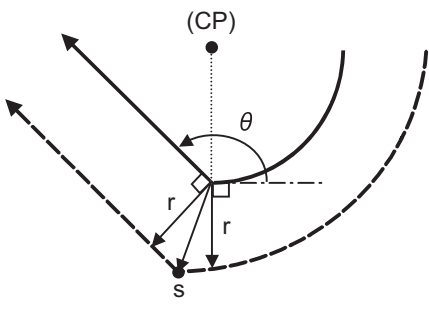
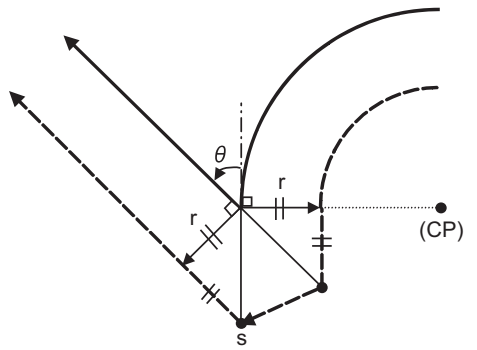
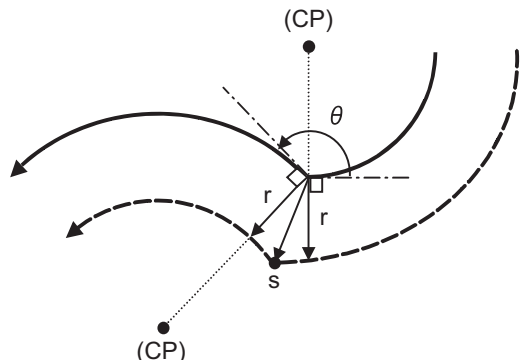
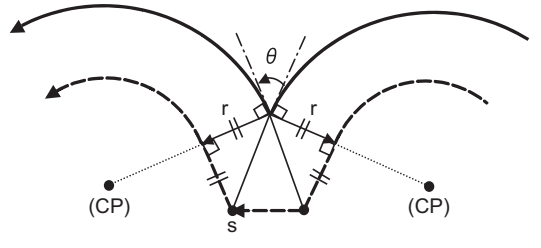
Operation in compensation mode

Calculate the tool center path from the linear line/circular arc to perform compensation to the program path (G00, G01, G02, G03).

Even if the same compensation command (G41, G42) is issued in the compensation mode, the command will be ignored.

When 4 or more blocks without a movement command are continuously commanded in the compensation mode, overcutting or undercutting will occur.

(1) Machining an outside corner

<p>Linear -> Linear ($90^\circ \leq \theta < 180^\circ$)</p> 	<p>Linear -> Linear ($0^\circ < \theta < 90^\circ$)</p> 
<p>Linear -> Linear ($90^\circ \leq \theta < 180^\circ$)</p> 	<p>Linear -> Linear ($0^\circ < \theta < 90^\circ$)</p> 
<p>Circular -> Linear ($90^\circ \leq \theta < 180^\circ$)</p> 	<p>Circular -> Linear ($0^\circ < \theta < 90^\circ$)</p> 
<p>Circular -> Circular ($90^\circ \leq \theta < 180^\circ$)</p> 	<p>Circular -> Circular ($0^\circ < \theta < 90^\circ$)</p> 

(CP) Center of circular

r : Compensation amount

s : Stop point with single block

← Program path

← Nose R center path

(2) Machining an inside corner

<p>Linear -> Linear (Obtuse angle)</p>	<p>Linear -> Linear (Acute angle)</p>
<p>Linear -> Circular (Obtuse angle)</p>	<p>Linear -> Circular (Acute angle)</p>
<p>Circular -> Linear (Obtuse angle)</p>	<p>Circular -> Linear (Acute angle)</p>
<p>Circular -> Circular (Obtuse angle)</p>	<p>Circular -> Circular (Acute angle)</p>

(CP) Center of circular

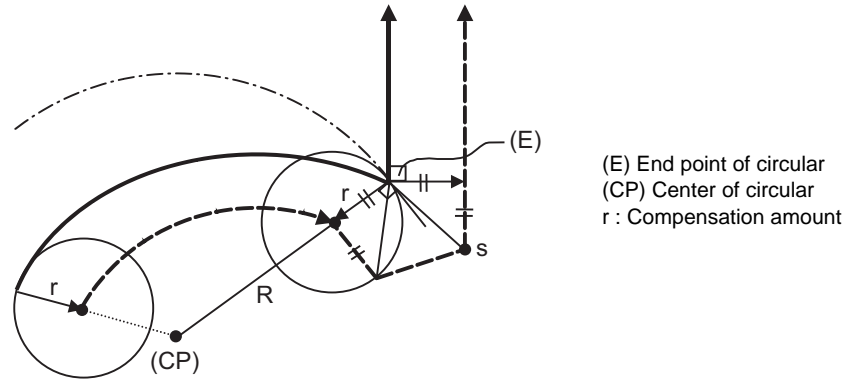
r : Compensation amount

s : Stop point with single block

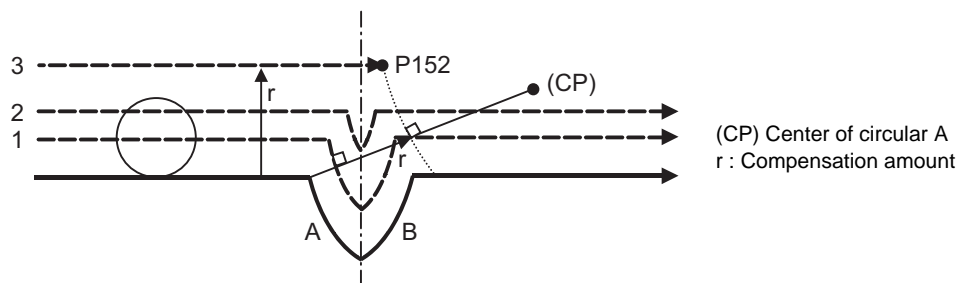
← Program path

← Nose R center path

- (3) When the circular end point is not on the circular
Spiral circular command ... The area from the arc start point to the end point is interpolated as a spiral arc.
Normal circular command ... If the error after compensation is within the parameter value, it is interpolated as a spiral arc.



- (4) When the inner intersection point does not exist
In cases like the figure below, the intersection point of circulars A and B may not exist depending on the compensation amount.
In such cases, program error (P152) appears and the tool stops at the end point of the previous block.
In the pattern 1 and 2 in this figure, machining is possible because compensation amount r is small.
In pattern 3, compensation r is so large that an intersection does not exist and program error (P152) will occur.



← Program path

← Nose R center path

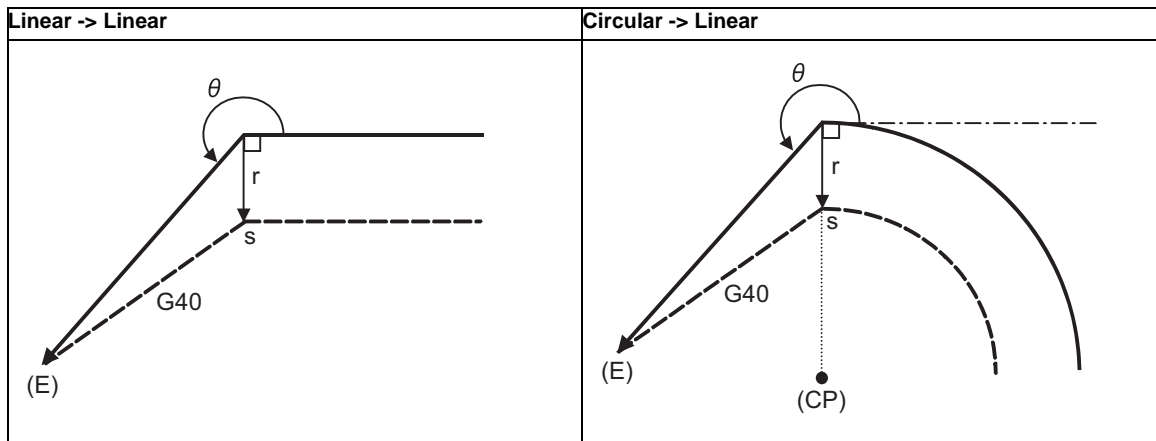
Tool radius compensation cancel

If either of the following conditions is met in the tool radius compensation mode, the compensation will be canceled. However, this must be any movement command except a circular command.
 If the compensation is canceled by a circular command, program error (P151) will occur.

- (1) The G40 command has been executed.
 The cancel mode is established once the compensation cancel command has been read, 5-block pre-reading is suspended and 1-block pre-reading will be operated.

Tool radius compensation cancel operation

- (1) Machining an inside corner

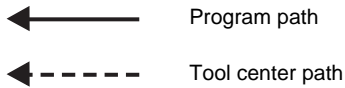


(E) End point

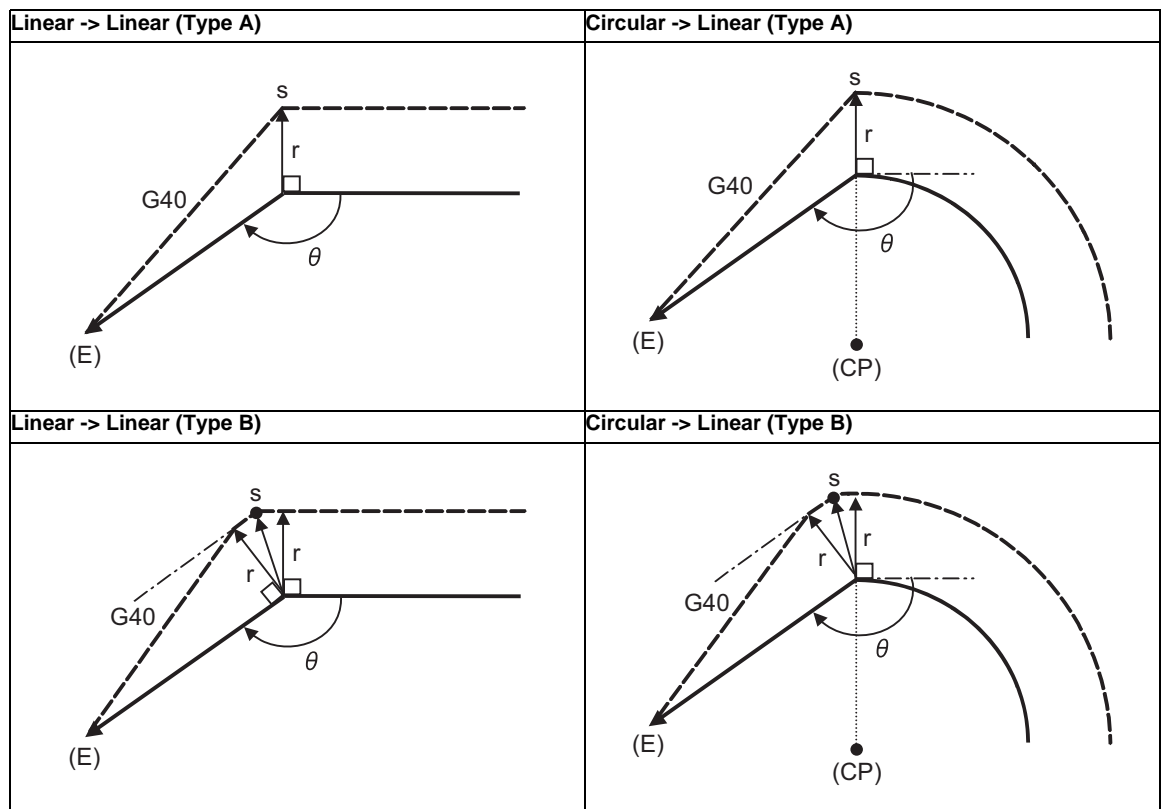
(CP) Center of circular

r : Compensation amount

s : Stop point with single block



(2) Machining an outside corner (obtuse angle) [$90^\circ \leq \theta < 180^\circ$]

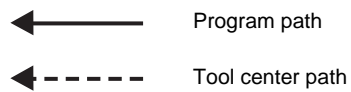


(E) End point

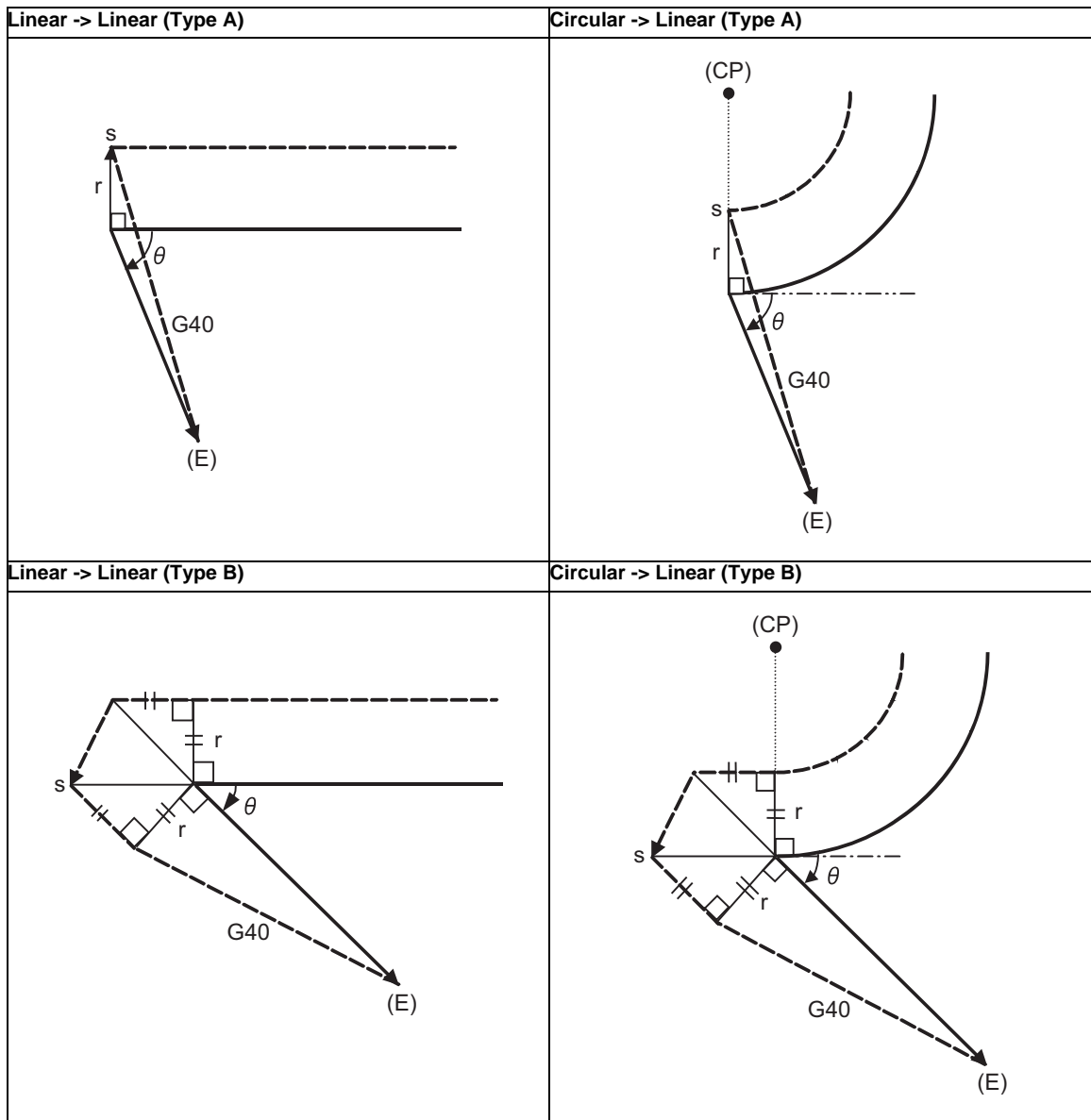
(CP) Center of circular

r : Compensation amount

s : Stop point with single block



(3) Machining an outside corner (acute angle) [$\theta < 90^\circ$]

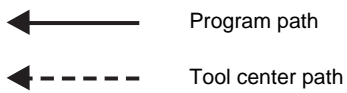


(E) End point

(CP) Center of circular

r : Compensation amount

s : Stop point with single block



Changing the compensation direction during tool radius compensation

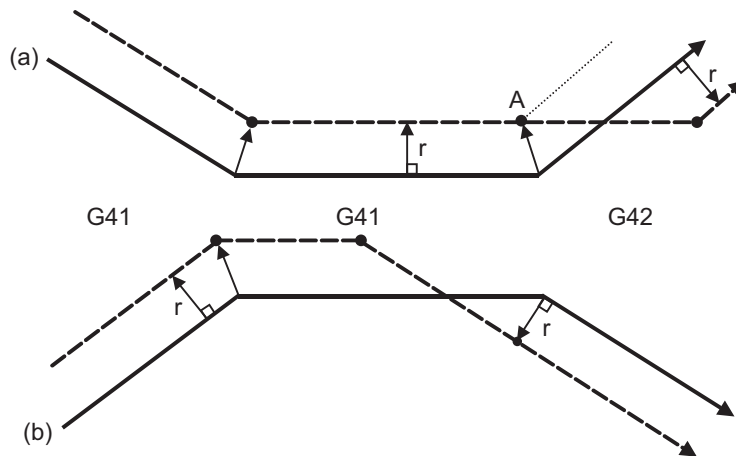
G code	Compensation direction
G41	Left-side compensation
G42	Right-side compensation

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.

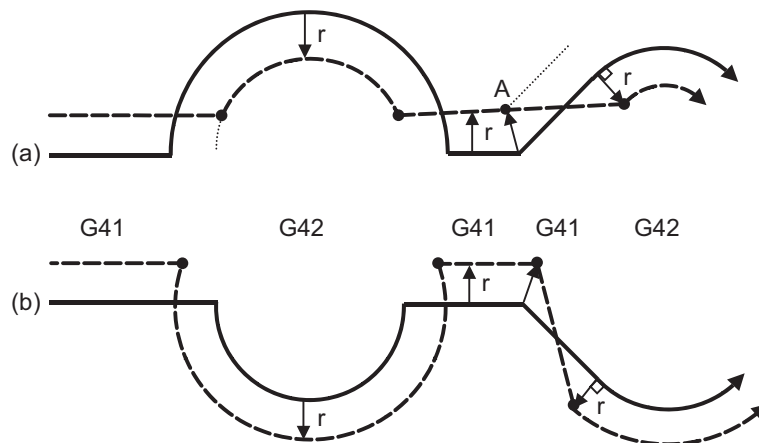
(1) Linear -> Linear

- (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 an intersection (A) at the change of compensation direction
- (b) When there is no intersection at the change of compensation direction

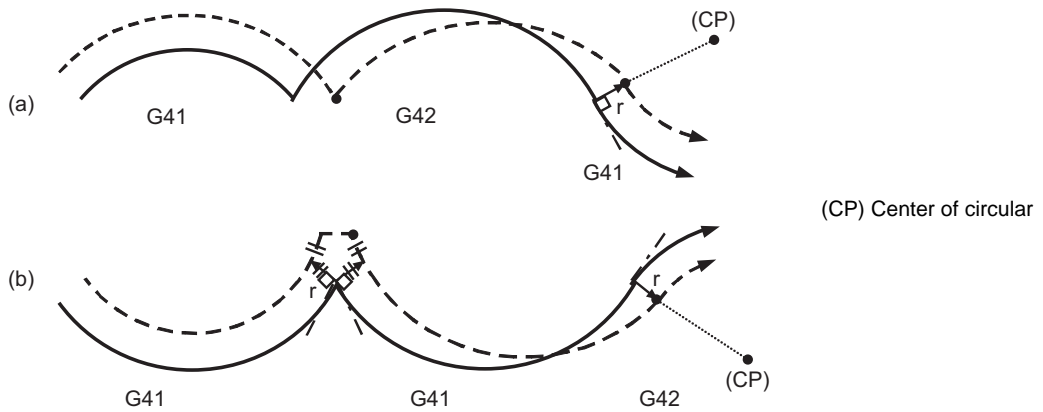


← Program path

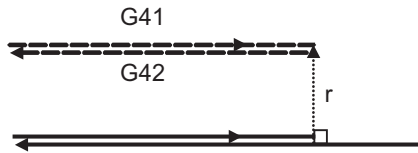
← Nose R center path

(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



(4) Linear return

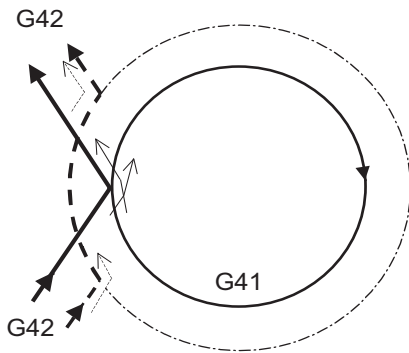


(5) Arc exceeding 360° due to compensation method

In the cases below, the arc may exceed 360°.

- a. Changing the compensation direction by switching between G41/G42.

If the arc exceeds 360°, compensation will be performed as shown in the figure and uncut section will be left.



```

:
G42 G01 X_ Y_;
G41 G02 X_ Y_ I_ J_;
G42 G01 X_ Y_;
:

```



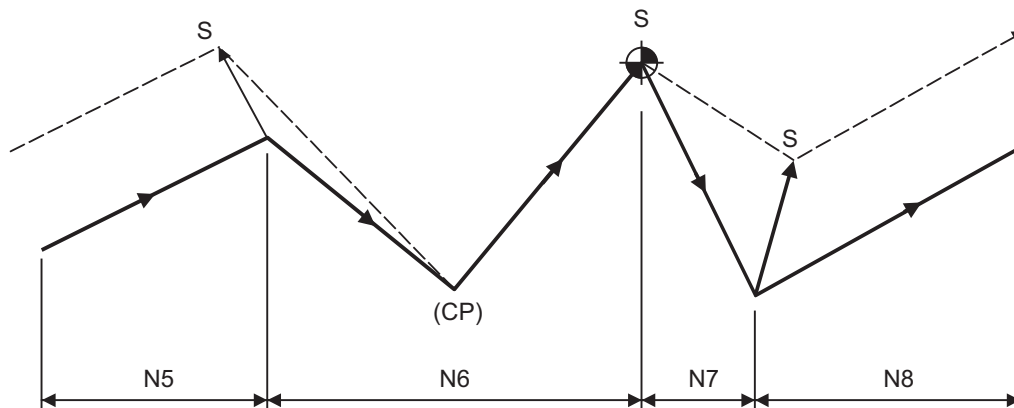
Command for eliminating compensation vectors temporarily

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 point return command

Compensation vector temporarily becomes 0 at the intermediate point (Reference point if there is no intermediate point).



(G41) :

N5 G01 X60. Y30. ;

N6 G28 X50. Y-40. ;

N7 X30. Y-60. ;

N8 X70. Y40. ;

:

(CP) Intermediate Point

(2) The compensation vector will be eliminated temporarily with the G53 command (Basic machine coordinate system selection).

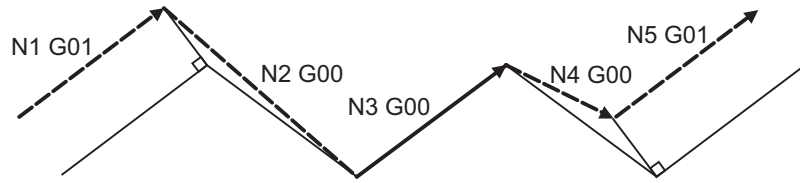
(Note 1) The compensation vectors do not change with the coordinate system setting (G92) command.

(3) Positioning (G00) commands

When G00 command is issued, the tool radius compensation is temporarily canceled.

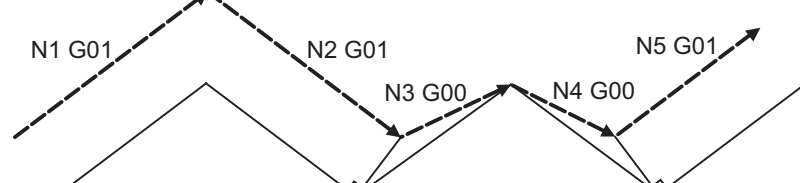
```

:
N1 G01 X_Y_F_;
N2 G00 X_Y_;
N3 G00 X_Y_;
N4 G00 X_Y_;
N5 G01 X_Y_;
:
    
```



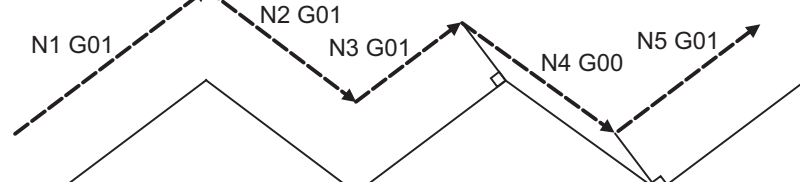
```

:
N1 G01 X_Y_F_;
N2 G01 X_Y_;
N3 G00 X_Y_;
N4 G00 X_Y_;
N5 G01 X_Y_;
:
    
```



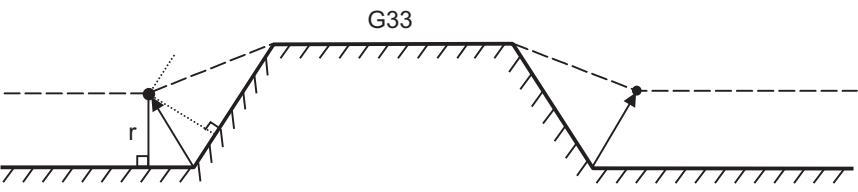
```

:
N1 G01 X_Y_F_;
N2 G01 X_Y_;
N3 G01 X_Y_;
N4 G00 X_Y_;
N5 G01 X_Y_;
:
    
```



(4) G33 thread cutting command

Tool nose radius compensation does not apply to the G33 block.



(5) Compound type fixed cycle for turning machining

When a compound type fixed cycle for turning machining I command (G70, G71, G72, G73) is issued, the tool nose radius compensation will temporarily be canceled, since the finished shape already includes the compensation amount and, upon completion of the cutting, operation will automatically return to the compensation mode.

Blocks without movement and M commands inhibiting pre-reading

The following blocks are known as blocks without movement.

M03 ;	M command
S12 ;	S command
T45 ;	T command
G04 X500 ;	Dwell
G22 X200. Y150. Z100. ;	Machining prohibited region setting
G10 L10 P01 R50 ;	Compensation amount setting
G92 X600. Y400. Z500. ;	
(G17)Z40 ;	Movement outside the compensation plane
G90 ;	G code only
G91 X0 ;	Movement amount 0

M00, M01, M02, and M30 are treated as M codes which prohibit pre-reading.

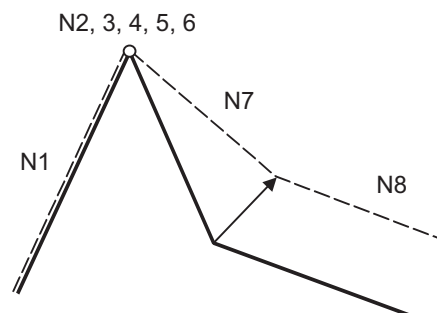
- (1) When commanded at the compensation start
Compensation vector cannot be created when there are four or more successive blocks without movement, or when pre-reading prohibiting M command is issued.

```

N1 X30.Y60. ;
N2 G41 ;
N3 G4 X1000 ;
N4 F100 ;
N5 S500 ;
N6 M3 ;
N7 X20.Y-50. ;
N8 X50.Y-20. ;

```

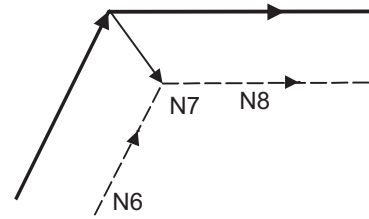
} Block without movement



(2) When commanded during the compensation mode

Compensation vector will be created as normal when there are not four or more successive blocks without movement, or when pre-reading prohibiting M command is not issued.

```
N6 G91X100. Y200. ;
N7 G04 X P1000 ;    ... Block without movement
N8 X200. ;
```

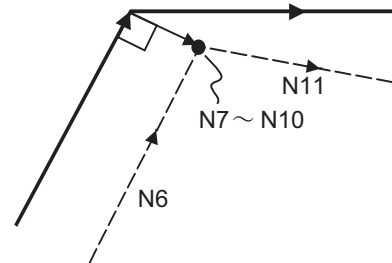


Block N7 is executed at N7 in the figure.

When four or more blocks without movement follow in succession or if there is a pre-reading inhibiting M code, the compensation vectors are created perpendicularly at the end point of the previous block. This may cause cutting.

```
N6 X100. Y200. ;
N7 G4 X1000 ;
N8 F100 ;
N9 S500 ;
N10 M4 ;
N11 X100. ;
```

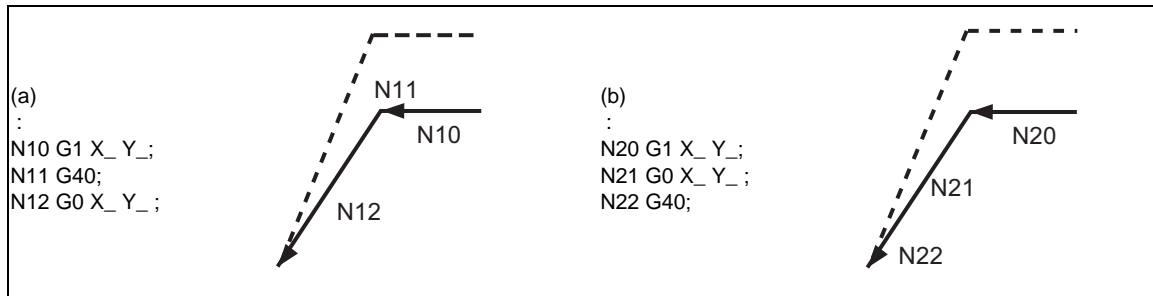
} Block without movement



(3) Compensation cancel alone

(a) G1 precedes the block containing G40: Compensation is canceled by a movement command following G40.

(b) G0 precedes the block containing G40: Compensation is canceled by a G0 command before G40.



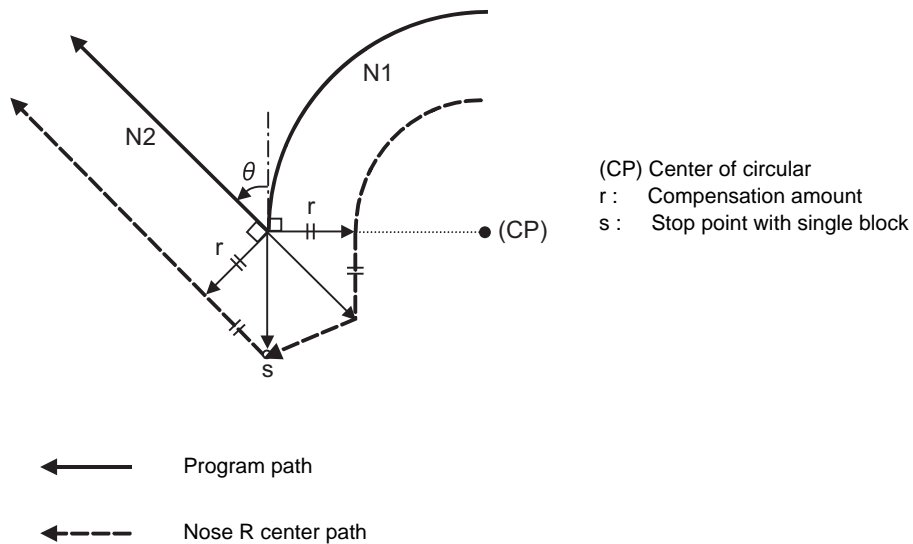
(Note) In program (a), if G13.1 is commanded after G40 without a movement command, cancellation is done at G13.1 block.

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. Consequently, in the single block mode, the previous block + corner movement will be executed as a single block and the remaining joint movement + following block will be executed as a single block in the next operation.



6.8.11.2 Interference Check



Function and purpose

When tool radius is larger than the program path, a tool, compensated 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.

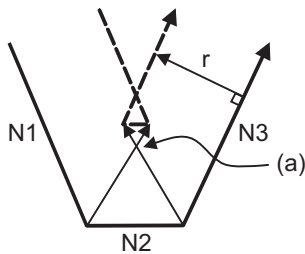
	Function	Parameter		Operation
		#8102 COLL. ALM OFF	#8103 COLL. CHK OFF	
(1)	Interference check alarm function	0	0	Operation stops with a program error(P153) before executing a block which will cause cutting.
(2)	Interference check avoidance function	1	0	The tool path is changed to prevent cutting from occurring. Operation stops with a program error(P153) when it is unable to change.
(3)	Interference check invalid function	0/1	1	Cutting continues as is, even if the workpiece is cut into. Use in the fine segment program.



Detailed description

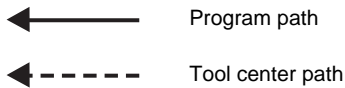
Conditions viewed as interference

When there is a movement command in three of the five pre-read blocks, and if the compensation calculation vectors which are created at the contacts of movement commands intersect each other, it will be viewed as interference.



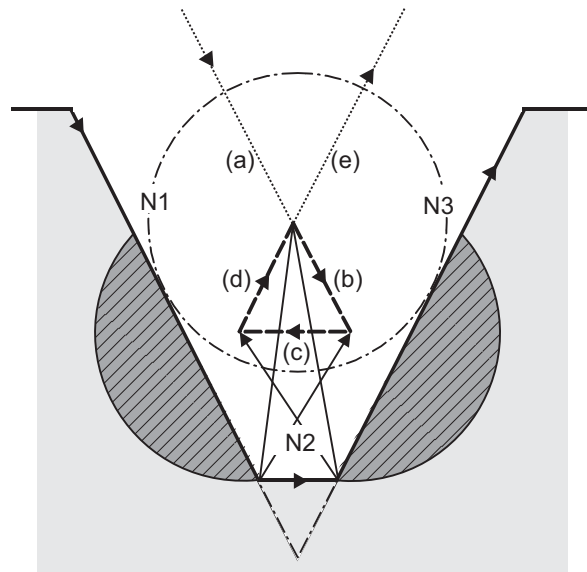
r : Compensation amount

(a) Vectors intersect



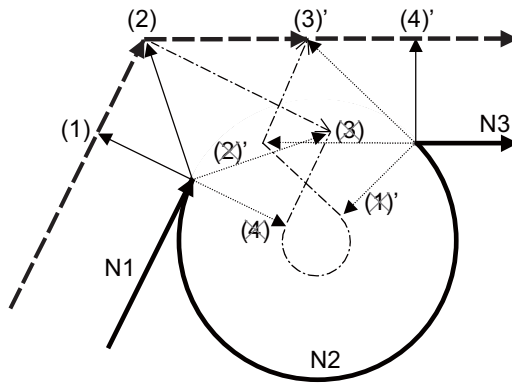
(Example 1) When operating a program including a short segment with a tool with a large radius
Cutting will occur in the shaded section.

```
(G41)
N1 G1 X50. Y-100. ;
N2 X70. Y-100. ;
N3 X120. Y0 ;
```



- (1) With alarm function
An alarm is given before N1 is executed. The buffer correction function can thus be used to change N1 to the following, enabling machining to continue: N1 G1 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.

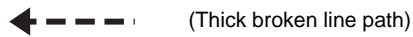


Interference check processing

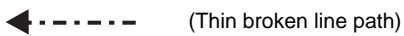
Vectors (1) (4)' check	-->	No interference	
↓			
Vectors (2) (3)' check	-->	No interference	
↓			
Vectors (3) (2)' check	-->	Interference	--> Erase vectors (3) (2)'
			↓
			Erase vectors (4) (1)'

(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 which connects these vectors, as the interference avoidance path.

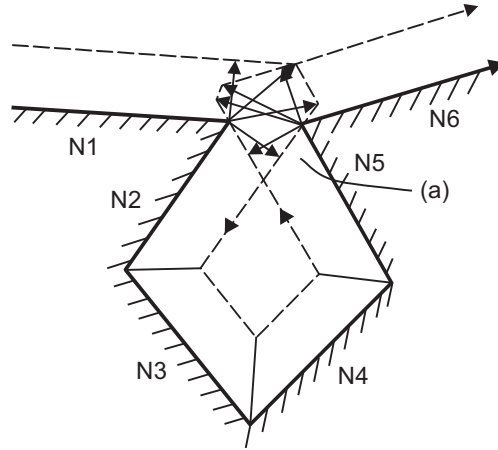


(3) With interference check invalid function
The tool center path will follow the path which connects (1), (2), (3), (4), (1)', (2)', (3)', (4)', as the interference avoidance path while cutting.

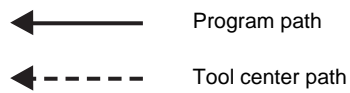


When interference check cannot be executed

- (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 do not have movement)
- (2) When there is an interference following the fourth movement block

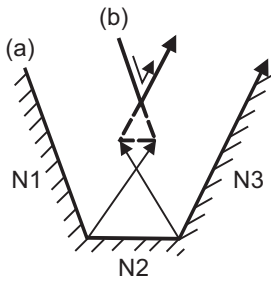


(a) Interference check is not possible

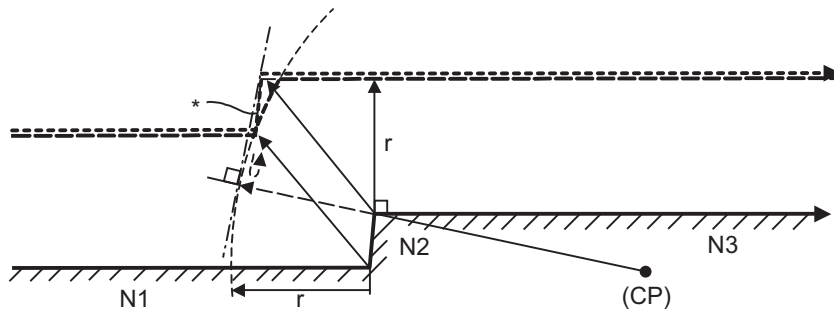
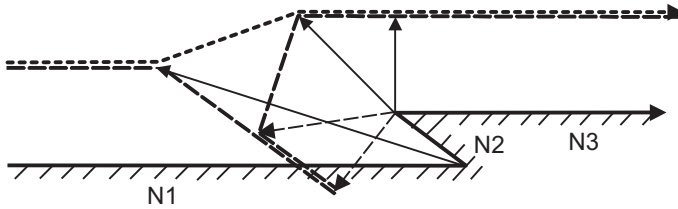


Operation when interference avoidance function is valid

The movement will be as shown below when the interference avoidance check is valid.

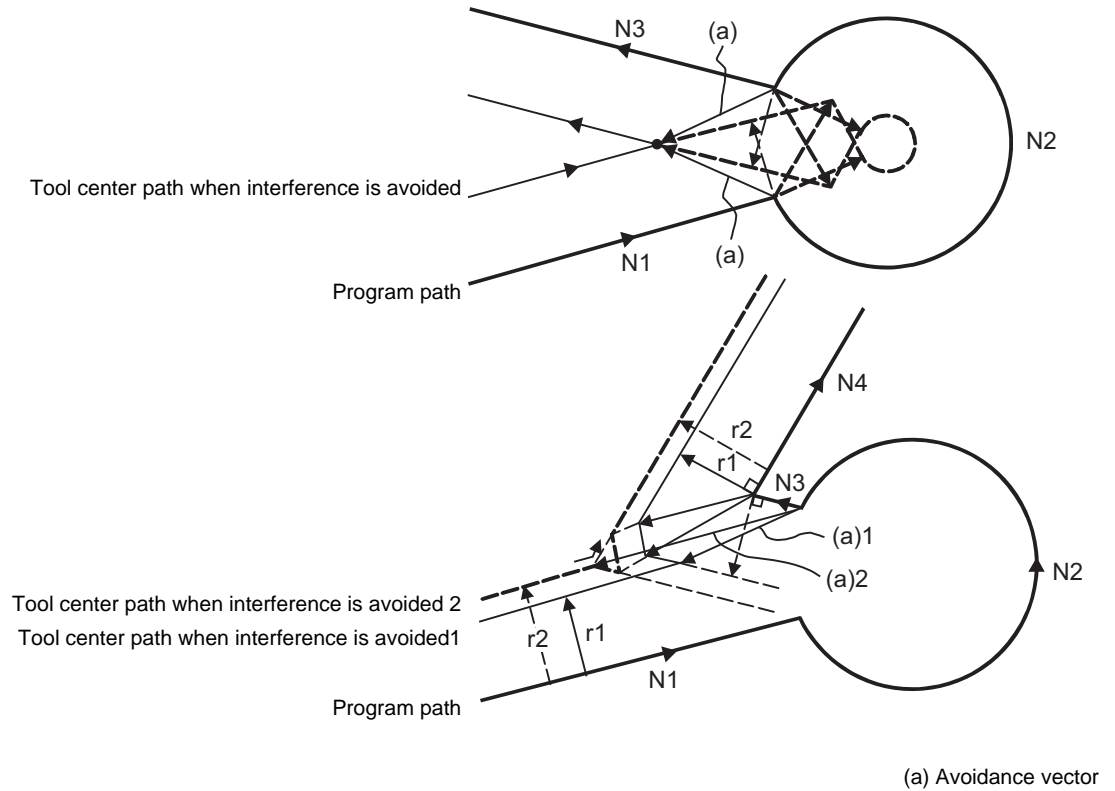


(a) Program path
(b) Tool center path

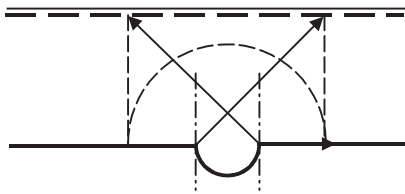


- ← Program path
- ← - - - - - Tool center path without interference check
- ← - - - - * - - - - Tool center path when interference is avoided (*: Linear movement)
- ← Valid vector
- ← - - - - - Invalid vector

If all of the line vectors for the interference avoidance are deleted, create a new avoidance vector as shown in below to avoid the interference.



In the case of the figure below, the groove will be left uncut.



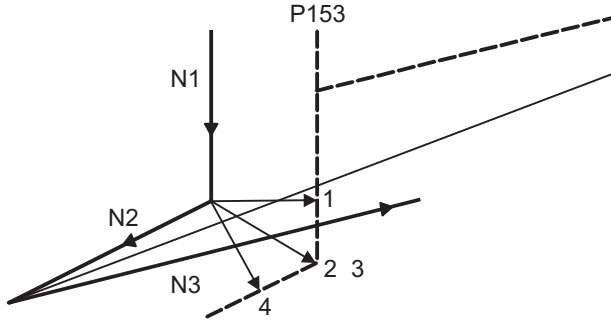
- ← Program path
- ← Tool center path
- ← Tool center path when interference is avoided

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 the vectors at the end of its own block have been deleted.

When, as shown in the figure below, 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.

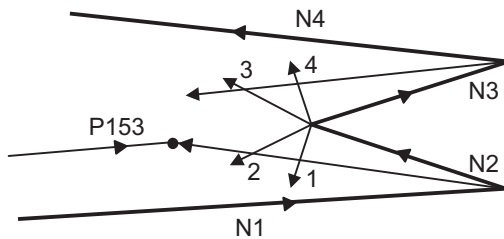


- (2) When the interference check avoidance function has been selected

(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.

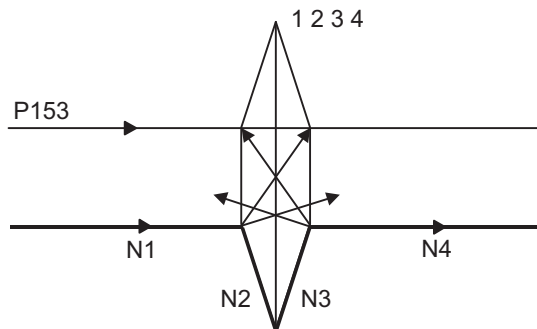
When, in the figure below, the N2 interference check is conducted, the N2 end point vectors are all deleted but the N3 end point vectors are regarded as valid.

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.

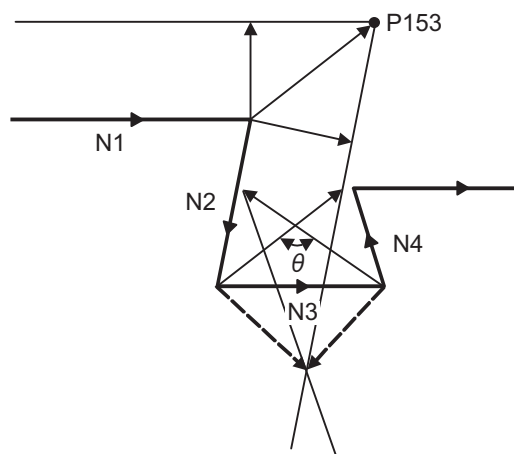
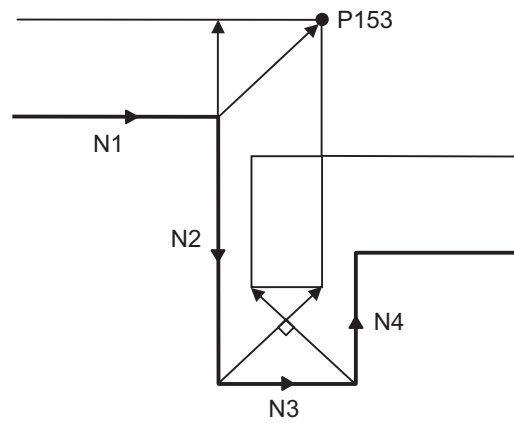
Program error (P153) now occurs before executing N1 and the operation stops.



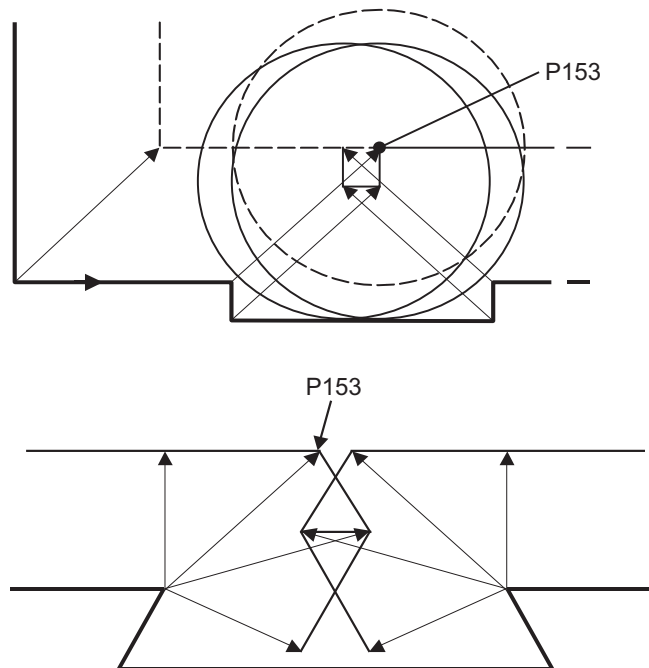
(Example 2) When avoidance vectors cannot be created

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, 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.

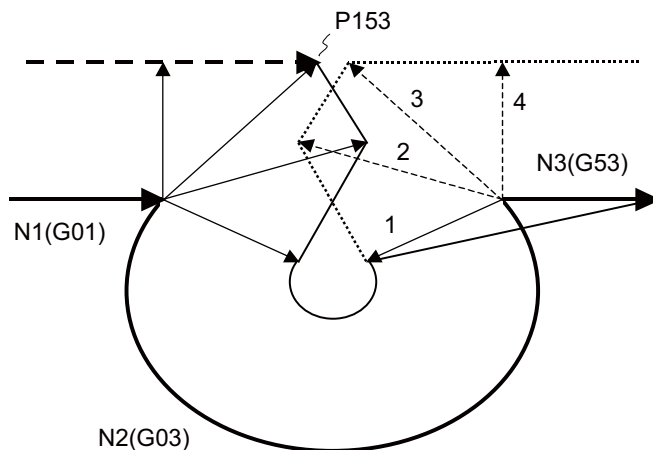


(Example 4) Command for eliminating compensation vectors temporarily

Interference check will be executed even at the end point of the block right before the command to eliminate compensation vectors temporarily, as well as when the command is not issued. It may be regarded as an interference even if there is actually no interference. If regarded as an interference, program error (P153) will occur.

In the figure below, only vector 1 is left as an end point vector in N2 because of the N3 G53 command to temporarily eliminate compensation vectors. However, the interference check will still be conducted to vector 1 to 4 and an interference will be detected.

Program error (P153) now occurs at the end point of the previous block and the operation stops.



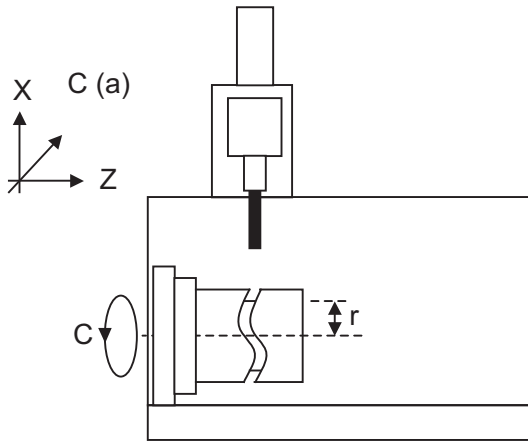
- ← Program path
- ← - - - - Tool center path
- Tool center path when the interference check is invalid
- Tool center path when the interference check is invalid in N3 due to a command (G01 etc.) not to eliminate compensation vectors.
- ← Valid vector
- ← - - - - Invalid vector (Invalid, however, subject to interference check)

6.9 Cylindrical Interpolation ; G07.1 (only 6 and 7 in G code list)



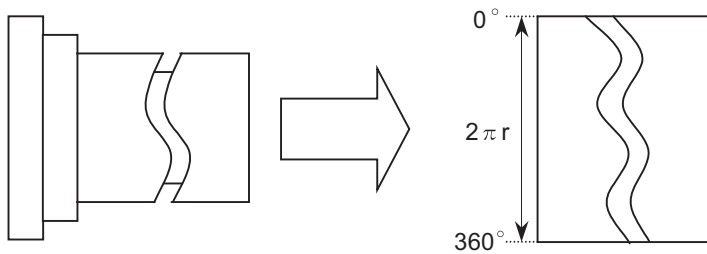
Function and purpose

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 movement in the cylindrical coordinates to conduct a contour control when machining.



(a) Hypothetical C axis

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 rotation axis and its orthogonal axis, grooves and other shapes can be machined on the side of the cylinder.



Command format

G07.1 C__ ; ... Cylindrical interpolation mode start/cancel

C	Cylinder radius value (When rotation axis name is "C") - Radius value \neq 0: Cylindrical interpolation mode start - Radius value = 0: Cylindrical interpolation mode cancel
---	--



Detailed description

- (1) The coordinate commands in the interval from the start to cancellation of the cylindrical interpolation mode will be the cylindrical coordinate system.

G07.1 C	Cylinder radius value;	Cylindrical interpolation mode start (Cylindrical interpolation will start)
:	:	(The coordinate commands in this interval will be the cylindrical coordinate system)
:	:	
:	:	
G07.1 C0 ;		Cylindrical interpolation mode cancel (Cylindrical interpolation will be canceled)

- (2) G107 can be used instead of G07.1.
 (3) Command G07.1 is an independent block. A program error (P33) will occur if this command is issued in the same block as other G codes.
 (4) Program the rotation axis with an angle degree.
 (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 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 segment feed rate on the developed cylinder by F. F is in mm/min or inch/min unit.

Cylindrical interpolation accuracy

In the cylindrical interpolation mode, the movement amount of the rotation 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.

Related parameters

- #1516 mill_ax (Milling axis name)
- #8111 Milling Radius
- #1267 ext03/bit0 (G code type)
- #1270 ext06/bit7 (Handling of C axis coordinate during cylindrical interpolation)

Plane selection

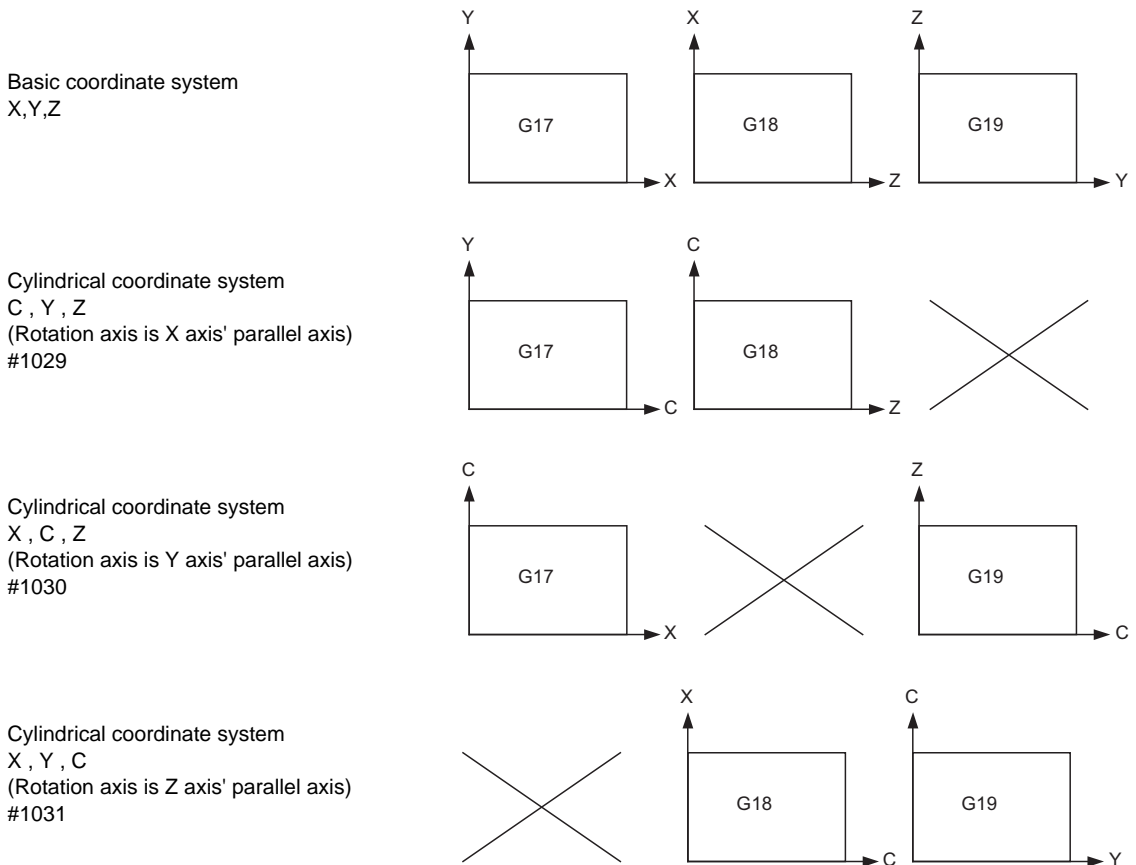
The axis used for cylindrical interpolation must be set with the plane selection command. (Note)
 Use parameters (#1029, #1030 and #1031) to set which parallel axis corresponds to the rotation axis.
 The circular interpolation and tool radius compensation, etc., can be designated on that plane.

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.

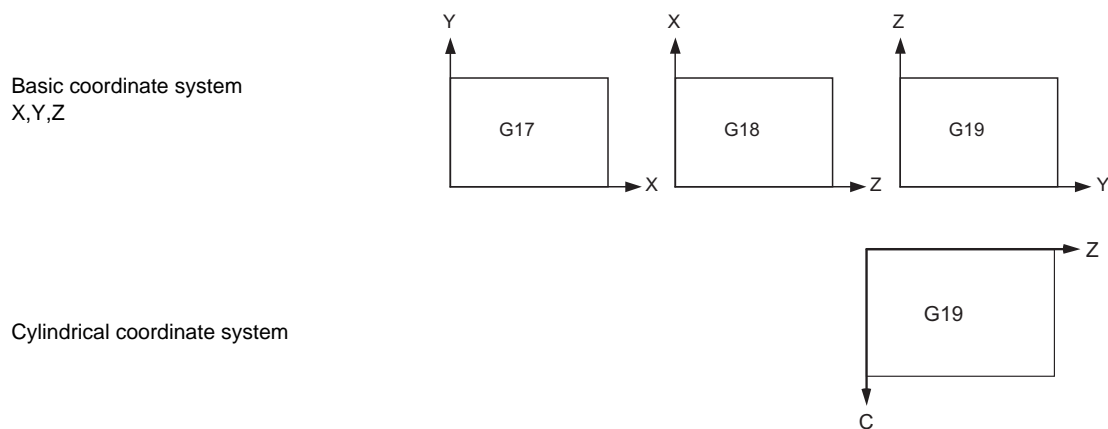
When the axis address is commanded while selecting the plane, the axes will travel. To prevent the axes from traveling, command the address with the incremental value.

(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
```



(Note) Depending on the model or version, the Z-C plane (Y-Z cylindrical plane) will be automatically selected with G07.1 and G19.
The circular interpolation and tool radius compensation, etc., can be designated on that plane.





Program example

<Program>

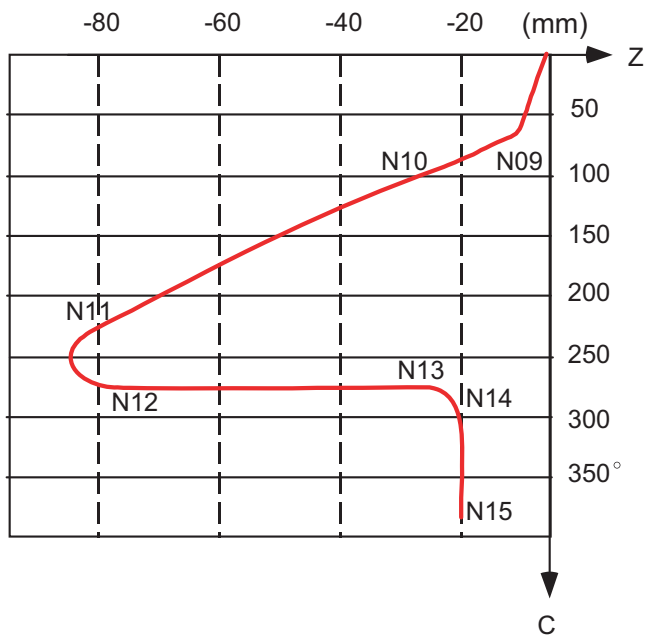
```

N01 G28 XZC ;
N02 T020 ;
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 G0 X100. Z100. ;
N20 M25 ;
N21 M30 ;
    
```

<Parameter>

```

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





Relation with other functions

Circular interpolation

- (1) Circular interpolation between the rotation axis and linear axis is possible during the cylindrical interpolation mode.
- (2) An R specification command can be issued with circular interpolation. (I, J and K cannot be designated.)

Tool 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 is canceled, the position of the axis in the G07.1 command block is interpreted as the position applied after the tool radius compensation is canceled and the following operations are performed.

Cutting feed per minute (asynchronous feed)

- (1) The feed per minute (asynchronous) mode is forcibly set when the cylindrical interpolation mode is started.
- (2) When the cylindrical interpolation mode is canceled, the feed per revolution (synchronous) will return to the state before the cylindrical interpolation mode was started.

Constant surface speed control

- (1) A program error (P485) will occur if G07.1 is commanded in the constant surface speed control mode (G96).

Miscellaneous functions

- (1) The miscellaneous function (M) and 2nd miscellaneous function 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.
- (3) Issue the T command before starting cylindrical interpolation. Program error (P481) will occur if T command is issued in the cylindrical interpolation mode.

```
:  
:  
T1212 ;          T command before cylindrical interpolation -> Valid  
G0 X100. Z0. ;  
G19 Z C ;  
G07.1 C100. ;  
:  
T1200 ;          ... T command in cylindrical interpolation mode -> Program error  
:  
G07.1 C0 ;
```

- (4) 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:
 - Machine coordinate will not change even if G12.1 is executed.
 - The workpiece coordinate will change to that of the post tool length compensation when G07.1 is executed.
 (Even if the cylindrical interpolation is canceled, this workpiece coordinate will not be canceled.)

F command during cylindrical interpolation

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

- (1) When G94 is commanded just before G07.1
 If there is no F command in the cylindrical interpolation, the previous F command feedrate will be used. After the cylindrical interpolation mode is canceled, the F command feedrate set at the start of the cylindrical interpolation mode or the last F command feedrate set during cylindrical interpolation will continue to be the feedrate.
- (2) When G95 is commanded just before G07.1
 The previous F command feedrate cannot be used during cylindrical interpolation, thus a new F command must be issued. After the cylindrical interpolation mode is canceled, the feedrate will return to the state before the cylindrical interpolation mode was started.

When there is no F command in G07.1

Previous mode	No F command	After G07.1 is canceled
G94	Previous F is used	<-
G95	Program error (P62)	F just before G07.1 is used

When F is commanded in G07.1

Previous mode	With F command	After G07.1 is canceled
G94	Commanded F is used	<-
G95	Commanded F is used *1	F just before G07.1 is used
*1) Moves with the feed per minute command during G07.1.		



Restrictions and precautions

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

G code	Details
G00	Positioning
G01	Linear interpolation
G02	Circular interpolation (CW)
G03	Circular interpolation (CWW)
G04	Dwell
G09	Exact stop check
G22/23	Chuck barrier ON/OFF
G40-42	Tool radius compensation
G50.2	Polygon machining mode cancel
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)
G79-89	Fixed cycle for drilling
G90/91	Absolute/incremental value command
G94	Asynchronous feed
G98	Hole drilling cycle initial return
G99	Hole drilling cycle R point return

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

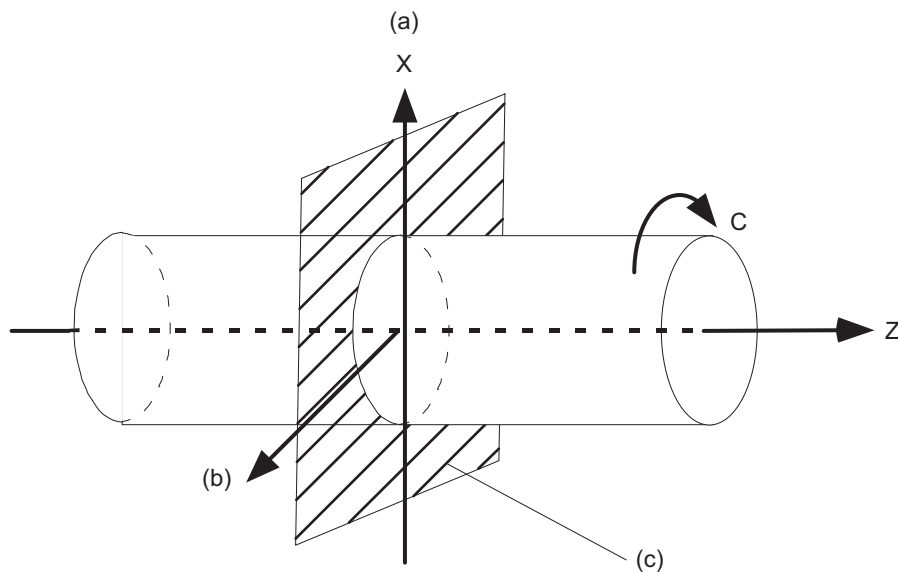
- (2) The cylindrical interpolation mode is canceled when the power is turned ON or reset.
- (3) A program error (P484) will occur if any axis commanded during cylindrical interpolation has not completed the reference position return.
- (4) Tool radius compensation must be canceled before canceling the cylindrical interpolation mode.
- (5) When the cylindrical interpolation mode is canceled and switched to the cutting mode, the plane selected before the cylindrical interpolation will be restored.
- (6) Program cannot be restarted (program restart) when the block is in the cylindrical interpolation.
- (7) A program error (P486) will occur if the cylindrical interpolation command is issued during the mirror image.
- (8) When the cylindrical interpolation mode is started or canceled, the deceleration check is performed.
- (9) A program error (P481) will occur if the cylindrical interpolation or the pole coordinate interpolation is commanded during the cylindrical interpolation mode.
- (10) The G84, G88 synchronous tapping cycles cannot be used in the cylindrical interpolation mode. The asynchronous tapping can be used during cylindrical interpolation mode; however, the synchronous tapping must not be commanded.

6.10 Polar Coordinate Interpolation ; G12.1,G13.1/G112,G113 (Only 6, 7 in G code list)



Function and purpose

This function converts the commands programmed with the orthogonal coordinate axis into linear axis movement (tool movement) and rotation axis movement (workpiece rotation), and controls the contour. The plane that uses the linear axis as the plane's 1st orthogonal axis, and the intersecting hypothetical axis as the plane's 2nd axis (hereafter "polar coordinate interpolation plane") is selected. Polar coordinate interpolation is carried out on this plane. The workpiece coordinate system zero point is used as the coordinate system zero point during polar coordinate interpolation.



- (a) Linear axis
- (b) Rotation axis (hypothetical axis)
- (c) Polar coordinate interpolation plane (G17 plane)

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



Command format

G12.1; ... Polar coordinate interpolation mode start

G13.1; ... Polar coordinate interpolation mode cancel



Detailed description

- (1) The coordinate commands in the interval from the start to cancellation of the polar coordinate interpolation mode will be the polar coordinate interpolation.

```
G12.1;    Polar coordinate interpolation mode start (Polar coordinate interpolation will start)
:
:          (The coordinate commands in this interval will be the polar coordinate interpolation)
:
G13.1;    Polar coordinate interpolation mode cancel (Polar coordinate interpolation is canceled)
```

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

Plane selection

The linear axis and rotation axis used for polar coordinate interpolation must be set beforehand with parameters.

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

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

- (2) A program error (P485) will occur if the plane selection command (G16 to G19) is issued during the polar coordinate interpolation mode.

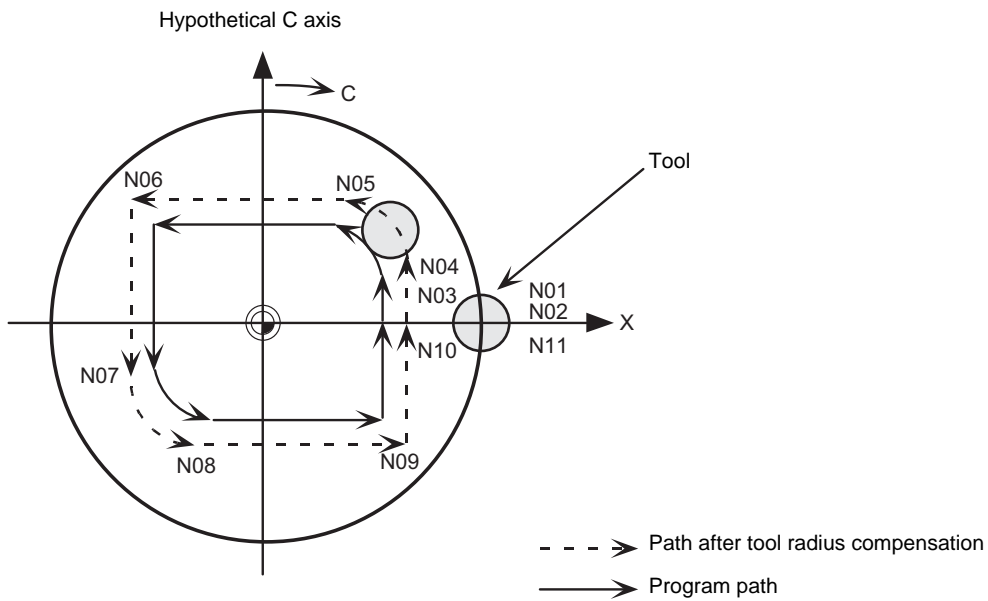
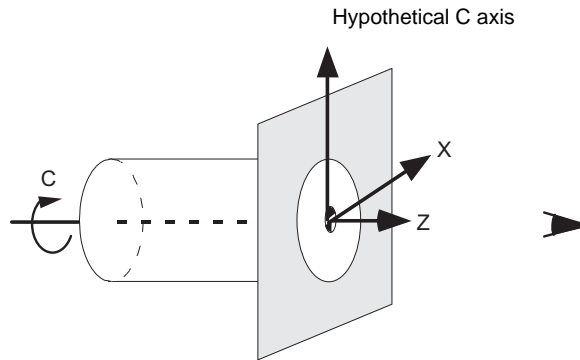
(Note) Depending on the model or version, parameter (#1533) may not be provided. In this case, the operation will be the same as when the parameter (#1533) is blank (no setting).

Related parameters

#1516 mill_ax (Milling axis name)
 #1517 mill_c (Milling interpolation hypothetical axis name)
 #8111 Milling Radius



Program example



<Program>

```

:
:
N00 T0101;
:
:
N01 G17 G90 G0 X40.0 C0 Z0;      Setting of start position
N02 G12.1;                      Polar coordinate interpolation mode: Start
N03 G1 G42 X20.0 F2000;         Actual machining start
N04 C10.0;
N05 G3 X10.0 C20.0 R10.0;
N06 G1 X-20.0;                  Shape program
N07 C-10.0;
N08 G3 X-10.0 C-20.0 I10.0 J0;  (Follows orthogonal coordinate values on X-C hypothetical axis plane.)
N09 G1 X20.0;
N10 C0;
N11 G40 X40.0;
N12 G13.1;                      Polar coordinate interpolation mode: Cancel
:
:
M30 ;
    
```



Relation with other functions

Program commands during polar coordinate interpolation

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

Circular interpolation on polar coordinate plane

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

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

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

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

Tool radius compensation

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

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

Cutting asynchronous feed

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

Miscellaneous 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.
- (3) Issue the T command before starting polar coordinate interpolation. Program error (P481) will occur if T command is issued in the polar coordinate interpolation mode.

```

:
T1212 ;          ... T command before polar coordinate interpolation -> Valid
G0 X100. Z0. ;
G12.1;
:
T1200 ;          ... T command in polar coordinate interpolation mode -> Program error
:
G13.1;
    
```

- (4) Complete the tool compensation operation (movement of tool length and wear compensation amount) before executing the polar coordinate interpolation.
 If the tool compensation operation is not completed when the polar coordinate interpolation start command has been issued, the followings will occur:
 - Machine coordinate will not change even if G12.1 is executed.
 - When G12.1 is executed, the workpiece coordinate will changed to that of the post tool length compensation.
 (Even if the polar coordinate interpolation is canceled, this workpiece coordinate will not be canceled.)

F command during polar coordinate interpolation

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

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

If there is no F command in the polar coordinate interpolation, the previous F command feedrate will be used.

After the polar coordinate interpolation mode is canceled, the F command feedrate set at the start of the polar coordinate interpolation mode or the last F command feedrate set during polar coordinate interpolation will continue to be the feedrate.

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

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

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

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

Previous mode	No F command	After G13.1
G94	Previous F is used	<-
G95	Program error (P62)	F just before G12.1 is used

[When F is commanded in G12.1]

Previous mode	With F command	After G13.1
G94	Commanded F is used	<-
G95	Commanded F is used *1	F just before G12.1 is used
*1) Moves with the feed per minute command during G12.1.		

Hole drilling axis in the fixed cycle for drilling command

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

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



Restrictions and precautions

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

G code	Details
G00	Positioning
G01	Linear interpolation
G02	Circular interpolation (CW)
G03	Circular interpolation (CWW)
G04	Dwell
G09	Exact stop check
G22/23	Chuck barrier ON/OFF
G40-42	Tool radius compensation
G61	Exact stop mode
G64	Cutting mode
G65	Macro call (simple call)
G66	Macro modal call (modal call)
G66.1	Macro modal call (block call per macro)
G67	Macro modal call cancel (modal call cancel)
G80-89	Fixed cycle for drilling
G90/91	Absolute/incremental value command
G94	Asynchronous feed
G98	Hole drilling cycle initial return
G99	Hole drilling cycle R point return

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

- (2) Program cannot be restarted (program restart) when the block is in the cylindrical interpolation.
- (3) Before commanding polar coordinate interpolation, set the workpiece coordinate system so that the center of the rotation axis is at the coordinate system zero point. Do not change the coordinate system during the polar coordinate interpolation mode. (G50, G52, G53, relative coordinate reset, G54 to G59, etc.)
- (4) The feedrate during polar coordinate interpolation will be the interpolation speed on the polar coordinate interpolation plane (orthogonal coordinate system).
(The relative speed with the tool will vary according to the polar coordinate conversion.)
When passing near the center of the rotation axis on the polar coordinate interpolation plane (orthogonal coordinate system), the rotation axis side feedrate after polar coordinate interpolation will be very high.
- (5) The axis movement command outside of the plane during polar coordinate interpolation will move unrelated to the polar coordinate interpolation.
- (6) The current position displays during pole coordinate interpolation will all indicate the actual coordinate value. However, the "remaining movement amount" indicates the movement amount on the polar coordinate input plane.
- (7) The polar coordinate interpolation mode is canceled when the power is turned ON or reset.
- (8) A program error (P484) will occur if any axis commanded during polar coordinate interpolation has not completed the reference position return.
- (9) Tool radius compensation must be canceled before canceling the polar coordinate interpolation mode.
- (10) When the polar coordinate interpolation mode is canceled and switched to the cutting mode, the plane selected before the polar coordinate interpolation will be restored.
- (11) A program error (P486) will occur if the polar coordinate interpolation command is issued during the mirror image.
- (12) A program error (P481) will occur if the cylindrical interpolation or the polar coordinate interpolation is commanded during the polar coordinate interpolation mode.
- (13) The G84, G88 synchronous tapping cycles cannot be used in the polar coordinate interpolation mode. The asynchronous tapping can be used during the polar coordinate interpolation mode; however, the synchronous tapping command must not be issued.

6.11 Exponential Interpolation ; G02.3,G03.3



Function and purpose

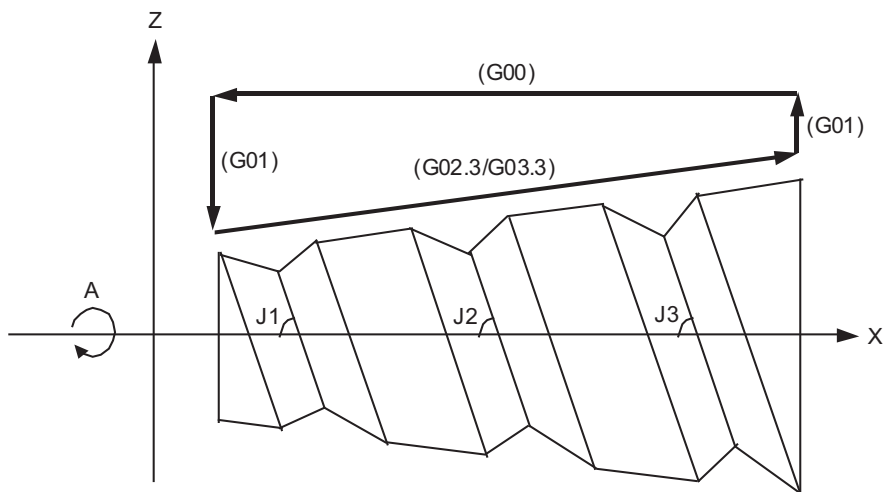
Exponential function interpolation changes the rotation axis into an exponential function shape in respect to the linear axis movement.

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

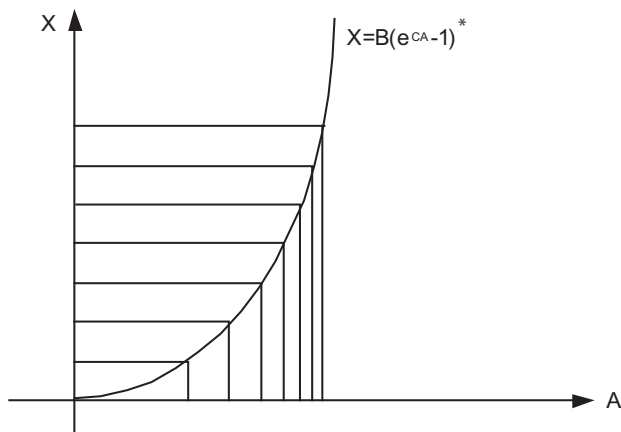


Torsion angle: $J1 = J2 = J3$

A : A axis (rotation axis)

X : X axis (linear axis)

- Relation of linear axis and rotation axis



A : A axis (rotation axis)

X : X axis (linear axis)

* : {B, C... constant}



Command format

G02.3 Xx1 Yy1 Zz1 Ii1 Jj1 Rr1 Ff1 Qq1 ; ... Forward rotation interpolation

G03.3 Xx1 Yy1 Zz1 Ii1 Jj1 Rr1 Ff1 Qq1 ; ... Negative rotation interpolation

X	X axis end point (Note 1)
Y	Y axis end point (Note 1)
Z	Z axis end point (Note 1)
I	Angle i1 (Note 2)
J	Angle j1 (Note 2)
R	Constant value r1 (Note 3)
F	Initial feedrate (Note 4)
Q	Feedrate at end point (Note 5)

(Note 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 rotation axis designated with parameter "#1515 expRotax" is specified, linear interpolation without exponential function interpolation will take place.

(Note 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.

(Note 3) The command unit is as follows.

Setting unit	#1003 = B	#1003 = C	#1003 = D	#1003 = E	Unit
Metric 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.

(Note 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 rotation 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.

(Note 5) The command unit is as follows.

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

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

Command the composite feedrate that includes the rotation axis.

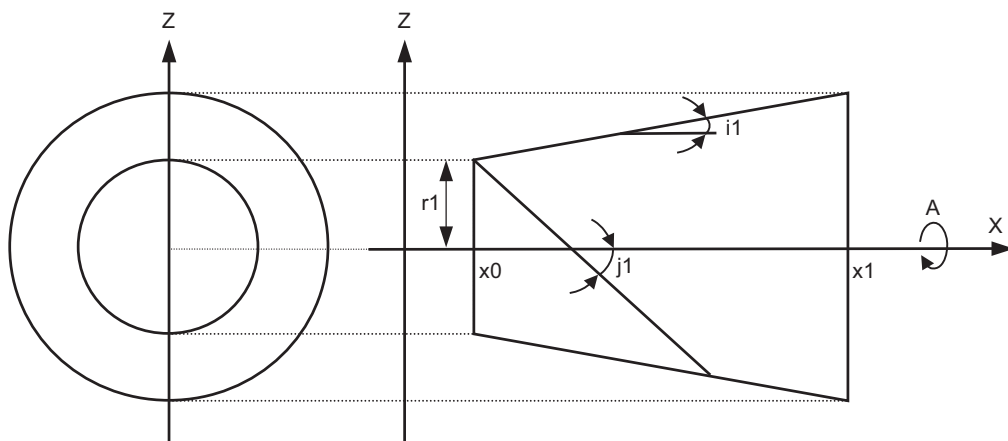
The normal F modal value will not change by the address Q command.

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 rotation axis (A) in the G02.3/G03.3 command is defined in the following manner.

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

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

However,

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

$\omega = 0$ during forward rotation (G02.3), and $\omega = 1$ during reverse rotation (G03.3).

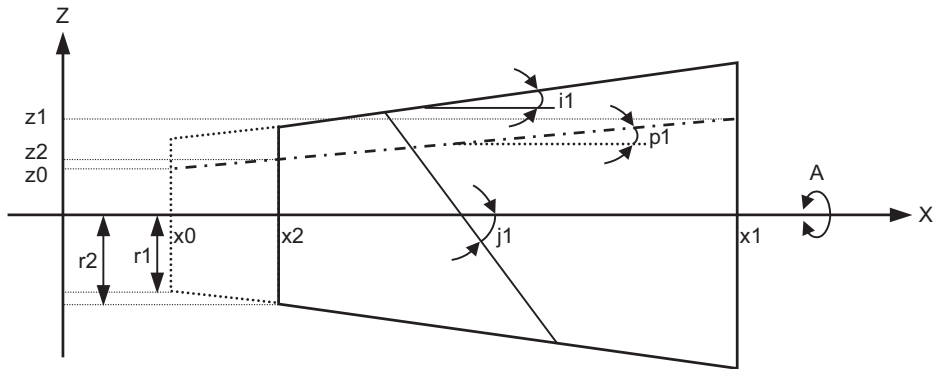
θ is the rotation angle (radian) from the rotation axis' start point

The rotation axis' rotation angle (θ) is as follows according to expression (1).

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

Machining example

- Uniform helix machining of taper shape



<Relational expression of exponential function in machining example>

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

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

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

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

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

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

A(θ): Absolute value from A axis (rotation axis) start point

r1: Exponential function interpolation constant value (address R command)

r2: Workpiece left edge radius

x2: X axis (linear axis) position at workpiece left edge

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 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):

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

According to expression (3), the slot base gradient angle (p1) is determined 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

G2.3(Equivalent to G3.3 if $j1 < 0$)					
		X movement direction > 0		X movement direction < 0	
		$i1 > 0$	$i1 < 0$	$i1 > 0$	$i1 < 0$
Command					
Operation					
Machining program example	N10 G28XYZC; N20 G91G0 X100. Z100.; N30 G2.3 X100. Z100. I50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X100. Z200.; N30 G2.3 X100. Z-100. I-50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X-100. Z100.; N30 G2.3 X-100. Z100. I50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X-100. Z200.; N30 G2.3 X-100. Z-100. I-50. J80. R105. F500.; N40 M30;	

G3.3(Equivalent to G2.3 if $j1 < 0$)					
		X movement direction > 0		X movement direction < 0	
		$i1 > 0$	$i1 < 0$	$i1 > 0$	$i1 < 0$
Command					
Operation					
Machining program example	N10 G28XYZC; N20 G91G0 X100. Z100.; N30 G3.3 X100. Z100. I50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X100. Z200.; N30 G3.3 X100. Z-100. I-50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X-100. Z100.; N30 G3.3 X-100. Z100. I50. J80. R105. F500.; N40 M30;	N10 G28XYZC; N20 G91G0 X-100. Z200.; N30 G3.3 X-100. Z-100. I-50. J80. R105. F500.; N40 M30;	

(E) End point

(S) Start 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 rotation 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 rotation axis designated with the parameter (#1515 expRotax) is commanded.
- (3) Tool length offset and nose R compensation cannot be used during the G02.3/G03.3 mode. Note that the tool length offset which started interpolation before the G02.3/G03.3 mode will normally continue.
- (4) A program error (P481) will occur if commands are issued during the pole coordinate interpolation, cylindrical interpolation or milling interpolation modes.
- (5) Program error (P612) will occur if commands are issued during the mirror image.
- (6) G02.3/G03.3 will function with feed per minute even during the feed per revolution mode.
- (7) If the parameter "#1515 expRota" setting is the same axis name as the initial C axis, the axis selected with the C axis selection signal will interpolate as the rotation axis.

7

Feed Functions

7.1 Rapid Traverse Rate



Function and purpose

The rapid traverse rate can be set independently for each axis. The available speed ranges are from 1 mm/min to 10000000 mm/min. The upper limit is subject to the restrictions limited by the machine specifications. Refer to the specifications manual of the machine for the rapid traverse rate settings. Two paths are available for positioning: the interpolation type where the area from the start point to the end point is linearly interpolated or the non-interpolation type where movement proceeds at the maximum speed of each axis. The type is selected with parameter "#1086 G0Intp". The positioning time is the same for each type.

(Note) Rapid traverse override

Override can be applied by an external input signal for both manual and automatic rapid traverse.

There are 2 types which are determined by the PLC specifications.

Type1 : Override in 4 steps: 1%, 25%, 50%, 100%

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

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 by the cutting feedrate clamp parameter (whose setting range is the same as that for the cutting feedrate).

The cutting feedrate is assigned with address F and numerals.

The cutting feedrate is valid for the G01, G02, G03, G33 and G34 commands.

Examples Feed per minute (asynchronous feed)

Metric input

Feedrate					
G1	X100.	Z100.	F200 ;	200.0mm/min	F200 or F200.000 gives the same rate.
G1	X100.	Z100.	F123.4 ;	123.4mm/min	
G1	X100.	Z100.	F56.789 ;	56.789mm/min	

Inch input

Feedrate					
G1	X100.	Z100.	F200 ;	2.0inch/min	
G1	X100.	Z100.	F200. ;	200.0inch/min	
G1	X100.	Z100.	F123.4 ;	123.4inch/min	
G1	X100.	Z100.	F56.789 ;	56.789inch/min	

Speed range that can be commanded (when input setting unit is 1 μ m)

Command mode	Feedrate command range	Remarks
mm/min	0.001 to 10,000,000	
inch/min	0.0001 to 1000000	
° /min	0.001 to 10,000,000	

(Note 1) A program error (P62) will occur when there is no F command in the first cutting command (G01, G02, G03, G33, G34) after the power has been turned ON.

7.3 F1-digit Feed



Function and purpose

By setting the F1-digit feed parameter, the feedrate which has been set to correspond to the 1-digit number following the F address serves as the command value.

When F0 is assigned, the rapid traverse rate is established and the speed is the same as for G00. (G modal does not change, but the acceleration/deceleration method follows the rapid traverse setting.)

When F1 to F5 is assigned, the feedrate set to correspond to the command serves as the command value.



Detailed description

- (1) To validate the F1-digit feed, set the parameter "#1079 F1digit" to "1".
- (2) The override function of the feedrate which is set in accordance to the F1-digit is performed by using the 1st manual handle. (Feedrate cannot be changed with the 2nd or 3rd handle.)

The amount by which the feedrate can be increased or reduced is determined by the following formula.

$$\pm \Delta F = \frac{FM}{K} \times (\pm P)$$

Here, "+" means increase, and "-" means reduction.

- K : Operation constant
(This is the number of FM divisions, and is the calculated constant of the increment/decrement speed per scale of the manual handle pulse generator.)
This is set with the base specification parameter "#1507 F1_K".
- FM : This is the clamp speed of F1 to F5
This is set with the base specification parameter "#1506 F1_FM".
- P : The number of manual handle pulses

Set the corresponding speed of F1 to F5 with the base specification parameters "#1185 spd_F1" to "#1189 spd_F5" respectively.

The increase/reduction range is from "0" to the set value of the parameter "#1506 F1_FM".

The error "M01 operation error 0104" will occur when the feedrate is 0.

- (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 number of manual handle pulses is 1 pulse per scale unit regardless of the scaling factor.
- (7) During a F1-digit command, the F1-digit number and F1-digit command signal are output as the PLC signals.

F1-digit and G commands

- (1) 01 group G command in same block as F1-digit commands

	Executed feedrate	Modal display rate	G modal
G0F0 F0G0	Rapid traverse rate	0	G0
G0F1 F1G0	Rapid traverse rate	1	G0
G1F0 F0G1	Rapid traverse rate	0	G1
G1F1 F1G1	F1 contents	1	G1

- (2) F1-digit and unmodal commands may be assigned in the same block. In this case, the unmodal command is executed and at the same time the F1-digit modal command is updated.

Example of arithmetic constant K setting

When the handle scale unit is to be made ± 10 mm/min.

FM is made 15000 mm/min:

$$\Delta F = 10 = \frac{15000}{K}$$

Therefore, K is 1500.

The feedrate is made F (1 to 5) ± 10 (mm/min) by rotating the handle by one scale unit.

The conditions for validation of the speed change with the handle

When all the following conditions are established, the speed change is valid with the handle.

- (1) The operation must be automatic (program run, MDI).
- (2) The operation must be in automatic start.
- (3) The operation must be in cutting feed, and F1-digit feedrate command must be issued.
- (4) The F1-digit feed parameter must be ON.
- (5) The F1-digit feed switch must be ON.
- (6) The operation must not be in machine lock.
- (7) The operation must not be in dry run. .
- (8) Upper limit of F1-digit feedrate (#1506) and F1-digit feedrate change constant (#1507) are not zero.

Conversion of the F1-digit feedrate changing with handle operation

Once the F1-digit feedrate is changed with manual handle, the changed feedrate is valid until parameter is changed or the power is turned OFF. Changed feedrate can be checked only during operation. Note that if the parameter "#1246 set18/bit16"(switchover of the F1-digit feedrate changing method) is set to "1", even after the power is turned OFF, operation is kept at the changed feedrate by changing to the feedrate after changing the parameter of F1-digit feedrate. In this case, if the screen is not switched, the changed parameter values are not reflected to display.



Precautions

- (1) If F1-digit feedrate changing valid signal is turned ON when F1-digit feed is commanded and changing feedrate with the manual handle is valid and applied to the following conditions, it is clamped at the upper limit of F1-digit feedrate.
F1-digit feedrate > Upper limit of F1-digit feedrate
- (2) When the F1-digit feedrate is 0 with the handle operation during the F1-digit feedrate command, the alarm "0104 F1-digit feed rate zero" will occur. This alarm can be eliminated when the F1-digit feedrate is set in larger value than 0 with the handle operation.
- (3) When K is 0, the speed is not changed.
- (4) F1 to F5 are invalid in the G00 mode and the rapid traverse rate is established instead.
- (5) 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.
- (6) 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.
- (7) 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.
- (8) 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 (°)/mm.
- (9) 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).
- (10) When both the F1-digit feed command and inverse time feed command are present, the inverse time feed command will have the priority.
- (11) When both the F1-digit feedrate changing and the manual speed command are present, the manual speed command will have the priority.
- (12) In the synchronous tapping command, the speed cannot be changed with the handle.

7.4 Feed Per Minute/Feed Per Revolution (Asynchronous Feed/Synchronous Feed) ; G94,G95



Function and purpose

Feed per minute (asynchronous feed)

By issuing the G94 command, the commands from that block are issued directly by the numerical value following F as the feedrate per minute (mm/min, inch/min).

Feed per revolution (synchronous feed)

By issuing the G95 command, the commands from that block are issued directly by the numerical value following F as the feedrate per spindle revolution (mm/rev, inch/rev).

When this command is used, the rotary encoder must be attached to the spindle.



Command format

G94; ... Feed per minute (mm/min) (asynchronous feed)

G95; ... Feed per revolution (mm/rev) (synchronous feed)



Detailed description

G94/G95 commands are modal commands.

(Ex.) After the G95 command is assigned, the G95 command is valid until the G94 command is next assigned.

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

Metric input

Input Setting unit	B(0.001mm)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(mm/min)	F(mm/rev)
Minimum command unit	1(=1.000) (1.=1.000)	1(=0.0001) (1.=1.00)
Command range	0.001 - 1000000.000	0.0001 - 999.9999

Input Setting unit	C(0.0001mm)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(mm/min)	F(mm/rev)
Minimum command unit	1(=1.0000) (1.=1.0000)	1(=0.00001) (1.=1.00)
Command range	0.0001 - 1000000.0000	0.00001 - 999.99999

Input Setting unit	D(0.00001mm)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(mm/min)	F(mm/rev)
Minimum command unit	1(=1.00000) (1.=1.00000)	1(=0.000001) (1.=1.00)
Command range	0.00001 - 1000000.00000	0.000001 - 999.999999

Input Setting unit	E(0.000001mm)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(mm/min)	F(mm/rev)
Minimum command unit	1(=1.000000) (1.=1.000000)	1(=0.0000001) (1.=1.00)
Command range	0.000001 - 1000000.000000	0.0000001 - 999.9999999

Inch input

Input Setting unit	B(0.0001inch)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(inch/min)	F(inch/rev)
Minimum command unit	1(=0.0100) (1.=1.0000)	1(=0.000001) (1.=1.000)
Command range	0.0001 - 100000.0000	0.00001 - 99.99999

Input Setting unit	C(0.00001inch)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(inch/min)	F(inch/rev)
Minimum command unit	1(=0.01000) (1.=1.00000)	1(=0.0000001) (1.=1.000)
Command range	0.00001 - 100000.00000	0.000001 - 99.999999

Input Setting unit	D(0.000001inch)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(inch/min)	F(inch/rev)
Minimum command unit	1(=0.010000) (1.=1.000000)	1(=0.00000001) (1.=1.000)
Command range	0.000001 - 100000.000000	0.0000001 - 99.9999999

Input Setting unit	E(0.0000001inch)	
Command Mode	Feed per minute	Feed per revolution
Command Address	F(inch/min)	F(inch/rev)
Minimum command unit	1(=0.0100000) (1.=1.0000000)	1(=0.000000001) (1.=1.000)
Command range	0.0000001 - 100000.0000000	0.00000001 - 99.99999999

- (2) The effective rate (actual movement speed of machine) under per-revolution feed conditions is given in the following formula (Formula 1).

$$FC = F \times N \times OVR \quad \dots\dots \text{(Formula 1)}$$

FC : Effective rate (mm/min, inch/min)

F : Commanded feedrate (mm/rev, inch/rev)

N : Spindle rotation speed (r/min)

OVR: Cutting feed override

When a multiple number of axes have been commanded at the same time, the effective rate FC in formula 1 applies in the vector direction of the command.



Precautions

- (1) The effective rate (mm/min or inch/min), which is produced by converting the commanded speed, the spindle rotation speed and the cutting feed override into the per-minute speed, appears as the FC on the monitor 1. Screen of the setting and display unit.
- (2) When the above effective rate exceeds the cutting feed clamp rate, it is clamped at that clamp rate.
- (3) If the spindle rotation speed is zero when feed per revolution is executed, the error "M01 operation error 0105" occurs.
- (4) Feedrate during the machine lock becomes the command speed.
- (5) Under dry run conditions, feed per minute applies and movement results at the manual feedrate (mm/min or inch/min).
- (6) Whether feed per minute (G94) or feed per revolution (G95) is to be established when the power is turned ON or when M02 or M30 is executed can be selected by setting parameter "#1074 I_Sync".

7.5 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.

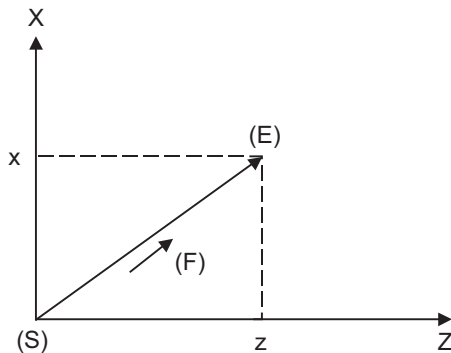


Detailed description

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 Z) are to be controlled:



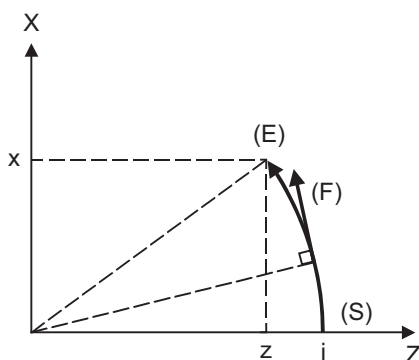
$$f_x = f \times \frac{x}{\sqrt{x^2 + z^2}}$$

$$f_z = f \times \frac{z}{\sqrt{x^2 + z^2}}$$

- (S) Tool start point
- (E) Tool end point
- (F) Speed in this direction is "f".
- f_x : Feedrate for X axis
- f_z : Feedrate for Z axis

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.

(Note) When the circular interpolation function is used and the tool is moved along the circumference of an arc by the linear control axis, 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".

(Example) When the feedrate is designated as "f" and the linear axes (X and Z) are to be controlled using the circular interpolation function:

In this case, the feedrate of the X and Z 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

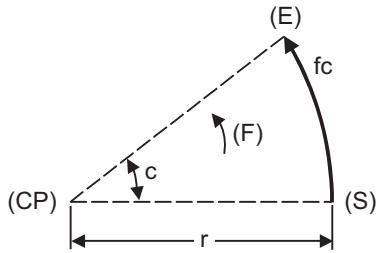
When rotary axes are to be controlled, the designated feedrate functions as the rotary speed of the rotary axes or, in other words, as an angular speed.

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, in order to make the cutting feed (linear feed) in the tool advance direction "fc" :

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

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

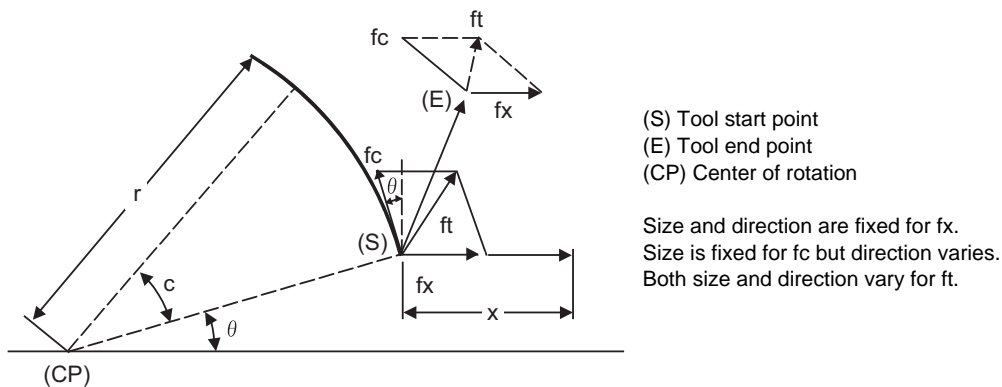
$$f = f_c \times \frac{180}{\pi \cdot r}$$

When linear and rotary axes are to be controlled at the same time

The controller proceeds in exactly the same way whether linear or rotary axes are to be controlled. When a rotary axis is to be controlled, the numerical value assigned by the coordinate word (C, H) is the angle and the numerical values assigned by the feedrate (F) are all handled as linear speeds. In other words, 1° of the rotary axis is treated as being equivalent to 1mm of the linear axis. Consequently, when both linear and rotary axes are to be controlled simultaneously, in the components for each axis of the numerical values assigned by F will be the same as previously described "When controlling linear axes". However, although in this case both the size and direction of the speed components based on linear axis control do not vary, the direction of the speed components based on rotary axis control will change along with the tool movement (their size will not change). This means, as a result, that the combined tool advance direction feedrate will vary along with the tool movement.

(Example) When the feedrate is designated as "f" and Linear (X) and rotary (C) axes are to be controlled simultaneously

In the X-axis incremental command value is "x" and the C-axis incremental command values is "c":



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

$$f_x = f \times \frac{x}{\sqrt{x^2 + c^2}} \quad \dots\dots (1)$$

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

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

$$f_c = \omega \times \frac{\pi \times r}{180} \quad \dots\dots (3)$$

If the speed in the tool advance direction at start point (S) is "ft" and the component speeds in the X-axis and Y-axis directions are "ftx" and "fty", respectively, then these can be expressed as:

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

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

Where r is the distance between center of rotation and tool (in mm units)
 θ is the angle between the (S) point and the X axis at the center of rotation (in units °)

The combined speed "ft" according to (1), (2), (3), (4) and (5) is:

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

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

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

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

"ft" in formula (6) is the speed at the (S) point and the value of θ changes as the C axis rotates, which means that the value of "ft" will also change. Consequently, in order to keep the cutting feed "ft" as constant as possible the angle of rotation which is designated in one block must be reduced to as low as possible and the extent of the change in the θ value must be minimized.

7.6 Thread Cutting Mode



Function and purpose

F command or E commands for thread leads can be issued for the thread cutting mode (G33, G34, G76 G78 commands).

The thread leads command range is shown below.

Thread cutting metric input

Input setting unit	B(0.001mm)		
Command Address	F(mm/rev)	F(mm/rev)	E(ridges/inch)
Minimum command unit	1(=0.0001) (1.=1.0000)	1(=0.00001) (1.=1.00000)	1(=1.00) (1.=1.00)
Command range	0.0001 - 999.9999	0.00001 - 999.99999	0.03 - 999.99

Input setting unit	C(0.0001mm)		
Command Address	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(=1.000) (1.=1.000)
Command range	0.00001 - 999.99999	0.000001 - 999.999999	0.026 - 222807.017

Input setting unit	D(0.00001mm)		
Command Address	F(mm/rev)	E(mm/rev)	E(ridges/inch)
Minimum command unit	1(=0.000001) (1.=1.000000)	1(=0.0000001) (1.=1.0000000)	1(=1.0000) (1.=1.0000)
Command range	0.000001 - 999.999999	0.0000001 - 999.9999999	0.0255 - 224580.0000

Input setting unit	E(0.000001mm)		
Command Address	F(mm/rev)	E(mm/rev)	E(ridges/inch)
Minimum command unit	1(=0.0000001) (1.=1.0000000)	1(=0.00000001) (1.=1.00000000)	1(=1.00000) (1.=1.00000)
Command range	0.0000001 - 999.9999999	0.00000001 - 999.99999999	0.02540 - 224719.00000

Thread cutting inch input

Input setting unit	B(0.0001inch)		
Command Address	F(inch/rev)	E(inch/rev)	E(ridges/inch)
Minimum command unit	1(=0.00001) (1.=1.00000)	1(=0.000001) (1.=1.000000)	1(=1.000) (1.=1.000)
Command range	0.00001 - 39.37007	0.000001 - 39.370078	0.025 - 9999.999

Input setting unit	C(0.00001inch)		
Command Address	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(=1.0000) (1.=1.0000)
Command range	0.000001 - 39.370078	0.0000001 - 39.3700787	0.0254 - 9999.9999

Input setting unit	D(0.000001inch)		
Command Address	F(inch/rev)	E(inch/rev)	E(ridges/inch)
Minimum command unit	1(=0.0000001) (1.=1.0000000)	1(=0.00000001) (1.=1.00000000)	1(=0.00001) (1.=1.00000)
Command range	0.0000001 - 39.3700787	0.00000001 - 39.37007873	0.02540 - 9999.99999

Input setting unit	E(0.0000001inch)		
Command Address	F(inch/rev)	E(inch/rev)	E(ridges/inch)
Minimum command unit	1(=0.00000001) (1.=1.00000000)	1(=0.000000001) (1.=1.000000000)	1(=1.000000) (1.=1.000000)
Command range	0.00000001 - 39.37007873	0.000000001 - 39.370078736	0.025400 - 9999.999999

7.7 Automatic Acceleration/Deceleration

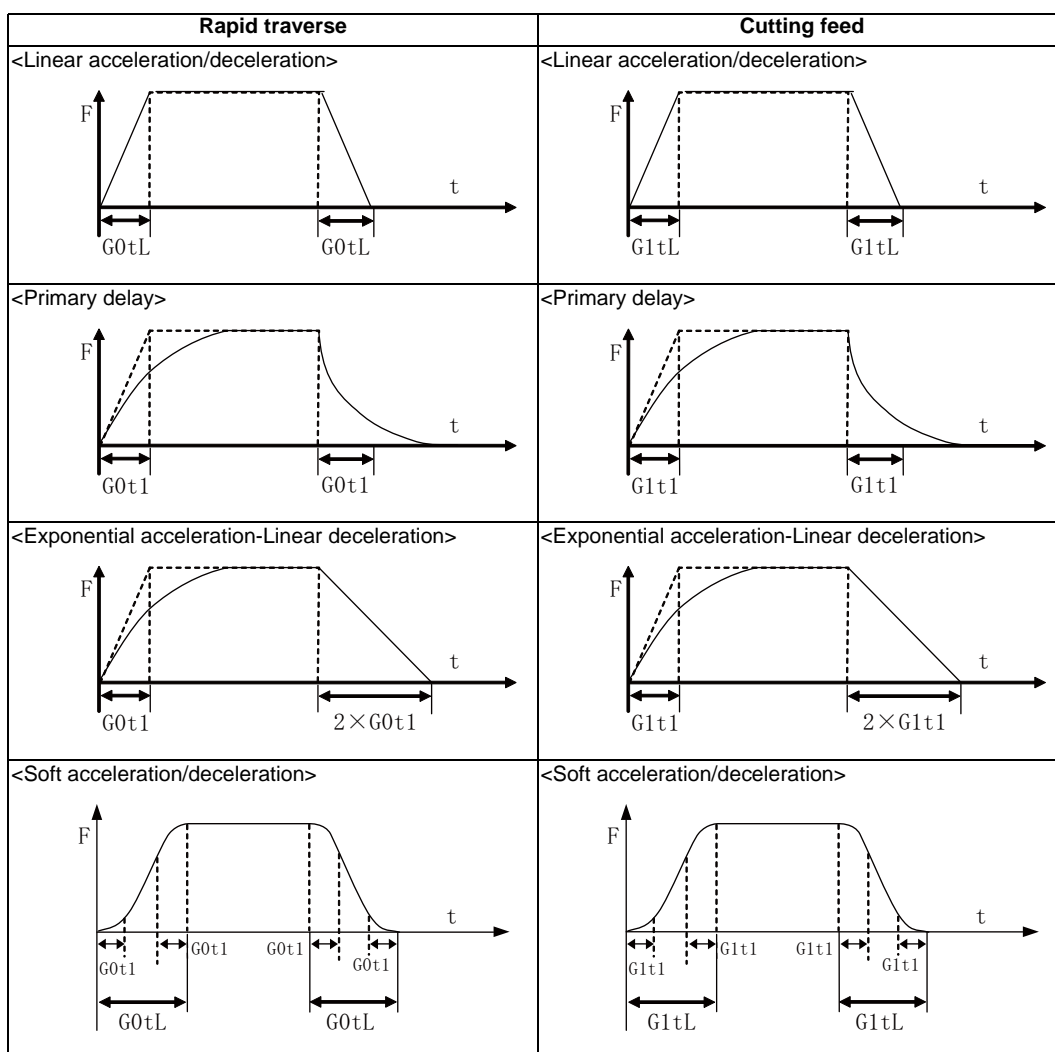


Function and purpose

Acceleration/deceleration is applied to axis traverse automatically. The acceleration/deceleration patterns are linear acceleration/deceleration, primary delay acceleration/deceleration, exponential acceleration-linear deceleration and soft acceleration/deceleration. Select those patterns with the axis specification parameter "#2003 smgst".

For rapid traverse feed, acceleration/deceleration is always made for each block, and the time constant can be set for each axis separately.

Refer to the following figures for the acceleration/deceleration patterns of rapid feed and cutting feed.



G0tL:G0 time constant(linear)

G0t1:G0 time constant(primary delay)/The 2nd step time constant of the soft acceleration/deceleration

G1tL:G1 time constant(linear)

G1t1:G1 time constant(primary delay)/The 2nd step time constant of the soft acceleration/deceleration

(Note 1) The rapid traverse feed acceleration/deceleration patterns are effective for the following:
G00, G27, G28, G29, G30, rapid traverse feed in manual run, JOG feed, incremental feed, return to reference position.
It is invalid to G31 and handle feed.

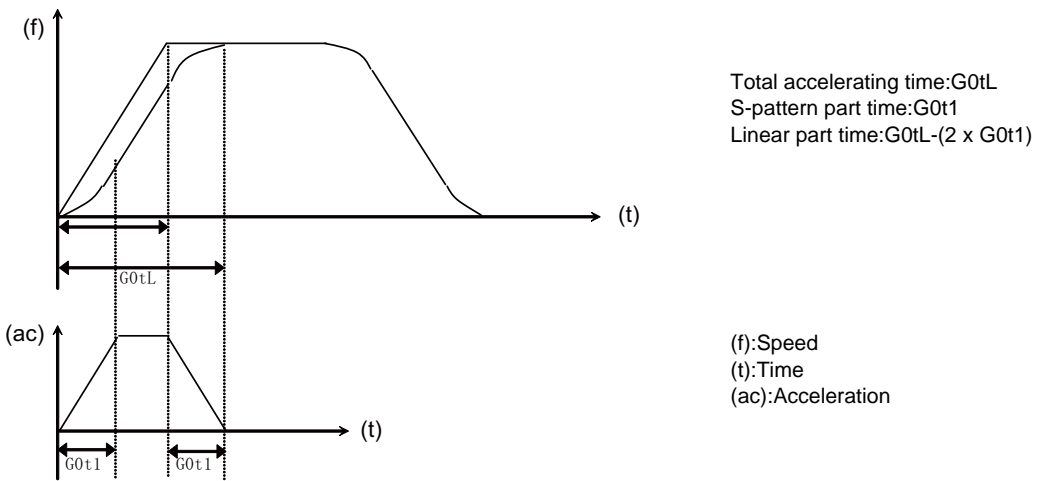
(Note 2) Acceleration/deceleration in handle feed mode is usually performed according to the acceleration/deceleration pattern for cutting feed. However, a parameter can be specified to select a pattern with no acceleration/deceleration (step).



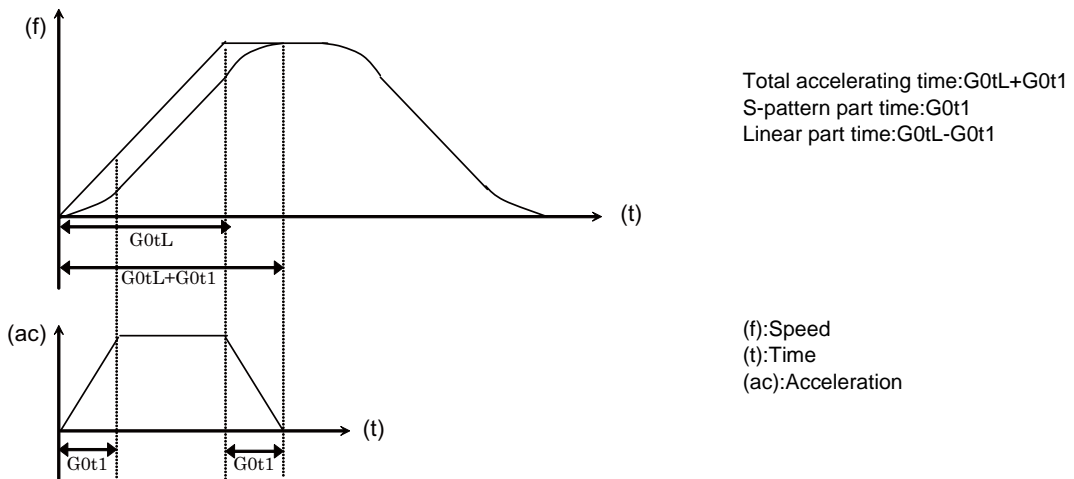
Detailed description

- (1) Acceleration/deceleration are automatically controlled both in automatic and manual operations. Select acceleration/deceleration patterns with the axis specification parameter "#2003 smgst(acceleration/deceleration mode)".
- (2) Acceleration/deceleration time of soft acceleration/deceleration can be changed by the basic specification parameter "#1219 aux03/bit7".

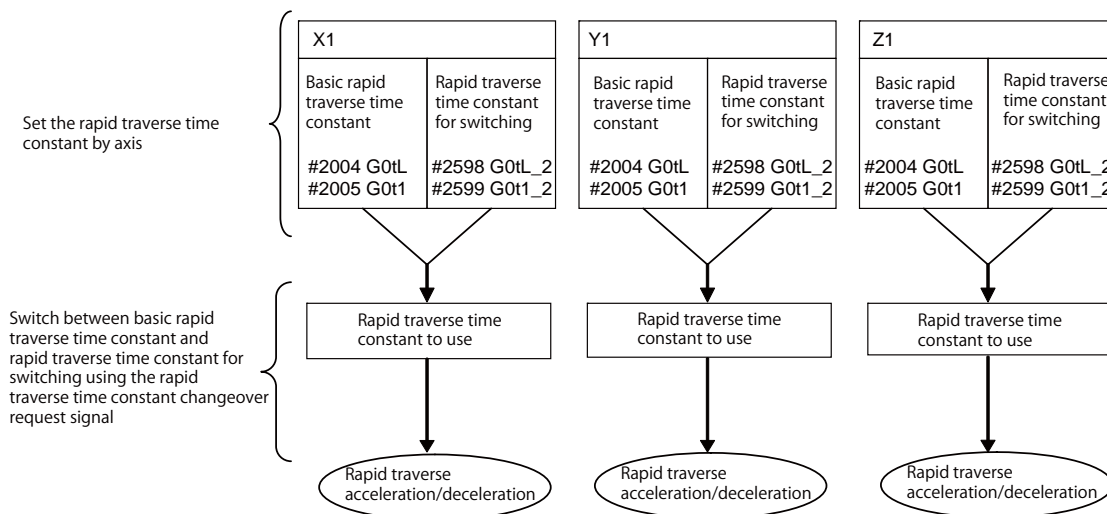
[When the basic specification parameter "#1219 aux03/bit7" is OFF.]



[When the basic specification parameter "#1219 aux03/bit7" is ON.]



(3) Use the rapid traverse time constant changeover request signal to switch the rapid traverse constant.



[Parameter]

Rapid traverse time constants for switching and for basic are set in the following parameters.

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

(Note) When #2598 is "0", use #2004. When #2599 is "0", use #2005.

[How to switch time constants]

Turn ON the rapid traverse time constant changeover request signal so that the rapid traverse time constant is changed to the rapid traverse time constant for switching.

Turn OFF the rapid traverse time constant changeover request signal so that the rapid traverse time constant is changed to the basic rapid traverse time constant.

[Timing to switch time constants]

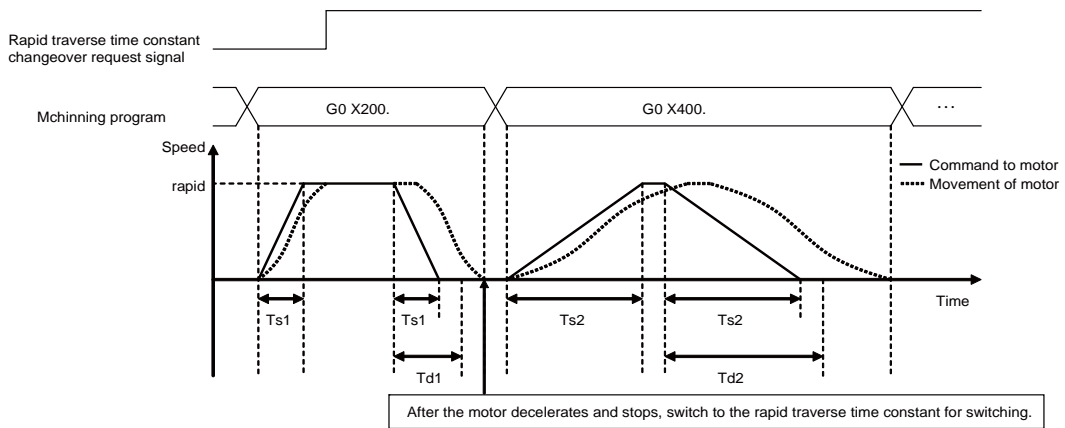
Automatic operation: When switching is required in rapid traverse, the axis is decelerated to stop at the end of the block, and then the time constant is changed.

Automatic operation: When switching is required in cutting traverse with stopped axis, switch immediately the time constant.

Manual operation: When switching is required in cutting feed or at a standstill, the time constant is changed immediately.

[Operation example]

Rapid traverse time constant switching 1.(turn on the rapid traverse time constant changeover request signal during the axis travel.)



rapid : Rapid traverse speed (Axis specification parameter #2001 rapid)

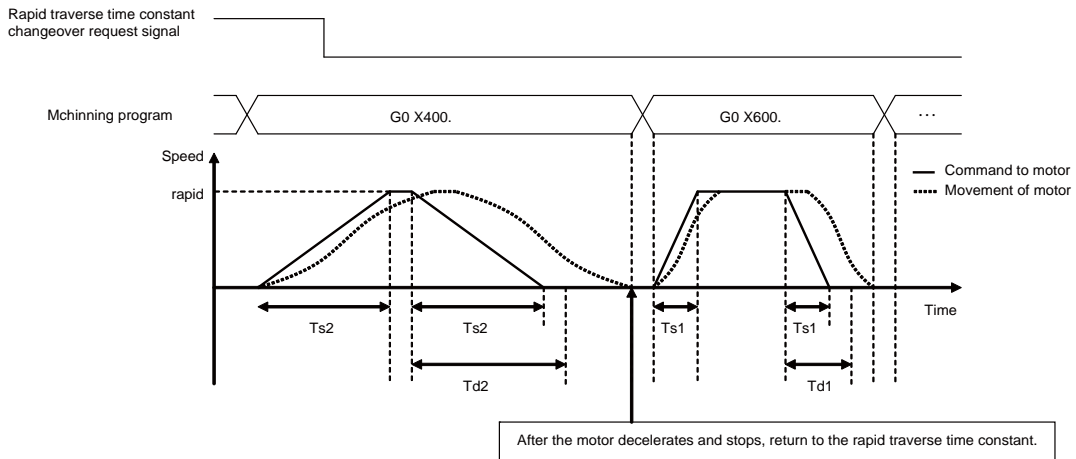
Ts1 : Acceleration/deceleration time (Axis specification parameter #2004 G0tL(G0 time constant) = 100ms)

Ts2 : Acceleration/deceleration time (Axis specification parameter #2598 G0tL_2(G0 time constant 2) = 300ms)

Td1 : Command deceleration check time (Td1 = Ts1 + (0 to 14 ms))

Td2 : Command deceleration check time (Td2 = Ts2 + (0 to 14 ms))

Rapid traverse time constant switching 2.(turn off the rapid traverse time constant changeover request signal during the axis travel.)



rapid : Rapid traverse speed (Axis specification parameter #2001 rapid)

Ts1 : Acceleration/deceleration time (Axis specification parameter #2004 G0tL(G0 time constant) = 100ms)

Ts2 : Acceleration/deceleration time (Axis specification parameter #2598 G0tL_2(G0 time constant 2) = 300ms)

Td1 : Command deceleration check time (Td1 = Ts1 + (0 to 14 ms))

Td2 : Command deceleration check time (Td2 = Ts2 + (0 to 14 ms))



Precautions

- (1) When the time constant < 2nd step time constant under the soft acceleration/deceleration, the feed rate pattern of the time constant and 2nd step time constant are interchanged.

7.8 Rapid Traverse Constant Inclination Acceleration/Deceleration



Function and purpose

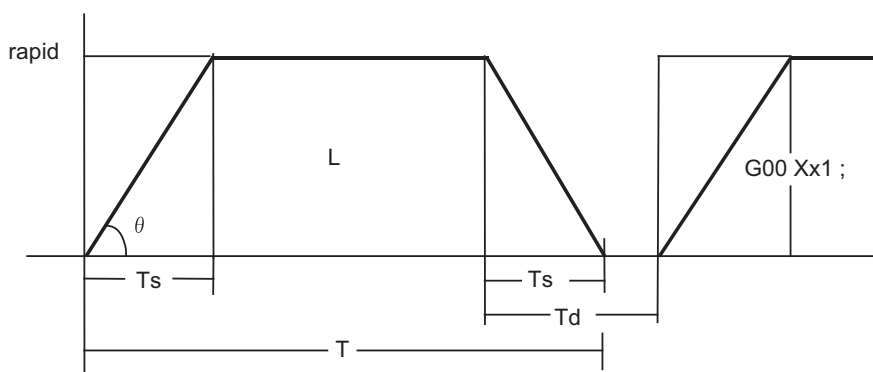
This function performs acceleration and deceleration at a constant inclination during linear acceleration/ deceleration in the rapid traverse mode. Compared to the method of acceleration /deceleration after interpolation, the constant inclination acceleration/deceleration method makes for improved cycle time.



Detailed description

- (1) Rapid traverse constant inclination acceleration/deceleration are valid only for a rapid traverse command. Also, this function is effective only when the rapid traverse command acceleration/ deceleration mode is linear acceleration and linear deceleration.
- (2) The acceleration/deceleration patterns in the case where rapid traverse constant inclination acceleration/ deceleration are performed are as follows.

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



$$T = \frac{L}{\text{rapid}} + T_s$$

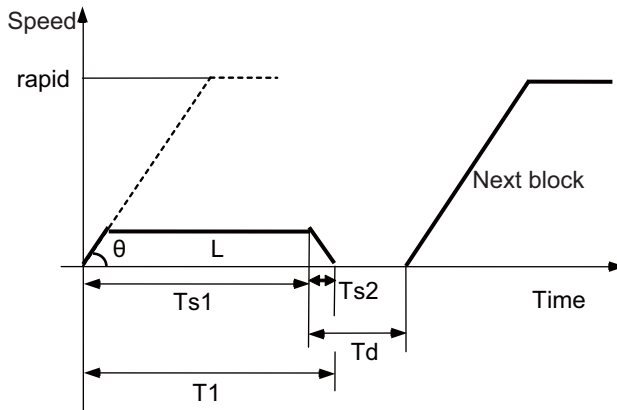
$$T_d = T_s + (0 \sim 14\text{ms})$$

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

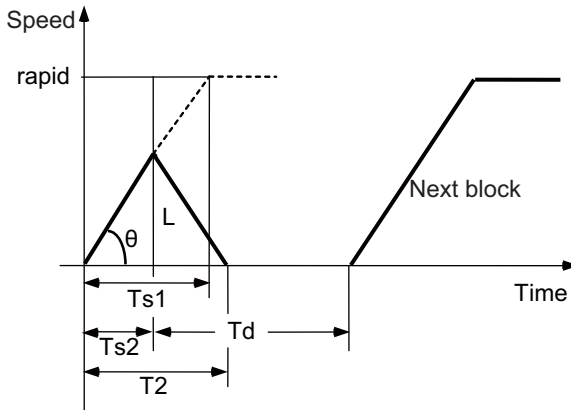
- rapid : Rapid traverse rate
- Ts : Acceleration/deceleration time constant
- Td : Command deceleration check time
- θ : Acceleration/deceleration inclination
- T : Interpolation time
- L : Interpolation distance

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

In case of time-constant acceleration/deceleration:



In case of inclination-constant acceleration/deceleration:



$$T1 = Ts1 + Ts2$$

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

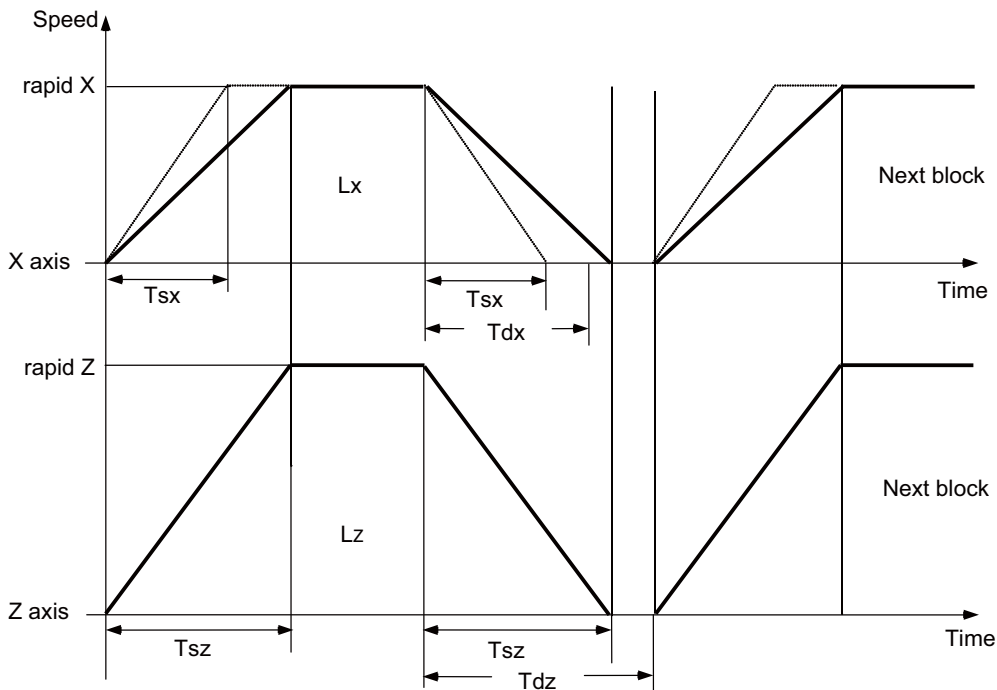
$$Td = T2 / 2 + (0 \text{ to } 14 \text{ ms})$$

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

- rapid : Rapid traverse rate (Axis specification parameter #2001 rapid)
- Ts1 : Acceleration/deceleration time (Axis specification parameter #2004 G0tL)
- Ts2 : Acceleration/deceleration time to reach the maximum speed
- Td : Command deceleration check time
- θ : Acceleration/deceleration inclination
- T1 : Interpolation time (time-constant acceleration/deceleration)
- T2 : Interpolation time (inclination-constant acceleration/deceleration)
- L : Interpolation distance

- (3) When 2-axis simultaneous interpolation (linear interpolations) is performed during rapid traverse constant inclination acceleration and deceleration, the acceleration (deceleration) time is the longest value of the acceleration (deceleration) times determined for each axis by the rapid traverse rate of commands executed simultaneously, the rapid traverse acceleration and deceleration time constant, and the interpolation distance, respectively. Consequently, linear interpolation is performed even when the axes have different acceleration and deceleration time constants.

[2-axis simultaneous interpolation (When linear interpolation is used, $T_{sx} < T_{sz}$, $L_x \neq L_z$)]

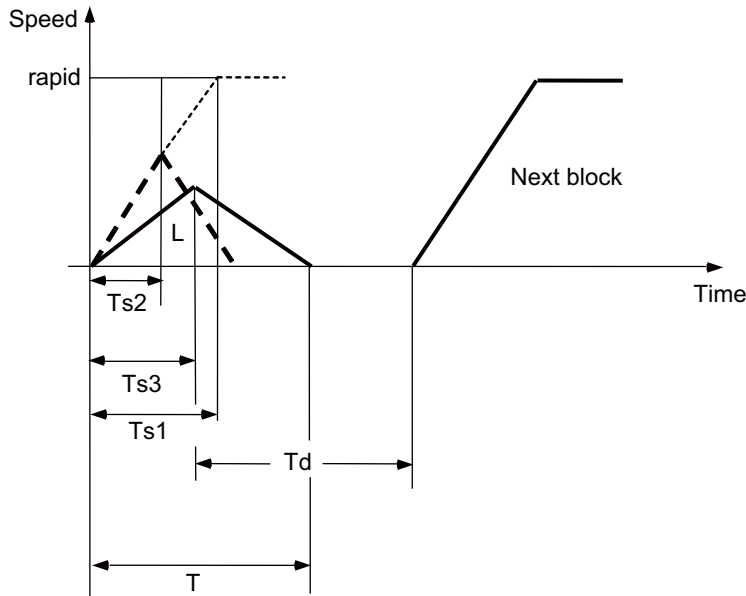


When T_{sz} is greater than T_{sx} ,
 T_{dz} is also greater than T_{dx} , and
 $T_d = T_{dz}$ in this block.

- T_{sx} : X axis acceleration/deceleration time
- T_{sz} : Z axis acceleration/deceleration time
- T_{dx} : X axis commanded deceleration check time
- T_{dz} : Z axis commanded deceleration check time
- L_x : X axis interpolation distance
- L_z : Z axis interpolation distance

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

[When the interpolation distance is so short that the acceleration/deceleration time is shorter than the minimum time constant for constant inclination acceleration/deceleration]



$$T = 2 \times Ts2$$

$$Td = \frac{T}{2} + (0 \text{ to } 14 \text{ ms})$$

- rapid : Rapid traverse speed (Axis specification parameter #2001 rapid)
- Ts1 : Acceleration/deceleration time (Axis specification parameter #2004 G0tL)
- Ts2 : Acceleration/deceleration time to reach the maximum speed
- Ts3 : Minimum time for inclination-constant acceleration/deceleration (Axis specification parameter #2198 G0tMin)
- Td : Command deceleration check time
- T1 : Interpolation time
- L : Interpolation distance

- (5) Use the rapid traverse time constant changeover request signal to switch the rapid traverse constant. The time constant is switched in the block next to where the rapid traverse time constant changeover request signal is turned ON/OFF.

	Basic rapid traverse time constant (signal OFF)	Rapid traverse time constant for switching (signal ON)
Rapid traverse time constant	#2004 G0tL	#2598 G0tL_2
Rapid traverse 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.

- (6) The program format of G0 (rapid traverse command) when rapid traverse constant inclination acceleration/deceleration are executed is the same as when this function is invalid (time constant acceleration/deceleration).
- (7) This function is valid only for G0 (rapid traverse).



Precautions

- (1) When "#2003 smgst Acceleration and deceleration modes" is set to the soft acceleration/deceleration, and "#1219 aux03 bit7: Time constant setting changeover for soft acceleration/deceleration" is set to "1: Accelerating 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-inclination acceleration/deceleration.
- (2) When "#2003 smgst Acceleration and deceleration modes" is set to the soft acceleration/deceleration, if the acceleration/deceleration time is shorter than $G0tL$ (or $G1tL$), the 2nd step time constant will be reduced by the same rate as the 1st step time constant.
- (3) If a commanded travel distance in a block is small, acceleration/deceleration time becomes quite short when the constant-inclination acceleration/deceleration method is enabled. Although this does contribute to reducing the cycle time, this can also be a cause of machine vibrations. In such case set the minimum time constant for constant-inclination acceleration/deceleration in parameter "#2198 $G0tMin$ ". By this setting, the acceleration/deceleration speed is controlled so that the acceleration/deceleration time should not fall below the preset value.

7.9 Cutting Feed Constant Inclination Acceleration/Deceleration



Function and purpose

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



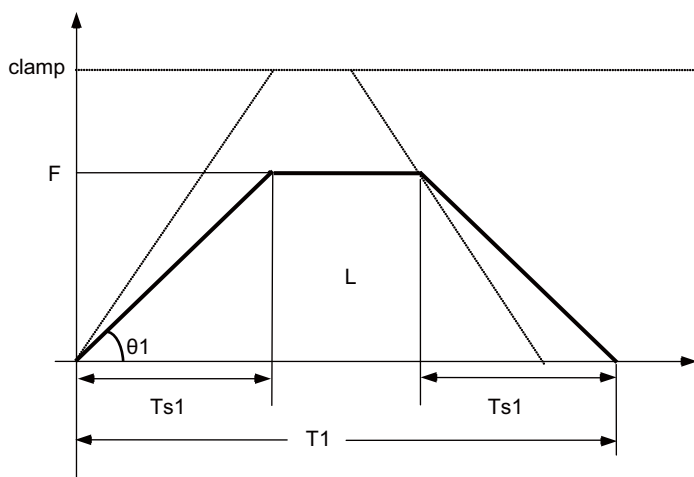
Detailed description

- (1) Cutting feed constant inclination acceleration/deceleration function is effective only when the commanded cutting feed acceleration/deceleration mode is linear method or soft method in a linear interpolation command.
- (2) The program format of linear interpolation when cutting feed constant inclination acceleration/deceleration is executed is the same as when this function is invalid (time constant acceleration/deceleration).

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

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

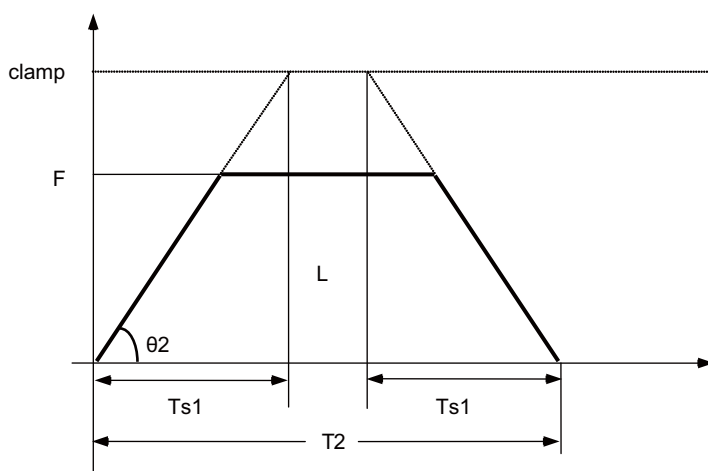
In case of time-constant acceleration/deceleration:



$$T1 = \frac{L}{F} + Ts1 \times 2$$

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

In case of inclination-constant acceleration/deceleration:



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

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

clamp : Maximum cutting feed rate (Axis specification parameter "#2002 clamp")

F : Cutting feed rate

Ts1 : Acceleration/deceleration time (Axis specification parameter "#2007 G1tL")

$\theta 1$: Acceleration/deceleration inclination (time-constant acceleration/deceleration)

$\theta 2$: Acceleration/deceleration inclination (inclination-constant acceleration/deceleration)

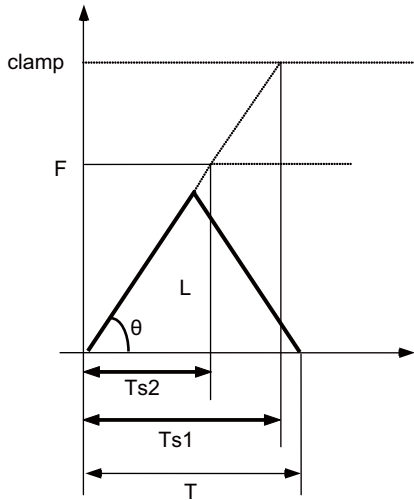
T1 : Interpolation time (time-constant acceleration/deceleration)

T2 : Interpolation time (inclination-constant acceleration/deceleration)

L : Interpolation distance

In the case of time-constant acceleration/deceleration, the acceleration/deceleration inclination is determined by the cutting feed rate. In the case of inclination-constant acceleration/deceleration, it's determined by the maximum cutting feed rate; therefore, the cycle time will be shorter than in the former case.

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



$$T = 2 \times \sqrt{Ts2 \times L / \text{clamp}}$$

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

clamp : Maximum cutting feed rate (Axis specification parameter "#2002 clamp")

F : Cutting feed rate

Ts1 : Acceleration/deceleration time (Axis specification parameter "#2007 G1tL")

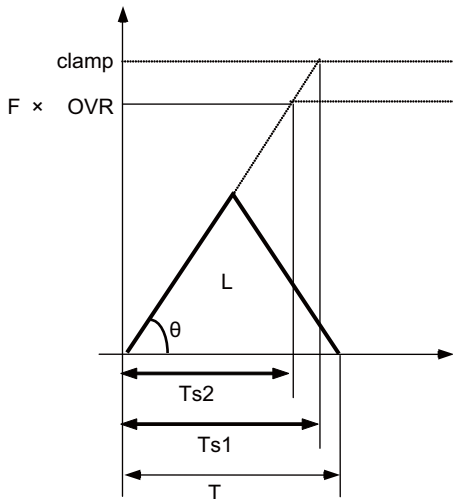
Ts2 : Acceleration/deceleration time to reach the cutting feed rate

θ : Acceleration/deceleration inclination

T : Interpolation time

L : Interpolation distance

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



$$T = 2 \times \sqrt{Ts2 \times L \times \text{OVR} / \text{clamp}}$$

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

clamp : Maximum cutting feed rate (Axis specification parameter "#2002 clamp")

F : Cutting feed rate

OVR : Maximum override value for cutting feed inclination-constant acceleration/deceleration (Base Specifications Parameter "#1367 G1AccOVRMax")

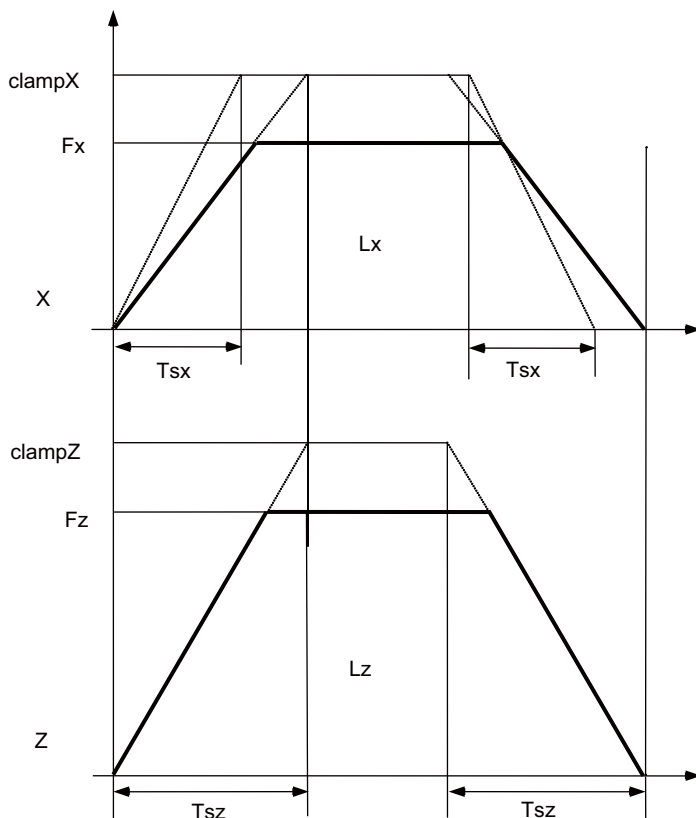
Ts1 : Acceleration/deceleration time (Axis specification parameter "#2007 G1tL")

Ts2 : Acceleration/deceleration time to reach the cutting feed rate

θ : Acceleration/deceleration inclination

[2-axis simultaneous interpolation (When $T_{sx} < T_{sz}$, $L_x \neq L_z$)]

When 2-axis simultaneous interpolation is performed during linear interpolation inclination-constant acceleration/deceleration, the acceleration/deceleration time of each axis will be set to the longest one among the acceleration/deceleration times determined by maximum cutting feed rates (axis specification parameter "#2002 clamp"), cutting feed acceleration/deceleration time constant (axis specification parameter "#2007 G1tL"), cutting feed rates (F) and interpolation distances (L) of the simultaneously commanded axes. Should the acceleration/deceleration time constant of an axis be greater than the cutting feed acceleration/deceleration time constant (axis specification parameter "#2007 G1tL"), that axis accelerates/decelerates in accordance with the cutting feed acceleration/deceleration time constant (axis specification parameter "#2007 G1tL").

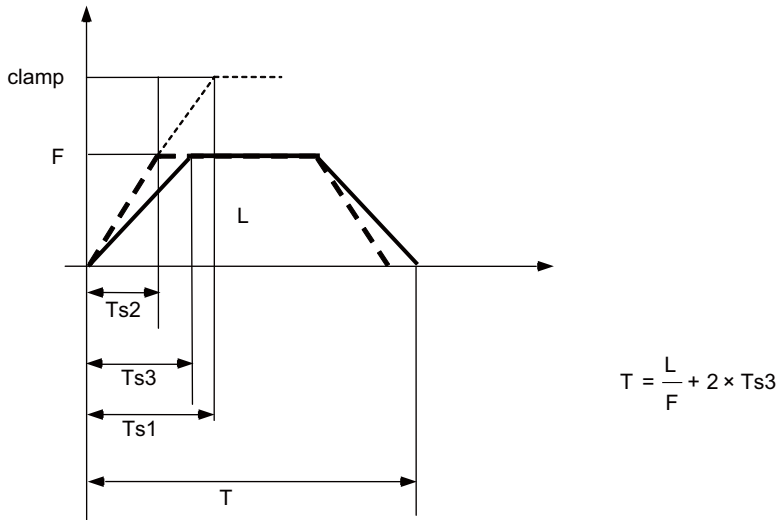


- Tsx : X axis acceleration/deceleration time
- Tsz : Z axis acceleration/deceleration time
- Lx : X axis interpolation distance
- Lz : Z axis interpolation distance
- Fx : X axis feed rate
- Fz : Z axis feed rate

When $T_{sx} < T_{sz}$, the acceleration/deceleration time of the block (T_s) will be equal to the acceleration/deceleration time T_{sz} (Z axis acceleration/deceleration time).

[When the feed rate is so low that the acceleration/deceleration time is shorter than the minimum time constant for inclination-constant acceleration/deceleration]

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



clamp : Maximum cutting feed rate (Axis specification parameter "#2002 clamp")

F : Cutting feed rate

Ts1 : Acceleration/deceleration time (Axis specification parameter "#2007 G1tL")

Ts2 : Acceleration/deceleration time to reach the cutting feed rate

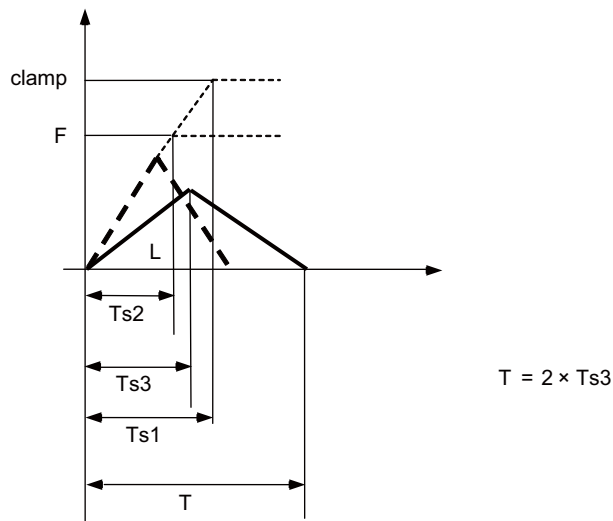
Ts3 : Minimum time for inclination-constant acceleration/deceleration (Axis specification parameter "#2199 G1tMin")

T : Interpolation time

L : Interpolation distance

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

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



clamp : Maximum cutting feed rate (Axis specification parameter "#2002 clamp")

F : Cutting feed rate

Ts1 : Acceleration/deceleration time (Axis specification parameter "#2007 G1tL")

Ts2 : Acceleration/deceleration time to reach the cutting feed rate

Ts3 : Minimum time for inclination-constant acceleration/deceleration (Axis specification parameter "#2199 G1tMin")

T : Interpolation time

L : Interpolation distance



Precautions

- (1) If a value greater than 100 (%) is designated for cutting feed override under cutting feed inclination-constant acceleration/deceleration control, the acceleration/deceleration inclination becomes steeper as the feedrate increases. To use the cutting feed override function at a rate higher than 100%, set the parameter "#1367 G1AccOVRMax" accordingly. When the setting of this parameter is between 0 and 99, the override value is handled as 100% even if the specified cutting feed override is over 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 and deceleration modes" is set to the soft acceleration/deceleration, and "#1219 aux03 bit7: Time constant setting changeover for soft acceleration/deceleration" is set to "1: Accelerating time is obtained by $G0tL+G0t1 (G1tL+G1t1)$ ", acceleration/deceleration speed is adjusted to prevent the sum of the 1st step and 2nd step acceleration/deceleration times from going below the minimum time constant for inclination-constant acceleration/deceleration.
- (4) When "#2003 smgst Acceleration and deceleration modes" 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 (G1) feed rate is low, acceleration/deceleration time becomes quite short when the inclination-constant acceleration/deceleration method is enabled. Although this does contribute to reduce the cycle time, this can also be a cause of machine vibrations. Such vibrations can be prevented by setting the minimum time constant for inclination-constant acceleration/deceleration in parameter "#2198 G0tMin". By this setting, the acceleration/deceleration speed is controlled so that the acceleration/deceleration time should not fall below the preset value.

7.10 Speed Clamp



Function and purpose

This function exercises control over the actual cutting feedrate in which override has been applied to the cutting feedrate command so that the speed clamp value which has been preset independently for each axis is not exceeded.

(Note) Speed clamping is not applied to synchronous feed and thread cutting.

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 selected with the parameter. (Refer to the section "Deceleration check")

The in-position width is set into parameter the servo parameter "#2224 sv024" or "#2077 G0inps", "#2078 G1inps".



Command format

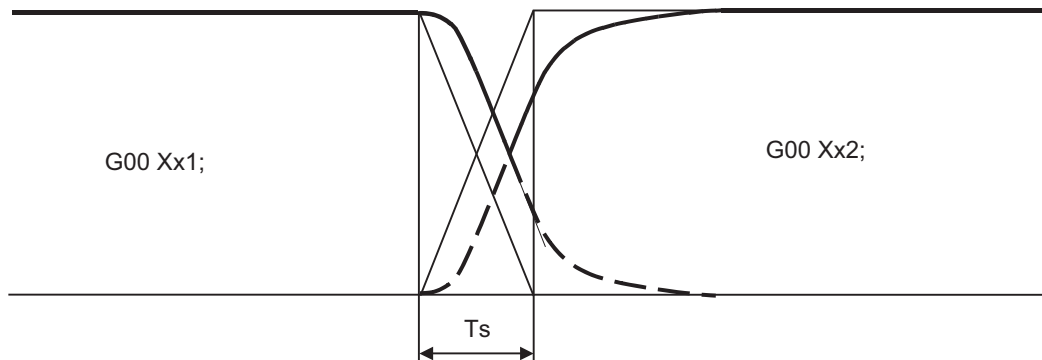
G09 G01 (G02, G03) ; ... Exact stop check

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

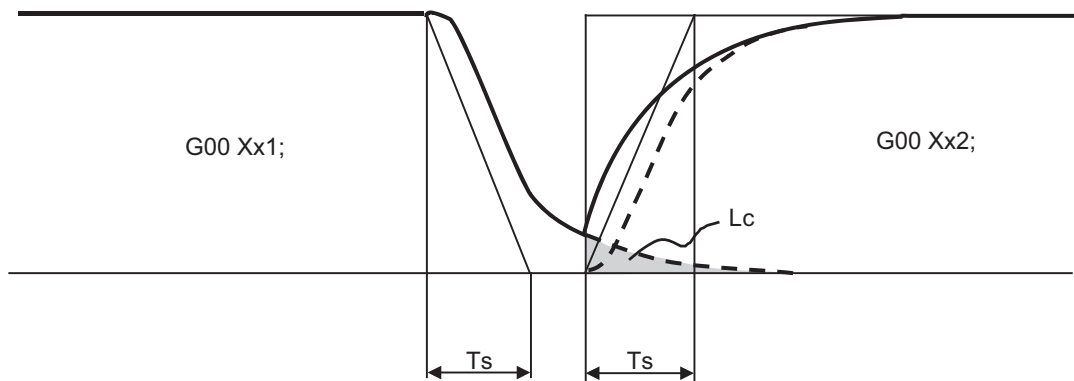


Detailed description

[With continuous cutting feed]



[With cutting feed in-position check]



Ts : Cutting feed acceleration/deceleration time constant

Lc : In-position width

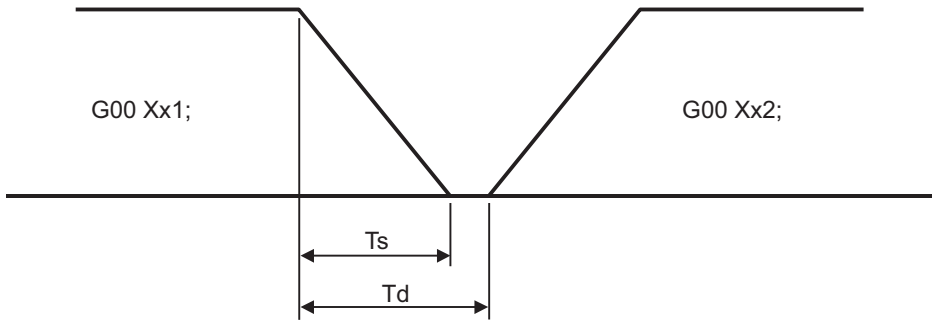
As shown in the figure below, the remaining distance (shaded area in the above figure) of the previous block when the next block is started can be set into the servo parameter "#2224 sv024" as the in-position width "Lc". The in-position width is designed to reduce the roundness at the workpiece corners 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.

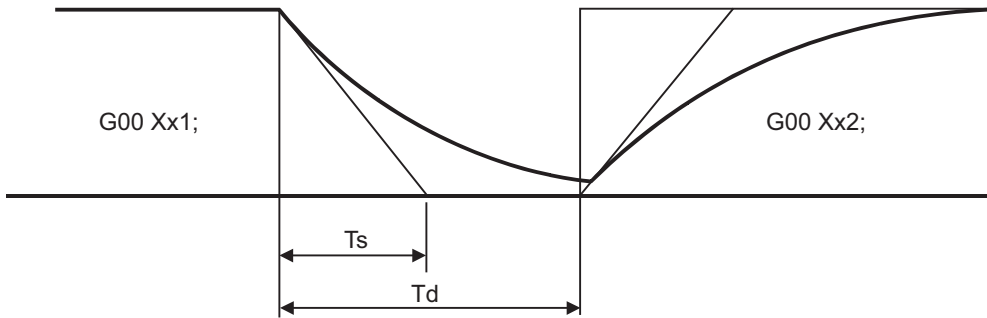
With deceleration check

(1) With linear acceleration/deceleration



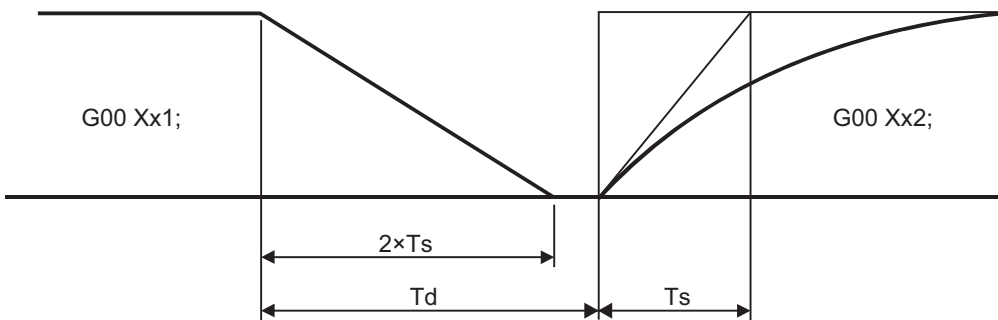
TS: Acceleration/deceleration time constant
 Td: Deceleration check time $Td = Ts + (0 \text{ to } 7\text{ms})$

(2) With exponential acceleration/deceleration



TS: Acceleration/deceleration time constant
 Td: Deceleration check time $Td = 2 \times Ts + (0 \text{ to } 7\text{ms})$

(3) With exponential acceleration/linear deceleration



TS: Acceleration/deceleration time constant
 Td: Deceleration check time $Td = 2 \times Ts + (0 \text{ to } 7\text{ms})$

The time required for the deceleration check during cutting feed is the longest among the cutting feed deceleration check times of each axis determined by the cutting feed acceleration/deceleration time constants and by the cutting feed acceleration/ deceleration mode of the axes commanded simultaneously.

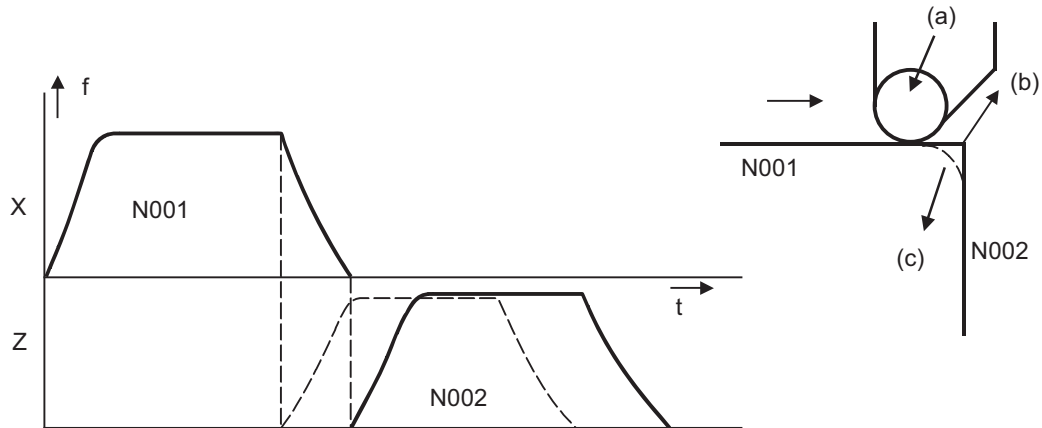
(Note 1) To execute exact stop check in a fixed cycle cutting block, insert command G09 into the fixed cycle subprogram.



Program example

N001 G09 G01 X100.000 F150 ;	The commands in the following block are started once the deceleration check time or in-position state has been checked after the machine has decelerated and stopped.
N002 Z100.000 ;	

[Exact stop check result]



(a) Tool

(b) With G09

(c) Without G09

f: Commanded speed

t: Time

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.

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



Function and purpose

The deceleration check is a function that determines the method of the check at the completion of the axis movement block's movement.

The deceleration check includes the in-position check and commanded speed check method.

The G0 and G1 deceleration check method combination can be selected. (Refer to section "Deceleration check combination".)

With this function, the deceleration check in the reverse direction of G1 -> G0 or G1 -> G1 can be changed by changing the parameter setting.

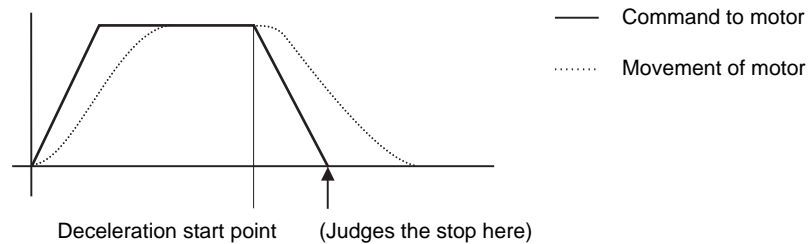


Detailed description

Types of deceleration check

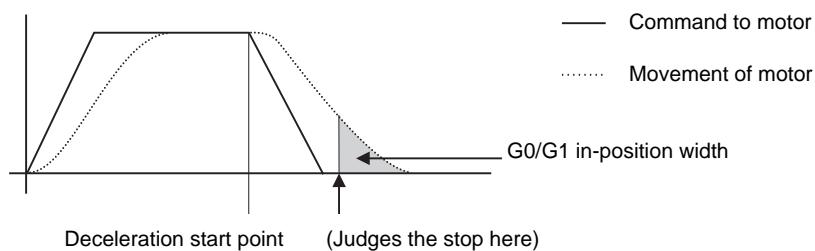
(1) Command speed check

With the commanded speed check, the completion of deceleration is judged when the command to the motor is completed.



(2) In-position check

With the in-position check, the completion of deceleration is judged when the motor moves to the in-position width designated with the parameter.



Designating deceleration check

The deceleration check by designating a parameter includes "deceleration check specification type 1" and "deceleration check specification type 2". The setting is selected with the parameter "#1306 InpsTyp".

- (1) Deceleration check specification type 1 ("#1306 InpsTyp" = 0)

The G0 and G1 deceleration check method can be selected with the base specification parameter deceleration check method 1 (#1193 inpos) and deceleration check method 2 (#1223 AUX07/BIT-1).

Parameter	Rapid traverse command	Parameter	Other than rapid traverse command (G1: other than G0 command)	
			G1+G9 -> XX	G1 -> XX
inpos (#1193)	G0 -> XX (G0+G9 -> XX)	AUX07/BIT-1 (#1223/BIT-1)	G1+G9 -> XX	G1 -> XX
0	Command deceleration check	0	Command deceleration check	No deceleration check
1	In-position check	1	In-position check	

(Note 1) XX expresses all commands.

(Note 2) "#1223 aux07" is the part system common parameter.

- (2) Deceleration check specification type 2 ("#1306 InpsTyp" = 1)

Rapid traverse and cutting feed in-position are designated with the "#1193 inpos" parameter.

Parameter	Command block		
	G0	G1+G9	G1
#1193 inpos			
0	Command deceleration check	Command deceleration check	No deceleration check
1	In-position check	In-position check	No deceleration check

(Note 1) "#1193 inpos" is the parameter per part system.

(Note 2) "G0" means the rapid traverse, and "G1" means the cutting feed.

7.13.1 G1 -> G0 Deceleration Check



Detailed description

In G1 -> G0 continuous blocks, the parameter "#1502 G0lpfg" can be changed to change the deceleration check in the reverse direction.

	Same direction	Reverse direction
G0lpfg:0		<p>(a) The acceleration rate is excessive due to the composite speed of G1 and G0.</p>
G0lpfg:1		<p>Command deceleration</p>



Program example

When there is a deceleration check in the movement of several axes:

(1)

G91 G1 X100. Z100. F4000 ; G0 X-100. Z120. ;	A deceleration check is carried out, because the X axis moves in the reverse direction in the program.
---	--

(2)

G91 G1 X100. Z-100. F4000 ; G0 X80. Z100. ;	A deceleration check is carried out, because the Z axis moves in the reverse direction in the program.
--	--

(3)

G90 G1 X100. Z100. F4000 ; G0 X80. Z100. ;	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 Z0)
---	--

(4)

G91 G1 X100. Z100. F4000 ; G0 X100. Z100. ;	A deceleration check is not carried out, because both the X axis and the Z axis move in the same direction in the program.
--	--

(5)

G91 G1 X100. Z80. F4000 ; G0 X80. ;	A deceleration check is not carried out, because the X axis moves in the same direction, and there is no Z axis movement command in the program.
--	--

7.13.2 G1 -> G1 Deceleration Check



Detailed description

In G1 -> G1 continuous blocks, the parameter "#1503 G1lpfg" can be changed to change the deceleration check of the reverse direction.

	Same direction	Reverse direction
G1lpfg:0		
G1lpfg:1		



Program example

When there is a deceleration check in the movement of several axes:

(1)

G91 G1 X100. Z100. F4000 ; G1 X-100. Z120. ;	A deceleration check is carried out, because the X axis moves in the reverse direction in the program.
---	--

(2)

G91 G1 X100. Z-100. F4000 ; G1 X80. Z100. ;	A deceleration check is carried out, because the X axis moves in the reverse direction in the program.
--	--

(3)

G90 G1 X100. Z100. F4000 ; G1 X80. Z120. ;	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 Z0)
---	--

(4)

G91 G1 X100. Z100. F4000 ; G1 X100. Z100. ;	A deceleration check is not carried out, because both the X axis and the Z axis move in the same direction in the program.
--	--

(5)

G91 G1 X100. Z80. F4000 ; G1 X80. ;	A deceleration check is not carried out, because the X axis moves in the same direction, and there is no Z axis movement command in the program.
--	--

7.14 Automatic Corner Override ; G62



Function and purpose

When cutting with nose R compensation, to prevent machining surface distortion due to the increase in the cutting load during cutting of corners, this command automatically applies an override on the cutting feedrate so that the cutting amount is not increased for a set time at the inside corner or automatic corner R.

Automatic corner override is valid until the nose R compensation cancel (G40), exact stop check mode (G61), tapping mode (G63), or cutting mode (G64) command is issued.



Command format

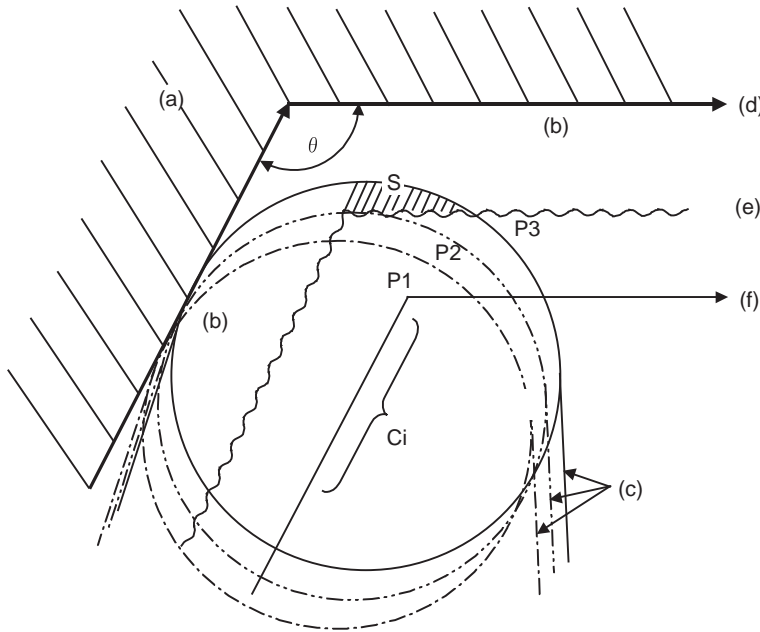
G62 ; ... Automatic Corner Override
--



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.



- (a) Workpiece
 - (b) Machining allowance
 - (c) Tool
 - (d) Programmed path (finished shape)
 - (e) Workpiece surface shape
 - (f) Nose R center path
- θ : Max. angle at inside corner
 Ci : Deceleration range (IN)

[Operation]

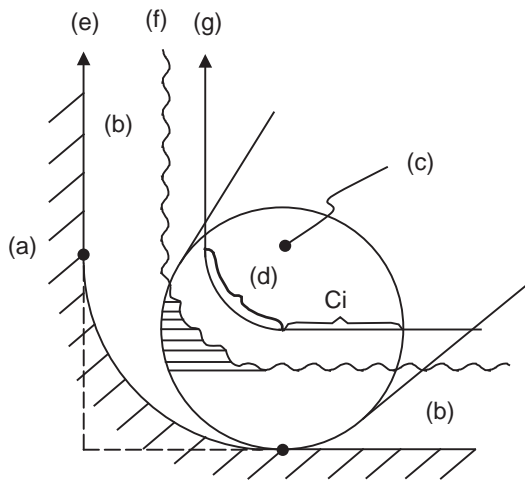
- (1) When automatic corner override is not to be applied :
 When the tool moves in the order of P1 -> P2 -> P3 in the above figure, the machining allowance at P3 increase by an amount equivalent to the area of shaded section S and so that tool load increases.
- (2) When automatic corner override is to be applied :
 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.

#	Parameter	Setting range
#8007	OVERRIDE	0 to 100 [%]
#8008	Max. angle at inside corner	0 to 180 [°]
#8009	DSC. ZONE	0 to 99999.999 [mm] or 0 to 3937.000 [inch]

Automatic corner R



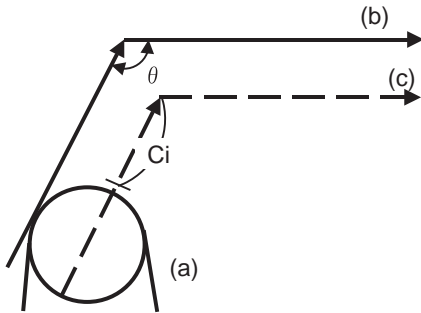
- | | |
|------------------------|-----------------------------|
| (a) Workpiece | (b) Machining allowance |
| (c) Corner R center | (d) Corner R section |
| (e) Programmed path | (f) Workpiece surface shape |
| (g) Nose R center path | |

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



Application example

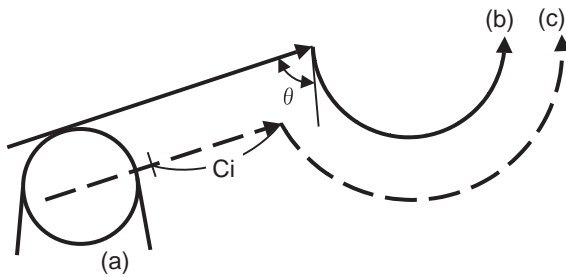
(1) Linear - linear corner



- (a) Tool
- (b) Program
- (c) Nose R center

The override set in the parameter is applied at Ci.

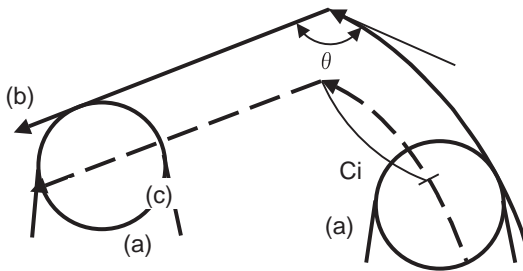
(2) Linear - arc (outside offset) corner



- (a) Tool
- (b) Program
- (c) Nose R center

The override set in the parameter is applied at Ci.

(3) Arc (inside offset) - linear corner

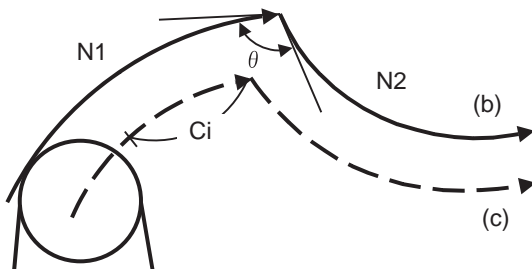


- (a) Tool
- (b) Program
- (c) Nose R center

The override set in the parameter is applied at Ci.

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

(4) Arc (inside offset) - arc (outside offset) corner



- (a) Tool
- (b) Program
- (c) Nose R center

The override set in the parameter is applied at Ci.



Relation with other functions

Function	Override at corner
Cutting feed override	Automatic corner override is applied after cutting feed override has been applied.
Override cancel	Automatic corner override is not canceled by override cancel.
Speed clamp	Valid after automatic corner override
Dry run	Automatic corner override is invalid.
Synchronous feed	Automatic corner override is applied to the synchronous feedrate.
Thread cutting	Automatic corner override is invalid.
G31 skip	Program error occurs with G31 command during nose R compensation.
Machine lock	Valid
Machine lock high speed	Automatic corner override is invalid.
G00	Invalid
G01	Valid
G02,G03	Valid



Precautions

- (1) Automatic corner override 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 nose R compensation mode is entered.
- (3) Automatic corner override will not be applied on a corner where the nose R compensation is started or canceled.
- (4) Automatic corner override will not be applied on a corner where the nose R 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) Automatic corner override will not be applied when the maximum angle in the parameter is set to 0 or 180.
- (9) Automatic corner override will not be applied when the override in the parameter is set to 0 or 100.

7.15 Tapping Mode ; G63



Function and purpose

The G63 command allows the control mode best suited for tapping to be entered, as indicated below:

- (1) Cutting override is fixed at 100%.
- (2) Deceleration commands at joints between blocks are invalid.
- (3) Feed hold is invalid.
- (4) Single block is invalid.
- (5) In-tapping mode signal is output.

G63 is released by the exact stop check mode (G61), automatic corner override (G62), or cutting mode (G64) command.

The machine is in the cutting mode status when its power is turned ON.



Command format

G63; ... Tapping mode

7.16 Cutting Mode ; G64



Function and purpose

The G64 command allows the cutting mode in which smooth cutting surfaces are obtained to be established. Unlike the exact stop check mode (G61), the next block is executed continuously with the machine not decelerating and stopping between cutting feed blocks in this mode.

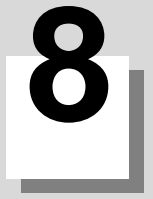
G64 is released by the exact stop check mode (G61), automatic corner override (G62), or tapping mode (G63).

The machine is in the cutting mode status when its power is turned ON.



Command format

G64; ... Cutting mode



Dwell

8.1 Dwell (Time Designation) ; G04



Function and purpose

The machine movement is temporarily stopped by the program command to make the waiting time state. Therefore, the start of the next block can be delayed. The waiting time state can be canceled by inputting the skip signal.



Command format

G04 X/U_/P_ ; ... Dwell (Time designation)

X/U/P	Dwell time
-------	------------

The input command unit for the dwell time depends on the parameter.

In addition to the address P and X, the address U (actually, the address corresponding to the X-axis designated with the #1014 incax) can be used. Note that this is invalid when the #1076 AbsInc is set to 0.

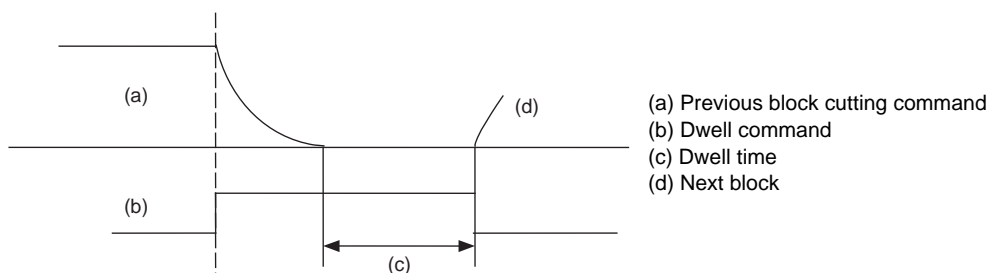


Detailed description

- (1) When designating the dwell time with X or U, the decimal point command is valid.
- (2) When designating the dwell time with P, the availability of the decimal point command can be selected with the parameter (#8112). When the decimal point command is invalid in the parameter setting, the command below the decimal point issued with P is ignored.
- (3) When the decimal point command is valid or invalid, the dwell time command range is as follows.

Command range when the decimal point command is valid	Command range when the decimal point command is invalid
0 to 99999.999(s)	0 to 99999999 (ms)

- (4) The dwell time setting unit applied when there is no decimal point can be made 1s by setting 1 in the parameter "#1078 Decpt2". This is effect only for X, U and P for which the decimal command is valid.
- (5) When a cutting command is in the previous block, the dwell command starts calculating the dwell time after the machine has decelerated and stopped. When it is commanded in the same block as an M, S, T or B command, the calculation starts simultaneously.
- (6) The dwell is valid during the interlock.
- (7) The dwell is valid even for the machine lock.
- (8) The dwell can be canceled by setting the parameter "#1173 dwlskp" beforehand. If the set skip signal is input during the dwell time, the remaining time is discarded, and the following block will be executed.





Program example

Command	Dwell time [s]			
	#1078 Decpt2 = 0		#1078 Decpt2 = 1	
	G04P DECIMAL PNT-N	G04P DECIMAL PNT-P	G04P DECIMAL PNT-N	G04P DECIMAL PNT-P
G04 X500 ;	0.5		500	
G04 X5000 ;	5		5000	
G04 X5. ;	5		5	
G04 X#100 ;	1000		1000	
G04 U500 ;	0.5		500	
G04 U5000 ;	5		5000	
G04 U5. ;	5		5	
G04 U#100 ;	1000		1000	
G04 P5000 ;	5		5	5000
G04 P12.345 ;	0.012	12.345	0.012	12.345
G04 P#100 ;	1	1000	1	1000

(Note 1) The above examples are the results under the following conditions.

- Input setting unit 0.001mm or 0.0001inch
- #100 = 1000 ;

(Note 2) "G04P DECIMAL PNT-P" is a control parameter (#8112).

(Note 3) If the input setting unit is 0.0001inch, the X before G04 will be multiplied by 10. For example for "X5. G04 ;", the dwell time will be 50 seconds.



Precautions and restrictions

- (1) When using this function, command X or U after G04 in order to make sure that the dwell is based on X or U.



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 with this controller, and up to 4 groups can be commanded in a single block.

The number of M commands that can be issued within the same block is determined by the parameter setting (#12005 Mfig).

(Example) G00 Xx Mm1 Mm2 Mm3 Mm4 ;

When five or more commands are issued in a block, only the last four will be valid.

Whether to BCD output or binary output the second miscellaneous function can be selected by a parameter. The eight commands of M00, M01, M02, M30, M96, M97, M98 and M99 are used as auxiliary commands for specific objectives and so they cannot be used as general auxiliary commands. Therefore, 92 miscellaneous functions are available.

Reference should be made to the instructions issued by the machine manufacturer 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 tape reader has read this function, it stops reading the next block. As far as the NC system's functions are concerned, only the tape reading is stopped. Whether such machine functions 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 tape reader reads the M01 command when the optional stop switch on the machine operation board is ON, it will stop and performs the same operation as the M00.

If the optional stop switch is OFF, the M01 command is ignored.

(Example)	The state and operation of optional stop switch
N10 G00 X1000 ;	Stops at N11 when switch is ON
N11 M01 ;	Next command (N12) is executed without stopping at N11 when switch is OFF
N12 G01 X2000 Z3000 F600 ;	
:	

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 tape rewinding. Whether the tape is actually rewound 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 tape rewinding 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 rewinding 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.

(Note 1) Independent signals are also output respectively for the M00, M01, M02 and M30 commands and these outputs are each reset by pressing the reset key.

(Note 2) M02 or M30 can be assigned by manual data input (MDI).

At this time, commands can be issued simultaneously with other commands just as with the tape.

Macro interruption; M96, M97

M96 and M97 are M codes for user macro interrupt control.

The M code for user macro interrupt control is processed internally, and is not output externally.

To use M96 and M97 as miscellaneous functions, change to another M code with the parameter (#1109 subs_M, #1110 M96_M and #1111 M97_M).

Subprogram call/completion : M98, M99

These commands are used as the return instructions from branch destination subprograms and branches to subprograms.

M98 and M99 are processed internally and M code signals and strobe signals are not output.

Internal processing with M00/M01/M02/M30 commands

Internal processing suspends pre-reading when the M00, M01, M02 or M30 command has been read. Other tape rewinding operations and the initialization of modals by resetting differ according the machine specifications.

9.2 Secondary Miscellaneous Functions (A8-digits, B8-digits or C8-digits)



Function and purpose

These serve to assign the indexing table positioning and etc. In this controller, they are assigned by an 8-digit number from 0 to 99999999 following address A, B or C. The machine maker determines which codes correspond to which positions.



Detailed description

Select the address A, B or C that is used for the second miscellaneous function by the 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 up to 4 sets in a block. The number of M commands to be issued within a same block is determined by the parameter setting (#12011 Bfig).
 Whether to BCD output or binary output the 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 which 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

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
- Deep hole drilling cycle 2 commands

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.



Detailed description

The index table index function carries out following operations.

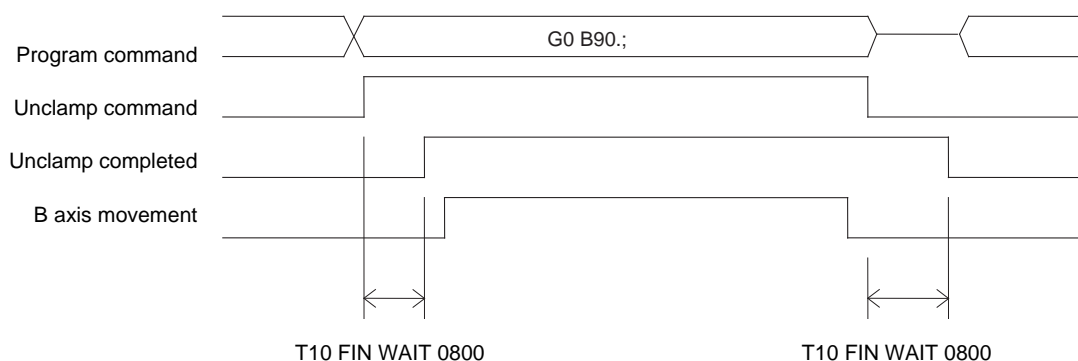
Command format example

(Example) G00 B90 ;

B: The axis that was designated by the index axis with parameter "#2076 index x".

- (1) Set "1" to the "index_x" parameter (#2076) of the axis in which index table indexing will be carried out.
- (2) The movement command (either absolute or incremental) for the selected axis is executed with the program command.
- (3) An unclamp operation is carried out before the axis movement.
- (4) The commanded axis movement starts after the unclamp operation completes.
- (5) The clamp operation is carried out after the movement is completed.
- (6) The next block is processed after the clamp operation completes.

[Operation time chart]





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 (G0/G1) at that time.
- (3) The unclamp command for the indexing axes is also issued when the index table indexing axes are commanded in the same block as other axes. Thus, the movement of other axes commanded in the same block is not carried out until the unclamp operation completes.
Note that the movement of other axes commanded in the same block is carried out during a non-interpolation commands.
- (4) Index table indexing axes are used as normal rotation axes, but this function performs an unclamp operation even for linear axes.
- (5) If some error that makes unclamp command OFF occurs during indexing axis movement in automatic operation, the unclamp state will remain, and the indexing axis will execute a deceleration stop. Other axes commanded in the same block will also execute a deceleration stop, except during non-interpolation commands.
- (6) If the axis movement is interrupted by an interlock, etc., during indexing axis movement, the unclamp state will remain.
- (7) The clamp and unclamp operations are not executed when the movement commands of the index table indexing axis are successively issued.
Note that the clamp and unclamp operations are executed even when the movement commands are continued during single block operation.
- (8) Make sure that the command position is at a position where clamping is possible.

9.4 M Code Output during Axis Traveling ; G117



Function and purpose

This function controls the timing at which miscellaneous functions are output, and it outputs a miscellaneous function when axis reaches at the designated position movement.



Command format

G117 Xx1 Zz1 Cc1 Mm1 Mm2 Mm3 Mm4; ... M Code Output during Axis Traveling

Xx1,Zz1,Cc1	Movement start points
Mm1,Mm2,Mm3,Mm4	Miscellaneous function



Detailed description

- (1) This command is issued independently immediately before the block with the movement command that activates the miscellaneous function.
- (2) Single block stop does not apply to this command.
- (3) The maximum number of groups to which the miscellaneous functions in the G117 block can be issued is as follows:

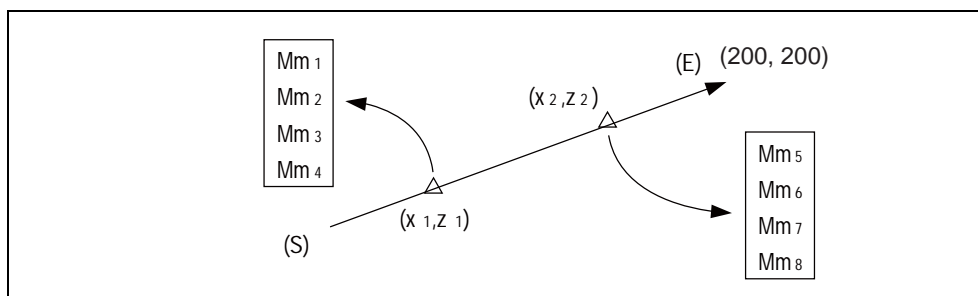
M commands	4 sets
S commands	2 sets
T commands	1 set
2nd miscellaneous function	1 set

- (4) This command can be issued in up to two consecutive blocks.
When issued in three or more consecutive blocks, the last two blocks will be valid.



Program example

```
G117 Xx1 Zz1 Mm1 Mm2 Mm3 Mm4 ;
G117 Xx2 Zz2 Mm5 Mm6 Mm7 Mm8 ;
G01 X200 Z200 ;
:
```



(E) End point

(S) Start point

10

Spindle Functions

10.1 Spindle Functions



Function and purpose

These functions are assigned with an 8-digit (0 to 99999999) number following the address S, and one group can be assigned in one 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.

10.2 Constant Surface Speed Control ; G96,G97



Function and purpose

These commands automatically control the spindle rotation speed in line with the changes in the radius coordinate values as cutting proceeds in the diametrical direction, and they serve to keep the cutting point speed constant during the cutting.



Command format

G96 S__ P__ ; ... Constant surface speed ON

S	Surface speed
P	Constant surface speed control axis

G97 ; ... Constant surface speed cancel



Detailed description

- (1) The constant surface speed control axis is set by parameter "#1181 G96_ax".
 - 0: Fixed at 1st axis (P command invalid)
 - 1: 1st axis
 - 2: 2nd axis
 - 3: 3rd axis
- (2) When the above-mentioned parameter is not zero, the constant surface speed control axis can be assigned by address P.
(Example) G96_ax = 1

Program	Constant surface speed control axis
G96 S100 ;	1st axis
G96 S100 P3 ;	3rd axis

- (3) Example of selection program and operation

```
F300;
:
G90 G96 G01 X50. Z100. S200 ; The spindle rotation speed is controlled so that the surface speed is 200m/min.
:
G97 G01 X50. Z100. F300 S500 ; The spindle rotation speed is controlled to 500r/min.
:
M02 ; The modal returns to the initial value.
```

- (4) The spindle subject to control is determined with the following.
 - For multiple-spindle control I (#1300 ext36 bit0 = 0), the spindle is determined by the spindle selection command in the G group 20.
 - For multiple-spindle control II (#1300 ext36 bit0 = 1), the spindle is determined by the spindle selection signal (SWS) from the PLC. .



Precautions

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 or may cause damage to the operators. Thus 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

(1) When the parameter "1146 Sclamp" is set to "0".

G96 S200 ; ... The spindle rotation speed is controlled so that the surface speed is 200m/min.
 G92 S4000 Q200 ; ... The spindle rotation speed is clamped up to 4000r/min and down to 200r/min.
 M3 ; ... The rotation command to the spindle

(2) When the parameter "1146 Sclamp" is set to "1".

G92 S4000 Q200 ; ... The spindle rotation speed is clamped up to 4000r/min and down to 200r/min.
 G96 S200 ; ... The spindle rotation speed is controlled so that the surface speed is 200m/min.
 M3 ; ... The rotation command to the spindle

(Note) For safety, issue the rotation command to the spindle after G92.

WARNING

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 result in breakage of tools or machine or may cause damage to the operators.

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

G92 S__ Q__ ; ... Spindle Clamp Speed Setting

S	Maximum clamp rotation speed
Q	Minimum clamp rotation speed



Detailed description

- Besides this command, parameters can be used to set the rotation speed range up to 4 stages in 1 r/min units to accommodate gear selection between the spindle and spindle motor. The lowest upper limit and highest lower limit are valid among the rotation speed ranges based on the parameters and based on "G92 S_ Q_ ;".
 - Set in the parameters "#1146 Sclamp" and "#1227 aux11/bit5" whether to carry out rotation speed clamp only in the constant surface speed mode or even when the constant surface speed is canceled.
- (Note 1) 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 EXECUTION	NO ROTATION SPEED CLAMP

(Note2) The address Q following the G92 command is handled as the spindle speed clamp command regardless of the constant surface mode.

- The command value of the spindle clamp 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.
When the power is turned ON, the setting will be cleared to 0.



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. Even when G92 S0 is commanded, the value of Q_ is kept enabled and Q_ is greater than S0.
- (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

1. 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/C Axis Control



Function and purpose

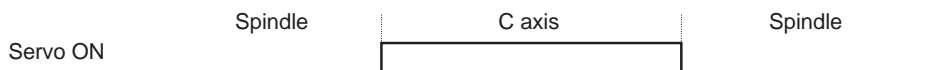
This function enables a spindle (MDS-D or later) to also be used as a C axis (rotation axis) by an external signal.



Detailed description

Spindle/C axis changeover

Changeover between the spindle and C axis is done by the C axis Servo ON signal.



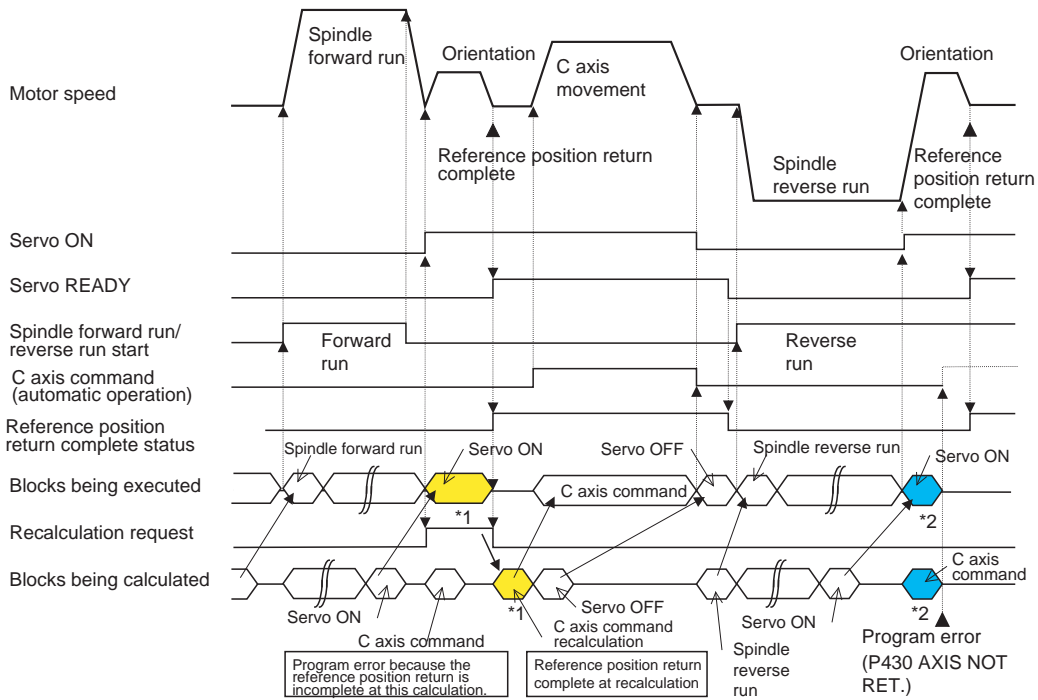
At servo OFF Spindle (C axis control not possible)

At servo ON C axis (spindle control not possible)

The C axis is in a reference position return incomplete state.

- (1) Reference position return state
Reference position return is incomplete when the Z phase has not been passed.
Reference position return is complete when the Z phase has been passed.
- (2) C axis position data
The NC's internal C axis position data is updated even for the spindle rotation during spindle control.
The C axis coordinate value counter is held during spindle control, and is updated according to the amount moved during spindle control when the C axis servo READY is turned ON. (The C axis position at servo ON may differ from the position just before the previous servo OFF.)

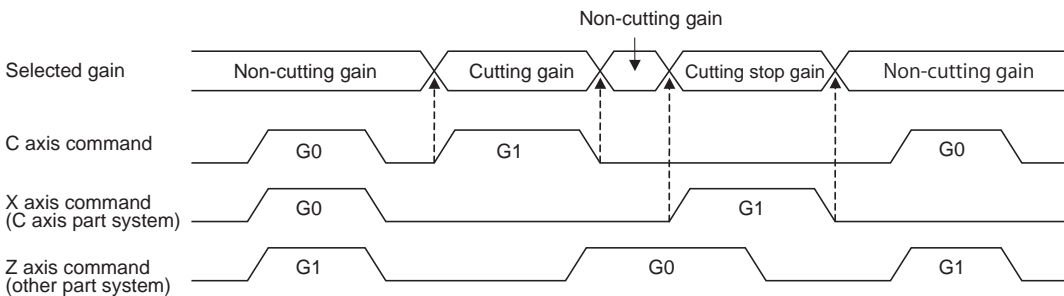
Changeover timing chart example



- (Note 1) For axis commands, the reference position return complete is checked at calculation. Thus, when the C axis servo ON command and C axis command are continuous, the program error (P430) will occur as shown above in *2.
- In response to this kind of situation, the following two processes must be carried out on user PLC, as shown above in *1.
- Input the recalculation request signal with a servo ON command.
 - Wait for the completion of the servo ON command until the C axis enters a servo READY state.

C axis gain

The C axis gain is changed over (the optimum gain is selected) by the C axis cutting condition. During C axis cutting feed, cutting gain is applied. During other axis' cutting feed (C axis face turning), non-cutting stop gain is applied. Non-cutting gain is applied in all other cases.



- (Note 1) The cutting feed of other part systems does not affect the C axis gain selection.
- (Note 2) There are 1st to 3rd cutting gains, which are selected with the ladder.

Deceleration check in movement including spindle/C-axis

The deceleration check in a movement command including the spindle/C-axis is as the table described below when the following condition is fulfilled. When the different values are set for the position loop gain in non-cutting mode (spindle parameter #3203 PGCO) and the position loop gain in cutting mode (spindle parameter #3330 PGC1 to #3333 PGC4). That is because a vibration and so on occurs in the machine when the gain is changed during the axis movement.

Parameter	Rapid traverse command	Parameter	Other than rapid traverse command (G1: other than G0 command)	
			G1 -> G0 (G0+G9 -> XX)	G1 -> G1
Inpos (#1193)	G0 -> XX (G0+G9 -> XX)	AUX07/bit1 (#1223/bit1)	G1 -> G0 (G0+G9 -> XX)	G1 -> G1
0	Command deceleration check	0	In-position check (Applicable only to SV024)	No deceleration check
1	In-position check	1		

(Note 1) When G1 command is issued, the in-position check is performed regardless of the deceleration check parameter.

(Note 2) XX expresses all commands.



Precautions and restrictions

- (1) The program error (P430) will occur if a C axis command is issued during servo OFF or during orientation.
- (2) Do not execute a servo OFF during a C axis command.
The remaining C axis commands will be cleared at servo ON.
(If servo OFF is executed during C axis control, the feed will stop and spindle control will occur.)
- (3) If servo ON is executed during spindle rotation, the rotation will stop and C axis control will occur.
- (4) Dog-type reference position return is not possible for the C axis.
Set the reference position return to the orientation method in the parameters (Spindle base specifications parameters "#3106 zrn_typ/bit8"= 0), or set the axis to "Axis without reference position (zero point)" (Zero point return parameters "#2031 noref: 1").

10.5 Spindle Synchronization



Function and purpose

In a machine having two or more spindles, this function controls the rotation speed and phase of one spindle (basic spindle) in synchronization with the rotation of the other spindle (synchronous spindle).

The function is used when the rotation speed of the two spindles must be matched, for example, if a workpiece grasped by the first spindle is to be grasped by a second spindle, or if the spindle rotation speed has to be changed when one workpiece is grasped by both the first and second spindles.

There are two types of spindle synchronization: Spindle synchronization I and Spindle synchronization II.

The spindle synchronization control I

The designation of the synchronous spindle and start/stop of the synchronization are executed by commanding G codes in the machining program.

The spindle synchronization function II

The selections of the synchronized spindle and synchronization start, etc., are all designated from the PLC. Refer to the instruction manual issued by the machine tool builder for details.

Common setting for the spindle synchronization control I and II

When the spindle synchronization control is carried out, the followings must be set.

- Chuck close
- Error temporary cancel
- Phase monitor
- Multi-speed acceleration/deceleration

For details, refer to the "Precautions for Using Spindle Synchronization Control".

10.5.1 Spindle Synchronization Control I ; G114.1



Function and purpose

With the spindle synchronization control I, the designation of the synchronous spindle and start/stop of the synchronization are executed by commanding G codes in the machining program.



Command format

G114.1 H__ D__ R__ A__ ; ... Spindle synchronization ON

H	Basic spindle selection
D	Synchronous spindle selection
R	Synchronous spindle phase shift amount
A	Spindle synchronization acceleration/deceleration time constant

G113 ; ... Spindle synchronization cancel

Spindle synchronization control ON (G114.1) command designates the basic spindle and synchronous spindle, and synchronizes the two designated spindles. By commanding the synchronous spindle phase shift amount, the phases of the basic spindle and synchronous spindle can be aligned.

Spindle synchronization cancel (G113) cancels the synchronous state of the two spindles rotating in synchronization with the spindle synchronization command.

Address	Meaning of address	Command range (unit)	Remarks
H	Basic spindle selection Select the No. of the spindle to be used as the basic spindle from the two spindles.	1 to 6 1: 1st spindle 2: 2nd spindle 3: 3rd spindle 4: 4th spindle 5: 5th spindle 6: 6th spindle	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 a spindle not serially connected is commanded, a program error (P700) will occur.
D	Synchronous spindle selection Select the No. of the spindle to be synchronized with the basic spindle from the two spindles.	1 to 6 or -1 to -6 1: 1st spindle 2: 2nd spindle 3: 3rd spindle 4: 4th spindle 5: 5th spindle 6: 6th spindle	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 spindle as that commanded for the basic spindle selection is designated, a program error (P33) will occur. The rotation direction of the synchronous spindle in respect to the basic spindle is commanded with the D sign. If a spindle not serially connected is commanded, a program error (P700) will occur.
R	Synchronous spindle phase shift amount 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) will occur. The commanded shift amount will be effective in the clockwise direction of the basic spindle. If there is no R command, the phases will not be aligned.
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) will occur. 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.



Detailed description

Rotation speed and rotation direction

- (1) The rotation speed and rotation direction of the basic spindle and synchronous spindle during spindle synchronization are the rotation speed and rotation direction commanded for the basic spindle. Note that the rotation direction of the synchronous spindle can be reversed from the basic spindle through the program.
- (2) The basic spindle's rotation speed and rotation direction can be changed during spindle synchronization.
- (3) The synchronous spindle's rotation command is also valid during spindle synchronization.
When spindle synchronization is commanded, if neither a forward run command nor reverse run command is commanded for the synchronous spindle, the synchronization standby state will be entered without starting the synchronous spindle's rotation. If the forward run command or reverse run command is input in this state, the synchronous spindle will start rotation. The synchronous spindle's rotation direction will follow the direction commanded in the program.
If spindle stop is commanded for the synchronous spindle during spindle synchronization (when both the forward run and reverse run commands are turned OFF), the synchronous spindle rotation will stop.
- (4) The rotation speed command (S command) and constant surface speed control are invalid for the synchronous 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 basic spindle even during spindle synchronization.

Rotation synchronization

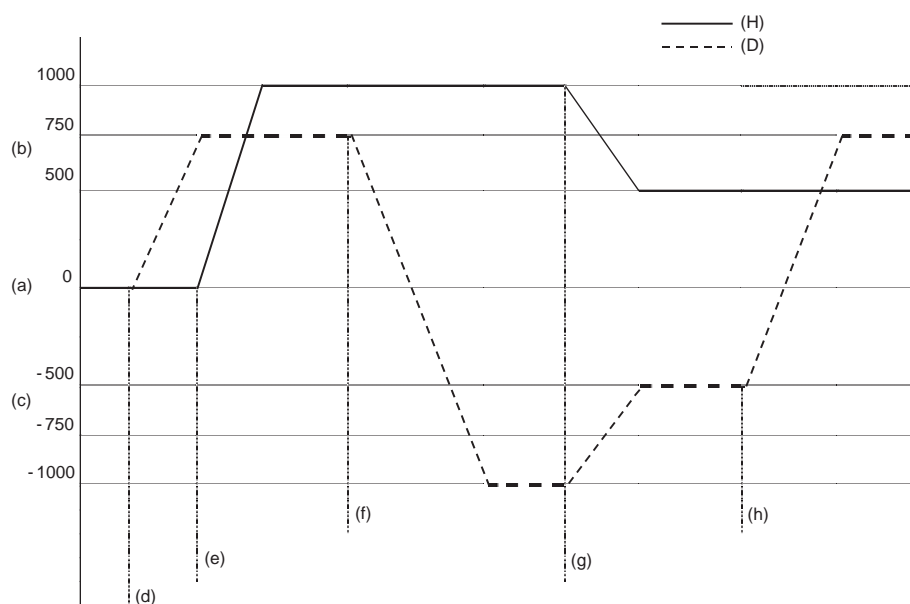
- (1) When rotation synchronization control (command with no R address) is commanded with the G114.1 command, the synchronous spindle rotating at an arbitrary rotation speed will accelerate or decelerate to the rotation speed commanded beforehand for the basic spindle, and will enter the rotation synchronization state.
- (2) If the basic 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 basic spindle can be controlled to the constant surface speed even when two spindles are grasping one workpiece.
- (4) Operation will take place in the following manner.

```

M23 S2=750 ;      ..... Forward rotate 2nd spindle (synchronous spindle) at 750 r/min (speed command)
:
M03 S1=1000 ;    ..... Forward rotate 1st spindle (basic spindle) at 1000 r/min (speed command)
:
G114.1 H1 D-2 ; ..... Synchronize 2nd spindle (synchronous spindle) to 1st spindle (basic spindle) with reverse
run.
:
S1=500 ;         ..... Change 1st spindle (basic spindle) rotation speed to 500 r/min.
:
G113 ;          ..... Cancel spindle synchronization

```

<Operation>



(H) Basic spindle

(D) Synchronous spindle

(a) Rotation speed (r/min)

(b) Forward run

(c) Reverse run

(d) 2nd spindle (synchronous spindle) forward run

(e) 1st spindle (basic spindle) forward run

(f) 2nd spindle (synchronous spindle) reverse run synchronization

(g) 1st spindle (basic spindle) rotation speed change

(h) Spindle synchronization cancel

Phase synchronization

- (1) When phase synchronization (command with R address) is commanded with the G114.1 command, the synchronous spindle rotating at an arbitrary rotation speed will accelerate or decelerate to the rotation speed commanded beforehand for the basic 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 basic 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 basic spindle can be controlled to the constant surface speed even when two spindles are grasping one workpiece.

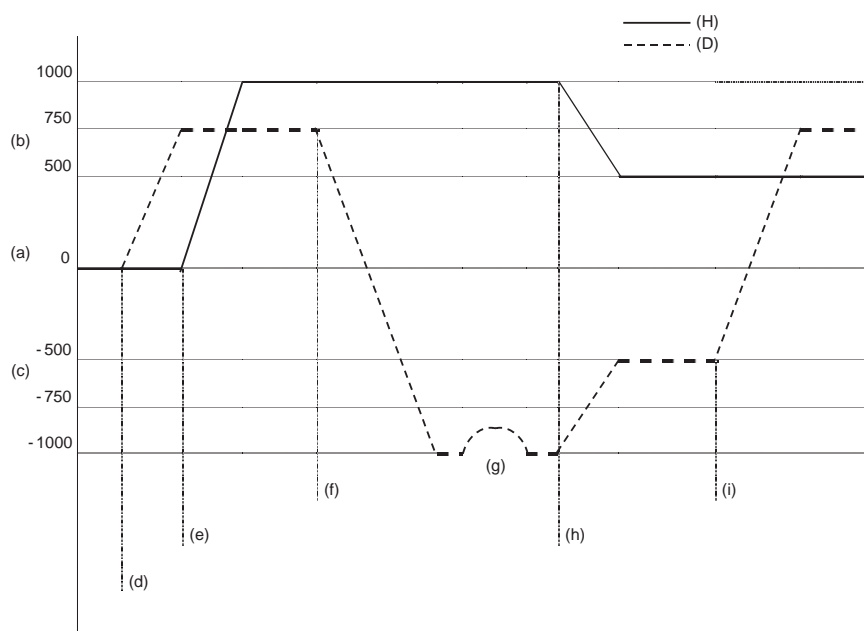
(4) Operation will take place in the following manner.

```

M23 S2=750 ;      ..... Forward rotate 2nd spindle (synchronous spindle) at 750 r/min (speed command)
:
M03 S1=1000 ;    ..... Forward rotate 1st spindle (basic spindle) at 1000 r/min (speed command)
:
G114.1 H1 D-2 Rxx ; ..... Synchronize 2nd spindle (synchronous spindle) to 1st spindle (basic spindle) with
reverse run. Shift phase of synchronous spindle by R command value.
:
:
S1=500 ;         ..... Change 1st spindle (basic spindle) rotation speed to 500 r/min.
:
G113 ;          ..... Cancel spindle synchronization

```

<Operation>



- | | |
|---|-------------------------|
| (H) Basic spindle | (D) Synchronous spindle |
| (a) Rotation speed (r/min) | (b) Forward run |
| (c) Reverse run | |
| (d) 2nd spindle (synchronous spindle) forward run | |
| (e) 1st spindle (basic spindle) forward run | |
| (f) 2nd spindle (synchronous spindle) reverse run synchronization | |
| (g) Phase alignment | |
| (h) 1st spindle (basic spindle) rotation speed change | |
| (i) Spindle synchronization cancel | |

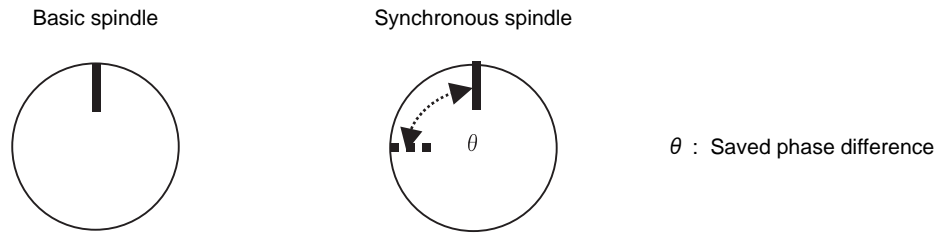
(Note 1) Acceleration/Deceleration at the phase synchronization is the step synchronization method when "#3130 syn_spec/bit1" = "0", the multi-step acceleration/deceleration method when "#3130 syn_spec/bit1" = "1".

Spindle synchronization phase shift amount calculation function

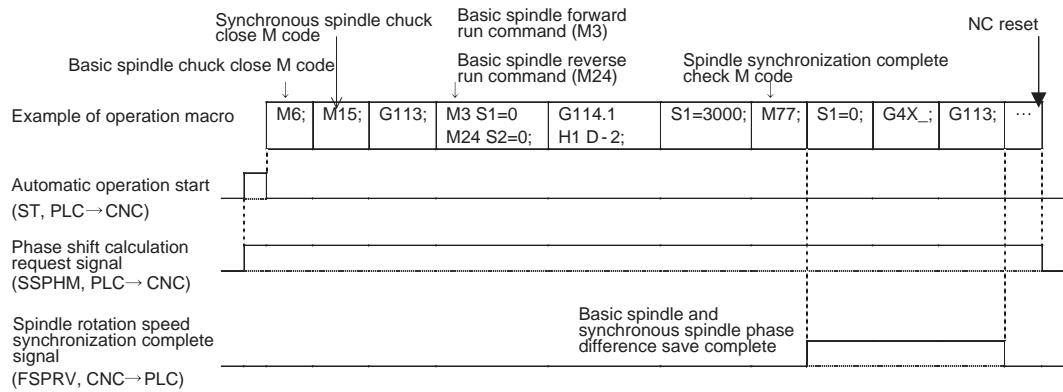
The spindle phase shift amount calculation function obtains and saves the phase difference of the basic spindle and synchronous spindle by turning the PLC signal ON when the phase synchronization command is executed. When the phase is positioned to the automatically saved phase difference before executing the phase synchronization control command, phases can be aligned easier when re-grasping profile materials.

[Saving the basic spindle and synchronous spindle phase difference]

- (1) Set a profile material in the basic spindle.
- (2) Set the profile material in the synchronous spindle.
- (3) Turn the phase shift calculation request signal (SSPHM) ON.
- (4) Input a rotation command, with 0 speed, for the basic spindle and synchronous spindle.
<Example> M3 S1=0 M24 S2 = 0;
- (5) Execute the rotation synchronization signal (with no R address command).
<Example> G114.1 H1 D-2;
- (6) Rotate the basic 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.

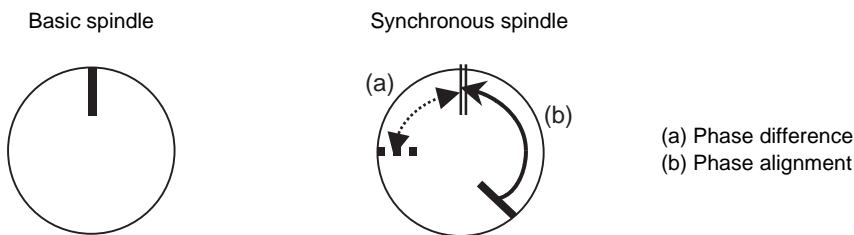


<Example of operation>

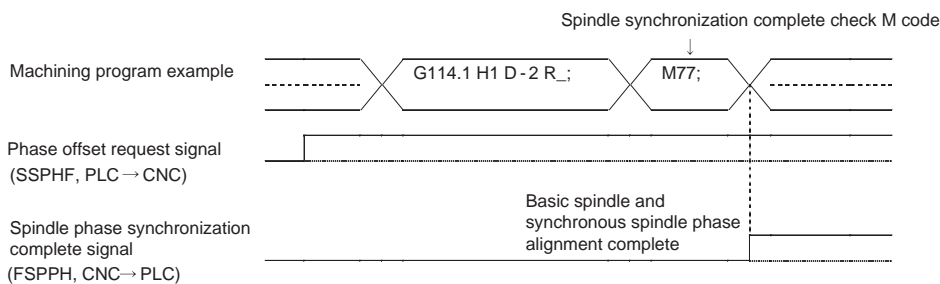


[Automatic phase alignment of basic spindle and synchronous spindle]

- (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 synchronous spindle phase shift amount designation R value is 0 is the same as the reference state (state obtained with phase shift calculation request signal).



<Example of operation>

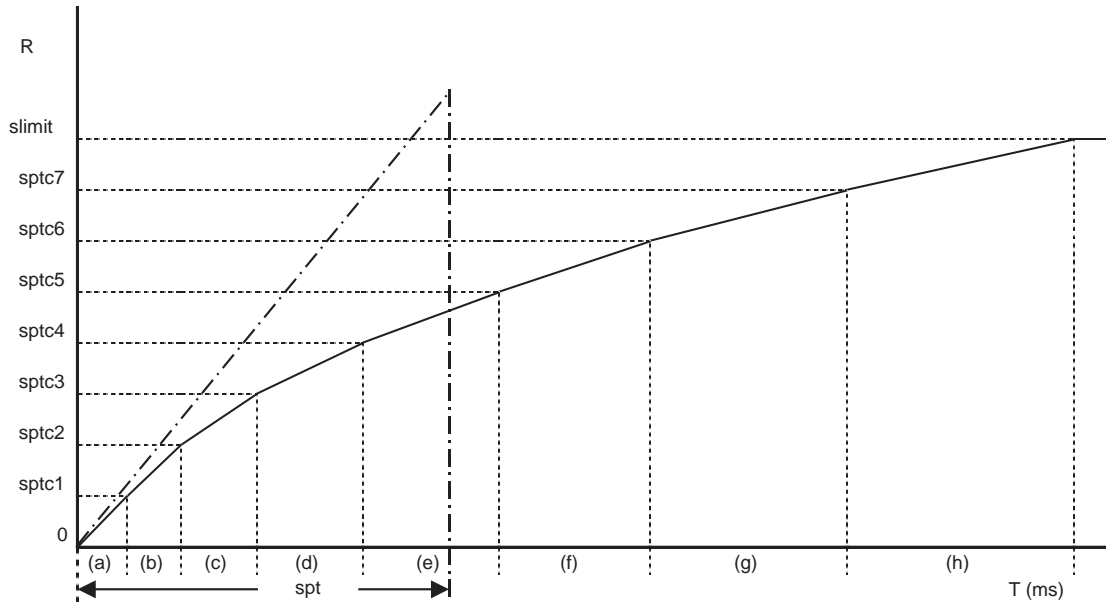


Multi-step acceleration/deceleration

Acceleration/deceleration time constants for up to eight steps can be selected according to the spindle rotation speed for the acceleration/deceleration during spindle synchronization.
The acceleration/deceleration in each step is as follows.

Time required from minimum rotation speed to maximum rotation speed in each step
= [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 (or A command when G114.1 is commanded) * sptc1/slimit
- Time required to reach sptc2 set rotation speed from sptc1 (b)
= spt (or A command when G114.1 is commanded) * spdiv1 * (sptc2 - sptc1)/slimit
- Time required to reach sptc3 set rotation speed from sptc2 (c)
= spt (or A command when G114.1 is commanded) * spdiv2 * (sptc3 - sptc2)/slimit
- Time required to reach sptc4 set rotation speed from sptc3 (d)
= spt (or A command when G114.1 is commanded) * spdiv3 * (sptc4 - sptc3)/slimit
- Time required to reach sptc5 set rotation speed from sptc4 (e)
= spt (or A command when G114.1 is commanded) * spdiv4 * (sptc5 - sptc4)/slimit
- Time required to reach sptc6 set rotation speed from sptc5 (f)
= spt (or A command when G114.1 is commanded) * spdiv5 * (sptc6 - sptc5)/slimit
- Time required to reach sptc7 set rotation speed from sptc6 (g)
= spt (or A command when G114.1 is commanded) * spdiv6 * (sptc7 - sptc6)/slimit
- Time required to reach sptc8 set rotation speed from sptc7 (h)
= spt (or A command when G114.1 is commanded) * spdiv7 * (slimit - sptc7)/slimit



R: Rotation speed (r/min) T: Time

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



Precautions

- (1) The spindle rotating with spindle synchronization will stop when emergency stop is applied.
- (2) The rotation speed clamp during spindle synchronization will follow the smaller clamp value set for the basic spindle or synchronous spindle.
- (3) Orientation of the basic spindle and synchronous 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 synchronous spindle during the spindle synchronization mode. Note that the modal will be updated, so this will be validated when spindle synchronization is canceled.
- (5) The constant surface speed control is invalid for the synchronous spindle during the spindle synchronization mode. Note that the modal will be updated, so this will be validated when spindle synchronization is canceled.
- (6) The rotation speed command (S command) and constant surface speed control for the synchronous spindle will be validated when spindle synchronization is canceled. Thus, the synchronous 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 (sppst) is invalid when using the spindle synchronous phase shift amount calculation function. (It is ignored.)
The spindle Z phase encoder position parameter (sppst) is valid when the phase offset request signal is OFF.
- (9) If the phase synchronization command (command with R address) is issued while the phase shift calculation request signal is ON, an operation error (1106) will occur.
- (10) If the phase shift calculation request signal is ON and the basic spindle or synchronous spindle is rotation while rotation synchronization is commanded, an operation error (1106) will occur.
- (11) If the phase synchronization command R0 (<Ex.> G114.1 H1 D-2 R0) is commanded while the phase offset request signal is ON, the basic spindle and synchronous spindle phases will be aligned to the phase difference of the basic spindle and synchronous spindle saved in the NC memory.
- (12) If a value other than the phase synchronization command R0 (<Ex.> G114.1 H1 D-2 R000) 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 basic spindle and synchronous spindle saved in the NC memory will be used to align the basic spindle and synchronous 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 basic spindle and synchronous spindle saved in the NC is valid only when the phase shift calculation signal is ON and for the combination of the basic spindle selection (H_) and synchronous spindle (D_) commanded with the rotation synchronization command (no R address). For example, if the basic spindle and synchronous spindle phase difference is saved as "G114.1 H1 D-2 ;", the saved phase difference will be valid only when the phase offset request signal is ON and "G114.1 H1 D_2 R*** ;" is commanded. If "G114.1 H2 D-1 R*** ;" is commanded in this case, the phase shift amount will not be calculated correctly.
- (15) The basic spindle and synchronous spindle phase difference saved in the NC is held until the next spindle synchronous phase shift calculation (rotation synchronization command is completed with phase shift calculation request signal ON).
- (16) When the spindle synchronization commands are being issued with the PLC I/F method (#1300 ext36/bit7 OFF), a program error (P610) will occur if the spindle synchronization is commanded with G114.1/ G113.
- (17) Always set the "Chuck close". If the "Chuck close" is not set, an excessive load may be applied on the machine or an alarm may occur.

Cautions on programming

- (1) To enter the rotation synchronization mode while the basic spindle and synchronous spindle are chucking the same workpiece, turn the basic spindle and synchronous spindle rotation commands ON before turning the spindle synchronization mode ON.

\$1 (1st part system)	\$2 (2nd part system)
:	:
M6; 1st spindle chuck close	:
:	M25 S2=0; 2nd spindle stops at S=0
:	:
!2;	!1; Timing synchronization operation between part systems
M5 S1=0; 1st spindle stops at S=0	M15; 2nd spindle chuck close
:	M24; 2nd spindle rotation command ON
M3; 1st spindle rotation command ON	:
!2;	!1; Timing synchronization operation between part systems
:	G114.1 H1 D-2 ; Rotation synchronization mode ON
:	:
S1=1500; Synchronous rotation at S=1500	:
:	:
S1=0; Both spindles stop	:
G113 Synchronization mode cancel	

- (2) To chuck the same workpiece with the basic spindle and synchronous spindle in the phase synchronization mode, align the phases before chucking.

\$1 (1st part system)	\$2 (2nd part system)
:	:
M6; 1st spindle chuck close	:
:	:
M3 S1=1500; 1st spindle rotation command ON	:
:	G114.1 H1 D-2 R0; Phase synchronization mode ON
:	:
:	M24; 2nd spindle rotation command ON
:	:
:	M15; 2nd spindle chuck close (Note 1)
:	:

(Note 1) Close the chuck after confirming that the spindle phase synchronization complete signal (X18AA) has turned ON (phase alignment complete).

CAUTION

1. Do not make the synchronous spindle rotation command OFF with one workpiece chucked by the basic spindle and synchronous spindle during the spindle synchronous mode. Failure to observe this may cause the synchronous spindle stop, and hazardous situation.

10.5.2 Spindle Synchronization Control II



Function and purpose

For spindle synchronization function II, selection of the synchronized spindle and synchronization start, etc., are all designated from the PLC. Refer to the instruction manual issued by the machine tool builder for details.



Detailed description

Basic spindle and synchronous spindle selection

Select the basic spindle and synchronous spindle for synchronous control from the PLC.

Device No.	Signal name	Abbreviation	Explanation
R7016	Basic spindle selection	-	Select a serially connected spindle to be controlled as the basic spindle. (0: 1st spindle) 1: 1st spindle 2: 2nd spindle 3: 3rd spindle 4: 4th spindle 5: 5th spindle 6: 6th spindle (Note1) Spindle synchronization will not take place if a spindle not connected in serial is selected. (Note2) If "0" is designated, the 1st spindle will be controlled as the basic spindle.
R7017	Synchronous spindle selection	-	Select a serially connected spindle to be controlled as the synchronous spindle. (0: 2nd spindle) 1: 1st spindle 2: 2nd spindle 3: 3rd spindle 4: 4th spindle 5: 5th spindle 6: 6th spindle (Note3) Spindle synchronization will not take place if a spindle not connected in serial is selected or if the same spindle as the basic spindle is selected. (Note4) If "0" is designated, the 2nd spindle will be controlled as the synchronous spindle.

Starting spindle synchronization

The spindle synchronization mode is entered by inputting the spindle synchronization signal (SPSY). The synchronous spindle will be controlled in synchronization with the rotation speed commanded for the basic spindle during the spindle synchronization mode.

When the difference of the basic spindle and synchronous spindle rotation speeds reaches the spindle synchronization rotation speed reach level setting value (#3050 sprlv), the spindle rotation speed synchronization complete signal (FSPRV) will be output.

The synchronous spindle's rotation direction is designated with the spindle synchronization rotation direction designation as the same as the basic spindle or the reverse direction.

Device No.	Signal name	Abbreviation	Explanation
Y18B0	Spindle synchronization	SPSY	The spindle synchronization mode is entered when this signal turns ON.
X18A8	In spindle synchronization	SPSYN1	This notifies that the mode is the spindle synchronization.
X18A9	Spindle rotation speed synchronization complete	FSPRV	This is turned ON when the difference of the basic spindle and synchronous spindle rotation speeds reaches the spindle rotation speed reach level setting value during the spindle synchronization mode. This signal is turned OFF when the spindle synchronization mode is canceled, or when an error exceeding the spindle rotation speed reach level setting value occurs during the spindle synchronization control mode.
Y18B2	Spindle synchronization rotation direction designation	-	Designate the basic spindle and synchronous spindle rotation directions for spindle synchronization. 0: The synchronous spindle rotates in the same direction of the basic spindle. 1: The synchronous spindle rotates in the reverse direction of the basic spindle.

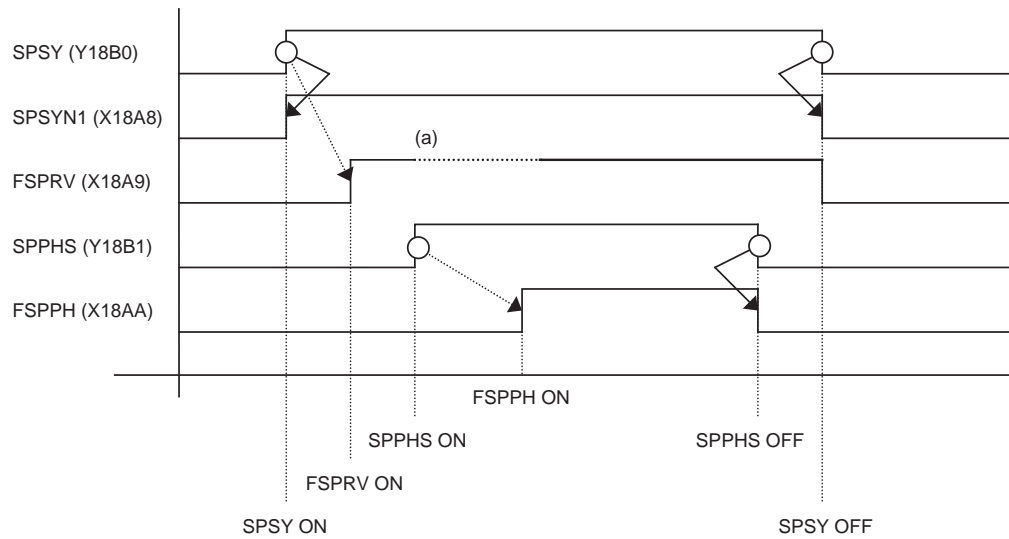
Spindle phase alignment

Spindle phase synchronization starts when the spindle phase synchronization signal (SPPHS) is input during the spindle synchronization mode.

The spindle phase synchronization complete signal is output when the spindle synchronization phase reach level setting value (#3051 spplv) is reached.

The synchronous spindle's phase shift amount can also be designated from the PLC.

Device No.	Signal name	Abbreviation	Explanation
Y18B1	Spindle phase synchronization control	SPPHS	Spindle phase synchronization starts when this signal is turned ON during the spindle synchronization mode. (Note1) If this signal is turned ON in a mode other than the spindle synchronization mode, it will be ignored.
X18AA	Spindle phase synchronization control	FSPPH	This signal is output when the spindle synchronization phase reach level is reached after starting spindle phase synchronization.
R7018	Phase shift amount setting	-	Designate the synchronous spindle's phase shift amount. Unit: 360° /4096



(a) Turns OFF temporarily to change the rotation speed during phase synchronization.

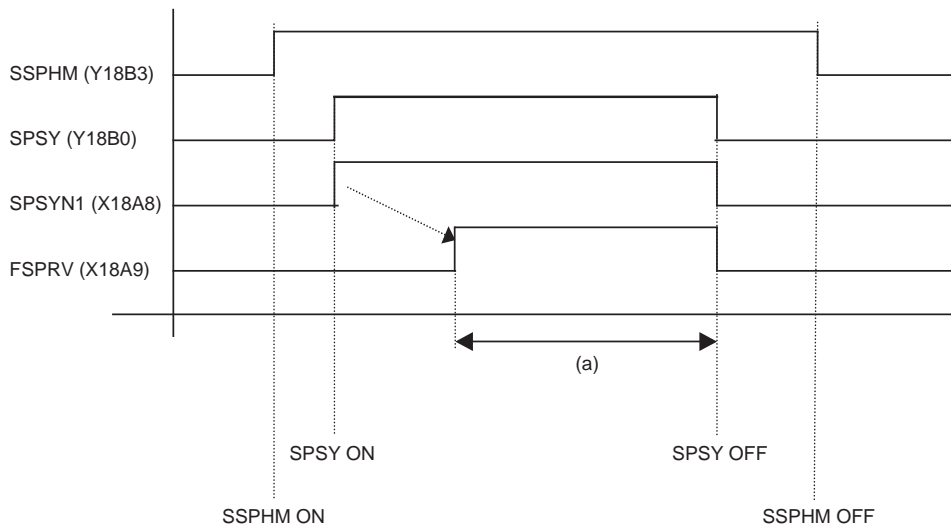
- SPSY : Spindle phase synchronization
- SPSYN1 : In spindle synchronization
- FSPRV : Spindle rotation synchronization complete
- SPPHS : Spindle phase synchronization control
- FSPPH : Spindle phase synchronization complete

Calculating the spindle synchronization phase shift amount and requesting phase offset

The spindle phase shift amount calculation function obtains and saves the phase difference of the basic spindle and synchronous spindle by turning the PLC signal ON during spindle synchronization. When calculating the spindle phase shift, the synchronous spindle can be rotated with the handle, so the relation of the phases between the spindles can also be adjusted visually.

If the spindle phase synchronization control signal is input while the phase offset request signal (SSPHF) is ON, the phases will be aligned using the position shifted by the saved phase shift amount as a reference. This makes aligning of the phases easier when grasping the material that the shape of one end differs from the other end.

Device No.	Signal name	Abbreviation	Explanation
Y18B3	Phase shift calculation request	SSPHM	If spindle synchronization is carried out while this signal is ON, the phase difference of the basic spindle and synchronous spindle will be obtained and saved.
Y18B4	Phase offset request	SSPHF	If spindle phase synchronization is carried out while this signal is ON, the phase will be aligned using the position shifted by the saved phase shift amount as a basic position.
R6516	Phase difference output	-	The delay of the synchronous spindle in respect to the basic spindle is output. Unit:360° /4096 (Note 1) If either the basic spindle or synchronous spindle has not passed through the Z phase, etc., and the phase cannot be calculated, -1 will be output. (Note 2) This data is output only while calculating the phase shift or during spindle phase synchronization.
R6518	Phase offset data	-	The phase difference saved with phase shift calculation is output. Unit:360° /4096 (Note 3) This data is output only during spindle synchronization.



(a) The phase difference in this interval is saved. (The synchronous spindle can be controlled with the handle.)

- SSPHM : Phase shift calculation request
- SPSY : Spindle synchronization
- SPSYN1 : In spindle synchronization signal
- FSPRV : Spindle synchronization complete signal

- (Note 1) The phase cannot be aligned while calculating the phase shift.
- (Note 2) The synchronous spindle cannot be rotated with the handle when the manual operation mode is set to the handle mode.



Precautions and restrictions

- (1) When carrying out spindle synchronization, a rotation command must be issued to both the basic spindle and synchronous spindle. The synchronous spindle's rotation direction will follow the basic spindle rotation direction and spindle synchronization rotation direction designation regardless of whether a forward or reverse run command is issued.
- (2) The spindle synchronization control mode will be entered even if the spindle synchronization control signal is turned ON while the spindle rotation speed command is ON. However, synchronous control will not actually take place. Synchronous control will start after the rotation speed command has been issued to the basic spindle, and then the spindle synchronization complete signal will be output.
- (3) The spindle rotating with spindle synchronization will stop when emergency stop is applied.
- (4) An operation error will occur if the spindle synchronization signal is turned ON while the basic spindle and synchronous spindle designations are illegal.
- (5) The rotation speed clamp during spindle synchronization will follow the smaller clamp value set for the basic spindle or synchronous spindle.
- (6) Orientation of the basic spindle and synchronous spindle is not possible during the spindle synchronization. To carry out orientation, turn the spindle synchronization signal OFF first.
- (7) The rotation speed command is invalid for the synchronous spindle during the spindle synchronization. The commanded rotation speed will be validated after spindle synchronization is canceled.
- (8) The constant surface speed control is invalid for the synchronous spindle during the spindle synchronization.
- (9) If the phase offset request signal is turned ON before the phase shift is calculated and then spindle phase synchronization is executed, the shift amount will not be calculated.
- (10) The spindle Z-phase encoder position parameters are invalid when phase offset is carried out.
- (11) If spindle phase synchronization is started while the phase shift calculation request signal is ON, the error "M01 operation error 1106" will occur.
- (12) Turn the phase shift calculation request signal ON when the basic spindle and synchronous spindle are both stopped. If the phase shift calculation request signal is ON while either of the spindles is rotating, the error "M01 operation error 1106" will occur.
- (13) The phase shift amount saved in the NC is held until the next phase shift is calculated. (This value is saved even when the power is turned OFF.)
- (14) Always set the "Chuck close". If the "Chuck close" is not set, an excessive load may be applied on the machine or an alarm may occur.

10.5.3 Precautions for Using Spindle Synchronization Control



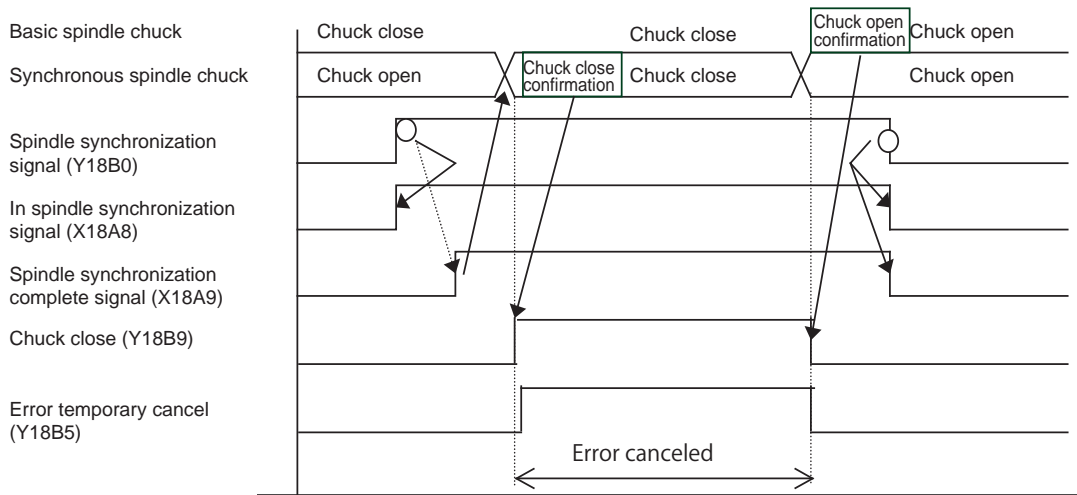
Precautions

Some PLC signals must be set when spindle synchronization control I or II is used. If these signals are not set, an excessive load or an alarm may occur. Refer to the instruction manual issued by the machine tool builder for details. In this section, each function and the signal are explained.

Chuck close signal

The synchronous spindle side carries out droop compensation while the chuck is opened, and aligns itself with the basic spindle. However, when the chuck is closed, the droop compensation is added, and the synchronization error with the base increases. Droop compensation is prevented with the chuck close signal and the position where the chuck is grasped is maintained with position compensation.

Device No.	Signal name	Abbreviation	Explanation
Y18B9	Chuck close	-	This turns ON when the chuck is closed. When this signal turns ON, the compensation between the basic spindle and synchronous spindle will change from droop compensation to position compensation.
X18AC	Chuck close confirmation	-	This turns ON when the chuck close signal is received during the spindle synchronization mode.



(Note 1) Use the error temporary cancel only when there is still an error between the spindle and synchronization with the chuck close signal.

Error temporary cancel function

When spindle synchronization is carried out while grasping the workpiece with the basic spindle and rotating, if the chuck is closed to grasp the workpiece with the synchronous spindle, the speed will fluctuate due to external factors and an error will occur. If spindle synchronization is continued without compensating this error, the workpiece will twist.

This torsion can be prevented by temporarily canceling this error.

Device No.	Signal name	Abbreviation	Explanation
Y18B5	Error temporary cancel	SPDRP0	The error is canceled when this signal is ON.

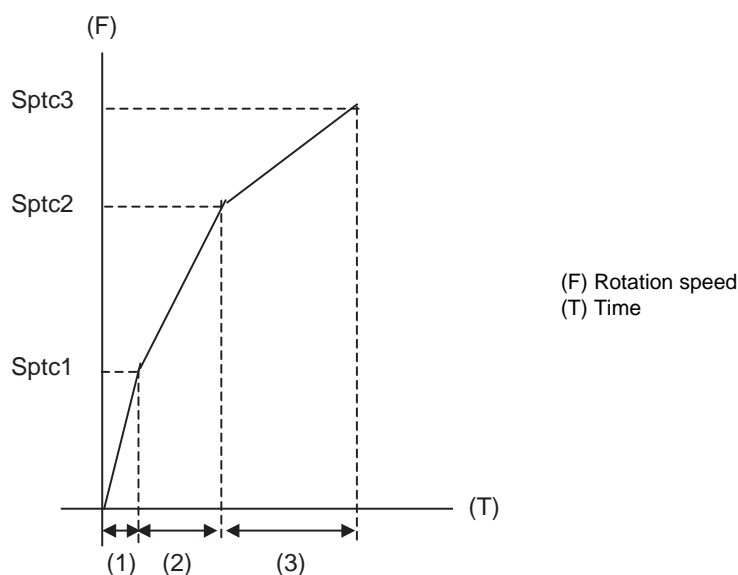
Phase error monitor

The phase error can be monitored during spindle phase synchronization.

Device No.	Signal name	Abbreviation	Explanation
R6519	Phase error monitor	-	The phase error during spindle phase synchronization control is output as a pulse unit.
R6520	Phase error monitor (lower limit value)	-	The lower limit value of the phase error during spindle phase synchronization control is output as a pulse unit.
R6521	Phase error monitor (upper limit value)	-	The upper limit value of the phase error during spindle phase synchronization control is output as a pulse unit.

Multi-step acceleration/deceleration

Up to eight steps of acceleration/deceleration time constants for spindle synchronization can be selected according to the spindle rotation speed.



- (1) Time required from stopped state to sptc1 setting rotation speed
 $spt * (sptc1/\text{maximum rotation speed})$
- (2) Time required from sptc1 to sptc2 setting rotation speed
 $spt * ((sptc2-sptc1)/\text{maximum rotation speed}) * spdiv1$
- (3) Time required from sptc2 to sptc3 setting rotation speed
 $spt * ((sptc3-sptc2)/\text{maximum rotation speed}) * spdiv2$

10.6 Tool Spindle Synchronization IA (Spindle-Spindle, Polygon) ; G114.2



Function and purpose

In a machine having a rotary tool controlled with a serial connection and having a spindle controlled with a serial connection as the workpiece axis, polygon machining can be carried out by controlling the workpiece axis rotation in synchronization with the rotation of the rotary tool axis.

The serial connection control of spindle and rotary tool axis can be carried out with MDS*-SP or MDS*-SPJ3.



Command format

G114.2 H__ D__ E__ L__ R__ ; ... Tool Spindle Synchronization IA (Spindle-Spindle, Polygon) ON

H	Rotary tool axis selection (Basic spindle)
D	Workpiece axis selection (Synchronous spindle)
E	Rotary tool axis rotation ratio designation
L	Workpiece axis rotation ratio designation
R	Synchronous spindle phase shift amount

G113 ; ... Tool Spindle Synchronization IA (Spindle-Spindle, Polygon) OFF

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) cancels the synchronous state of rotating two spindles by the spindle synchronization command.

Address	Meaning of address	Command range (unit)	Remarks
H	Rotary tool axis Of the two spindles, the spindle No. of the select rotation axis.	1 to number of spindles 1: 1st spindle 2: 2nd spindle 3: 3rd spindle 4: 4th spindle	- 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 a spindle that is not serially connected is selected, a program error (P700) will occur.
D	Workpiece axis Of the two spindles, select the spindle No. of the workpiece axis.	1 to number of spindles or -1 to - (number of spindles) 1,-1: 1st spindle 2,-2: 2nd spindle 3,-3: 3rd spindle 4,-4: 4th spindle	- 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 a spindle that is not serially connected is selected, a program error (P700) will occur.
E	Rotary tool axis rotation ratio Set the rotation ratio (Number of rotary tool gear teeth) of the rotary tool axis.	1 to 10	- 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 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	Synchronized spindle phase shift amount 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 is applied in the clockwise direction of the spindle. - If there is no R command, phase alignment will not be carried out.



Detailed description

Rotation axis and rotation direction

The rotary tool axis and workpiece axis rotation speed and rotation direction during tool spindle synchronization IA (Polygon) 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 (Polygon) mode 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.

Operation for spindle-spindle polygon

The workpiece axis will be controlled in the following manner.

- (1) When the spindle synchronization IA (polygon) mode is commanded, if neither a forward run command nor a reverse run command is input for the workpiece axis, the workpiece axis will not start rotating even if the polygon axis is rotating, and instead will wait for synchronization. If a forward run command or reverse run command is input for the workpiece axis in this state, the workpiece axis will start rotation.
- (2) If spindle stop is commanded (both forward run command and reverse run command are turned OFF) in respect to the workpiece axis during the spindle synchronization IA (polygon) mode, the workpiece axis rotation will stop even when the rotary tool axis is rotating.
- (3) The rotation command (S command) and constant surface speed control are invalid in respect to the workpiece axis during the tool spindle synchronization IA (polygon) mode. Note that the modal will be updated, so these will be effective after the spindle synchronization is canceled.
- (4) If a rotary tool axis rotation speed that exceeds the workpiece axis maximum rotation speed is commanded, the rotary tool axis rotation speed will be clamped so that the workpiece axis rotation speed does not exceed the workpiece axis maximum rotation speed.

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 sptc 1 set rotation speed from stopped state (a)

$$= \text{spt} * \text{sptc1} / \text{slimit}$$

Time required to reach sptc 2 set rotation speed from sptc 1 (b)

$$= \text{spt} * \text{spdiv1} * (\text{sptc2} - \text{sptc1}) / \text{slimit}$$

Time required to reach sptc 3 set rotation speed from sptc 2 (c)

$$= \text{spt} * \text{spdiv2} * (\text{sptc3} - \text{sptc2}) / \text{slimit}$$

Time required to reach sptc 4 set rotation speed from sptc 3 (d)

$$= \text{spt} * \text{spdiv3} * (\text{sptc4} - \text{sptc3}) / \text{slimit}$$

Time required to reach sptc 5 set rotation speed from sptc 4 (e)

$$= \text{spt} * \text{spdiv4} * (\text{sptc5} - \text{sptc4}) / \text{slimit}$$

Time required to reach sptc 6 set rotation speed from sptc 5 (f)

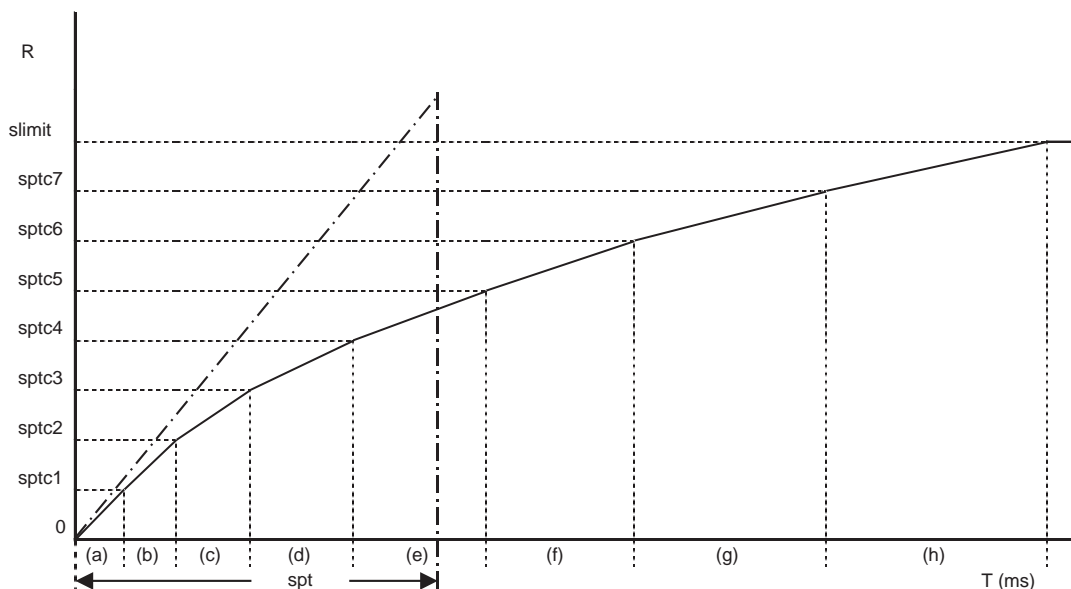
$$= \text{spt} * \text{spdiv5} * (\text{sptc6} - \text{sptc5}) / \text{slimit}$$

Time required to reach sptc 7 set rotation speed from sptc 6 (g)

$$= \text{spt} * \text{spdiv6} * (\text{sptc7} - \text{sptc6}) / \text{slimit}$$

Time required to reach sptc 8 set rotation speed from sptc 7 (h)

$$= \text{spt} * \text{spdiv7} * (\text{slimit} - \text{sptc7}) / \text{slimit}$$



R : Rotation speed (r/min) T : Time

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 synchronous spindle rotating at an arbitrary rotation speed will accelerate or decelerate to the rotation speed following the basic spindle and synchronous spindle rotation ratio command, and the spindle synchronization state will be entered. After that, the phase will be aligned to match the rotation phase commanded with the R address.
- (2) The spindle synchronization phase shift amount is commanded as the shift amount from the synchronous spindle's (workpiece axis) reference position (one rotation signal). There is not the shift amount in respect to the basic spindle (rotary tool axis).
- (3) If the basic spindle's (rotary tool axis) commanded rotation speed is changed during the spindle synchronization state, acceleration/deceleration will be carried out following the spindle acceleration/ deceleration set in the parameters while maintaining the synchronization state, and will reach the commanded rotation speed.

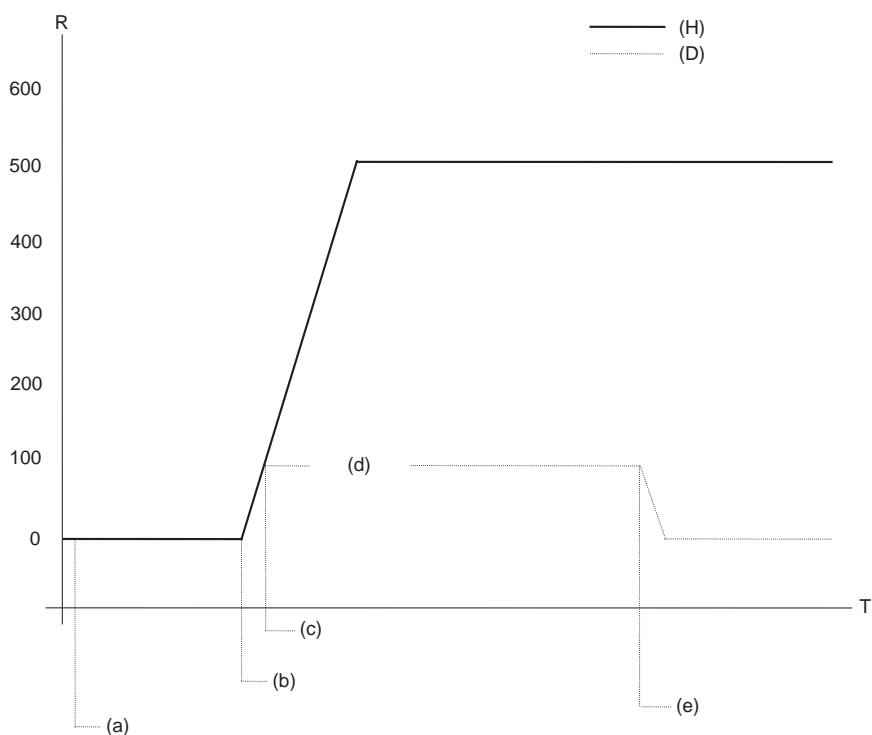
(4) The following type of operation will take place.

```

M03 S1=0;          Carries out forward run (speed command) 1st spindle (synchronous spindle)
:
Tx00;             Rotary tool selection
M83 S4=500;       Carries out forward run (speed command) 4th spindle (basic spindle)
:
G114.2 H4 D1 E1 L5 Rxx; Carries out forward run 1st spindle (synchronous spindle) and synchronizes with 4th
                    spindle (basic spindle).
                    Shifts synchronous spindle phase by amount of R command value.
:
:
MG113;           Cancels tool-spindle synchronization IA

```

<Operation>



R: Rotation speed (r/min)

T: Time

(H) Basic spindle

(D) Synchronous spindle

(a) 1st spindle (synchronous spindle) forward run

(b) 4th spindle (basic spindle) forward run

(c) 1st spindle (synchronous spindle) forward run synchronization

(d) Phase alignment

(e) Cancels spindle synchronization



Program example

```

:
:
M03 S1=0 ;           1st spindle forward run
Txx00 ;             Rotary tool selection
M83 S4=500 ;        4th spindle forward run
G00 X40.Z-5 ;
:
:
G114.2 H4 D1 E1 L10 R0 ; Tool Spindle Synchronization IA (Spindle-Spindle, Polygon mode) ON
                        Rotary tool axis: 4th spindle, Workpiece: 1st axis
                        Number of rotary tool gear teeth: 1,
                        Rotation ratio: Number of workpiece corners: 10
                        Synchronous spindle phase shift amount : 0°
                        S1 starts rotating by forward run in synchronization with S4.
                        The phase is aligned with shift amount 0° .
                        The S1 rotation speed is 50 r/min (S2:S1 = 10:1)

G99 ;               Synchronous feed mode selection

G00 X18. ;          1st cut in
G01 Z20.F0.1 ;      Z axis feedrate is 0.1 mm per workpiece axis rotation.
G00 X40. ;
  Z-5. ;
:
:
G00 X14. ;          Final cut in
G01 Z20.F0.1 ;      Z axis feedrate is 0.1 mm per workpiece axis rotation.
G00 X40. ;
  Z-5. ;

G113 ;              Spindle synchronization cancel

M85 ;               4th spindle stop
M05 ;               1st spindle stop
:

```



Precautions

- (1) If S is commanded in the same block as G114.2, the synchronization speed will be created at the previous S command until the S command ends, so the spindle speed may fluctuate momentarily. Thus, do not command S in the same block if possible.
- (2) Always command G114.2 in an independent block.
- (3) The tool spindle synchronization I (spindle-spindle, polygon) mode cannot be commanded during the spindle synchronization mode commanded with G114.*. the operation error "M01 1005" will occur.
- (4) If spindle-spindle polygon machining is commanded while the phase shift calculation request signal SSPHM is ON, the operation error "M01 1106" will occur.
- (5) Spindle-spindle polygon machining cannot be executed by designating a spindle being used for synchronous tapping with the G114.2 command. The operation error "M01 1007" 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. The 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 the 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.

$$\text{Spindle rotation speed} / \text{encoder rotation speed} = n \text{ ("n" is an integer of 1 or more)}$$
 If this relation is not established, the encoder's reference position will not stay at a constant position of 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.

$$(\text{Rotary tool axis spindle rotation speed} * \text{Number of rotary tool gear teeth}) / \text{encoder rotation speed} = n \text{ ("n" is an integer of 1 or more)}$$
- (2) During phase alignment control, phase alignment is carried out following each spindle encoder's reference position, so 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.
 So 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.

Precautions for programming

- (1) The axis address (X, Z, C) command in the same block as G114.2 will be ignored.
ex. G114.2 X_ ;
X_ : ignored.
- (2) If a modal is commanded in the same block as G114.2, the modal will be updated.
ex. G114.2 G01 ;
The group 01 modal is set to G01.
- (3) If a miscellaneous command (M, S, T) is commanded in the same block as G114.2, the miscellaneous command will be executed simultaneously with the change to the rotary tool machining mode.
ex. G114.2 M03 ;
M03 is executed simultaneously with G114.2.
- (4) If there is a group 00 G code command in the same block as G114.2, the G code commanded last in the block will have the priority.
ex. G114.2 G4 P30. ;
G4 P30. is executed.

10.7 Tool Spindle Synchronization IB (Spindle-Spindle, Polygon) ; G51.2 (only 6 and 7 in G code list)



Function and purpose

In a machine having a serial connection-controlled workpiece axis, and furthermore having a serial connection-controlled spindle as a rotary tool axis, spindle-spindle polygon machining can be carried out by controlling the rotation of the rotary tool axis in synchronization with the workpiece axis.

A parameter (#1501) is used to switch from spindle-spindle polygon machining (IB) to spindle-NC axis polygon machining (IC) or vice versa.

#1501 polyax = 0 : Spindle-spindle polygon machining (IB)
Other than 0 : NC axis-spindle polygon machining (IC)

The workpiece axis and rotary tool axis can be serially connected and controlled with MDS-*-SP or MDS-*-SPJ3.

This function is valid when the G code system is 6 or 7.



Command format

G51.2 H__D__P__Q__R__ ; ... Tool Spindle synchronization IB (Spindle-Spindle, Polygon mode) ON

H	Workpiece axis selection (Basic spindle)
D	Rotary tool axis selection (Synchronous spindle)
P	Rotary tool axis rotation ratio designation
Q	Workpiece axis rotation ratio designation
R	Synchronous spindle phase shift amount

G50.2 ; ... Tool Spindle synchronization IB (Spindle-Spindle, Polygon mode) OFF

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

Address	Meaning of address	Command range (unit)	Remarks
H	Workpiece axis selection Command the spindle No. of the workpiece axis.	1 to number of spindles	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 a spindle not serially connected is commanded, a program error (P33) will occur. If omitted, the spindle No. set in the parameters will be applied.
D	Rotary tool axis Command the spindle No. of the rotary tool axis.	1 to number of spindles	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 a spindle not serially connected is commanded, a program error (P33) will occur. If omitted, the spindle No. set in the parameters will be applied.
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 the workpiece axis.
R	Synchronous spindle phase shift amount designation 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. If there is no R command, the phase will be handled as R0. (only when #1239 set11/bit4 = 0)



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.

$$S_w = S_h * \frac{Q}{P}$$

S_w : Rotary tool axis rotation speed (r/min)
 S_h : 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 teeth)

- (3) The rotary tool axis rotation direction is determined by the sign of the rotary tool axis selection Q commanded with G51.2.
 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 spindle-spindle polygon machining is commanded, the relation of the workpiece axis and rotary tool axis rotation is held until spindle-spindle polygon machining cancel (G50.2) 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 spindle-spindle polygon machining 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 spindle-spindle polygon machining 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 speed command (S command) and constant surface speed control are invalid for the rotary tool axis during the spindle-spindle polygon machining mode. Note that the modal is updated, so these will be validated when the spindle-spindle polygon machining is canceled.
- (4) If a workpiece axis rotation speed that exceeds the rotary tool axis's maximum rotation speed is commanded, the workpiece axis rotation speed will be clamped so that the rotary tool axis rotation does not exceed the rotary tool axis's maximum rotation speed.

Acceleration/deceleration control

- (1) Acceleration/deceleration of the workpiece axis will be carried out linearly according to the spindle synchronization acceleration/deceleration time constant (spt) of the spindle selected as the workpiece axis.
- (2) By setting the spindle synchronization multi-speed acceleration/deceleration time constant changeover speed (spdct1 to 7) and the scale for the time constant changeover speed (spddiv1 to 7), 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

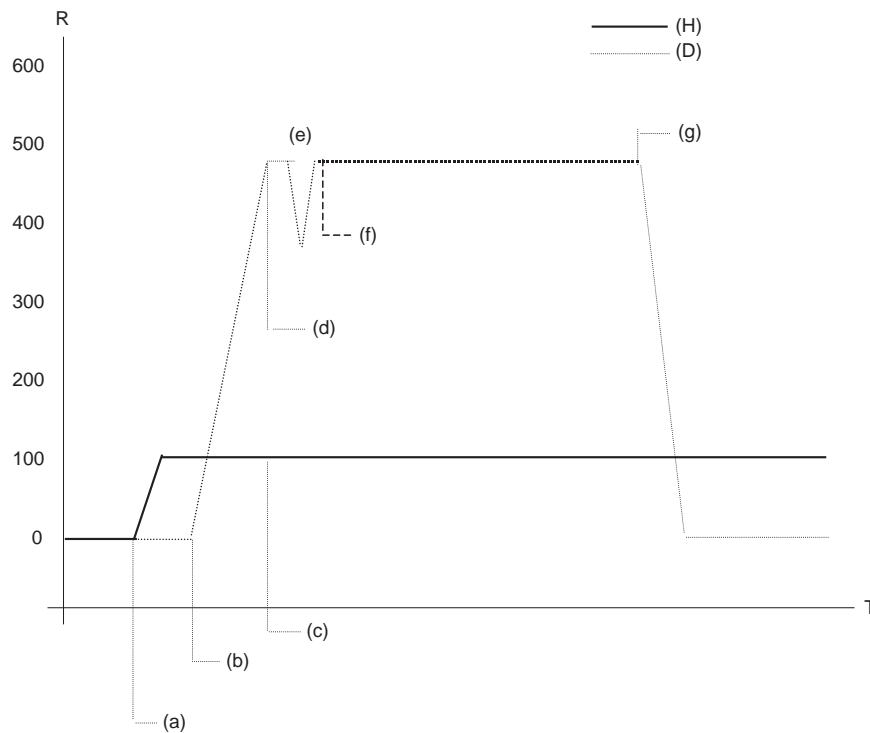
- (1) If the spindle-spindle polygon command (R = 0 with no R command) is commanded with G51.2, the workpiece axis spindle rotating at an arbitrary rotation speed will accelerate/decelerate to the rotation speed following the rotation ratio command of the workpiece axis spindle and rotary tool axis spindle. 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.
- (3) The following type of operation will take place.

<Program example 1> With phase error (#1239 set11/bit4 = 0)

```

:
Txx00;           Rotary tool selection
M03 S100;        Carries out forward run (speed command) 1st spindle (workpiece axis)
:
G51.2 H1 D3 P1 Q5 Rxx; Spindle-spindle polygon command, synchronize 3rd spindle (rotary tool axis) to 1st
:                  spindle (workpiece axis spindle) with forward run. Shifts synchronous spindle's phase
:                  by R command value.
:
G50.2;           Cancel spindle-spindle polygon mode.
    
```

<Operation>



- R: Rotation speed (r/min) T: Time (H) Workpiece axis (D) Rotary tool axis
- (a) 1st spindle (workpiece axis) forward run
 - (b) G51.2 command spindle-spindle polygon machining start 3rd spindle (rotary tool axis) forward run
 - (c) 1st spindle (workpiece axis) forward run synchronization
 - (d) 3rd spindle (rotary tool axis) forward run synchronization
 - (e) Phase alignment (f) Synchronization complete
 - (g) Spindle synchronization cancel

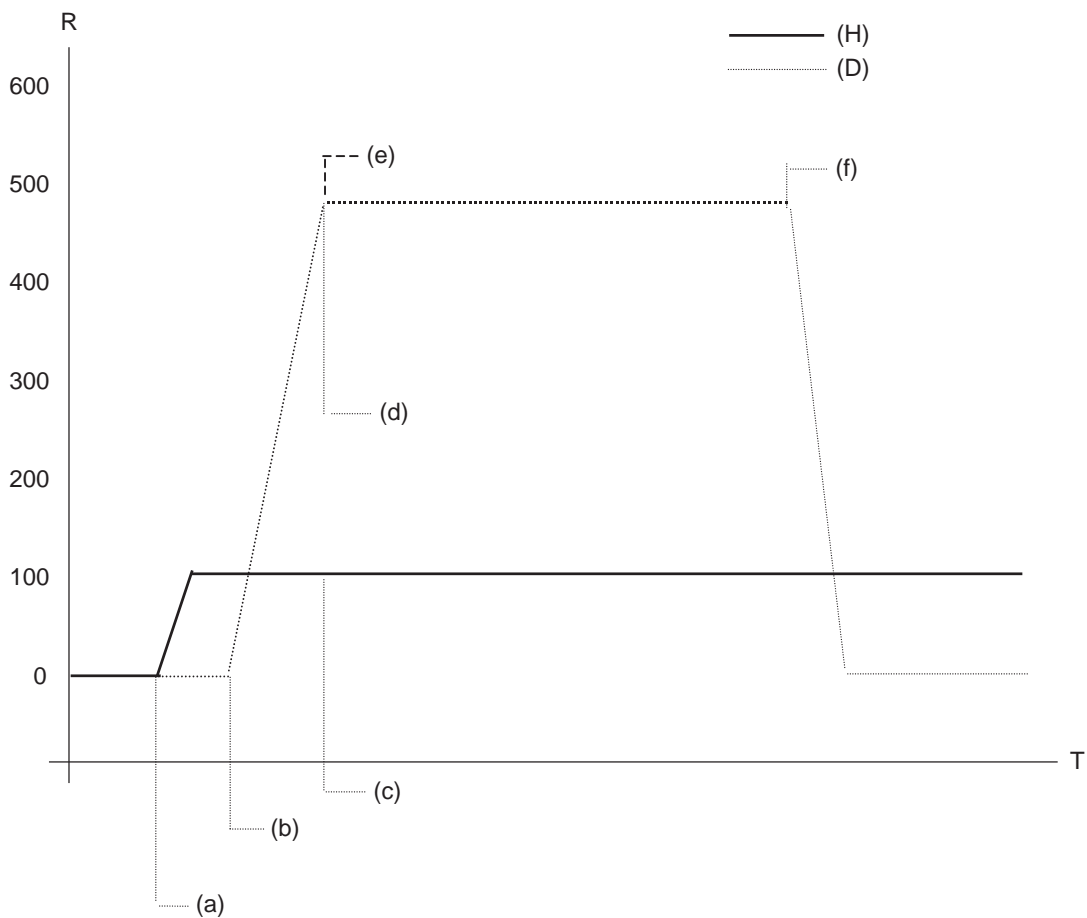
10.7 Tool Spindle Synchronization IB (Spindle-Spindle, Polygon) ; G51.2 (only 6 and 7 in G code list)

<Program example 2> No phase error (#1239 set11/bit4 = 1)

```

:
Txx00;           Rotary tool selection
M03 S100;        Carries out forward run (speed command) 1st spindle (workpiece axis)
:
G51.2 H1 D3 P1 Q5 ; Spindle-spindle polygon command, synchronize 3rd spindle (rotary tool axis) to 1st
:                spindle (workpiece axis spindle) with forward run.
:
G50.2;          Cancel spindle-spindle polygon mode.
    
```

<Operation>



- R: Rotation speed (r/min) T: Time (H) Workpiece axis (D) Rotary tool axis
- (a) 1st spindle (workpiece axis) forward run
- (b) G51.2 command spindle-spindle polygon machining start 3rd spindle (rotary tool axis) forward run
- (c) 1st spindle (workpiece axis) forward run synchronization
- (d) 3rd spindle (rotary tool axis) forward run synchronization
- (e) Synchronization complete (f) Spindle synchronization cancel



Program example

```

:
:
Txx00 ;           Rotary tool selection
M03 S500 ;       1st spindle forward run
G00 X40.Z-5 ;

G51.2 H1 D3 P1 Q3 R0 ;   Spindle-spindle polygon machining mode ON
                        Select 1st spindle as workpiece axis and 3rd spindle as rotary tool axis.
                        Designate rotation ratio as one workpiece corner and three rotary tool teeth.
                        Designate rotary tool axis spindle phase shift amount as 0° .
                        Start synchronous rotation of S3 to S1 with forward run.
                        The phase is aligned with shift amount 0° .
                        S3 rotation speed is 1500 r/min (S1:S3 = 1:3)
G99 ;           Synchronous feed mode selection

G00 X18. ;      If synchronization is not completed, wait to start cutting feed.
G01 Z20.F0.1 ; 1st cut in
G00 X40. ;
  Z-5. ;
:
:
:
G00 X14. ;      Final cut in
G01 Z20.F0.1 ;
G00 X40. ;
  Z-5. ;

G50.2 ;        Spindle-spindle polygon machining mode cancel
                3rd spindle stop

M05 ;         1st spindle stop
:
:

```



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 relation is not established, the encoder's reference position will not stay at a constant position of 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) Always command G51.2 and G50.2 in an independent block.
- (2) The R command can be omitted when entering the spindle-spindle polygon 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 while in the spindle-spindle polygon mode, command G51.2 again. In this case, R can be commanded independently. 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 part systems cannot be used simultaneously. The part system commanded first will be valid, and the operation error 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 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. Program error (P33) will occur if the spindle is not serially connected.
- (7) After G51.2 is commanded, the cutting feed block will not start until synchronization is established. (Operation will stop with the operation error 1033.)

10.8 Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon) ; G51.2 (only 6 and 7 in G code list)



Function and purpose

This function carries out polygon machining by controlling the workpiece (spindle) and tool axis (NC servo axis) to rotate in synchronization at the commanded ratio.

A parameter is used to switch from spindle-spindle polygon machining (IB) to spindle-NC axis polygon machining (IC) or vice versa.



Command format

G51.2 P__ Q__ ; ... Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon machining mode) ON

P,Q	Spindle and rotary tool axis rotation ratio (P__:Q:__) P : Spindle Q : Rotary tool axis Command range : Integer value between 1 and 9, -1 and -9 Rotation direction : Designate with a sign ("+" Forward rotation, "-" Reverse rotation)
-----	--

G50.2 ; ... Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon machining mode) OFF

In addition to the G50.2 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)



Detailed description

Explanation of operation

S1000 ;	The spindle rotation speed (workpiece rotation speed) is commanded.
G51.2 P1 Q2 ;	The polygon machining mode is entered with the G51.2 command.
: (Cutting into workpiece)	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 is designated with 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 "#3052 spplr (Spindle motor spindle relative polarity)".

P command sign	#3052 spplr	Rotation direction
(+)	0000	CW
(+)	0001	CCW
(-)	0000	CCW
(-)	0001	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



Program example

N10 G00 X100. Z20. ;	Positioning
N20 S1000 ;	Spindle (workpiece) rotation speed command
N30 G51.2 P1 Q2 ;	Spindle/tool axis rotation start (Spindle rotation speed 1000[r/min], tool axis rotation speed 2000[r/min])
N40 G01 X80. F10. ;	X axis cut in
N50 G04 X2. ;	Dwell
N60 G00 X100. ;	X axis retract
N70 G50.2 ;	Spindle/tool axis rotation stop

(Note) Always command G51.2 and G50.2 in independent blocks.



Precautions

- (1) The "spindle synchronization (polygon)" specifications must be provided to use this function. If G51.2 or G50.2 is commanded without the specifications, the program error (P39) will occur.
- (2) Always command G51.2 and G50.2 in independent blocks.
 - If the G51.2 (G50.2) command and group 0 G code are commanded in the same block, the G code commanded last in the block will have the priority.
 - If the G51.2 (G50.2) command and G code other than a group 0 code are commanded in the same block, the program error (P33) will occur.
- (3) While in the polygon machining mode, a movement command cannot be issued in the machining program for a servo 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 servo 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 during the polygon machining mode.
- (8) If the feedrate for the rotary tool axis exceeds the rapid traverse rate (axis specifications parameters "#2201 rapid") during the polygon machining mode, 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) The following functions cannot be used simultaneously with polygon machining.
 - Synchronous tapping
 - Thread cutting
- (11) 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.
- (12) 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.
- (13) By setting the spindle basic specifications parameters "#3106 zrn_typ/bit4" to "0", the polygon machining will start after the spindle returns to the zero point.

10.9 Tool Spindle Synchronization II (Hobbing) ; G114.3

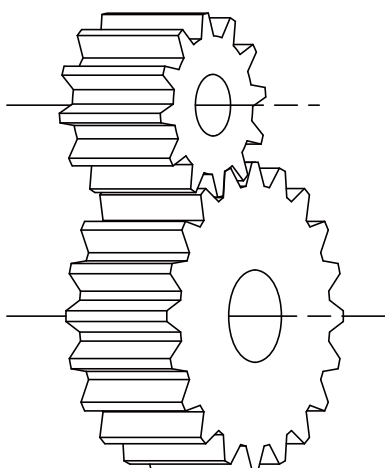


Function and purpose

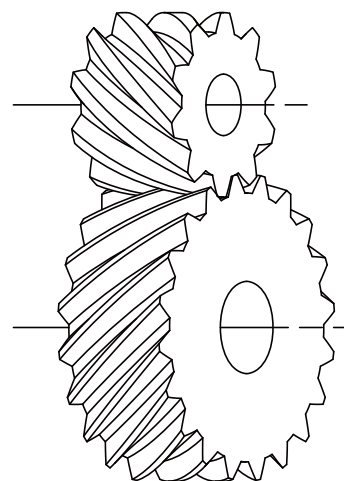
This function is to cut the gear with hob (hob cutter).

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.

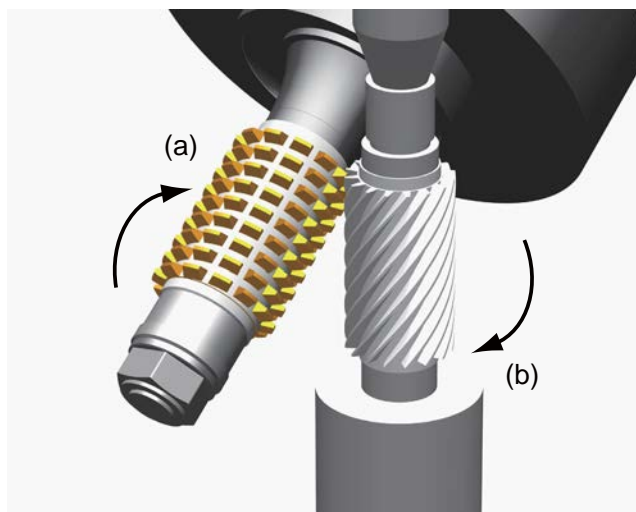


Spur gears



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.



Command format

Tool spindle synchronization II (hobbing) ON (for spur gear)

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 gear)

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

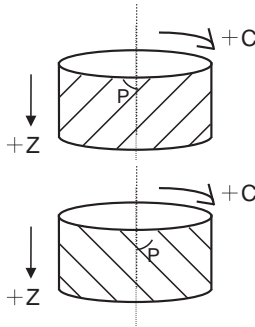
G113 ;

Tool spindle synchronization II (G114.3 for spur gear) in which two axes are synchronously rotated at different speeds is entered by designating the hob axis and workpiece axis, and by designating the rotation ratio (hob threads and number of gear teeth) for the two designated axes.

Tool spindle synchronization II (G114.3 for helical gears) is entered by additionally designating the gear torsion angle and module or diametral pitch.

Tool spindle synchronization II (hobbing) OFF cancels the synchronous state of the hob axis and workpiece axis rotating in synchronization with the tool spindle synchronous II (hobbing) command.

Address	Meaning of address	Command range (unit)	Remarks
H	Hob axis selection Set the spindle No. of the hob axis.	1 to 6	- If there is no command, a program error (P33) will occur. - If an analog connected spindle is selected, a program error (P33) will occur. - If disconnected spindle No. is designated, a program error (P35) will occur.
D	Workpiece axis selection Set the rotation No. of the workpiece axis.	-9 to -1, 1 to 9 ± 1 to 6 Axis No. (in part system) ± 9 : C axis	- If there is no command, a program error (P33) will occur. - 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 rotation axis, a program error (P33) will occur. - If C axis is selected when there is no C axis, a program error (P33) will occur.

Address	Meaning of address	Command range (unit)	Remarks
E	Hob axis rotation ratio designation Set 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). (Note 2)
L	Workpiece axis rotation ratio designation Set 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.	0 to 359999 (0 to 359.999°) Decimal point input possible (Note 3)	- The commanded shift amount will be applied in the workpiece axis counter's positive direction. - If there is no R command, phase alignment will not be carried out.
P	Gear torsion angle designation Command the torsion angle for the helical gear.	-89000 to 89000 (-89.000 to 89.000) Decimal point input possible (Note 4)	- 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 
Q	Module designation Command the normal module for helical gear. When inch input, command the diametral pitch.	Metric input Module designation 100 to 25000 0.1 to 25. (0.1 to 25 mm) Inch input Diametral pitch designation 1000 to 250000 0.1 to 25. (0.1 to 25inch ⁻¹) Decimal point input possible (Note 5)	If there is no Q command for helical gear (when P is designated), a program error (P33) will occur. For spur gear (when P is not designated, or P0 is commanded), the Q command will be ignored.

(Note 1) If a value exceeding the command range is commanded, a program error (P35) will occur.

(Note 2) When address E = 0 is commanded, the workpiece axis will not rotate.

Do not use this except for special cutting (cutting of only part of the gears, etc.).

(Note 3) The range which can be set depends on the input setting unit (parameter "#1003 iunit").

(Example) When the input setting unit is 0.000001°, the range is 0 to 359.999999°.

(Note 4) 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.

(Note 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 as follows:

For metric system: 100000 to 250000000

For inch system: 1000000 to 2500000000



Detailed description

Rotation ratio change during tool spindle synchronization II (hobbing) mode

The rotation ratio and the number of hob threads can be changed stopping neither the hob axis nor the workpiece axis during tool spindle synchronization control II (hobbing) mode.

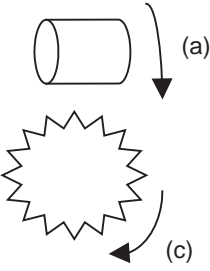
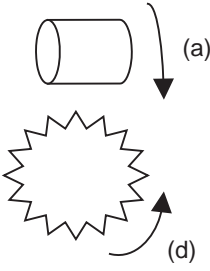
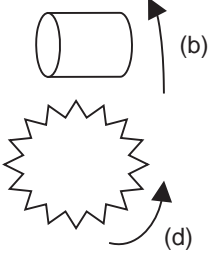
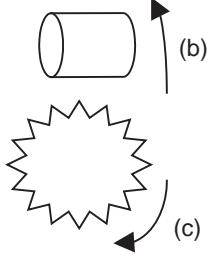
G114.3 E__L__P__Q__ ; Rotation ratio change	
E	Hob axis rotation ratio designation
L	Workpiece axis rotation ratio designation
P	Gear torsion angle designation
Q	Module or diametral pitch designation

- (1) The G114.3 command during the tool spindle synchronization control II (hobbing) mode can omit each address. 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 last command is used excluding L.)
 G114.3 L50;
- (2) If the followings are issued, a program error (P33) will occur.
 - (a) When R command (workpiece axis phase shift amount) is issued.
 - (b) When the hob axis No. and workpiece axis No. are changed.
 - (c) When other than 0 is commanded by E command in E=0 state, or 0 is commanded by E command in E ≠ 0 state.
- (3) The workpiece axis rotation speed may be changed by rotation ratio. Acceleration/deceleration time constant follows spindle multi-step acceleration/deceleration time constant based on spindle rotation region of hob axis calculated in consideration for rotation ratio at workpiece axis rotation speed.
- (4) Spindle rotation speed synchronization completion signal 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.

Rotation speed and rotation direction

The rotation speed and rotation direction of hob axis and workpiece axis during tool spindle synchronization control 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
(c) Workpiece axis: + direction

- (b) Hob axis: Reverse rotation
(d) Workpiece axis: - direction

- (2) The workpiece axis rotation speed is determined by the hob threads commanded with G114.3 and the number of gear teeth.

$$S_w = S_h * \frac{E}{L}$$

S_w : Workpiece axis rotation speed (r/min)

S_h : Hob axis rotation speed (r/min)

E : Hob axis rotation ratio (hob threads)

L : Workpiece axis rotation ratio (number of gear teeth)

- (3) The workpiece axis rotation direction is determined by the sign of the address D commanded with G114.3.
In other words, when the D sign is "+", the workpiece axis will rotate in the same direction as the hob axis, and when "-", the workpiece axis will rotate in the direction opposite the hob axis.
- (4) After tool spindle synchronization control II (hobbing) is commanded the relation of the hob axis and workpiece axis rotation is held in all operation modes of automatic and manual modes until spindle synchronization cancel (G113) 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 G114.3 is commanded during hob axis rotation, workpiece axis accelerates up to the speed in synchronization with hob axis according to the hobbing workpiece axis time constant (#2195 hob_tL) using the constant inclination 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 control II (hobbing) mode.
- (3) If a manual movement command is issued to the workpiece axis during the tool spindle synchronization control 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 workpiece axis will be output.

Note that, if the movement command is issued in the manual reference position return mode, an operation error (0005) occurs.

An automatic movement command can be issued to the workpiece axis during the tool spindle synchronization control 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 control 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 a servo OFF signal is input for the workpiece axis during the tool spindle synchronization control II (hobbing) mode, the tool spindle synchronization control 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 control II (hobbing) mode.
 - (a) When workpiece axis is a rotary-type rotation axis
The axis will rotate in the 0.000 to 359.999 range in the normal manner.
 - (b) When workpiece axis is a linear-type rotation axis (all coordinate values linear type)
The axis will rotate in the 360° range including the machine coordinate position and workpiece coordinate position when hobbing starts.
 - (c) When workpiece axis is a linear-type rotation axis (workpiece coordinate values linear type)
The axis will rotate in the 360° range including the workpiece coordinate position when hobbing starts.

(Example)

Coordinate value 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 zero point return, a program error (P430) will occur.

Acceleration/deceleration control

- (1) The hob 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.

Phase alignment control (Machine configuration that the phase alignment is possible)

To carry out phase alignment during hobbing, the spindle detector 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 (Operation when the zero point of hob axis is not established)

When the zero point of hob axis is not established by the hob axis rotation after turning the power ON or the spindle gear changeover, carry out phase alignment by following operations.

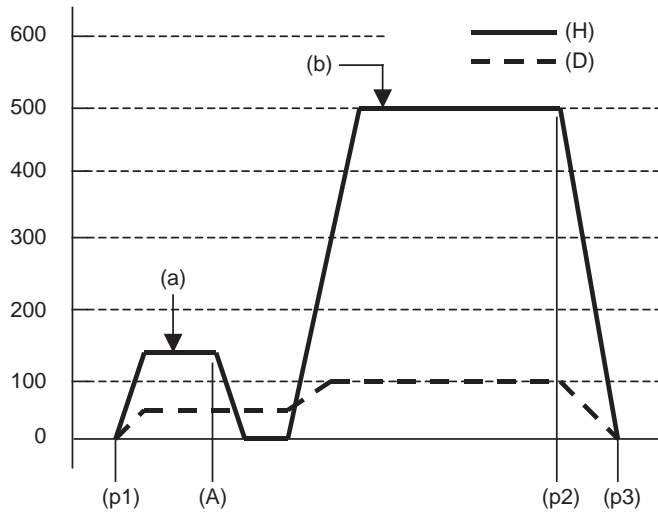
(The zero point of hob axis is established within the range of (p1) - (A) in the figure.)

- (1) When tool spindle synchronization control II (with R command) is commanded with G114.3, the rotation axis commanded as the workpiece axis will enter the tool spindle synchronization II (hobbing) control state.
- (2) The hob axis will start rotation at the Z phase detection speed (parameter "#3109 zdetspd") set in the parameters with the first S command issued for the hob axis after the hobbing control state is entered. At this time, the workpiece axis will reach the rotation speed following the rotation ratio command for the hob axis and workpiece axis.
If this command rotation speed is 0 (r/min), the hob axis will not start rotating, and instead will wait for the next S command.
- (3) The hob axis and workpiece axis phases will be aligned in this state.
- (4) 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.

(5) The operation example is as follows.

Txx00;	Select the rotary tool.	
M83 S4=0;	Forward run the fourth spindle (hob axis). (Rotation speed is 0)	
:		
G114.3 H4 D9 E1 L5 R0;	Hobbing mode (phase alignment at phase difference 0) ON	
S4=500;	Rotate the fourth spindle (hob axis) at 500r/min.	(p1)
:		
M85;	Stop fourth spindle.	(p2)
G113 ;	Cancel the hobbing mode.	(p3)

<Operation>



(H) Hob axis
 (D) Workpiece axis
 (a) Z phase detection speed (r/min)
 (b) Command rotation speed (r/min)

Phase alignment control (Operation when the zero point of hob axis is established)

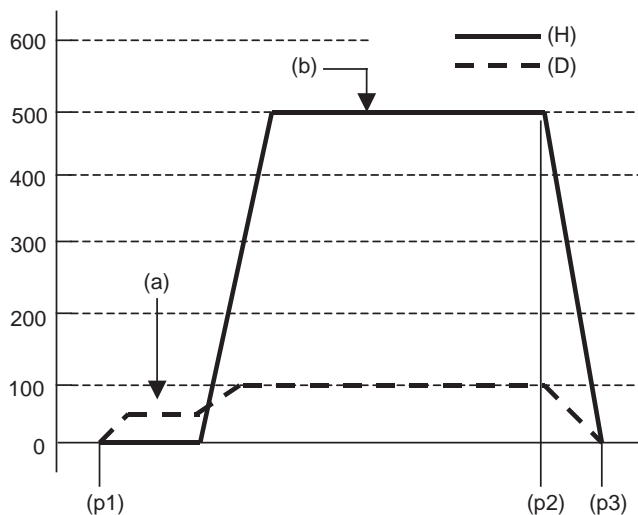
When the zero point of hob axis has already been established, the zero point of hob axis establishment is omitted.

Thus, the process finishes fast compared with the case that the zero point of hob axis is not established.

- (1) When tool spindle synchronization control II (with R command) is commanded with G114.3, the rotation axis commanded as the workpiece axis will enter the tool 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) When the state of the hob axis stopping and workpiece axis rotating, the phase alignment is carried out.
- (4) 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.
- (5) The operation example is as follows.

Txx00;	Select the rotary tool.	
M83 S4=0;	Forward run the fourth spindle (hob axis). (Rotation speed is 0)	
:		
G114.3 H4 D9 E1 L5 R0;	Hobbing mode (phase alignment at phase difference 0) ON	
S4=500;	Rotate a fourth spindle (hob axis) at 500r/min.	(p1)
:		
M85;	Stop the fourth spindle.	(p2)
G113 ;	Cancel the hobbing mode.	(p3)

<Operation>



- (H) Hob axis
 (D) Workpiece axis
 (a) Workpiece axis rotation speed based on Z phase detection speed (r/min)
 (b) Command rotation speed (r/min)

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

OFF: No compensation

ON: Hob axis delay (advance) is compensated with workpiece axis

#3134 spttc 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 with the workpiece axis command in the machining program.

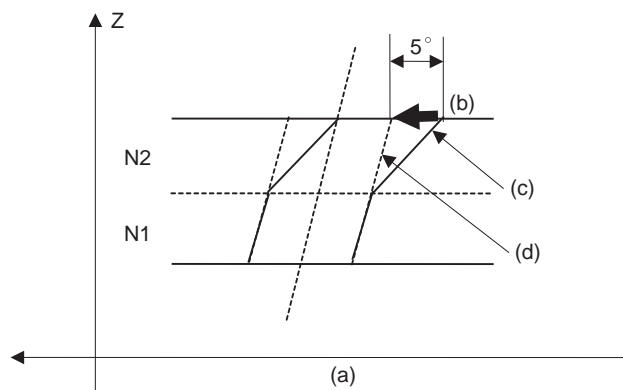
(a) Command the workpiece axis compensation amount as an incremental value.

(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 absolute value for the workpiece axis during the tool spindle synchronization control II (hobbing) mode, a program error (P32) will occur.

<Program example>

```
G114.3 H1 D9 E1 L10 P30. Q100. ;
S1 = 100;
G94 ;
N1 G01 Z20. F10;
N2 G91 G01 Z20. C5.;           <- Workpiece axis compensation amount
:
:
:
```



- (a) Workpiece axis (C axis) rotation direction
- (b) Workpiece axis compensation amount
- (c) Shape caused by insufficient rigidity
- (d) Command shape

Feedforward control during tool spindle synchronization II (hob machining) 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 (#3135 sfd_g).
- (2) The workpiece axis feedforward control is controlled according to hob axis feedforward gain (parameter "#3135 sfd_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

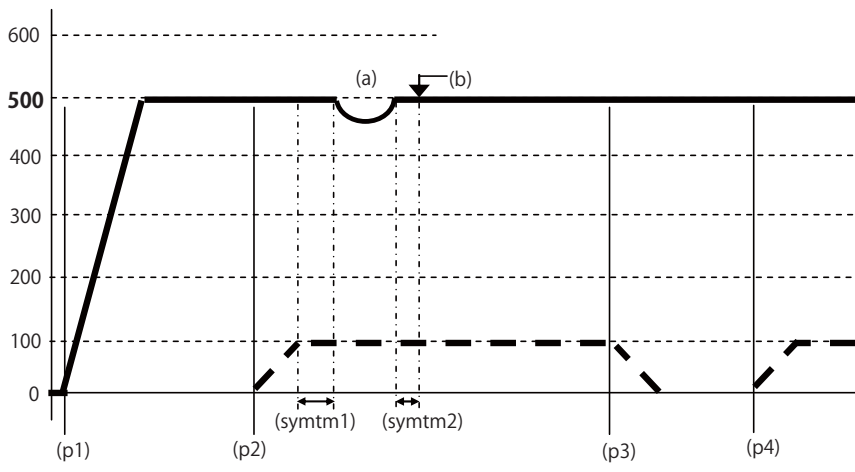
G114.3 or G113 can be commanded during hob axis rotation.

- (1) When G114.3 is commanded 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 (#2195 hob_tL) and uses the constant inclination acceleration/deceleration control. If the setting of the hobbing workpiece axis time constant is outside the setting range, set the maximum value in the range.
- (2) After the workpiece axis finishes acceleration, phase alignment is carried out between hob and workpiece axes if G114.3 contains R command.
- (3) Synchronization is established after phase alignment is completed.
- (4) When G113 is commanded during hob axis rotation, workpiece axis decelerates to a stop. This deceleration follows the hobbing workpiece axis time constant (#2195 hob_tL), and uses the constant inclination acceleration/deceleration. If the setting of hobbing workpiece axis time constant is outside the setting range, set the maximum value in the range.
- (5) Below is the operation example:

```

Txx00;           Select a rotary axis
M** S4=500;      Forward run the 4th spindle (hob spindle) (at 500r/min).           (p1)
:
G114.3 H4 D9 E1 L5 R0;  Hobbing mode ON (phase alignment with the phase difference of zero) (p2)
:
G113;           Hobbing mode OFF (p3)
:
G114.3 H4 D9 E1 L5 R0  Hobbing mode ON (phase alignment with the phase difference of zero) (p4)
:
    
```

<Operation>



————— Hob axis
 - - - - - Workpiece axis

(a) Aligning the phase
 Follows the setting of acceleration/deceleration during phase alignment.
 When #3130 sync_spec/bit1=0, the type is step alignment type.
 When #3130 sync_spec/bit1=1, the type is multi-step type

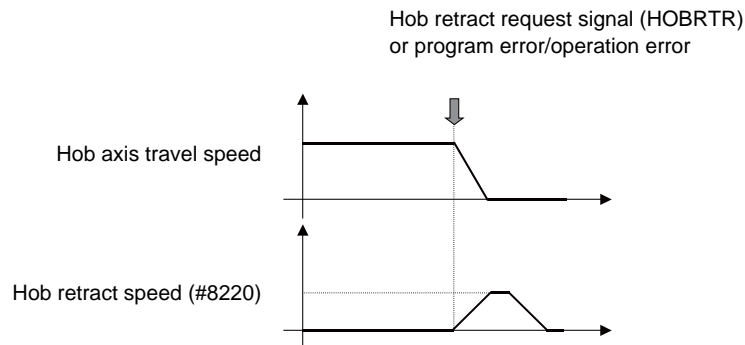
(b) Spindle speed sync completed
 (symtm1) Phase sync start confirmation time
 (symtm2) Phase sync end confirmation time

Retract during Hobbing Mode

Operating retract during hobbing mode prevents a destruction of the work when hobbing is interrupted.

When Hob retract request signal is input during hobbing mode, the control can retract the axis designated by parameter.

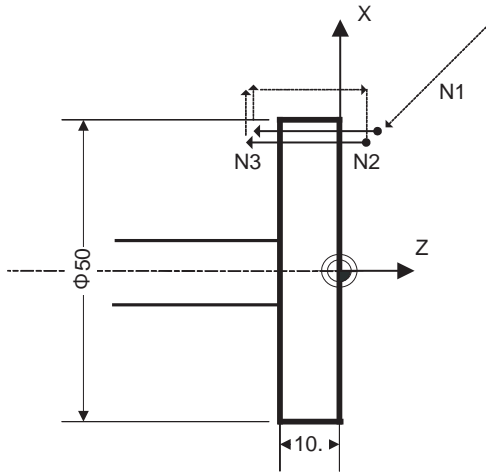
Retract operation can also be carried out when a program error or operation error occurs.



- (1) Retract operation is carried out when Hob retract request signal (YCDE) is turned ON during hobbing mode.
- (2) When a program operation error occurs by setting "#19406(Hob retract ON at alarm" during hobbing mode, retract operation is carried out.
Note however that even when #19406 (Hob retract ON at alarm) is ON, retract is not carried out at an alarm regardless of the setting of the parameter if Hob alarm retract inhibit signal (YCDF) is ON.
- (3) Retract is carried out in automatic operation mode.
But when it is in automatic mode, retract can also be performed when not in automatic operation.
- (4) After retract operation is completed, automatic operation stops.
- (5) The movement amount in retract operation is determined by either the parameter "#8219 Hob retract amount 1" or "#8220 Hob retract amount 2", which is specified by the Hob retract amount selection signal (YB20).
- (6) The parameters "#8219 Hob retract amount 1" or "#8220 Hob retract amount 2" are handled as radius values.
- (7) Retract speed is set for each axis using the parameter "#8221 Hob retract speed".
- (8) In hob retract signal (XCAE) is ON during retract operation.
- (9) When retract operation is completed, Hob retract complete signal (XCAF) turns ON.
- (10) When retract is performed, acceleration and deceleration are carried out based on the travel command's acceleration/deceleration mode. However when the parameter "#19407 Hob retract acceleration/ deceleration OFF" is set to 1, step-wise acceleration/deceleration is carried out.
- (11) Retract is not carried out for axes that are in movement.
- (12) If mirror image is set for the retract axis, mirror image will be reflected on the retract operation. Therefore, the retract is carried out in a direction opposite to the setting.



Program example



(1) Spur gear machining without phase alignment

```

:
:
Txx00;           Rotary tool selection
M** S4=0;       S4 spindle stop (Rotation command forward
                rotation ON)
N1  G00 X48. Z5.;
:
:
G114.3 H4 D9 E1 L10:  Tool spindle synchronization II (hobbing) mode ON
                    Hob axis: S4 spindle, workpiece axis: C axis
                    Hob threads: 1, rotation ratio: 10 teeth
S4=500;         C axis starts forward run synchronized with S4.
                The C axis rotation speed is 50r/min (S4 : C = 10 : 1).

M**;           Spindle speed synchronization completion
                confirmation

G94;           Asynchronous feed mode selection

N2  G01 Z-15. F10;   First cut
     G00 X54.;
     Z5.;
     X46.;
N3  G01 Z-15. F10;   Second cut
     G00 X54.;

S4=0;         S4 spindle stop
G113;        Tool spindle synchronization II (hobbing) mode OFF
:
    
```

(2) Helical gear machining with phase alignment

	:	
	:	
	Txx00;	Rotary tool selection
	M** S4=0;	S4 spindle stop (Rotation command forward rotation ON)
N1	G00 X48. Z5.;	
	:	
	:	
	G114.3 H4 D9 E3 L10 P30. Q2000 R0.;	Tool spindle synchronization II (hobbing) mode ON
		Hob axis: S4 spindle, workpiece axis: C axis
		Hob threads: 3, rotation ratio: 10 teeth
		torsion angle : 30° , module : 2mm
		Phase alignment at phase difference 0
	S4=500;	C axis starts forward run synchronized with S4. The C axis rotation speed is 150r/min (S4 : C = 10 : 3).
	M**;	Spindle speed synchronization completion confirmation
	G94;	Asynchronous feed mode selection
N2	G01 Z-15. F10;	First cut
	G00 X54.;	
	Z5.;	
	X46.;	
N3	G01 Z-15. F10;	Second cut
	G00 X54.;	
	S4=0;	S4 spindle stop
	G113;	Tool spindle synchronization II (hobbing) mode OFF
	:	

(3) When hobbing several times successively with hob axis rotated
 Spur gear machining with phase alignment

Txx00;	Rotary tool selection
M** S4=0;	S4 spindle stop (Rotation command forward rotation ON)
G00 X48.Z5.;	
:	
:	
G114.3 H4 D9 E3 L10 R0;	Tool spindle synchronization II (hobbing) mode ON Hob axis:S4 spindle, workpiece axis: C axis, Hob threads:3, 10 teethe Phase alignment at phase difference 0
S4=500;	C axis starts forward run synchronized with S4. The C axis rotation speed is 150r/min (S4 : C = 10 : 3).
M**;	Spindle speed synchronization completion confirmation
G94;	Asynchronous feed mode selection
G01 Z-15.F10;	First cut
G00 X54.;	
Z5.;	
X46.;	
:	
G113;	Tool spindle synchronization II (hobbing) mode OFF
:	
G114.3 H4 D9 E3 L10 R0;	Tool spindle synchronization II (hobbing) mode ON
M**;	Spindle speed synchronization completion confirmation
G01 Z-15.F10;	Nth cut
G00 X54.;	
Z5.;	
X44.;	
:	
G113;	Tool spindle synchronization II (hobbing) mode OFF
:	
G114.3 H4 D9 E3 L10 R0;	Tool spindle synchronization II (hobbing) mode ON
M**;	Spindle speed synchronization completion confirmation
G01 Z20. F10;	Final cut
G00 X40.;	
Z-5.;	
:	
G113;	Tool spindle synchronization II (hobbing) mode OFF
:	

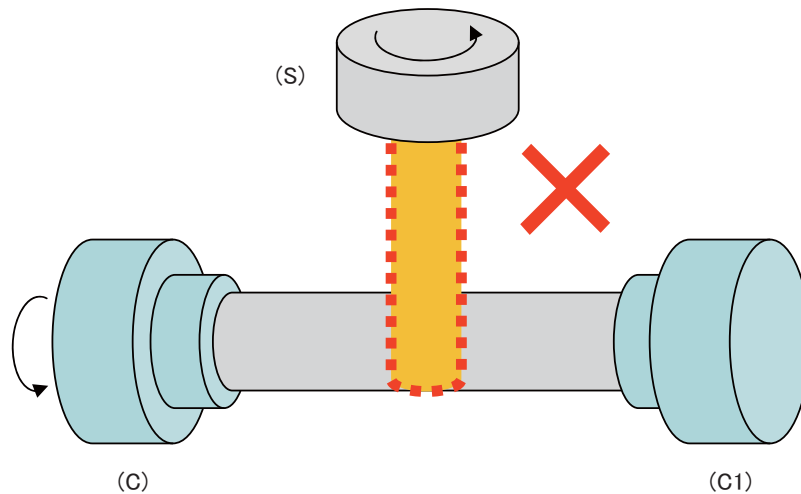


Precautions and restrictions

- (1) The tool spindle synchronization control II (hobbing) mode cannot be commanded during the spindle synchronization control I (spindle rotation synchronization) mode or the tool spindle synchronization control I (polygon machining) mode. Only one of the three types of spindle synchronization control can be commanded at once.
- (2) Even if the reset signal is input, the tool spindle synchronization control II (hobbing) maintains synchronization.
However, the synchronization is canceled at emergency stop.
- (3) When cutting helical gear, correct cutting feed will not be possible in the synchronous feed mode, so always cut in the asynchronous feed mode.
- (4) 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.
- (5) The linear-type rotation axis for the absolute position system cannot be used as the hobbing workpiece axis. If used, an error "Z70 Absolute position lost 0002" will occur after the power is turned OFF/ON.
- (6) If hobbing control is carried out using the linear-type rotation 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.
- (7) If address E is set to 0, phase alignment will not be carried out. Address R will be ignored even if commanded.
- (8) When "Hob axis delay (advance) allowable angle" (parameter "#3133 spherr") is "0", "Hob axis delay excess" (X18B3) is not output.
- (9) During acceleration/deceleration of hob axis, "Hob axis delay angle" (R6516) and "Maximum hob axis delay angle" (R6517) are not updated.
- (10) Issue the commands for the workpiece axis from the machining program as a G0 incremental value or G1 incremental value. When an absolute value is commanded, program error (P32) will occur.
- (11) 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 (servo axis): #2203 SV003, #2204 SV004, #2257 SV057
 - Workpiece axis (spindle/C axis) #13002 SP002, #13035 SP035/BITC
- (12) If door interlock 1 or door interlock 2 is ON during the tool spindle synchronization control II (hobbing) mode, the tool spindle synchronization control II (hobbing) will be canceled because synchronization cannot be maintained.
- (13) When the mixed synchronization control is commanded in the part system commanded hobbing, "M01 operation error 1035" will occur. The hobbing which uses the axis mixed synchronization controlling is possible.
- (14) Issue G114.3 command (Hobbing mode ON) with workpiece axis stopped. When commanding G114.3 (Hobbing mode ON) during workpiece axis rotation, hobbing mode will be ON after the workpiece axis stop is confirmed.
- (15) Retract amount is set by parameter for each axis. If all the axes' retract amount are zero, retract will not carry out the operation.
- (16) Retract operation is not performed while Hob retract complete signal is ON.
- (17) If you turn OFF the Hob retract request signal while retract operation is carried out using this signal, retract operation continues.
- (18) If you turn ON the Hob alarm retract inhibit signal while retract operation is carried out due to a program or operation error, retract operation continues.
- (19) When reset or emergency stop is input during retract operation, retract axis stops moving. Retract is not restarted even when you cancel emergency stop.
- (20) If retract axis detect OT limit, all the axes except for hob and workpiece axes will stop.
- (21) When the parameter "#19407 Hob retract acceleration deceleration OFF" is set to "1", and retract speed is relatively fast, a servo alarm (excessive error etc.) may occur.
- (22) Machine lock is enabled for retract axis.

- (23) Automatic interlock is enabled for retract axis. When not in automatic operation, automatic interlock is enabled for retract axis.
- (24) Cutting feed override or rapid traverse override is not reflected on retract axis.
- (25) External deceleration is enabled for retract axis.
- (26) Dry-run is disabled for retract axis.
- (27) When performing hob machining after retract, the tool is separated from the workpiece by amount of the retract, thus hobbing cannot be correctly performed.
- (28) Retract can be performed for an axis that is under synchronization control. Retraction for a master axis causes the slave axis to move.
- (29) Retract can be performed for an axis that is under inclined axis control. In accordance with the inclined axis' movement, the basic axis moves by the compensation amount.
- (30) When performing retract operation in automatic operation, retract operation can be interrupted by inputting Automatic operation stop signal. But when performing retract in a mode other than automatic operation, retract will not be interrupted by Automatic operation stop signal input.
- (31) If retract operation is interrupted by stopping automatic operation or by switching the operation mode (automatic to manual), retract will not resumed even when you activate automatically after the interruption.
- (32) Pre-interpolation acceleration/deceleration is disabled for a hob retract axis. Post-interpolation acceleration/deceleration is applied instead.
- (33) Do not issue a hob machining command (tool spindle synchronization 2) when workpiece axis (C) and rear workpiece axis (C1) are in synchronization control. When hob machining is commanded during synchronization control, rear workpiece axis (C1) and workpiece axis (C) will not operate in synchronization, and this may twist the workpiece.

[Hob machining model: Workpiece axes in synchronization between C and C1]



(S) : Spindle axis (hob axis)
 (C) : Workpiece axis
 (C1) : Rear workpiece axis

10.10 Multiple-spindle Control



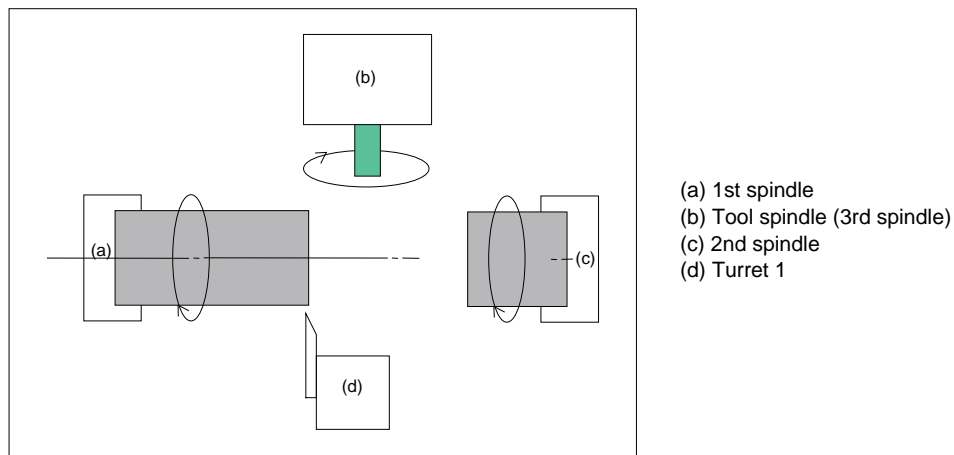
Function and purpose

Multiple spindle control is a function used to control the spindle following 2nd spindle in a machine tool that has a 1st spindle and a spindle following 2nd spindle.

The spindle control method differs between multiple-spindle control I and multiple-spindle control II. Whether to use multiple-spindle control I or multiple-spindle control II depends on the parameter (#1300 ext36/bit0) setting.

Multiple-spindle control I: Control following the spindle selection command (G43.1, etc.) and spindle control command ([S*****;] or [S O =*****;]), etc.
(ext36/bit0 = 0)

Multiple-spindle control II: Control following the external signal (spindle command selection signal, spindle selection signal) and spindle control command ([S***** ;] only), etc.
(ext36/bit0 = 1)
The spindle selection command [S O =*****;] cannot be used.



10.10.1 Multiple-spindle Control I (spindle control command) ; S ○ =



Function and purpose

In addition to the S***** command, the 1st to 4th spindles can be identified and commanded with the S ○ = ***** command.



Command format

S ○ = ***** ; ... Multiple-spindle Control I (spindle control command)

○	Designate the spindle No. with one numeric character (1: 1st spindle/2: 2nd spindle/3: 3rd spindle/4: 4th spindle). A variable can be designated.
*****	Rotation speed or constant surface speed command. A variable can be designated.

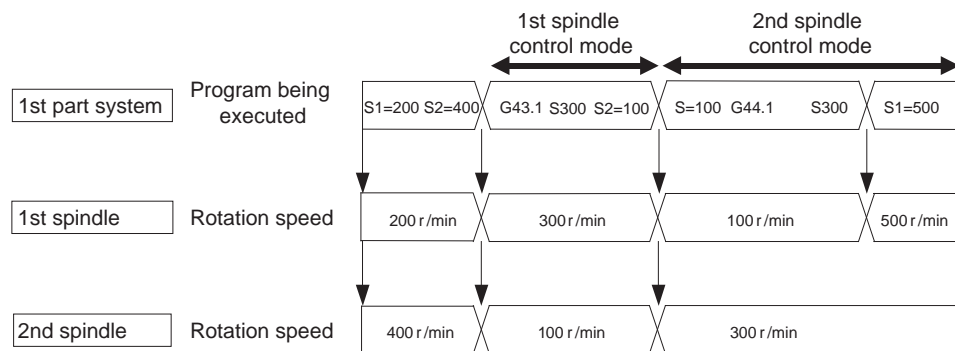
(Note 1) A program error (P35) will occur if the ○ value is not 1 to 4.

(Note 2) A program error (P33) will occur if G47.1 is issued in a modal.



Detailed description

- Each spindle command is delimited by the details of ○ .
(Example)
S1 = 3500 ; 1st spindle 3500(r/min) command
S2 = 1500 : 2nd spindle 1500(r/min) command
- Multiple spindles can be commanded in one block.
- If two or more commands are issued to the same spindle in a block, the command issued last will be valid.
(Example) S1 = 3500 S1 = 3600 S1 = 3700 ; S1 = 3700 will be valid.
- The S***** command and S ○ =***** command can be used together.
The spindle targeted for the S***** command is delimited by the spindle selection command.
(Example) When G44.1 spindle No. is 2



- The commands for each spindle can be commanded from any machining program in either part system.

10.10.2 Multiple-spindle Control I (spindle selection command) ; G43.1,G44.1,G47.1



Function and purpose

The following S commands (S****) can be changed for the 1st spindle to 4th spindle by issuing the spindle selection command (G43.1, etc. [G group 20]).



Command format

G43.1 ; ... 1st spindle control mode ON

G44.1 ; ... Selected spindle control mode ON (Selected spindle No. is set with SnG44.1).

G47.1 ; ... All spindles simultaneous control mode ON



Detailed description

- (1) The selected spindle No. is set with the parameter (#1534 SnG44.1).
- (2) The spindle selection commands are modal G codes.
- (3) A program error (P33) will occur if the spindle selection command is issued while the multiple-spindle control II mode is valid.
- (4) The spindle control mode entered when the power is turned ON or reset can be set with the parameters for each part system. The state when the power is turned ON or reset is as follows.

G group 20 modal state	Set with setup parameter "#1199 Sselect". 0: G43.1 1: G44.1 2: G47.1
G44.1 spindle No.	Set with setup parameter "#1534 SnG44.1". 0: 2nd spindle 1: 1st spindle 2: 2nd spindle 3: 3rd spindle 4: 4th spindle

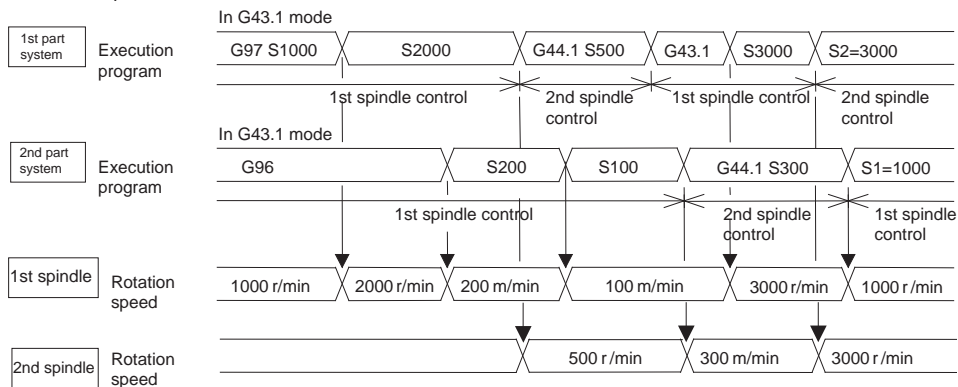
- (5) If the S command is issued in the same block as the spindle selection command, it will be valid for the spindle switched to with the spindle selection command.

(Example)
 G43.1 S100 ; -> Command 100 rotations for 1st spindle.
 S200 G44.1 S300 ; -> After commanding 200 rotations for 1st spindle, command 300 rotations for 2nd spindle.
 (SnG44.1 = 2)

- (6) If a spindle that does not exist is set, the 2nd spindle will be used. Note that if there is only one spindle, the 1st spindle will be used.

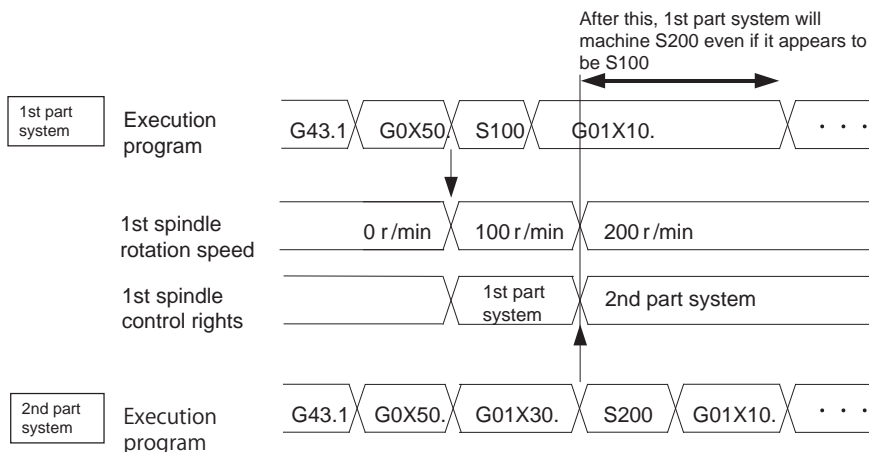
- (7) Commands can be issued from any part system.

When G44.1 spindle No. is 2



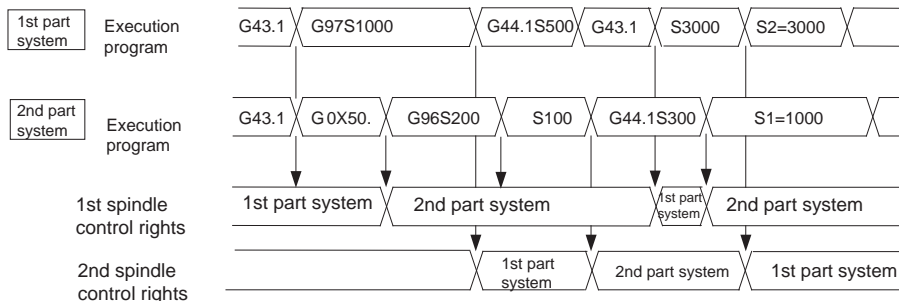
Spindle control rights

If the constant surface speed control, S command and spindle related M command are commanded randomly from each part system to one spindle, the spindle may not operate correctly. For example, if S is commanded (feed per minute) during constant surface speed control with 1st part system, the rotation speed will not change from the 1st part system side, and the actual rotation speed will rely on 2nd part system from which S was commanded last. These commands must be commanded from only one part system, or commanded simultaneously while using standby, etc.

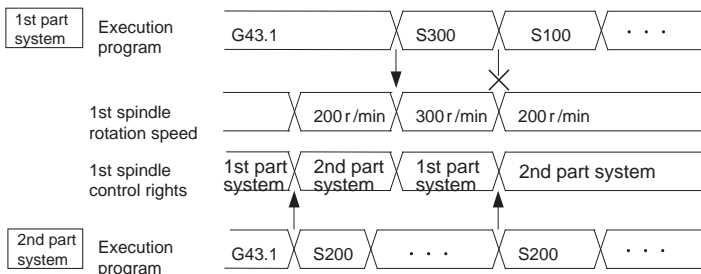


[Spindle control rights transfer conditions] When G44.1 spindle No. is 2

(1) The part system that commanded the S command last has the spindle control rights.



(2) If different S commands are executed simultaneously in two part systems, the part system with the larger part system No. will have the priority. That part system will also have the control rights.





Relation with other functions

The following functions change after the spindle selection command.

- (1) S command (S****)
The S command in G97 (rotation speed command)/G96 (constant surface speed command) will be the command for the spindle designated with the spindle selection command.
- (2) Spindle clamp speed command
The spindle clamp speed command issued with G92 S_Q_ will also follow the spindle selection command mode.
- (3) Per rotation command (synchronous feed)
Even if F is commanded in the G95 mode, the per rotation feedrate for the spindle designated with the spindle selection command will be applied.
- (4) Constant surface speed control spindle changeover
The constant surface speed control will also follow the spindle selection command mode.
If spindle other than the current mode is commanded with S ○ =****, the spindle rotation speed command designated with ○ will have the priority.

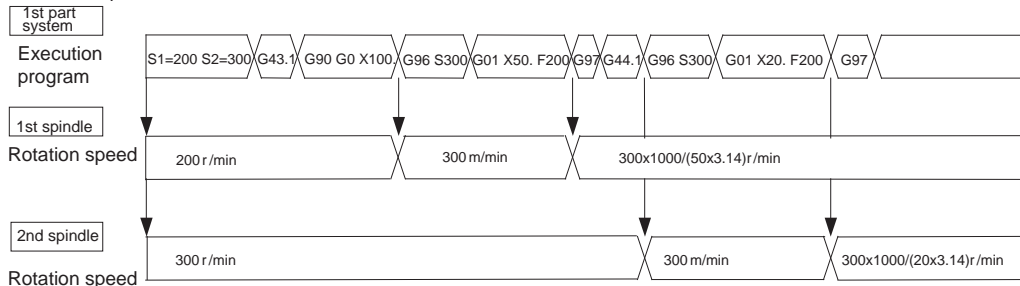
(Example)

S2 = **** in G43 system mode 2nd spindle rotation speed command

S1 = **** in G44 system 2nd spindle selection mode 1st spindle rotation speed command

Note that the constant surface speed command in the constant surface speed control (G96) will be modal information independent from the rotation speed command.

When G44.1 spindle No. is 2



10.10.3 Multiple-spindle Control II



Function and purpose

Multiple-spindle control II is a function that designates which spindle to select with the signals from PLC. The command is issued to the spindle with one S command.

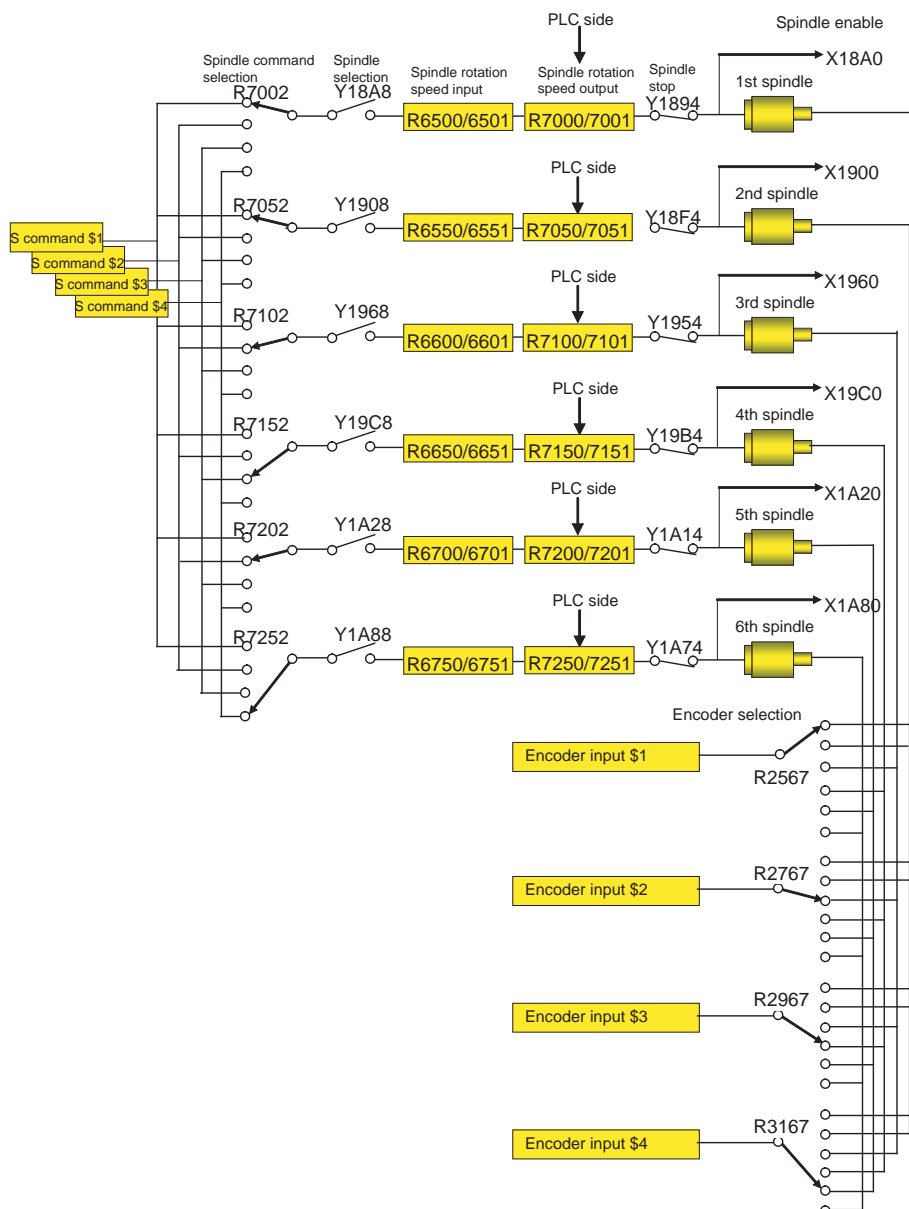


Detailed description

(1) Spindle command selection, spindle selection

The S command to the spindle is output as the rotation speed command to the selected spindle when the spindle selection signal (SWS) from the PLC turns ON. The selected spindle rotates at the output rotation speed. The spindle whose selection is canceled when the spindle selection signal (SWS) turns OFF maintains the speed at which it was rotating at before being canceled. This allows each axis to be simultaneously rotated at differing rotation speeds.

The spindle command selection signal is used to determine which part system each spindle receives the S command from.



(Note) Refer to the PLC Interface Manual for details on each signal.



Relation with other functions

- (1) Spindle clamp speed setting (G92)
 This is valid only on the spindle selected with the spindle selection signal (SWS).
 The spindle not selected with the spindle selection signal (SWS) maintains the speed at which it was rotating at before being canceled.
 (The spindle clamp speed is maintained with the G92 command.)
- (2) Constant surface speed control
 Constant surface speed control can be applied on all spindles.
 The spindle rotation speed is automatically controlled during constant surface speed control, so when machining with constant surface speed, the spindle selection signal (SWS) for that spindle must be left ON.
 The spindle not selected with the spindle selection signal (SWS) maintains the speed at which it was rotating at before being canceled.
- (3) Thread cutting/synchronous feed
 The threads are cut with the spindle selected with the spindle selection signal (SWS). The encoder feedback selected with the encoder selection signal is used.
- (4) Polygon machining (servo-spindle)
 The polygon machining spindle is selected with the spindle selection signal (SWS). Do not select the polygon machining spindle multiple times, and do not change the polygon machining spindle selection signal during the polygon machining mode.
 If a C axis mode command is issued to the polygon machining spindle, the "M01 operation error 1026" will occur. When the C axis command is canceled, the error will be canceled and machining will resume.
 If synchronous tapping is commanded during polygon machining, a program error (P34) will occur.
- (5) Synchronous tapping
 The synchronous tapping spindle is selected with the spindle selection signal (SWS).
 Select the synchronous tapping spindle before issuing the synchronous tapping command. Do not change the synchronous tapping spindle selection signal during the synchronous tapping mode.
 If a C axis mode command is issued to the synchronous tapping spindle, the "M01 operation error 1026" will occur. When the C axis command is canceled, the error will be canceled and machining will resume.
 If a polygon machining command is issued to the synchronous tapping spindle, the "M01 operation error 1026" will occur. When the polygon machining command is canceled, the error will be canceled and machining will resume.
- (6) Asynchronous tapping
 The asynchronous tapping spindle is selected with the spindle selection signal (SWS).
 Select the asynchronous tapping spindle before issuing the tapping command. Input a calculation request to change the asynchronous tapping spindle selection. Do not change the asynchronous tapping spindle selection signal during the asynchronous tapping mode.
- (7) Tapping retract
 The tapping retract spindle is selected with the spindle selection signal (SWS).
 Select the spindle for which the tapping cycle execution is stopped before turning the tapping retract signal ON. If tapping retract is executed when a different spindle is selected, the "M01 operation error 1032" will occur. Do not change the spindle selection signal during tapping retract.



Restrictions

- (1) The S manual numerical value command is invalid when multiple-spindle control II is valid.
- (2) Setup parameter "#1199 Sselect" is invalid when multiple-spindle control II is valid.
- (3) The spindle control mode changeover G code cannot be used when multiple-spindle control II is valid. A program error (P34) will occur.
- (4) The "S1=***" and "S2=***" commands are invalid when multiple-spindle control II is valid. The program error (P33) will occur.
- (5) The spindle gear shift command output signal (GR1/GR2) is not output when multiple-spindle control II is valid.

Tool Functions

11.1 Tool Functions (T8-digit BCD)



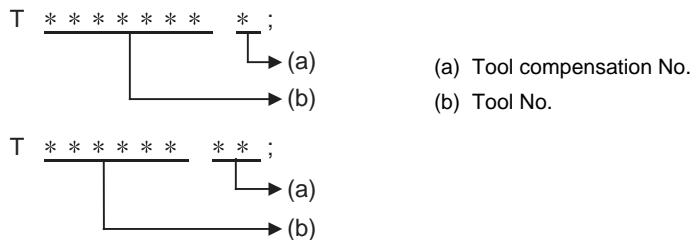
Function and purpose

The tool functions are also known as T functions and they assign the tool numbers and tool compensation number. The designations are made by the 8 digits (0 to 99999999) following the address T. These commands are used with the higher-order six or seven digits indicating the tool No., and the lower-order one or two digits indicating the compensation No.

Which is to be used is determined by the setting in the parameter "#1097 T1digit". The available T commands differ according to each machine, so refer to the instruction manual issued by the machine tool builder. One set of T commands can be issued in one block.



Command format



Refer to the instructions issued by the machine tool builder for the correspondence between the actual tools and the tool Nos. commanded in the program.

BCD codes and start signals are output.

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 the movement command.
- (2) The T function is executed simultaneously with the movement command.

Processing and completion sequences are required for all T commands.

Tool Compensation Functions

12.1 Tool Compensation



Function and purpose

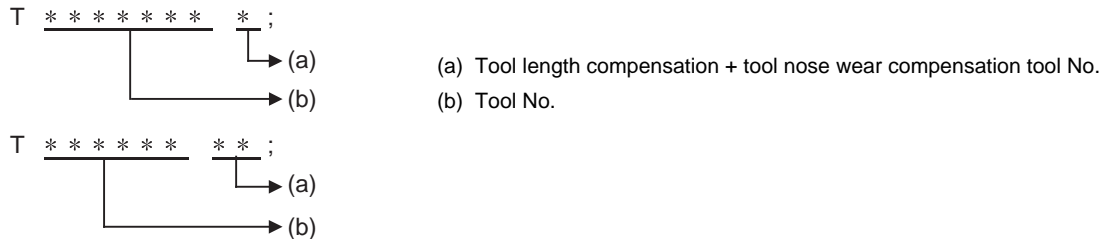
Tool compensation is performed by the T functions which are commanded with the 3-, 4- or 8-digit number following address T. There are two types of tool compensation: tool length compensation and tool nose wear compensation. And there are two ways to issue the commands: to designate both compensations by the last 1 or 2 digits of the T command or to designate the tool nose wear compensation by the last 1 or 2 digits of the T command and the tool length compensation by the tool No. Parameter "#1098 TLno." is used to switch between them. Parameter "#1097 T1digit" is also used to switch between the last 1 or 2 digits for the compensation.

One set of T commands can be issued in one block.

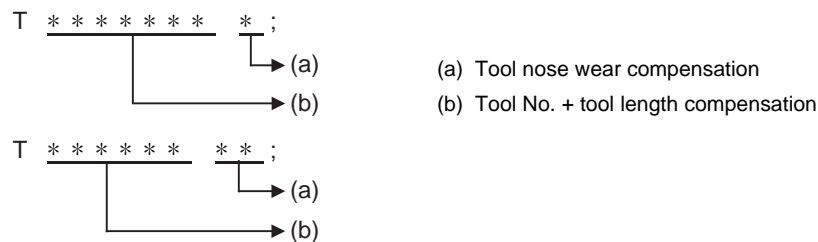


Command format

- (1) When designating the tool length and tool nose wear compensation No. using the last 1 or 2 digits of the T command.



- (2) When differentiating between the tool length compensation No. and tool nose wear compensation No.



The lower two digits of the tool No. are the tool length compensation No.

- (Note 1) When multi-part system

There are cases when each part system has tool data and when the part systems share common tool data. This can be selected with parameter (#1051 MemTol).

Parameter #1051 MemTol 0: Tool data provided for each part system

1: Tool data shared by part systems

When the tool data is shared by the part systems, the compensation amount of tool command for all part systems will be the same (when the same tool compensation No. is commanded).

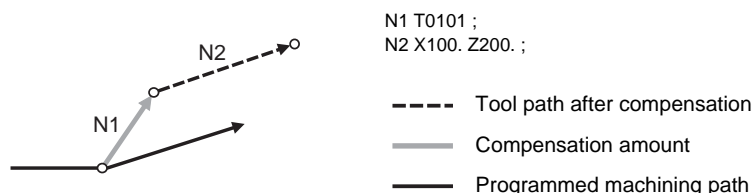
12.1.1 Tool Compensation Start



Detailed description

There are two ways to execute tool compensation and these can be selected by parameters: executing compensation when the T command is executed or executing compensation in the block with a movement command instead of performing compensation when the T command is executed.

(1) Compensation with T command execution



Tool length compensation and tool nose wear compensation are conducted simultaneously.

(Note 1) The movement for a compensation with the T command execution is rapid traverse in a G00 modal. The movement for other modals is cutting feed.

(Note 2) When performing compensation with T command execution, the path is compensated as a linear movement in a circular modal.

(Note 3) When performing compensation with T command execution, if the following G commands are issued in the same block as the T command, compensation will not be performed until other G commands are issued.

However, if an axis is specified by the command, compensation will be performed only to the specified axis.

G04 : Dwell

G11 : Parameter input by program cancel

G92 : Coordinate system setting

G92.1 : Workpiece coordinate system pre-setting

G113 : Spindle synchronization cancel

G114.1 : Spindle synchronization I

G114.2 : Tool spindle synchronization IA

G114.3 : Tool spindle synchronization II

(Note 4) The following commands will temporarily cancel the compensation amount for the axis with any movement commanded: automatic reference position return (G28), 2nd, 3rd, and 4th reference position return (G30) or basic machine coordinate system selection (G53).

(Note 5) If G53 command is issued in the same block as the T command, the compensation amount for the axis with no movement commanded will employ the amount in the previous block.

(2) Compensation with movement command



Tool length compensation and tool nose wear compensation are conducted simultaneously.

(Note 1) When performing compensation with a movement command, compensation is performed if the compensation amount is lower than the parameter "#1084 RadErr" when the first compensation is performed with a circular command. If the amount is higher, the program error (P70) will occur. (This also applies when the circular command and T command are in the same block for compensation with T command execution.)

12.1.2 Expanded Method at Starting Tool Compensation



Function and purpose

By setting the parameter "#1100 Tmove", the compensation operation when T is commanded is selected whether that is carried out when the T command is executed or carried out with superimposed on the movement command.

Also, depending on the setting of the parameter, it is possible to compensate only the wear amount when the T command is executed and to compensate the tool length by superimposing on the movement command.



Detailed description

To compensate only the wear amount when the T command is executed and to compensate the tool length by superimposing on the movement command, set the "#1100 Tmove" to 2.

When 2 is set, the length compensation will follow the setting 1 (superimposed on movement command), and the wear compensation will follow the setting 0 (move when T is commanded).

The compensation operation for each "#1100 Tmove" setting is as shown below.

Length compensation amount: 7.000mm Wear compensation amount: 0.500mm

Machining program example	Machine value		
	#1100 Tmove = 0	#1100 Tmove = 1	#1100 Tmove = 2
G28 X.;	0.000	0.000	0.000
G0 T116 ;	7.500	0.000	0.500
G0 X100. ;	107.500	107.500	107.500
:			

(Note 1) When performing wear compensation with T command execution, if the following G commands are issued in the same block as the T command, compensation will not be performed until other G commands are issued.

However, if an axis is specified by the command, compensation will be performed only to the specified axis.

G04 : Dwell

G11 : Parameter input by program cancel

G92 : Coordinate system setting

G92.1 : Workpiece coordinate system pre-setting

G113 : Spindle synchronization cancel

G114.1 : Spindle synchronization I

G114.2 : Tool spindle synchronization IA

G114.3 : Tool spindle synchronization II

(Note 2) After the compensation amount is temporarily canceled, the compensation amount will return for each movement command axis.

The movement at this time will follow the parameter "#1101 Tabsmv" for both length compensation and wear compensation.

G command involving temporary cancel: G28, G30, G53

(Note 3) If G53 command is issued in the same block as the T command, the compensation amount for the axis with no movement commanded will employ the amount in the previous block.

- (1) Relation with "#1101 Tabsmv" (tool compensation amount superimposition during absolute command)
When "#1100 Tmove" is set to 2, the compensation operation will follow the "#1101 Tabsmv" setting.
- (2) Parameters

# No.	Item	Details	Setting range
1100	Tmove	Designate the timing to perform tool length compensation and wear compensation. 0: Compensates when T command is executed. 1: Superimposes and compensates with the movement command in the block where the T command is located. If there is no movement command in the same block, compensation will be executed after the movement command is superimposed in the next movement command block. 2: Compensates wear amount when the T command is executed. Superimposes and compensates a tool length compensation amount with the movement command in the same block. If there is no movement command in the same block, compensation will be executed after the movement command is superimposed in the next movement command block.	0 to 2
1101	Tabsmv	Designate the type of movement command superimposed when Tmove is set to 1 or 2. 0: Compensates regardless of the movement command type. 1: Compensates only at the movement command in the absolute value command.	0 / 1

12.2 Tool Length Compensation

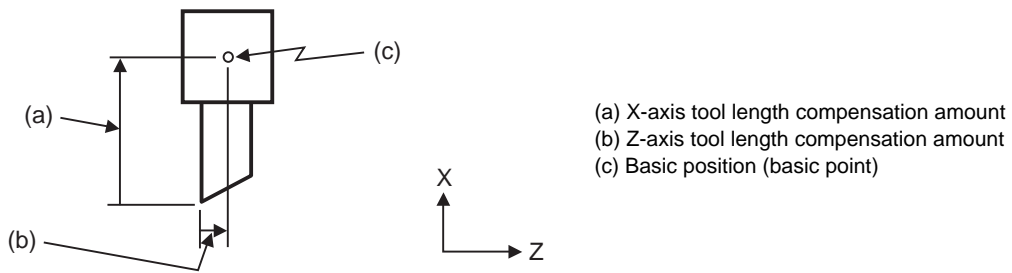


Detailed description

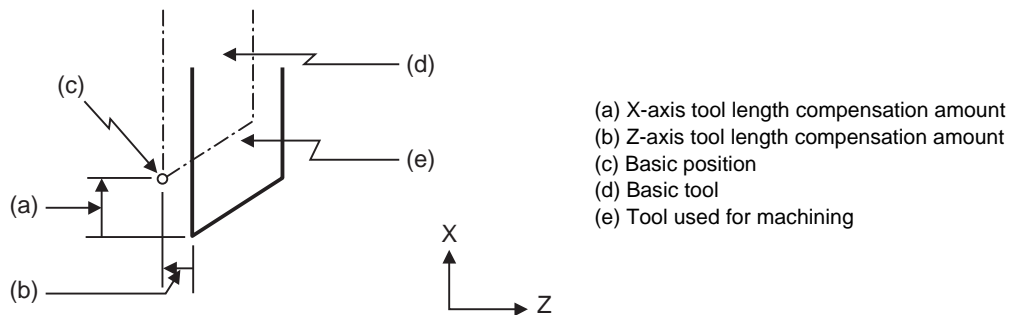
Tool length compensation amount setting

This function compensates the tool length with respect to the programmed basic position. This position may generally be set to either the center position of the turret or the tool nose position of the basic tool.

(1) Center position of turret

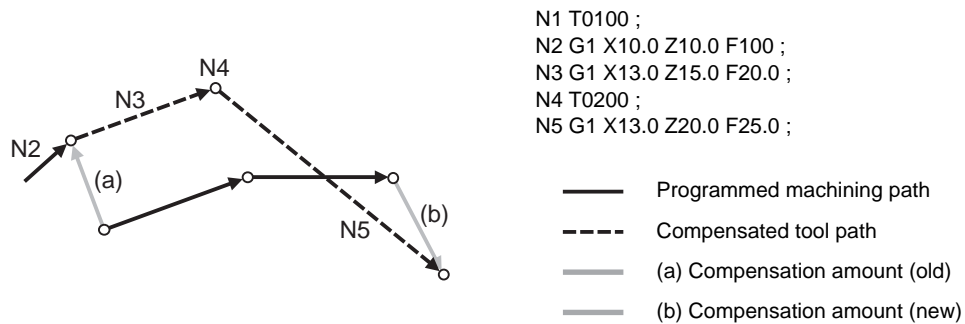


(2) Tool nose position of basic tool



Tool length compensation No. change

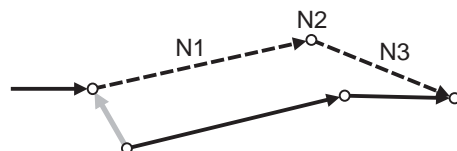
When tool Nos. are changed, the tool length compensation corresponding to the new tool Nos. are added to the movement amounts in the machining program.



In this example, the tool length is compensated with the tool No. and compensation is performed in the block with the movement command.

Tool length compensation cancel

- (1) When the compensation No. of 0 has been assigned
Compensation is canceled when 0 has been assigned as the tool length compensation No. by the T command.

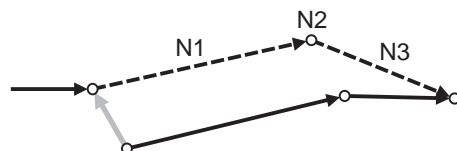


```
N1 X10.0 Z10.0 F10 ;
N2 T0000 ;
N3 G1 X10.0 Z20.0 ;
```

In this case, compensation is performed by the block with the movement command.

— Programmed machining path
- - - Compensated tool path
— Compensation amount

- (2) When the assigned compensation amount is 0
Compensation is canceled when the compensation amount in the tool length compensation No. assigned by the T command is 0.



```
N1 G1 X10.0 Z10.0 F10 ;
N2 T0100 ;
N3 G1 X10.0 Z20.0 ;
```

In this case, compensation is performed by the block with the movement command.

— Programmed machining path
- - - Compensated tool path
— Compensation amount



Precautions

- (1) When G28, G29 or G30 is commanded, the compensation is temporarily canceled. Therefore, the machine moves to the position where the compensation was canceled and the compensation amount is stored in the memory. This means that with the next movement command the machine will move to the compensation position.
- (2) When G28, G29 or G30 and the compensation cancel are commanded in the same block, the machine is moved to the position where the compensation was canceled, however the compensation amount remains stored in the memory. This means that the display coordinate may be displayed including the compensation amount. Issue these command in the separate block so that the compensation amount should not be stored in the memory.
- (3) Even if the compensation amount of the compensation No. currently selected by MDI is changed during automatic operation, the changed compensation amount will not be valid unless a T command with the same No. is executed again.
- (4) The tool length compensation and tool nose wear compensation amounts are cleared by resetting and by emergency stop. They can be retained by parameter "#1099 Treset".

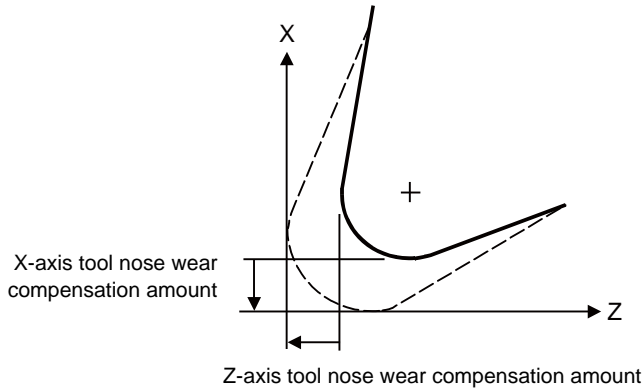
12.3 Tool Nose Wear Compensation



Detailed description

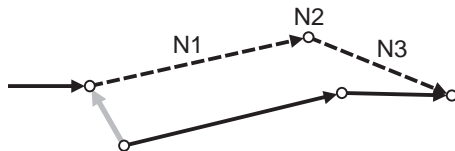
Tool nose wear compensation amount setting

The wear sustained by the tool being used can be compensated.



Tool nose wear compensation cancel

Tool nose wear compensation is canceled when 0 has been assigned as the compensation No.



```
N1 G1 X10.0 Z10.0 F10 ;
N2 T0100 ;
N3 G1 X10.0 Z20.0 ;
```

In this case, compensation is performed by the block with the movement command.

- Programmed machining path
- - - Compensated tool path
- Compensation amount



Precautions

- (1) When G28, G29 or G30 is commanded, the compensation is temporarily canceled. Therefore, the machine moves to the position where the compensation was canceled and the compensation amount is stored in the memory. This means that with the next movement command the machine will move to the compensation position.
- (2) When G28, G29 or G30 and the compensation cancel are commanded in the same block, the machine is moved to the position where the compensation was canceled, however the compensation amount remains stored in the memory. Issue these command in the separate block so that the compensation amount should not be stored in the memory.
- (3) Even if the compensation amount of the compensation No. currently selected by MDI is changed during automatic operation, the changed compensation amount will not be valid unless a T command with the same No. is executed again.
- (4) The tool length compensation and tool nose wear compensation amounts are cleared by resetting and by emergency stop. They can be retained by parameter "#1099 Treset".

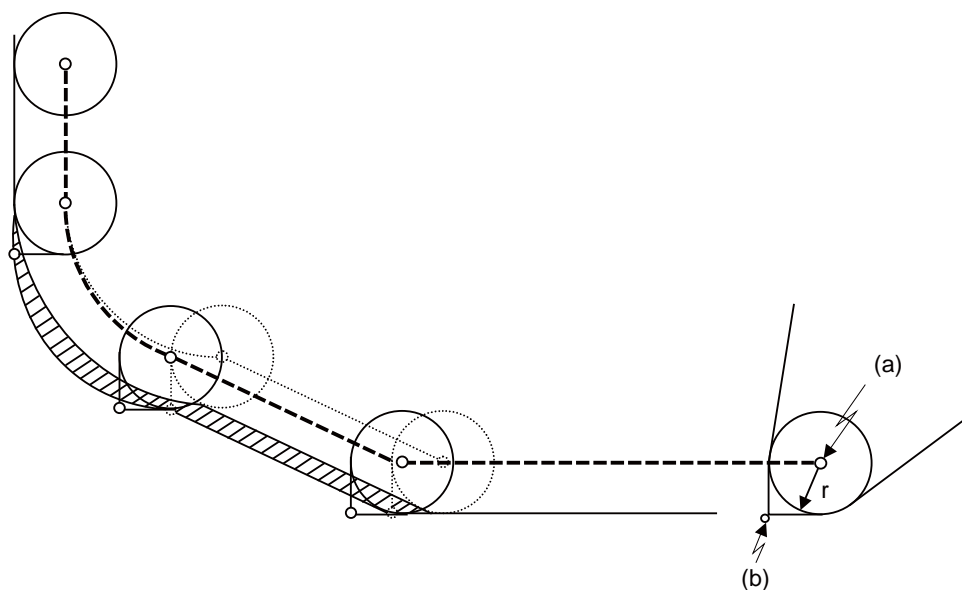
12.4 Tool Nose R Compensation ; G40,G41,G42,G46



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 is a function for automatically calculating and offsetting this error by setting the tool nose radius value.

These command codes enable to choose the offset direction to be fixed or automatically identified.



- (a) Tool nose center
- (b) Hypothetical tool nose point
- (r) Tool nose R

- Tool nose center path with no tool nose radius compensation (Shaded part indicates the cutting shape gap)
- Tool nose center path with tool nose radius compensation



Command format

G40(Xx/Uu Zz/Ww) ; ... Tool nose radius compensation cancel

G41(Xx/Uu Zz/Ww) ; ... Tool nose radius compensation left

G42(Xx/Uu Zz/Ww) ; ... Tool nose radius compensation right

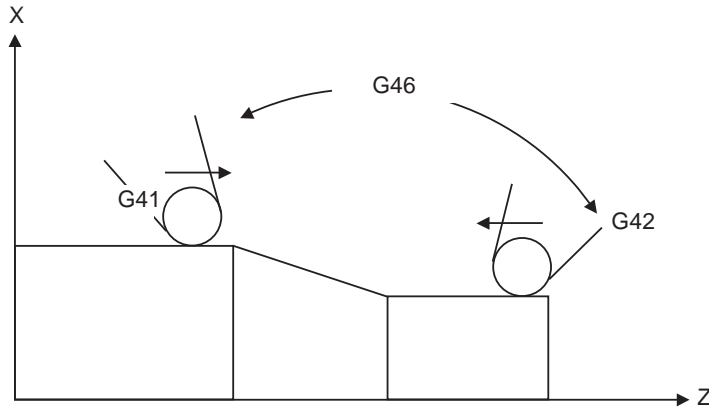
G46(Xx/Uu Zz/Ww) ; ... Tool nose radius compensation (automatic direction identification) ON

X_/U_	X axis end point coordinate (X is the absolute value of workpiece coordinate system, U is the incremental value from current position)
Z_/W_	Z axis end point coordinate (Z is the absolute value of workpiece coordinate system, W is the incremental value from current position)

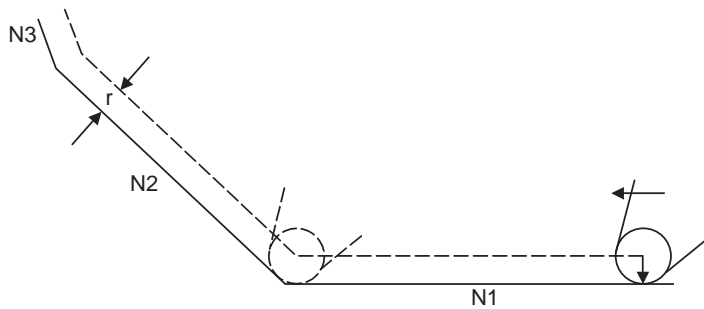


Detailed description

- (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. G46 automatically identifies the compensation direction by the preset hypothetical tool nose point and movement commands in the machining program. G40 cancels the tool nose radius compensation mode.



- (2) Tool nose radius 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 radius. In the figure below, "r" is the tool nose radius compensation amount (nose radius). The tool nose radius 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 valid also for fixed cycles (G77 to G79) and rough cutting cycles (G70, G71, G72, G73). However, in the rough cutting cycles, cutting will be done with the compensation mode canceled since the finished shape already includes the compensation amount and, upon completion of the cutting, operation will automatically return to the compensation mode.
- (5) Compensation mode will be temporarily canceled in 1 block before the thread cutting command block.
- (6) A tool nose radius compensation (G41 or G42) command can be issued during tool nose radius compensation (G46). For this, compensation mode does not need to be canceled with G40.
- (7) The compensation plane, movement axes and next advance direction vector follow the plane selection command designated by G17, G18 or G19.
 - G17 ... X-Y plane X, Y, I, J
 - G18 ... Z-X plane Z, X, K, I
 - G19 ... Y-Z plane Y, Z, J, K

12.4.1 Tool Nose Point and Compensation Directions



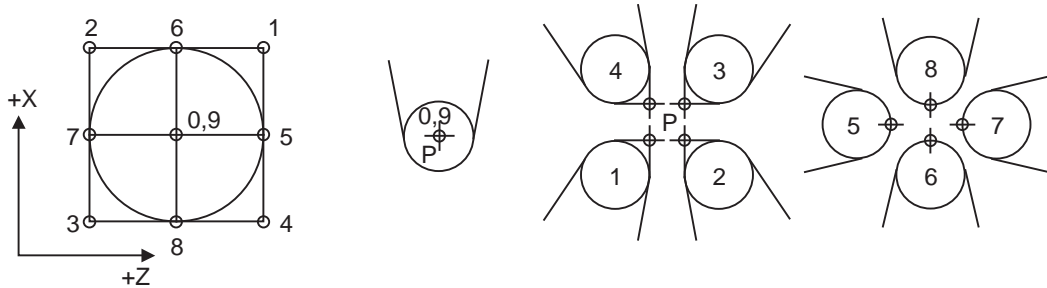
Detailed description

Tool nose point

Because a tool nose is generally rounded, the programmed tool nose position is adjusted to a point P shown in the examples figures below.

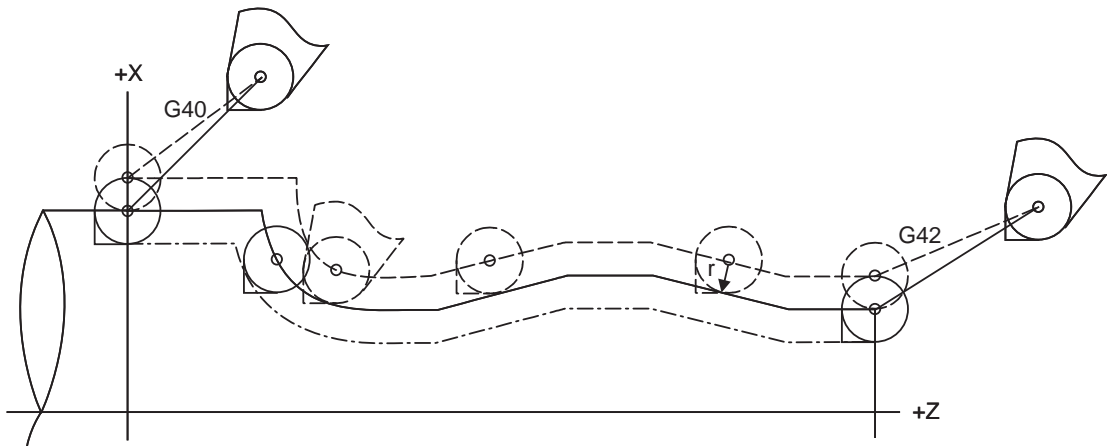
For tool nose radius compensation, select one of the points in the figures below for each tool length numbers and preset the positional relationship.

(Selects 1 to 8 in the G46 mode and 0 to 9 in the G41/G42 mode.)



Tool nose point and compensation operation

- (1) When the nose R center is adjusted to the machining start position (Tool nose point 0 or 9)



- Programmed path or machining shape with tool nose radius compensation
- Nose R center path with tool nose radius compensation
- · - · - · - Machining shape with no tool nose radius compensation

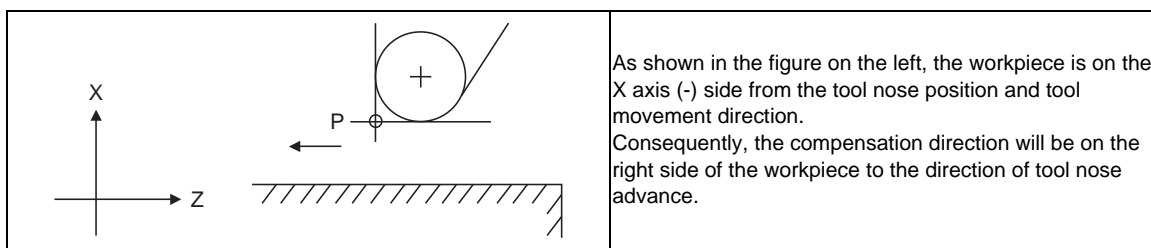
- (3) When the compensation direction during tool nose radius compensation corresponds to a "x" in the table below, the previous direction will be resumed.

[How to determine the compensation direction by the movement vectors and tool nose point in command G46]

		Compensation direction of tool nose									
		Tool nose points									
		①	②	③	④	⑤	⑥	⑦	⑧		
Direction of tool nose advance	Movement vector (tool nose points 1 to 4)		Right	Right	Left	Left	x	Right	x	Left	→
			x	Right	x	Left	Left	Right	Right	Left	
			Left	Right	Right	Left	Left	x	Right	x	↑
			Left	x	Right	x	Left	Left	Right	Right	
	Movement vector (tool nose points 5 to 8)		Left	Left	Right	Right	x	Left	x	Right	←
			x	Left	x	Right	Right	Left	Left	Right	
			Right	Left	Left	Right	Right	x	Left	x	↓
			Right	x	Left	x	Right	Right	Left	Left	

- (Note 1) "x" marks in the table indicate that the compensation direction cannot be determined by the movement vector (tool nose points).
- (Note 2) The "↗" mark denotes a movement vector in the 45° direction. (The other movement vectors are based on this.)
- (Note 3) The "◊" mark denotes a movement vector with a range larger than 45° and smaller than 135°. (The other movement vectors are based on this.)

(Example)When tool nose point 3, movement vector in the Z axis (-) direction (when movement vector is -)



12.4.2 Tool Nose Radius Compensation Operations



Detailed description

Tool nose radius compensation cancel mode

The tool nose radius compensation cancel mode is established by any of the following conditions.

- (1) After the power has been turned 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 tool radius compensation cancel command (G40) has been executed
- (5) After tool No. 0 has been selected (T00 has been executed)

The offset vectors are zero in the compensation cancel mode, and the tool nose point path coincides with the programmed path.

Programs including tool nose radius compensation must be terminated in the compensation cancel mode.

Tool nose radius compensation start (start-up)

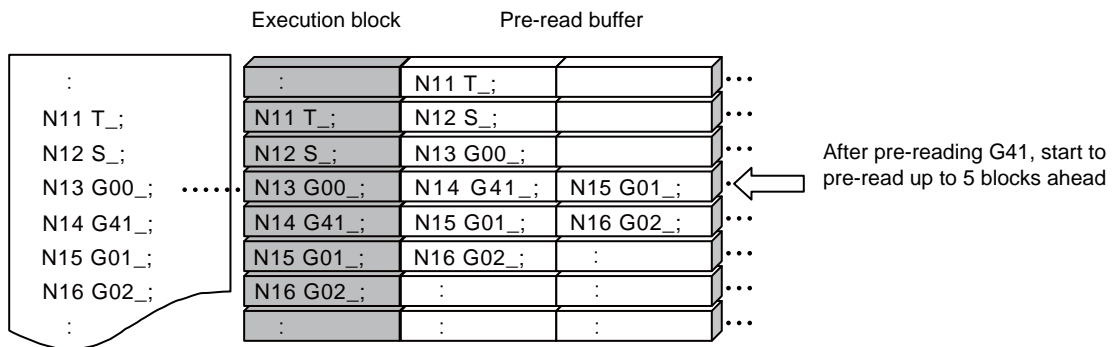
Tool nose radius compensation starts when all the following conditions are satisfied in the compensation cancel mode.

- (1) The movement command is issued after G41, G42 or G46.
- (2) The movement command is not a circular command.

Before starting a compensation, 2 to 5 blocks are pre-read for the intersection point calculation regardless of single block operation or continuous operation. (Two blocks are pre-read if there is a movement command, up to 5 blocks are pre-read if not)

Similarly, during compensation mode, up to five blocks are pre-read for compensation calculation.

[Control mode transition diagram]



There are two ways of starting the compensation operation: type A and type B.

The type to be used is selected by setting of parameter "#8157 tool nose comp type B".

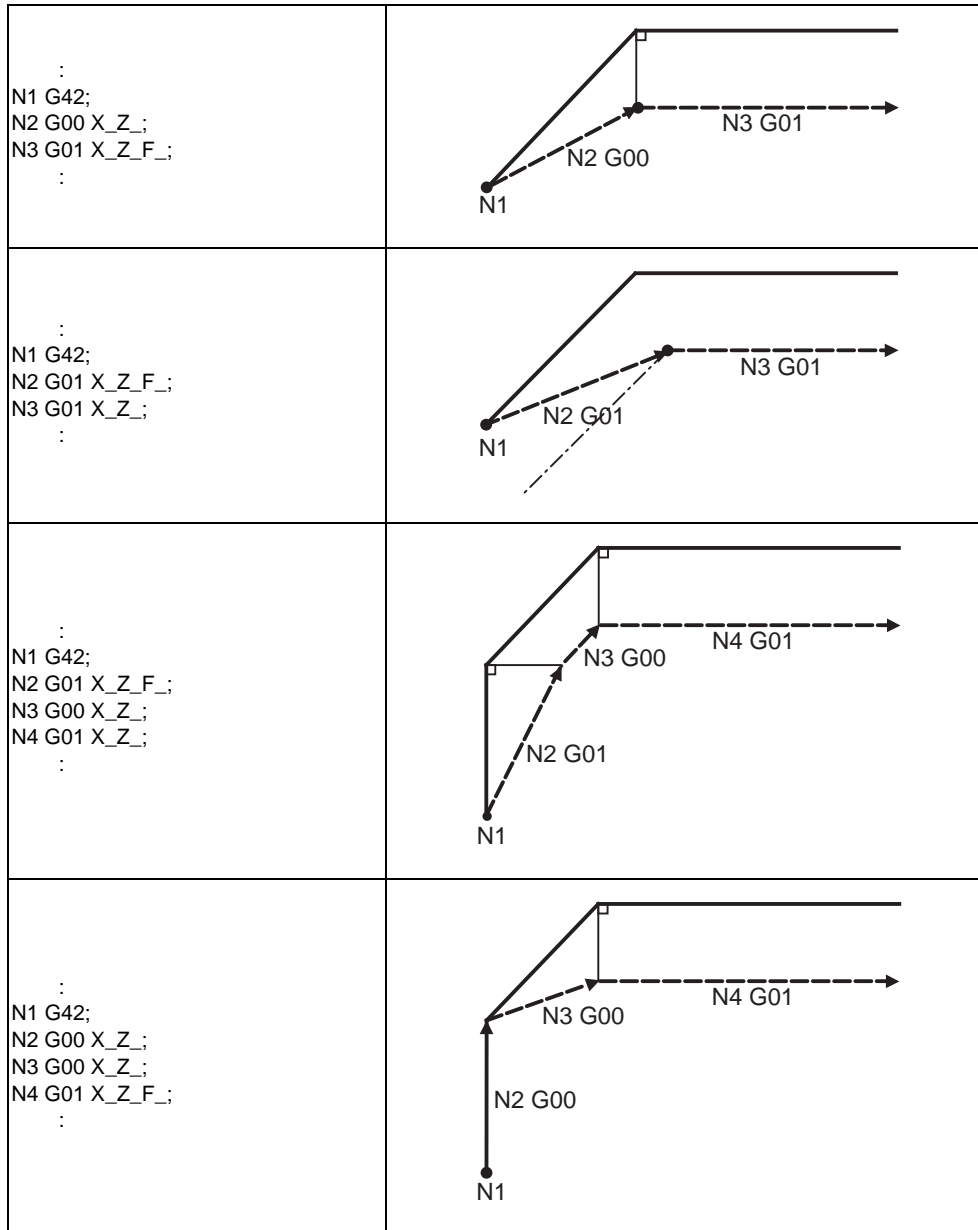
This type is used in common with the compensation cancel type.

#8157 Nose R comp type B	Type	Explanation
0	Type A	When starting up/canceling a command block with tool nose radius compensation and 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.
1	Type B	When starting up/canceling a command block with tool nose radius compensation and radius compensation, Type B will conduct intersection operation processing to the command block and the next block.

Start operation for tool nose radius compensation

When starting tool nose radius compensation, the tool will not move as much as the specified compensation amount by either G41, G42 or G46 command alone. Tool nose radius compensation cannot be applied to the G00 command. It can be applied from G01, G02 or G03 command. Note that even if there is an axis command, the tool nose radius compensation will not be applied unless there is movement.

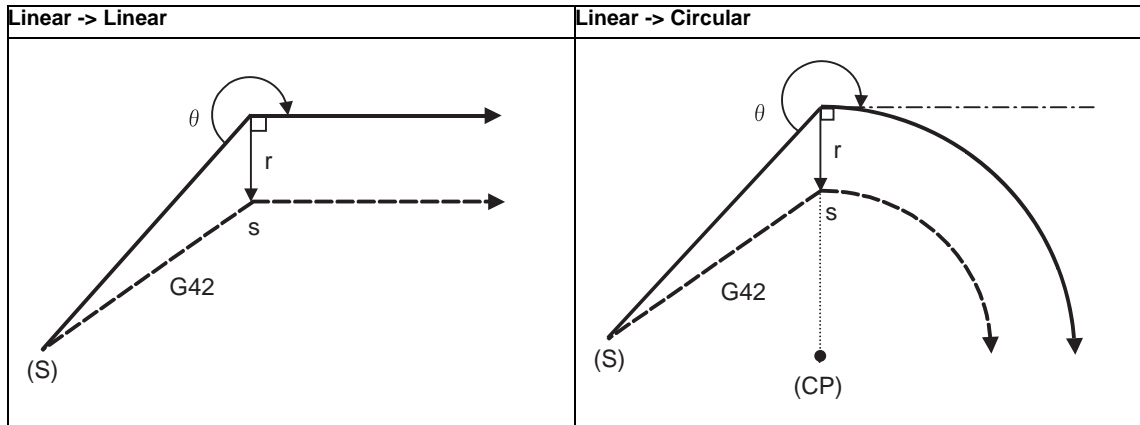
(1) When an independent G41/G42/G46 command is issued at an inside corner



— Program path

- - - Nose R center path

(2) When a G41/G42/G46 command is issued at an inside corner, in the same block as a movement command



(S) Start point

(CP) Circular radius

r : Nose R radius

s : Stop point with single block

—— Program path

----- Nose R center path

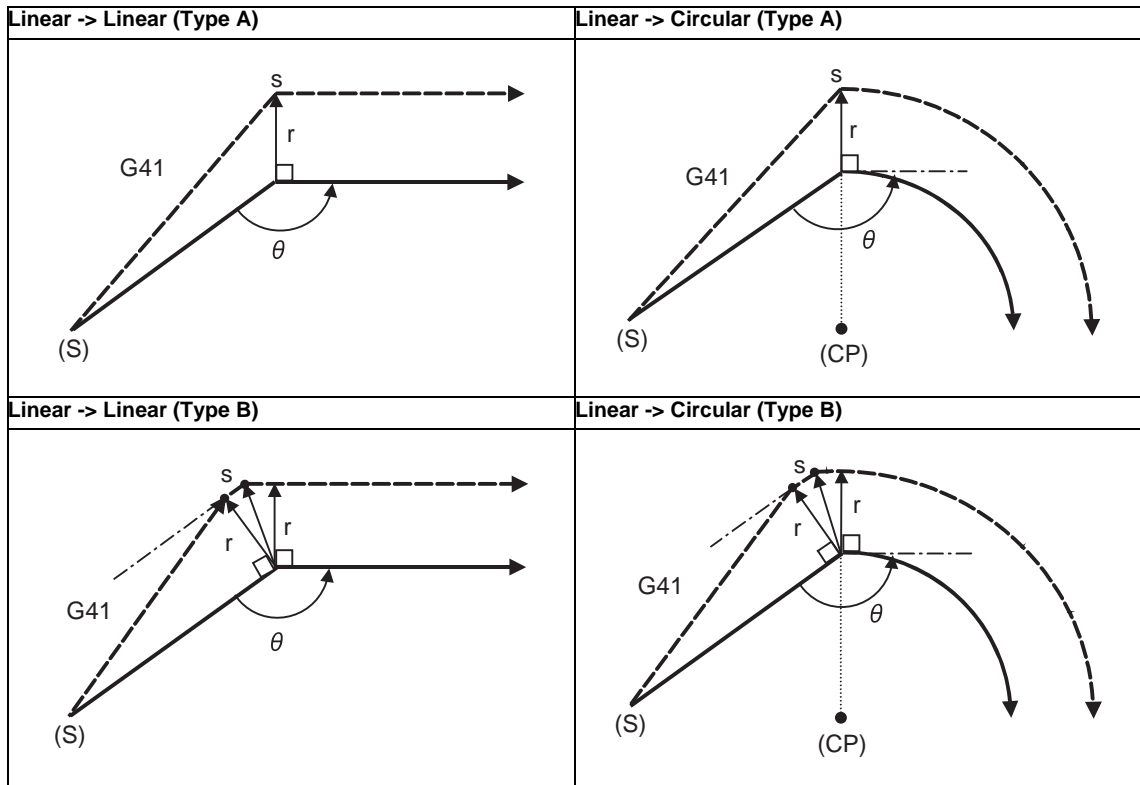
(3) When an independent G41/G42/G46 command is issued at an outside corner (obtuse angle)

	Type A	Type B
<pre> : N1 G41; N2 G00 X_Z_; N3 G01 X_Z_F_; : </pre>		
<pre> : N1 G41; N2 G01 X_Z_F_; N3 G01 X_Z_; : </pre>		
<pre> : N1 G41; N2 G01 X_Z_F_; N3 G00 X_Z_; N4 G01 X_Z_; : </pre>		
<pre> : N1 G41; N2 G00 X_Z_; N3 G00 X_Z_; N4 G01 X_Z_F_; : </pre>		

— Program path

- - - - - Nose R center path

- (4) When a G41/G42/G46 command is issued at an outside corner (obtuse angle), in the same block as a movement command [$90^\circ \leq \theta < 180^\circ$]



(S) Start point

(CP) Circular radius

r : Nose R radius

s : Stop point with single block

— Program path

- - - - - Nose R center path

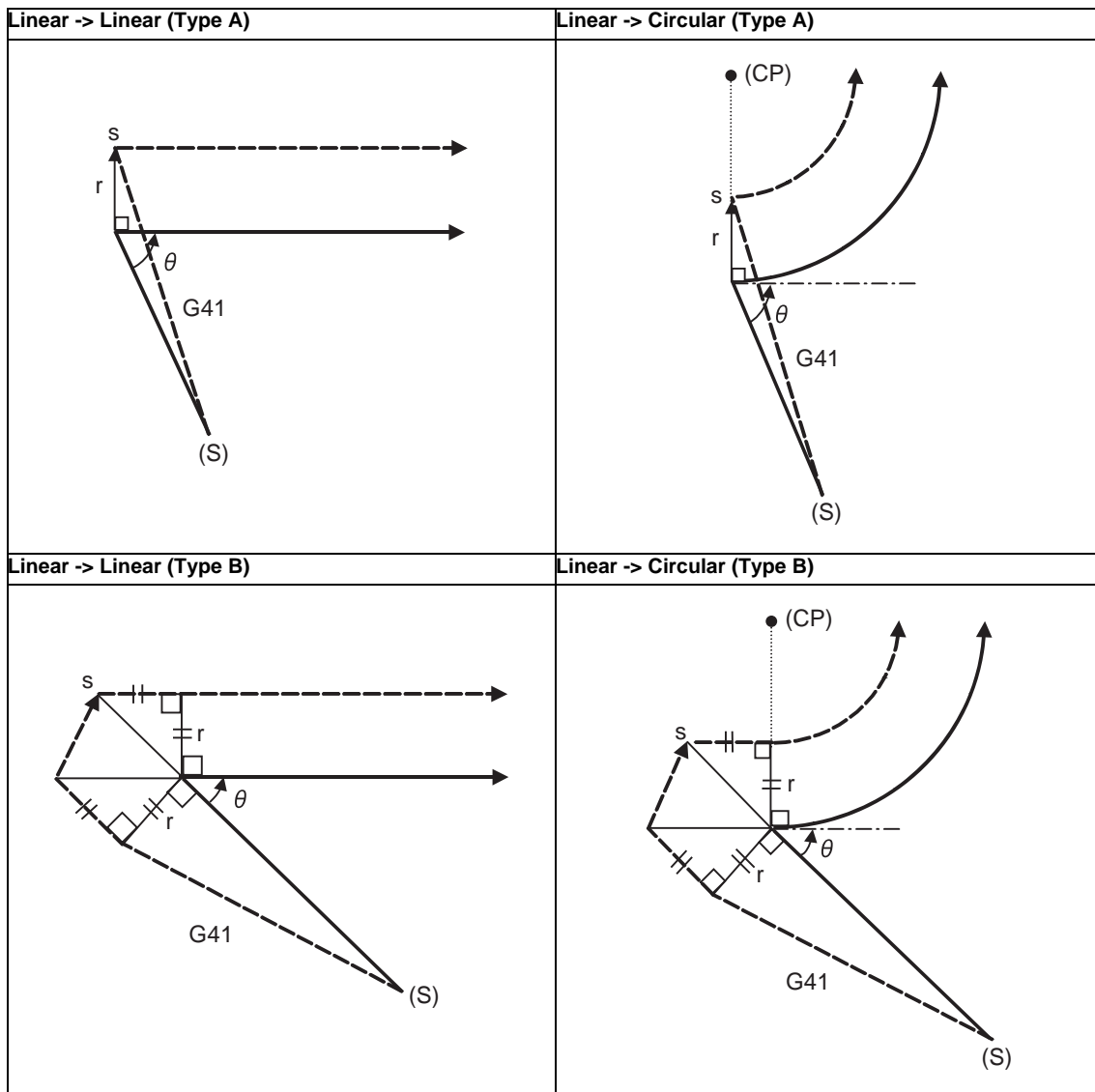
(5) When an independent G41/G42/G46 command is issued at an outside corner (acute angle)

	Type A	Type B
<pre> : N1 G41; N2 G00 X_Z_; N3 G01 X_Z_F_; : </pre>		
<pre> : N1 G41; N2 G01 X_Z_F_; N3 G01 X_Z_; : </pre>		
<pre> : N1 G41; N2 G01 X_Z_F_; N3 G00 X_Z_; N4 G01 X_Z_; : </pre>		
<pre> : N1 G41; N2 G00 X_Z_; N3 G00 X_Z_; N4 G01 X_Z_F_; : </pre>		

— Program path

- - - - - Nose R center path

(6) When a G41/G42/G46 command is issued at an outside corner (acute angle), in the same block as a movement command [$\theta < 90^\circ$]



(S) Start point

(CP) Circular radius

r : Nose R radius

s : Stop point with single block

— Program path

- - - - - Nose R center path

(Note 1) 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 straight line/circular arc to perform compensation to the program path (G00, G01, G02, G03).

Even if the same compensation command (G41, G42, G46) is issued in a tool nose radius compensation (G41, G42, G46) mode, the command will be ignored.

When 4 or more blocks without movement command are continuously specified in the compensation mode, overcutting or undercutting will occur.

When the M00 command is issued during tool nose radius compensation, pre-reading will be prohibited.

(1) Machining an outside corner

<p>Linear -> Linear ($90^\circ \leq \theta < 180^\circ$)</p>	<p>Linear -> Linear ($0^\circ < \theta < 90^\circ$)</p>
<p>Linear -> Linear ($90^\circ \leq \theta < 180^\circ$)</p>	<p>Linear -> Linear ($0^\circ < \theta < 90^\circ$)</p>
<p>Circular -> Linear ($90^\circ \leq \theta < 180^\circ$)</p>	<p>Circular -> Linear ($0^\circ < \theta < 90^\circ$)</p>
<p>Circular -> Circular ($90^\circ \leq \theta < 180^\circ$)</p>	<p>Circular -> Circular ($0^\circ < \theta < 90^\circ$)</p>

(CP) Circular radius
 r : Nose R radius
 s : Stop point with single block

— Program path

- - - - - Nose R center path

(2) Machining an inside corner

<p>Linear -> Linear (Obtuse angle)</p>	<p>Linear -> Linear (Acute angle)</p>
<p>Linear -> Circular (Obtuse angle)</p>	<p>Linear -> Circular (Acute angle)</p>
<p>Circular -> Linear (Obtuse angle)</p>	<p>Circular -> Linear (Acute angle)</p>
<p>Circular -> Circular (Obtuse angle)</p>	<p>Circular -> Circular (Acute angle)</p>

(CP) Circular radius

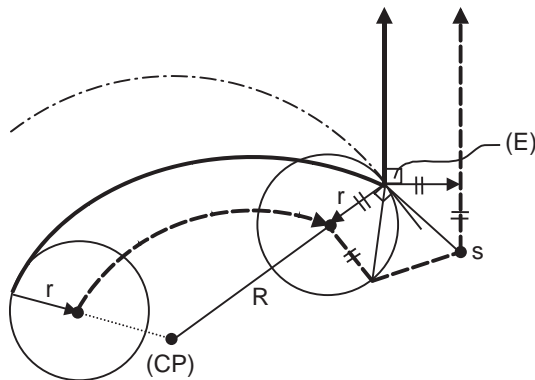
r : Nose R radius

s : Stop point with single block

— Program path

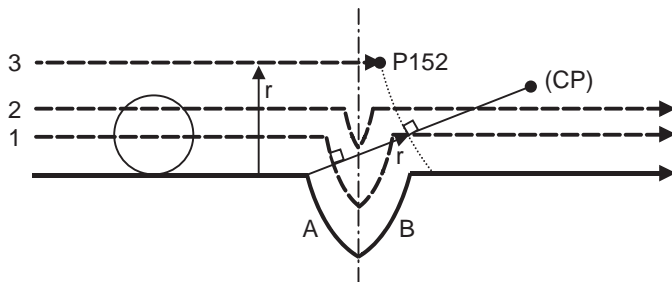
- - - Nose R center path

- (3) When the circular end point is not on the circular
 If the error after performing the compensation is within the parameter (#1084 RadErr) range, circular's start point to end point will be interpolated as a spiral circular.



(E) End point of circular
 (CP) Center of circular
 r : Nose R radius

- (4) When the inner intersection point does not exist
 In cases like the figure below, the intersection point of circulars A and B may not exist depending on the compensation amount.
 In such cases, program error (P152) will be displayed and the tool stops at the end point of the previous block.
 In the pattern 1 and 2 of this figure, machining is possible because nose R radius r is small.
 In pattern 3, nose R radius r is so large that an intersection does not exist and program error (P152) will occur.



(CP) Center of circular A
 r : Nose R radius

— Program path

- - - - - Nose R center path

Tool nose R compensation cancel

Tool nose R compensation mode will be canceled when any of the following conditions is met.

However, this must be any movement command except a circular command.

If the compensation is canceled by a circular command, program error (P151) will occur.

- (1) The G40 command has been executed.
- (2) Tool No. T00 is executed.

The cancel mode is established once the compensation cancel command has been read, 5-block pre-reading is suspended and 1-block pre-reading is made operational.

Tool Nose Radius Compensation Operations

Tool nose radius compensation cancel command results as follows.

- (1) If the command before G40 is G00 after the tool nose radius compensation is completed, the tool nose radius compensation is temporarily stopped, and the tool nose radius compensation is canceled in that state by an independent G40 command.
- (2) If there is an interpolation command before G40 after the tool nose radius compensation is completed, the tool nose radius compensation is not canceled by the independent G40 command, so the nose R center can stop in a vertical position. Instead, the tool nose radius compensation is canceled by the first axis movement command after G40. The nose radius compensation will not be canceled by the axis command if there is no axis movement. If there is no axis movement command after G40, and the program finishes by M02, etc., the tool nose radius compensation will remain valid. Resetting will cancel the tool nose radius compensation, but the operation will not be canceled.
- (3) When an independent T00 command is issued in a block, a tool nose radius cancel mode will result in that block, and the axis will move to the tool nose radius cancel position.

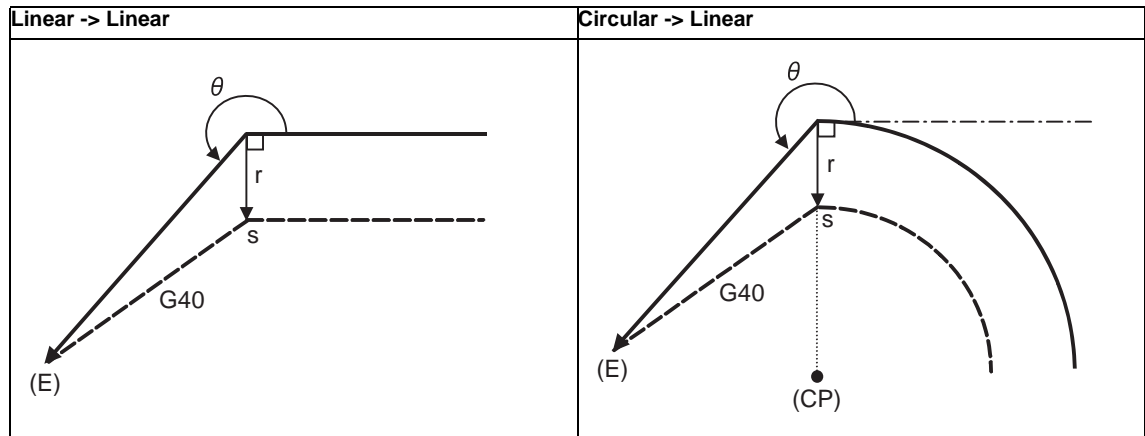
(4) Relation of an inside corner/outside corner and cancel
 (a)-1 When an independent G40 command is issued at an inside corner

<pre> : N1 G01 X_Z_F_; N2 G00 X_Z_; N3 G40; : </pre>	
<pre> : N1 G01 X_Z_F_; N2 G01 X_Z_; N3 G40; N4 M05; N5 G00 X_Z_; : </pre>	
<pre> : N1 G01 X_Z_F_; N2 G00 X_Z_; N3 G01 X_Z_; N4 G40; N5 M05; N6 G01 X_Z_; : </pre>	
<pre> : N1 G01 X_Z_F_; N2 G00 X_Z_; N3 G00 X_Z_; N4 G40; : </pre>	

— Program path

- - - - - Nose R center path

(a)-2 When a G40 command is issued at an inside corner in the same block as a movement command



(E) End point

(CP) Center of circular

r : Nose R radius

s : Stop point with single block

—— Program path

----- Nose R center path

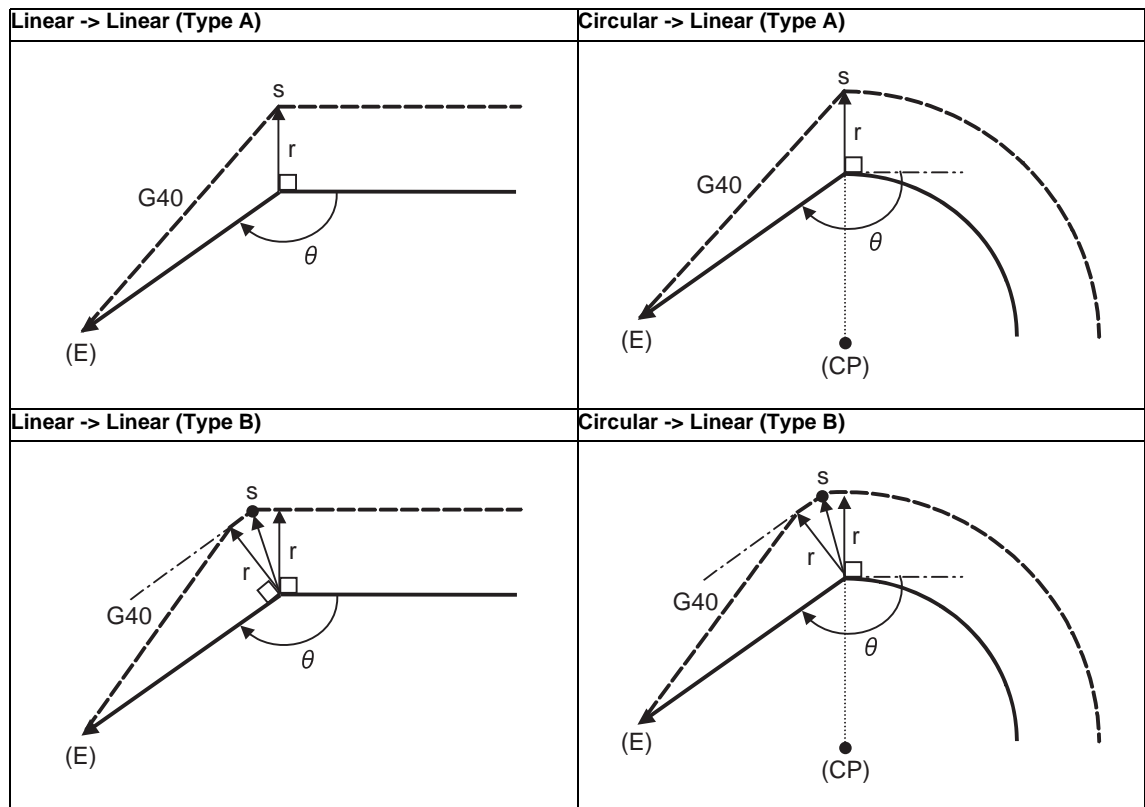
(b)-1 When an independent G40 command is issued at an outside corner (obtuse angle)

	Type A	Type B
<pre> : N1 G01 X_Z_F_; N2 G00 X_Z_; N3 G40; : </pre>		
<pre> : N1 G01 X_Z_F_; N2 G01 X_Z_; N3 G40; N4 M05; N5 G00 X_Z_; : </pre>		
<pre> : N1 G01 X_Z_F_; N2 G00 X_Z_; N3 G01 X_Z_; N4 G40; N5 M05; N6 G01 X_Z_; : </pre>		
<pre> : N1 G01 X_Z_F_; N2 G00 X_Z_; N3 G00 X_Z_; N4 G40; : </pre>		

— Program path

- - - - - Nose R center path

(b)-2 When a G40 command is issued at an outside corner (obtuse angle), in the same block as a movement command



(E) End point

(CP) Center of circular

r : Nose R radius

s : Stop point with single block

— Program path

--- Nose R center path

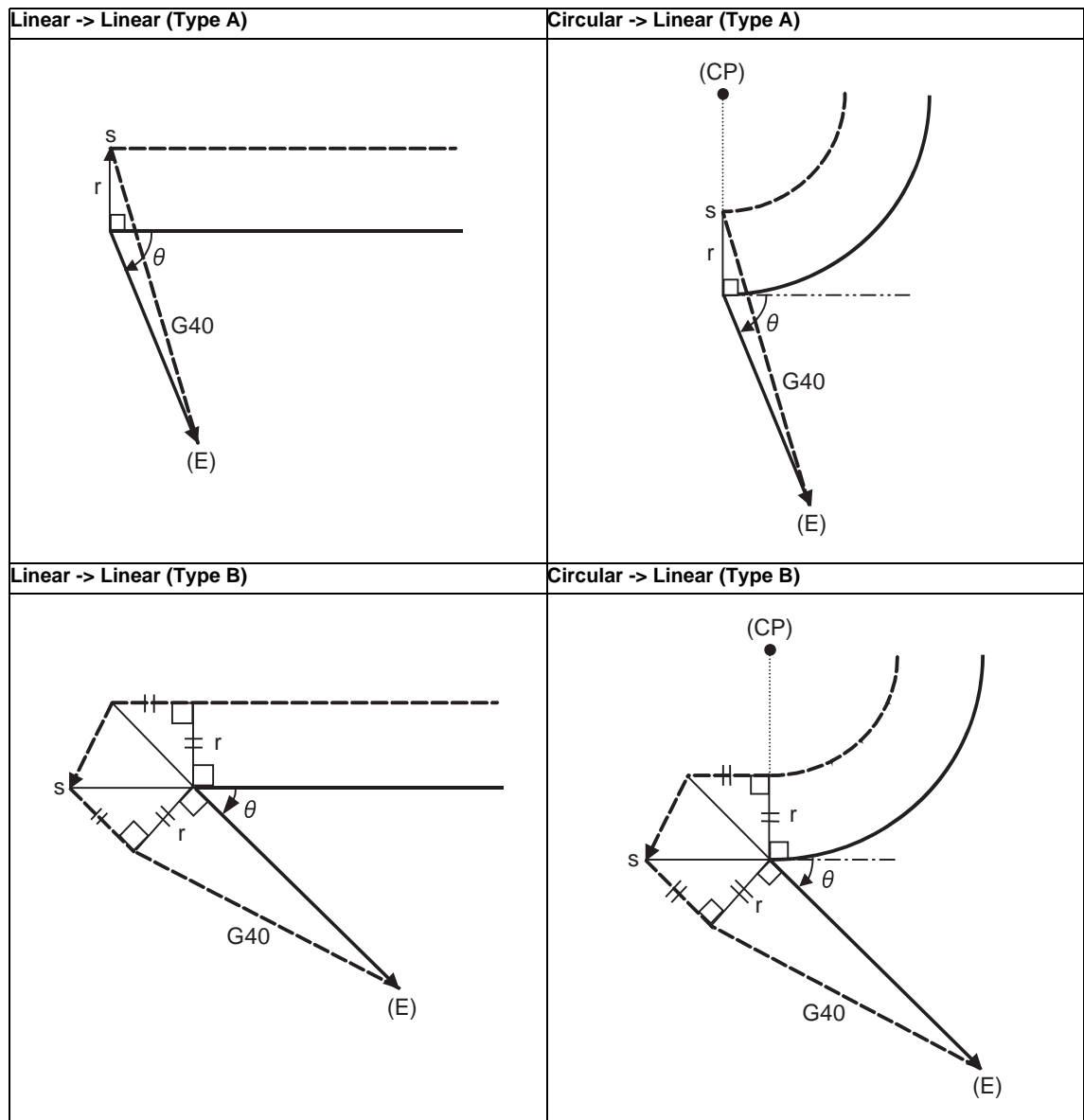
(c)-1 When an independent G40 command is issued at an outside corner (acute angle)

	Type A	Type B
<pre> : N1 G01 X_Z_F_; N2 G00 X_Z_; N3 G40; : </pre>		
<pre> : N1 G01 X_Z_F_; N2 G01 X_Z_; N3 G40; N4 M05; N5 G00 X_Z_; : </pre>		
<pre> : N1 G01 X_Z_F_; N2 G00 X_Z_; N3 G01 X_Z_; N4 G40; N5 M05; N6 G01 X_Z_; : </pre>		
<pre> : N1 G01 X_Z_F_; N2 G00 X_Z_; N3 G00 X_Z_; N4 G40; : </pre>		

— Program path

--- Nose R center path

(c)-2 When a G40 command is issued at an outside corner (acute angle), in the same block as a movement command



(E) End point

(CP) Center of circular

r : Nose R radius

s : Stop point with single block

— Program path

--- Nose R center path

12.4.3 Other Operations during Tool Nose Radius Compensation



Detailed description

Changing the compensation direction during tool nose radius compensation

The compensation direction is determined by the tool nose radius compensation commands (G41, G42).

G code	Compensation direction
G41	Left-side compensation
G42	Right-side compensation

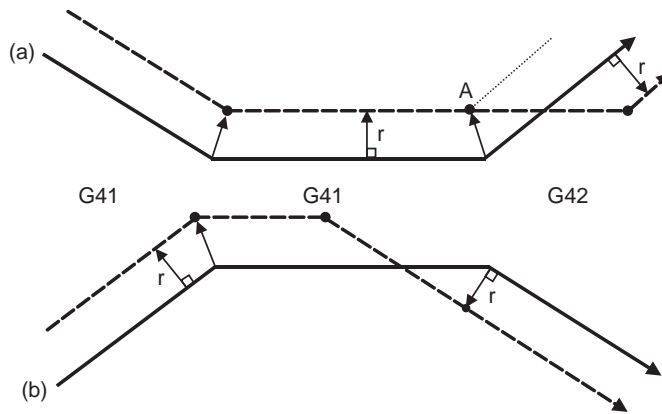
The compensation direction can be changed by changing the compensation command during the compensation mode without canceling the mode.

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

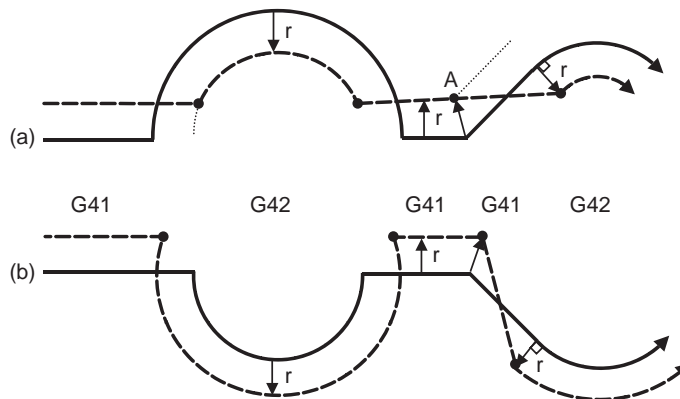
(b) When there is no intersection at the change of compensation direction



(2) Linear <--> Circular

(a) When there is an intersection (A) at the change of compensation direction

(b) When there is no intersection at the change of compensation direction



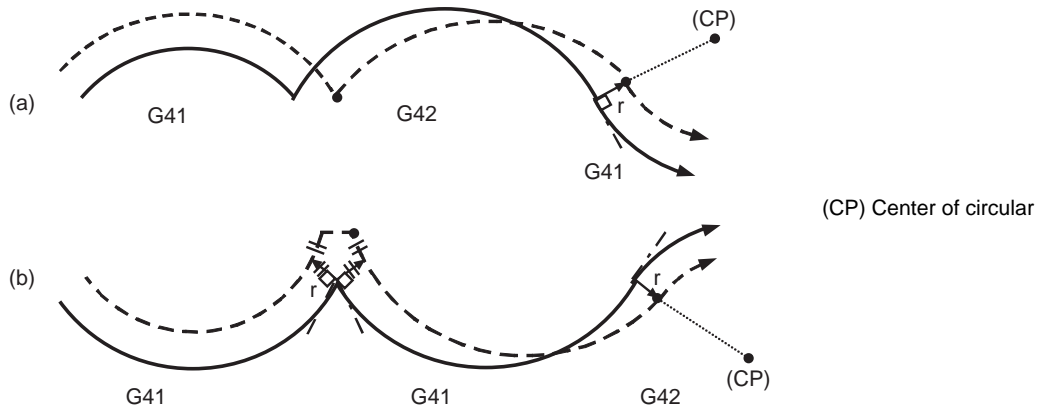
— Program path

- - - - - Nose R center path

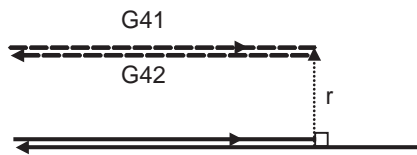
(3) Circular -> Circular

(a) When there is an intersection at the change of compensation direction

(b) When there is no intersection at the change of compensation direction



(4) Linear return

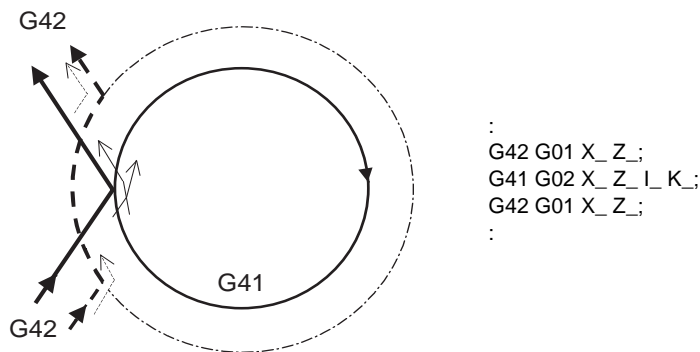


(5) Arc exceeding 360° due to compensation method

In the cases below, it is possible that the arc may exceed 360°.

a. With compensation direction selection based on G41/G42

If the arc exceeds 360°, compensation will be performed as shown in the figure and uncut section will be left.



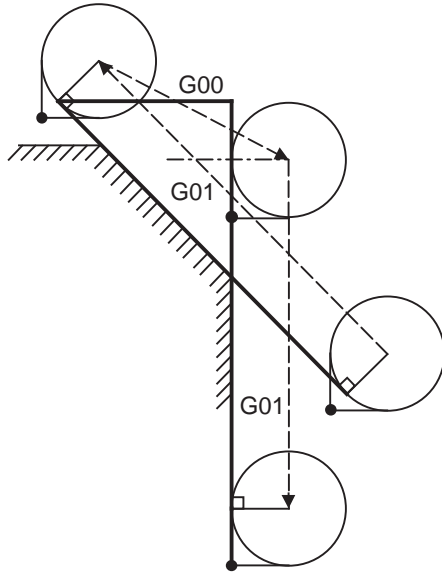
— Program path

--- Nose R center path

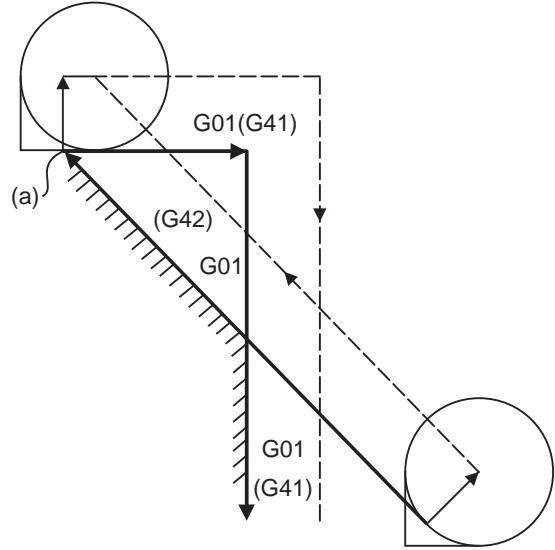
- - - - - Uncut section

Tool nose radius compensation of path closed by G46/G41/G42

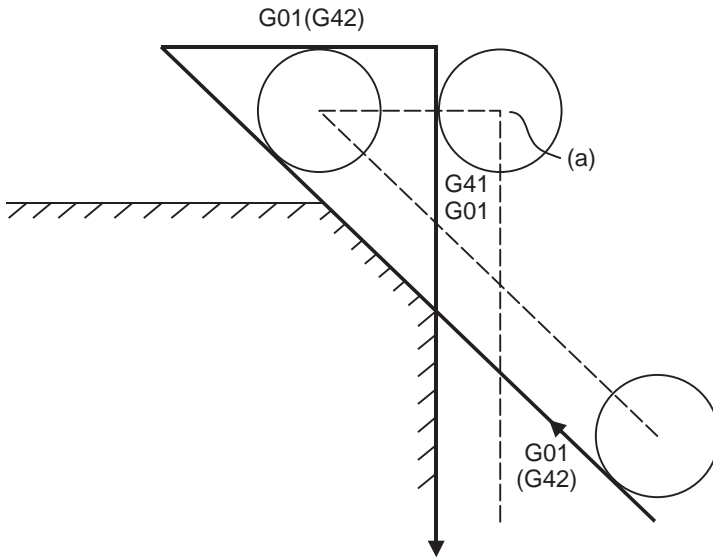
(1) G46 command operation



(2) G42 -> G41 command operation (When commanding G41 at (a))



(3) G42 -> G41 command operation (When commanding G41 at (a))



— Program path

--- Nose R center path

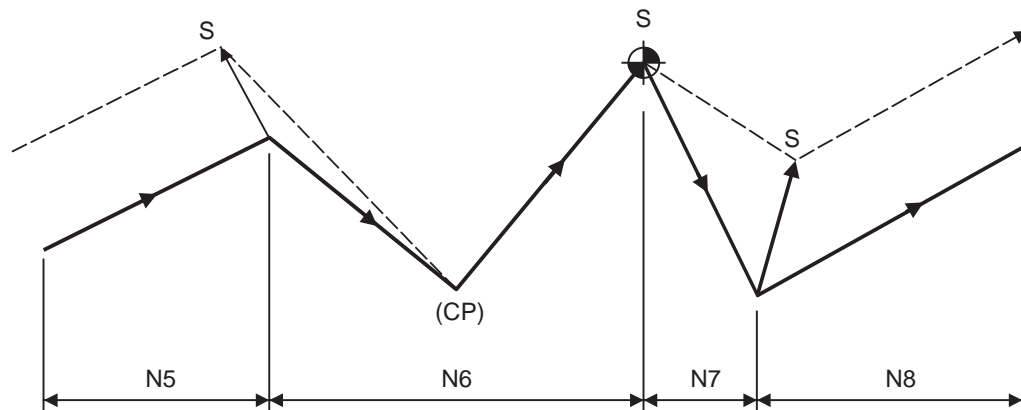
Command for eliminating compensation vectors temporarily

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 point return command

Compensation vector temporarily becomes 0 at the intermediate point (Reference point if there is no intermediate point).



(G41) :

N5 G01 U 30. W 60. ;

N6 G28 U-40. W 50. ;

N7 U-60. W 30. ;

N8 U 40. W 70. ;

:

(CP) Intermediate Point

(2) The compensation vector will be eliminated temporarily with the G53 command (Basic machine coordinate system selection).

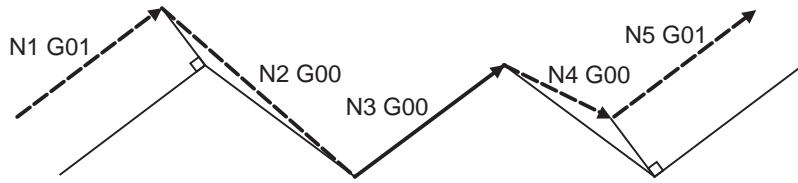
(Note 1) The compensation vectors do not change with the coordinate system setting (G92) command.

(3) Positioning (G00) commands

Tool nose radius compensation is temporarily canceled with G00 commands.

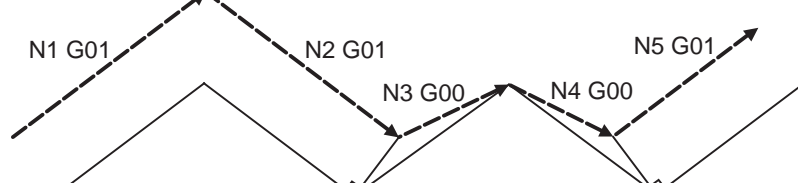
```

:
N1 G01 X_Z_F_;
N2 G00 X_Z_;
N3 G00 X_Z_;
N4 G00 X_Z_;
N5 G01 X_Z_;
:
    
```



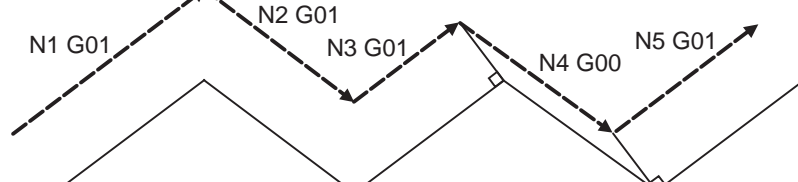
```

:
N1 G01 X_Z_F_;
N2 G01 X_Z_;
N3 G00 X_Z_;
N4 G00 X_Z_;
N5 G01 X_Z_;
:
    
```



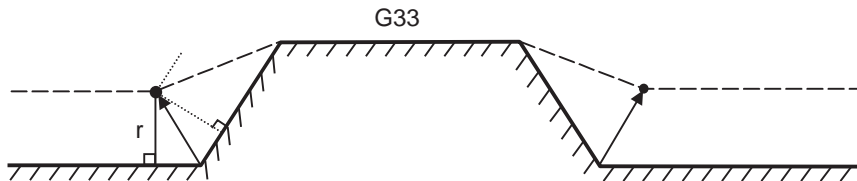
```

:
N1 G01 X_Z_F_;
N2 G01 X_Z_;
N3 G01 X_Z_;
N4 G00 X_Z_;
N5 G01 X_Z_;
:
    
```



(4) G33 thread cutting command

Tool nose radius compensation will not be applied to the G33 block.



(5) Compound type fixed cycle for turning machining

When a compound type fixed cycle for turning machining I command (G70, G71, G72, G73) is issued, the tool nose radius compensation will temporarily be canceled.

The finished shape to which tool nose radius compensation has been applied is cut with the compensation cancel state, and upon completion, operation will automatically return to the compensation mode.

Blocks without movement

The following blocks are known as blocks without movement.

M03 ;	M command
S12 ;	S command
T0101 ;	T command
G04X500 ;	Dwell
G10P01R50 ;	Compensation amount setting
G92X600. Z500. ;	Coordinate system setting
Y40. ;	Movement outside the compensation plane
G00 ;	G code only
U0 ;	Zero movement amount

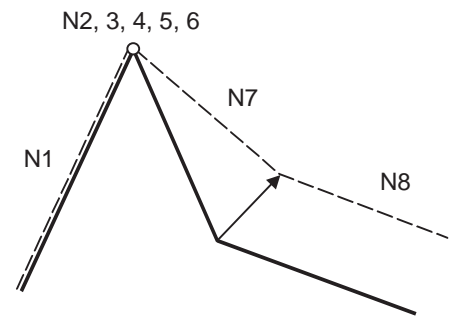
- (1) When commanded at the compensation start
Compensation vector cannot be created when there are four or more successive blocks without movement, or when pre-read prohibiting M command is issued.

```

N1 U60.W30.T0101 ;
N2 G41 ;
N3 G4 X1000 ;
N4 F100 ;
N5 S500 ;
N6 M3 ;
N7 U- 50.W20. ;
N8 U- 20.W50. ;

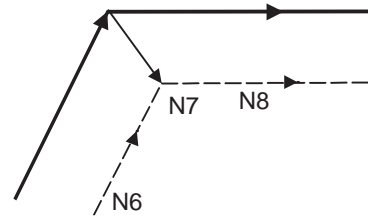
```

} Block without movement



- (2) When command is issued during 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 U200. W100. ;
N7 G04X P1000 ;    ... Block without movement
N8 W200. ;
```



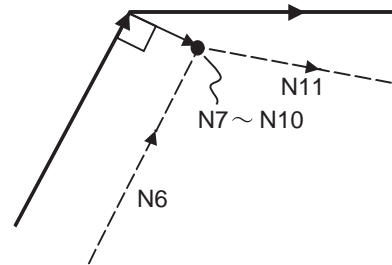
Block N7 is executed at N7 in the figure.

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.

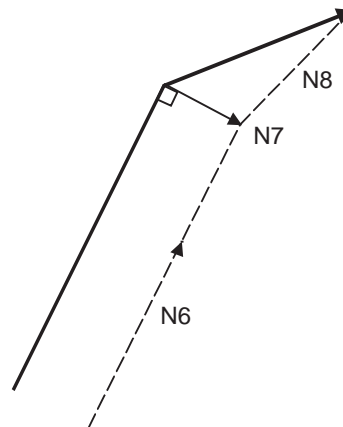
```
N6 U200. W100. ;
N7 G4 X1000 ;
N8 F100 ;
N9 S500 ;
N10 M4 ;
N11 W100. ;
```

} Block without movement



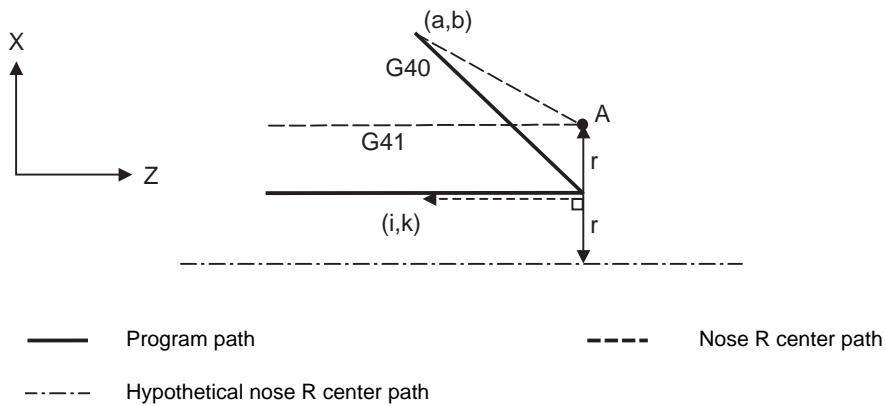
- (3) When commanded together with compensation cancel
 Only the compensation vectors are canceled when a block without movement is commanded together with the G40 command.

```
N6 U200. W100. ;
N7 G40 M5 ;
N8 U50. W100. ;
```



When I, J, K are commanded in G40

A perpendicular vector is created in the block before G40.

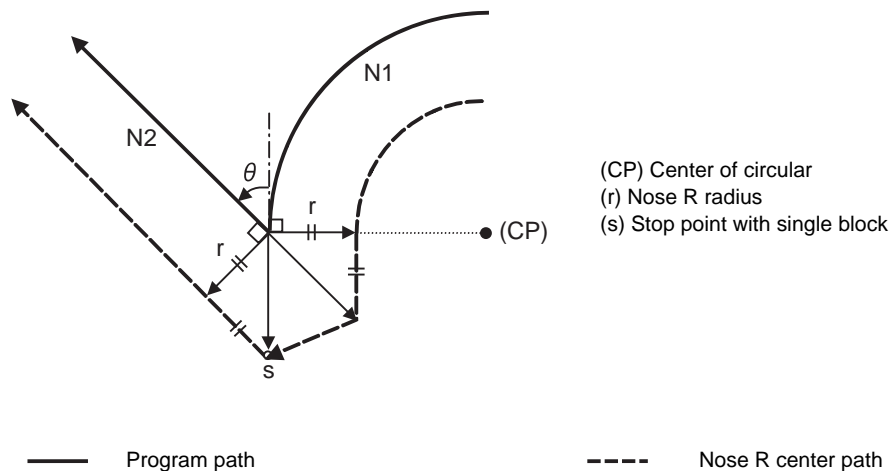


Corner movement

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 those vectors. This action is called corner movement.

When the vectors do not coincide, the tool moves in order to machine the corner.

In the single block mode operation, the previous block and corner movement are executed in a single block and the remaining movement and following block are executed in a single block in the next operation.



12.4.4 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

G18 (Z-X plane) G41/G42 X__ Z__ I__ K__ ;

Assign a linear command (G00, G01) in a movement mode.



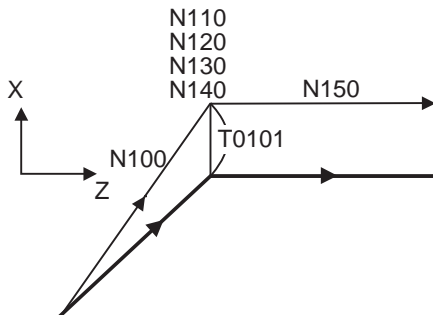
Detailed description

I, K type vectors (G18 X-Z plane selection)

This section describes the new I, K type vectors (G18 plane) created by this command. (Similar descriptions apply to vector I, J for the G17 plane and to J, K for the G19 plane.)

As shown in the following figures, I, K type vectors create compensation vectors which are perpendicular to the direction designated by I, K and equivalent to the compensation amount, without the intersection point calculation of the programmed path. The I, K 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, K is commanded at compensation start

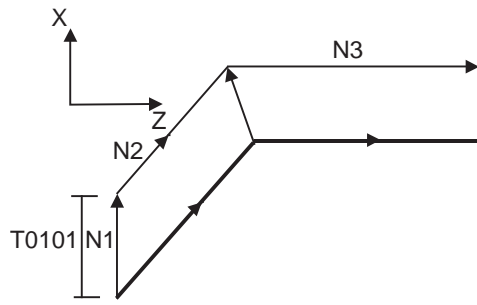


```

(G40)
:
N100 G41 U100. W100.K150.T0101 ;
N110 G04 X1000 ;
N120 G01 F1000 ;
N130 S500 ;
N140 M03 ;
N150 Z150. ;
:
    
```

- ← Program path
- ← Nose R center path

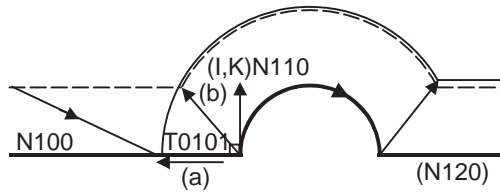
(2) When there are no movement commands at the compensation start.



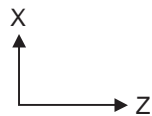
```
(G40)
:
N1 G41 K150. T0101 ;
N2 U100. W100. ;
N3 W150. ;
:
```

← Program path
 ← Nose R center path

(3) When I, K has been commanded in the mode (G18 plane)



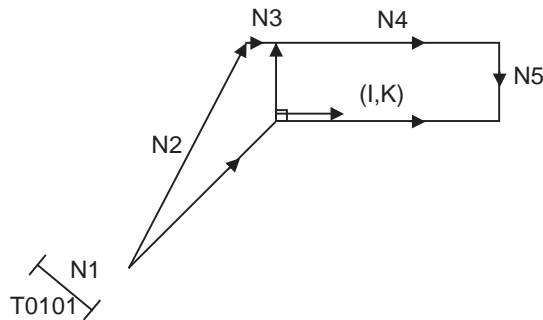
```
:
(G18 G41)
N100 G41 T0101 G01 W150. I50. ;
N110 G02 W100. I50. ;
N120 G01 W100. ;
:
```



(a) I, K type vector (b) Intersection point calculation type vector

← Program path
 ← Nose R center path
 - - - - - Path after intersection point calculation

(4) When I, K has been commanded in a block without movement



```
N1 G41 T0101 G01 F1000 ;
N2 U100. W100. ;
N3 G41 K50. ;
N4 W150. ;
N5 G40 ;
```


Offset vector direction

(1) In G41 mode

Direction produced by rotating the direction commanded by I, K by 90° to the left when looking at the zero point from the forward direction of the Y axis (3rd axis).

(Example 1) With K100.		(Example 2) With K-100.	
	(0, 100) IK direction		(0, -100) IK direction
	Offset vector direction		Offset vector direction

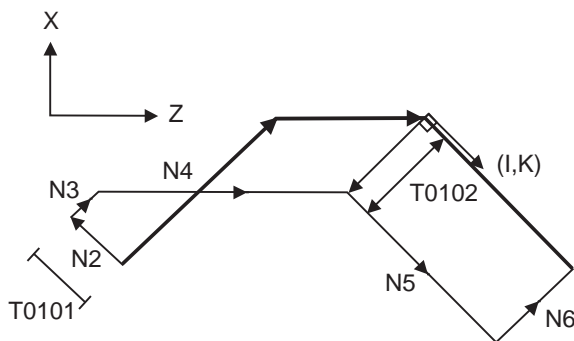
(2) In G42 mode

Direction produced by rotating the direction commanded by I, K by 90° to the right when looking at the zero point from the forward direction of the Y axis (3rd axis).

(Example 1) With K100.		(Example 2) With K-100.	
	(0, 100) IK direction		(0, -100) IK direction
	Offset vector direction		Offset vector direction

Selection of offset modal

G41 and G42 modals can be switched over at any time.



```

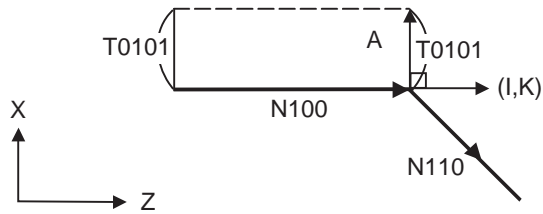
N1 G28 X0Z0 ;
N2 G41 T0101 F1000 ;
N3 G01 U100. W100. ;
N4 G42 W100. I - 100. K100.T0102 ;
N5 U- 100. W100. ;
N6 G40 ;
N7 M02 ;
%
```

Compensation amount for offset vectors

The compensation amount is determined by the offset No. (modal) in a block with the I, K designation.

<Example 1>

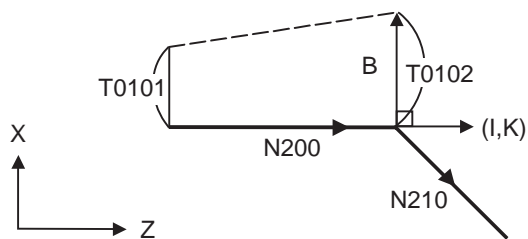
Vector A is the compensation amount registered in tool offset No. modal 1 of the N100 block.



```
(G41 T0101)
:
N100 G41 W150. K50. ;
N110 U- 100. W100. ;
:
```

<Example 2>

Vector B is the compensation amount registered in tool offset No. modal 2 of the N200 block.

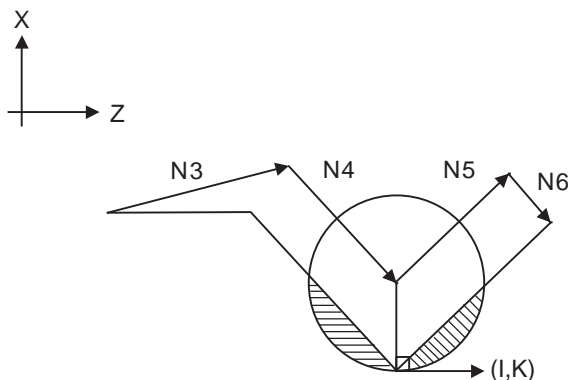


```
(G41 T0101)
:
N200 G41 W150. K50. T0102 ;
N210 U - 100. W100. ;
:
```



Precautions

- (1) Issue the I, K type vector in a linear mode (G0, G1). If it is in an arc mode at the start of compensation, program error (P151) will occur.
When it is in the offset mode as well as in the arc mode, I, K will be designated at the center of the circular.
- (2) When the I, K 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 X0Z0 ;
N2 G41 T0101 F1000 ;
N3 W100. ;
N4 G41 U-100. W100. K10. ;
N5 U100. W100. ;
N6 G40 ;
N7 M02 ;
    
```

- (3) 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/G42	I,K (J)	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, K type vector No insertion block

12.4.5 Interrupts during Tool Nose Radius Compensation



Detailed description

MDI interruption

Tool nose radius compensation is valid in any automatic operation mode - whether tape, memory or MDI mode.

The figure below shows what happens by MDI interruption after stopping the block during tape or memory mode.

S in the figure indicates the stop position with single block.

(1) Interrupt without movement (tool path does not change)

Automatic operation

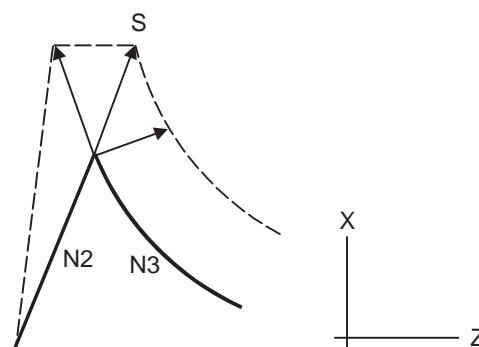
N1 G41 T0101;

N2 U50. W20. ;

N3 G3 U-40. W40. R70. ;

MDI interruption

<--- S1000 M3;



(2) Interrupt with movement

The compensation vectors are automatically re-calculated in the movement block after interrupt.

With linear interrupt

Automatic operation

N1 G41 T0101;

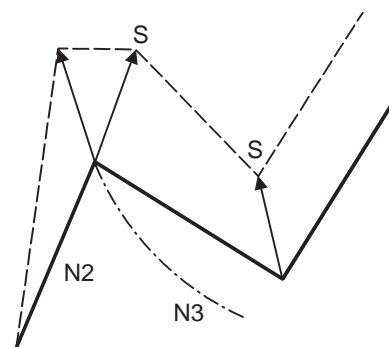
N2 U50. W20. ;

N3 G3 U-40. W40. R70. ;

MDI interruption

<--- U-30. W50. ;

U50. W30. ;



With circular interruption

Automatic operation

N1 G41 T0101;

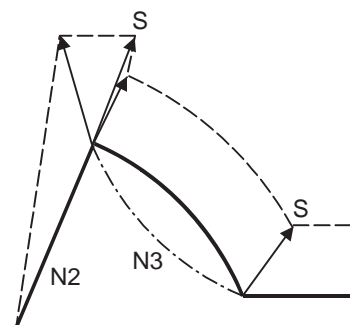
N2 U50. W20. ;

N3 G3 U-40. W40. R70. ;

MDI interruption

<--- G2 U-40. W40. R70. ;

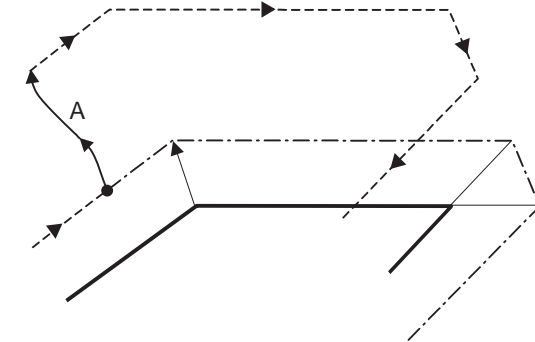
G1 W40. ;



Manual interruption

(1) Interrupt with manual absolute OFF.

The tool path will deviate from the compensated path by the interrupt amount.

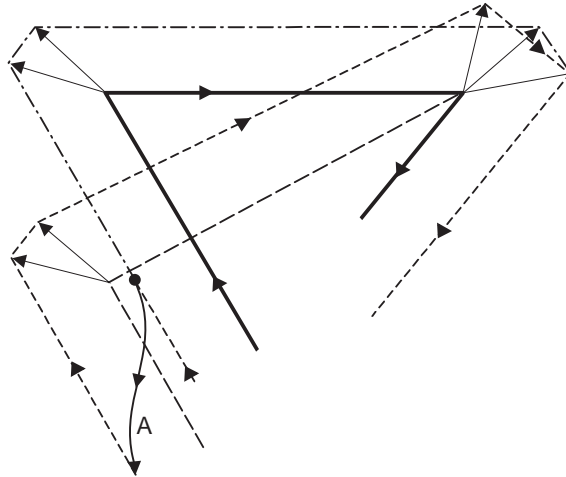


- Program path
- - - - - Tool path after compensation
- ← Interrupt (A)
- ← - - - - Tool path after interrupt

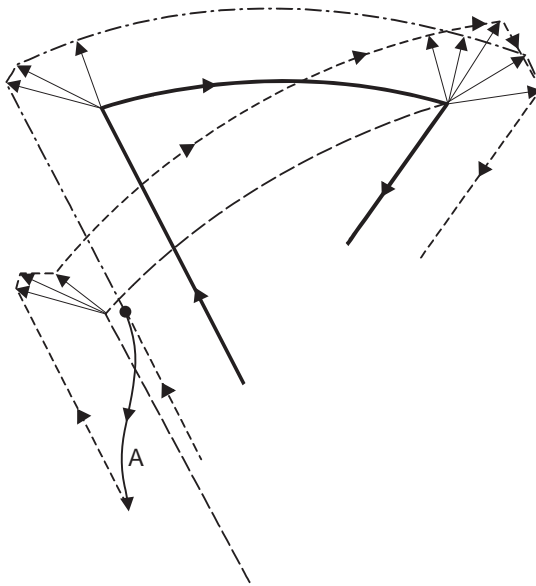
(2) Interrupt with manual absolute ON.

In the incremental value mode, the same operation will be performed as the manual absolute OFF.
 In the absolute value mode, however, the tool returns to its original path at the end point of the block following the interrupted block, as shown in the figure.

[Line-Line-Line]



[Line-arc-Line]



- Program path
- Tool path after compensation
- ← Interrupt (A)
- ←----- Tool path after interrupt

12.4.6 General Precautions for Tool Nose Radius Compensation



Precautions

Assigning the compensation amounts

- (1) The compensation amount is normally assigned by designating the No. of the compensation amount by the last 1 or 2 digits of the T code. Depending on the machine specifications, the high-order digits may be used. The T code will remain valid once designated until another T code is subsequently commanded. Besides being used to designate the compensation amounts for tool nose radius compensation, the T codes are also used to designate the compensation amounts for tool length 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.

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 ("P112" when a plane different from the one used during the compensation is commanded)
 - G31 (P608)
 - G74,G75,G76 (P155)
 - G81 to G89 (P155)
- (2) A program error (P158) will occur when a tool nose point other than 1 to 8 is designated during the G46 mode.
- (3) A program error (P156) will occur when the compensation direction is not determined by the movement vector of the initial cutting command even when the tool nose radius compensation operation has started in the G46 mode and 5 blocks have been pre-read.
- (4) A program error (P151) will occur when a circular command is issued in the first or last block of the tool nose radius compensation.
- (5) A program error (P157) will occur when the compensation direction is reversed in the G46 mode. A parameter can be set to move the tool in the same compensation direction. (Control parameter "#8106 G46 NO REV-ERR")
- (6) A program error (P152) will occur during tool nose radius compensation when the intersection point of single block skip in the interference block processing, cannot be calculated.
- (7) A program error will occur when there is an error in one of the pre-read blocks during tool nose radius compensation.
- (8) A program error (P153) will occur when an interference occurs under no interference avoidance conditions during tool nose radius compensation.
- (9) A program error (P150) will occur when a tool nose radius compensation command is issued even though the tool nose radius compensation specification is not provided.

12.4.7 Interference Check



Function and purpose

A tool, whose tool nose has been compensated under the tool nose radius compensation function by the usual two-block pre-read, 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.

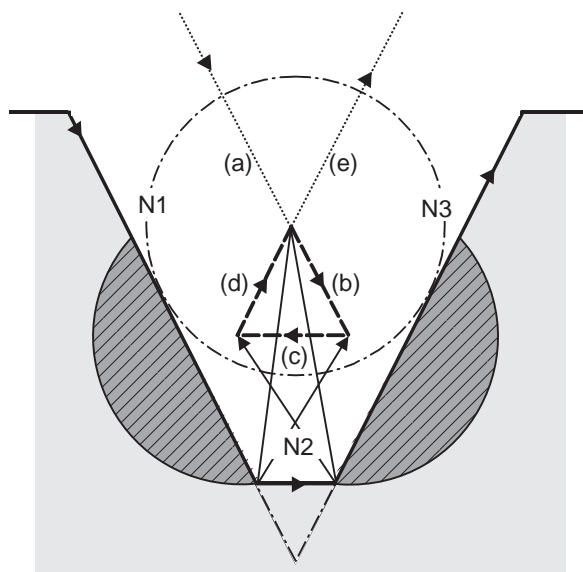
	Function	Parameter		Operation
		#8102 COLL. ALM OFF	#8103 COLL. CHK OFF	
(1)	Interference check alarm function	0	0	Operation stops with a program error(P153) before executing a block which will cause cutting.
(2)	Interference check avoidance function	1	0	The tool path is changed to prevent cutting from occurring. Operation stops with a program error(P153) when it is unable to change.
(3)	Interference check invalid function	0/1	1	Cutting proceeds unchanged even when cutting occurs. Use in the fine segment program.



Detailed description

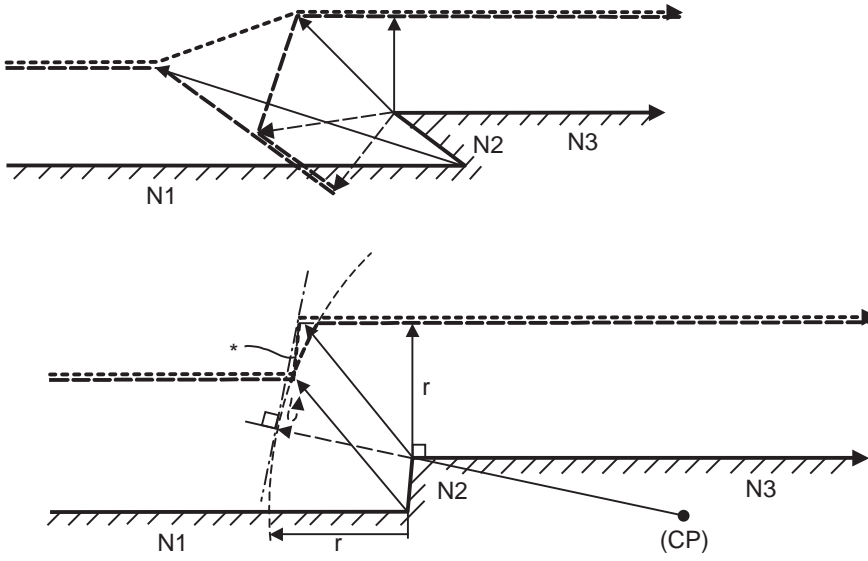
(Example)

```
(G41)
N1 G1 X100. Z50.;
N2 Z70;
N3 X300. Z120.;
```



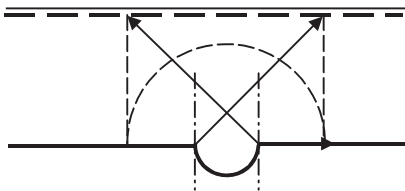
- (1) With alarm function
An alarm is given when N1 is executed. The buffer correction function can thus be used to change N1 to the following, enabling machining to continue: N1 G1 X-100. Z-20. ;
- (2) With avoidance function
The intersection of N1 and N3 is calculated to create interference avoidance vectors.
Tool nose R center path is (a) -> (e).
- (3) With interference check invalid function
The tool passes while cutting the N1 and N3 line.
Tool nose R center path is (a)->(b)->(c)->(d)->(e) .

Operation when interference avoidance function is valid



- ← Program path
- ← - - - - - Nose R center path when interference check is invalid
- ← - - - - - Tool center path when interference is avoided (*: Linear movement)
- ← Valid vector
- ← - - - - - Invalid vector

In the case of the figure below, the groove will be left uncut.

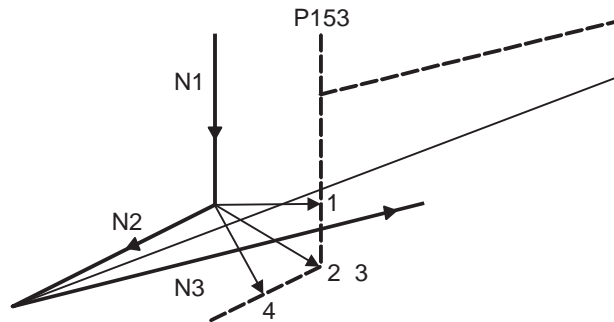


- ← Program path
- ← - - - - - Nose R center path
- ← - - - - - Tool center path when interference is avoided

Interference check alarm

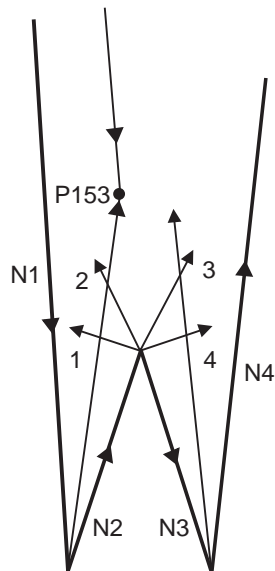
The interference check alarm occurs under the following conditions.

- (1) When the interference check alarm function has been selected
 When all the vectors at the end of its own block have been deleted.
 When, as shown in the figure below, 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.

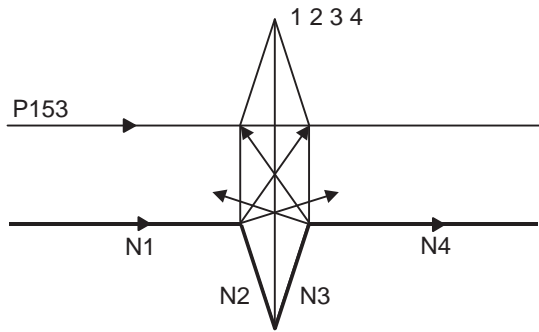


- (2) When the interference check avoidance function has been selected

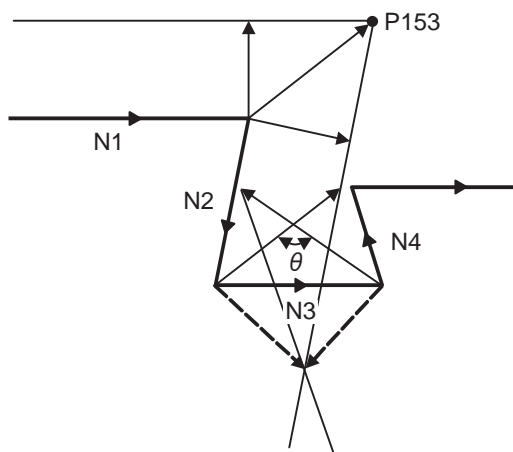
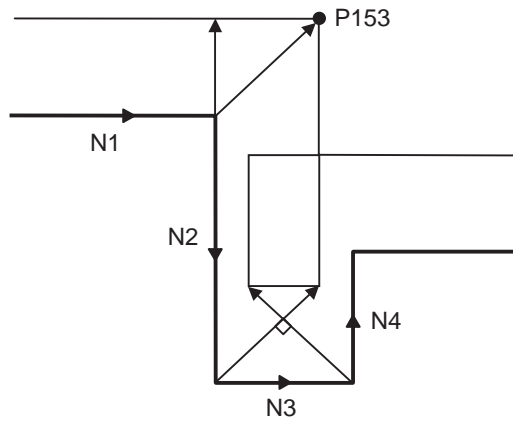
- (Ex. 1) When there are valid vectors at the end point of the following blocks even when all the vectors at the end point of its own block have been deleted.
 When, in the figure below, the N2 interference check is conducted, the N2 end point vectors are all deleted but the N3 end point vectors are regarded as valid.
 Program error (P153) now occurs at the N1 end point and the operation stops.



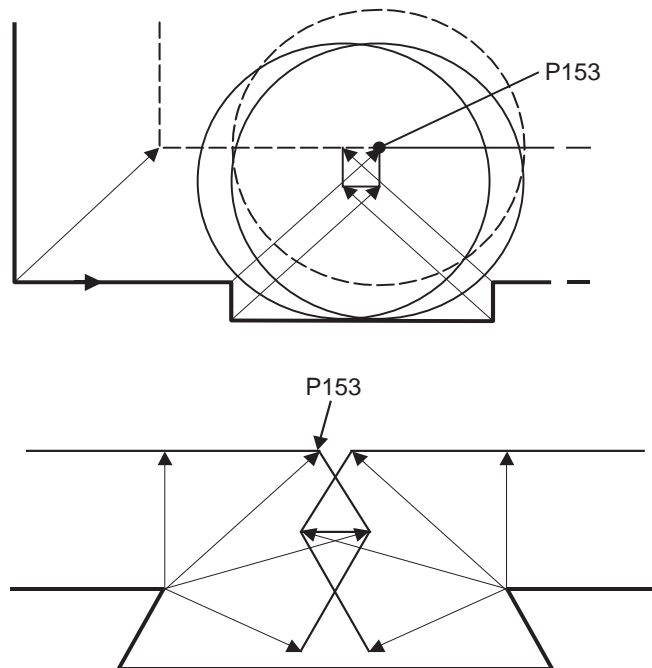
In the case shown in the figure below, the tool will move in the reverse direction at N2. Program error (P153) now occurs before executing N1 and the operation stops.



- (Ex. 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.

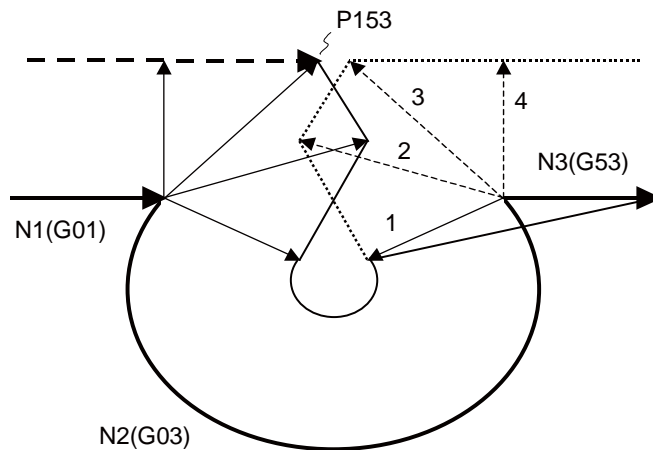


- (Ex. 3) When the program advance direction and the advance direction after compensation are reversed
When grooves, narrower than the tool nose diameter with parallel or widening bottom, are programmed, it will still be regarded as interference even if there is actually no interference.



(Ex. 4) When the end point vector at the end point of the most recent block to temporarily eliminate the compensation vector interferes
 Interference check will be executed even at the end point of the most recent block to eliminate the compensation vector temporarily in a similar way to check the interference at the end vector point when the compensation vector will not be eliminated. Therefore, it may be regarded as an interference even though there is 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 since the N3 is G53 command to temporarily eliminate compensation vectors. However, the interference check will still be conducted to vector 1 to 4 and an interference will be detected. Program error (P153) now occurs at the end point of the previous block and the operation stops.



- ← Program path
- ← - - - - - Nose R center path
- Nose R center path when the interference check is invalid
- Nose R center path when the interference check is invalid in N3 due to a command (G01 etc.) not to eliminate compensation vectors.
- ← Valid vector
- ← - - - - - Invalid vector (Invalid, however, subject to interference check)

12.5 Compensation Data Input by Program ; G10 L2/L10/L11, G11



Function and purpose

The amount of tool compensation and workpiece offset can be set or changed by the G10 command. When commanded with absolute value (X, Z, R), the compensation amounts serve as the new amounts; when commanded with incremental values (U, W, C), the new compensation amounts are equivalent to the commanded amounts plus the current compensation amount settings.



Command format

Workpiece offset input

G10 L2 P__ X__ (U__)Z__ (W__) ;

P	Compensation No.
X	X axis compensation amount (absolute)
U	X axis compensation amount (incremental)
Z	Z axis compensation amount (absolute)
W	Z axis compensation amount (incremental)

Tool length compensation input

G10 L10 P__ X__ (U__)Z__ (W__)R__ (C__)Q__ ;

P	Compensation No.
X	X axis compensation amount (absolute)
U	X axis compensation amount (incremental)
Z	Z axis compensation amount (absolute)
W	Z axis compensation amount (incremental)
R	Nose R compensation amount (absolute)
C	Nose R compensation amount (incremental)
Q	hypothetical tool nose point

Tool nose wear compensation input command

G10 L11 P__ X__ (U__)Z__ (W__)R__ (C__)Q__ ;

P	Compensation No.
X	X axis compensation amount (absolute)
U	X axis compensation amount (incremental)
Z	Z axis compensation amount (absolute)
W	Z axis compensation amount (incremental)
R	Nose R compensation amount (absolute)
C	Nose R compensation amount (incremental)
Q	Hypothetical tool nose point

When there is no L command with tool length compensation input (L10) or tool nose wear compensation input (L11).

Tool length compensation input command : P = 10000 + Compensation No.

Tool nose wear compensation input command : P = Compensation No.

Compensation input cancel

G11;



Detailed description

(1) The following table shows the compensation Nos. and the setting ranges of the hypothetical tool nose points.

Address	Meaning of address	Setting range		
		L2	L10	L11
P	Compensation No.	0: External workpiece offset	When L command is present 1 to Max. number of tool compensation sets	When L command is/is not present 1 to Max. number of tool compensation sets
		1: G54 workpiece offset		
		2: G55 workpiece offset	When L command is not present 10001 to 10000 + Max. number of tool compensation sets	
		3: G56 workpiece offset		
		4: G57 workpiece offset		
		5: G58 workpiece offset		
6: G59 workpiece offset				
Q	Hypothetical tool nose point		0 to 9	

(Note 1) The maximum number of tool compensation sets for P (compensation No.) with tool compensation input (L10 or L11) is up to a total of 80 with the addition of options.
(The number of sets will differ according to the model so check the specifications.)

(2) The setting range for the compensation amount is given below.
Program error (P35) occurs for any value not listed in the table after command unit conversion.
With an incremental value command, the setting range for the compensation amount is the sum of the present setting value and command value.

Setting	Tool length compensation amount		Wear compensation amount	
	Metric system	Inch system	Metric system	Inch system
#1004=B	± 9999.999 (mm)	± 999.9999 (inch)	± 9999.999 (mm)	± 999.9999 (inch)
#1004=C	± 9999.9999 (mm)	± 999.99999 (inch)	± 9999.9999 (mm)	± 999.99999 (inch)
#1004=D	± 9999.99999 (mm)	± 999.999999 (inch)	± 9999.99999 (mm)	± 999.999999 (inch)
#1004=E	± 9999.999999 (mm)	± 999.9999999 (inch)	± 9999.999999 (mm)	± 999.9999999 (inch)



Precautions

- (1) Compensation amount setting range check
The maximum value of the wear compensation amount and the maximum additional value for the wear compensation input check respectively take precedence for a single-time compensation amount in the maximum value and incremental value command of the wear compensation amount, and when an amount greater than these values has been commanded, program error (P35) will occur.
- (2) G10 is an unmodal command and is valid only in the commanded block.
- (3) Compensation input can be performed similarly for the 3rd axis but even when the C axis has been designated as the 3rd axis, address C is handled as an incremental command value of the nose R in the L10 or L11 command.
- (4) If an illegal L No. or compensation No. is commanded, the program errors (P172 and P170) will occur respectively.
- (5) When the P command is omitted, it will be handled as the currently selected workpiece offset input.
- (6) A program error (P35) will occur when the compensation amount exceeds the setting range.
- (7) X, Z and U, W are input together in a single block but when an address that commands the same compensation input (X,U or Z, W) is commanded, the address which is input last is valid.
- (8) Compensation will be input even if one address following G10L(2/10/11) P_ is commanded. A program error (P33) will occur when not even a single command has been assigned.

(Example) G10 L10 P3 Z50. ;

↓

[Tool length data]

#	Z
3	50.000

Input as per left.

- (9) Decimal point is valid for compensation amount.
- (10) G40 to G42 are ignored when they have been commanded in the same block as G10.
- (11) Do not command G10 in the same block as the fixed cycles and subprogram call commands. This will cause malfunctioning and program errors.
- (12) When T command has been issued in the same block as G10 with a parameter "#1100 Tmove" set to "0", the compensation will be accomplished in the following block.
- (13) For the multiple C axis system, both C axis workpiece offset are rewritten with the workpiece offset input.

12.6 Tool Life Management II ; G10 L3, G11



Function and purpose

Tool life management divides the tools being used into several groups, and manages the life (usage time, number of uses) of the tools in each group. When it comes to the end of life, a similar spare tool in the same group will be selected in order. This tool life management function with spare tools allows unmanned operation over a long time.

- (1) Number of life-managed tools
 1-part system : Maximum 80 tools
 Multi-part system : Maximum 40 tools/part system
- (2) Number of groups
 1-part system : Maximum 80 groups
 Multi-part system : Maximum 40 groups/part system
- (3) Group No.
 1 to 9999
- (4) Number of tools in one group
 Maximum 16 tools
- (5) Life time
 0 to 999999 minutes (approx. 16667 hours)
- (6) Number of lives
 0 to 999999 times

The tool life management data can be set from the NC program or from the Tool Life Management screen. Refer to the Instruction Manual for the method of setting from the Tool Life Management screen. When using the NC program, register the data with the same method as programmable compensation data input.



Command format

Life management data registration start

```
G10 L3 ;
P__ L__ N__ R__ ; (First group)
T__ ;
T__ ;
P__ L__ N__ ; (Next group)
T__ ;
T__ ;
```

P	Group No., (1 to 9999)
L	Life per tool (0 to 999999 minutes, or 0 to 999999 times)
N	Method (0:Time management, 1: Number of times management)
R	Remaining life (0 to 999999 min. or 0 to 999999 times) (Note) Only when the life prediction function is enabled ("#1277 ext13/bit1"="1"), the remaining life can be set with address R. The R command without the life prediction function causes the program error (P33).
T	Tool No. The spare tools are selected in the order of the tool Nos. registered here. (Tool No. 1 to 999999. Compensation No. 1 to 80) Tn follows the specifications.

End of life management data registration

```
G11 ;
```



Program example

(1) Format

: T □□□□ 99 ;	Start use of □□□□ group tool
: :	
T □□□□ 88 ; : :	Cancel □□□□ group tool compensation (Equivalent to T Δ Δ 00: Δ Δ is No. of the tool being used)
M02(M30) ;	End of machining program

(2) Actual example

: T0199 ;	Start use of group 01 tool
: :	
T0188 ; : :	Cancel group 01 tool compensation. If the No. of the tool being used is 17, this is equivalent to T1700.
T0609 ; : :	Selects tool No. 06 and compensation No. 09. * Life management is not carried out for tool 06.
T0600 ; : :	Cancel of group 06 tool compensation.
T0299 ; : :	Start use of group 02 tool
T0199 ; :	Start use of group 01 tool If the selected tool has several compensation Nos., the second compensation No. will be selected.



Operation example

Example of tool selection operation (When one tool has several compensation Nos.)

- (1) To use several compensation Nos. with one tool, select the next compensation No. for each T □□□□ 99 command.
- (2) If T□□□□99 is commanded for more times than the number of registered compensation Nos., the last compensation No. will be selected, and the operation will continue. (Refer to following.)

Registration to group 1	Program	Tool selection
T1701	T0199 ;	Equivalent to T1701
	:	
T1702	T0199 ;	Equivalent to T1702
	:	
T1703	T0199 ;	Equivalent to T1703
	:	
T2104	T0199 ;	Equivalent to T1703
	:	:
(Group 1)	:	:

(Hereafter, same until tool 17 reaches the end of life.)

- (3) If the above program is executed after resetting with M02/M30, or by resetting with external reset, the head compensation No will again be selected.



Precautions

- (1) The tool life data is registered by executing the above program in the memory or MDI mode.
- (2) When the above program is executed, all data (group No., tool No., life data) registered previously will be deleted. The registered data is held even if the power is turned OFF.
- (3) The group No. designated with P does not have to be consecutive, but it should be set in ascending order if possible. Because Nos will be displayed in ascending order on the screen, this will make monitoring easier. The group No. cannot be commanded in duplicate.
- (4) If the life data (L_) is omitted, the life data for that group will be "0". If N_ which specifies method is omitted, the method for that group will follow the basic specification parameter "#1106 Tcount".
- (5) If the remaining life data (R_) is omitted, the remaining life data for that group will be "0". When the remaining life data is "0", the "Life prediction" signal is not output for the designated tool. If N_ which specifies method is omitted, the method for that group will follow the basic specification parameter "#1106 Tcount". If the remaining life data (R_) is issued in larger value than life value (R_>L_), a program error (P35) will occur.
If the data L_ is omitted and the remaining life data (R_) is issued, a program error (P33) will occur.
- (6) Programming with a sequence No. is not possible between G10 L3 and G11.
- (7) If the usage data count valid signal (YC8A) is ON, G10 L3 cannot be commanded. (Program error(P177))

12.6.1 Counting the Tool Life



Function and purpose

The tool life can be counted with the time-count type or number of uses-count type. The count method and timing for the number of uses-count type can be changed to type 2 with the parameter setting (#1277 ext13/bit0).

If the usage data is equivalent to or more than the life data as a result of the count up, a spare tool in the group will be selected with the next group selection command (T □□□□ 99), and the newly selected tool will be counted.

When all tools in the group have reached their lives and a spare tool cannot be selected, the count will continue.



Detailed description

Counting the time of uses when the time-count type method is selected

In the cutting mode (G01, G02, G03, G31, G33, etc.), the length of time in which the tool is used is counted with 100ms units.

The time is not counted during dwell, machine lock, miscellaneous function lock or dry run. Whether to count or not during single block can be changed by setting the parameter.

- (Note)
- The maximum value for the life is 999999 minutes.
 - The data is displayed in a minute unit on the Tool Life Management screen.

When the number of uses-count type method is selected: Type 1 (#1277 ext13/bit0: 0)

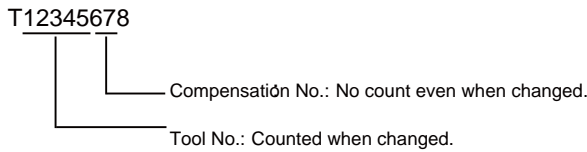
The number of uses is counted when the No. of the tool being used is changed with the tool selection command (T □□□□ 99) and when the program is in the cutting mode (excluding machine lock, miscellaneous function lock and dry run states).

The use is not counted if the cutting mode is not entered even once after the No. changes.

Whether to count or not during single block can be changed with the parameter setting.

- (Note) - The maximum value for the life is 999999 minutes.
- If only the compensation No. of the tool being used is changed, counting will not be conducted.

Example: When T code of tool being used is T12345678



<<Operation example 1>>

```

T0199          (1)
:
:
T0299
:
:
T0199          (2)
:
:
T0299
:
:
T0199          (3)
```

Group 01 has been used three times.

<<Operation example 2>>

```

T0199          (1)
:
:
T0299
:
:
T0199
```

Group 01 has been used once.

* The number of uses is for one program execution. If the program is executed again after resetting, it will be counted.

When the number of uses-count type method is selected: Type 2 (#1277 ext13/bit0: 1)

- (1) The groups, used for cutting between the start to reset of the machining program, is added with only "1". The count is made at the reset.
- (2) If recount M is commanded, the group used up to that point will be added with "1" to the counter.

(Note 1) A count is not made in the machine lock, miscellaneous function lock or dry run states.

(Note 2) During single block, select whether to count or not, with the parameter.

(Note 3) The maximum value for the life is 999999 times.

Re-selecting same tool group

When tool data of a tool with different compensation Nos. exists, the re-count M codes enable selecting the first compensation No.

Use the base specification parameter "#1108 TirectM" to set the re-count M code.

[Example of re-count M]

When the re-count M code parameter "#1108 TirectM" is "35":

(1) Tool selection when re-count M is not used:

Register to group 1	Program	Tool selection	
T1701	T0199 ;	Equivalent to T1701	
	⋮		
T1702	T0199 ;	Equivalent to T1702	
	⋮		
T1703	T0199 ;	Equivalent to T1703	
	⋮		
T2104	T0199 ;	Equivalent to T1703	
	⋮		
	T0199 ;	Equivalent to T1703	
	⋮		
	T0199 ;	Equivalent to T1703	
	⋮		
	T0199 ;	Equivalent to T1703	
	⋮		
(Group 1)		(Hereafter, the same until tool No. 17 reaches its life)	

(2) Tool selection when re-count M is used:

Register to group 1	Program	Tool selection	
T1701	T0199 ;	Equivalent to T1701	
	⋮		
T1702	T0199 ;	Equivalent to T1702	
	⋮		
T1703	T0199 ;	Equivalent to T1703	
	⋮		
T2104	M35 ;	Equivalent to T0188	→ Re-count M command (As MF is ON, FIN is returned)
	T0199 ;	Equivalent to T1701	
	⋮		
	T0199 ;	Equivalent to T1702	
	⋮		
	T0199 ;	Equivalent to T1703	
	⋮		
(Group 1)		(Hereafter, the same until tool No. 17 reaches its life)	

(Note 1) In the case of "number of uses-count type" life management, the usage data is counted up by one when cutting is carried out after re-count M code is commanded.

Therefore, in the case of the above example, count up is only once in a), but count up is made once again after M35 in b).

In the case of "time-count type" life management, the length of time during cutting is counted, thus the re-count M code command does not affect the usage data.

(Note 2) Among the settings of #1108 TirectM, the setting 0 disables re-count M code, and 1, 2 and 30 are not allowed to use.

Program Support Functions

13.1 Fixed Cycles 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
G77	Longitudinal cutting cycle
G78	Thread cutting cycle
G79	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.
The following G code cancel commands are available.

 G00, G01, G02, G03
 G09,
 G10,G11
 G27, G28, G29, G30
 G33,G34
 G37,
 G92,
 G52,G53
 G65,
- (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 (G77 to G79) 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.

13.1.1 Longitudinal Cutting Cycle ; G77



Function and purpose

The longitudinal cutting cycle performs continuous straight and taper cutting in the longitudinal direction.



Command format

Straight cutting

G77 X/U__ Z/W__ F__ ;

X/U	X axis end point coordinates
Z/W	Z axis end point coordinates
F	Feedrate

Taper cutting

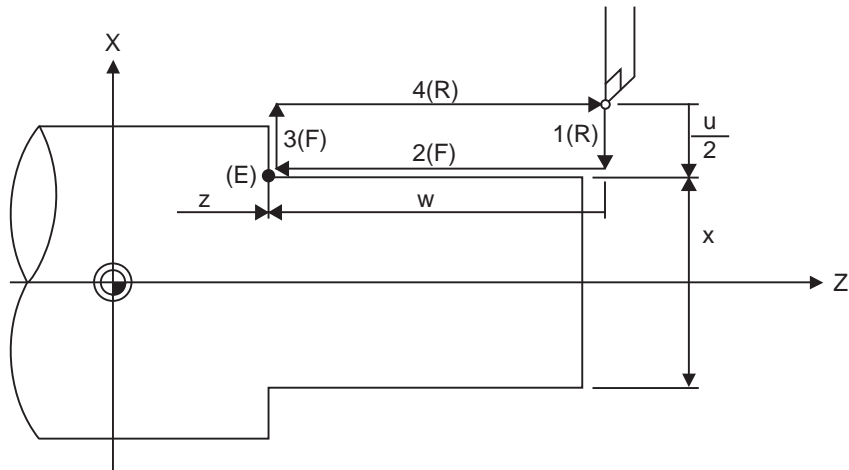
G77 X/U__ Z/W__ R__ F__ ;

X/U	X axis end point coordinates
Z/W	Z axis end point coordinates
R	Taper depth (radius designation, incremental value, sign required)
F	Feedrate



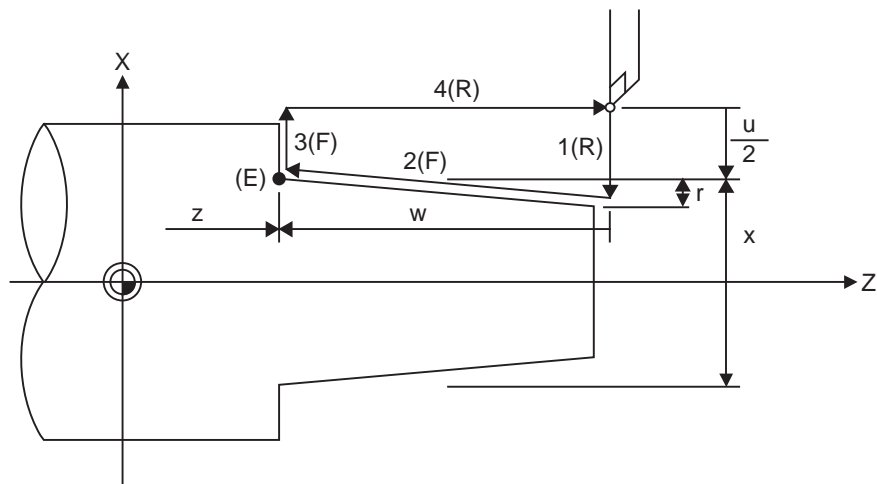
Detailed description

Straight cutting



- (R) Rapid traverse
- (F) Cutting feed
- (E) End point coordinates

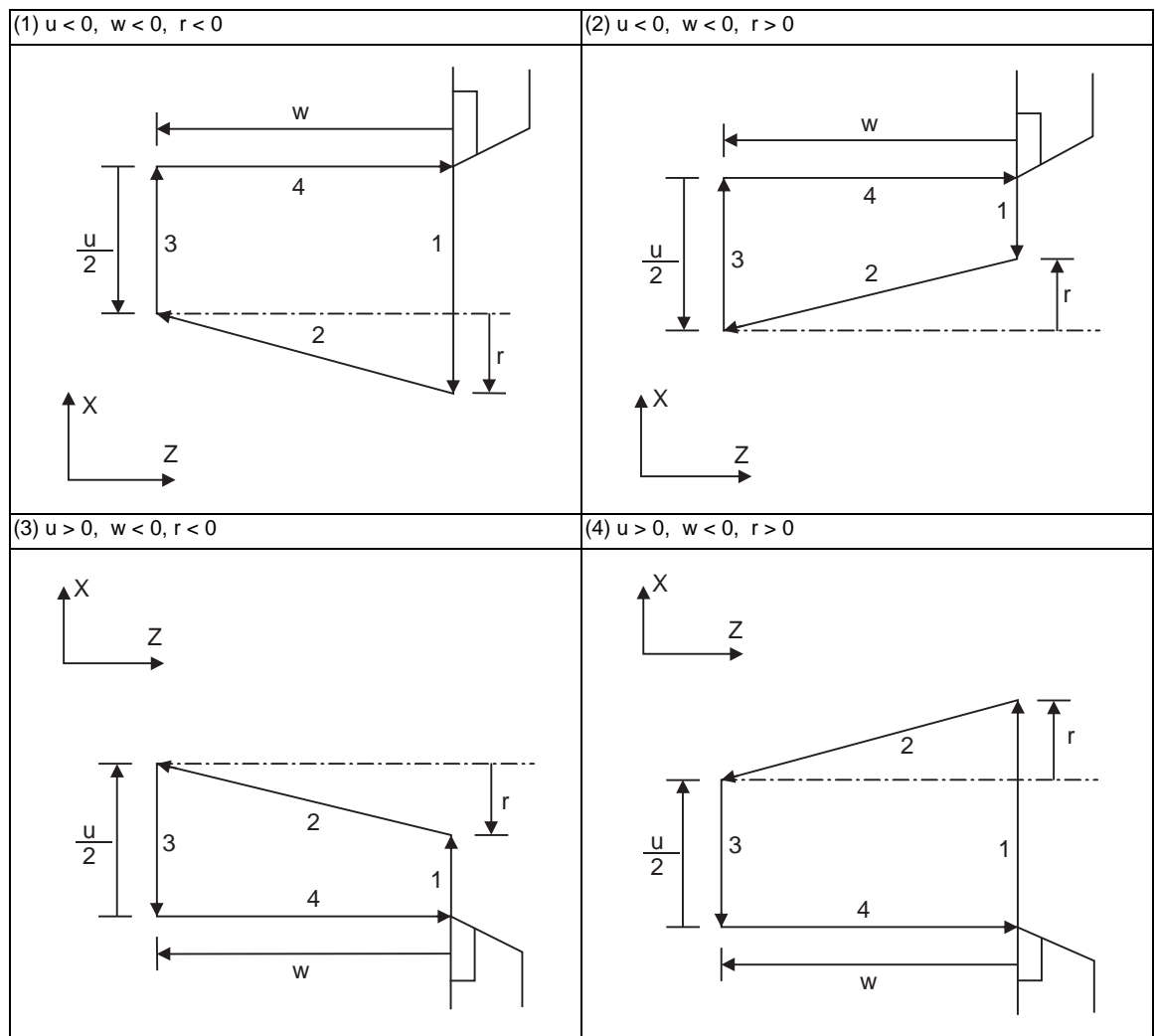
Taper cutting



- (R) Rapid traverse
- (F) Cutting feed
- (E) End point coordinates

With a single block, the tool stops at the end points of operations 1, 2, 3 and 4.

Depending on signs of u , w and r , the following shapes are created.



Program error (P191) will occur in (2) and (3) unless the following condition is satisfied.

$$|u/2| \geq |r|$$

13.1.2 Thread Cutting Cycle ; G78



Function and purpose

Thread cutting cycle is a fixed cycle which performs straight and taper thread cutting.
 (Note) The operation of thread cutting is same as the thread cutting command(G33).



Command format

Straight thread cutting

G78 X/U__ Z/W__ F/E__ Q__ ;

X/U	X axis end point coordinates
Z/W	Z axis end point coordinates
F/E	Lead of long axis (axis which moves most) direction
Q	Thread cutting start shift angle

Taper thread cutting

G78 X/U__ Z/W__ R__ F/E__ Q__ ;

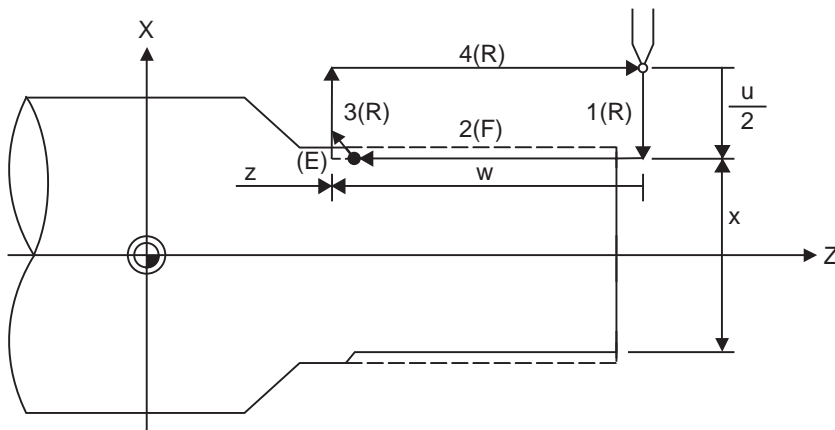
X/U	X axis end point coordinates
Z/W	Z axis end point coordinates
R	Taper depth (radius designation, incremental value, sign required)
F/E	Lead of long axis (axis which moves most) direction
Q	Thread cutting start shift angle



Detailed description

Straight thread cutting

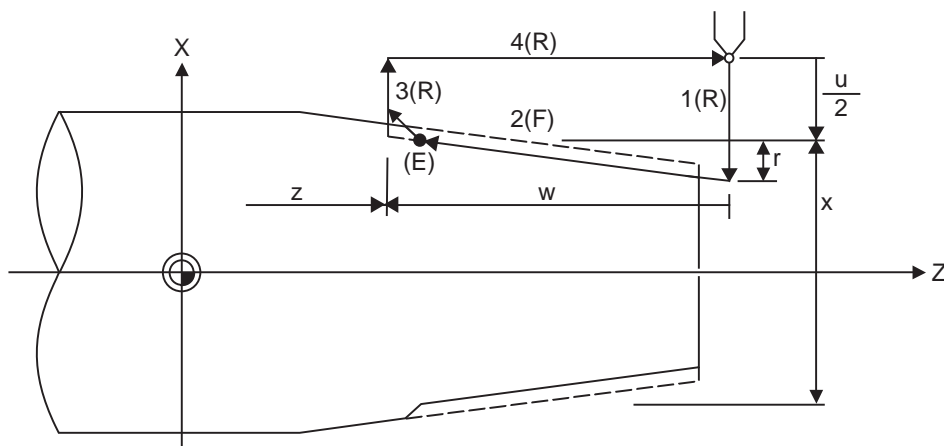
With a single block, the tool stops at the end points of operations 1, 3 and 4.



- (R) Rapid traverse
- (F) Thread cutting cycle
- (E) End point coordinates

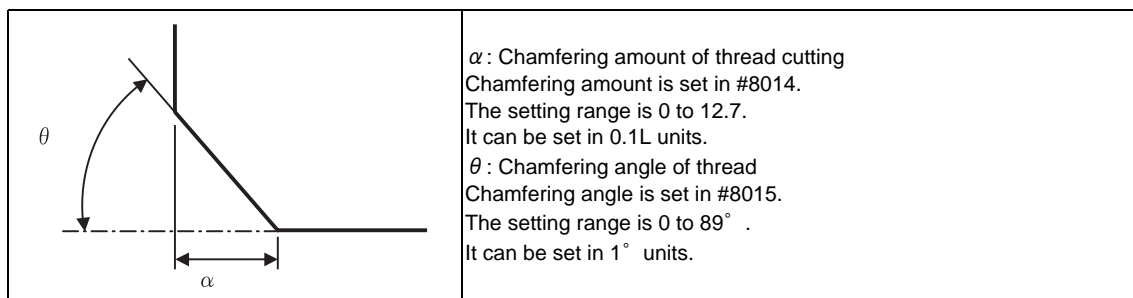
Taper thread cutting

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

(1) Details for chamfering



If chamfering amount of thread cutting up is bigger than a thread lead length, a program error(P192) will occur before starting thread cutting.

Set a parameter to select the operation in thread cutting cycle when a result of the chamfering cutting up exceeds the cycle start point.

<#1270 ext06/bit4 = 0>

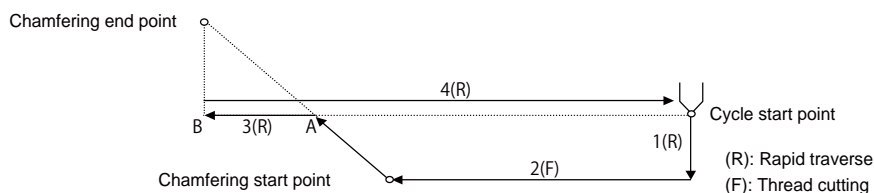
A program error(P192) will occur before thread cutting is started.

<#1270 ext06/bit4 = 1>

A program error(P192) will not occur.

After the cutting up 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.

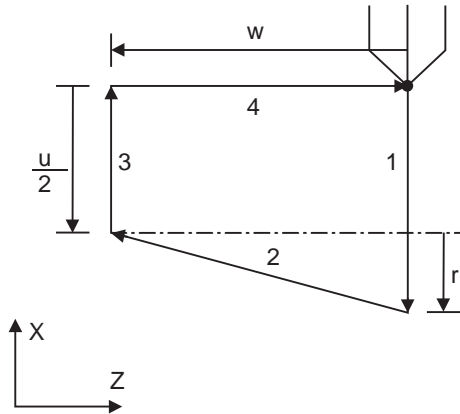
Cutting up operation when #1270 ext06/bit4 is set to "1"



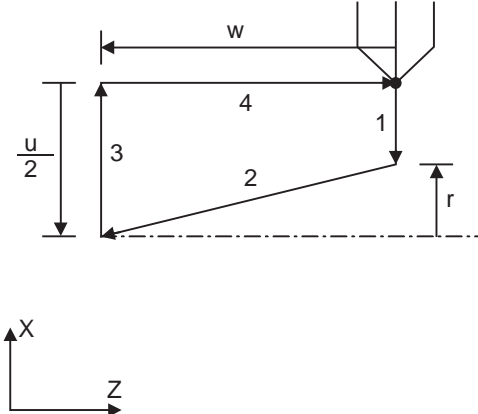
(R): Rapid traverse
(F): Thread cutting

- (2) When the feed hold function is applied during a thread cutting cycle, automatic operation will stop if it is applied when thread cutting is not being executed or when cutting command is issued but the axis is yet to move. If thread cutting is proceeding when the function is applied, the operation stops at the next movement completion position (completion of operation 3) of the thread cutting.
- (3) The dry run valid/invalid status does not change during thread cutting.
- (4) Depending on signs of u, w and r, the following shapes are created.

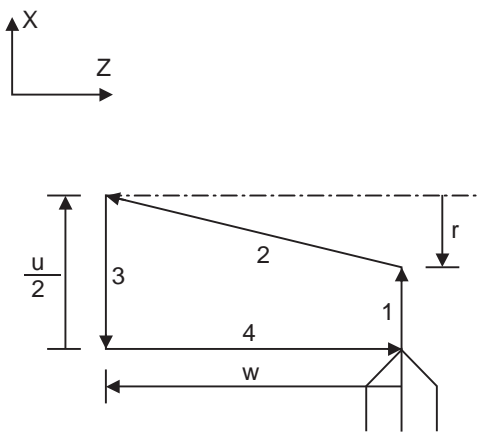
(a) $u < 0, w < 0, r < 0$



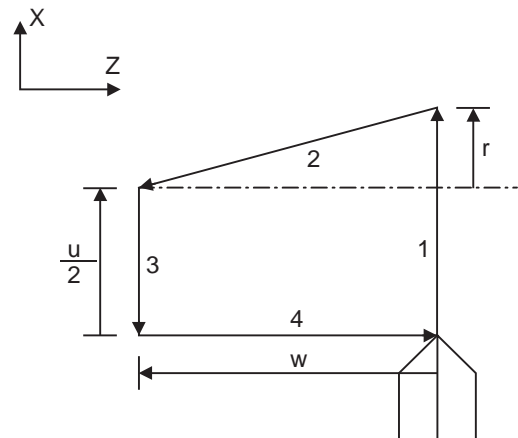
(b) $u < 0, w < 0, r > 0$



(c) $u > 0, w < 0, r < 0$



(d) $u > 0, w < 0, r > 0$



Program error (P191) will occur in (b) and (c) unless the following condition is satisfied.

$$|u/2| \geq |r|$$

- (Note 1) The thread cutting start shift angle is not modal. If there is no Q command with G78, this will be handled as "Q0".
- (Note 2) If a value exceeding 360.000 is command in G78 Q, this will be handled as "Q360.000".
- (Note 3) G78 cuts one row with one cycle. To cut two rows, change the Q value, and issue the same command.

13.1.3 Face Cutting Cycle ; G79



Function and purpose

The face cutting cycle performs continuous straight and taper cutting in the face direction.



Command format

Straight cutting

G79 X/U__ Z/W__ F__ ;

X/U	X axis end point coordinates
Z/W	Z axis end point coordinates
F	Feedrate

Taper cutting

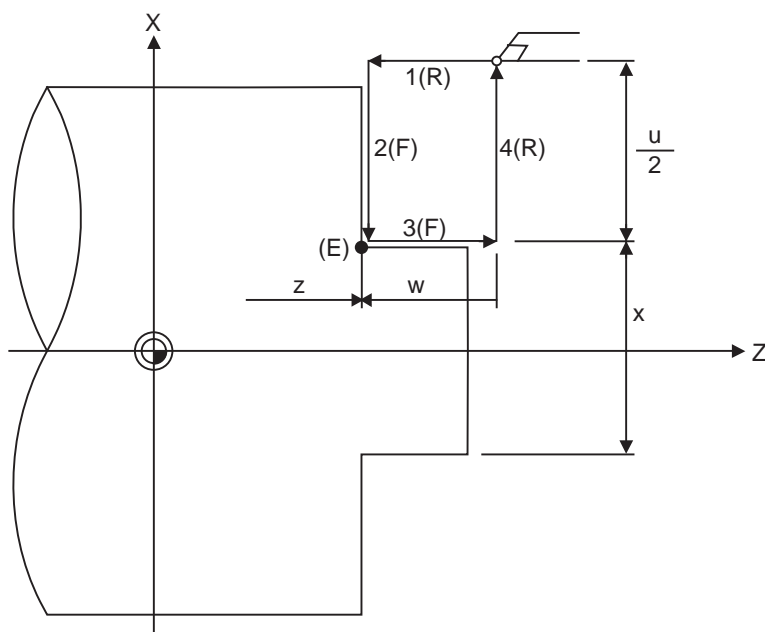
G79 X/U__ Z/W__ R__ F__ ;

X/U	X axis end point coordinates
Z/W	Z axis end point coordinates
R	Taper depth (radius designation, incremental value, sign required)
F	Feedrate



Detailed description

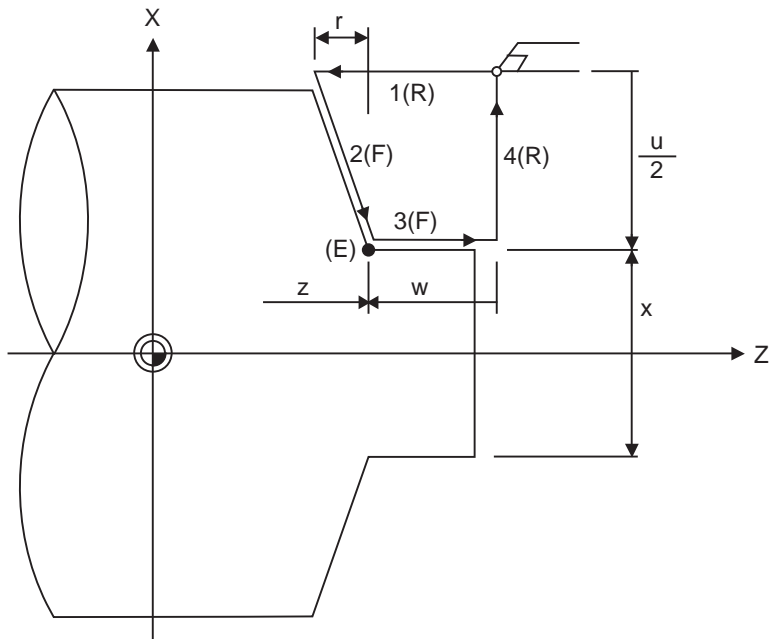
Straight cutting



(R) Rapid traverse
(E) End point coordinates

(F) Cutting feed

Taper cutting



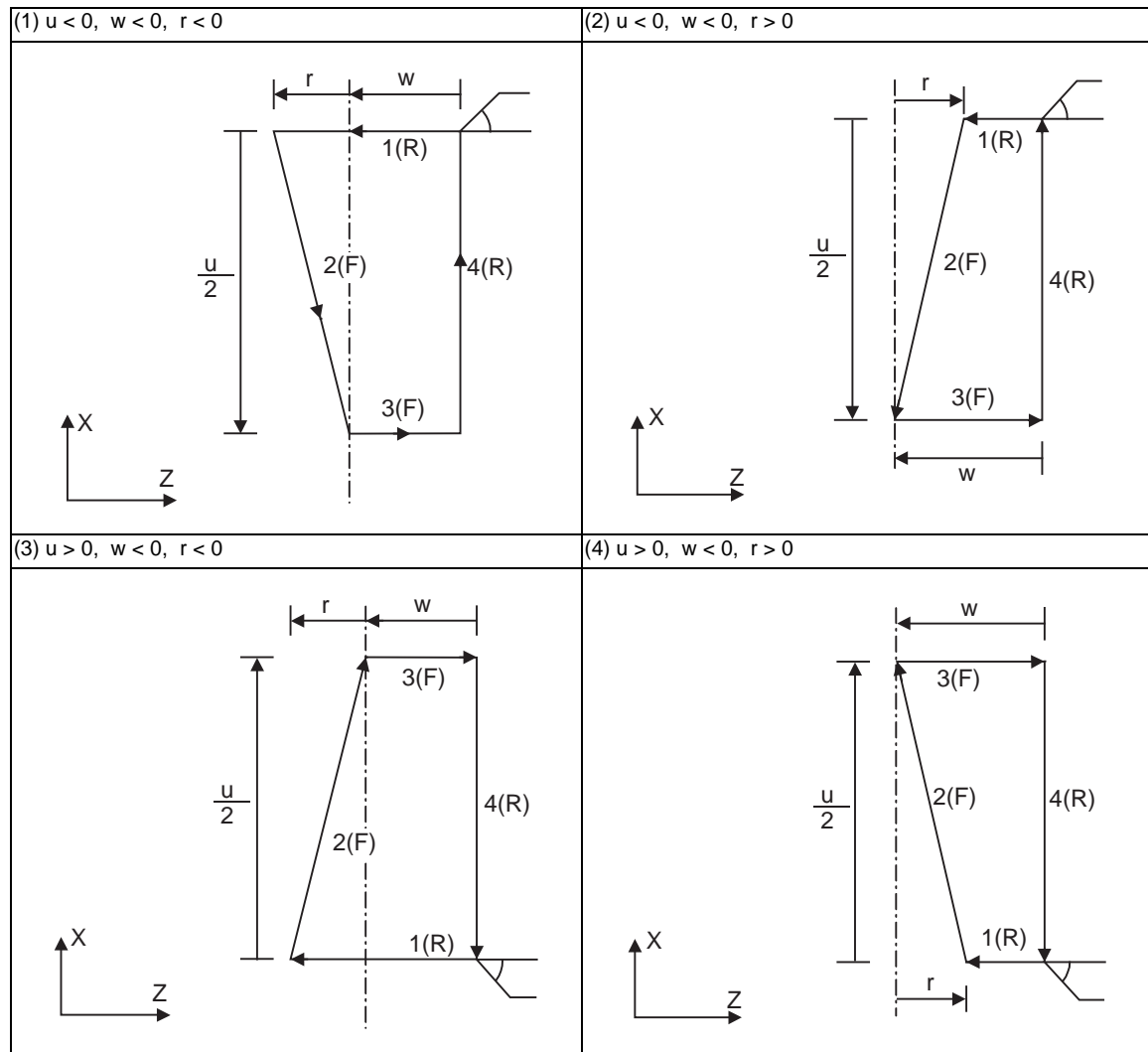
(R) Rapid traverse

(F) Cutting feed

(E) End point coordinates

With a single block, the tool stops at the end points of operations 1, 2, 3 and 4.

Depending on signs of u , w and r , the following shapes are created.



Program error (P191) will occur in (2) and (3) unless the following condition is satisfied.

$$|w| \geq |r|$$

13.2 Fixed Cycles for Turning Machining (MITSUBISHI CNC special format) ; G77,G78,G79



Function and purpose

When performing rough cutting and other cutting 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.

When using the MITSUBISHI CNC special format (#1265 ext01/bit1 ON), some addresses differ from the normal format. This section gives details on the formats different from the normal ones.

Refer to section "Fixed cycles for turning machining" for details on the fixed cycles for turning machining.



Command format

Longitudinal cutting cycle

G77 X/U__ Z/W__ I__ F__ ;

X/U	X axis end point coordinates
Z/W	Z axis end point coordinates
I	Taper depth (radius designation, incremental value, sign required)
F	Feedrate

Thread cutting cycle

G78 X/U__ Z/W__ I__ F/E__ Q__ ;

X/U	X axis end point coordinates
Z/W	Z axis end point coordinates
I	Taper depth (radius designation, incremental value, sign required)
F/E	Lead of long axis (axis which moves most) direction
Q	Thread cutting start shift angle

Face cutting cycle

G79 X/U__ Z/W__ K__ F__ ;

X/U	X axis end point coordinates
Z/W	Z axis end point coordinates
K	Taper depth (radius designation, incremental value, sign required)
F	Feedrate



Detailed description

(1) Comparison of MITSUBISHI CNC special format and normal format

Some addresses in the MITSUBISHI CNC special format differ from the normal format.

Function	MITSUBISHI CNC special format	Normal format	Difference from normal format
Longitudinal turning cycle	G77 X Z I F; or G77 U W I F;	G77 X Z R F; or G77 U W R F;	Taper depth Designate with R -> I
Thread cutting cycle	G78 X Z I F; or G78 U W I F;	G78 X Z R F; or G78 U W R F;	Taper depth Designate with R -> I
Face turning cycle	G79 X Z K F; or G79 U W K F;	G79 X Z R F; or G79 U W R F;	Taper depth Designate with R -> K

13.3 Compound Type Fixed Cycle for Turning Machining



Function and purpose

This function enables to perform a prepared fixed cycle by commanding a program in a block.

The types of fixed cycles are listed below.

G code	Function	
G70	Finishing cycle	Compound type fixed cycle for turning machining I
G71	Longitudinal rough cutting cycle (finished shape chamfering)	
G72	Face rough cutting cycle (finished shape chamfering)	
G73	Formed Material Rough Cutting Cycle	
G74	Face Cut-Off Cycle	Compound type fixed cycle for turning machining II
G75	Longitudinal cut-off cycle	
G76	Compound thread cutting cycle	

The compound type fixed cycle for turning machining I (G70 to G73) cannot be used if the finished shape program is not registered in the memory.

13.3.1 Longitudinal Rough Cutting Cycle ; G71



Function and purpose

This function calls the finished shape program and, while automatically calculating the tool path, performs rough cutting in the longitudinal direction.



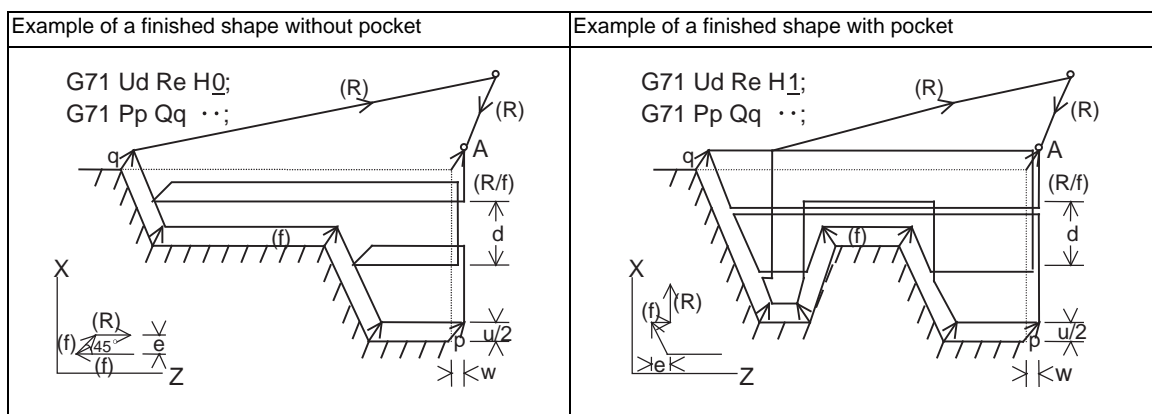
Command format

**G71 Ud Re Hh ;
G71 Aa Pp Qq Uu Ww Ff Ss Tt ; ... Longitudinal Rough Cutting Cycle**

This fixed cycle requires two blocks.

However, when using a value set by a parameter, the first block can be omitted.

Ud	Cut amount (modal) ... Reversible parameter "#8051 G71 THICK" is also available for setting Unit: μm / 1/10000 inch radius value command
Re	Retract amount (modal) ... Reversible parameter "#8052 G71 PULL UP" is also available for setting Unit: μm / 1/10000 inch radius value command
Hh	Pocket machining (modal)Reversible parameter "#8110 G71/G72 POCKET" is also available for setting 0: No pocket machining 1: Pocket machining
Aa	Finished shape program No. (Program being executed when omitted) Use a parameter to specify a 4- or 8-digit program No. starting with O.
Pp	Finished shape start sequence No. (Head of program when omitted)
Qq	Finished shape end sequence No. (To the end of program when omitted) If M99 is commanded before Q, the program will end at M99 even if Q is commanded.
Uu	X axis direction finishing allowance (When omitted, X axis direction finishing allowance is handled as 0.) Unit: μm / Follows 1/10000inch diameter/radius value command (#1019 dia).
Ww	Z axis direction finishing allowance (When omitted, Z axis direction finishing allowance is handled as 0.) Unit: μm / 1/10000 inch radius value command
Ff	Cutting speed (When omitted, the cutting speed (modal) just before G71 will be applied)
Ss,Tt	Spindle command, tool command



(R) Rapid traverse

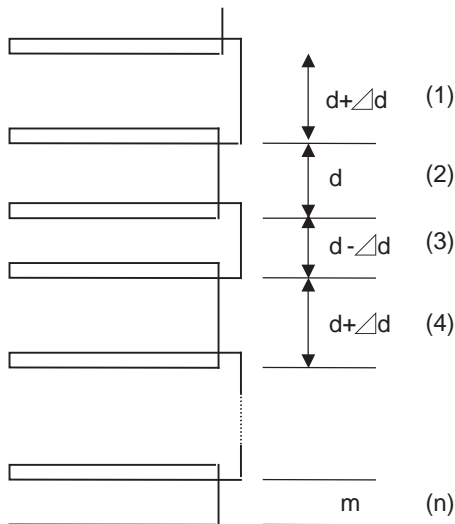
(f) Cutting feed

A : Rough cutting start point

(Note) A reversible parameter enables to use parameter setting value without issuing a program command and also, the value can be changed by the program command.

Cut amount: Ud

- (1) Designate the cut amount by Ud or parameter "#8051 G71 THICK".
Also, the amount can be changed for each cutting by setting the cut change amount (Δd) using the parameter.
- (2) Using the following parameters, select the response when a cut amount commanded in the program is deeper than the cutting depth of the finished shape.
Cut amount designation (#1271 ext07/bit7)
<ext07/bit7=0>
If a cut amount commanded in a program is deeper than the cutting depth of the finished shape ($d < \Delta d$), a program error (P204) occurred.
<ext07/bit7=1>
A program error will not occur even if a designated cut amount is deeper than the cutting depth of the finished shape, and the command will be executed as one cut.
Note that a cut amount must be between 0 and 99.999mm. A program error will occur if a value exceeds this range.
- (3) At the end of the cutting, remaining part will be the cut amount, but if this part is smaller than the value set in the parameter, finishing rough cutting will be executed instead of cutting.
 - Cut amount (d):
[Machining parameter] "#8051 G71 THICK" 0 to 99.999mm
 - Cutting change amount (Δd):
[Machining parameter] "#8017 G71 DELTA-D" 0 to 99.999mm
 - Minimum cut amount for final cut:
[Machining parameter] "#8016 G71 MINIMUM" 0 to 99.999mm



(1) to (n) : 1st cutting to last cutting
m : Remainder

Cutting method and retract amount: Re

- (1) Designate the retract amount by Re or the parameter "#8052 G71 PULL UP" (0 to 99.999mm).
- (2) The cutting method differs according to whether pocket machining is ON or OFF.
 - Pocket machining OFF.....Retracts in 45° direction in respect to workpiece
 - Pocket machining ONTraces shape
 When the pocket machining is ON, cutting direction differs depending on "#1272 ext08/bit0" setting.
 Cut-in amount designation [Base specification parameter] "#1272 ext08/bit0"
 0 : The pull up direction is Z axis direction when pocket machining is ON
 1 : The pull up direction is X axis direction when pocket machining is ON
 However, when "#1272 ext08/bit0" is "1", pocket machining will be performed only when the first movement block after the finished shape start block, is designated to X and Z axes.

#1272 ext08	<When pocket machining is OFF>	<When pocket machining is ON>
bit0 = 0		
bit0 = 1		

Pocket machining: Hh

- (1) Pocket machining ON/OFF can be set by Hh and a parameter.
 #8110 G71/G72 POCKET
 0: No pocket machining
 1: Pocket machining
 However, when "#1272 ext08/bit0" is "1", pocket machining will be performed only when the first movement block after the finished shape start block, is designated to X and Z axes.

Finished shape start block and finished shape end block:Aa,Pp,Qq

Designate the finished shape start block and finished shape end block by Aa,Pp,Qq.

If calling a subprogram numbered with O is enabled, a program number starting with O and specified by A command value is called.

When A command value is less than the digit number set with parameter "#8129 subprogram number selection", increase the digit number of the command value by adding leading zeros.

In the following cases, a subprogram numbered without O and specified by A command value is called even when calling a subprogram numbered with O is enabled.

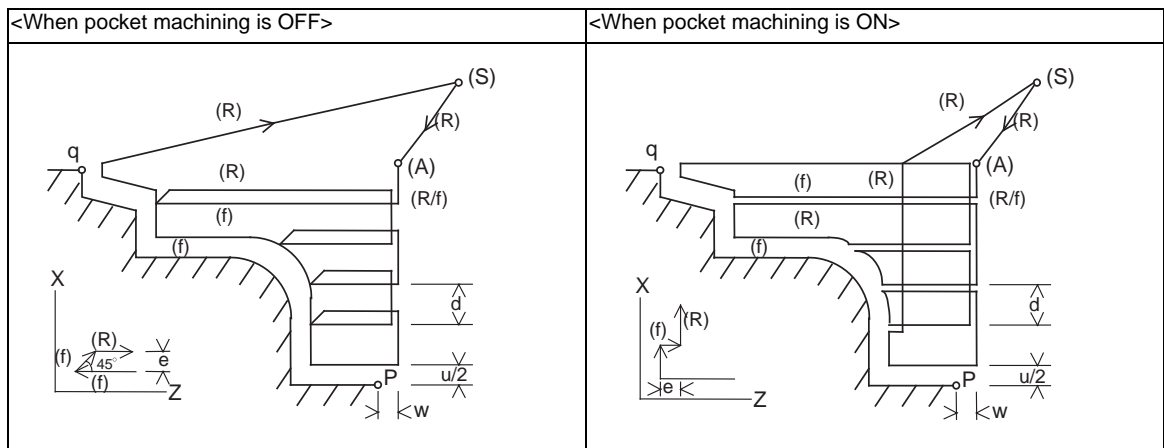
- When the digit number of A command value is greater than that of the parameter "#8129 subprogram number selection".
- When a specified subprogram number starting with O does not exist.

When A command is omitted, program command P, Q being executed, will be applied.

And when A command is omitted, the program will be executed in the order of G71 and then the program following Qq (finished shape end sequence No.)

A file name can be designated instead of address A by enclosing the file name in brackets <>. (The file name can have up to 32 characters including the extension.)

The maximum number of finished shape blocks is 200 including the blocks inserted with corner chamfer, corner R command and nose R command. A program error (P202) will occur if the numbers of blocks exceed the limit.



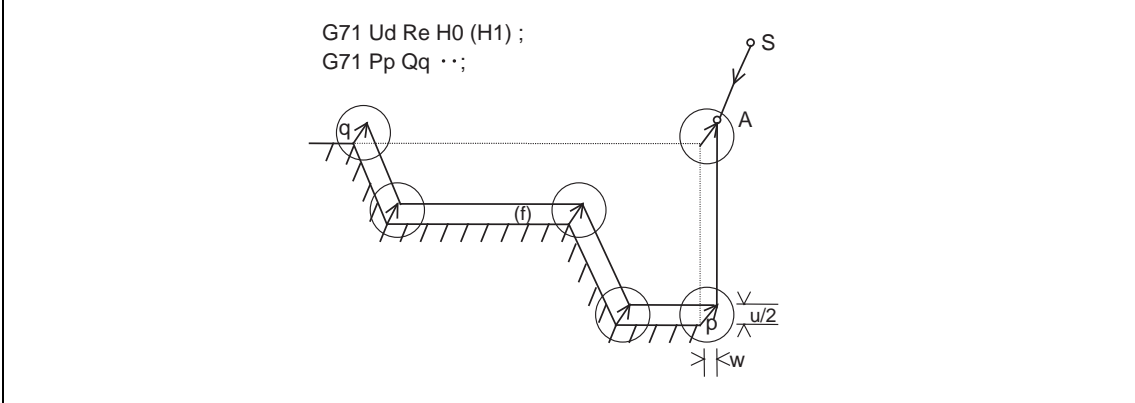
- (R) Rapid traverse
- (f) Cutting feed
- (S) Cycle command point
- (A) Rough cutting start point

<When creating a finished shape program in a machining program other than the one currently being executed>	<When creating a finished shape program in a program currently being executed>
<p>G71 A__ P__ Q__ A: Finished shape program No. P: Finished shape start sequence No. (From head of finished shape program when omitted) Q: Finished shape end sequence No. (To end of finished shape program or M99 when omitted)</p>	<p>G71 P__ Q__ P: Finished shape start sequence No. Q: Finished shape end sequence No.</p>
<p>Program currently being executed G0 X80.0 Z75.0 T0101; <- (a) G71 U10. R3.; <- (b) G71 A100 P10 Q20 U3. W1.5 F500 S1500; <- (c) G70 A100 P10 Q20; <- (d)</p> <p>Shape program O100 G28 XZ; N10 G0 X15.0 Z65.0; G1 Z55. F450; G1 X30.0; G3 X40.0 Z50.0 R5.0; : G1 Z42.0; G1 X50.0 Z35.0; N20 G1 X60.0; N30 G0 X13.0 Z68.0;</p> <p>Program execution order (a)(b)(c)(d) (N10-N20...Finished shape program)</p>	<p>Program currently being executed G0 X80.0 Z75.0 T0101; <- (a) G71 U10. R3.; <- (b) G71 P10 Q20 U3. W1.5 F500 S1500; <- (c) N10 G0 X15.0 Z65.0; G1 Z55. F450; G1 X30.0; G3 X40.0 Z50.0 R5.0; G1 Z42.0; G1 X50.0; G1 X55.0 Z35.0; N20 G1 X60.0; G70 A100 P10 Q20; <- (d)</p> <p>Program execution order (a)(b)(c)(d) (N10-N20...Finished shape program)</p> <p>* G70 command: Finishing cycle The workpiece is finished following the finishing program, and then the axis returns to the G70 command position.</p>

finishing allowance: Uu,Ww

When the finishing allowance is designated, the Uu/Ww part will be left uncut from the finished shape.
 X axis finishing allowance ...The finishing allowance is left uncut in the rough cutting start point direction.

<Finishing allowance when pocket machining is OFF or when machining an open section when pocket machining is ON (finished shape start block X axis is at base)>

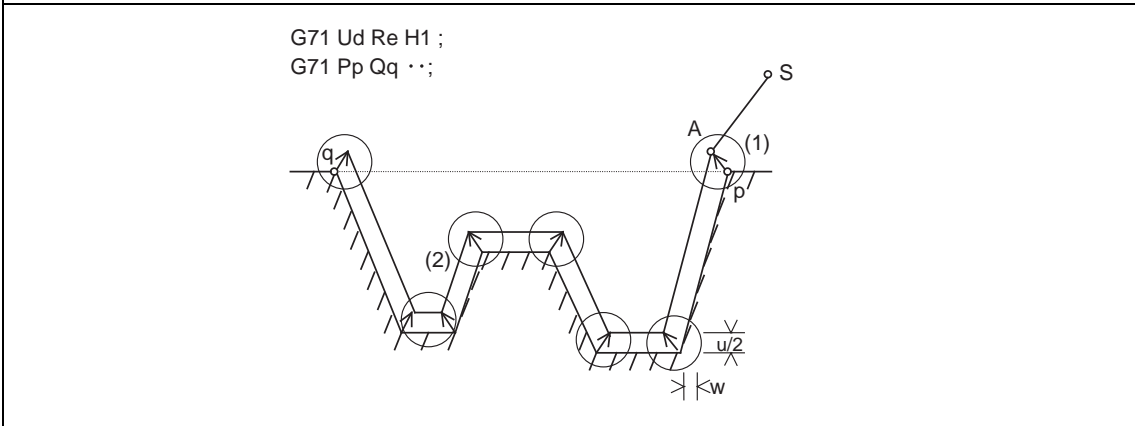


S : Cycle command point
 p : Finished shape start block
 q : Finished shape end block

A : Rough cutting start point

Z axis finishing allowance ...The finishing allowance is left uncut in the rough cutting start point direction.
 Note that if down cutting (block commanded in direction in which X axis faces hole base) is executed when pocket machining is ON and if cutting is started from the middle of the longitudinal direction, the Z axis finishing allowance direction will be reversed.

<Finishing allowance of mid-section when pocket machining is ON (Finished shape start block X axis = finished shape end block X axis)>



S : Cycle command point
 p : Finished shape start block
 q : Finished shape end block

A : Rough cutting start point

The block that starts cutting from the middle of the longitudinal direction will have a reversed Z axis finishing allowance direction. (1)

(Note) During tool nose R compensation, programming must be done so that the X axis positions of p and q after the tool nose R compensation will be the same value. (Refer to "Cutting shape" (5).)

The block that executes down cutting has a reversed Z axis finishing allowance direction. (2)



Detailed description

Validity of pocket machining

It is judged that there is a pocket section when down cutting (finished shape block in which previous movement block has no X axis movement command, or the X axis moves in the reverse direction of the hole base and then moves toward the hole base) is issued between the block following the finished shape start block and the end block.

(Note 1) During tool nose radius compensation, the validity of the pocket section is judged in respect to the finished shape after tool nose radius compensation.

(Note 2) If the finished shape has a pocket, program the pocket in a way that causes no interference between the tool and workpiece.

Machining of open section and machining of mid-section

The rough cutting cycle includes machining with one end of the workpiece is open, and machining which starts at a mid-section of a workpiece.

Machining of open sectionPossible when pocket machining is either OFF or ON

Machining of mid-sectionPossible only when pocket machining is ON

(Entire rough cutting cycle is handled as one pocket.)

	#1271 ext07/bit5 = 0	#1271 ext07/bit5 = 1
Open section		
Mid-section		

- S : Cycle command point
- A : Rough cutting start point
- B : Finished shape start block
- C : Finished shape end block

(Note) #1271 ext07/bit5: Selection of cut start position
 0: Determined by finished shape program
 1: Determined by cycle start point

Rough cutting direction

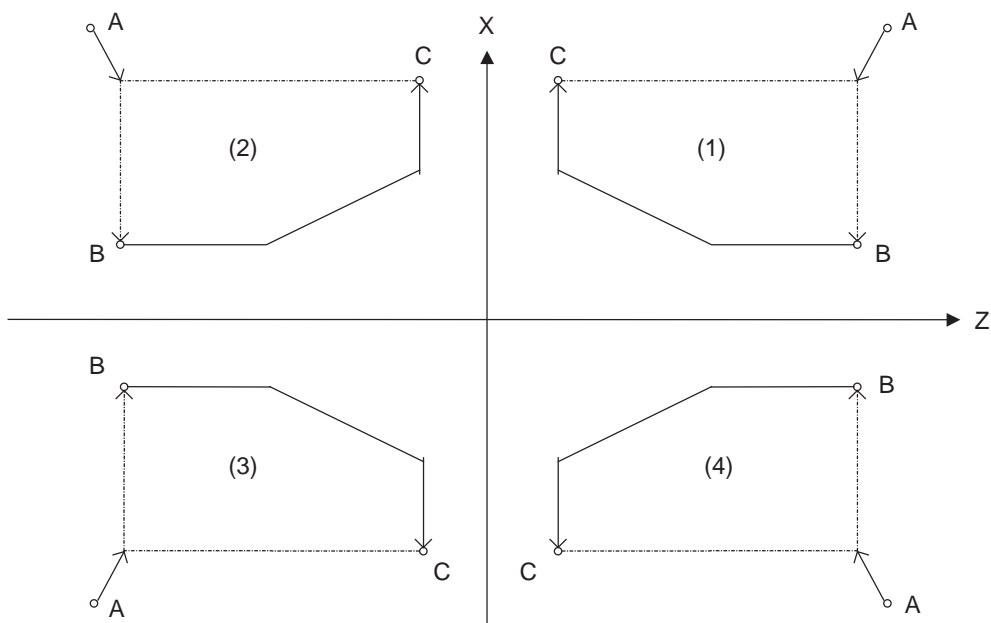
<Rough cutting direction when pocket machining is OFF>

[Automatically determine according to finished shape (#1273 ext09/bit2=0)]

The rough cutting direction is determined in the following manner according to the finished shape.

C (finished shape end block) X axis > B (finished shape start block) X axis	(1) or (2)
C (finished shape end block) X axis < B (finished shape start block) X axis	(3) or (4)
C (finished shape end block) Z axis > B (finished shape start block) Z axis	(2) or (3)
C (finished shape end block) Z axis < B (finished shape start block) Z axis	(1) or (4)

A program error (P203) will occur if C (finished shape end block) X axis = B (finished shape start block) X axis or C (finished shape end block) Z axis = B (finished shape start block) Z axis.



- A : G71 cycle command point
- B : Finished shape start block
- C : Finished shape end block

The finished shape program is A → B → C.

[Designate with program (#1273 ext09/bit2=1)]

The finishing allowance direction follows the sign of the finishing allowance designated in the program.

<Rough cutting direction when pocket machining is ON>

Select one of the followings.

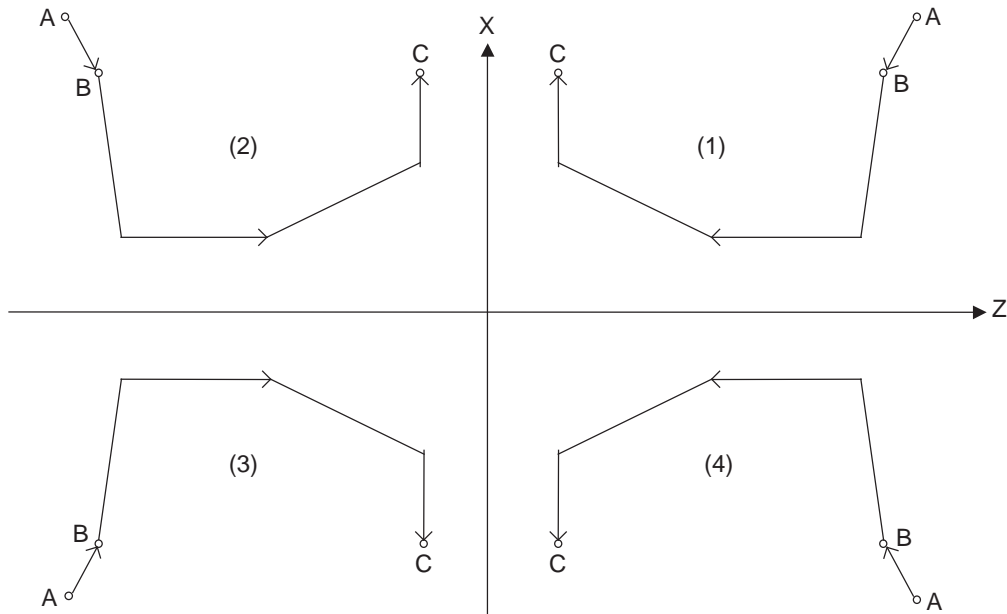
[Automatically determine according to finished shape (#1273 ext09/bit2=0)]

The rough cutting direction is determined in the following manner according to the finished shape.

A (G71 cycle command point) X axis > B (finished shape start block) X axis, and B (finished shape start block) X axis <= C (finished shape end block) X axis	(1) or (2)
A (G71 cycle command point) X axis < B (finished shape start block) X axis, and B (finished shape start block) X axis >= C (finished shape end block) X axis	(3) or (4)
C (finished shape end block) Z axis > B (finished shape start block) Z axis	(2) or (3)
C (finished shape end block) Z axis < B (finished shape start block) Z axis	(1) or (4)

(Note 1) A program error (P203) will occur if C (finished shape end block) Z axis = B (finished shape start block) Z axis

(Note 2) When setting the B point X axis as the base, the method is the same as the "<Rough cutting direction when pocket machining is OFF>".



A : G71 cycle command point
B : Finished shape start block
C : Finished shape end block

The finished shape program is $A \rightarrow B \rightarrow C$.

[Designate with program (#1273 ext09/bit2=1)]

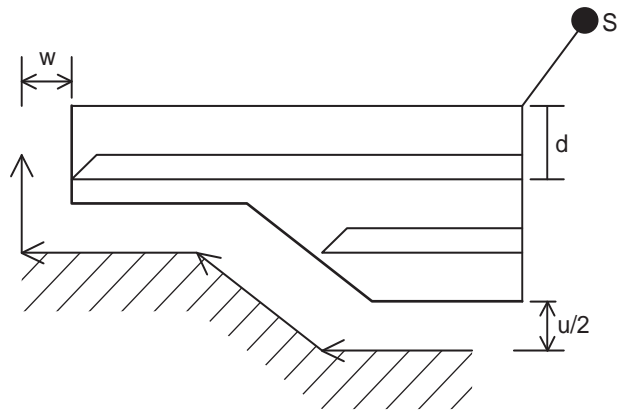
The finishing allowance direction follows the sign of the finishing allowance designated in the program.

Finished shape

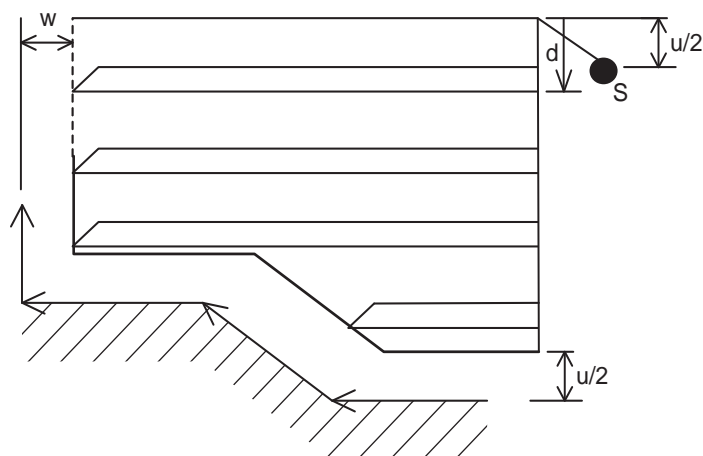
Selection of cut start position (#1271 ext07/bit5)

The cut start position is calculated from the final position of the finished shape program, but this can be changed to the cycle start point.

<ext07/bit5 = 0>



<ext07/bit5 = 1>



S : Cycle start point
 u/2, w : Finishing allowance
 d : First cut

<Finished shape in Z axis direction when pocket machining is OFF>

The Z axis direction finishing allowance must always change monotonously (only increment, or only decrement).

A program error (P203) will occur if the shape is illegal.

However, the program error can be avoided in the case of a minute inversion by setting a tolerance error range.

<Example of illegal Z axis direction when pocket machining is OFF>	
Z axis direction reverses (program error (P203))	Z axis direction reverses at middle of arc (program error (P203))

The program error does not occur and machining continues as long as a reversed travel distance of Z axis is within the tolerance error range (#8060 G71 ERR).

<Finished shape in X axis direction when pocket machining is OFF>

As a principle, the X axis direction finishing allowance must change monotonously (increment or decrement).

At sections that do not change monotonously, the rough cutting cycle will not be executed at the indent section (pocket), but the finishing allowance will be traced as commanded.

When programming, be aware that if the pocket's X axis depth is not less than the cut amount, the cutting load will increase at the pocket section.

<Example of illegal X axis direction when pocket machining is OFF>	
Smaller than cut amount d (Program error (P203))	There is a pocket when pocket machining is OFF. (An error will not occur, but the pocket section will be rough cut with one cut.)

A : G71 cycle command point

B : Finished shape start block

C : Finished shape end block

<Finished shape in Z axis direction when pocket machining is ON>

The Z axis direction finishing allowance must always change monotonously (only increment, or only decrement).

Sections that do not change monotonously will have a cover.

<Example of section where Z axis direction does not change monotonously when pocket machining is ON>	
The Z axis direction reverses, so the finished shape is created with a cover at the shaded section. (The shaded section is not cut)	The Z axis direction reverses in the middle of the arc, so the finished shape is created with a cover at the shaded section. (The shaded section is not cut)

- A : Cycle command point
- B : Finished shape start block
- C : Finished shape end block

The shape is changed (cover) even when the reversed travel distance of Z axis is within the tolerable error range (#8060 G71 ERR).

<Finished shape in X axis direction when pocket machining is ON>

As a principle, the finished shape in the X axis direction must change monotonously (increment or decrement). The direction must reverse only at the indent shape (pocket).

During pocket machining, the block in finishing, must not exceed the C (finished shape end block) X axis position.

If it does, a program error (P203) will occur.

<Example of illegal X axis direction when pocket machining is ON>	
Smaller than cut amount d (Program error (P203))	Block in finishing is higher than finished end point block (Program error (P203))

- A : Cycle command point
- B : Finished shape start block
- C : Finished shape end block

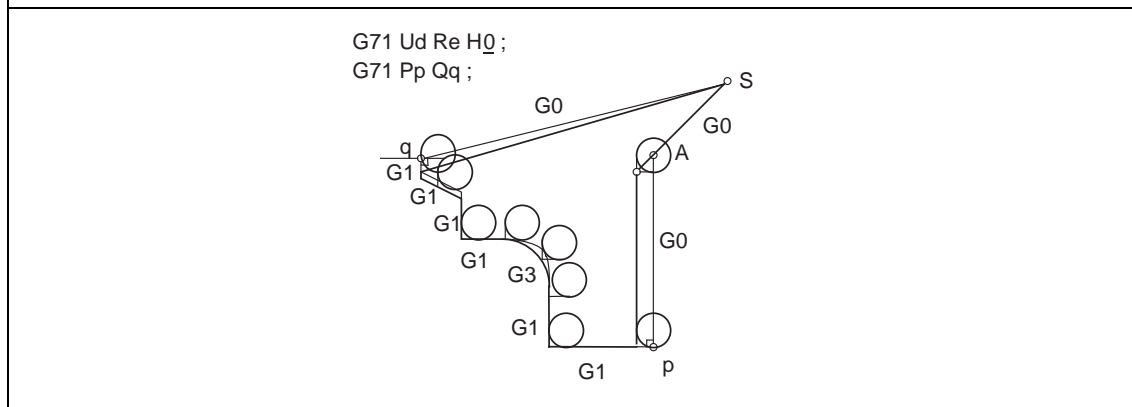
Finished shape during tool nose R compensation

Selection of tool nose R compensation (#1271 ext07/bit6)

<ext07/bit6=0>

If there is a G71 command during tool nose R compensation, the G71 cycle command point will be at the position where tool nose R compensation is temporarily canceled. The finished shape will be created while tool nose R compensation is applied.

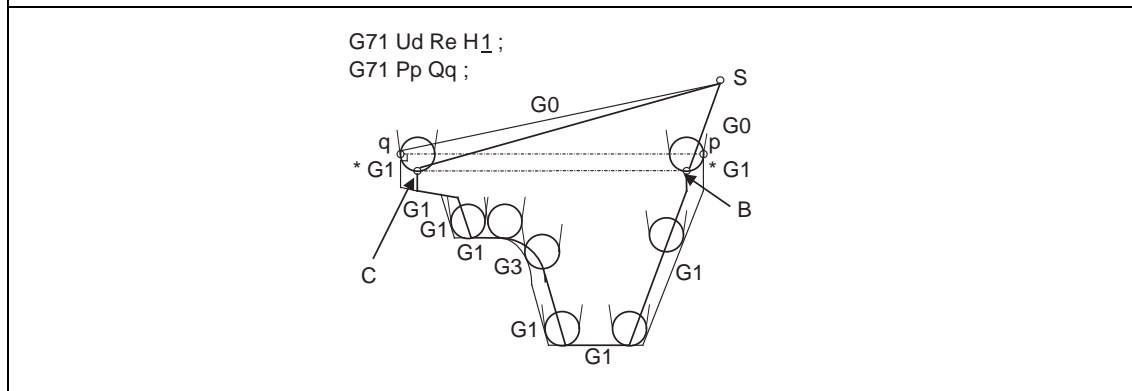
<Example of tool nose No. 3 (pocket machining OFF) ... Machining of open section>



S : Cycle command point

A : Rough cutting start point

<Example of tool nose No. 8 (pocket machining ON) ... Machining of mid-section>



S : Cycle command point

B : Finished shape start block after nose R compensation

C : Finished shape end block after nose R compensation

(Note) To machine a mid-section while pocket machining is ON, programming must be done so that the X axis positions of the finished shape start block and finished shape end block are the same after tool nose R compensation is applied.

Precautions for machining mid-section

The mid-section will be machined when the X axis positions of the finished shape start block and finished shape end block are the same after nose R compensation.

Programming must be done so that only the X axis moves (more than nose R diameter) in the block with * mark in the above figure.

<ext07/bit6=1>

Rough cutting will be executed according to the finished shape program without the tool nose R compensation.



Program example

Machining of open section (example with pocket machining OFF)

<pre>G0 X80.0 Z75.0 T0101; <- (1) G71 U10. R3. H0; <- (2) G71 P10 Q20 U3.W1.5 F500 S1500; <- (3) N10 G0 X15.0 Z65.0; G1 Z55. F450; G1 X30.0; G3 X40.0 Z50.0 R5.0; G1 Z42.0; G1 X50.0; G1 X55.0 Z35.0; N20 G1 X60.0; G70 P10 Q20; <- (4)</pre> <p>Program execution order (1)(2)(3)(4) (N10-N20...Finished shape program)</p>	<p>G71 Ud Re H0 ;</p>
---	-----------------------

Machining of open section (example with pocket machining ON)

<pre>G0 X80.0 Z75.0 T0101; <- (1) G71 U10. R3. H1; <- (2) G71 P10 Q20 U3.W1.5 F500 S1500; <- (3) N10 G0 X15.0 Z65.0; G1 Z55. F450; G1 X30.0; G3 X40.0 Z50.0 R5.0; G1 Z42.0; G1 X50.0; G1 X55.0 Z35.0; N20 G1 X60.0; G70 P10 Q20; <- (4)</pre> <p>Program execution order (1)(2)(3)(4) (N10-N20...Finished shape program)</p>	
---	--

Machining of mid-section (example with pocket machining ON)

<pre>G0 X80.0 Z75.0 T0101; <- (1) G71 U10. R3. H1; <- (2) G71 P10 Q20 U3.W1.5 F500 S1500; <- (3) N10 G0 X60.0 Z73.0; G1 X15.0 Z65.0; G1 Z55. F450; G1 X30.0; G3 X40.0 Z50.0 R5.0; G1 Z42.0; G1 X50.0; G1 X55.0 Z35.0; N20 G1 X60.0; G70 P10 Q20; <- (4)</pre> <p>Program execution order (1)(2)(3)(4) (N10-N20...Finished shape program)</p>	<p>G71 Ud Re H1 ;</p>
---	-----------------------

S : Cycle command point
(R) Rapid traverse

A : Rough cutting start point
(f) Cutting feed



Precautions

Refer to "Precautions for Compound Type Fixed Cycle for Turning Machining; G70 to G76".

13.3.2 Face Rough Cutting Cycle ; G72



Function and purpose

This function calls the finished shape program and, while automatically calculating the tool path, performs rough cutting in the face direction.



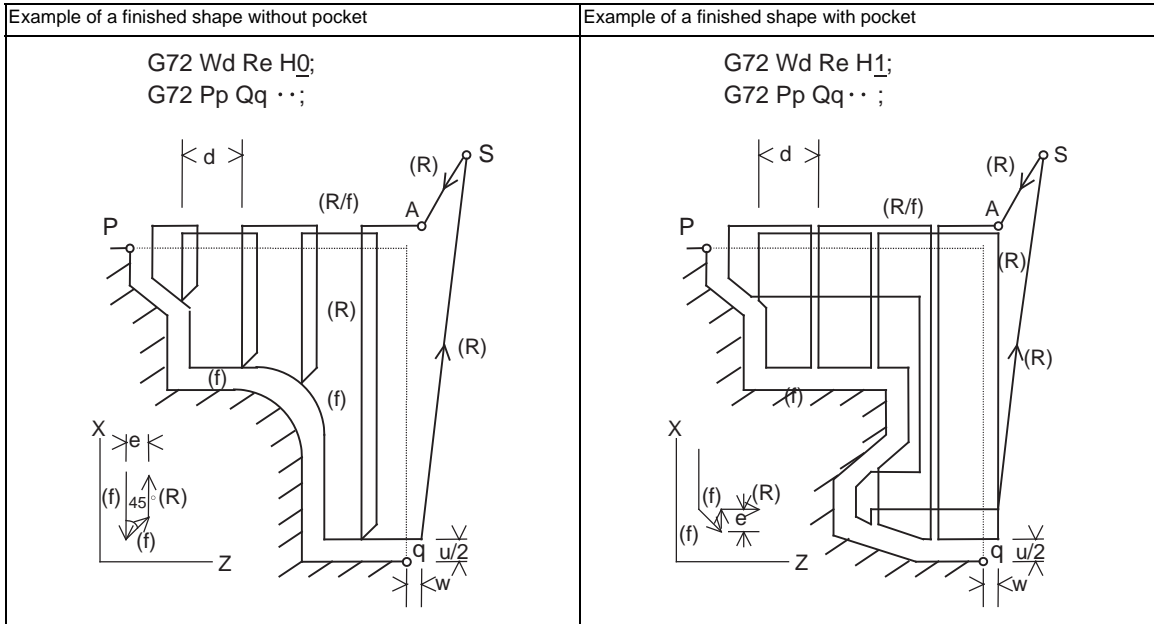
Command format

G72 Wd Re Hh ;
G72 Ae Pp Qq Uu Ww Ff Ss Tt ; ... Face Rough Cutting Cycle

This fixed cycle requires two blocks.

However, when using a value set by a parameter, the first block can be omitted.

Wd	Cut amount (modal) ...Reversible parameter "#8051 G71 THICK" is also available for setting Unit: μ m/ 1/10000 inch radius value command
Re	Retract amount (modal) ...Reversible parameter "#8052 G71 PULL UP" is also available for setting Unit: μ m/ 1/10000 inch radius value command
Hh	Pocket machining (modal) ...Reversible parameter "#8110 G71/G72 POCKET" is also available for setting 0: No pocket machining 1: No pocket machining
Aa	Finished shape program No. (Program being executed when omitted) Use a parameter to specify a 4- or 8-digit program No. starting with O.
Pp	Finished shape start sequence No. (Head of program when omitted)
Qq	Finished shape end sequence No. (To the end of program when omitted) If M99 is commanded before Q, the program will end at M99 even if Q is commanded.
Uu	X axis direction finishing allowance (When omitted, X axis direction finishing allowance is handled as 0.) Unit: μ m/ Follows 1/10000inch diameter/radius value command (#1019 dia).
Ww	Z axis direction finishing allowance (When omitted, Z axis direction finishing allowance is handled as 0.) Unit: μ m/ 1/10000 inch radius value command
Ff	Cutting speed (When omitted, the cutting speed (modal) just before G72 will be applied)
Ss,Tt	Spindle command, tool command



S : Cycle command point

A : Rough cutting start point

(R) Rapid traverse

(f) Cutting feed

(Note) A reversible parameter enables to use parameter setting value without issuing a program command and also, the value can be changed by the program command.



Detailed description

Refer to "Longitudinal Rough Cutting Cycle" for details of the functions.



Precautions

Refer to "Precautions for Compound Type Fixed Cycle for Turning Machining; G70 to G76".

13.3.3 Formed Material Rough Cutting Cycle ; G73



Function and purpose

This function calls the finished shape program, automatically calculates the tool path and performs rough cutting while cutting the workpiece into the finished shape.

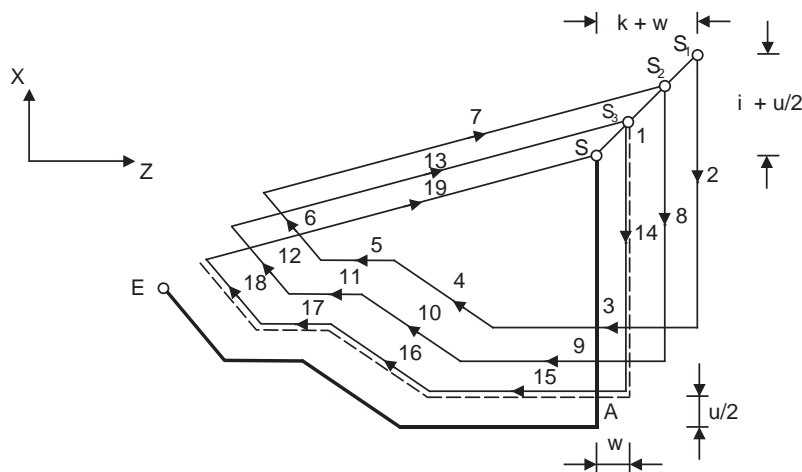


Command format

G73 Ui Wk Rd ;
G73 Aa Pp Qq Uu Ww Ff Ss Tt ; ... Formed Material Rough Cutting Cycle

This fixed cycle requires two blocks.

Ui Wk Rd	X-axis direction cutting allowance i Z-axis direction cutting allowance k Number of divisions d	- Cutting allowances when P, Q commands are not issued - Modal data - Sign is ignored - Radius designation applies to the cutting allowance.
Aa	Finished shape program No.	(program being executed when omitted) A file name can be designated instead of address A by enclosing the file name in brackets <>. (The file name can have up to 32 characters including the extension.)
Pp	Finished shape start sequence No.	(program head when omitted)
Qq	Finished shape end sequence No.	(up to end of program when omitted) If M99 is commanded before Qq, the program will end at M99.
Uu Ww	X-axis direction finishing allowance u Z-axis direction finishing allowance w	- Cutting allowance when P, Q commands are issued. - Sign is ignored - Diameter/Radius designation changes in accordance with the parameters (#1019 dia). - The shift direction is determined by the shape. For details, refer to the "finishing allowance direction :Uu, Ww " for G71.
FfSsTt	Cutting feedrate (F function) Spindle speed (S function) Tool command (T function)	The F, S and T commands in the finished shape program are ignored, and the value in the rough cutting cycle command or the value before the command will be valid.



S : G73 cycle command point

A : Finished shape start block

E : Finished shape end block

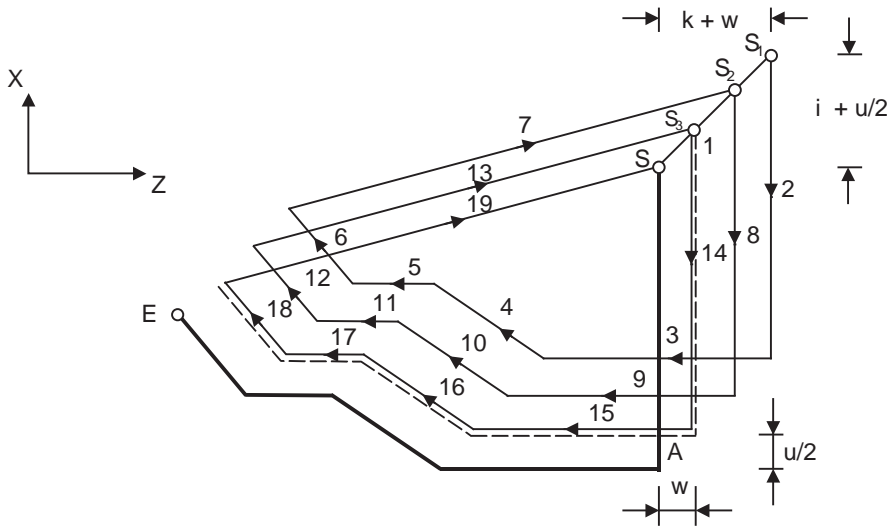
(Note) With a single block, operation stops at the end point of each block.



Detailed description

Finished shape

In the program, S -> A -> E in the figure below are commanded.
 The section between A and E must be a shape with monotonous changes in both the X axis and Z axis directions.

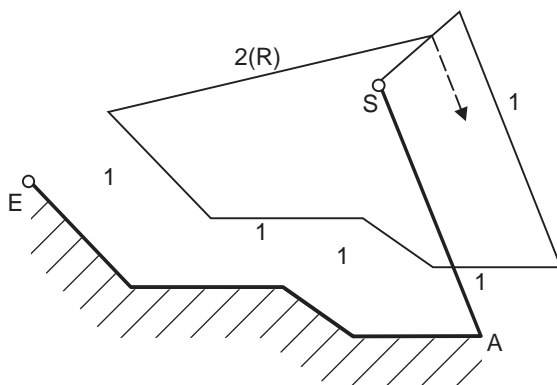


- S : G73 cycle command point
- A : Finished shape start block
- E : Finished shape end block

(Note) With a single block, operation stops at the end point of each block.

1 cycle configuration

1 cycle is configured as shown below.



- 1 : Machining with shape profiling (based on shape program)
- 2 : Return to the next command point (rapid traverse)
- S : G73 cycle command point
- A : Finished shape start block
- E : Finished shape end block

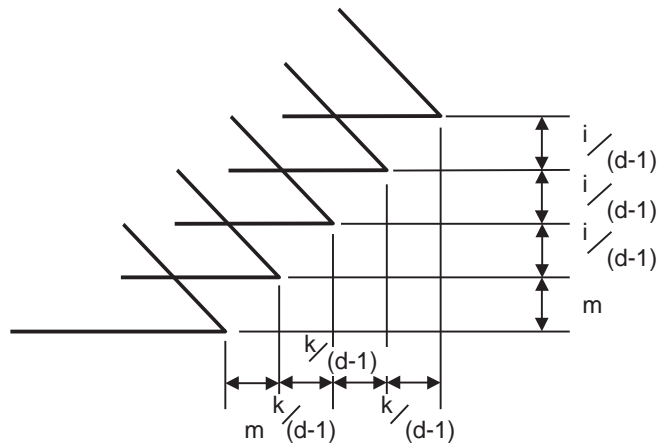
Cut amount

The cut amount is calculated by dividing the cutting allowances (i, k) by the number of divisions (d-1).

X axis direction $i/(d-1)$

Z axis direction $k/(d-1)$

When the allowance is not divisible, chamfering will be performed and adjustment will be made at the final pass.



m : Remainder

Tool nose R compensation

<#1271 ext07/bit6 = 0>

When this cycle is commanded with the tool nose R compensation mode still in force, the compensation is temporarily canceled immediately before this cycle and started at the head block of the finished shape program. So the compensation is applied to the finished shape program covered by this cycle and this cycle is executed for the compensated shape.

<#1271 ext07/bit6 = 1>

Like G71 and G72 cycle, rough cutting is carried out to the shape which tool nose radius compensation is not carried out.

Cutting direction

Determined according to finish shape (#1273 ext09/bit2=0)

The shift direction for the cutting is determined by the shape in the finishing program, as shown in the table below.

	1	2	3	4
Drawing				
Initial X axis	- direction	-	+	+
Complete Z axis	- direction	+	+	-
X axis cutting	+ direction	+	-	-
Z axis cutting	+ direction	-	-	+

S : G73 cycle command point

A : Finished shape start block

E : Finished shape end block

Determined according to commanded finish allowance/cutting allowance (ext09/bit2=1)

The shift direction for cutting follows the sign of the cutting allowance, etc., designated in the program.



Precautions

Refer to "Precautions for Compound Type Fixed Cycle for Turning Machining; G70 to G76".

13.3.4 Finishing Cycle ; G70



Function and purpose

After rough cutting have been carried out by the G71 to G73 commands, finishing cutting can be performed by the following command.



Command format

G70 A__ P__ Q__ ; ... Finishing cycle

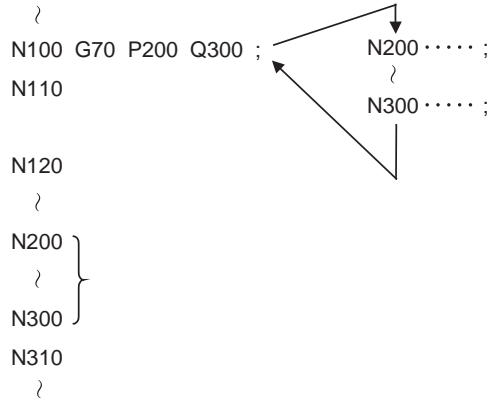
A	Finished shape program No. (Program being executed when omitted)
P	Finished shape start sequence No. (Head of program when omitted)
Q	Finished shape end sequence No. (To the end of program when omitted) If M99 is commanded before Q, the program will end at M99.



Detailed description

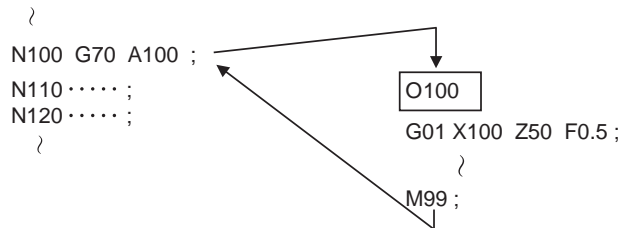
- (1) The F, S and T commands in the finished shape program are valid during the finishing cycle.
- (2) When the G70 cycle is completed, the tool returns to the start point by rapid traverse and the next block is read.

(Example 1) When a sequence No. is designated



N200-N300 ... Finished shape program

(Example 2) When a program No. is designated



In both Example 1 and Example 2, after the N100 cycle is executed, the N110 block will be executed next.



Precautions

Refer to "Precautions for Compound Type Fixed Cycle for Turning Machining; G70 to G76".

13.3.5 Face Cut-Off Cycle ; G74



Function and purpose

The G74 fixed cycle automatically performs grooving in the face direction of the workpiece by commanding the coordinates of the groove end point, cut amount, cutter shift amount and cutter escape at the bottom of the cut. The machining program commands are as follows.



Command format

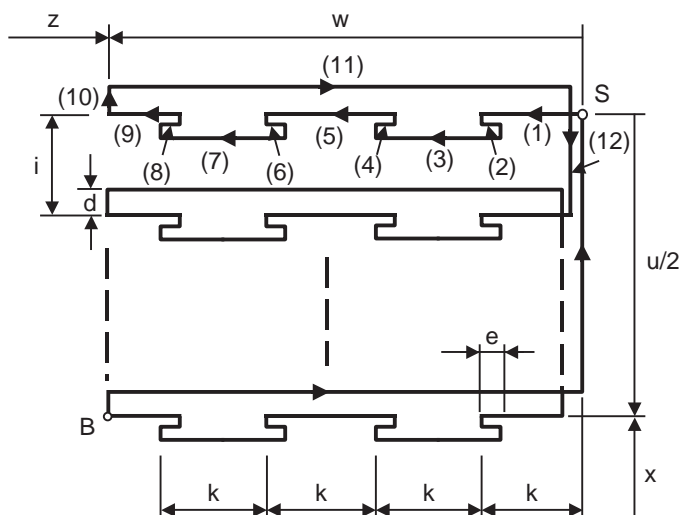
G74 Re ;
G74 X(U)x Z(W)z Pi Qk Rd Ff ; ... Face Cut-Off Cycle

This fixed cycle requires two blocks.

However, when using a value set by a parameter, the first block can be omitted.

Re	Return amount (no X/U, P commands) (modal) ... Reversible parameter "#8056 G74 RETRACT" is also available for setting
X/Ux	B point X coordinate (absolute/incremental value)
Z/Wz	B point Z coordinate (absolute/incremental value)
Pi	Tool shift amount (radius designation, incremental value, sign not required)
Qk	Cut amount (radius designation, incremental value, sign not required)
Rd	Escape at the bottom of the cut When there is no sign, the tool escapes even at the bottom of the first cut. When a - sign is attached, the tool escapes from the second cut without escaping the first cut.
Ff	Feedrate

(Note) A reversible parameter enables to use parameter setting value without issuing a program command and also, the value can be changed by the program command.



S : Start point

B : End point

Operations (9) and (12) immediately before the final cycle are executed with the remainder amount.

Operations (2), (4), (6), (8), (10), (11) and (12) are executed at the rapid traverse rate.

With a single block, operation stops at each block.



Detailed description

- (1) When X/U and P are omitted or when the values of "x" and "i" are zero, operation will apply to the Z axis only. Note that when there is an Rd command and no sign, the tool will escape at the bottom of the cut.
- (2) When X/U or Z/W command is not issued, Re setting value will be the return amount. Even when G74 Pi Qk Rd ; is commanded, Rd is regarded as Re, and the return amount will be set.
- (3) The escape direction does not change whether - sign is attached to the Rd command or not.
- (4) A program error (P204) will occur in the following cases.
 - (a) When " i " is zero or P is not commanded even though X/U is commanded.
 - (b) When tool shift amount " i " is larger than the "x" movement amount.
 - (c) When the escape amount "d" is larger than the shift amount " i " .
 - (d) When the return amount "e" is larger than the cut amount "k".
 - (e) When the cut in amount "k" is larger than the hole depth "w".



Precautions

Refer to "Precautions for Compound Type Fixed Cycle for Turning Machining; G70 to G76".

13.3.6 Longitudinal Cut-off Cycle ; G75



Function and purpose

The G75 fixed cycle automatically performs grooving in the longitudinal direction of the workpiece by commanding the coordinates of the groove end point, cut amount, cutter shift amount and cutter escape at the bottom of the cut.



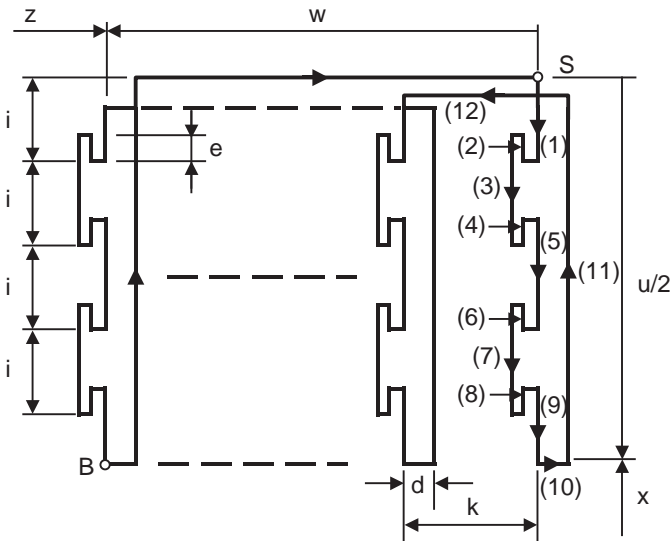
Command format

G75 Re ;
G75 X(U)x Z(W)z Pi Qk Rd Ff ; ... Longitudinal cut-off cycle

This fixed cycle requires two blocks.

However, when using a value set by a parameter, the first block can be omitted.

Re	Return amount (no X/U, P commands) (modal) ... Reversible parameter "#8056 G74 RETRACT" is also available for setting
X/Ux	B point X coordinate (absolute/incremental value)
Z/Wz	B point Z coordinate (absolute/incremental value)
Pi	Cut amount (radius designation, incremental value, sign not required)
Qk	Tool shift amount (radius designation, incremental value, sign not required)
Rd	Escape at the bottom of the cut When there is no sign, the tool escapes even at the bottom of the first cut. When a - sign is attached, the tool escapes from the second cut without escaping the first cut.
Ff	Feedrate



S : Start point

B : End point

Operations (9) and (12) immediately before the final cycle are executed with the remainder amount.

Operations (2), (4), (6), (8), (10), (11) and (12) are executed at the rapid traverse rate.

With a single block, operation stops at each block.

(Note) A reversible parameter enables to use parameter setting value without issuing a program command and also, the value can be changed by the program command.



Detailed description

- (1) When Z/W and Q are omitted or when the values of "z" and "k" are zero, operation will apply to the X axis only (slotting). Note that when there is an Rd command and no sign, the tool will escape at the bottom of the cut.
- (2) When X/U or Z/W command is not issued, Re setting value will be the return amount. Even when G75 Pi Qk Rd ; is commanded, Rd is regarded as Re, and the return amount will be set.
- (3) The escape direction does not change whether - sign is attached to the Rd command or not.
- (4) A program error (P204) will occur in the following cases.
 - (a) When "k" is zero or Q is not commanded even though Z/W is commanded.
 - (b) When tool shift amount "k" is larger than the "z" movement amount.
 - (c) When the escape amount "d" is larger than the shift amount "k".
 - (d) When the return amount "e" is larger than the cut amount "i".
 - (e) When the cut in amount "i" is larger than the hole depth "u/2".



Precautions

Refer to "Precautions for Compound Type Fixed Cycle for Turning Machining; G70 to G76".

13.3.7 Compound Thread Cutting Cycle ; G76



Function and purpose

The G76 fixed cycle enables to cut the workpiece at a desired angle by designating the thread cutting start point and end point, and it automatically performs cutting so that the cutting cross section (cutting torque) per cutting pass is constant.

Various longitudinal direction threads can be cut by bearing in mind the command value for the thread end point coordinate and taper height component.

With compound thread cutting cycle, chamfering can be operated. Refer to "Thread cutting cycle ; G78" for details on chamfering.

(Note) The operation of thread cutting is same as the thread cutting command(G33).



Command format

**G76 Pmra Q Δdmin Rd ;
G76 X/U Z/W Ri Pk Q Δd FI ; ... Compound thread cutting cycle**

This fixed cycle requires two blocks.

However, if "#1222 aux06/bit5" = "1" and "#1265 ext01/bit0" = "0" the 1st block can be omitted.

If the 1st block is omitted, the value set with the parameter is used.

Address		Significance
P	m	Number of cutting passes for finishing: 00 to 99 (times) (modal) (Reversible parameter "#8056 G76 TIMES" is also available for setting)
	r	Chamfering amount: 00 to 99 (0.1mm/rev) (modal) (Reversible parameter "#8014 CDZ-VALE" is also available for setting) The chamfering width based on thread lead "λ" is designated by a 2-digit integer without decimal point between the ranges from 0.0 to 9.9.
	a	Tool nose angle (thread angle): 00 to 99 (°) (modal) (Reversible parameter "#8059 G76 ANGLE" is also available for setting) The angle from 0° to 99° is assigned in 1° units. "m", "r" and "a" are commanded in succession during address P. (Example) When m=5, r=1.5 and a=0° , P is 051500 (P051500). The leading and trailing zeroes cannot be omitted.
Q	Δdmin	Minimum cut amount If the calculated cut amount is smaller than Δdmin, it is clamped by Δdmin. (When omitted, operations differ depending on #1222/bit4 setting) (Note 6)
R	d	Finishing allowance 0 to 9999 (μ m) (modal) (Reversible parameter "#8057 G76 LAST-D" is also available for setting)
X/U		X-axis end point coordinate of thread The X coordinate of the end point for the thread is commanded by an absolute or incremental value.
Z/W		Z-axis end point coordinate of thread The Z coordinate of the end point for the thread is commanded by an absolute or incremental value.
R	i	Taper height component (radius value) for thread Straight thread when "i" is zero
P	k	Thread height This is commanded by a positive radius value.
Q	Δd	Cut amount The cut amount of the first cutting pass is commanded by a positive radius value.
F	l	Thread lead

(Note 1) A reversible parameter enables to use parameter setting value without issuing a program command and also, the value can be changed by the program command.

(Note 2) The two G76 commands above cannot be assembled in a block.

The data commanded by P, Q and R are automatically identified according to the presence or absence of the X/U and Z/W axis addresses.

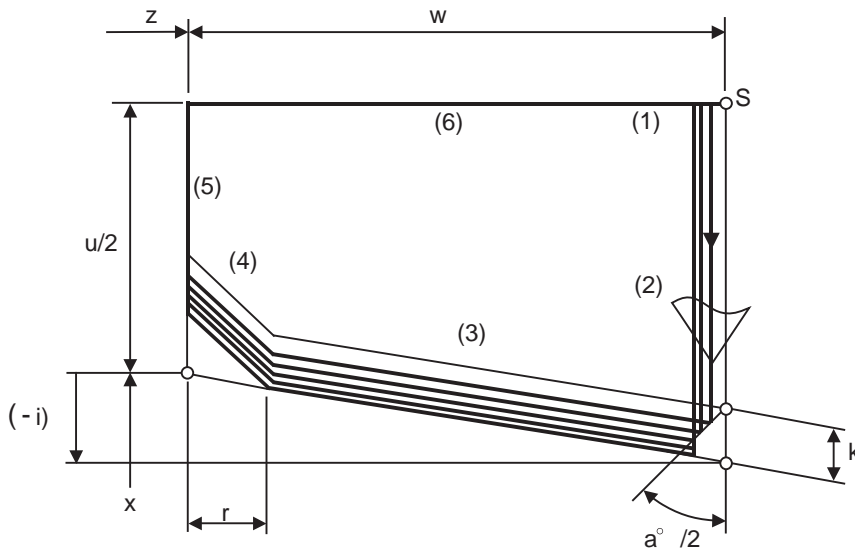
- (Note 3) The above "r" modal data can be rewritten by the program commands as well as using parameter (#8014 CDZ-VALE) settings.
- (Note 4) The chamfering amount designation is valid even for thread cutting fixed cycles.
- (Note 5) A program error (P204) will occur in the following cases.
 - When "a" is outside the rated value
 - When both or one of the X and Z commands is not issued, or when the start and end point coordinates coincides in both or one of the X and Z commands.
 - When the thread is larger than the movement of the X axis to the thread bottom
- (Note 6) The precautions for the thread cutting cycle (G78) should be observed.



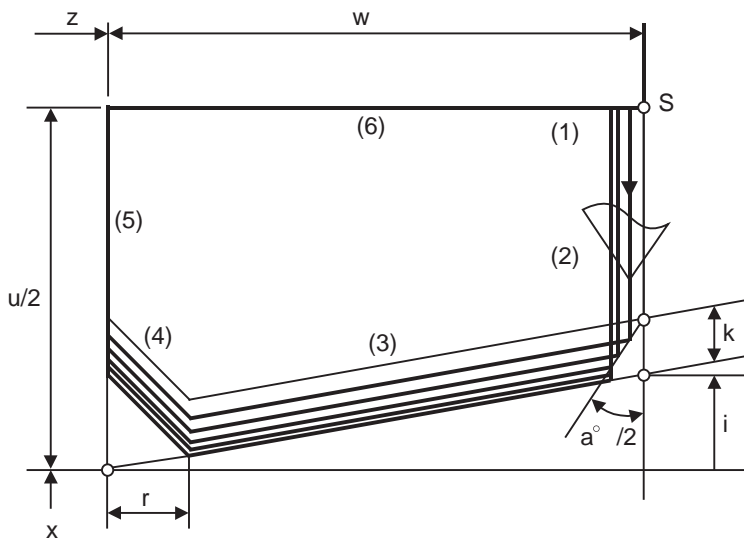
Detailed description

1 cycle configuration

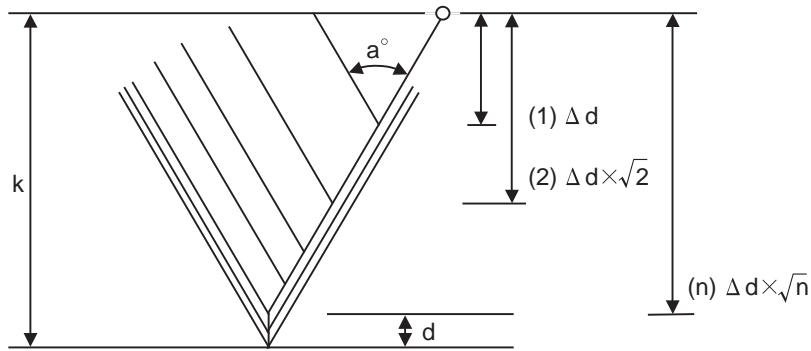
The tool moves at rapid traverse for operations (1), (2), (5) and (6) in the cycle and at the cutting feed based on the F designation for operations (3) and (4) during 1 cycle.



When Ri is negative



When Ri is positive



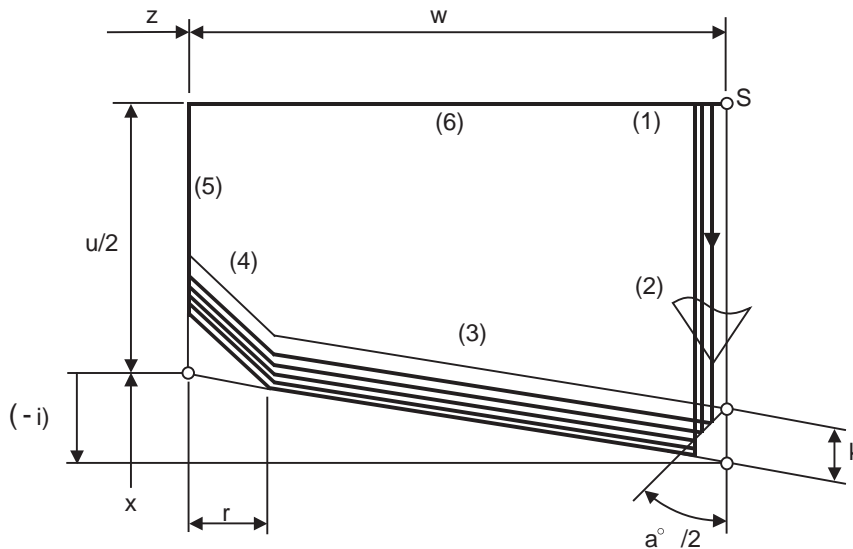
k : Thread height

d : Finishing allowance (cut "m" times)

(1) to (n) : 1st cutting to nth cutting

Interrupt operation

- (1) When the feed hold button is pressed during thread cutting, an automatic operation will stop upon completion of a block without thread cutting. (The automatic operation pause lamp turns on immediately and it goes off when automatic operation stops.)
If feed hold is applied when thread cutting is not executed, or when the thread cutting command is issued but the axis is yet to move, the automatic operation pause lamp will turn on, and the automatic operation will pause.
- (2) The tool stops upon completion of operations (1), (4) and (5) in the figure, if the following operations are conducted during the G76 command execution.
 - When the automatic operation mode is switched to another automatic operation mode
 - When automatic operation is changed to manual operation
 - When single block operation is conducted

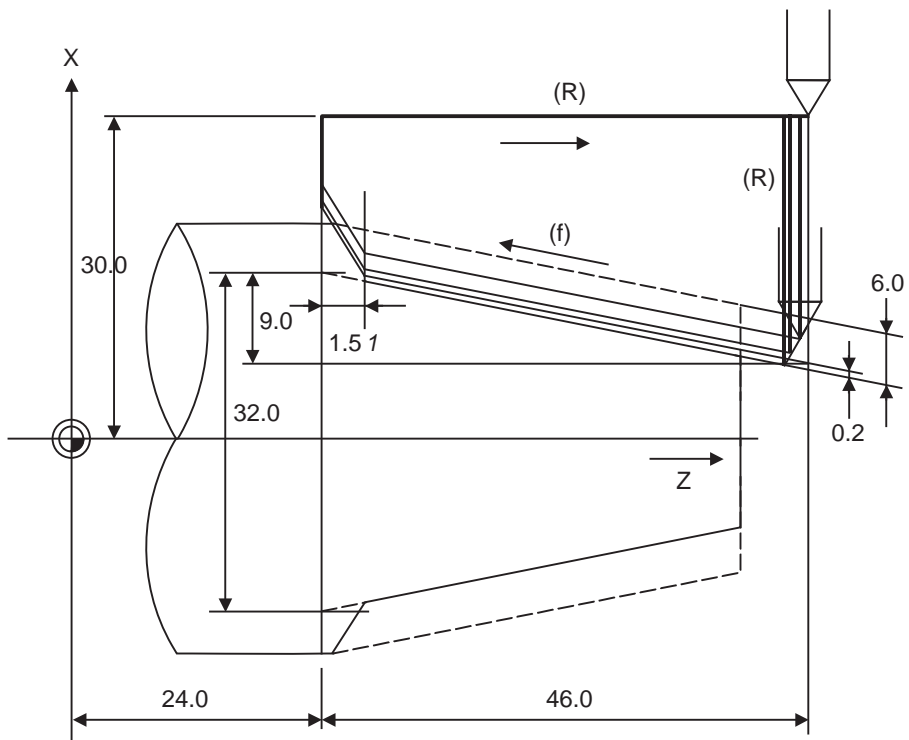


When Ri is negative

- (3) The dry run valid/invalid status does not change when G76 is being executed.



Program example



(R) Rapid traverse

(f) Cutting feed

G76 P011560 R0.2 ;

G76 U-28.0 W-46.0 R-9.0 P6.0 Q3.5 F4.0 ;



Precautions

Refer to "Precautions for Compound Type Fixed Cycle for Turning Machining; G70 to G76".

13.3.8 Precautions for Compound Type Fixed Cycle for Turning Machining; G70 to G76



Precautions

- (1) Command all required parameters in a compound type fixed cycle for turning machining command block.
- (2) Provided that the finished shape program is registered in the memory, compound type fixed cycle for turning machining I commands can be executed in the memory, MDI or tape mode.
- (3) When executing G70 to G73 command, ensure that the sequence No. of the finished shape program which is designated with P and Q is not duplicated in that program.
- (4) The finished shape program specified by P and Q in the G71 to G73 blocks should be prepared so that all the commands including those for corner chamfering, corner rounding and the automatic insertion blocks based on tool nose radius compensation, are created within 200 blocks. If this number is exceeded, program error (P202) will occur.
If the cutting start position is the cycle start position (#1271 ext07/bit5 =1), the finished shape program can have up to 199 blocks. (When the cycle start point is located out of the finished shape program's final position.)
- (5) The finished shape program which is designated by the G71 to G73 blocks should be a program with monotonous changes (increases only or reductions only) for both the X and Z axes.
- (6) Blocks without movement in the finished shape program are ignored.
- (7) N, F, S, M, and T commands in the finished shape program are ignored.
- (8) When any of the following commands exists in a finished shape program, program error (P201) will occur.
 - (a) Commands related to reference position return (G27, G28, G29, G30)
 - (b) Thread cutting (G33)
 - (c) Fixed cycles
 - (d) Skip functions (G31, G37)
- (9) If subprogram call or macro call command exists in the finished shape program, these commands will also be executed.
- (10) Except for thread cutting cycles, operation stops at the end (start) point of each block in the single block mode.
- (11) Note that, depending on whether the sequence No. or program No. is designated, the next block after the completion of the G71, G72 or G73 command will differ.

(a) When a sequence No. is designated	(b) When a program No. is designated
<p>The next block is the one designated by Q.</p> <pre style="margin-left: 40px;"> N100 G71P200 Q500 U_W_... ; N200 N300 } N400 } N500 } N600 </pre> <p>Operation moves to the N600 block upon completion of the cycle.</p>	<p>The next block is the next block of the cycle command.</p> <pre style="margin-left: 40px;"> N100 G71A100 U_W_... ; N200 N300 N400 </pre> <div style="border: 1px solid black; padding: 5px; margin-left: 100px; width: fit-content;"> <p>O100 N10X100.Z50. ; N20 : : :</p> </div> <p>Operation moves to the N200 block upon completion of the cycle.</p>

(12) The next block after the completion of the G70 command is the next block of the command block.

```

:
N100 .... ;
N200 .... ;
N300 .... ;
N400 .... ;
N500 .... ;
:
N1000 G70 P200 Q500 ; (or G70A100 ; )
N1100 .... ;
:
    
```

Operation moves to the N1100 block upon completion of the G70 command.

(13) It is possible to apply a manual interruption while a compound type fixed cycle for turning machining command (G70 to G76) is being executed. However, upon completion of the interrupt, the tool must first be returned to the position where the interrupt was applied and then the compound type fixed cycle for turning machining must be restarted.

If it is restarted without the tool having been returned, all subsequent movements will deviate by an amount equivalent to the manual interruption amount.

(14) Compound type fixed cycle for turning machining commands are unmodal. So they must be issued every time they are required.

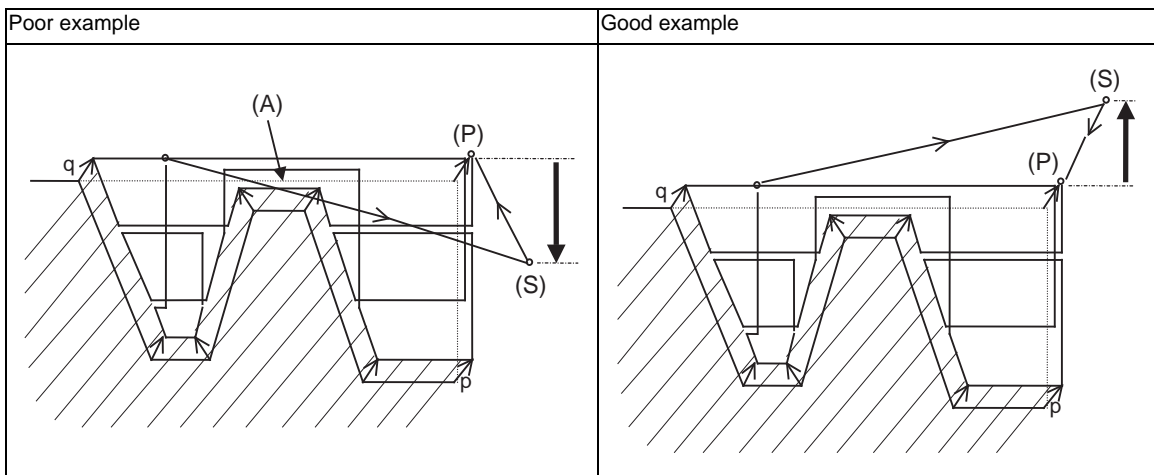
(15) Program error (P203) will occur with the G71 and G72 commands when there is no further movement of the Z axis in the second block or the Z axis has moved in the opposite direction because of tool nose radius compensation.

(16) A program error (P204) will occur if the finished shape program is issued in a same program with G70 to G73 and both P and Q are not designated.

Note that if A, P and Q are all not designated in G71 to G73, this will be interpreted as the first block of the compound type fixed cycle for turning machining I, and an error will not occur.

(17) Set the cycle start point in the final block of the compound type fixed cycle for turning machining I so that the tool retracts from the workpiece cutting face. This is necessary to prevent the cutter from interfering with the convex sections of the workpiece.

The figure below shows a poor example. The position (A) is located higher than the cycle start point and an interference will occur in the final block.



(S) Cycle command point

(18) Pocket machining is an option. If the option is not provided, H1 will operate as H0 even if designated.

(19) The common variable settings are ignored in the finished shape program. The setting values before compound type fixed cycle for turning machining I command will be valid.

(20) If there is a circular command in the first movement block for the finished shape when pocket machining is OFF, a program error (P203) will occur.

13.4 Compound Type Fixed Cycle for Turning Machining (MITSUBISHI CNC special format) ; G71,G73,G74,G76



Function and purpose

This function enables to perform a prepared fixed cycle by commanding a program in a block. When using the MITSUBISHI CNC special format (#1265 ext01/bit0 ON), some addresses differ from the normal format. With the normal format, G71 to G76 are commanded in two blocks. However with the MITSUBISHI CNC special format, these are commanded in one block. This section gives details on the formats different from the normal ones.

Refer to section "13.3 Compound type fixed cycle for turning machining" for details on the compound type fixed cycle for turning machining.



Command format

Longitudinal rough cutting cycle, face rough cutting cycle

G71 P__ Q__ U__ W__ D__ F__ S__ T__ ;

P	Finished shape start sequence No.
Q	Finished shape end sequence No.
U	X-axis direction finishing allowance
W	Z-axis direction finishing allowance
D	Cut amount
F	Cutting speed
S	Spindle speed
T	Tool command

(Same for G72)

Formed material rough cutting cycle

G73 P__ Q__ U__ W__ I__ K__ D__ F__ S__ T__ ;

P	Finished shape start sequence No.
Q	Finished shape end sequence No.
U	X-axis direction finishing allowance
W	Z-axis direction finishing allowance
I	X-axis direction cutting allowance
K	Z-axis direction cutting allowance
D	Number of divisions
F	Cutting speed
S	Spindle speed
T	Tool command

Face cut-off cycle, longitudinal cut-off cycle

G74 X(U)___ Z(W)___ I___ K___ F___ D___ ;

X(U)	X axis slotting end point coordinate
Z(W)	Z axis slotting end point coordinate
I	Tool shift amount (X/U axis direction movement amount)
K	Cut amount (Z/W axis direction cut amount)
F	Cutting speed
D	Tool escape amount at cutting base

(Same for G75)

Compound thread cutting cycle

G76 X(U)___ Z(W)___ I___ K___ D___ F___ A___ Q___ P___ ;

X(U)	X-axis end point coordinate of thread
Z(W)	Z-axis end point coordinate of thread
I	Radius value at thread section
K	Thread height
D	1st cut amount
F	Thread lead
A	Thread angle
Q	Thread cutting start shift angle (0.001 to 360.000 [°])
P	Zigzag thread cutting at P2 command [For specific models only (No specification for M700V/M70V Series)]



Detailed description

Check of command format

This checks whether the normal command format is being used with the MITSUBISHI CNC special format.

[When normal format is selected (#1265 ext01/bit0 OFF)]

- (1) Check of command format
When the format check selection parameter (#1222 aux06/bit5) is set to "0", a program error (P33) will occur in the following cases.
 - There is no command to the 1st block.
 - A MITSUBISHI CNC special format is commanded.
- (2) Check of command format
A program error (P32) will occur in the following cases.
 - When address I, K or D is commanded in the G71, G72, G73, G74 or G75 block.
 - When address I, K, D or A is commanded in the G76 block.

[When MITSUBISHI CNC special format is selected (#1265 ext01/bit0 ON)]

- (1) Check of command format
A program error (P33) will occur if a normal format is commanded.
- (2) Check of command format
A program error (P32) will occur in the following cases.
 - When address R or A is commanded in the G71, G72 or G73 block.
 - When address P, Q or R is commanded in the G74 or G75 block.
 - When address R is commanded in the G76 block.

Comparison of MITSUBISHI CNC special format and normal format

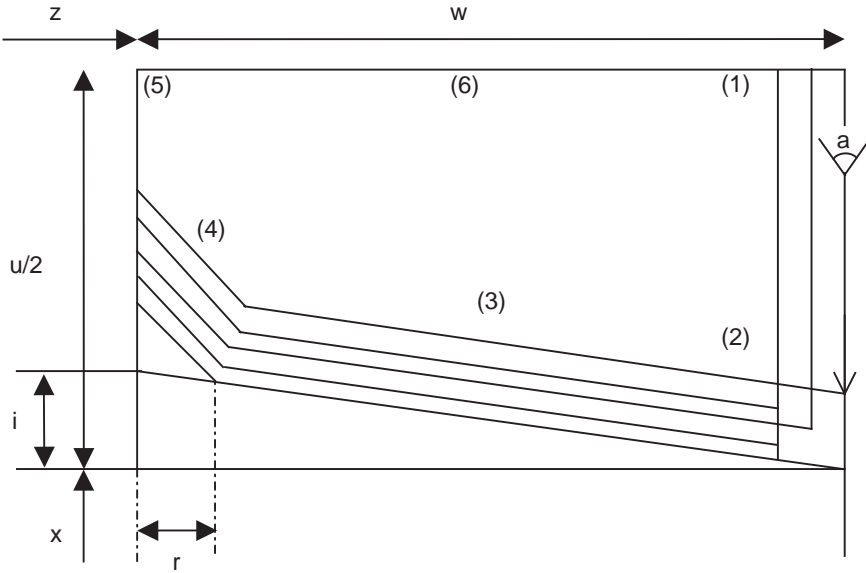
While the normal format is commanded using two blocks, the MITSUBISHI CNC special format requires only one block. Some addresses differ, and some are designated with the parameters.

Function	MITSUBISHI CNC special format	Normal format	Difference from normal format
Longitudinal Rough Cutting Cycle I	G71 P Q U W D F S T;	(1)G71 U R; (2)G71 A P Q U W F S T;	Cut amount Designate with U (1) -> D Retract amount R (1) -> Parameter (#8052 PULL UP) No A command (finished shape program No.)
Face Rough Cutting Cycle I	G72 P Q U W D F S T;	(1)G72 W R; (2)G72 A P Q U W F S T;	Cut amount Designate with W (1) -> D Retract amount R (1) -> Parameter (#8052 PULL UP) No A command (finished shape program No.)
Formed Material Rough Cutting Cycle	G73 P Q U W I K D F S T;	(1)G73 U W R; (2)G73 A P Q U W F S T;	X-axis direction cutting allowance U(1) -> I Z-axis direction cutting allowance W(1) -> K Number of divisions R(1) -> D No A command (finished shape program No.)
Face Cut-Off Cycle	G74 X Z I K F D; or G74 U W I K F D;	(1)G74 R; (2)G74 X Z P Q R F; or (1)G74 R; (2)G74 U W P Q R F;	X axis direction movement amount P(2) -> I Z axis direction cut amount Q(2) -> K Escape amount at cutting base R(2) -> D Return amount R (1) -> parameter (#8056 G74 RETRACT)
Outer Diameter Cut-Off Cycle	G75 X Z I K F D; or G75 U W I K F D;	(1)G75 R; (2)G75 X Z P Q R F; or (1)G75 R; (2)G75 U W P Q R F;	X axis direction cut amount P(2) -> I Z axis direction movement amount Q(2) -> K Escape amount at cutting base R(2) -> D Return amount R (1) -> parameter (#8056 G74 RETRACT)
Compound thread cutting cycle	G76 X_ Z_ I_ K_ D_ F_ A_ Q_ ;	(1)G76 Pmra R_ ; (2)G76 X_ Z_ R_ P_ Q_ F_ ;	Radius value at thread section R(2) -> I Thread height P(2) -> K 1st cut amount Q(2) -> D (Q in the MITSUBISHI CNC special format is the thread cutting start shift angle) Thread angle P(1)a -> A Thread cutting P(1)r -> parameter (#8014 CDZ-VALE) Number of finish cuts P (1) m -> parameter (#8058 TIMES) Finishing allowance R (1) -> parameter (#8057 G76 LAST-D)

Zigzag thread cutting [For specific models only (No specification for M700V/M70V Series)]

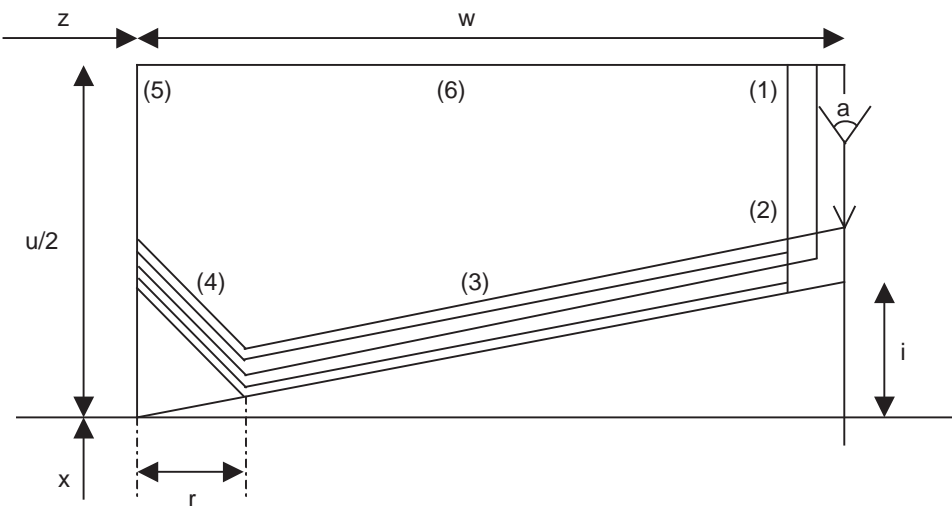
By commanding P2 in the G76 block of the compound thread cutting cycle, zigzag thread cutting with a constant cut amount can be performed.

When li (taper height component at thread section) is negative

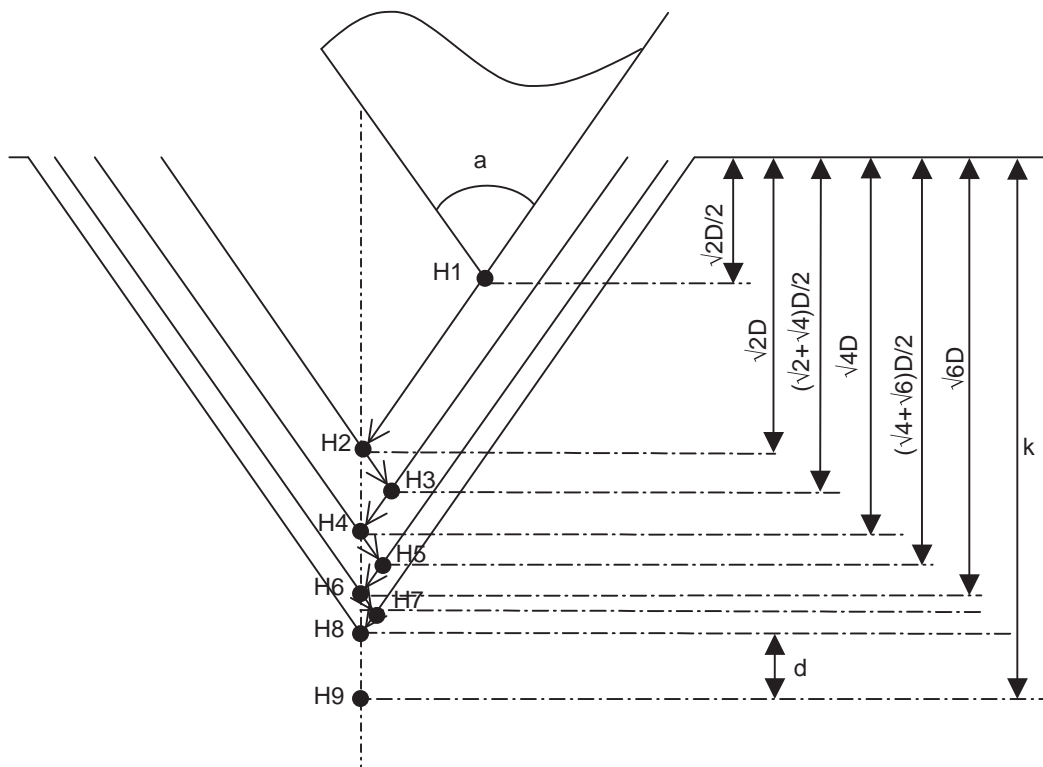


i : Taper height component for thread
 r : Chamfering amount (Set by #8014 CDZ-VALE)

When li (taper height component at thread section) is positive



i : Taper height component (radius value) for thread
 r : Chamfering amount (Set by #8014 CDZ-VALE)



a : Thread angle
 k : Thread height
 d : Finishing allowance (Set by "#8057 LAST-D")
 (Number of finish cuts are set by "#8058 TIMES")

The cut amount increases at a set rate.

Relational expression of cut amount

$$H1 = H2 * 1/2 = (\sqrt{2} * D) * 1/2$$

$$H2 = \sqrt{2} * D$$

$$H3 = H2 + (H4 - H2) * 1/2 = (\sqrt{2} + \sqrt{4}) * D * 1/2$$

$$H4 = \sqrt{4} * D$$

$$H5 = H4 + (H6 - H4) * 1/2 = (\sqrt{4} + \sqrt{6}) * D * 1/2$$

$$H6 = \sqrt{6} * D$$

:

:

$$Hn = (\sqrt{(n-1)} + \sqrt{(n+1)}) * \Delta d / 2 \quad (n = 1, 3, 5, 7, \dots)$$

$$Hn = \sqrt{n} * \Delta d \quad (n = 2, 4, 6, 8, \dots)$$



Restrictions and precautions

- (1) Always designate P and Q in G71 to G73. A program error (P204) will occur if one or both of them are not designated.
- (2) The thread cutting start shift angle is not modal. If there is no Q command with G76, this will be handled as "Q0".
- (3) If a value exceeding 360.000 is command in G76 Q, a program error (P35) will occur.
- (4) G76 cuts one row with one cycle. To cut two rows, change the Q value, and issue the same command.

13.5 Fixed Cycle for Drilling



Function and purpose

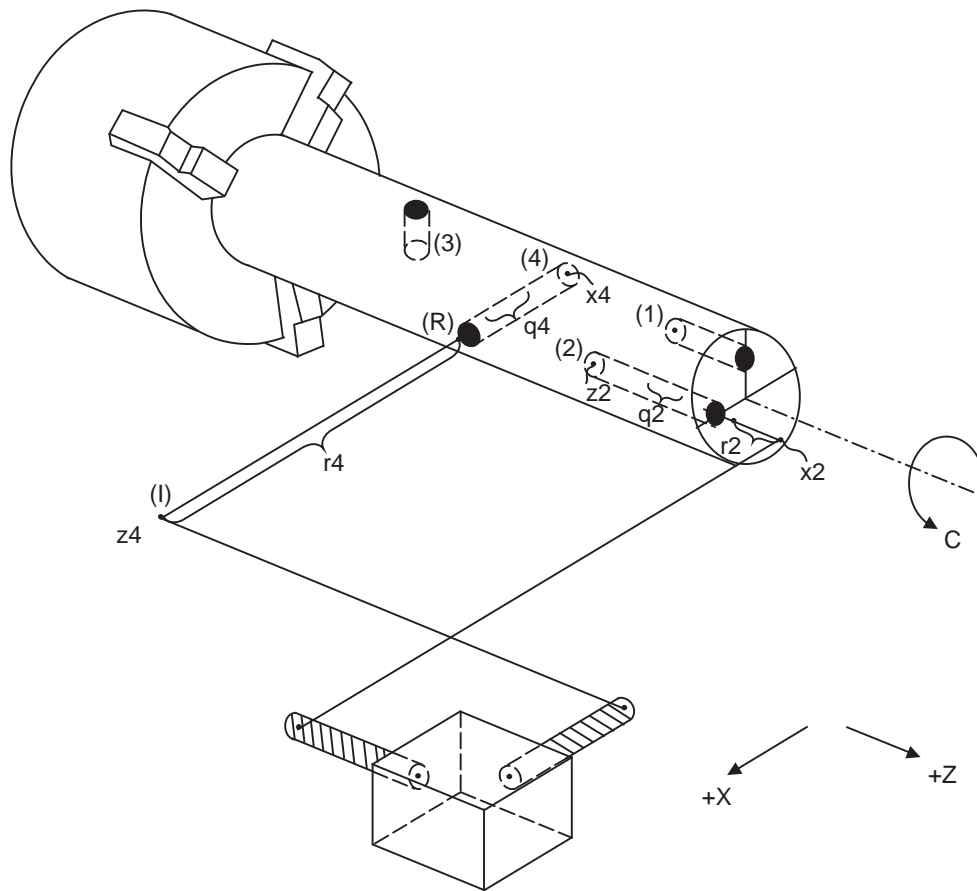
These fixed cycles are used to perform prepared working sequences of machining programs such as positioning, hole drilling, boring and tapping in a block. When performing a same machining repeatedly, it can be executed by commanding only the axis position. The types of fixed cycles are listed below.

(Note) The X axis is designated as the hole drilling axis by longitudinal hole drilling fixed cycle. But the Y axis can also be designated as the hole drilling axis by using the longitudinal hole drilling axis selection function. "Type II" in the table explains details for using the longitudinal hole drilling axis selection function.

G code	Hole drilling axis	Drilling Hole drilling start	Operation at hole bottom	Return operation	Retract at high speed	Application	Type II	
							Longitudinal hole drilling axis selection signal ON/OFF	Hole drilling axis
G80	-	-	-	-	-	Cancel	-	-
G83	Z	Cutting feed Intermittent feed	In-position check, Dwell	Rapid traverse	Possible	Face Deep Hole Drilling Cycle I	ON	Z
							OFF	
G84 (G84.1)	Z	Cutting feed	In-position check, Dwell, Spindle reverse rotation	Cutting feed	-	Face Tapping Cycle (Face Reverse Tapping Cycle) Pecking tapping cycle Deep-hole tapping cycle	ON	Z
							OFF	
G85	Z	Cutting feed	In-position check, Dwell	Cutting feed	-	Face Boring Cycle	ON	Z
							OFF	
G87	X	Cutting feed Intermittent feed	In-position check, Dwell	Rapid traverse	Possible	Deep Hole Drilling Cycle I	ON	Y
							OFF	X
G88 (G88.1)	X	Cutting feed	In-position check, Dwell, Spindle reverse rotation	Cutting feed	-	Longitudinal tapping cycle (Longitudinal reverse tapping cycle) Pecking tapping cycle Deep-hole tapping cycle	ON	Y
							OFF	X
G89	X	Cutting feed	In-position check, Dwell	Cutting feed	-	Longitudinal boring cycle	ON	Y
							OFF	X
G83.2	Z/X	Cutting feed Intermittent feed	In-position check, Dwell	Rapid traverse	Possible	Deep hole drilling cycle 2	ON	Z/X
							OFF	

A fixed cycle mode can be canceled by G80 command and other hole machining modes or G command in the 01 group. At the same time, various other data will also be cleared to zero.

The hole drilling axes and the positioning for the fixed cycle for drilling are shown in the outline drawing below.



(I) Initial point

(R) R point

- (1) G83 Xx1 Cc1 Zz1 Rr1 Qq1 Pp1 Ff1 Kk1 ; Face deep hole drilling cycle
- (2) G83 Xx2 Cc2 Zz2 Rr2 Qq2 Pp2 Ff2 Kk2 ;
- (3) G87 Zz3 Cc3 Xx3 Rr3 Qq3 Pp3 Ff3 Kk3 ; Longitudinal deep hole drilling cycle
- (4) G87 Zz4 Cc4 Xx4 Rr4 Qq4 Pp4 Ff4 Kk4 ;

During the hole drilling cycle, the C axis (spindle) is clamped so that it does not move.

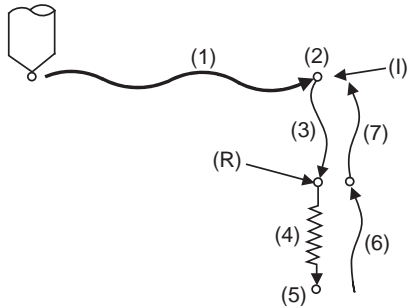
Commands M03, M04 and M05 (forward rotation, reverse rotation and stop) operate to the rotary tools.



Detailed description

Basic operations of fixed cycle for drilling

The actual operation consists of the following seven movements.



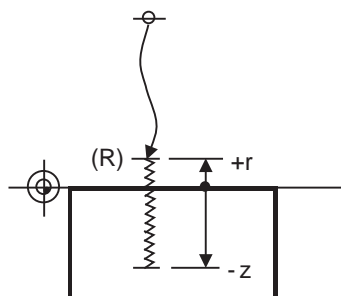
(I) Initial point

(R) R point

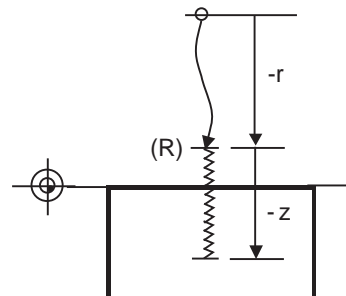
- (1) This denotes the positioning (by rapid traverse) to the X (Z) and C axis initial point.
If ",I" (Positioning axis in-position width) is designated, in-position check will be carried out after the block is completed.
- (2) This will be output if the M code for C-axis clamping is issued.
- (3) This denotes the positioning (by rapid traverse) to the R point.
- (4) Hole machining is conducted by cutting feed.
If ",J" (Hole drilling axis in-position width) is designated, in-position check will be carried out after the block completes. Note that in case of deep drilling cycle 1 or 2, in-position check will not be carried out for hole drillings in the program. It will be carried out at designated hole bottom position (the last hole drilling).
- (5) This operation takes place at the hole bottom position and it differs according to the fixed cycle modes including the rotary tool reverse rotation (M04), rotary tool forward rotation (M03) and dwell. .
- (6) This denotes the returning to the R point.
- (7) This denotes the returning to the initial point by rapid traverse.
(Operations 6 and 7 may become a single operation depending on the fixed cycle mode.)
- (Note 1) In the synchronous tapping command, the position of an in-position check will differ depending on the parameter setting.
(Refer to "Tapping Cycle; G84 (Face)/G88 (Longitudinal), Reverse Tapping Cycle; G84.1 (Face)/G88.1 (Longitudinal)"(General format))
- (Note 2) Whether the fixed cycle is to be completed at operation 6 or 7 can be selected by G98/G99 commands. (Refer to "Initial point and R point level return; G98, G99")

Difference between absolute value command and incremental value command

For absolute value



For incremental value



(R) R point

13.5.1 Face Deep Hole Drilling Cycle 1 (Longitudinal deep hole drilling cycle 1) ; G83 (G87)



Command format

Face deep hole drilling cycle 1

G83 X/U__ C/H__ Z/W__ Rr Qq Pp Ff Kk Mm ;

X/U C/H	Designation of hole position initial point (absolute/incremental value) ... Data for positioning X and C axes
Z/W	Designation of hole bottom position (absolute/incremental value from R point) (modal)
Rr	Designation of R point (incremental value from initial point) (sign ignored) (modal)
Qq	Designation of cut amount for each cutting pass with G83 (G87). Always incremental value, radius value (sign ignored)
Pp	Designation of dwell time at hole bottom point. (modal) Relationship between time and designated value is same as for G04 designation.
Ff	Designation of feed rate for cutting feed (modal)
Kk	Designation of number of repetitions, 0 to 9999 (standard value = 1)
Mm	Designation of miscellaneous command

Longitudinal deep hole drilling cycle 1

G87 Z/W__ C/H__ X/U__ Rr Qq Pp Ff Kk Mm ;

Z/W C/H	Designation of hole position initial point (absolute/incremental value) ... Data for positioning Z and C axes
X/U	Designation of hole bottom position (absolute/incremental value from R point) (modal)
Rr	Same as face deep hole drilling cycle 1
Qq	Same as face deep hole drilling cycle 1
Pp	Same as face deep hole drilling cycle 1
Ff	Same as face deep hole drilling cycle 1
Kk	Same as face deep hole drilling cycle 1
Mm	Same as face deep hole drilling cycle 1

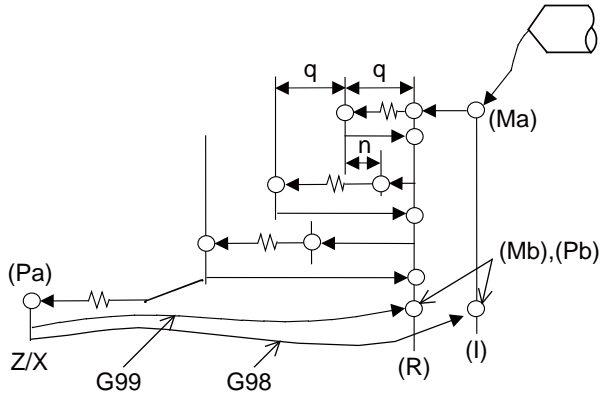
Precautions

- (1) For the longitudinal deep hole drilling cycle 1(G87), designate Z/W to the hole position initial point and X/U to the hole bottom position.
- (2) The designation of the hole position initial point is unmodal. When G83(G87) command is to be executed continuously, designate them block by block.
- (3) Q command is unmodal. Designate them block by block.
- (4) K command is unmodal. When K command is not issued, it is regarded as K1. When K0 is designated, the hole machining data are stored in the memory but no holes will be machined.



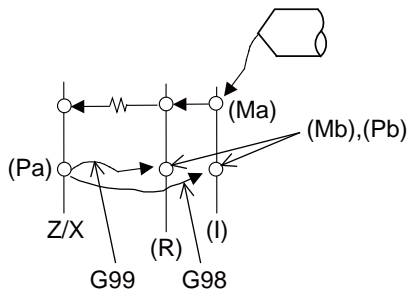
Detailed description

When the Q command is issued (deep hole drilling)



- (1) Return amount "d" is set by the parameter (#8013 G83 n). The tool returns at rapid traverse.
- (2) (Ma) ... The M code (Mm) is output when there is a C-axis clamping M code command (Mm).
- (3) (Mb) ... The C-axis unclamping M code (C-axis clamp M code + 1 = Mm + 1) is output when there is a C-axis clamping M code command (Mm).
- (4) (Pa) ... Dwell is performed for the duration equivalent to the time designated by P.
- (5)(Pb) ' ... After the C-axis unclamping M code (Mm + 2) is output, dwell is performed for the duration equivalent to the time set by the parameter (#1184 clmp_D).

When the Q command is not present (drilling)



G83 (G87) X(z) C Z(x) R r Pp F f Kk Mm ;
 See "When the Q command is present (deep hole drilling)" for details on (Ma),(Mb),(Pa),(Pb).



Precautions

Refer to "Precautions When Using a Fixed Cycle for Drilling".

13.5.2 Face Tapping Cycle (Longitudinal tapping cycle) / Face Reverse Tapping Cycle (Longitudinal reverse tapping cycle) ; G84 (G88) / G84.1 (G88.1)



Command format

Face tapping cycle

G84(G84.1) X/U__ C/H__ Z/W__ Rr1 Pp Ff Kk Dd Ss1 ,Ss2 ,Rr2 Mm ;

G84(G84.1)	G84 Face tapping cycle mode G84.1 Face tapping cycle mode (the tap rotation direction is reversed)
X/U C/H	Designation of hole position initial point (absolute/incremental value) ... Data for positioning X and C axes
Z/W	Designation of hole bottom position (absolute/incremental value from R point) (modal)
Rr1	Designation of R point (incremental value from initial point) (sign ignored) (modal)
Pp	Designation of dwell time at hole bottom point. Relationship between time and designated value is same as for G04 designation. (modal)
Ff	During synchronous tapping: Designation of drilling axis feed amount (tapping pitch) per spindle revolution (modal) During asynchronous tapping: Designation of the feedrate for cutting feed (modal)
Kk	Designation of number of repetitions, 0 to 9999 (standard value = 1)
Dd	Designation of spindle used in tapping cycle (modal) (Setting range: 1 to No. of spindles)
Ss1	Designation of spindle rotation speed (During synchronous tapping, the S command in the command format with a spindle No. (example: "S2 = 2000") is ignored.)
,Ss2	Designation of spindle rotation speed at return (Valid only during synchronous tapping mode. This setting is ignored during other modes.)
,Rr2	Synchronization method selection (r2 = 1: synchronous, r2 = 0: asynchronous) (When omitted, the mode will follow the setting of parameter "#8159 Synchronous tap".)
Mm	Designation of miscellaneous command

Longitudinal tapping cycle

G88(G88.1) Z/W__ C/H__ X/U__ Rr1 Pp Ff Kk Dd Ss1 ,Ss2 ,Rr2 Mm ;

G88(G88.1)	G88 Longitudinal tapping cycle mode G88.1 Longitudinal reverse tapping cycle mode (the tap rotation direction is reversed)
Z/W C/H	Designation of hole bottom position (absolute/incremental value from R point) ... Data for positioning Z and C axes
X/U	Designation of hole bottom position (absolute/incremental value from R point) (modal)
Rr1	Same as face tapping cycle.
Pp	Same as face tapping cycle.
Ff	Same as face tapping cycle.
Kk	Same as face tapping cycle.
Dd	Same as face tapping cycle.
Ss1	Same as face tapping cycle.
,Ss2	Same as face tapping cycle.
,Rr2	Same as face tapping cycle.
Mm	Same as face tapping cycle.

Pecking Tapping Cycle/Deep-hole Tapping Cycle

G84(G88, G84.1, G88.1) X/U__ C/H__ Z/W__ Rr1 Qq Ff (Ee) Pp Ss1 ,Ss2 ,li ,Jj ,Rr2 Dd Kk Mm ;

G84 (G88, G84.1, G88.1)	G84 Face tapping cycle mode G88 Longitudinal tapping cycle mode G84.1 Face tapping cycle mode (the tap rotation direction is reversed) G88.1 Longitudinal reverse tapping cycle mode (the tap rotation direction is reversed)
X/U C/H (For G84, G84.1)	Designation of hole position initial point (absolute/incremental value) ... Data for positioning X and C axes
Z/W (For G84, G84.1)	Designation of hole bottom position (absolute/incremental value from R point) (modal)
Z/W C/H (For G88, G88.1)	Designation of hole bottom position (absolute/incremental value from R point) ... Data for positioning Z and C axes
X/U (For G88, G88.1)	Designation of hole bottom position (absolute/incremental value from R point) (modal)
Rr1	Designation of R point (incremental value from initial point) (sign ignored) (modal)
Qq	Specify the depth of cut per pass. Always specify using a radial increment (sign ignored)(modal)
Ff	During synchronous tapping: Designation of drilling axis feed amount (tapping pitch) per spindle revolution (modal) During asynchronous tapping: Designation of the feedrate for cutting feed (modal)
Ee	Specify the number of tap threads per 1-inch feed of Z axis
Pp	Designation of dwell time at hole bottom point. Relationship between time and designated value is same as for G04 designation. (modal)
Ss1	Designation of spindle rotation speed (During synchronous tapping, the S command in the command format with a spindle No. (example: "S2 = 2000") is ignored.)
,Ss2	Designation of spindle rotation speed at return (Valid only during synchronous tapping mode. This setting is ignored during other modes.)
,li	Specify the positioning axis in-position width
,Jj	Specify the drilling axis in-position width
,Rr2	Synchronization method selection (r2 = 1: synchronous, r2 = 0: asynchronous) (When omitted, the mode will follow the setting of parameter "#8159 Synchronous tap".)
Dd	Designation of spindle used in tapping cycle (modal) (Setting range: 1 to No. of spindles)
Kk	Designation of number of repetitions, 0 to 9999 (standard value = 1)
Mm	Designation of miscellaneous command

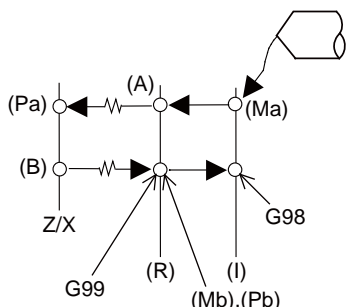
Precautions

- (1) The designation of the hole position initial point is unmodal. When tapping cycle command is to be executed continuously, designate them block by block.
- (2) If a value other than zero is specified to address Q while the option of pecking tapping cycle or deep-hole tapping cycle is ON, either pecking or deep-hole tapping cycle is executed instead of normal tapping cycle.
 When the parameter "#1272 ext08/bit4" is "0" ... Pecking tapping cycle
 When the parameter "#1272 ext08/bit4" is "1" ... Deep-hole tapping cycle
- (3) K command is unmodal. When K command is not issued, it is regarded as K1. When K0 is designated, the hole machining data are stored in the memory but no holes will be machined.
- (4) Set a minus value to D command when designating the reverse tapping cycle by G84/G88 with "#1309 Gtype" setting.
- (5) When multiple-spindle control II is valid, the spindle is selected from the PLC so do not issue a D command. A program error (P32) will occur if commanded.
- (6) When D command is omitted, the spindle designated with the spindle selection command will be selected.
- (7) ,S command is held as modal information.
 When the value is smaller than the spindle rotation speed (S command), the value of the spindle speed is valid even at the return.
 When the spindle speed at the return is not "0", the value of tap return override (#1172 tapovr) is invalid.



Detailed description

Normal tapping cycle(When Q is not designated)



(Ma) The M code (Mm) is output when there is a C-axis clamping M code command (Mm).

(Mb) The C-axis unclamping M code (C-axis clamp M code + 1 = Mm + 1) is output when there is a C-axis clamping M code command (Mm).

(Pa) Dwell is performed for the duration of time commanded by P.

(Pb) After the C-axis unclamping M code (Mm + 2) is output, dwell is performed for the duration of time set by the parameter (#1184 clmp_D).

(R) R point

(I) Initial point

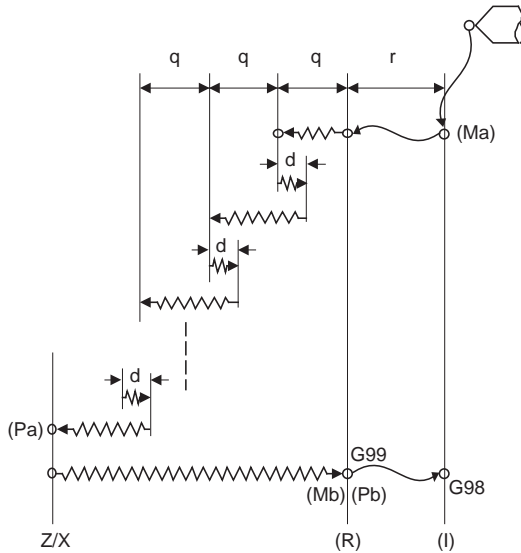
With the reverse tapping cycle G84.1 (G88.1), the tapping rotation direction is reversed.

- (1) When G84(G88) is executed, the override will be canceled and the override will automatically be set to 100%.
- (2) 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(G88) execution, the block stops after returning operation is completed.
- (3) During single block operation, the axis will not stop at the turning point of tapping cycle.
- (4) During the G84 (G88) modal, the "Tapping" NC output signal will be output.
- (5) During the G84 (G88) synchronous tapping modal, the M3, M4, S code, etc. will not be output.
- (6) The tool stops at the R point and the rotary tool forward rotation signal is output. (A)
- (7) The rotation of the rotary tool is reversed at the hole bottom and tapping is performed. (B)
- (8) The fixed cycle subprograms should be edited if the rotary tool stop (M05) command is required before the rotary tool reverse (M04) or forward rotation (M03) signal is output.
- (9) Whether to set the reverse tapping cycle G codes to G84.1/G88.1 or G84/G88 (address D value is minus) is determined according to the basis specification parameter "#1309 Gtype".
- (10) When it is interrupted by such as the emergency stop during the tapping cycle, a tool can be taken out from the workpiece after performing the tap retract.

Pecking tapping cycle (When Q command is designated #1272 ext08/bit4=0)

In deep-hole tapping, the load applied to the tool can be reduced by designating the depth of cut per pass and cutting the workpiece by a multiple number of times.

In the 2nd and subsequent cuts, the tool retracts with reverse tap rotation by the retract distance (d) specified by parameter from the previous cut position. And then it cuts the depth (q + d) with forward tap rotation. (In G84.1/G88.1, forward tapping and reverse tapping are reversed.)



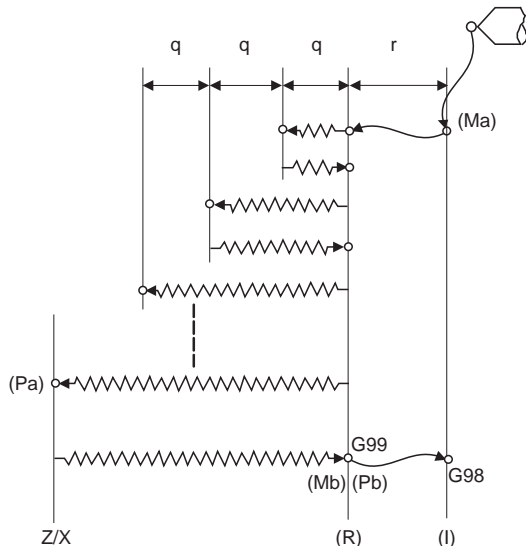
- (Ma) The M code (Mm) is output when there is a C-axis clamping M code command (Mm).
- (Mb) The C-axis unclamping M code (C-axis clamp M code + 1 = Mm + 1) is output when there is a C-axis clamping M code command (Mm).
- (Pa) Dwell is performed for the duration of time commanded by P.
- (Pb) After the C-axis unclamping M code (Mm + 2) is output, dwell is performed for the duration of time set by the parameter (#1184 clmp_D).
- (R) R point
- (I) Initial point
- d Retract amount (parameter "#8018 G84/G74 n")

- (1) When G84 (G88/G84.1/G88.1) is executed, the override will be canceled and the override will automatically be set to 100% in cutting operation. During a pulling operation from the hole bottom to the R point, the override set by the parameter "#1172 tapovr" will be enabled. Note that, however, it will be disabled during a return operation by "return amount d".
- (2) 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 (G88) execution, the tool does not stop immediately during cutting or returning, but stops after completing an R point return.
- (3) During single block operation, the tool does not stop when cutting or returning, but stops after completing an R point/initial point return.
- (4) During the G84 (G88/G84.1/G88.1) mode, the NC signal "Tapping" will be output.
- (5) During the G84 (G88/G84.1/G88.1) synchronous tapping mode, the M3, M4, M5, or S code will not be output.
- (6) If the command value of F becomes extremely small such as around "F < 0.01mm/rev" during synchronous tapping, the spindle rotates unsmoothly. So make sure to command a value larger than 0.01mm/rev. The unit of F can be selected between mm/rev and mm/min.
- (7) 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.
- (8) If the operation is interrupted by a cause such as an emergency stop or reset during the pecking tapping cycle, a tap retract is executed when the tap retract signal is input.
- (9) When the reference position return signal is input during the pecking tapping cycle, a tap retract is carried out, and a reference position return will be executed from the end point of the tap retract.

Deep-hole tapping cycle (When Q command is designated "#1272 ext08/bit4" = 1)

In deep-hole tapping, the load applied to the tool can be reduced by designating the depth of cut per pass and cutting the workpiece by a multiple number of times.

When repeating cutting by the depth of cut (q) in forward tapping, the tool is returned to the R point in reverse tapping after every cutting. (In G84.1/G88.1, forward tapping and reverse tapping are reversed.)



(Ma) The M code (Mm) is output when there is a C-axis clamping M code command (Mm).

(Mb) The C-axis unclamping M code (C-axis clamp M code + 1 = Mm + 1) is output when there is a C-axis clamping M code command (Mm).

(Pa) Dwell is performed for the duration of time commanded by P.

(Pb) After the C-axis unclamping M code (Mm + 2) is output, dwell is performed for the duration of time set by the parameter (#1184 clmp_D).

(R) R point

(I) Initial point

- (1) When G84 (G88/G84.1/G88.1) is executed, the override will be canceled and the override will automatically be set to 100% in cutting operation. And the override set by 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)
- (2) 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 (G88) execution, the tool does not stop immediately during cutting or returning, but stops after completing an R point return.
- (3) During single block operation, the tool does not stop when cutting or returning, but stops after completing an R point/initial point return.
- (4) During the G84 (G88/G84.1/G88.1) mode, the NC signal "Tapping" will be output.
- (5) During the G84 (G88/G84.1/G88.1) synchronous tapping mode, the M3, M4, M5 or S code will not be output.
- (6) If the command value of F becomes extremely small such as around "F < 0.01mm/rev" during synchronous tapping, the spindle rotates unsmoothly. So make sure to command a value larger than 0.01mm/rev. The unit of F can be selected between mm/rev and mm/min.
- (7) 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.
- (8) 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.
- (9) 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.

Synchronous/asynchronous tap selection

- (1) Selecting with a program command

Tap cycle ",R0/1" command

G84 (G88) Xx1 Cc1 Zz1 Rr1 Pp1 Ff1 Kk1 Dd1 Ss1 ,Ss2 ,Rr2 Mm1 ;

When r2 = 1, the synchronous tapping mode will be applied, and when r2 = 0, the asynchronous tapping mode will be applied.

- (2) Selecting with parameters

User parameters

#	Item	Details	Setting range
8159	Synchronous tap	0: Handles the tapping cycles as the tapping cycles with a floating tap chuck. (asynchronous tapping) 1: Handles the tapping cycles as the tapping cycles without a floating tap chuck.(Synchronous tapping)	0/1

The tapping command will be synchronous tapping cycle when this parameter is turned on.

- (3) Selecting with M function

Basic specification parameters

#	Item	Details	Setting range
1272(PR)	ext08 bit1	M-function synchronous tapping cycle valid.	0 : Invalid 1 : Valid

Synchronous tapping can be selected with the M function when this parameter is ON.

This parameter requires the CNC to be turned OFF after the settings. Turn the power OFF and ON to enable the parameter settings.

Basic 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.

(Note1) Do not use M00, 01, 02, 30, 96, 97, 98 and 99.

(Example)

M29 ; (M command synchronous tapping)

G84 Z50. R20. F2.; or G84 Z50. R20. F2. M29 ;

The selection of synchronous and asynchronous tapping by the three items above will follow the combination shown below.

	Combination											
	0	0	0	0	1	1	1	1	No command			
Program command (,R0/1)	0	0	0	0	1	1	1	1				
#8159 Synchronous tap	0	0	1	1	0	0	1	1	0	0	1	1
M function code (M**)	x	○	x	○	x	○	x	○	x	○	x	○
Synchronous/asynchronous selection	A	A	A	A	S	S	S	S	A	S	S	S

x : Does not command

A : Asynchronous tap

○ : Commands

S : Synchronous tapping in-position

Canceling synchronous tapping selection

To cancel the synchronous tapping selection, command reset, G80 (fixed cycle for drilling cancel), 01 group G code or other fixed cycle G codes.

Feed per minute command of the synchronous tapping

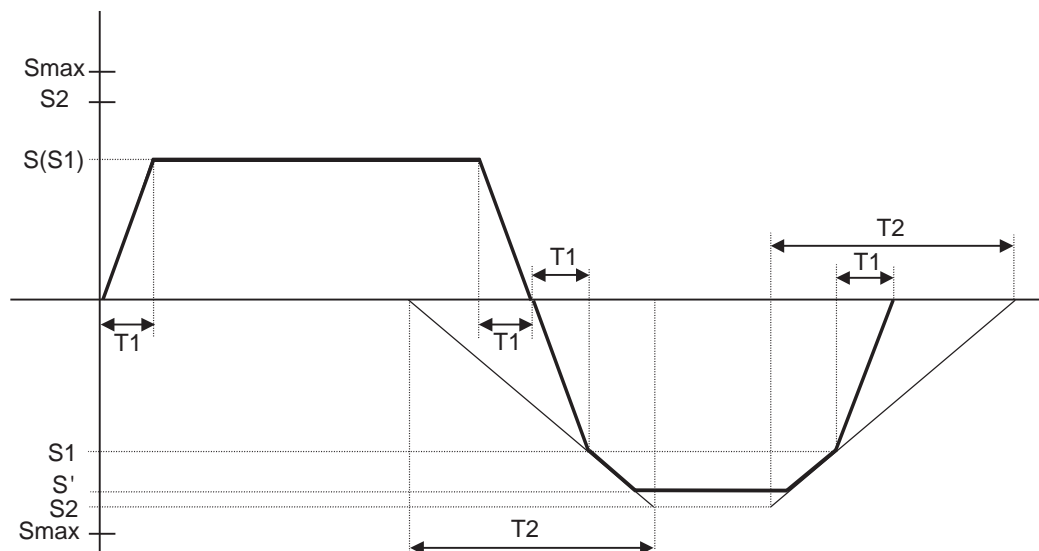
Enable the feed per minute command of the synchronous tapping by the setting of parameter "#1268 ext04/bit2". When this parameter is valid, G94 and G95 modal will be applied.

	During G94 modal (feed per minute)	During G95 modal (feed per revolution)
#1268/bit2 = ON	Feed per minute	Feed per revolution
#1268/bit2 = OFF	Feed per revolution	Feed per revolution

Spindle acceleration/deceleration pattern during synchronous tapping

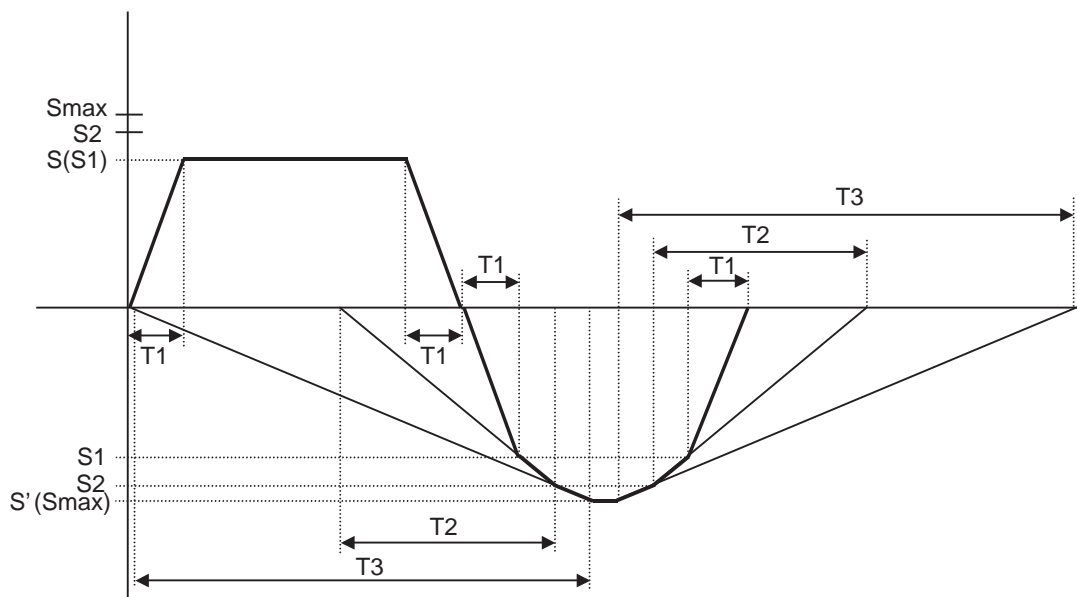
This function enables to make spindle acceleration/deceleration pattern closer to that of the speed loop by dividing the spindle and drilling axis acceleration/deceleration pattern into up to three stages during synchronous tapping. The acceleration/deceleration pattern can be set up to three stages for each gear. 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 \leq synchronous tapping changeover spindle rotation speed 2



- S : Command spindle rotation speed
 S' : Spindle rotation speed during return
 S1 : Tapping rotation speed (spindle basic specification parameters #3013 to #3016)
 S2 : Synchronous tapping changeover spindle rotation speed 2 (spindle basic specification parameters #3037 to #3040)
 Smax : Maximum rotation speed (spindle basic specification parameters #3005 to #3008)
 T1 : Tapping time constant (spindle basic specification parameters #3017 to #3020)
 T2 : Synchronous tapping changeover time constant 2 (spindle basic 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 basic specification parameters #3013 to #3016)
- S2 : Synchronous tapping changeover spindle rotation speed 2 (spindle basic specification parameters #3037 to #3040)
- Smax : Maximum rotation speed (spindle basic specification parameters #3005 to #3008)
- T1 : Tapping time constant (spindle basic specification parameters #3017 to #3020)
- T2 : Synchronous tapping changeover time constant 2 (spindle basic specification parameters #3041 to #3044)
- T3 : Synchronous tapping changeover time constant 3 (spindle basic specification parameters #3045 to #3048)

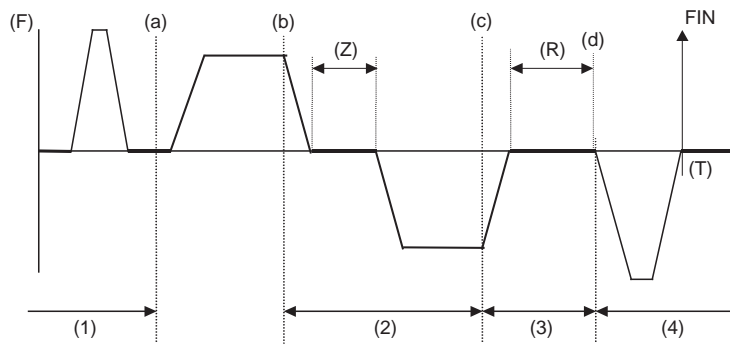
In-position check during synchronous tapping

Synchronous tapping in-position check settings

#1223 aux07				"P" designation of G84/G74 command	In-position check during synchronous tapping
bit3	bit4	bit5	bit2		
Synchronous tapping In-position check	Hole bottom	R point:	I point -> R point:		
0	-	-	-	-	Carry out in-position check at I point -> R point/R point/hole bottom.
1	-	-	-	No "P" designation Example: G84 F1. Z-5. S1000 R-5.	Hole bottom: Do not carry out in-position check. R point: Do not carry out in-position check. I point -> R point; Do not carry out in-position check.
1	1	1	1	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	Hole bottom: Carry out in-position check by tapping in-position width. R point: Carry out in-position check. I point -> R point; Carry out in-position check.
1	1	0	1	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	Hole bottom: Carry out in-position check by tapping in-position width. R point: Do not carry out in-position check. I point -> R point; Carry out in-position check.
1	0	1	1	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	Hole bottom: Do not carry out in-position check. R point: Carry out in-position check. I point -> R point; Carry out in-position check.
1	0	0	1	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	Hole bottom: Do not carry out in-position check. R point: Do not carry out in-position check. I point -> R point; Carry out in-position check.
1	1	1	0	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	Hole bottom: Carry out in-position check by tapping in-position width. R point: Carry out in-position check. I point -> R point; Do not carry out in-position check.
1	1	0	0	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	Hole bottom: Carry out in-position check by tapping in-position width. R point: Do not carry out in-position check. I point -> R point; Do not carry out in-position check.
1	0	1	0	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	Hole bottom: Do not carry out in-position check. R point: Carry out in-position check. I point -> R point; Do not carry out in-position check.
1	0	0	0	"P" designation Example: G84 F1. Z-5. S1000 P0 R-5.	Hole bottom: Do not carry out in-position check. R point: Do not carry out in-position check. I point -> R point; Do not carry out in-position check.

(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

(F): Feed rate

(T): Time

(a) In-position completion of the G0 feed from the R point

(b) G1 deceleration start at tapping cut-in

(c) G1 deceleration start at tapping return

(d) Start of G0 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 Tapinps

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 tapping in-position check	Hole bottom	R point	I point -> R point:				
0	-	-	-	Time designated by "P". Several 10ms as processing time when no "P".	Operation determined by setting of inpos (#1193) and aux 07 (#1223): bit 1 parameters.	Operation determined by setting of inpos (#1193) and aux 07 (#1223): bit 1 parameters.	Operation determined by setting of inpos (#1193) and aux 07 (#1223): bit 1 parameters.
1	0	0	1	Larger value of "P" and TapDwl (#1313) is valid. No dwell 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	Larger value of "P" and TapDwl (#1313) is valid. No dwell 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	Larger value of "P" and TapDwl (#1313) is valid. No dwell 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	Larger value of "P" and TapDwl (#1313) is valid. Several 10ms 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	Larger value of "P" and TapDwl (#1313) is valid. No dwell executed if both values are 0.	Wait until time in the left column elapses.		
1	0	1	0	Larger value of "P" and TapDwl (#1313) is valid. No dwell 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	Larger value of "P" and TapDwl (#1313) is valid. No dwell 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	Larger value of "P" and TapDwl (#1313) is valid. Several 10ms 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.	

(Note 1) The I point refers to the initial point.

(Note 2) Note that vibration or deterioration in accuracy may occur when invalidating the in-position check at R point.



Precautions

Refer to "Precautions When Using a Fixed Cycle for Drilling".

13.5.3 Face Boring Cycle (Longitudinal boring cycle) ; G85 (G89)



Command format

Face boring cycle mode

G85 X/U__ C/H__ Z/W__ Rr Pp Ff Kk Mm ;

X/U C/H	Designation of hole position initial point (absolute/incremental value)
Z/W	Designation of hole bottom position (absolute/incremental value from R point) (modal)
Rr	Designation of R point (incremental value from initial point) (sign ignored) (modal)
Pp	Designation of dwell time at hole bottom point. Relationship between time and designated value is same as for G04 designation. (modal)
Ff	Designation of feedrate for cutting feed (modal)
Kk	Designation of number of repetitions, 0 to 9999 (standard value = 1)
Mm	Designation of miscellaneous command

Longitudinal boring cycle mode

G89 Z/W__ C/H__ X/U__ Rr Pp Ff Kk Mm ;

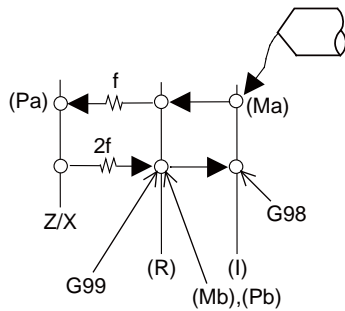
Z/W C/H	Designation of hole position initial point (absolute/incremental value)
X/U	Designation of hole bottom position (absolute/incremental value from R point) (modal)
Rr	Designation of R point (incremental value from initial point) (sign ignored) (modal)
Pp	Designation of dwell time at hole bottom point. Relationship between time and designated value is same as for G04 designation. (modal)
Ff	Designation of feedrate for cutting feed (modal)
Kk	Designation of number of repetitions, 0 to 9999 (standard value = 1)
Mm	Designation of miscellaneous command

Precautions

- (1) For the longitudinal boring cycle (G89), designate Z/W to the hole position initial point and X/U to the hole bottom position.
- (2) The designation of the hole position initial point is unmodal. When G85(G89) command is to be executed continuously, designate them block by block.
- (3) Q command is unmodal. Designate them block by block.
- (4) K command is unmodal. When K command is not issued, it is regarded as K1. When K0 is designated, the hole machining data are stored in the memory but no holes will be machined.



Detailed description



- (1) See "Face Deep Hole Drilling Cycle 1; G83" for details on (Ma),(Mb),(Pa),(Pb).
- (2) The tool returns to the R point at a cutting feed rate which is double the designated feed rate command. However, it does not exceed the maximum cutting feed rate.



Precautions

Refer to "Precautions When Using a Fixed Cycle for Drilling".

13.5.4 Deep Hole Drilling Cycle 2 ; G83.2



Function and purpose

The deep hole drilling cycle 2 drills deep holes in the X-axis or Z-axis direction by commanding the X or Z coordinate of the end point and the cut amount at cutting feed.



Command format

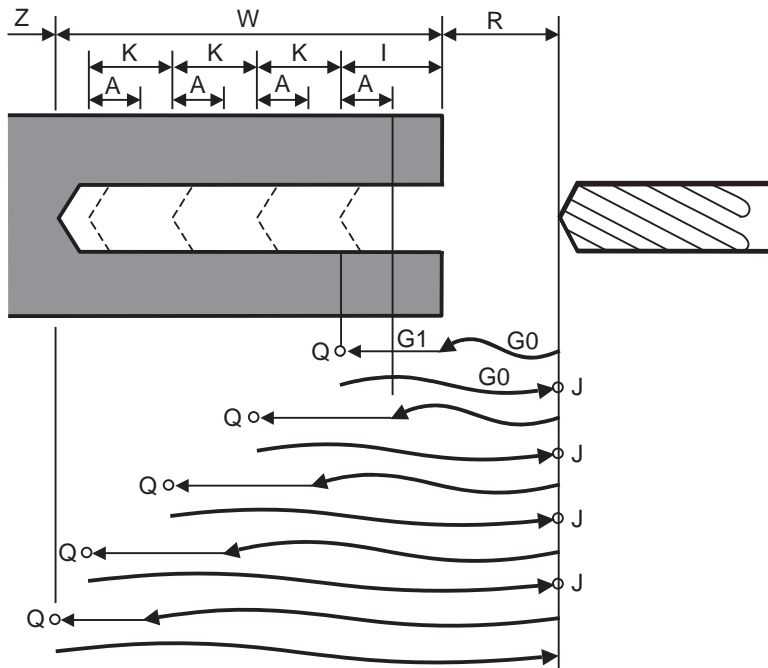
G83.2 W/Z/U/X__ R__ I__ K__ A__ Q__ J__ F__ ; ... Deep hole drilling cycle 2

W/Z/U/X	Incremental value from hole drilling start point/coordinates of hole bottom (sign valid)
R	Incremental value from present position up to hole drilling start point (sign ignored) (always radius value with incremental value.)
I	Cut amount of first cutting pass (sign ignored) (always radius value with incremental value.)
K	Cut amount of second and subsequent cutting passes (sign ignored) (always radius value with incremental value.)
A	Drill stop safety distance for second and subsequent cutting passes (sign ignored) (always radial value with incremental value.)
Q	Dwell time at cut point (sign ignored, decimal point invalid)
J	Dwell time at return point (sign ignored, decimal point invalid)
F	Cutting feedrate

(Note 1) When A command is not issued, the parameter "#8013 G83 n" setting value is used.

(Note 2) If the cut amount for either the first cutting pass (address "I") or the second and subsequent passes (address "K") is not designated (including a command value = 0), the one designated will be used as I = K= command value.

If both commands are not designated, hole drilling will be performed down to the hole bottom at once.



Q : Dwell time at cut point

J : Dwell time at return point

With single block operation, block stops upon completion of the deep hole drilling cycle 2 commands.



Detailed description

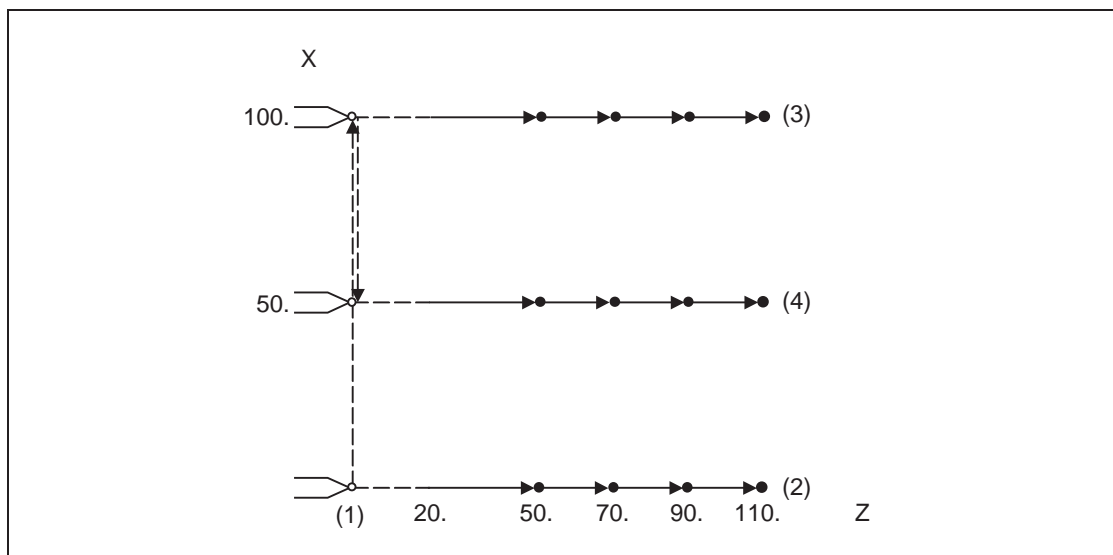
- (1) When the axis address of the hole drilling axes is commanded several times in a block, the last address will be valid.
- (2) A program error (P33) will occur in the following commands.
 - (a) When both hole drilling axis X (command address X or U) and hole drilling axis Z (command address Z or W) are commanded.
 - (b) When an axis other than X or Z (command addresses except X, U, Z and W) is commanded.
- (3) When the feed hold button is pressed while the deep hole drilling cycle 2 is being executed, the automatic operation stops and the remainder of the cycle will be executed when automatic operation is restarted.
- (4) When an interruption is made by manual operation while the automatic operation is stopped (manual ABS switch ON), the operation in the deep hole drilling cycle 2 modal is shifted from the automatic operation restart by an amount equivalent to the movement caused by the interrupt.



Program example

(when deep hole drilling cycle 2 is used as a modal command)

G28 XZ;	
G0 X0. Z0.;	...(1)
G83.2 Z110.R20.I30.K20.A5.Q1000 J500 F300. ;	...(2)
X100.;	...(3)
X50.;	...(4)
M02 ;	



Precautions

Refer to "Precautions When Using a Fixed Cycle for Drilling".

13.5.5 Fixed Cycle for Drilling Cancel; G80



Detailed description

This cancels the fixed cycle for drilling. The hole machining mode and hole machining data are both canceled.

13.5.6 Precautions When Using a Fixed Cycle for Drilling



Precautions

- (1) When G84 or G88 fixed cycle is commanded, the rotary tool must be rotated to the specified direction beforehand by miscellaneous functions (M3, M4).
- (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 not data, the hole drilling operation will not be executed. Note that even with X axis data, if that block is a dwell (G04) time command, the hole drilling will not be executed.
- (3) Command hole machining data (Q, P (A,I,K,Q,J for G83.2)) in the block in which hole drilling is carried out (block containing data for the basic axis, additional axis or R). The modal data is not updated if these data are commanded in blocks without hole drilling operation.
- (4) The F modal may change when a reset is applied during execution of G85 (G89).
- (5) The fixed cycle for drilling can also be canceled by 01 group G codes in addition to G80. If these are commanded in the same block as the fixed cycle, the fixed cycle will be ignored.

m = 01 group code n = fixed cycle for drilling code
 Gm Gn X C Z R Q P K F ;

Gm: Executed	Gn: Ignored	X C Z: Executed
R Q P K: Ignored	F: Memorized	

(Example)

```
G01 G83 X100. C30. Z50. R-10. Q10. P1 F100. ;
G83 G01 X100. C30. Z50. R-10. Q10. P1 F100. ;
```

In both cases, G01 X100. C30. Z50. F100 is executed.

- (6) When miscellaneous functions are commanded in the same block as fixed cycle commands, they are output simultaneously with the initial positioning. Note that when the M code of the C axis clamp set in the parameters (#1183 clmp_M) is commanded in the same block, the M code will be output after positioning. After returning to the return point (G98 mode: initial point/G99 mode: R point) after hole drilling, the M code of the C axis unclamp (clamp M + 1) will be output, and the axis dwells for the time set in the parameter (#1184 clmp_D). When the number of rotations is designated, the above control will be carried out only for the first rotation, except for M codes of the C axis clamp. C axis clamp/unclamp M commands work in modal, and are output at every rotation until canceled by the fixed cycle cancel command.
- (7) When tool length offset command (T function) is issued in the fixed cycle for drilling, they are executed according to the tool length offset function.
- (8) A program error (P155) will occur when a fixed cycle for drilling is commanded during tool nose R compensation.
- (9) For the G code lists 2,4,6 the initial point level return is fixed. The return level cannot be changed with a G98/G99 command. Note that a deferent function will be executed if G98/G99 is commanded.
- (10) 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.
 G0 X100. ,I10.0 ;
 X-200. ;
- (11) If a tapping axis is under machine lock, normal synchronous tapping is applied even though high-speed synchronous tapping option is enabled.

13.5.7 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

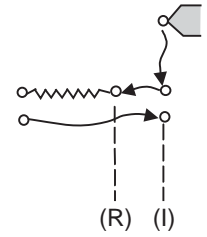
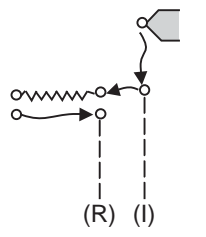
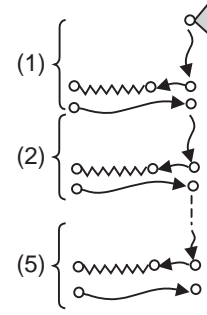
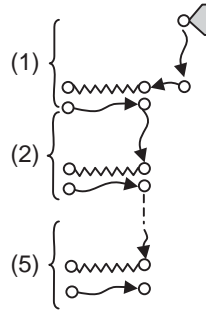
G98; ... Initial level return

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	G90 G83 X100. Z-50. R25. F1000 ;	 <p>Initial level return is executed.</p>	 <p>R point level return is executed.</p>
Two or more executions	G90 G83 X100. Z-50. R25. L5 F1000 ;	 <p>Initial level return is executed for all times.</p>	

13.5.8 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) When the workpiece coordinates change, re-program the addresses Z and R, even if the values are the same.

(Example)G54 Xx1 Zz1;

G83 Xx1 Zz2 Rr2 ;

G55 Xx3 Zz2 Rr2 ; ... Re-command even if Z and R are the same as the previous value.

Xx4 ;

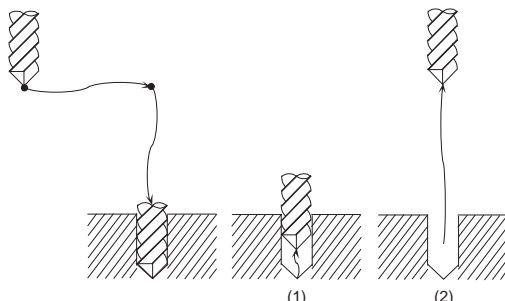
Xx5 ;

13.5.9 Drilling Cycle High-Speed Retract



Function and purpose

This function retracts the drill from the hole bottom at high speed in drilling machining. This helps extending the drill life by reducing the time of drilling in vain at hole bottom.



The drill moves up at high-speed (1) and returns to the initial point or R point in rapid traverse (2).



Command format

The command format is the same as fixed cycle.



Detailed description

- (1) 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.
Set the lost motion compensation type 2 or 3 to the servo parameter. Then set the following parameters to adjust the retract amount.
 - #2170 Lmc1QR (Lost motion compensation gain 1 for high-speed retract)
(correspond to "#2216 SV016 LMC1 Lost motion compensation 1")
 - #2171 Lmc2QR (lost motion compensation gain 2 for high-speed retract)
(correspond to "#2241 SV041 LMC2 Lost motion compensation 2")
 Set the following parameters for lost motion compensation timing or lost motion compensation 3 spring constant viscous coefficient, apart from the ordinary lost motion compensations.
 - #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")
 If the drilling axis is synchronously controlled, set the same value in each parameters for primary and secondary axes.
- (2) 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.
- (3) When the inclined axis control is valid and also, the hole drilling axis is an inclined axis or the basic axis of an inclined axis, a high-speed retract will not be performed. The drill moves in the ordinary rapid traverse.
- (4) For axes of control axis synchronization across part systems, a high-speed retract of the secondary axis will not be performed.

13.5.10 Acceleration/Deceleration Mode Change in Hole Drilling Cycle



Function and purpose

This function switches the acceleration/deceleration mode for hole drilling cycle between the inclination-constant method and the acceleration/deceleration after interpolation.



Command format

The command formats are the same as those of the fixed cycles G83, G87, and G83.2.



Detailed description

With parameter "#1253 set25/bit2 Acceleration/deceleration mode change in hole drilling cycle" enabled, operation will be as follows.

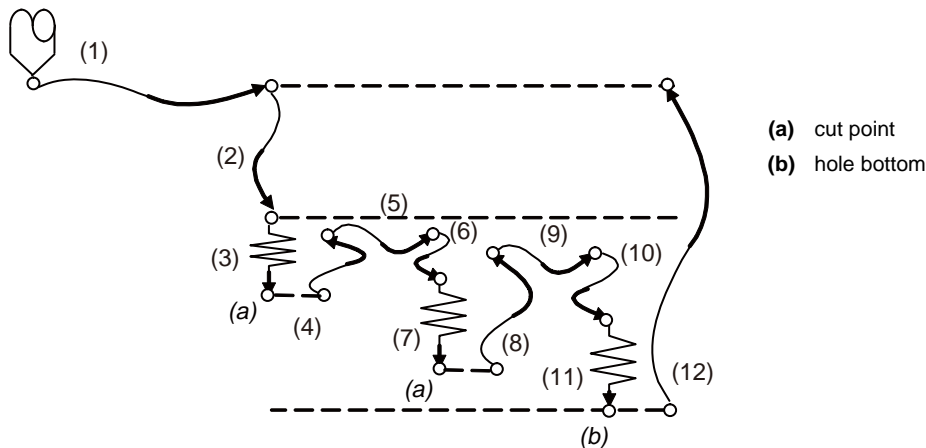
- (1) Acceleration/deceleration mode will be either linear or soft method. (Unless soft acceleration/deceleration is applied, the linear method will always be applied.)
- (2) The operation follows the parameter settings that determine whether the inclination-constant or post-interpolation acceleration/deceleration is applied. "#2001 rapid (rapid traverse rate)" and "#2004 G0tL (G0 time constant (linear))" provide G0 (rapid traverse) acceleration/deceleration inclination, and "#2002 clamp (cutting feedrate for clamp)" and "#2007 G1tL (G1 time constant (linear))" provide G1 (cutting feed) acceleration/deceleration inclination. Refer to "Rapid Traverse Constant Inclination Acceleration/Deceleration" for details of constant inclination acceleration/deceleration.



Operation example

Operation example of "acceleration/deceleration mode change in hole drilling cycle" being enabled

The below illustrates the processes of hole-bottom deceleration check of a drilling axis following the parameter "#19417 Hole dec check 2" settings.



#19417		G83	G87	G83.2
0	(a)Cut point	Perform no deceleration check.		
	(b)Hole bottom	Perform no deceleration check.		
1	(a)Cut point	Perform command deceleration check.		
	(b)Hole bottom	Perform command deceleration check.		
2	(a)Cut point	Perform command deceleration check.	Perform in-position check (sv024).	
	(b)Hole bottom	Perform in-position check (sv024).		

13.6 Fixed Cycle for Drilling (MITSUBISHI CNC special format)



Function and purpose

These fixed cycles are used to perform prepared sequences of machining programs, such as positioning, hole drilling, boring and tapping in a block. The machining sequences available are listed in the table below. This function is valid when the MITSUBISHI CNC special format is added (#1265 ext01/bit2 is ON).

G code	Hole machining start (-Z direction)	Operation at hole bottom		Return operation +Z direction	Application
		Dwell	Spindle		
G80	-	-	-	-	Cancel
G81	Cutting feed	-	-	Rapid traverse	Drill, spot drilling cycle
G82	Cutting feed	Yes	-	Rapid traverse	Drill, counter boring cycle
G83	Intermittent feed	Yes	-	Rapid traverse	Deep hole drilling cycle
G83.1	Intermittent feed	Yes	-	Rapid traverse	Stepping cycle
G84	Cutting feed	Yes	Reverse rotation	Cutting feed	Tapping cycle Pecking tapping cycle Deep-hole tapping cycle
G84.2	Cutting feed	Yes	Reverse rotation	Cutting feed	Synchronous tapping cycle Pecking tapping cycle Deep-hole tapping cycle
G85	Cutting feed	-	-	Cutting feed	Boring cycle
G89	Cutting feed	Yes	-	Cutting feed	Boring cycle

A fixed cycle mode can be canceled by G80 command and other hole machining modes or G command in the 01 group. At the same time, various other data will also be cleared to zero.

For the basic operations of the fixed cycle for drilling, refer to "Basic operations of fixed cycle for drilling" in "Fixed Cycle for Drilling; G80 to G89" (General format).



Detailed description

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 command, 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. The basic axis will be the hole drilling axis if there is no designation.

- (Ex.1) When G17 (XY 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) Changeover of the hole drilling axis must be done with the fixed cycle canceled.

(Note 2) A program error will occur if the hole drilling axis does not exist.

Programmable in-position width command in fixed cycle

This commands the in-position width for commanding the fixed cycle from the machining program.

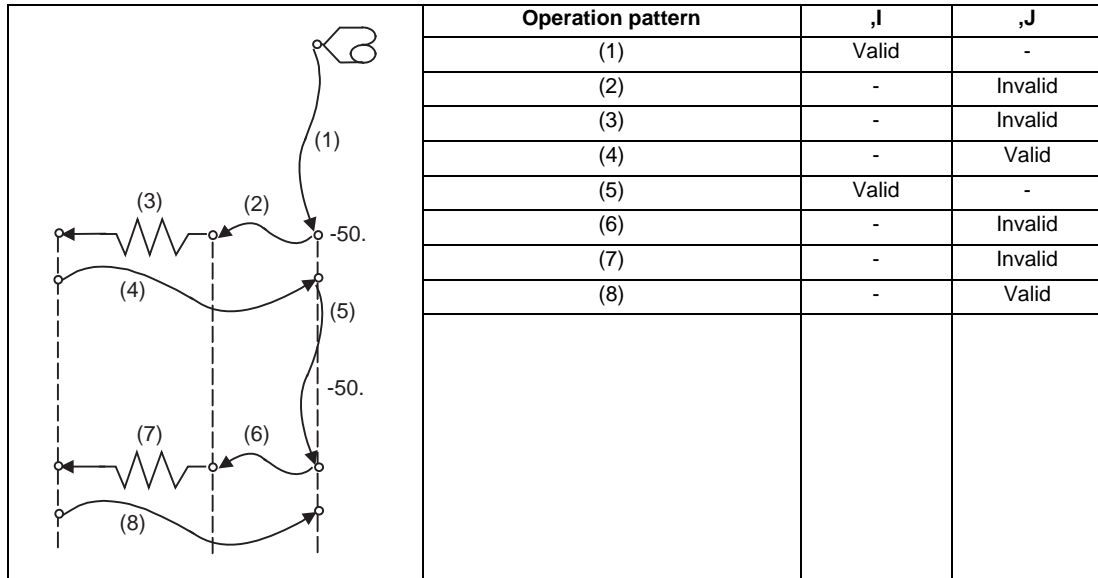
The in-position width is commanded with the ",I" address is commanded to the positioning axis, and the ",J" address is commanded to the hole drilling axis.

Address	Significance	Command range (unit)	Remarks
,I	Positioning axis in-position width (position error amount)	1 to 999.999 (mm)	When a value exceeding the command range is issued, a program error will occur. Program error (P35)
,J	Drilling axis in-position width (position error amount)	1 μm unit	

In-position check in fixed cycle

When L (number of repetitions) is designated two or more times in the fixed cycle, the commanded in-position width will be valid in the repetition block (5) to (8) below.

G81 U-50. W-50. R-50. L2 F2000 ,I0.2 ,J0.3 ;



Operation when number of repetitions L is designated

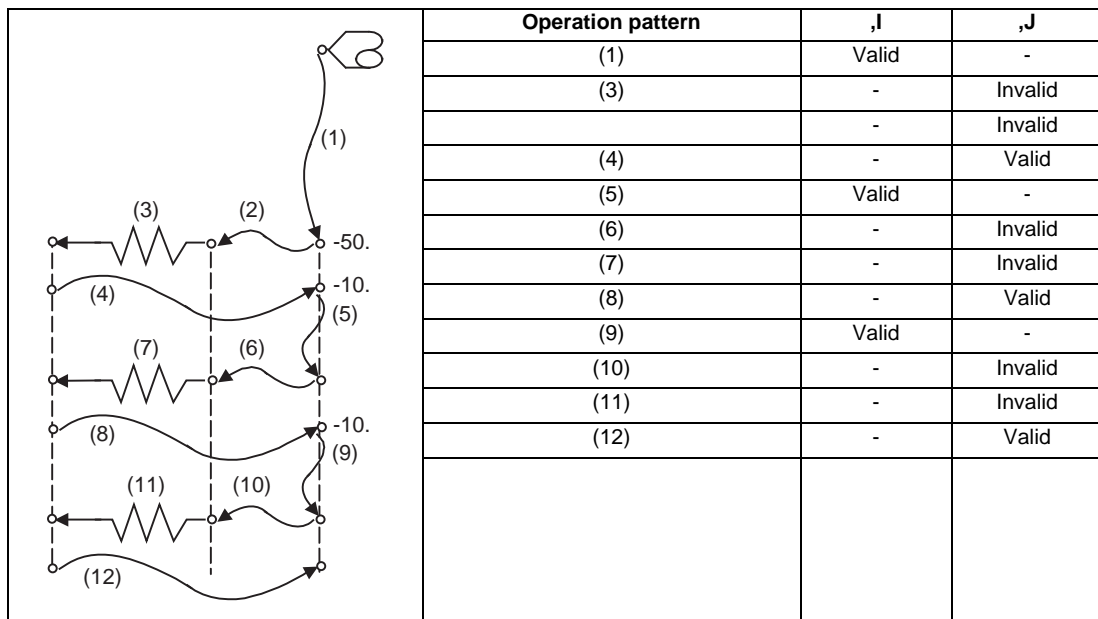
In the following machining program, the commanded in-position width is valid in the block in the figure below. In the (B) block, the in-position width (, I) commanded regarding to positioning in the previous block (A) is invalid (5). However, when returning from the hole bottom, the in-position width (, 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).

G81 U-50. W-50. R-50. F2000 ,I0.2 ,J0.3 ; (A)

U-10.; (B)

U-10.,I0.2; (C)



Operation in fixed cycle modal

13.6.1 Drilling Cycle, Spot Drilling Cycle ; G81



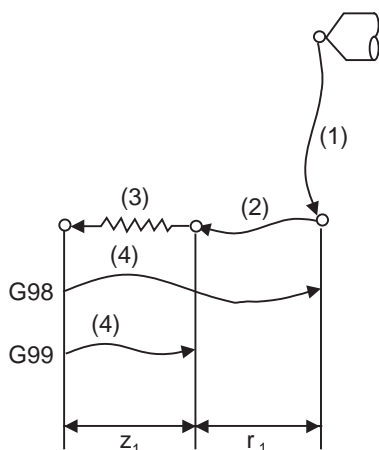
Command format

G81 Xx1(U) Zz1(W) Rr1 Ff1 LI1 ,li1 ,Jj1 ;

G81	Drilling cycle mode, spot drilling cycle mode
Xx1(U)	Designation of hole position initial point (absolute/incremental value)
Zz1(W)	Designation of hole bottom position (absolute/incremental value from R point)
Rr1	Designation of R point (incremental value from initial point) (sign ignored)
Ff1	Cutting feedrate
LI1	Designation of number of repetitions (0 to 9999)
,li1	Positioning axis in-position width
,Jj1	In-position width for hole drilling (a) G81 (drilling, spot drilling)



Detailed description



The operation stops at after the (1), (2) and (4) commands during single block operation.

Operation pattern	i1	j1	Program
(1)	Valid	-	G0Xx1
(2)	-	Invalid	G0Zr1
(3)	-	Invalid	G1Zz1 Ff1
(4)	-	Valid	G98 mode G0Z - (z1+r1) G99 mode G0Z - z1

13.6.2 Drilling Cycle, Counter Boring Cycle ; G82



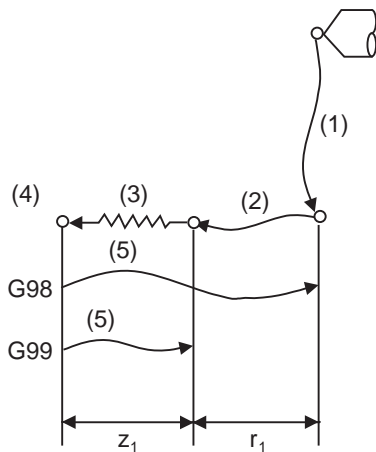
Command format

G82 Xx1(U) Zz1(W) Rr1 Ff1 Pp1 Ll1 ,li1 ,Jj1 ;

G82	Drilling cycle mode, counter boring cycle mode
Xx1(U)	Designation of hole position initial point (absolute/incremental value)
Zz1(W)	Designation of hole bottom position (absolute/incremental value from R point)
Rr1	Designation of R point (incremental value from initial point) (sign ignored)
Ff1	Cutting feedrate
Ll1	Designation of number of repetitions (0 to 9999)
Pp1	Designation of dwell time at hole bottom position (decimal points will be ignored)
,li1	Positioning axis in-position width
,Jj1	Drilling axis in-position width



Detailed description



Operation pattern	i1	j1	Program
(1)	Valid	-	G0Xx1
(2)	-	Invalid	G0Zr1
(3)	-	Invalid	G1Zz1 Ff1
(4)	-	-	G4Pp1(Dwell)
(5)	-	Valid	G98 mode G0Z - (z1+r1) G99 mode G0Z - z1

The operation stops at after the (1), (2) and (5) commands during single block operation.

13.6.3 Deep Hole Drilling Cycle ; G83



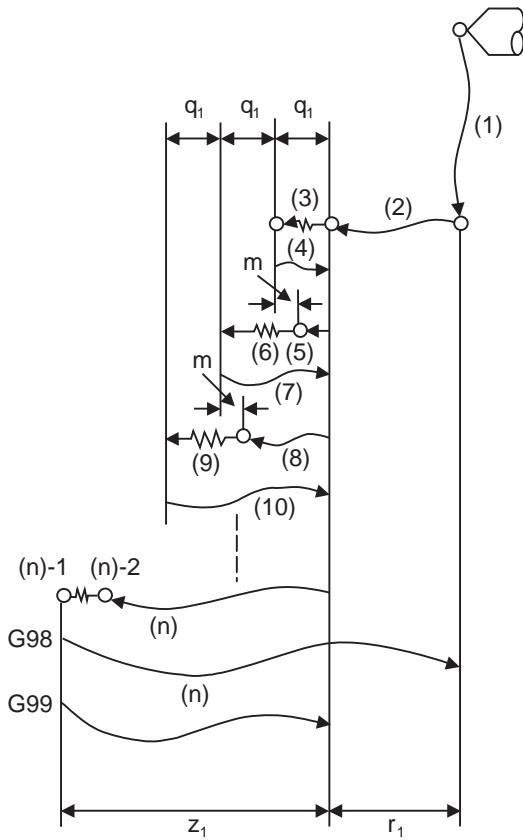
Command format

G83 Xx1(U) Zz1(W) Rr1 Qq1 Pp1 Ff1 LI1 ,li1 ,Jj1 ;

G83	Deep hole drilling cycle mode
Xx1(U)	Designation of hole position initial point (absolute/incremental value)
Zz1(W)	Designation of hole bottom position (absolute/incremental value from R point)
Rr1	Designation of R point (incremental value from initial point) (sign ignored)
Qq1	Cut amount for each cutting pass (incremental value)
Pp1	Designation of dwell time at hole bottom position (decimal points will be ignored)
Ff1	Cutting feedrate
LI1	Designation of number of repetitions (0 to 9999)
,li1	Positioning axis in-position width
,Jj1	Drilling axis in-position width



Detailed description



m will differ according to the parameter "#8013 G83 n". Program so that $q_1 > m$.

Operation pattern	,I	,J	Program
(1)	Valid	-	G0Xx1
(2)	-	Invalid	G0Zr1
(3)	-	Invalid	G1Zq1 Ff1
(4)	-	Invalid	G0Z-q1
(5)	-	Invalid	G0Z(q1-m)
(6)	-	Invalid	G1Z(q1+m) Ff1
(7)	-	Invalid	G0Z -2*q1
(8)	-	Invalid	G0Z (2*q1-m)
(9)	-	Invalid	G1Z(q1+m) Ff1
(10)	-	Invalid	G0Z-3*q1
	:		
n-1	-	Invalid	G4Pp1(Dwell)
N	-	Valid	G98 mode G0Z - (z1+r1) G99 mode G0Z - z1

As shown above, when executing a second and the following cuttings in the G83, the movement will change from rapid traverse to cutting feed at several m[mm] before the position where the previous cutting was done. After reaching the hole bottom, the axis will return according to the G98 or G99 mode. The operation stops at after the (1), (2) and (n) commands during single block operation.

13.6.4 Stepping Cycle ; G83.1



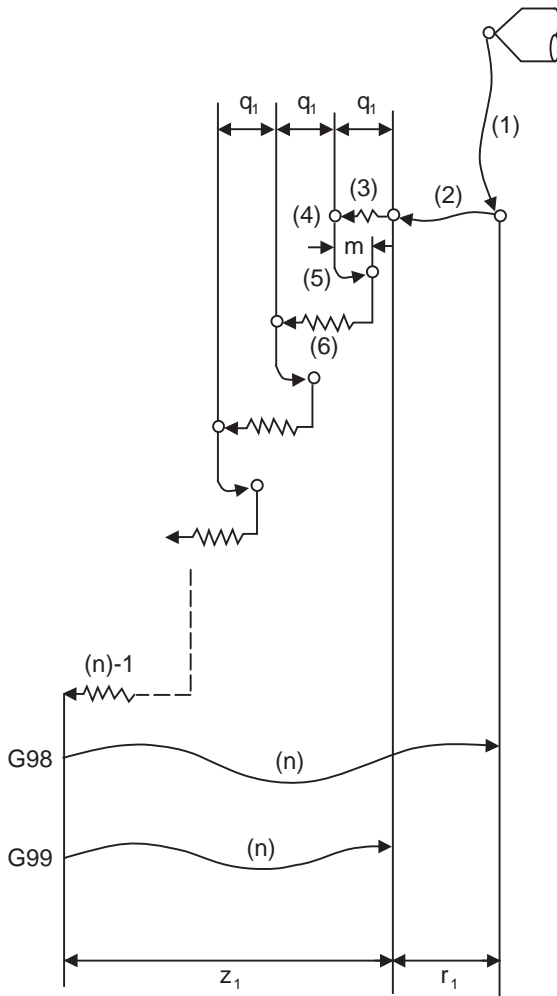
Command format

G83.1 Xx1(U) Zz1(W) Qq1 Rr1 Ff1 Pp1 LI1 ,li1 ,Jj1 ;

G83.1	Stepping cycle mode
Xx1(U)	Designation of hole position initial point (absolute/incremental value)
Zz1(W)	Designation of hole bottom position (absolute/incremental value from R point)
Qq1	Cut amount for each cutting pass (incremental value)
Rr1	Designation of R point (incremental value from initial point) (sign ignored)
Ff1	Cutting feedrate
Pp1	Designation of dwell time at hole bottom position (decimal points will be ignored)
LI1	Designation of number of repetitions (0 to 9999)
,li1	Positioning axis in-position width
,Jj1	Drilling axis in-position width



Detailed description



Operation pattern	,I	,J	Program
(1)	Valid	-	G0Xx1
(2)	-	Invalid	G0Zr1
(3)	-	Invalid	G1Zq1 Ff1
(4)	-	-	G4Pp1
(5)	-	Invalid	G0Z-m
(6)	-	Invalid	G1Z(q1+m) Ff1
	:		
n	-	Valid	G98 mode G0Z - (z1+r1) G99 mode G0Z - z1

As shown above, when executing a second and the following cuttings in the G83, the movement will return several m[mm] at rapid traverse and then change to cutting feed. The return amount m will differ according to the parameter "#8012 G73 n".

The operation stops at after the (1), (2) and (n) commands during single block operation.

13.6.5 Tapping Cycle ; G84



Command format

G84 Xx1(U) Zz1(W) Rr1 Qq1 Ff1 Pp1 Ll1 ,li1 ,Jj1 ;

G84	Tapping cycle mode
Xx1(U)	Designation of hole position initial point (absolute/incremental value)
Zz1(W)	Designation of hole bottom position (absolute/incremental value from R point)
Rr1	Designation of R point (incremental value from initial point) (sign ignored)
Qq1	Designation of cut amount for each cutting pass with G83 (G87). Always incremental value, radius value (sign ignored)
Ff1	Cutting feedrate
Pp1	Designation of dwell time at hole bottom position (decimal points will be ignored)
Ll1	Designation of number of repetitions (0 to 9999)
,li1	Positioning axis in-position width
,Jj1	Drilling axis in-position width

If a value other than zero is specified to address Q while the option of pecking tapping cycle or deep-hole tapping cycle is ON, either pecking or deep-hole tapping cycle is executed instead of normal tapping cycle.

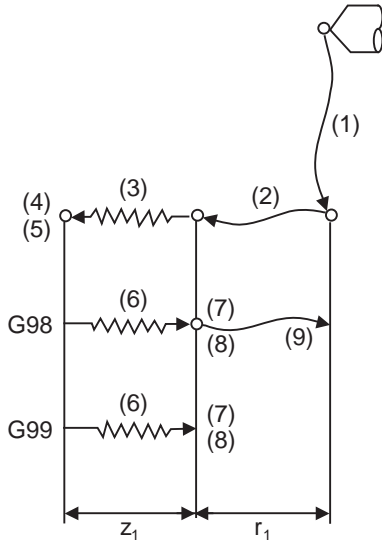
When the parameter "#1272 ext08/bit4" is "0" ... Pecking tapping cycle

When the parameter "#1272 ext08/bit4" is "1" ... Deep-hole tapping cycle



Detailed description

Normal tapping cycle(When Q is not designated)



Operation pattern	i1	j1	Program
(1)	Valid	-	G0 Xx1
(2)	-	Invalid	G0 Zr1
(3)	-	Invalid	G1 Zq1 Ff1
(4)	-	-	G4 Pp1
(5)	-	-	M4 (Spindle reverse rotation)
(6)	-	Invalid	G1 Z-z1 Ff1
(7)	-	-	G4 Pp1
(8)	-	-	M3 (Spindle forward rotation)
(9)	-	Valid	G98 mode G0Z - r1 G99 mode No movement

When G84 is being 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 control parameter "G00 DRY RUN" is ON. If the feed hold button is pressed during G84 execution, the movement will not stop immediately, and instead, will stop after (6) during sequences (3) to (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 modal, the "Tapping" NC output signal will be output.

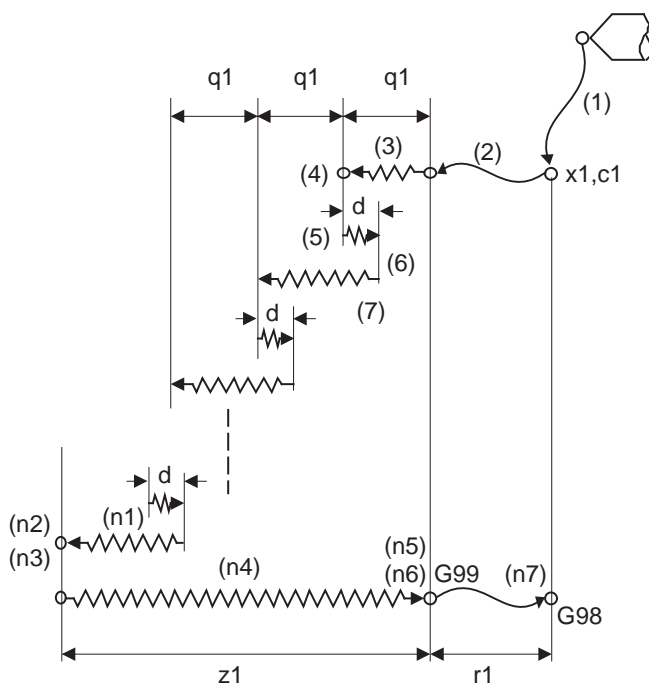
During the G84 synchronous tapping modal, the M3, M4, M5 and S code will not be output.

The feedrates for tapping retract are as follows.

G94/G95	Control parameter F1-digit value	F command value	Feed designation
G94	OFF	F designation	Feed per minute
	ON	Other than F0 to F8	
			F0 to F8 (no decimal point)
G95	-	F designation	Feed per revolution

- is irrelevant to the setting

Pecking Tapping Cycle(When the Q command is designated #1272 ext08/bit4=0)



Operation pattern	i1	j1	Program
(1)	Valid	-	G0 Xx1 Cc1 ,li1
(2)	-	Invalid	G0 Zr1
(3)	-	Invalid	G1 Zq1 Ff1
(4)	-	-	M4 (Spindle reverse rotation)
(5)	-	Invalid	G1 Z-d Ff1
(6)	-	-	M3 (Spindle forward rotation)
(7)	-	Invalid	G1 Z(q1+d) Ff1
:			
(n1)	-	Invalid	G1 Z(z1-q1*n) Ff1
(n2)	-	-	G4 Pp1
(n3)	-	-	M4
(n4)	-	Invalid	G1 Z-z1 Ff1
(n5)	-	-	G4 Pp1
(n6)	-	-	M3
(n7)	-	Valid	G98 mode G0Z-r1 ,lj1 G99 mode No movement

When G84.2 is executed, the override will be canceled and the override will automatically be set to 100%. During a pulling operation from the hole bottom to R point, the override set by the parameter "#1172 tapovr" will be enabled. Note however, it will be disabled during a return operation by the return amount d.

Set the return amount d in the parameter "#8018 G84/G74 n".

Dry run is valid when for a positioning command the parameter "#1085 G00 DRY RUN" is "1".

If the feed hold button is pressed during G84.2 execution, the operation stops immediately during a rapid traverse feed of (1), (2) and (n7).

For (3) to (n4), however, the operation will not stop immediately, but will stop after (n4) is completed.

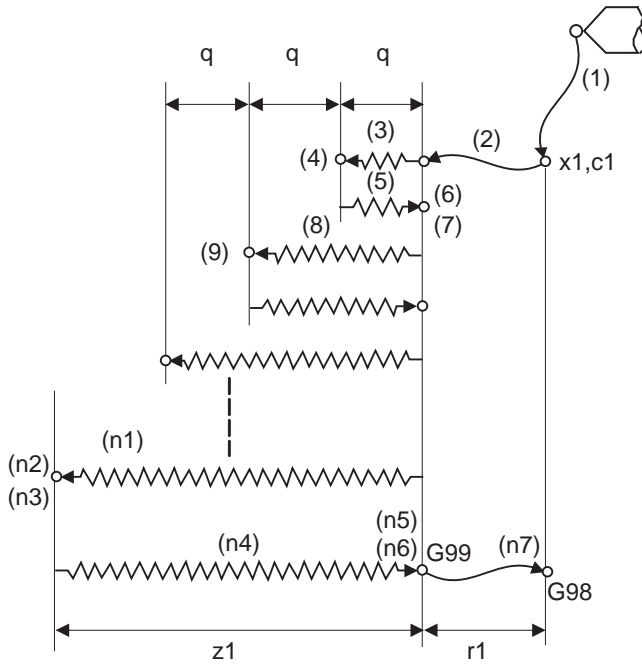
The operation stops in the positions where the commands (1), (2) and (n7) are completed during single block operation.

During the G84.2 mode, the NC signal "Tapping" will be output.

During the G84.2 synchronous tapping mode, the M3, M4, M5, or S code will not be output.

When the command value of Q is "0", this command is treated as a normal tapping cycle.

Deep-hole Tapping Cycle(When the Q command is designated #1272 ext08/bit4=1)



Operation pattern	i1	j1	Program
(1)	Valid	-	G0 Xx1 Cc1
(2)	-	Invalid	G0 Zr1
(3)	-	Invalid	G1 Zq1 Ff1
(4)	-	-	M4 (Spindle reverse rotation)
(5)	-	Invalid	G1 Z-d Ff1
(6)	-	-	G4 Pp1
(7)	-	-	M3 (Spindle forward rotation)
(8)	-	Invalid	G1 Z(2*q1)Ff1
(9)	-	-	
:			
(n1)	-	Invalid	G1 Zz1 Ff1
(n2)	-	-	G4 Pp1
(n3)	-	-	M4
(n4)	-	Invalid	G1 Z-z1 Ff1
(n5)	-	-	G4 Pp1
(n6)	-	-	M3
(n7)	-	Valid	G98 mode G0Z-r1 G99 mode No movement

When G84.2 is executed, the override will be canceled and the override will automatically be set to 100%. The override set with the parameter "#1172 tapovr" is disabled.

Dry run is valid when for a positioning command the parameter "#1085 G00 DRY RUN" is "1".

If the feed hold button is pressed during G84.2 execution, the operation stops immediately during a rapid traverse feed of (1), (2) and (n7).

For (3) to (n4), however, the operation will not stop immediately, but will stop after (n4) is completed.

The operation stops in the positions where the commands (1), (2) and (n7) are completed during single block operation.

During the G84.2 mode, the NC signal "Tapping" will be output.

During the G84.2 synchronous tapping mode, the M3, M4, M5, or S code will not be output.

When the command value of Q is "0", this command is treated as a normal tapping cycle.

13.6.6 Synchronous Tapping Cycle ; G84.2



Command format

G84.2 Xx1(U) Zz1(W) Rr1 Qq1 Ff1 Pp1 Ll1 Ss1 ,Ss2 ,li1 ,Jj1 ;

G84.2	Synchronous tapping cycle mode
Xx1(U)	Designation of hole position initial point (absolute/incremental value)
Zz1(W)	Designation of hole bottom position (absolute/incremental value from R point)
Rr1	Designation of R point (incremental value from initial point) (sign ignored)
Qq1	Designation of cut amount for each cutting pass with G83 (G87). Always incremental value, radius value (sign ignored)
Ff1	Cutting feedrate
Pp1	Designation of dwell time at hole bottom position (decimal points will be ignored)
Ll1	Designation of number of repetitions (0 to 9999)
Ss1	Spindle rotation speed
,Ss2	Designation of spindle rotation speed at return
,li1	Positioning axis in-position width
,Jj1	Drilling axis in-position width

If a value other than zero is specified to address Q while the option of pecking tapping cycle or deep-hole tapping cycle is ON, either pecking or deep-hole tapping cycle is executed instead of normal tapping cycle.

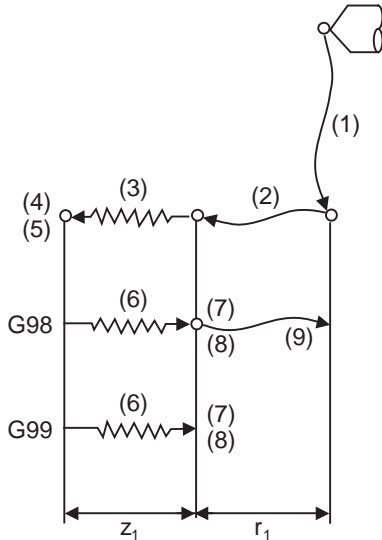
When the parameter "#1272 ext08/bit4" is "0" ... Pecking tapping cycle

When the parameter "#1272 ext08/bit4" is "1" ... Deep-hole tapping cycle



Detailed description

Normal tapping cycle(When Q is not designated)



Operation pattern	i1	j1	Program
(1)	Valid	-	G0 Xx1
(2)	-	Invalid	G0 Zr1
(3)	-	Invalid	G1 Zq1 Ff1
(4)	-	-	G4 Pp1
(5)	-	-	M4 (Spindle reverse rotation)
(6)	-	Invalid	G1 Z-z1 Ff1
(7)	-	-	G4 Pp1
(8)	-	-	M3 (Spindle forward rotation)
(9)	-	Valid	G98 mode G0Z - r1 G99 mode No movement

When G84.2 is being 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 control parameter "G00 DRY RUN" is ON. If the feed hold button is pressed during G84.2 execution, the movement will not stop immediately, and instead, will stop after (6) during sequences (3) to (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 G84.2 modal, the "Tapping" NC output signal will be output.

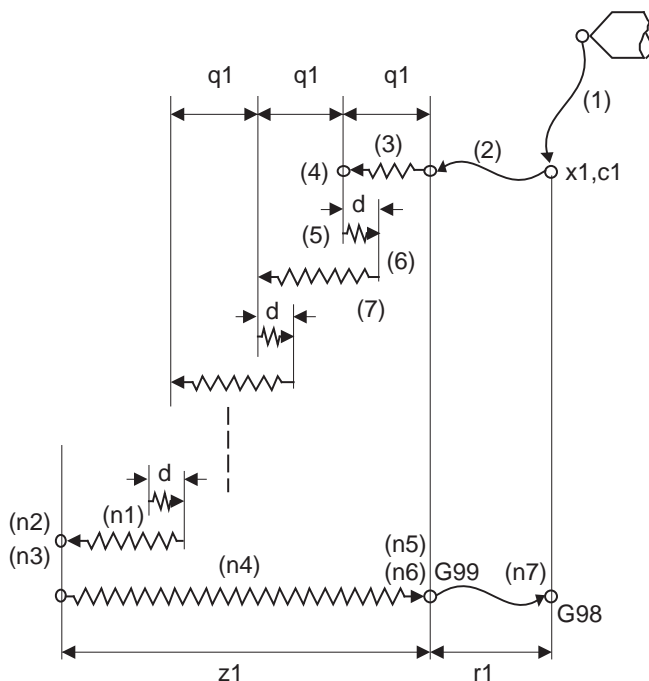
During the G84.2 synchronous tapping modal, the M3, M4, M5 and S code will not be output.

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. 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.

The feedrates for tapping retract are as follows.

Address	Meaning of address	Command range (unit)	Remarks
.S	Spindle rotation speed during return	0 to 99999 (r/min)	The data is held as modal information. If the value is smaller than the spindle rotation speed, the spindle rotation speed value will be valid even during return. If the spindle rotation speed is not 0 during return, the tapping retract override value will be invalid.

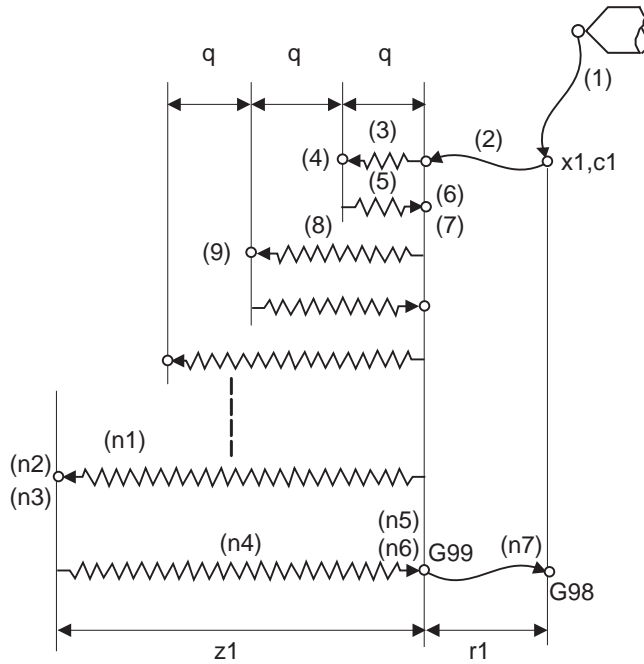
Pecking Tapping Cycle(When the Q command is designated #1272 ext08/bit4=0)



Operation pattern	i1	j1	Program
(1)	Valid	-	G0 Xx1 Cc1 ,li1
(2)	-	Invalid	G0 Zr1
(3)	-	Invalid	G1 Zq1 Ff1
(4)	-	-	M4 (Spindle reverse rotation)
(5)	-	Invalid	G1 Z-d Ff1
(6)	-	-	M3 (Spindle forward rotation)
(7)	-	Invalid	G1 Z(q1+d) Ff1
:			
(n1)	-	Invalid	G1 Z(z1-q1*n) Ff1
(n2)	-	-	G4 Pp1
(n3)	-	-	M4
(n4)	-	Invalid	G1 Z-z1 Ff1
(n5)	-	-	G4 Pp1
(n6)	-	-	M3
(n7)	-	Valid	G98 mode G0Z-r1 ,lj1 G99 mode No movement

When G84.2 is executed, the override will be canceled and the override will automatically be set to 100%. During a pulling operation from the hole bottom to R point, the override set by the parameter "#1172 tapovr" will be enabled. Note however, it will be disabled during a return operation by the return amount d. Set the return amount d in the parameter "#8018 G84/G74 n". Dry run is valid when for a positioning command the parameter "#1085 G00 DRY RUN" is "1". If the feed hold button is pressed during G84.2 execution, the operation stops immediately during a rapid traverse feed of (1), (2) and (n7). For (3) to (n4), however, the operation will not stop immediately, but will stop after (n4) is completed. The operation stops in the positions where the commands (1), (2) and (n7) are completed during single block operation. During the G84.2 mode, the NC signal "Tapping" will be output. During the G84.2 synchronous tapping mode, the M3, M4, M5, or S code will not be output. When the command value of Q is "0", this command is treated as a normal tapping cycle.

Deep-hole Tapping Cycle(When the Q command is designated #1272 ext08/bit4=1)



Operation pattern	i1	j1	Program
(1)	Valid	-	G0 Xx1 Cc1
(2)	-	Invalid	G0 Zr1
(3)	-	Invalid	G1 Zq1 Ff1
(4)	-	-	M4 (Spindle reverse rotation)
(5)	-	Invalid	G1 Z-d Ff1
(6)	-	-	G4 Pp1
(7)	-	-	M3 (Spindle forward rotation)
(8)	-	Invalid	G1 Z(2*q1)Ff1
(9)	-	-	
:			
(n1)	-	Invalid	G1 Zz1 Ff1
(n2)	-	-	G4 Pp1
(n3)	-	-	M4
(n4)	-	Invalid	G1 Z-z1 Ff1
(n5)	-	-	G4 Pp1
(n6)	-	-	M3
(n7)	-	Valid	G98 mode G0Z-r1 G99 mode No movement

When G84.2 is executed, the override will be canceled and the override will automatically be set to 100%. The override set with the parameter "#1172 tapovr" is disabled.

Dry run is valid when for a positioning command the parameter "#1085 G00 DRY RUN" is "1".

If the feed hold button is pressed during G84.2 execution, the operation stops immediately during a rapid traverse feed of (1), (2) and (n7).

For (3) to (n4), however, the operation will not stop immediately, but will stop after (n4) is completed.

The operation stops in the positions where the commands (1), (2) and (n7) are completed during single block operation.

During the G84.2 mode, the NC signal "Tapping" will be output.

During the G84.2 synchronous tapping mode, the M3, M4, M5, or S code will not be output.

When the command value of Q is "0", this command is treated as a normal tapping cycle.

Spindle acceleration/deceleration pattern during synchronous tapping

Refer to "Face Tapping Cycle; G84 (Longitudinal tapping cycle; G88)/ Face Reverse Tapping Cycle; G84.1 (Longitudinal reverse tapping cycle; G88.1" (General format).

In-position check during synchronous tapping

Refer to "Face Tapping Cycle; G84 (Longitudinal tapping cycle; G88)/ Face Reverse Tapping Cycle; G84.1 (Longitudinal reverse tapping cycle; G88.1" (General format).

13.6.7 Boring Cycle ; G85



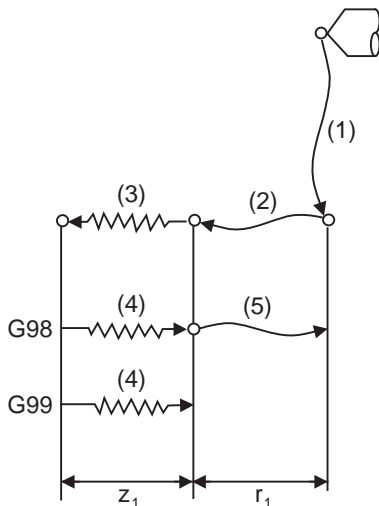
Command format

G85 Xx1(U) Zz1(W) Rr1 Ff1 Ll1 ,li1 ,Jj1 ;

G85	Boring cycle mode
Xx1(U)	Designation of hole position initial point (absolute/incremental value)
Zz1(W)	Designation of hole bottom position (absolute/incremental value from R point)
Rr1	Designation of R point (incremental value from initial point) (sign ignored)
Ff1	Cutting feedrate
Ll1	Designation of number of repetitions (0 to 9999)
,li1	Positioning axis in-position width
,Jj1	Drilling axis in-position width



Detailed description



Operation pattern	i1	j1	Program
(1)	Valid	-	G0Xx1
(2)	-	Invalid	G0Zr1
(3)	-	Invalid	G1Zq1 Ff1
(4)	-	Invalid	G1Z-z1 Ff1
(5)	-	Valid	G98 mode G0Z - r1 G99 mode No movement

The operation stops at after the (1), (2), (4) or (5) commands during single block operation.

13.6.8 Boring Cycle ; G89



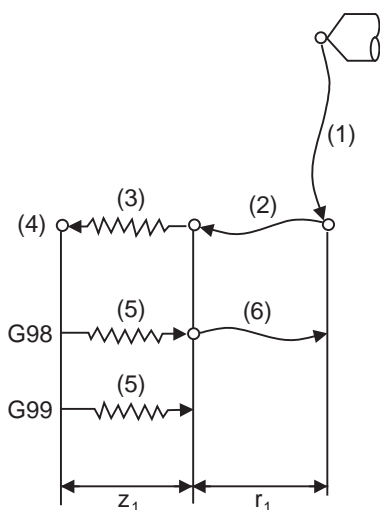
Command format

G89 Xx1(U) Zz1(W) Rr1 Ff1 Pp1 LI1 ,li1 ,Jj1;

G89	Boring cycle mode
Xx1(U)	Designation of hole position initial point (absolute/incremental value)
Zz1(W)	Designation of hole bottom position (absolute/incremental value from R point)
Rr1	Designation of R point (incremental value from initial point) (sign ignored)
Ff1	Cutting feedrate
Pp1	Designation of dwell time at hole bottom position (decimal points will be ignored)
LI1	Designation of number of repetitions (0 to 9999)
,li1	Positioning axis in-position width
,Jj1	Drilling axis in-position width



Detailed description



Operation pattern	i1	j1	Program
(1)	Valid	-	G0Xx1
(2)	-	Invalid	G0Zr1
(3)	-	Invalid	G1Zq1 Ff1
(4)	-	-	G4 Pp1
(5)	-	Invalid	G1Z-z1 Ff1
(6)	-	Valid	G98 mode G0Z - r1 G99 mode No movement

The operation stops at after the (1), (2), (5) or (6) commands during single block operation.

13.6.9 Precautions for Using Fixed Cycle for Drilling (MITSUBISHI CNC special format)



Precautions

- (1) Before commanding the fixed cycle, the spindle must be rotated to a specific direction by an M function (M3 or M4).
- (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 not 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) in the block where hole drilling is executed. (Block containing data on basic axis, additional axis or R)
- (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 cycle for drilling

Gm Gn X Z R Q P L F ;

Gm: Executed

Gn: Ignored

X Z: Executed

R Q P L: Ignored

F: Memorized

Note that for the G02 and G03 commands, R will be handled as the circular radius.

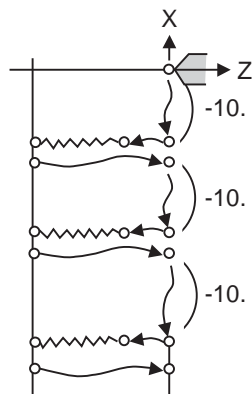
- (5) If an M function is commanded in the same block as the fixed cycle command, the M code and MF will be output during the initial positioning. The axis will move to the next operation with FIN (finish signal). If there is a designation of No. of times, the above control will be executed only for the first drilling.
- (6) 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.
- (7) 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) G83.1 X_ Y_ Z_ R_ Q_ P_ F_ L0_ ;

Memorize only the codes with an address

- (8) 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.
- (9) Other subprograms cannot be called from the fixed cycle subprogram.
- (10) Decimal points in the movement command of the fixed cycle subprogram will be ignored.
- (11) If the No. of repetitions L is 2 or more during the incremental value mode, the positioning will also be incremented each time.

(Example) G81 U-10. Z-50. R-20. F100. L3 ;



- (12) 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.
- (13) If inclinations 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 inclinations, the previous stage's inclination will be valid.
- (14) If the values set in the spindle basic specification parameter; tapping rotation speed and synchronous tapping changeover spindle rotation speed 2, exceed the maximum rotation speed, the spindle rotation speed will be clamped at the maximum rotation speed.
- (15) If the spindle rotation speed is not 0 during return, the tapping retract override value will be invalid.
- (16) 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.
(Example) G0 X100. ,I10.0
X-200. ;
- (17) 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.
- (18) 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.
- (19) 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.
- (20) Synchronous or asynchronous tapping can be selected.

Basic specification parameters

#	Item	Detail	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 tapping will follow the combination shown below.

Program command (G84/G84.2)	Combination											
	0	0	0	0	1	1	1	1	No command			
#8159 Synchronous tap	0	0	1	1	0	0	1	1	0	0	1	1
M function code (M**)	x	○	x	○	x	○	x	○	x	○	x	○
Synchronous/asynchronous selection	A	A	A	A	S	S	S	S	A	S	S	S

0: G84 (tapping cycle) x : Does not command A : Asynchronous tapping

1: G84.2 (synchronous tapping) ○ : Commands S : Synchronous tapping

(Note 1) Do not use M00, 01, 02, 30, 98 and 99.

(Note 2) Depending on the model, selection by the M function may not be available.

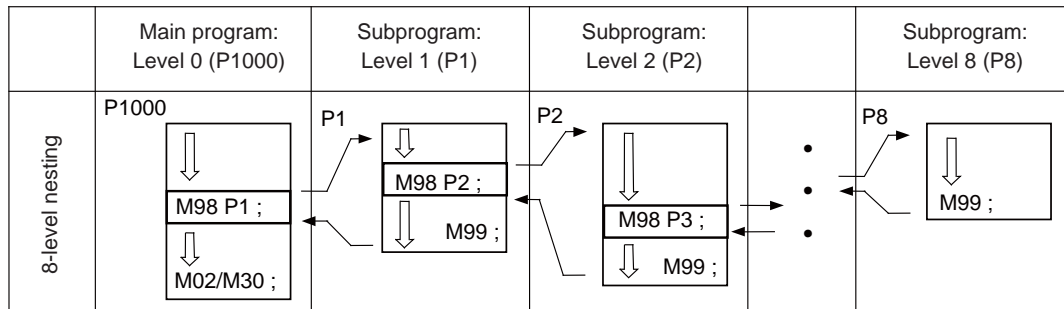
13.7 Subprogram Control; M98, M99, M198

13.7.1 Subprogram Call ; M98,M99



Function and purpose

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. 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 and the nesting depth can include as many as 8 levels.



The table below shows the functions which 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 (Note 2)	×	○	○	×
5. Subprogram nesting level call (Note 3)	×	○	○	×
6. Fixed cycles	×	×	○	○
7. Editing subprogram for fixed cycle	×	×	○	○

(Note 1) ○ denotes available functions and × denotes unavailable functions.

(Note 2) Variables cannot be transferred with the M98 command but variable commands in subprograms can be used provided that the variable command option is available.

(Note 3) A maximum of 8 nesting levels form the nesting depth.



Command format

Subprogram call

M98 P__ H__ L__ ,D__ ;

M98 <file name> H__ L__ ,D__ ;

P	<p>Program No. of subprogram to be called (own program if omitted) Note that P can be omitted only during memory mode and MDI mode. (Max. 8 digits) Use a parameter to specify a 4- or 8-digit subprogram No. starting with O. However, if the commanded value is bigger than the digit number set with parameter, a subprogram call is carried out as commanded.</p>
<File name>	<p>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> ;</p>
H	<p>Sequence No. in subprogram to be called (head block if omitted) (Max. 5 digits)</p>
L	<p>Number of subprogram repetitions (When omitted, this is interpreted as L1, and is not executed when L0.) (1 to 9999 times depending on the 4-digit value) For instance, For instance, M98 P1 L3 ; is equivalent to the following: M98 P1 ; M98 P1 ; M98 P1 ;</p>
,D	<p>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".</p>

Return to main program from subprogram

M99 P__ ;

P	Sequence No. of return destination (returned to block that follows the calling block)
---	---



Detailed description

Creating and registering subprograms

Subprograms have the same format as machining programs for normal memory mode, except that the subprogram completion instruction M99 (P_) ; must be registered as an independent block in the last block.

O***** ;	Program No. as subprogram No.
..... ; : ; ;	Main body of subprogram
M99 ;	Subprogram return command
%(EOR)	Registration completion code

- (1) The above program is registered by editing operations at the setting and display unit. For further details, refer to the section on "program editing" in the Instruction Manual.
- (2) Only those subprogram Nos. ranging from 1 to 99999999 designated by the optional specifications can be used. When there are no program Nos. on the tape, they are registered as the setting No. for "program input."
- (3) Up to 8 nesting levels can be used for calling programs from subprograms and program error (P230) will occur if this number is exceeded.
- (4) Main programs and subprograms are registered in the order they were read without distinction. Therefore, main programs and subprograms should not be given the same Nos. (If they are, error "E11" will be displayed at registration.)
- (5) Main programs can be executed during memory, tape or MDI mode but subprograms must be in the memory mode.
- (6) Besides the M98 command, subprogram nesting is subject to the following commands:
 - G65 : Macro call
 - G66 : Modal call
 - G66.1 : Modal call
 - G code call
 - Miscellaneous function call
 - MDI interruption
 - Automatic tool length measurement
 - Macro interruption
 - Multiple-step skip function
- (7) Subprogram nesting is not subject to the following commands which can be called even beyond the 8th nesting level.
 - Fixed cycles
 - Pattern cycles
- (8) To repeatedly use the subprogram, it can be repeated l1 times by programming M98 Pp1 Ll1;.
- (9) When using the multi-part system, if the subprogram attributed to the part system with the call command is empty, the subprogram call operation will change according to the parameters.

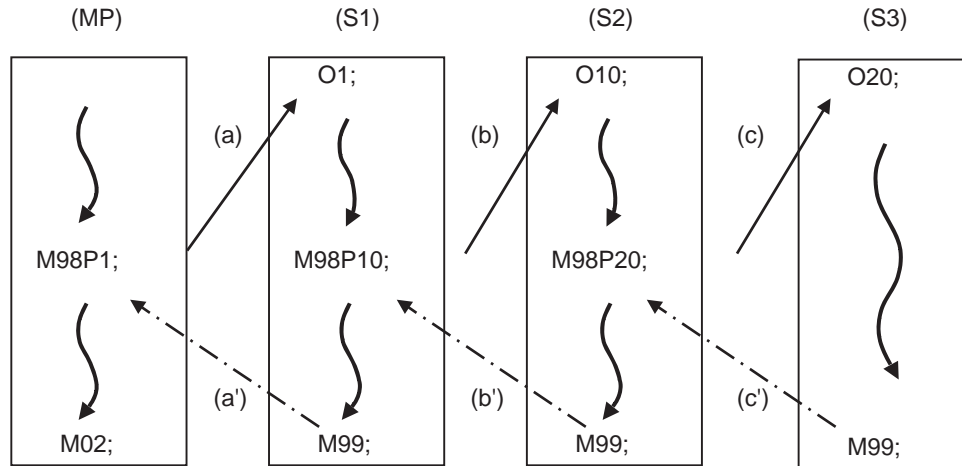
#1050 MemPrg	#1285 ext21/bit1	Details
0,2,4,6	-	The subprogram registered in the common memory of the part systems is called out.
1,3,5,7	OFF	The subprogram registered in the memory for the selected part system is called out.
	ON	The subprogram registered in the memory for the selected part system is called out. If the subprogram in the selected part system is empty, the subprogram with the same No. in the 1st part system is called out.



Program example

Program example 1

When there are 3 subprogram calls (known as 3 nesting levels)



(M) Main program

(S1) Subprogram 1

(S2) Subprogram 2

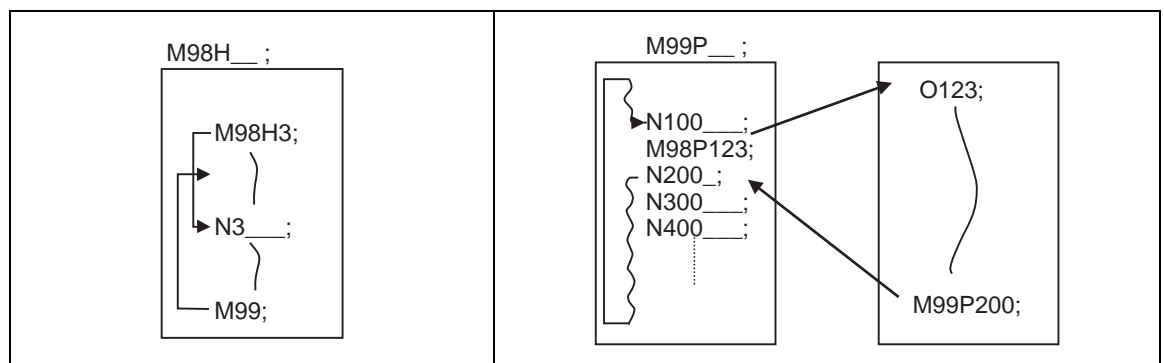
(S3) Subprogram 3

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) 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 same 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.

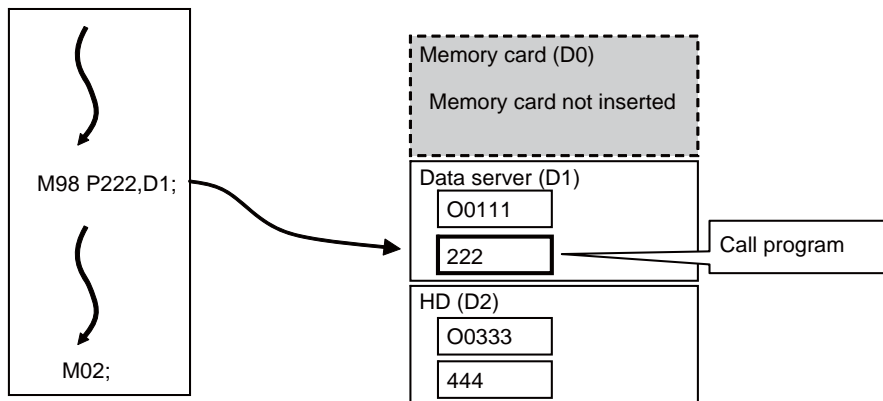
(Note) To prevent any unintended program call, avoid using such program names that may be confused.
(For example, 123, O0123 and O00000123 can be considered identical.)

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.

Main program



[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 disc)

(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.)

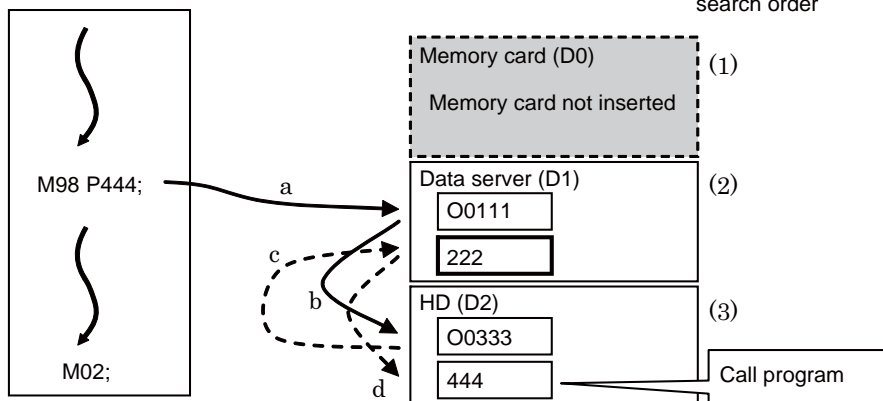
If a subprogram with O No. is not found, subprograms with a name designated with the P command are searched in order of the subprogram search setting. (Refer to the broken lines c,d.)

If none of the designated subprogram storage locations are subject to search, memories are searched.

* 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.

Main program

Subprogram search order



[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 disc)
- #8890 Subpro srch odr D0 = 1
- #8891 Subpro srch odr D1 = 2
- #8892 Subpro srch odr D2 = 3

13.7.2 Subprogram Call ; M198



Function and purpose

Programs registered in the data server (CF card in the control unit) of M700VW Series, or the front IC card (CF card) of M700VS/M70V Series can be called as subprograms. To call a program in the CF card as a subprogram, command in the main program as shown below.



Command format

Subprogram call

M198 P__ L__ ;

M198 <File name> L__ ;

P	Program No. in CF card to be called as subprogram. (Max. 8 digits) Use a parameter to specify a 4- or 8-digit subprogram No. starting with O. However, if the commanded value is bigger than the digit number set with parameter, a subprogram call is carried out as commanded.
<File name>	File name A file name can be specified instead of a program No. In this case, enclose the file name with brackets <>. (The file name can have up to 32 characters including the extension.)
L	Number of subprogram repetitions. (Max. 4 digits) This can be omitted. (In this case, the subprogram will be called once.) When "LO" is designated, the subprogram call will not be executed.

(Note) Sequence No. call (M198 H***) cannot be commanded.

Return to main program from subprogram

M99 ;



Detailed description

- (1) The subprogram can be called with the M198 command once in the subprogram nest. The subprogram can be called only from the memory or MDI program.
- (2) The section from the head of the program to the first LF (line feed code, 0x0A hexadecimal) is invalid, and is not run or displayed. Note that if the head starts with a 0 No., the program will be valid from the head.
- (3) A program registered in an IC card can be executed from only one part system. A program error will occur if it is attempted to execute the programs in the IC card simultaneously by two or more part systems. If all the part system is reset when the error occurred, programs will be displayed as only "%" except for the first part system.
- (4) Refer to "Subprogram call ; M98,M99" for <File name> and calling the subprogram with O No.

13.8 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 condition of executing the program.



Command format

```
##** = 00000000 ;
```

```
##** = [formula] ;
```



Detailed description

Variable expressions

		Example
#m	m = value consisting of 0 to 9	#100
# [f]	f = one of the followings in the formula	# [-#120]
	Numerical value m	123
	Variable	#543
	Formula Operator Formula	#110+#119
	- (minus) formula	-#120
	[Formula]	[#119]
	Function [formula]	SIN [#110]

(Note 1) The 4 standard operators are +, -, * and /.

(Note 2) Functions cannot be used unless the user macro specifications are available.

(Note 3) Error (P241) will occur when a variable No. is negative.

(Note 4) Examples of incorrect variable expressions are given below.

Incorrect	Correct
#6/2	# [6/2] (#6/2 is regarded as [#6] /2)
#--5	# [-[-5]]
#- [#1]	# [-#1]

Types of variables

The following table gives the types of variables.

Type		No.		Function
Common variables		Common variables 1	Common variables 2	- Can be used in common throughout main, sub and macro programs. - When using common variables in the multi-part system, the number of common variable shared among the part systems can be specified by the parameter "#1052 MemVal". (Note 5)
1st part system	100 sets	500 to 549	100 to 149	
	200 sets	500 to 599	100 to 199	
	300 sets	500 to 699	100 to 199	
	600 sets	500 to 999 100100 to 800199 (Note 7)	100 to 199	
	700 sets	400 to 999 (Note 4) 100100 to 800199 (Note 7)	100 to 199	
	8000 sets	400 to 999(Note 4) 100100 to 800199(Note 7) 900000 to 907399	100 to 199	
Multi-part system (n = number of part systems)	50 + 50 * n sets	500 to 549	100 to 149 * n	
	100 + 100 * n sets	500 to 599	100 to 199 * n	
	200 + 100 * n sets	500 to 699	100 to 199 * n	
	500 + 100 * n sets	500 to 999 100100 to 800199 (Note 7)	100 to 199 * n	
	600 + 100 * n sets	400 to 999 (Note 4) 100100 to 800199 (Note 7)	100 to 199 * n	
	7900+100 sets	400 to 999(Note 4) 100100 to 800199(Note 7) 900000 to 907399	100 to 199 *n	
Local variables		1 to 33		Can be used as local variables in macro programs.
System variable		From 1000		Application is fixed by system.
Fixed cycle variables		1 to 32		Local variables in fixed cycle programs.

- (Note 1) All common variables are retained even when the power is turned OFF.
- (Note 2) When the power is turned OFF or reset, the common variables can be set to <null> by setting the parameter (#1128 RstVC1, #1129 PwrVC1).
- (Note 3) The common variables are divided into the following two types.
 Common variables 1 : Used in common through all part systems
 Common variables 2 : Used in common in the programs of the part system
- (Note 4) Common variable address #400s can be used only when there are 700 sets or more of common variables and the parameter "#1336 #400_Valtype" is "1". If address #400s common variable is used when the set of common variable is less than 700 sets or the parameter "#1336 #400_Valtype" is "0", a program error (P241) will occur.
 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.
- (Note 5) When the parameter "#1052 MemVal" is set to "1" in multi-part system, a part or all of common variables "#100 to #199" and "#500 to #999" can be shared and used between part systems. Set the number of common variables by the parameter "#1303 V1comN" (#100 to set value) and "#1304 V0comN" (#500 to set value).

(EX)

Set "10" to "#1303 V1comN"

#100 to #109 : Common for the part systems

#110 to #199 : Each part system

Set "5" to "#1304 V0comN"

#500 to #504 : Common for the part systems

#505 to #999 : Each part system

When the parameter "#1052 MemVal" is set to "0", the common variables #100 to #199 are used for each part system, and variables #500 to #999 are common for the part systems. Address #400s and #900000 to #908399 which can be used with the 8000 sets of variable, which can be used as common variable with 700 or more sets of variable, is common for the part systems regardless of the setting of parameter "#1052 MemVal".

- (Note 6) When inputting the common variable data, if the following illegal variable No. data exist in the input file, the illegal variable No. data is ignored and only the correct common variable data will be input.
- Variable data which is not common variables such as local variables (#1 to #33) and system variables (from #1000), etc.
 - Variable data with which conditions of number of common variable sets do not match.

(Example)

If the #100 to #199, #500 to #599 exist in the input file when the common variable 100 sets (#100 to #149, #500 to #549), the #150 to #199, #550 to #599 are ignored, and #100 to #149, #500 to #549 will be input.

(Note 7) When the parameter "#1316 CrossCom" is set to "1", the common variables #100100 to #800199 can be used as common for the part systems. The part system common variable which can be used is shown in the table below.

Variable sets		Common variables 1 (When "#1316 CrossCom" = "1")
Variable sets specification	600 sets (500 + 100)	#100100 to #100199 (Equivalent to # 100 to #199 in 1st part system) #200100 to #200199 (Equivalent to # 100 to #199 in 2nd part system) #300100 to #300199 (Equivalent to # 100 to #199 in 3rd part system)
	700 sets (600 + 100)	#400100 to #400199 (Equivalent to # 100 to #199 in 4th part system) #500100 to #500199 #600100 to #600199
	8000 sets (7900 + 100)	#700100 to #700199 #800100 to #800199

(Example)

<1-part system>

#100100=200 ; Equivalent to #100 = 200 ;
 #200105=#100 ; "200" is set to #200105
 #300110=#100100 ; "200" is set to #300110
 #800199=#500120 ; The variable value of #500120 is set to #800199

<Multi-part system>

Common variables for each part system #100 to #199" in other part system can be used.

\$1

#200100=-100; "-100" is set to #100 of 2nd part system.
 #101=#200102 ; The variable value of #102 of 2nd part system is set to #101
 #300105=#200103 ; The variable value of #103 of 2nd part system is set to #105 of 3rd part system
 #110=#500107 ; The variable value of #500107 is set to #110

- The PLC data reading function cannot be used, which uses system variables #100100 to #100110, and variables #100100 to #100110 are used as common variables.

- The setting of number of common variables shared between the part systems (The parameter #1052 MemVal" is set to "1") becomes invalid, thus the movement is the same as "0" is set.

- When the parameters (#1128 RstVC1, #1129 PwrVC1) are set to "1", the operation is as follows.

"#1128 RstVC1" (When reset, variables is empty)

Common variables shared between the part systems equivalent to #100 to #199 of 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 PwrVC1" (When power-ON, variable is empty)

Common variables shared between the part systems equivalent to #100 to #199 of valid part system are cleared.

(Example) In 1st part system, #100100 to #100199 are cleared.

In 2nd part system, #100100 to #100199 and #200100 to #200199 are cleared.

- Common variables shared between the part systems #100100 to #800199 can be displayed and set on the common variable screen.

- If common variables #100100 to #800199 are used when the number of sets of common variables is less than 600 sets or the parameter "#1316 CrossCom" is "0", a program error (P241) will occur.

Variable quotations

Variables can be used for all addresses except O, N and / (slash).

- (1) When the variable value is used directly:
X#1 Value of #1 is used as the X value.
- (2) When the complement of the variable value is used:
X-#2 Value with the #2 sign changed is used as the X value.
- (3) When defining variables:
#3 = #5 Variable #3 uses the equivalent value of variable #5.
#1 = 1000 Variable #1 uses the equivalent value 1000 (which is treated as 1000.).
- (4) When defining the variable arithmetic formula:
#1 = #3 + #2 - 100 Value of the operation result of #3 + #2 - 100. is used as the #1 value.
X [#1 + #3 + 1000] Value of the operation result of #1 + #3 + 1000 is used as the X value.

(Note 1) A variable cannot be defined in the same block as an address. It must be defined in a separate block.

Incorrect	Correct
X#1 = #3 + 100 ;	#1 = #3 + 100 ; X#1 ;

- (Note 2) Up to five sets of square parentheses [] may be used.
#543 = -[[[[[#120]/2+15.]*3-#100]/#520+#125+#128]*#130+#132]
- (Note 3) There are no restrictions on the number of characters and number of variables for variable definition.
- (Note 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.
- (Note 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
- (Note 6) Variable quotations are always regarded as having a decimal point at the end.
When #100 = 10
X#100 ; is treated as X10.

13.9 User Macro

13.9.1 User Macro

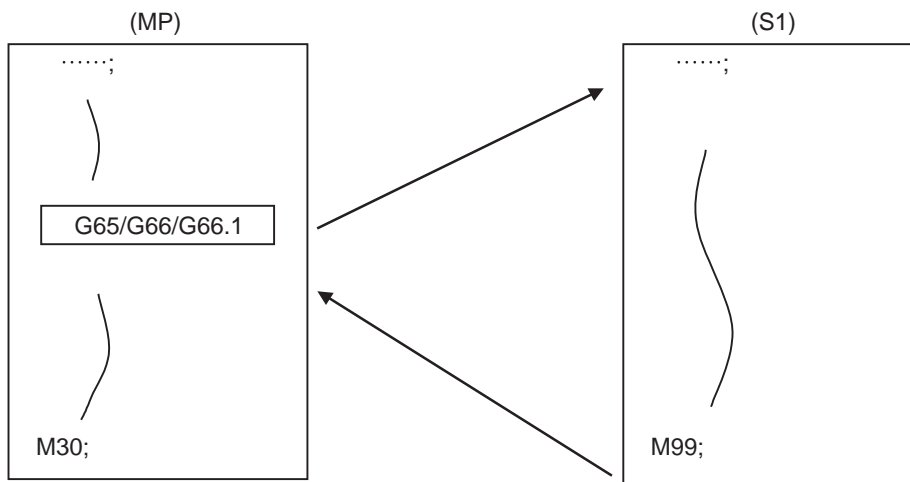


Function and purpose

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.



(MP) Main program

(S1) Macro program (subprogram)

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.

13.9.2 Macro Call Instruction



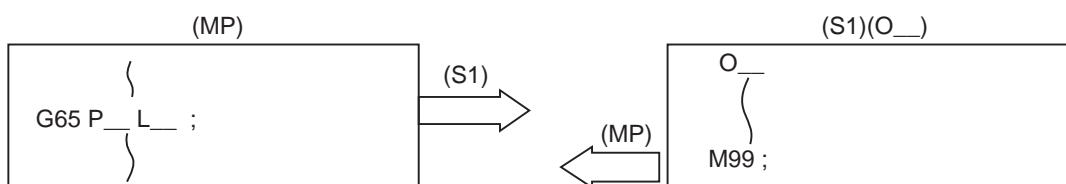
Function and purpose

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.

13.9.2.1 Simple Macro Calls ; G65



Function and purpose



M99 is used to terminate the user macro subprogram.

(MP) Main program

(S1) Subprogram



Command format

G65 P__ L__ argument ; ... Simple macro calls

G65 <File name> L__ argument ; ... Simple macro calls

P	Program No. 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:
Argument	Specify variable data



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	x	x *
H	#11	○	○
I	#4	○	○
J	#5	○	○
K	#6	○	○
L	#12	x	x *
M	#13	○	○
N	#14	x	x *
O	#15	x	x
P	#16	x	x *
Q	#17	○	○
R	#18	○	○
S	#19	○	○
T	#20	○	○
U	#21	○	○
V	#22	○	○
W	#23	○	○
X	#24	○	○
Y	#25	○	○
Z	#26	○	○

○ : Can be used
 x : Cannot be used
 * : Can be used while G66.1 command is modal

Argument designation II

Format : A__ B__ C__ I__ J__ K__ I__ J__ K__ ...

- (a) In addition to address A, B and C, up to 10 groups of arguments with I, J, K serving as 1 group can be designated.
- (b) When the same address is duplicated, designate the addresses in the specified order.
- (c) Addresses which do not need to be designated can be omitted.
- (d) The following table shows the correspondence between the addresses which can be designated by argument designation II and the variable numbers in the user macro main body.

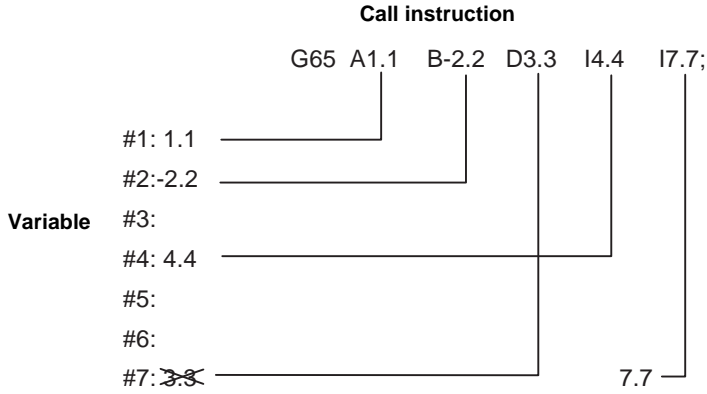
Argument specification II address	Variable in macro		Argument specification II address	Variable in macro
A	#1		J5	#17
B	#2		K5	#18
C	#3		I6	#19
I1	#4		J6	#20
J1	#5		K6	#21
K1	#6		I7	#22
I2	#7		J7	#23
J2	#8		K7	#24
K2	#9		I8	#25
I3	#10		J8	#26
J3	#11		K8	#27
K3	#12		I9	#28
I4	#13		J9	#29
J4	#14		K9	#30
K4	#15		I10	#31
I5	#16		J10	#32
			K10	#33

(Note 1) Subscripts 1 to 10 for I, J, and K indicate the order of the specified command sets. They are not required to specify instructions.

Using arguments designations I and II together

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 program number starting with O and specified by P command value is called with a priority.

However, when P command value is less than the digit number set with parameter "#8129 subprogram number selection", increase the digit number of command value by adding leading zeros.

(Example)

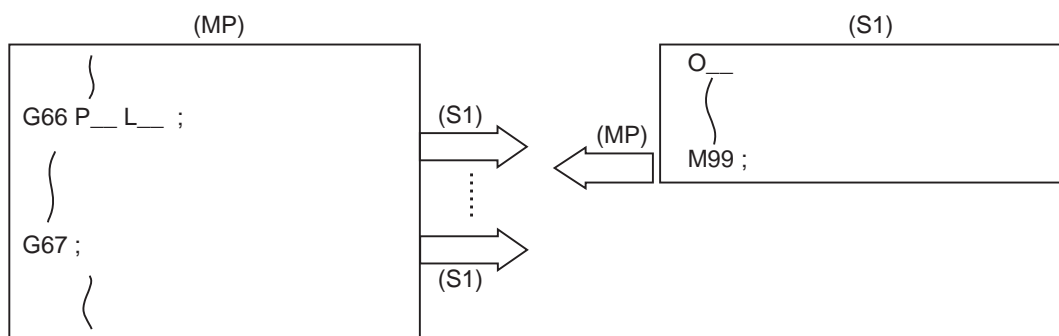
When parameter "#8129 subprogram number selection"="1", call the subprogram "O0012" with "G65 P12" command.

- (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 subprogram number selection".
 - A subprogram starting with commanded O No. does not exist.

13.9.2.2 Modal Call A (Movement Command Call) ; G66



Function and purpose



(MP) Main program

(S1) Subprogram

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. 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:
Argument	Specify variable data

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.
- (3) If calling a subprogram numbered with O is enabled, a program number starting with O and specified by P command value is called with a priority.
However, when P command value is less than the digit number set with parameter "#8129 subprogram number selection", increase the digit number of command value by adding leading zeros.
(Example)
When parameter "#8129 subprogram number selection"="1", call the subprogram "O0012" with "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 subprogram number selection".
 - A subprogram starting with commanded O No. does not exist.

13.9.2.3 Modal Call B (for each block) ; G66.1



Function and purpose

The specified user macro subprogram is called unconditionally for each command block which 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. 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
Argument	Specify variable data

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 G65P__ is assigned at the head of a block.
(Example 1)
In "G66.1 P1000 ; " mode,
N100 G01 G90 X100. Z100. F400 R1000 ; is same as
N100 G65 P1000 G01 G90 X100. Z200. F400 R1000 ;.
- (Note 1) 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 which 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 program number starting with O and specified by P command value is called with a priority.
However, when P command value is less than the digit number set with parameter "#8129 subprogram number selection", increase the digit number of command value by adding leading zeros.
(Example)
When parameter "#8129 subprogram number selection"="1", call the subprogram "O0012" with "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 subprogram number selection".
 - A subprogram starting with commanded O No. does not exist.

13.9.2.4 G Code Macro Call



Function and purpose

User macro subprogram with prescribed program numbers can be called merely by issuing the G code command.



Command format

G** argument ; ... G code macro call	
G**	G code for macro call



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.
 - a : M98 P $\Delta\Delta\Delta\Delta$;
 - b : G65 P $\Delta\Delta\Delta\Delta$ Argument ;
 - c : G66 P $\Delta\Delta\Delta\Delta$ Argument ;
 - d : G66.1 P $\Delta\Delta\Delta\Delta$ Argument ;
 When the parameters corresponding to c and d above are set, issue the cancel command (G67) either in the user macro or after the call code has been commanded so as to cancel the modal call.
- (2) The correspondence between the "*" which conducts the macro call and the macro program number P $\Delta\Delta\Delta\Delta$ to be called is set by parameters.
- (3) Up to 10 G codes from G100 to G999 can be used with this instruction. G codes from G01 to G99 can also be made available for use by parameter setting "#1081 Gmac_P" even though they are used in the system.
- (4) These commands cannot be issued in a program which has been called by a G code macro. If issued in such a program, they will be handled as ordinary G commands.

13.9.2.5 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

M ; (or S** ; T** ; B** ;) ... Miscellaneous command macro call**

M**	M code for macro call (or S, T, B code)
-----	---



Detailed description

- (1) The above instruction functions in the same way as the instructions below, however, the correspondence between M codes and instructions can be set by parameters. (Same for S, T and B codes)

a:M98 P**** ;	M98, M** are not output.
b:G65 P**** M** ;	
c:G66 P**** M** ;	
d:G66.1 P**** M** ;	

When the parameters corresponding to c and d above are set, issue the cancel command (G67) either in the user macro or after the call code has been commanded so as to cancel the modal call.

- (2) The correspondence between the "M**" which conducts the macro call and the macro program number P**** to be called is set by parameters. Up to 10 M codes from M00 to M9999 can be registered. Note that the codes to be registered should exclude those basically required for the machine and M0, M1, M2, M30, M96 to M99, and M198.
 - (3) As with M98, it is displayed on the screen display of the setting and display unit but the M codes and MF are not output.
 - (4) Even if the registered miscellaneous commands above are issued in a user macro subprogram which are called by an M code, it will not be regarded as a macro call and will be handled as a normal miscellaneous command. (Same for S, T and B codes)
 - (5) All S, T and B codes call the subprograms in the prescribed program numbers of the corresponding S, T and B functions.
 - (6) Up to 10 M codes can be set.
- (Note) If a value 1 to 3 is set to "#7002 M[01] Type", a macro call equivalent to G65, G66 or G66.1 will be executed. In this case the alphabet before M,S,T,B code macro is not treated as an argument.

(Preparation)

Register M06 to M code macro.

(Command)

M06 T02 ... The value of T is treated as variable #20 in macro. At the same time a value will be inserted to T code.

T02 M06 ... A value will not be input to variable #20. A value will be input to T code.

13.9.2.6 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 no 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 8 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.

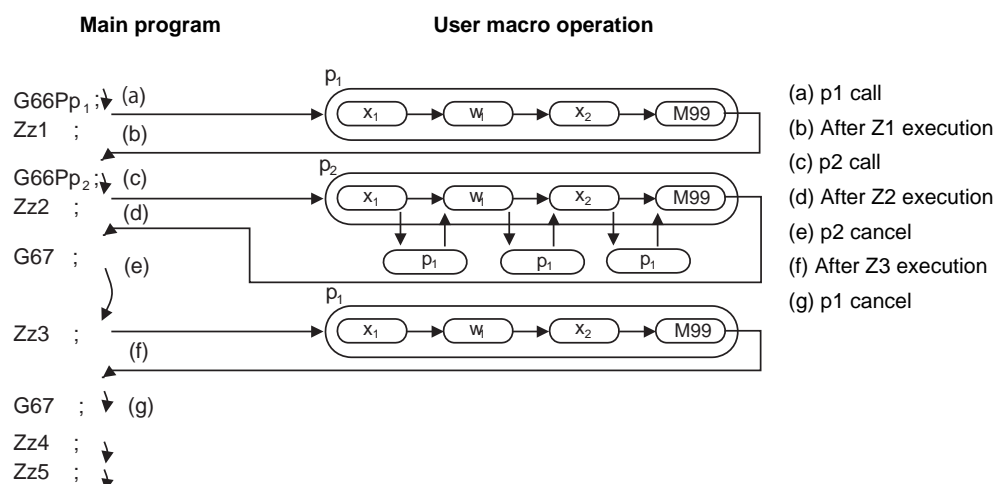
Macro call command nesting depth

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.

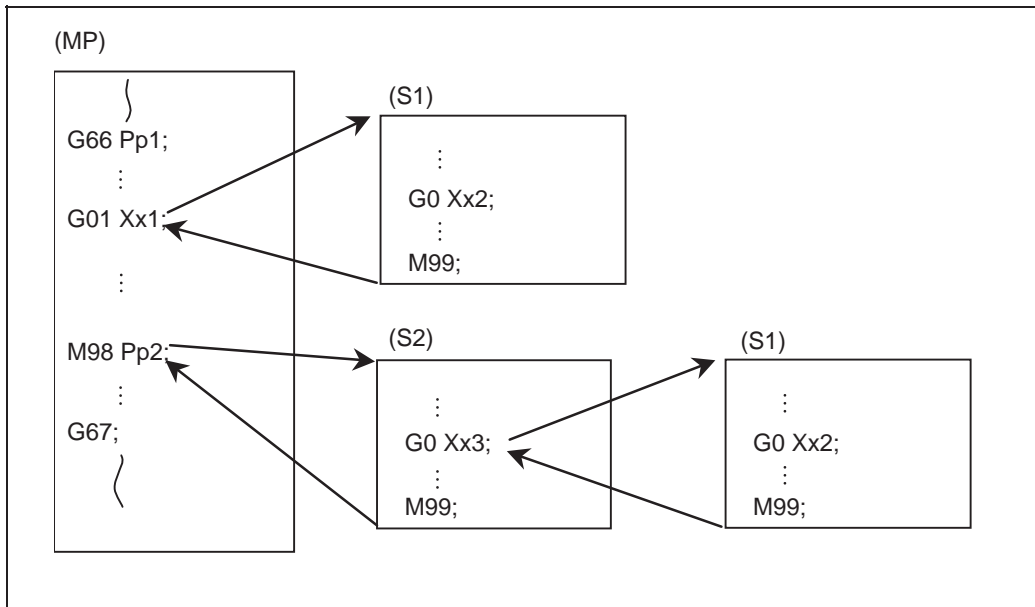
- (Note 1) When a G65, G66, G66.1 G code macro call or miscellaneous command macro call is conducted, this is regarded as a nesting level and the level of the local variables is also incremented by one.
- (Note 2) With modal call A, the designated user macro subprogram is called every time a movement 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)



(Note 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)



(MP) Main program

(S1),(S2) Subprogram

When the program numbers of p1 and p2 are same, the program numbers of (S1) and (S2) will be same.

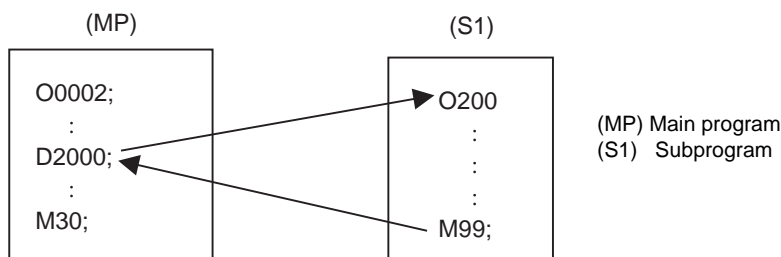
13.9.3 ASCII Code Macro



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.

(Execution example 1) M98 type

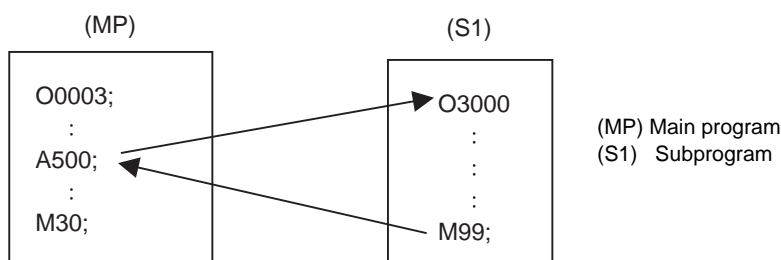


After outputting 2000 to common variable #146, the program No. 200 subprogram is called with the M98 subprogram call type.

Parameter

#7401 (ASCII [01] Valid/Invalid)	1 (Valid)
#7402 (ASCII [01] Code)	D
#7403 (ASCII [01] Type)	0 (M98 type)
#7404 (ASCII [01] Program No.)	200
#7405 (ASCII [01] Variable)	146

(Execution example 2) G65 type



After outputting 500 to local variable #1, the program No. 3000 subprogram is called out with the G65 macro call type.

Parameter

#7411 (ASCII [02] Valid/Invalid)	1 (Valid)
#7412 (ASCII [02] Code)	A
#7413 (ASCII [02] Type)	1 (G65 type)
#7414 (ASCII [02] Program No.)	3000
#7415 (ASCII [02] Variable)	100 (Not used)



Command format

<input type="checkbox"/> **** ; ... Designates the address and code	
<input type="checkbox"/>	ASCII code for calling out a macro (one character)
****	Value or expression output to variable (Setting range: ± 999999.9999)



Detailed description

- (1) The command above performs same operations as 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 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).

(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



Program example

The figure below describes a program that controls the tailstock with address A and machines a long thin workpiece.

#7411 (ASCII [02] Valid/Invalid)	1 (Valid)
#7412 (ASCII [02] Code)	A
#7413 (ASCII [02] Type)	1 (G65 type)
#7414 (ASCII [02] Program No.)	500
#7415 (ASCII [02] Variable)	100 (Not used)

Main program

G28 XZ;

A1; Call O500

G00 X20. Z0.;

G01 X15. F100;

G01 Z100.

F200;

X50 Z120. ;

A0; Call O500

M30 ;

Subprogram

O500;

IF[#1 EQ 0]GOTO

10;

G53 G00 A-205.; Tailstock advance

G53 G01 A-200.

F100;

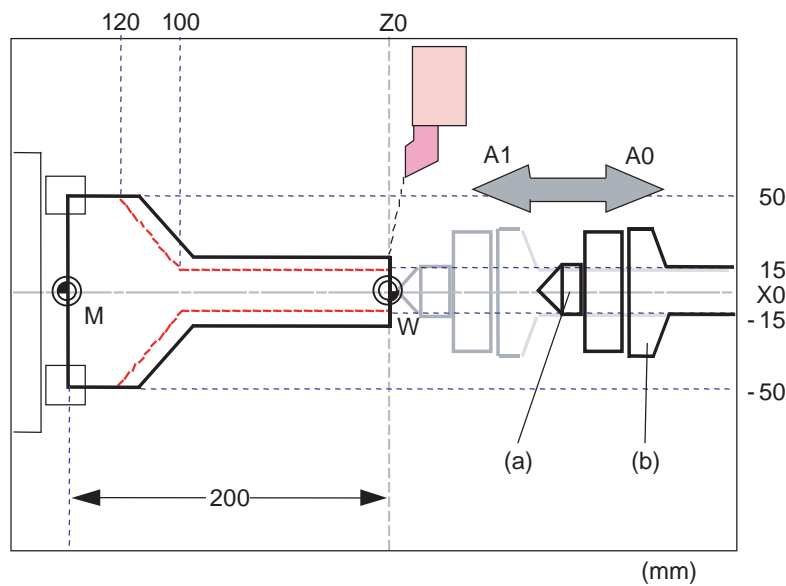
GOTO N20;

N10 M26;

G53 G0 A-400.; Tailstock retract

N20 M99;

Address A is converted into G65 P500.



(a) Tailstock spindle

(b) Tailstock A spindle

Press with A1

Pull with A0



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 command

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 macro call command

Counting the main program as 0, up to eight 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 cycle
- (8) Macro interruption

Order of command priority

If "M" is designated for the ASCII code address, it may overlap with the codes basically necessary for that machine. In this case, commands will be identified with the following priority using code values.

- (1) M98,M99 (subprogram call command)
M00 (program stop command), M01 (optional stop command)
M02,M30,M198,M199 (end command)
M96,M97 (macro interruption command)

(2) When corresponding to ASCII code macro command

(3) Used as normal command

"S", "T" and "B" will also overlap with the miscellaneous command macro call. However, if these commands do not correspond to the identification, the ASCII code macro will be executed.

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 (5) 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. 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)

13.9.4 Variable



Function and purpose

Both the variable specifications and user macro specifications are required for the variables which 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 PwrVC1".)



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>

```
G0 X#1 Z1000 ; .....Equivalent to G0 Z1000 ;
G0 X#1 + 10 Z1000 ; ..... Equivalent to G0 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> ≥ 0 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 used only for integers. For comparison of numeric values with decimals, GE, GT, LE, and LT should be used.

13.9.5 Types of Variables

13.9.5.1 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 #519. It must be composed of not more than 7 alphanumeric characters and it must begin with a letter. Do not use "#" in variable names. It causes an alarm when the program is executed.

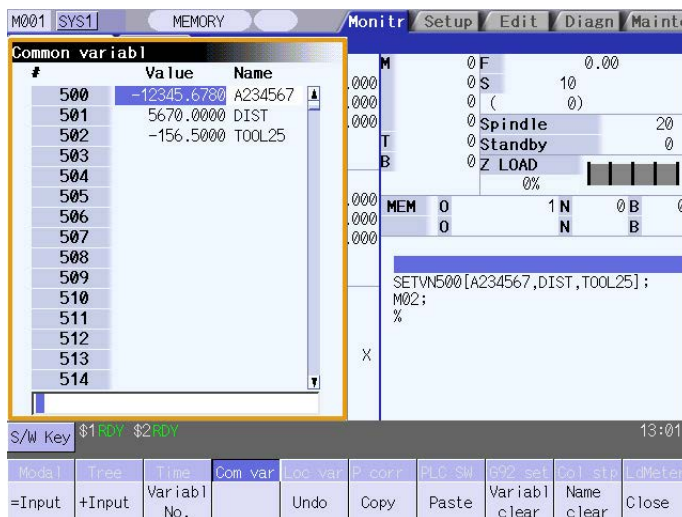
SETVn [NAME1,NAME2,] ;

n	Head No. of variable to be named
NAME1	#n name (variable name)
NAME2	#n + 1 name (variable name)

Variable names are separated by a comma (,).

- Once variable names have been set, they will not be cleared even when the power is turned off.
- 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];
- 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) Do not use characters (SIN, COS, etc.) predetermined by the NC and used for operation commands at the head of a variable name.

13.9.5.2 Local Variables (#1 to #33)



Detailed description

Local variables can be defined as an <argument> when a macro subprogram is called, and also used locally within main programs and subprograms. They can be duplicated because there is no relationship between macros. (up to 4 levels)

G65 P__ L__ <argument> ;

P	Program No.
L	Number of repetitions

The <argument> is assumed to be Aa1 Bb1 Cc1..... Zz1.

The following table shows the correspondences 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.	Call command		Argument address	Local variable No.
G65 G66	G66.1			G65 G66	G66.1		
○	○	A	#1	○	○	Q	#17
○	○	B	#2	○	○	R	#18
○	○	C	#3	○	○	S	#19
○	○	D	#7	○	○	T	#20
○	○	E	#8	○	○	U	#21
○	○	F	#9	○	○	V	#22
x	x *	G	#10	○	○	W	#23
○	○	H	#11	○	○	X	#24
○	○	I	#4	○	○	Y	#25
○	○	J	#5	○	○	Z	#26
○	○	K	#6			-	#27
x	x *	L	#12			-	#28
○	○	M	#13			-	#29
x	x *	N	#14			-	#30
x	x	O	#15			-	#31
x	x *	P	#16			-	#32
						-	#33

"x" 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.

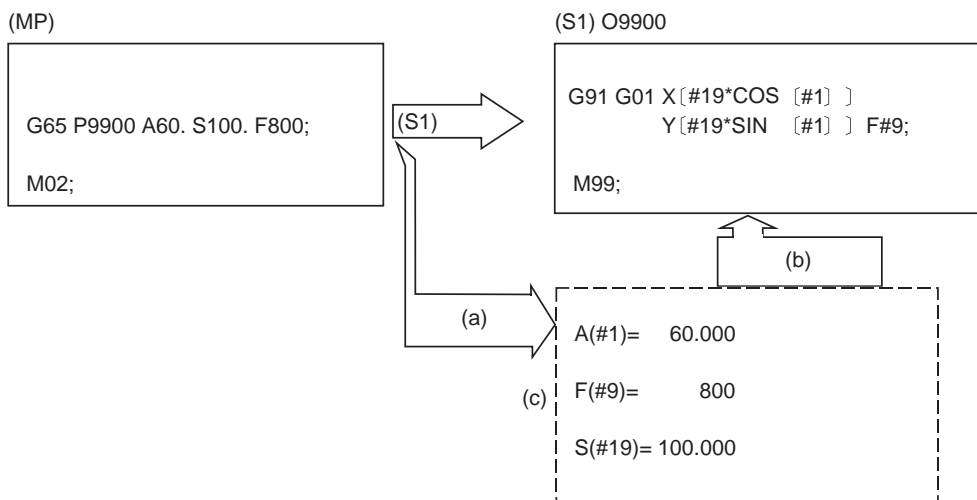
The hyphen (-) mark indicates that there is no corresponding address.

[Argument designation II]

Argument designation II address	Variable in macro		Argument designation II address	Variable in macro
A	#1		J5	#17
B	#2		K5	#18
C	#3		I6	#19
I1	#4		J6	#20
J1	#5		K6	#21
K1	#6		I7	#22
I2	#7		J7	#23
J2	#8		K7	#24
K2	#9		I8	#25
I3	#10		J8	#26
J3	#11		K8	#27
K3	#12		I9	#28
I4	#13		J9	#29
J4	#14		K9	#30
K4	#15		I10	#31
I5	#16		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.

- (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.)



(MP) Main program

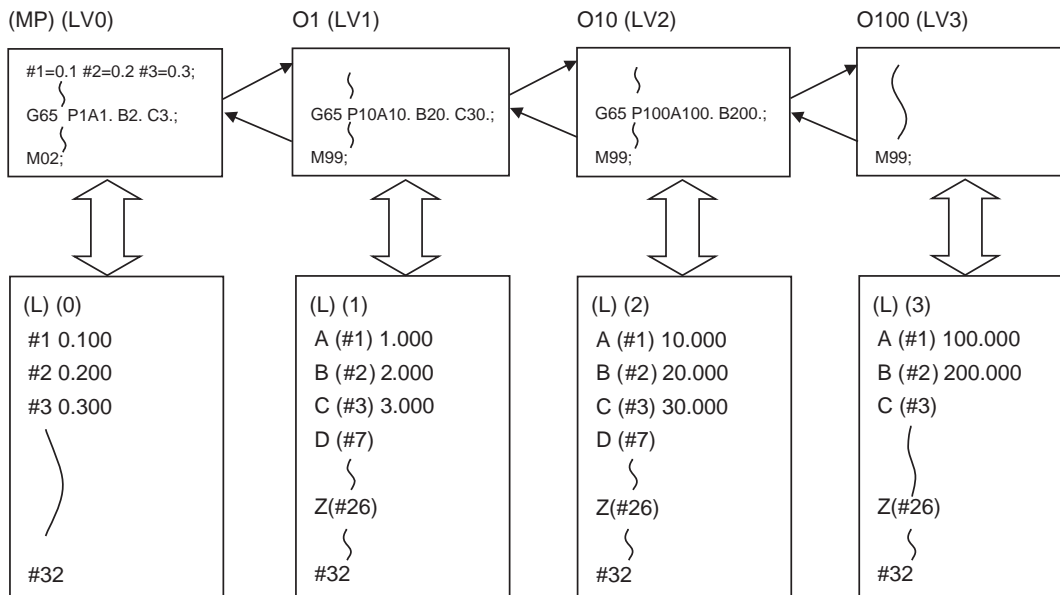
(S1) Subprogram

(a) Local variables set by argument

(b) Refer to the local variables and control the movement, etc.

(c) Local variable data table

- (2) 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.



(MP) Main program

(L) Local variables

(LV0 - 3) Macro level 0 - 3

The status of the local variables is displayed on the setting and display unit. Refer to the Instruction Manual for details.

13.9.5.3 Macro Interface Inputs/Outputs (#1000 to #1035, #1100 to #1135, #1200 to #1295, #1300 to #1395)

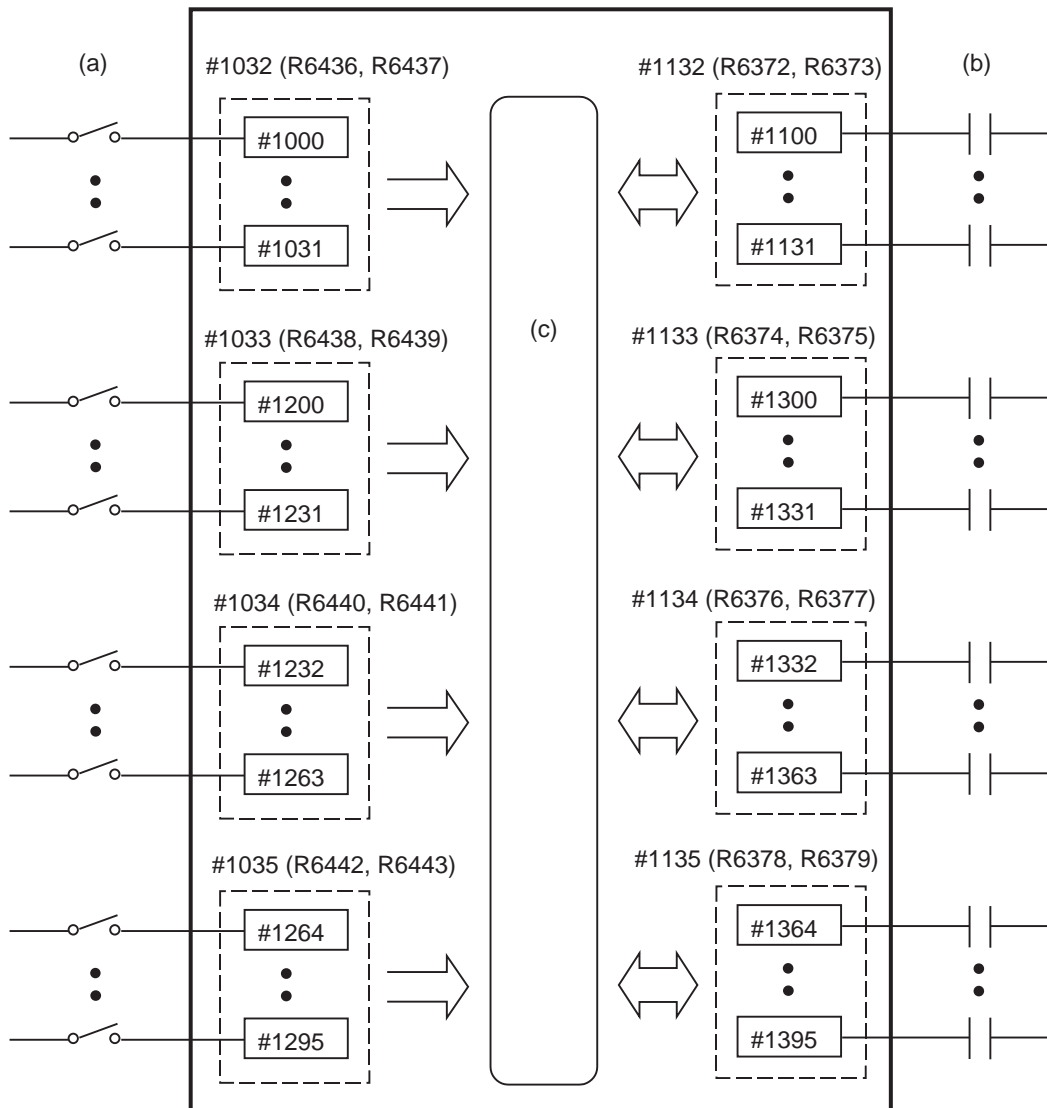


Function and purpose

The status of the interface input signals can be ascertained by reading out the values of variable numbers #1000 to #1035, #1200 to #1295.

The interface output signals can be sent by substituting values in variable Nos. #1100 to #1135, #1300 to #1395.

Example of 1st part system



- (a) Input signal
- (b) Output signal
- (c) Macro instructions



Detailed description

Macro interface inputs (#1000 to #1035, #1200 to #1295) : PLC -> NC

A variable value which has been read out can be only 1 or 0 (1:contact closed, 0:contact open). All the input signals from #1000 to #1031 can be read at once by reading out the value of variable No. #1032. Similarly, the input signals #1200 to #1231, #1232 to #1263, and #1264 to #1295 can be read by reading the values of the variable Nos. #1033 to #1035.

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.

Whether it is per part system or common among part systems depends on the setting of the base specification parameter "#1230 set02/bit7".

\$1	\$2	\$3	\$4
R6436,7	R6444,5	R6452,3	R6460,1

Following correspondence tables are example of the 1st part system.

System variable	No. of points	Interface input signal	System variable	No. of points	Interface input signal
#1000	1	Register R6436 bit 0	#1016	1	Register R6437 bit 0
#1001	1	Register R6436 bit 1	#1017	1	Register R6437 bit 1
#1002	1	Register R6436 bit 2	#1018	1	Register R6437 bit 2
#1003	1	Register R6436 bit 3	#1019	1	Register R6437 bit 3
#1004	1	Register R6436 bit 4	#1020	1	Register R6437 bit 4
#1005	1	Register R6436 bit 5	#1021	1	Register R6437 bit 5
#1006	1	Register R6436 bit 6	#1022	1	Register R6437 bit 6
#1007	1	Register R6436 bit 7	#1023	1	Register R6437 bit 7
#1008	1	Register R6436 bit 8	#1024	1	Register R6437 bit 8
#1009	1	Register R6436 bit 9	#1025	1	Register R6437 bit 9
#1010	1	Register R6436 bit 10	#1026	1	Register R6437 bit 10
#1011	1	Register R6436 bit 11	#1027	1	Register R6437 bit 11
#1012	1	Register R6436 bit 12	#1028	1	Register R6437 bit 12
#1013	1	Register R6436 bit 13	#1029	1	Register R6437 bit 13
#1014	1	Register R6436 bit 14	#1030	1	Register R6437 bit 14
#1015	1	Register R6436 bit 15	#1031	1	Register R6437 bit 15

System variable	No. of points	Interface input signal
#1032	32	Register R6436, R6437
#1033	32	Register R6438, R6439
#1034	32	Register R6440, R6441
#1035	32	Register R6442, R6443

System variable	No. of points	Interface input signal	System variable	No. of points	Interface input signal
#1200	1	Register R6438 bit 0	#1216	1	Register R6439 bit 0
#1201	1	Register R6438 bit 1	#1217	1	Register R6439 bit 1
#1202	1	Register R6438 bit 2	#1218	1	Register R6439 bit 2
#1203	1	Register R6438 bit 3	#1219	1	Register R6439 bit 3
#1204	1	Register R6438 bit 4	#1220	1	Register R6439 bit 4
#1205	1	Register R6438 bit 5	#1221	1	Register R6439 bit 5
#1206	1	Register R6438 bit 6	#1222	1	Register R6439 bit 6
#1207	1	Register R6438 bit 7	#1223	1	Register R6439 bit 7
#1208	1	Register R6438 bit 8	#1224	1	Register R6439 bit 8
#1209	1	Register R6438 bit 9	#1225	1	Register R6439 bit 9
#1210	1	Register R6438 bit 10	#1226	1	Register R6439 bit 10
#1211	1	Register R6438 bit 11	#1227	1	Register R6439 bit 11
#1212	1	Register R6438 bit 12	#1228	1	Register R6439 bit 12
#1213	1	Register R6438 bit 13	#1229	1	Register R6439 bit 13
#1214	1	Register R6438 bit 14	#1230	1	Register R6439 bit 14
#1215	1	Register R6438 bit 15	#1231	1	Register R6439 bit 15

System variable	No. of points	Interface input signal	System variable	No. of points	Interface input signal
#1232	1	Register R6440 bit 0	#1248	1	Register R6441 bit 0
#1233	1	Register R6440 bit 1	#1249	1	Register R6441 bit 1
#1234	1	Register R6440 bit 2	#1250	1	Register R6441 bit 2
#1235	1	Register R6440 bit 3	#1251	1	Register R6441 bit 3
#1236	1	Register R6440 bit 4	#1252	1	Register R6441 bit 4
#1237	1	Register R6440 bit 5	#1253	1	Register R6441 bit 5
#1238	1	Register R6440 bit 6	#1254	1	Register R6441 bit 6
#1239	1	Register R6440 bit 7	#1255	1	Register R6441 bit 7
#1240	1	Register R6440 bit 8	#1256	1	Register R6441 bit 8
#1241	1	Register R6440 bit 9	#1257	1	Register R6441 bit 9
#1242	1	Register R6440 bit 10	#1258	1	Register R6441 bit 10
#1243	1	Register R6440 bit 11	#1259	1	Register R6441 bit 11
#1244	1	Register R6440 bit 12	#1260	1	Register R6441 bit 12
#1245	1	Register R6440 bit 13	#1261	1	Register R6441 bit 13
#1246	1	Register R6440 bit 14	#1262	1	Register R6441 bit 14
#1247	1	Register R6440 bit 15	#1263	1	Register R6441 bit 15

System variable	No. of points	Interface input signal	System variable	No. of points	Interface input signal
#1264	1	Register R6442 bit 0	#1280	1	Register R6443 bit 0
#1265		Register R6442 bit 1	#1281		Register R6443 bit 1
#1266		Register R6442 bit 2	#1282		Register R6443 bit 2
#1267		Register R6442 bit 3	#1283		Register R6443 bit 3
#1268		Register R6442 bit 4	#1284		Register R6443 bit 4
#1269		Register R6442 bit 5	#1285		Register R6443 bit 5
#1270		Register R6442 bit 6	#1286		Register R6443 bit 6
#1271		Register R6442 bit 7	#1287		Register R6443 bit 7
#1272		Register R6442 bit 8	#1288		Register R6443 bit 8
#1273		Register R6442 bit 9	#1289		Register R6443 bit 9
#1274		Register R6442 bit 10	#1290		Register R6443 bit 10
#1275		Register R6442 bit 11	#1291		Register R6443 bit 11
#1276		Register R6442 bit 12	#1292		Register R6443 bit 12
#1277		Register R6442 bit 13	#1293		Register R6443 bit 13
#1278		Register R6442 bit 14	#1294		Register R6443 bit 14
#1279		Register R6442 bit 15	#1295		Register R6443 bit 15

Macro interface outputs (#1100 to #1135, #1300 to #1395) : NC -> PLC

Output signals can only be 0 or 1.

All the output Nos. from #1100 to #1131 can be sent at once by substituting a value in variable No. #1132.

Similarly, the output signals #1300 to #1311, #1332 to #1363, and #1364 to #1395 can be sent by substituting values to the variable Nos. #1133 to #1135. (2^0 to 2^{31})

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 among part systems depends on the setting of the base specification parameter "#1230 set02/bit7".

\$1	\$2	\$3	\$4
R6372,3	R6380,1	R6388,9	R6396,7

The following correspondence tables are examples of the 1st part system.

System variable	No. of points	Interface output signal	System variable	No. of points	Interface output signal
#1100	1	Register R6372 bit 0	#1116	1	Register R6373 bit 0
#1101	1	Register R6372 bit 1	#1117	1	Register R6373 bit 1
#1102	1	Register R6372 bit 2	#1118	1	Register R6373 bit 2
#1103	1	Register R6372 bit 3	#1119	1	Register R6373 bit 3
#1104	1	Register R6372 bit 4	#1120	1	Register R6373 bit 4
#1105	1	Register R6372 bit 5	#1121	1	Register R6373 bit 5
#1106	1	Register R6372 bit 6	#1122	1	Register R6373 bit 6
#1107	1	Register R6372 bit 7	#1123	1	Register R6373 bit 7
#1108	1	Register R6372 bit 8	#1124	1	Register R6373 bit 8
#1109	1	Register R6372 bit 9	#1125	1	Register R6373 bit 9
#1110	1	Register R6372 bit 10	#1126	1	Register R6373 bit 10
#1111	1	Register R6372 bit 11	#1127	1	Register R6373 bit 11
#1112	1	Register R6372 bit 12	#1128	1	Register R6373 bit 12
#1113	1	Register R6372 bit 13	#1129	1	Register R6373 bit 13
#1114	1	Register R6372 bit 14	#1130	1	Register R6373 bit 14
#1115	1	Register R6372 bit 15	#1131	1	Register R6373 bit 15

System variable	No. of points	Interface output signal
#1132	32	Register R6372, R6373
#1133	32	Register R6374, R6375
#1134	32	Register R6376, R6377
#1135	32	Register R6378, R6379

System variable	No. of points	Interface output signal	System variable	No. of points	Interface output signal
#1300	1	Register R6374 bit 0	#1316	1	Register R6375 bit 0
#1301	1	Register R6374 bit 1	#1317	1	Register R6375 bit1
#1302	1	Register R6374 bit 2	#1318	1	Register R6375 bit 2
#1303	1	Register R6374 bit 3	#1319	1	Register R6375 bit 3
#1304	1	Register R6374 bit 4	#1320	1	Register R6375 bit 4
#1305	1	Register R6374 bit 5	#1321	1	Register R6375 bit 5
#1306	1	Register R6374 bit 6	#1322	1	Register R6375 bit 6
#1307	1	Register R6374 bit 7	#1323	1	Register R6375 bit 7
#1308	1	Register R6374 bit 8	#1324	1	Register R6375 bit 8
#1309	1	Register R6374 bit 9	#1325	1	Register R6375 bit 9
#1310	1	Register R6374 bit 10	#1326	1	Register R6375 bit 10
#1311	1	Register R6374 bit 11	#1327	1	Register R6375 bit 11
#1312	1	Register R6374 bit 12	#1328	1	Register R6375 bit 12
#1313	1	Register R6374 bit 13	#1329	1	Register R6375 bit 13
#1314	1	Register R6374 bit 14	#1330	1	Register R6375 bit 14
#1315	1	Register R6374 bit 15	#1331	1	Register R6375 bit 15

System variable	No. of points	Interface output signal	System variable	No. of points	Interface output signal
#1332	1	Register R6376 bit 0	#1348	1	Register R6377 bit 0
#1333	1	Register R6376 bit 1	#1349	1	Register R6377 bit 1
#1334	1	Register R6376 bit 2	#1350	1	Register R6377 bit 2
#1335	1	Register R6376 bit 3	#1351	1	Register R6377 bit 3
#1336	1	Register R6376 bit 4	#1352	1	Register R6377 bit 4
#1337	1	Register R6376 bit 5	#1353	1	Register R6377 bit 5
#1338	1	Register R6376 bit 6	#1354	1	Register R6377 bit 6
#1339	1	Register R6376 bit 7	#1355	1	Register R6377 bit 7
#1340	1	Register R6376 bit 8	#1356	1	Register R6377 bit 8
#1341	1	Register R6376 bit 9	#1357	1	Register R6377 bit 9
#1342	1	Register R6376 bit 10	#1358	1	Register R6377 bit 10
#1343	1	Register R6376 bit 11	#1359	1	Register R6377 bit 11
#1344	1	Register R6376 bit 12	#1360	1	Register R6377 bit 12
#1345	1	Register R6376 bit 13	#1361	1	Register R6377 bit 13
#1346	1	Register R6376 bit 14	#1362	1	Register R6377 bit 14
#1347	1	Register R6376 bit 15	#1363	1	Register R6377 bit 15

System variable	No. of points	Interface output signal	System variable	No. of points	Interface output signal
#1364	1	Register R6378 bit 0	#1380	1	Register R6379 bit 0
#1365	1	Register R6378 bit 1	#1381	1	Register R6379 bit 1
#1366	1	Register R6378 bit 2	#1382	1	Register R6379 bit 2
#1367	1	Register R6378 bit 3	#1383	1	Register R6379 bit 3
#1368	1	Register R6378 bit 4	#1384	1	Register R6379 bit 4
#1369	1	Register R6378 bit 5	#1385	1	Register R6379 bit 5
#1370	1	Register R6378 bit 6	#1386	1	Register R6379 bit 6
#1371	1	Register R6378 bit 7	#1387	1	Register R6379 bit 7
#1372	1	Register R6378 bit 8	#1388	1	Register R6379 bit 8
#1373	1	Register R6378 bit 9	#1389	1	Register R6379 bit 9
#1374	1	Register R6378 bit 10	#1390	1	Register R6379 bit 10
#1375	1	Register R6378 bit 11	#1391	1	Register R6379 bit 11
#1376	1	Register R6378 bit 12	#1392	1	Register R6379 bit 12
#1377	1	Register R6378 bit 13	#1393	1	Register R6379 bit 13
#1378	1	Register R6378 bit 14	#1394	1	Register R6379 bit 14
#1379	1	Register R6378 bit 15	#1395	1	Register R6379 bit 15

(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 with resetting.)

(Note 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.

Any number except 0 and <Blank> is treated as 1.

Any value less than 0.00000001 is indefinite.

13.9.5.4 Tool Compensation



Detailed description

Tool compensation data can be read and set using the variable numbers.

	Variable No. range		Details
	#1120 TofVal = 0	#1120 TofVal = 1	
#10001 to #10000 + n	#2001 to #2000 + n	#2701 to #2700 + n	X shape compensation amount
#11001 to #11000 + n	#2701 to #2700 + n	#2001 to #2000 + n	X wear compensation amount
#12001 to #12000 + n			Additional axis shape compensation amount
#13001 to #13000 + n			Additional axis wear compensation amount
#14001 to #14000 + n	#2101 to #2100 + n	#2801 to #2800 + n	Z shape compensation amount
#15001 to #15000 + n	#2801 to #2800 + n	#2101 to #2100 + n	Z wear compensation amount
#16001 to #16000 + n	#2201 to #2200 + n	#2901 to #2900 + n	R shape compensation amount
#17001 to #17000 + n	#2901 to #2900 + n	#2201 to #2200 + n	R wear compensation amount
#18001 to #18000 + n	#2301 to #2300 + n		Nose compensation amount

"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.

The tool compensation data is configured as data with a decimal point in the same way as other variables.

When "# 10001=1000;" is programmed, 1000.000 is set in tool compensation data.

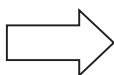
The additional axes' tool compensation can be used for only either the 3rd axis or 4th axis.

Which axis to use is selected by the parameter "#1520 Tchg34".

The variable No. corresponding to the #2000s' shape/wear compensation amount can be changed with the parameter "#1120 TofVal".

Programming example

```
#101=1000;
#10001=#101;
#102=#10001;
```



Common variables

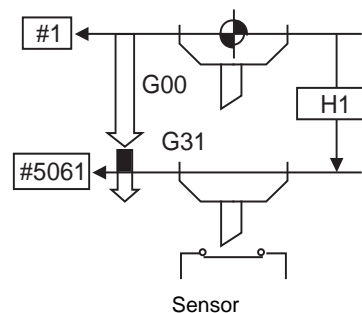
```
#101=1000.0
#102=1000.0
```

Tool compensation data

```
H1=1000.000
```

(Example 1) Calculation and tool offset data setting

```
G28X0 T0101; Reference position return
M06; Tool change (T0101)
#1=#5001; Start point memory
G00 X-200. ; Rapid traverse to safe position
G31 X-50. F100 ; Skip measurement
#10001=#5061-#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.

#5001 is the X axis start point position and #5061 indicates the position at which the skip signal is input while G31 is being executed in the X axis skip coordinates.

(Note 2) For multi-part system

There are cases when each part system has tool data and when the part systems share common tool data. This can be selected with parameter (#1051 MemTol).

Parameter #1051 MemTol

0 : Tool data provided for each part system

1 : Tool data shared by part systems

When the tool data is shared by the part systems, the read amount and substitution value for all part systems' tool compensation variables will be the same (when the same variable No. is commanded).

13.9.5.5 Workpiece Coordinate System Compensation (#5201 - #532n)



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) 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.

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	

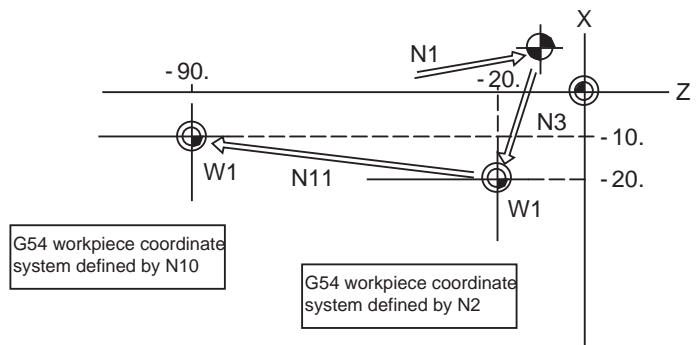
(Example 1)

```

N1 G28 X0 Z0 ;
N2 #5221=-20. #5222=-20. ;
N3 G00 G54 X0 Z0 ;

N10 #5221=-10. #5222=-90. ;
N11 G00 G54 X0 Z0 ;

M02 ;
    
```



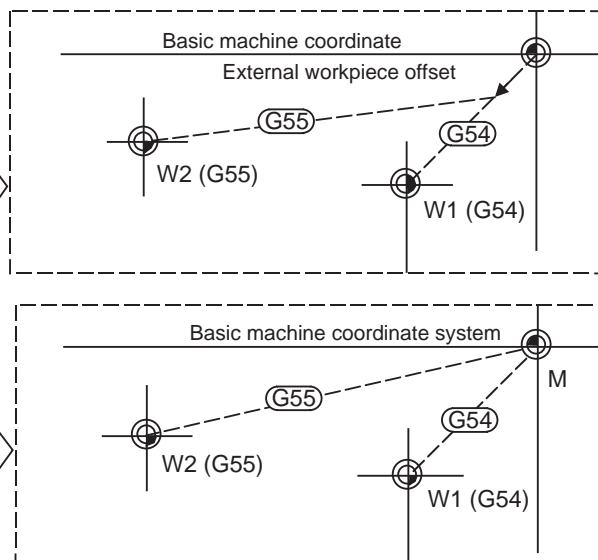
(Example 2)

```

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 system compensation values without changing the position of the workpiece coordinate systems.

13.9.5.6 NC Alarm (#3000)



Detailed description

The NC unit can be forcibly set to the alarm state by using variable No. #3000.

```
#3000= 70 (CALL #PROGRAMMER #TEL #530) ;
```

70	Alarm No.
CALL #PROGRAMMER #TEL #530	Alarm message

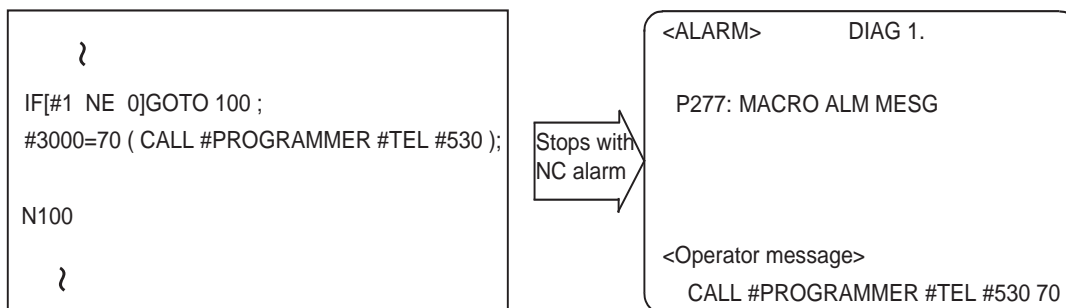
Any alarm number from 1 to 9999 can be specified.

The alarm message must be written in 31 or less characters.

NC alarm 3 signal (program error) is output.

The "P277: MACRO ALM MESSG" appears in the <ALARM> column on "DIAG 1." screen and the alarm message " (CALL #PROGRAMMER #TEL #530)" and the alarm No. (70) will appear in the <Operator message>.

Example of program (alarm when #1 = 0)



- (Note 1) If zero or any number greater than 9999 is specified for the alarm No., the number will be invalid and it will not display. However, the operation will be in the alarm status, and the specified alarm message will appear.
- (Note 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.
- (Note 3) When 32 or more characters are specified for the alarm message, characters after the 32nd character will not display.
- (Note 4) A space included in an alarm message character string is ignored, and characters after the 32nd character will not display. To split the character string insert a character such as "." (period).

13.9.5.7 Integrating Time (#3001, #3002)

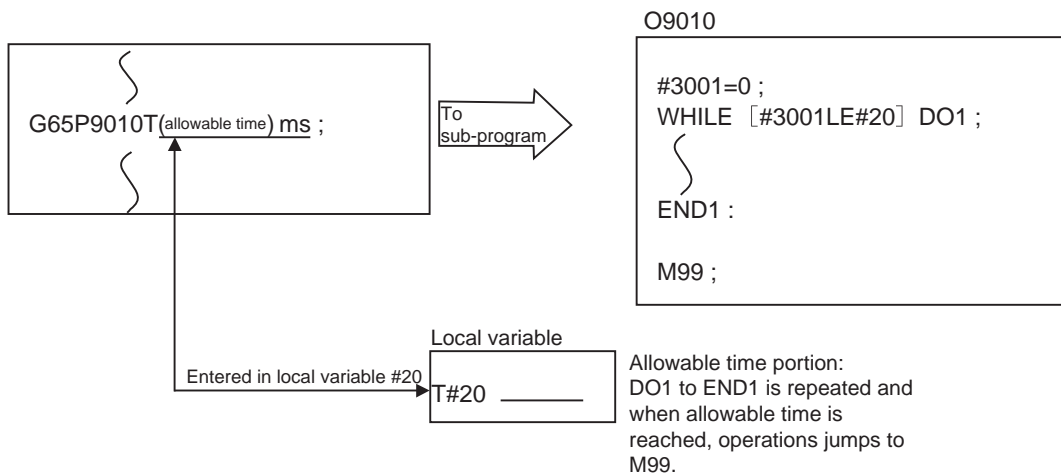


Detailed description

The integrating time during the power is turned ON or the automatic start is running, can be read or values can be substituted by using variable Nos. #3001 and #3002.

Type	Variable No.	Unit	Contents when power is switched on	Initialization of contents	Count condition
Power ON	3001	1ms	Same as when power is switched off	Substitute values to variables	At all times while power is ON
Automatic start	3002				In-automatic start

The integrating time returns to zero in about 2.44×10^{11} ms (approximately 7.7 years).



13.9.5.8 Suppression of Single Block Stop and Miscellaneous Function Finish Signal Waiting (#3003)



Detailed description

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	Single block stop	Miscellaneous function finish signal
0	Not suppressed	Wait
1	Suppressed	Wait
2	Not suppressed	Not wait
3	Suppressed	Not wait

(Note 1) Variable No. #3003 is set to zero by NC reset.

13.9.5.9 Feed Hold, Feedrate Override, G09 Valid/Invalid (#3004)



Detailed description

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.

Contents (value)	#3004		
	Bit 0	Bit 1	Bit 2
	Feed hold	Feedrate override	G09 check
0	Valid	Valid	Valid
1	Invalid	Valid	Valid
2	Valid	Invalid	Valid
3	Invalid	Invalid	Valid
4	Valid	Valid	Invalid
5	Invalid	Valid	Invalid
6	Valid	Invalid	Invalid
7	Invalid	Invalid	Invalid

(Note 1) Variable No. #3004 is set to zero by NC reset.

(Note 2) The functions are valid when the above bits are 0, and invalid when they are 1.

(Note 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.

13.9.5.10 Message Display and Stop (#3006)



Detailed description

By using variable No. #3006, the operation stops after the previous block is executed and, if message display data is commanded, the corresponding message and the stop No. will be indicated on the operator message area.

#3006 = 1(TAKE FIVE);	
1 to 9999	Stop No. (When Nos. other than 1 - 9999 are set, the command will be invalidated.)
TAKE FIVE	Message (Nothing will be displayed if no message is designated.)

The message should be written in 31 or less characters and should be enclosed by round parentheses.

13.9.5.11 Mirror Image (#3007)



Detailed description

By reading variable No. #3007, it is possible to ascertain the status of mirror image of the each axis at the point.

Each axis corresponds to a bit of #3007.

When the bits are 0, the mirror image function is invalid.

When the bits are 1, the mirror image function is valid.

#3007

Bit	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
nth axis													4	3	2	1

13.9.5.12 G Command Modals (#4001-#4021, #4201-#4221)



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	Barrier check	G22 : 22, G23 : 23
#4005	#4205	Feed designation	G94 : 94, G95 : 95
#4006	#4206	Inch/metric	G20 : 20, G21 : 21
#4007	#4207	Tool nose R compensation	G40 : 40, G41 : 41, G42 : 42, G46 : G46
#4008	#4208	No variable No.	
#4009	#4209	Fixed cycle	G80 : 80, G70-G79 : 70-79, G83-G85 : 83-85, G83.2 : 83.2, G87-G89 : 87-89
#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		
#4016	#4216	No variable No.	
#4017	#4217	Constant surface speed control	G96 : 96, G97 : 97
#4018	#4218	Balance cut	G14 : 14, G15 : 15
#4019	#4219		
#4020	#4220		
#4021	#4221		

(Example)

```
G28 X0 Z0 ;
G00 X150. Z200 ;
G65 P300 G02 W-30. K-15. F1000 ;
M02 ;
O300
#1 = #4001 ; = -> Group 01 G modal (pre-read) #1 = 2.0
# = #4201 ; = -> Group 01 G modal (now being executed) #2 = 0.0
G#1 W#24 ;
M99 ;
%
```

13.9.5.13 Other Modals (#4101 - #4120, #4301 - #4320)



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	Execution		Pre-read	Execution	
#4101	#4301		#4111	#4311	
#4102	#4302	2nd miscellaneous function B	#4112	#4312	
#4103	#4303		#4113	#4313	Miscellaneous function M
#4104	#4304		#4114	#4314	Sequence number N
#4105	#4305		#4115	#4315	Program number O (Note1)
#4106	#4306		#4116	#4316	
#4107	#4307		#4117	#4317	
#4108	#4308		#4118	#4318	
#4109	#4309	Feedrate F	#4119	#4319	Spindle function S
#4110	#4310		#4120	#4320	Tool function T
			#4130	#4330	Extended workpiece coordinate system No. P

(Note 1) The programs are registered as files. When the program No. (file name) is read with #4115, #4315, the character string will be converted to a value.

(Example 1) The file name "123" is the character string 0x31, 0x32, 0x33, so the value will be $(0x31-0x30)*100 + (0x32-0x30)*10 + (0x33-0x30) = 123.0$. Note that if the file name contains characters other than numbers, it will be "blank".

(Example 2) If the file name is "123ABC", it contains characters other than numbers; so, the result will be "blank".

13.9.5.14 Position Information (#5001 - #5140 + n)

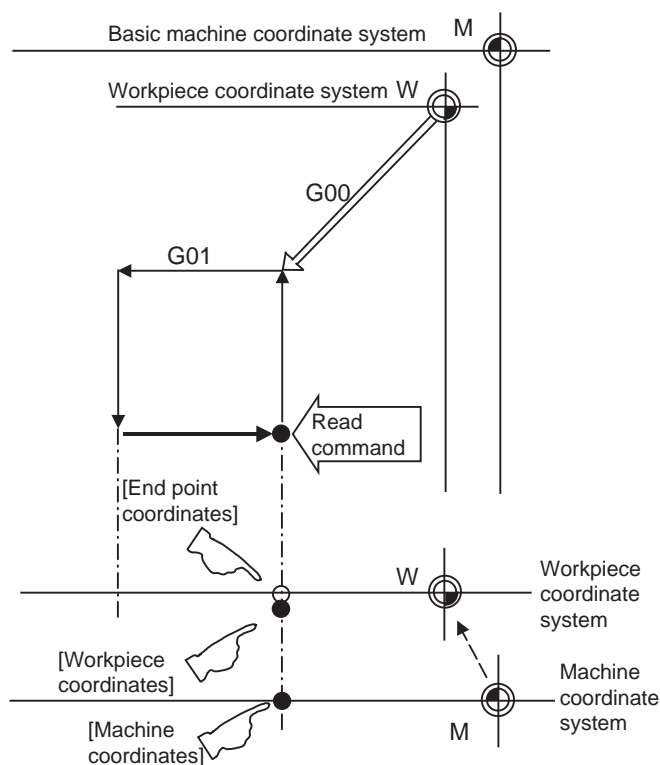


Detailed description

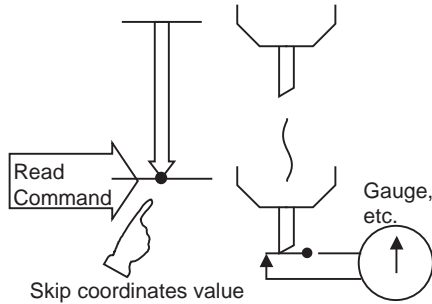
Using variable Nos. #5001 to #5104, it is possible to read the end point coordinates, machine coordinates, workpiece coordinates, skip coordinates, and servo deviation amounts in the last block.

Position information	Axis No.						Reading during movement
	1	2	3	4	...	n	
End point coordinate of the last block	#5001	#5002	#5003	#5004	...	#5000+n	Yes
Machine coordinate	#5021	#5022	#5023	#5024	...	#5020+n	No
Workpiece coordinate	#5041	#5042	#5043	#5044	...	#5040+n	No
Skip coordinate	#5061	#5062	#5063	#5064	...	#5060+n	Yes
Servo deviation amount	#5101	#5102	#5103	#5104	...	#5100+n	Yes
Macro interruption stop Start point coordinates	#5121	#5122	#5123	#5124	...	#5120+n	Yes
Macro interruption stop End point coordinates	#5141	#5142	#5143	#5144	...	#5140+n	Yes

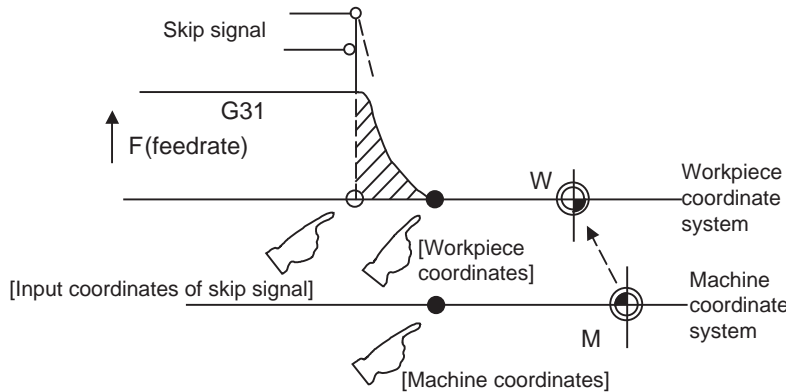
(Note) The number of axes which can be controlled differs according to the specifications.
The last digit of the variable No. corresponds to the control axis No.



- (1) The positions of the end point coordinates and skip coordinates are positions in 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.

The skip signal input coordinates value is the position in the workpiece coordinate system. The coordinate value in variable Nos. #5061 to #5060+n memorize the moments when the skip input signal during movement was input and so they can be read at any subsequent time.

For further details, refer to the section on "Skip Function".

13.9.5.15 External Workpiece Coordinate System Compensation (#2501, #2601)



Detailed description

The workpiece coordinate system compensation amount can be read using variables #2501 and #2601. By substituting a value in these variable Nos., the workpiece coordinate system compensation amount can be changed.

Axis No.	External workpiece coordinate compensation
1	#2501
2	#2601

13.9.5.16 Number of Workpiece Machining Times (#3901, #3902)



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.

Type	Variable No.	Data setting range
Number of workpiece machining times	#3901	0 to 999999
Maximum workpiece value	#3902	

(Note) The number of workpiece machining times must be a positive value.

13.9.5.17 Tool Life Management (#60000 - #63016)



Detailed description

Definition of variable Nos.

- (1) Designation of group No.

#60000

The tool life management data group No. to be read with #60001 to #63016 is designated by substituting a value in this variable No. If a group No. is not designated, the data of the group registered first is read. This is valid until reset.

- (2) Tool life management system variable No. (Read)

#60001 to #63016

| a | b | c | d | e |

| a | : "6" Fix (Tool life management)

| b | c | : Details of data classification

Data class	Details	Remarks
00	For control	Refer by data types
05	Group No.	Refer by registration No.
10	Tool No.	Refer by registration No.
15	Method	Refer by registration No.
20	Status	Refer by registration No.
25	Life time/No. of uses	Refer by registration No.
30	Usage time/No. of uses	Refer by registration No.

The group No., method, and life data are common for the group.

| d | e | : Registration No. or data type

Registration No.

1 to 16

Data type

Type	Details
1	Number of registered tools
2	Life current value
3	Tool selection No.
4	Number of remaining registered tools
5	Execution signal
6	Cutting time cumulative value (min)
7	Life end signal
8	Life prediction signal

List of variables

Variable No.	Item	Type	Details	Data range
60001	Number of registered tools	Common to system	Total number of tools registered in each group.	0 to 80
60002	Life current value	For each group (Designate Group No. #60000)	Usage time/No. of uses of tool being used. Usage data of tool being used (If tool uses several compensation Nos., the total of the usage data for each compensation No.).	0 to 999999 min 0 to 999999 times
60003	Tool selection No.		Registration No. of tool being used. Designated group's selected tool registration No. (If a tool is not selected, the first tool of ST:1, or if ST:1 is not used, the first tool of ST:0. When all tools have reached their lives, the last tool).	0 to 16
60004	Number of remaining registered tools		Total number of "usable" tools in the group. Number of tools registered in designated group, whose ST is 0: Not used.	0 to 16
60005	Execution signal		"1" when this group is used in program being executed. "1" when tool in designated group is selected.	0/1
60006	Cutting time cumulative value (min)		Indicates the time that this group is used in the program being executed.	
60007	Life end signal		"1" when lives of all tools in this group have expired. "1" when all registered tools in the designated group reach lives.	0/1
60008	Life prediction signal		"1" when selecting a new tool with the next command in this group. "1" when there are no tools in use (ST: 1) while there is an unused tool (ST: 0) in the specified group.	0/1
60500 +***	Group No.		Each group/ registration No. (Group No. #60000/ registration No. *** is designated.) Note that the group No./ method and life are common for the groups.	This group's No.
61000 +***	Tool No.	Tool No. and compensation No. Tool No. + compensation No. (When tool No. = 22 and compensation No. = 01, 2201=899H)		0 to 9999
61500 +***	Method	Whether to manage this group's life as time or number of uses. 0 : Time, 1: Number of uses		0/1
62000 +***	Status	Tool usage state 0 : Tool not used 1 : Tool in use 2 : Normal life tool 3 : Tool skip tool		0 to 3
62500 +***	Life time/No. of uses	This group's tool life value		0 to 999999 min 0 to 999999 times
63000 +***	Usage time/No. of uses			0 to 999999 min 0 to 999999 times



Program example

(1) Normal commands

```
#101 = #60001 ; ..... Reads the number of registered tools.
#102 = #60002 ; ..... Reads the life current value.
#103 = #60003 ; ..... Reads the tool selection No.
#60000 = 10 ; ..... Designates the group No. of the life data to be read.
                        Designated program No. is valid until reset.
#104 = #60004 ; ..... Reads the remaining number of registered tools in group 10.
#105 = #60005 ; ..... Reads the signal being executed in group 10.
#111 = #61001 ; ..... Reads the group 10, #1 tool No.
#112 = #62001 ; ..... Reads the group 10, #1 status.
#113 = #61002 ; ..... Reads the group 10, #2 tool No.
%
```

(2) When group No. is not designated.

```
#104 = #60004 ; ..... Reads the remaining number of registered tools in the group registered first.
#111 = #61001 ; ..... Reads the #1 tool No. in the group registered first.
%
```

(3) When non-registered group No. is designated. (Group 9999 does not exist.)

```
#60000 = 9999 ; ..... Designates the group No.
#104 = #60004 ; ..... #104 = -1
```

(4) When registration No. not used is designated. (Group 10 has 15 tools)

```
#60000 = 10 ; ..... Designates the group No.
#111 = #61016 ; ..... #111 = -1
```

(5) When registration No. out of the specifications is designated.

```
#60000 = 10 ;
#111 = #61017 ; ..... Program error (P241)
```

- (6) When tool life management data is registered with G10 command after group No. is designated.

```
#60000 = 10 ; ..... Designates the group No.
G10 L3 ; ..... Starts the life management data registration.
                  The group 10 life data is registered from G10 to G11.
P10 LLn NNn ; ..... 10 is the group No., Ln is the life per tool, Nn is the method.
TTn ; ..... Tn is the tool No.
:
G11 ; ..... Registers the life data with the G10 command.
#111 = #61001 ; ..... Reads the group 10, #1 tool No.
G10 L3 ; ..... Starts the life management data registration.
                  The life data other than group 10 is registered from G10 to G11.
P1 LLn NNn ; ..... 1 is the group No., Ln is the life per tool, Nn is the method.
TTn ; ..... Tn is the tool No.
:
G11 ; ..... Registers the life data with the G10 command.
                  (The registered data is deleted.)
#111 = #61001 ; ..... Group 10 does not exist. #111 = -1.
```



Precautions

- (1) If the tool life management system variable is commanded without designating a group No., the data of the group registered at the head of the registered data will be read.
- (2) If a non-registered group No. is designated and the tool life management system variable is commanded, "-1" will be read as the data.
- (3) If an unused registration No. tool life management system variable is commanded, "-1" will be read as the data.
- (4) Once commanded, the group No. is valid until NC reset.
- (5) When the tool life management I specifications are provided, the tool life management system variable specifications are not provided. A program error (P241) will occur if commanded.

13.9.5.18 Reading The Parameters (#100000-#100002, #100010)



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 in 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 (system in running program) 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 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 No./spindle No. designation

The axis/spindle No. of the parameter to be read in is designated by substituting an index value in 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
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 per part system [#1002 axisno (Number of axes)]

- #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]. (Program error (P39) occurs.)
- #100001 = 10; Designates [PLC axis].
- #110 = #100010; Reads the number of PLC axes.

(2) To read the axis parameter [#2037 G53ofs (#1 reference point)]

[Conditions]	[1st part system]		[2nd part system]	
	<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 [#1 reference point] for the 2nd 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 [#1 reference point] for the 2nd 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=300.000)

(3) To read each parameter for each part system, axis and spindle

#100002 = 1;	Designates [1st spindle].
#100000 = 3001;	Designates [#3001].
#101 = #100010;	Reads the [#3001 slimt1 (Limit rotation speed 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 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.

13.9.5.19 Reading PLC data (#100100-#100103,#100110)



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) 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 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 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.	Device designation value	Device	Unit	Device No.
0	M	Bit	M0 to M10239	10	F	Bit	F0 to F1023
1	D	Word	D0 to D2047	13	L	Bit	L0 to L511
2	C	Bit	C0 to C255	18	V	Bit	V0 to V255
4	X *	Bit	X0 to X1FFF	19	ST	Bit	ST0 to ST63
5	Y *	Bit	Y0 to Y1FFF	20	SD	Word	SD0 to SD127
6	R	Word	R0 to R32767	21	SB *	Bit	SB0 to SB1FF
7	T	Bit	T0 to T703	22	SW *	Word	SW0 to SW1FF
9	SM	Bit	SM0 to SM127	23	B *	Bit	B0 to B1FFF
				24	W *	Word	W0 to W1FFF

The unit indicates the amount of data per device No. "Word" is 16 bits, and "Bit" is one bit.

* is a device that expresses the device No. as a hexadecimal.

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 byte	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 byte	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 byte	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

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 bytes].
#101 = #100110;	Reads M0 to M7 (8 bits). (If M7 to M0 is 0001 0010, this will be #102 = 18 (0x12).)
#100102 = 102;	Designates [Signed two bytes].
#102 = #100110;	Reads M0 to M15 (16 bits). (If M15 to M0 is 1111 1110 1101 1100, this will be #102 = -292 (0xFEDC).)
#100102 = 4;	Designates [4 bytes].
#104 = #100110;	Reads M0 to M31 (32 bits). (If M31 to M0 is 0001 0010 0011 0100 0101 0110 0111 1000, #104 = 305419896 (0x12345678).)

(2) To read a word device

#100100 = 1;	Designates [D device].
#100101 = 0;	Designates [Device No. 0].
#100102 = 0;	Designates [Bit].
#100103 = 1;	Designates [Bit 1].
#100 = #100110;	Reads the D0 bit 1. (If D0 = 0x0102, #101 =1.)
#100102 = 1;	Designates [1 bytes].
#101 = #100110;	Reads the low-order byte of D0. (If D0 = 0x0102, #101 =2.)
#100102 = 2;	Designates [2 bytes].
#102 = #100110;	Reads D0. (If D0 = 0x0102, #102 =258.)
#100102 = 104;	Designates [Signed four bytes].
#104 = #100110;	Reads D0 and D1. (If D0 = 0xFFFFE and D1 = 0xFFFF, #104 =-2.)



Precautions

- (1) As the PLC data is read asynchronously from the ladder execution, the data is not necessarily the one which was gained when the program was executed. Be careful when reading devices which are changing.
- (2) If reading of a nonexistent device is attempted by designating the device No. and number of bytes, the 0 value will be read in only for the nonexistent section.

13.9.5.20 Time Reading Variables (#3001, #3002, #3011, #3012)



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 cumulative time during system variable #3002 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. Command range for year/month/date setting Year (YYYY) : 2000 to 2099 Month (MM) : 1 to 12 Date (DD) : 1 to maximum number of days in 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/DD. Command range for hour/minute/second setting Hour (HH) : 0 to 23 (24-hour system) Minute (MM) : 0 to 59 Second (SS) : 0 to 59

- (3) The cumulative time returns to 0 at approx. 2.44×10^{11} milliseconds (approx. 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 => #3001= 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.

13.9.5.21 R Device Access Variables (#50000 - #50749, #51000 - #51749, #52000 - #52749)



Function and purpose

By using variable Nos. #50000 to #50749, #51000 to #51749, #52000 to #52749, it is possible to read data (R8300 to R9799, R18300 to R19799, R28300 to R29799) and substitute value in the R device user backup area.

Variable No.	R device	
#50000	R8300,R8301	User backup area (1500 points)
#50001	R8302,R8303	
:		
#50749	R9798,R9799	

Variable No.	R device	
#51000	R18300,R18301	User backup area (1500 points)
#51001	R18302,R18303	
:		
#51749	R19798,R19799	

Variable No.	R device	
#52000	R28300,R28301	User backup area (1500 points)
#52001	R28302,R28303	
:		
#52749	R29798,R29799	



Detailed description

These variables read and write the two words of R device.

Data range of these variables is -2147483648 to 2147483647.

Depending on the setting of the PLC bit selection parameter "#6455/ bit0 to 2", 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).

#1041 I_inch	#1003 iunit			
	B	C	D	E
0 : Metric	Three digits after the decimal point	Four digits after the decimal point	Five digits after the decimal point	Six digits after the decimal point
1 : Inch	Four digits after the decimal point	Five digits after the decimal point	Six digits after the decimal point	Seven digits after the decimal point

These variables are retained even when the power is off.

These are common among part systems.

Access from a machining program to R device

[Reading variables]

When the variable #50000 is used in a machining program as shown below, the data set in device R8300 and R8301 will be referred.

G0 X#50000 ; 	Device	Value	#50000
	R8301	0x0001	0x1e240 (hexadecimal) = 123456 (decimal)
	R8300	0xe240	

- (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),
 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 #50000 in a machining program as shown below, data will be set in the device R8302 and R8303.

#50001 = 123 ;

- (1) When decimal point invalid is selected
 Regardless of the setting of the parameter "#1003 iunit"(inupt setting unit) and "#1041 I_inch"(initial inch), substituted value will be set in the R device.

#50001	Device	Value
123 (decimal) = 0x7b (hexadecimal)	R8303	0x0000
	R8302	0x007b

When a value with a decimal point is substituted to a variable like "#50001 = 123.456 ;", the numbers after the decimal point will be truncated and "123" will be set.

(2) When decimal point valid is selected

According to the settings of the parameter "#1003 iunit"(inupt setting unit) and "#1041 I_inch"(initial inch), values which are shifted for the number of decimals will be set in the R device, as shown below.

#1041 I_inch		0 : Metric			
#1003 iunit		B	C	D	E
#50001		123000 (decimal) = 0x1e078 (hexadecimal)	1230000 (decimal) = 0x12c4b0 (hexadecimal)	12300000 (decimal) = 0xbbaee0 (hexadecimal)	123000000 (decimal) = 0x754d4c0 (hexadecimal)
Device	R8303	0x0001	0x0012	0x00bb	0x0754
	R8302	0xe078	0xc4b0	0xae0	0xd4c0

#1041 I_inch		1 : Inch			
#1003 iunit		B	C	D	E
#50001		1230000 (decimal) = 0x12c4b0 (hexadecimal)	12300000 (decimal) = 0xbbaee0 (hexadecimal)	123000000 (decimal) = 0x754d4c0 (hexadecimal)	1230000000 (decimal) = 0x49504f80 (hexadecimal)
Device	R8303	0x0012	0x00bb	0x0754	0x4950
	R8302	0xc4b0	0xae0	0xd4c0	0x4f80

When a value with a decimal point is substituted to a variable like "#50001 = 123.456 ;", the value will directly be set.

#1041 I_inch		0 : Metric			
#1003 iunit		B	C	D	E
#50001		123456 (decimal) = 0x1e240 (hexadecimal)	1234560 (decimal) = 0x12d680 (hexadecimal)	12345600 (decimal) = 0xbc6100 (hexadecimal)	123456000 (decimal) = 0x75bca00 (hexadecimal)
Device	R8303	0x0001	0x0012	0x00bc	0x075b
	R8302	0xe240	0xd680	0x6100	0xca00

#1041 I_inch		1 : Inch			
#1003 iunit		B	C	D	E
#50001		1234560 (decimal) = 0x12d680 (hexadecimal)	12345600 (decimal) = 0xbc6100 (hexadecimal)	123456000 (decimal) = 0x75bca00 (hexadecimal)	1234560000 (decimal) = 0x4995e400 (hexadecimal)
Device	R8303	0x0012	0x00bc	0x075b	0x4998
	R8302	0xd680	0x6100	0xca00	0xe400

If the number of decimals of the substituted data exceeds the number of significant figures, the value will be rounded off to the number of significant figures and will be set.

When "#50001 = 123.4567899 ;".

#1041 I_inch		0 : Metric			
#1003 iunit		B	C	D	E
#50001		123457 (decimal) = 0x1e241 (hexadecimal)	1234568 (decimal) = 0x12d688 (hexadecimal)	12345679 (decimal) = 0xbc614f (hexadecimal)	123456790 (decimal) = 0x75bcd16 (hexadecimal)
Device	R8303	0x0001	0x0012	0x00bc	0x075b
	R8302	0xe241	0xd688	0x614f	0xcd16

#1041 I_inch		1 : Inch			
#1003 iunit		B	C	D	E
#50001		1234568 (decimal) = 0x12d688 (hexadecimal)	12345679 (decimal) = 0xbc614f (hexadecimal)	123456790 (decimal) = 0x75bcd16 (hexadecimal)	1234567899 (decimal) = 0x499602db (hexadecimal)
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"(inupt setting unit) and "#1041 I_inch"(initial inch), R device value of #50003 whose condition is true, will be "1".

#50003	Device	Value
1 (decimal) = 0x01 (hexadecimal)	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 (decimal) = 0x3e8 (hexadecimal)	10000 (decimal) = 0x2710 (hexadecimal)	100000 (decimal) = 0x186a0 (hexadecimal)	1000000 (decimal) = 0xf4240 (hexadecimal)
Device	R8307	0x0000	0x0000	0x0001	0x000f
	R8306	0x03e8	0x2710	0x86a0	0x4240

#1041 I_inch		1 : Inch			
#1003 iunit		B	C	D	E
#50003		10000 (decimal) = 0x2710 (hexadecimal)	100000 (decimal) = 0x186a0 (hexadecimal)	1000000 (decimal) = 0xf4240 (hexadecimal)	10000000 (decimal) = 0x989680 (hexadecimal)
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 (decimal) = 0xfffff85 (hexadecimal)	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 (decimal) = 0xffe1dc0 (hexadecimal)	-1234560 (decimal) = 0xffed2980 (hexadecimal)	-12345600 (decimal) = 0xff439f00 (hexadecimal)	-123456000 (decimal) = 0xf8a43600 (hexadecimal)
Device	R8309	0xffe	0xffed	0xff43
	R8308	0x1dc0	0x2980	0x9f00

#1041 I_inch	1 : Inch			
#1003 iunit	B	C	D	E
#50004	-1234560 (decimal) = 0xffed2980 (hexadecimal)	-12345600 (decimal) = 0xff439f00 (hexadecimal)	-123456000 (decimal) = 0xf8a43600 (hexadecimal)	-1234560000 (decimal) = 0xb66a1c00 (hexadecimal)
Device	R8309	0xffed	0xff43	0xf8a4
	R8308	0x2980	0x9f00	0x3600

[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"(inupt 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 #0<Blank> and #0 #0<Blank> with a conditional expression (EQ), it will not be formed.
- (3) A program error (P35) will occur when a value exceeding an effective range is substituted into these variables.
- (4) When these values are used as decimal point invalid, the settings of "#1078 Decpt2" (Decimal point type 2) 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.
The reading of these variables (reference to the R device value) during a graphic check will be as follows depending on the model.
M700VW : "0" is always read.
M700VS/M70V : The currently set R device value is read.

13.9.6 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 $\tan \theta$ uses $\sin \theta / \cos \theta$.
	#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]	Discard fractions less than 1
	#i = FUP [#k]	Add for fractions less than 1
	#i = LN [#k]	Natural logarithm
#i = EXP [#k]	Exponent with e (=2.718) as bottom	

(Note 1) A value without a decimal point is basically treated as a value with a decimal point at the end (1 = 1.000).

(Note 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

(Note 3) The <formula> after a function must be enclosed in the square parentheses [].

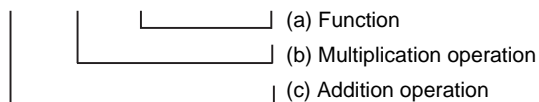


Detailed description

Sequence of operations

- (1) The sequence of the operations (a) to (c) is performed in the following order; the function, the multiplication operation and the addition operation.

#101=#111+#112*SIN [#113]



- (2) The part to be given priority in the operation sequence should be enclosed in square parentheses []. Up to 5 pairs of such parentheses, including those for the functions, may be used.

#101=SQRT [[[[#111-#112] *SIN [#113] +#114] *#115]] ;

First pair of brackets
 Second pair of brackets
 Third pair of brackets

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 compensation amount				
(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 with 3 over #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) Sin 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	



Precautions

(1) Notes on logical relation

EQ, NE, GT, LT, GE and LE conduct the same calculation as addition and subtraction. Pay careful attention to errors. For example, to determine whether or not #10 and #20 are equal in the following example, it is not always possible to judge correctly because of the error.

```
IF [#10 EQ #20]
```

Therefore when the difference between #10 and #20 falls within the designated error range, both values should be considered equal.

```
IF [ABS [#10 - #20] LT 200000]
```


13.9.7 Control Commands



Function and purpose

The flow of programs can be controlled by IF-GOTO- and WHILE-DO-END.



Detailed description

Branching

IF [conditional expression] GOTO n; (n = sequence number in the program)

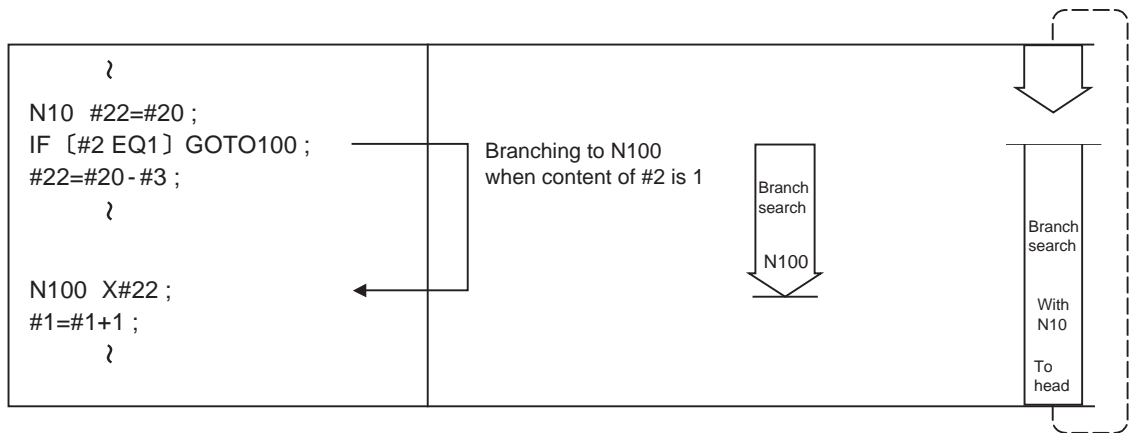
When the condition is satisfied, control branches to "n" and when it is not satisfied, the next block is executed. IF [conditional expression] can be omitted and, when it is, control branches to "n" unconditionally. The following types of [conditional expressions] are available.

#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.

(Note 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.

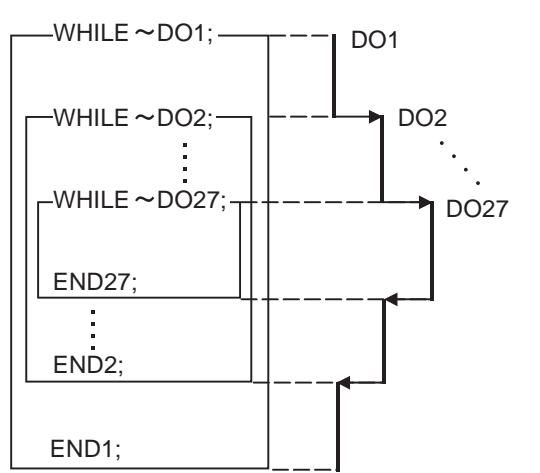
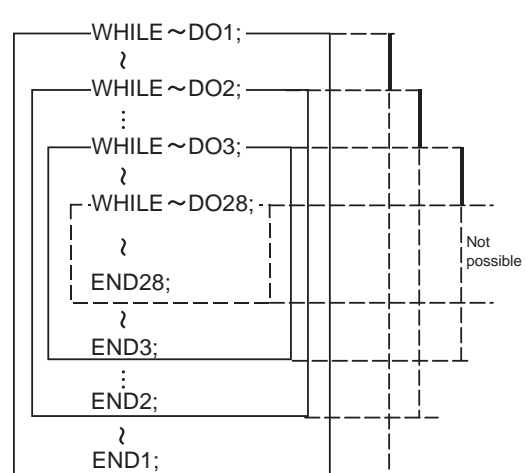
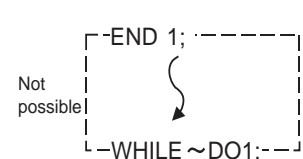
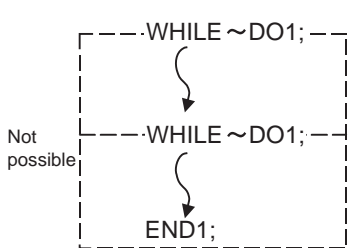
Repetitions

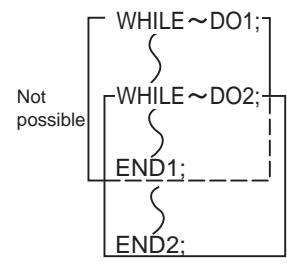
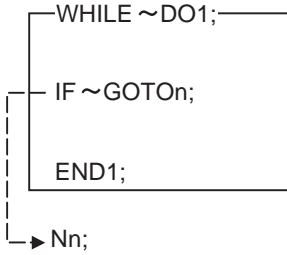
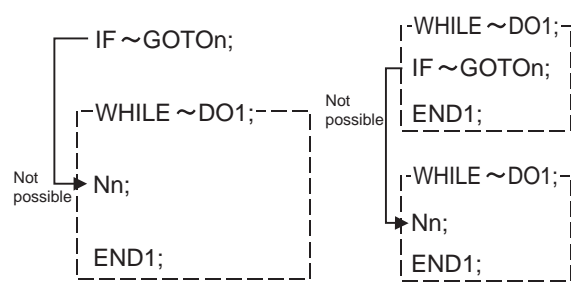
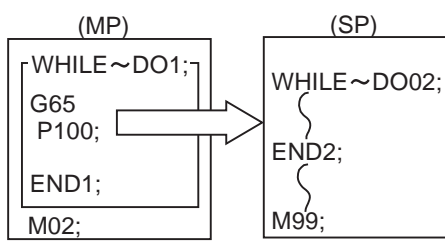
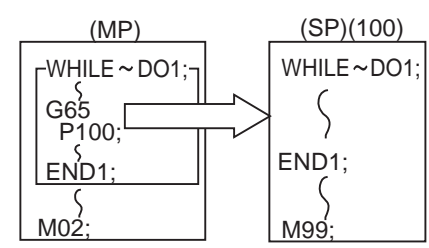
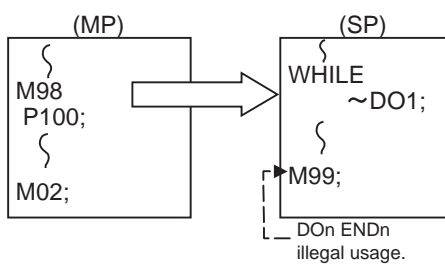
```

WHILE [conditional expression] DOm ; (m =1, 2, 3 ..... 127)
:
END m ;
    
```

While the conditional expression is established, the blocks from the following block to ENDm are repeatedly executed; when it is not established, execution moves to the block following ENDm. DOm may come before WHILE.

"WHILE [conditional expression] DOm" and "ENDm" must be used as a pair. If "WHILE [conditional expression]" is omitted, these blocks will be repeatedly ad infinitum. The repeating identification Nos. range from 1 to 127. (DO1, DO2, DO3, DO127) Up to 27 nesting levels can be used.

<p>(1) Same identification No. can be used any number of times.</p> <pre> WHILE ~DO1; END1; WHILE ~DO1; END1; </pre>	<p>(2) Any number may be used as the WHILE-DOm identification No.</p> <pre> WHILE ~DO1; } END1; } -WHILE ~DO3; } END3; } -WHILE ~DO2; } END2; } -WHILE ~DO1; } END1; </pre>
<p>(3) Up to 27 nesting levels can be used for WHILE-DOm. "m" is any number from 1 to 127 for the nesting depth.</p> 	<p>(4) The number of WHILE-DOm nesting levels cannot exceed 27.</p> 
<p>(Note) For nesting, "m" which has been used once cannot be used.</p> <p>(5) WHILE - DOm must be designated first and ENDm last.</p> 	<p>(6) WHILE - DOm and ENDm must correspond on a 1:1 (pairing) basis in a same program.</p> 

<p>(7) Two WHILE - DOm's must not overlap.</p> 	<p>(8) Branching externally out of the WHILE - DOm range, is possible.</p> 
<p>(9) No branching into WHILE - DOm, is possible.</p> 	<p>(10) Subprograms can be called by M98, G65 or G66 between WHILE - DOm's.</p> 
<p>(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.</p> 	<p>(12) A program error will occur in M99 if WHILE and END are not paired in the subprogram (including macro subprogram).</p> 

(MP) Main program

(SP) Subprogram

(Note) Even if a fixed cycle containing WHILE is called, the nesting level will be counted up.

13.9.8 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 interface or memory card.



Command format

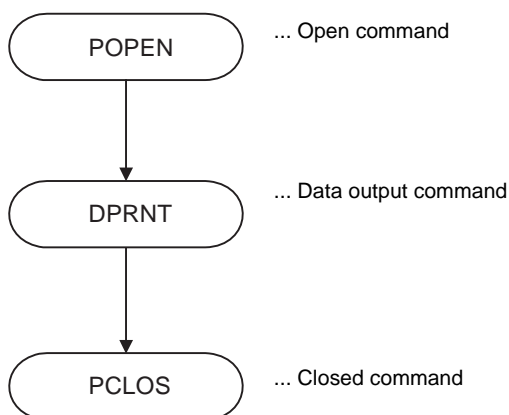
POPEN ... Open command

PCLOS ... Closed command

DPRNT ... Data output command

POPEN	For preparing the data outputs
PCLOS	For terminating the data outputs
DPRNT	For character output and digit-by-digit variable numerical output

Command sequence



Detailed description

Open command : POPEN

- (1) The command is issued before the series of data output commands.
- (2) The DC2 control code and % code are output from the NC system to the external output device.
- (3) Once POPEN; is issued, it will remain valid until PCLOS; is issued.

Close command : PCLOS

- (1) This command is issued when all the data outputs are completed.
- (2) The DC4 control code and % code are output from the NC system to the external output device.
- (3) This command is used together with the open command and it should not be issued unless the open mode has been established.
- (4) Issue the close command at the end of the program even when the operation is suspended by resetting or some other operation during data output.

Data output command : DPRNT

DPRNT [I1#v1 [d1 c1] I2#v2 [d2 c2]] ;

I1	Character string	
v1	Variable No.	
d1	Significant digits above decimal point	c + d <= 8
c1	Significant digits after decimal point	

- (1) The character output and decimal output of the variable values are done with ISO codes.
- (2) The commanded character string is output as it is by the ISO code.
Alphanumerics (A to Z, 0 to 9) and special characters (+, -, *, /) can be used.
Note that asterisk (*) is output as a space code.
- (3) The required significant digits above and below the decimal point of the variable values are each commanded within square parentheses. As a result, the commanded number of digits of variable values are output in ISO code in decimal notation from the high-order digits including the decimal point. In this case, trailing zeroes are not omitted.
- (4) Leading zeroes are omitted.
The omitted leading zero can be replaced by a space by the setting of a parameter.
This can justify the last digit of the data output to the printer.
- (5) Linefeed (LF) code will be output to the end of the output data.
And by setting the parameter "#9112 to #9512 DEV0 - 4 CR OUTPUT" to "1", (CR) code will be written in just before EOB (LF) code.
- (Note) A data output command can be issued even in two-part system mode. In this case, however, note that the output channel is shared by both part systems. So, be careful not to execute data output by both part systems simultaneously.

Data output port

- (1) The output port can be selected by the parameter "#9007 MACRO PRINT PORT".
- (2) When the port is a memory card, the file name of the port can be designated by the parameter "#9054 MACRO PRINT FILE".
- (3) When the port is a memory card, the port directory is fixed to root directory.



Example of use

<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	= DPRINT_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[]; (Note)
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;
%
```

(Note) 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 = DPRINT_OUT)>

Blank				Linefeed code							
(CR)	(LF)									(CR)	(LF)
V	A	L	-	C	H	E	C	K		(CR)	(LF)
1	2	3	4	5	6	7	8	9	0	(CR)	(LF)
			1	2	.	3	4	0	0	(CR)	(LF)
				0	.	0	0	0	0	(CR)	(LF)
-	2	3	4	5	6	7	8	9		(CR)	(LF)
							-	0		(CR)	(LF)
-	.	0	0	0	0	0	0	0	0	(CR)	(LF)
-	.	1	2	3	4	5	6	7	9	(CR)	(LF)

Values above the number of significant figures are rounded down

Values below the number of significant figures are rounded off



Precautions

- (1) An external output command during restart-searching is ignored.
After using the restart-search type 2 to restart-search between a POPEN command and a PCLOS command, execute a POPEN command by such as an MDI interruption before restarting the program.
- (2) An external output command during graphic check is ignored.
- (3) A program error (P460) will occur if an external output command is issued when the output device is unable to output due to a lack of connection, a low free space etc.
- (4) The NC automatically conducts a closing processing when it is reset between a POPEN command and a PCLOS command. So, execute a POPEN command by such as an MDI interruption before executing the rest of the machining program.
- (5) If a program error occurs between a POPEN command and a PCLOS command, NC will not automatically conduct a closing processing. So, there is no need to execute a POPEN command by such as an MDI interruption before executing the rest of the machining program.
- (6) If a program error occurs to the output port due to the setting of the memory card, execute an NC reset and close the output file before demounting the card.
- (7) When the output port is a memory card, the output file may be destroyed if the card is dismounted or the power is turned off without issuing a PCLOS command or NC reset after a POPEN command is issued.
- (8) As for M700VW 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.

13.9.9 Precautions



Precautions

When the user macro commands are employed, it is possible to use the M, S, T and other NC control commands together with the arithmetic, decision, branching and other macro commands for preparing the machining programs. When the former commands are made into executable statements and the latter commands into macro statements, the macro statement processing should be accomplished as quickly as possible in order to minimize the machining time, because such processing is not directly related to machine control.

By setting the parameter "#8101 macro single", the macro statements can be processed in parallel with the execution of the executable statement.

(During normal machining, set the parameter OFF to process all the macro statements together, and during a program check, set it ON to execute the macro statements block by block. Setting can be chosen depending on the purpose.)

Program example

N1 G91 G28 X0 Z0 ;(1)	
N2 G92 X0 Z0 ;(2)	
N3 G00 X-100. Z-100. ;(3)	
N4 #101 = 100. * COS[210.] ;(4)	(4),(5) Macro statements
N5 #103 = 100. * SIN[210.] ;(5)	
N6 G01 X#101 Z#103 F800 ;(6)	

Macro statements are:

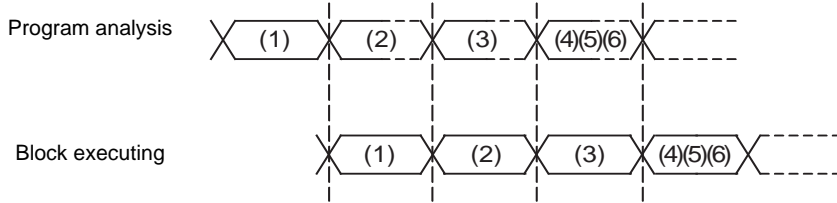
- (a) Arithmetic commands (block including =)
- (b) Control commands (block including GOTO, DO-END, etc.)
- (c) Macro call commands (including macro calls based on G codes and cancel commands G65, G66, G66.1, G67)

Execution statements refer to statements other than macro statements.

Flow of processing by the Program Example in the previous page

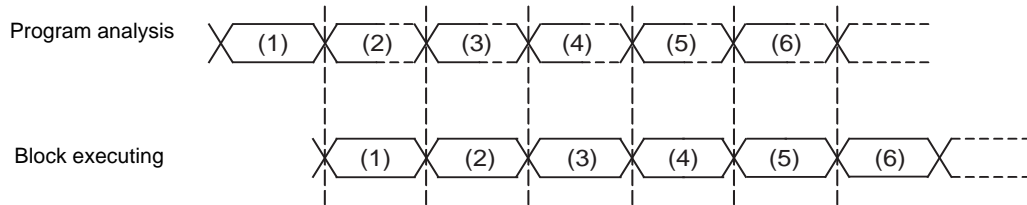
<Macro single OFF>

N4, N5 and N6 are processed in parallel with the control of the executable statement of N3. If the N4, N5 and N6 analysis is in time during N3 control, the machine movement will be continuously controlled.



<Macro single ON>

N4 is processed in parallel with the control of the executable statement of N3. After N3 is finished, N5 and N6 are analyzed, and then N6 is executed. So the machine control is held on standby during the N5 and N6 analysis time.



13.10 Mirror Image for Facing Tool Posts ; G68,G69



Function and purpose

In a machine in which the base turret and facing turret are integrated, this function is used to cut with the facing turret cutter using a program created with the base turret side.

The distance between the two turrets is set in the parameters beforehand.



Command format

G68 ; ... Mirror image for facing tool posts ON

G69 ; ... Mirror image for facing tool posts cancel

[T command mirror image for facing tool posts]

The mirror image for facing tool posts can be turned ON and OFF with the T command instead of the G68/G69 command.

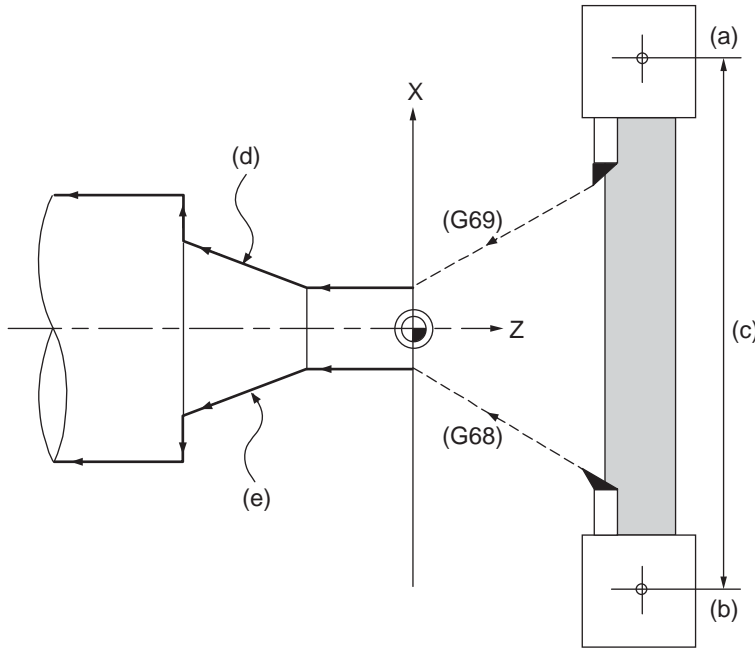
The T command for the G68 mode and the G69 mode is determined for each tool No. with the following base specification parameters.

#	Item		Details	Setting range (unit)
1119	Tmiron	Select the mirror image of each facing turret with T command	Selects whether the mirror image for facing tool posts with the T command is valid or invalid.	0: Invalid 1: Valid
1203	TmirS1	Select turret as facing turret with T command	The turret selection for the T command mirror image for facing tool posts corresponding to tool No. 1 to 32 is set.	0 to FFFFFFFF
1204	TmirS2	Select turret as facing turret with T command	The turret selection for the T command mirror image for facing tool posts corresponding to tool No. 33 to 64 is set.	0 to FFFFFFFF



Detailed description

When G68 is commanded, the following program coordinate system is shifted to the facing turret side of the axis for which the mirror image for facing tool posts is valid (hereafter unless noted in particular, the X axis will be described as the axis for which mirror image for facing tool posts is valid). The axis movement direction is reversed from the program command. When G69 is commanded, the following program coordinate system will be returned to the base turret side.



- (a) Base turret
- (b) Facing turret
- (c) Turret distance (parameter: radius value)
- (d) Programmed path
- (e) Facing turret path (Mirror image ON)

The mirror image for facing tool posts is valid for the following axes.

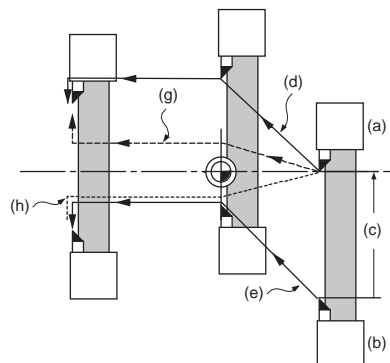
- (1) When parameter #1273 ext09 bit4=0
The mirror image for facing tool posts is valid for the 1st axis in the part system.
- (2) When parameter #1273 ext09 bit4=1
Mirror image for facing tool posts is valid for the axis determined in the following manner by the plane selected when mirror image for facing tool posts is selected. Note that even if the plane is changed during mirror image for facing tool posts, the axis for which mirror image for facing tool posts is valid will not change.

Plane selection	Axis for which mirror image for facing tool posts is valid
G17	J designated axis (#1027 base_J or #1030 aux_J axis in plane configuration)
G18	I designated axis (#1026 base_I or #1029 aux_I axis in plane configuration)
G19	K designated axis (#1028 base_K or #1031 aux_K axis in plane configuration)

Absolute value/incremental value

(1) Absolute value command

The command position for the Z axis is reversed symmetrically, and the base turret moves to the position shifted by the distance between cutters.



```
T0101;
G00 X0. ;
G68 ;
T0202 ;
G00 X10. Z0. ;
G01 Z-50. F400 ;
X20. ;
```

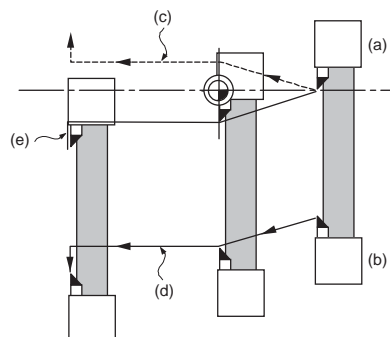
Mirror image for facing tool posts ON
Facing turret selection

- | | |
|--------------------------------|----------------------|
| (a) Base turret | (b) Facing turret |
| (c) Turret distance | (d) Base turret path |
| (e) Facing turret path | (g) Program |
| (h) Path for reversing program | |

* This figure does not incorporate the tool length.

(2) Incremental value command

The commanded X axis direction reverses, and the base turret moves.



```
T0101;
G00 X0. ;
G68 ;
T0202 ;
G00 U10. W-30. ;
G01 W-50. F400 ;
U10. ;
```

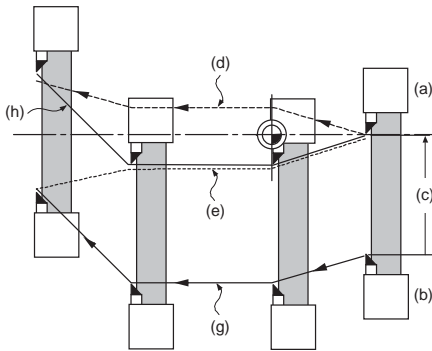
Mirror image for facing tool posts ON
Facing turret selection

- | | |
|---|------------------------|
| (a) Base turret | (b) Facing turret |
| (c) Program | (d) Facing turret path |
| (e) Base turret path (Path for reversing program) | |

* This figure does not incorporate the tool length.

(3) Changing from an incremental value command to an absolute value command

After changing to the absolute value, the same operation as “(1) Absolute value command” takes place.



```
T0101;
G00 X0. ;
G68 ;           Mirror image for facing tool posts ON
T0202 ;         Facing turret selection
G00 U10. W-30. ;
G01 W-50. F400 ;
X20. Z-80. ;
```

- (a) Base turret
- (b) Facing turret
- (c) Distance between cutters
- (d) Program
- (e) Path for reversing program
- (g) Facing turret path
- (h) Base turret path

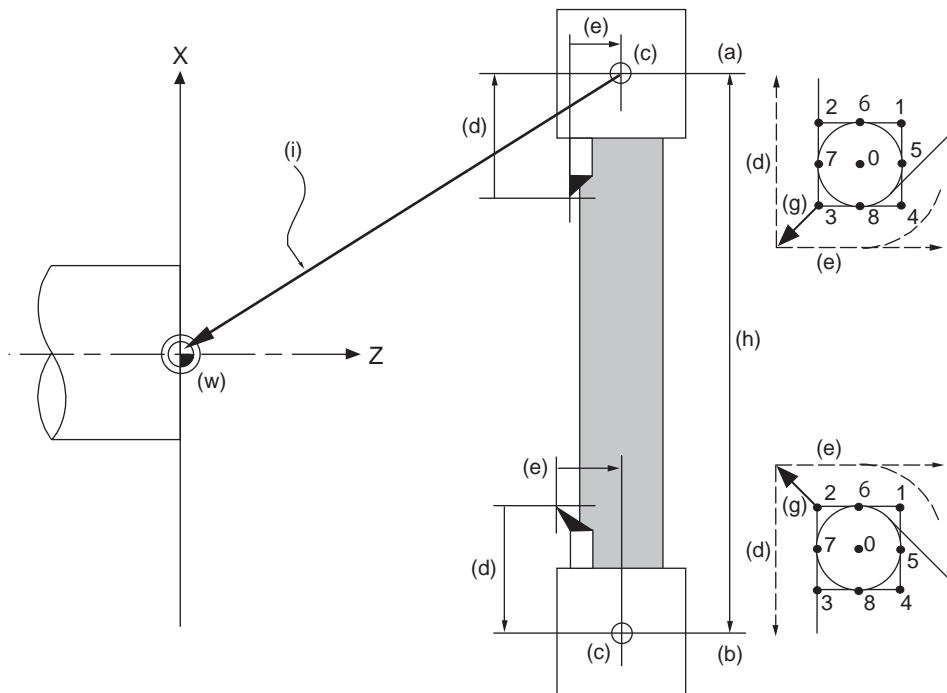
* This figure does not incorporate the tool length.

Displaying the workpiece coordinate values

The workpiece coordinate values of the axis for which mirror image for facing tool posts is valid are displayed in the following manner.

	#1273 ext09/bit3 = 0	#1273 ext09/bit3 = 1
In mirror image for facing tool posts mode	The workpiece coordinate values are at the position where the program command value sign has been reversed, and increases/decreases in the same manner as the actual movement, that is, movement on the machine coordinate system. (Indicates coordinate values for facing turret side.)	The workpiece coordinate values are the same as the program command value, and increases/decreases opposite the actual movement, that is, movement on the machine coordinate system.
At startup	Workpiece coordinate value = Workpiece coordinate value before mirror image for facing tool posts starts - distance between cutters (#1202 mirofs)	Workpiece coordinate value = {Workpiece coordinate value before mirror image for facing tool posts starts - distance between cutters (#1202 mirofs)} × (-1)
When movement is commanded while valid	Workpiece coordinate value = Workpiece coordinate value before movement + movement amount of machine coordinate	Workpiece coordinate value = Workpiece coordinate value before movement - movement amount of machine coordinate
When canceled	Workpiece coordinate value = Workpiece coordinate value before mirror image for facing tool posts is canceled - distance between cutters (#1202 mirofs)	Workpiece coordinate value = {Workpiece coordinate value before mirror image for facing tool posts is canceled × (-1)} + distance between cutters (#1202 mirofs)

Tool compensation of facing turret



- (a) Base turret
- (b) Facing turret
- (c) Tool length basic point
- (d) X tool length
- (e) Z tool length
- (g) Wear amount
- (h) Turret distance (parameter: radius value, X axis only)
- (i) Workpiece offset
- (w) Workpiece coordinate zero point

The original tool direction for the facing turret is shown in the drawing.

To set the facing turret tool to the same direction as the base turret tool, set the base specification parameter "#1118 mirr_A" to "1".

(1) Tool length offset

The tool length offset amount is the length from the tool nose to the tool length basic point. This also applies for the facing turret. Note that the offset amount setting value differs according to the tool length basic point position as shown below.

Tool length basic point and tool length offset

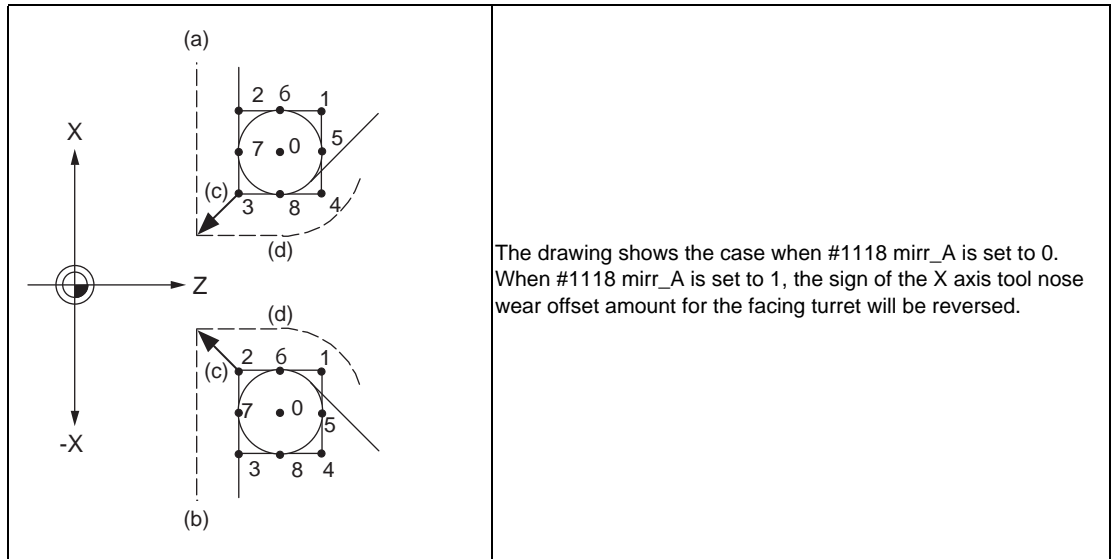
	Type A	Type B	Type C
Tool length basic point	Each turret basic point	Base turret basic point	Workpiece face center
Workpiece coordinate zero point	Workpiece face center	Workpiece face center	Workpiece face center
Turret distance	Distance between basic points of both turrets (radius value)	0	0
Workpiece offset	Workpiece coordinate zero point - base turret tool length basic point	Workpiece coordinate zero point - base turret tool length basic point	0
Tool length	Tool length basic point - tool nose position	Tool length basic point - tool nose position	Tool length basic point - tool nose position
Outline drawing			

- (a) Base turret
- (b) Facing turret
- (c) Tool length
- (d) Turret basic point
- (e) Turret distance
- (g) Workpiece offset
- (w) Workpiece coordinate zero point

The outline drawing in the table above shows the case when #1118 mirr_A is set to 0. When #1118 mirr_A is set to 1, the sign of the X axis tool length compensation amount for the facing turret will be reversed.

(2) Tool Nose Wear Compensation

The tool nose wear compensation amount is the length from the current tool nose to the original tool nose. The original tool nose is the tool nose when the tool length offset value was set.

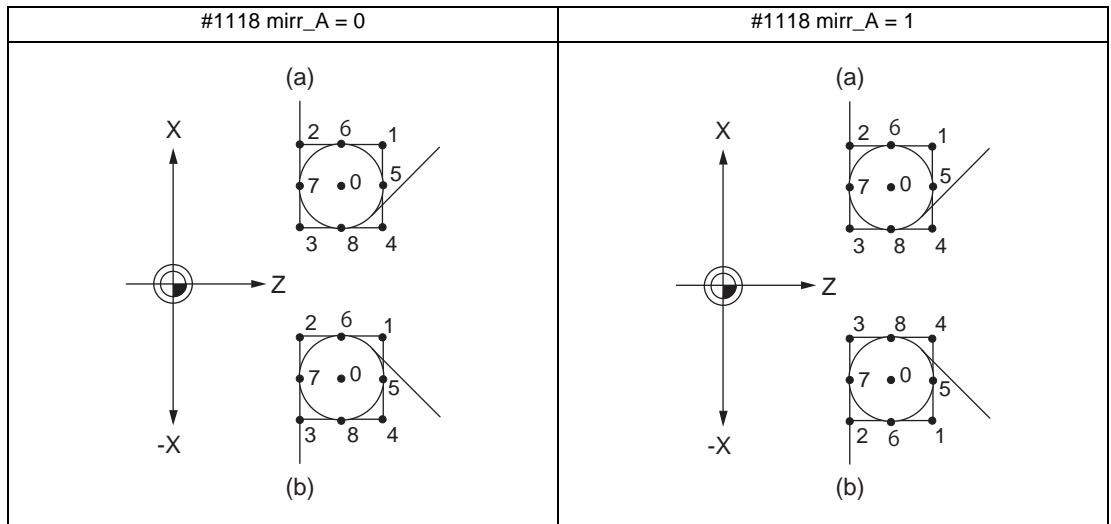


- (a) Base turret side
- (b) Facing turret side
- (c) Wear amount
- (d) Basic tool

(3) Tool nose point with nose R compensation

The tool nose point with nose R compensation is as follows.

Note that if the selected plane differs from when mirror image for facing tool posts was started, this will be handled as #1118 mirr_A = 0 even if it is #1118 mirr_A = 1.



- (a) Base turret side
- (b) Facing turret side

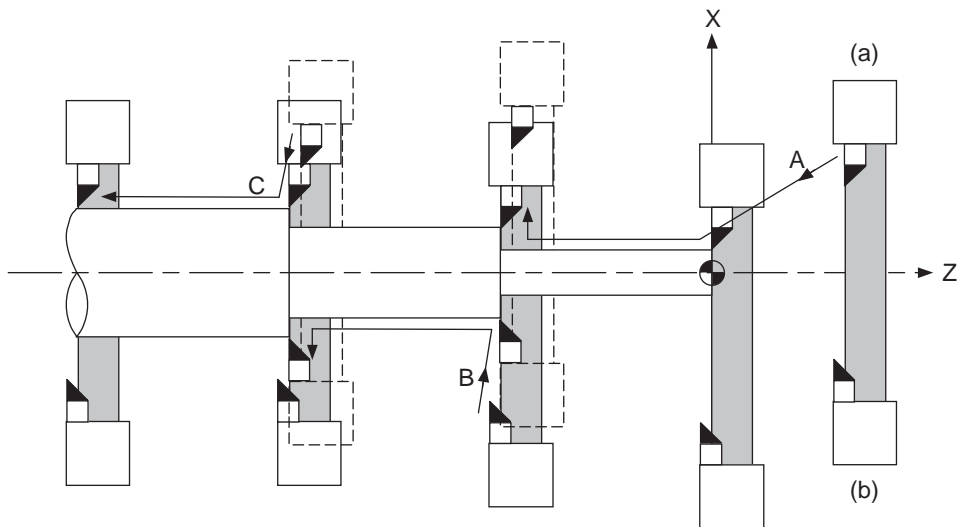
(4) Examples of setting the tool offset

	Type A				Type B				Type C			
	mirr_A = 0		mirr_A = 1		mirr_A = 0		mirr_A = 1		mirr_A = 0		mirr_A = 1	
	X	Z	X	Z	X	Z	X	Z	X	Z	X	Z
Workpiece offset	-100.	-150.	-100.	-150.	-100.	-150.	-100.	-150.	0.	0.	0.	0.
Turret distance (Setting value)	100.		100.		0.		0.		0.		0.	
Base turret tool length	40.	20.	40.	20.	40.	20.	40.	20.	40.	20.	40.	20.
Base turret tool wear amount	-4.	-2.	-4.	-2.	-4.	-2.	-4.	-2.	-4.	-2.	-4.	-2.
Base turret nose point	3		3		3		3		3		3	
Facing turret tool length	-30.	10.	30.	10.	170.	10.	-170.	10.	70.	-140.	-70.	-140.
Facing turret tool wear amount	2.	-1.	-2.	-1.	2.	-1.	-2.	-1.	2.	-1.	-2.	-1.
Facing turret nose point	2		3		2		3		2		3	



Program example

T0101 ; G00 X10. Z0. ; G01 Z-40. F400 ; X20. ;	Base turret selection	Machining with base turretA
G68 ; T0202 ; G00 X20. Z-40. ; G01 Z-80. F200 ; X30. ;	Mirror image for facing tool posts ON Facing turret selection	Machining with facing turretB
G69 ; T0101 ; G00 X30. Z-80. ; G01 Z-120. F400 ;	Mirror image for facing tool posts cancel Base turret selection	Machining with base turretC



(a) Base turret

(b) Facing turret



Relation with other functions

Reference position return (G28, G30)

Mirror image for facing tool posts will remain valid when moving to the intermediate point.

Mirror image for facing tool posts will be invalidated when moving past the intermediate point and during movement that ignores the intermediate point.

Mirror image for facing tool posts will be validated from the next block.

Machine coordinate system selection (G53)

Mirror image for facing tool posts will be invalidated during the movement.

Mirror image for facing tool posts will be validated from the next block.

Coordinate system setting (G92)

The following will take place on the axis for which mirror image for facing tool posts is valid.

Parameter		Details
#1273 ext09/bit3	0	The "command value $\times (-1)$ " coordinate system will be set as the coordinate system on the facing turret side.
	1	The commanded coordinate system will be set as the coordinate system on the facing turret side.

Reading the coordinate position with variables

The following will take place on the axis for which mirror image for facing tool posts is valid.

- (1) When previous block is end point coordinate system (#5001, etc.)
The "end point coordinate values on the base turret side in the workpiece coordinate system" will be read in.
- (2) For machine coordinate values (#5021, etc.)
The "coordinate values on the base turret side in the machine workpiece system" will be read in.
- (3) For machine coordinate values (#5041, etc.)
The "workpiece coordinate value in the mirror image for facing tool posts" will be read in.
- (4) For skip coordinate values (#5061, etc.)
If mirror image for facing tool posts is valid when skip is executed, the "workpiece coordinate values in the mirror image for facing tool posts" will be read in.

Resetting

- (1) When parameter #1210 RstGmd/bit14 = 0
The mirror image for facing tool posts will be canceled by reset.
- (2) When parameter #1210 RstGmd/bit14 = 1
The mirror image for facing tool posts state will be maintained even if it is reset.

External mirror image, parameter mirror image

- (1) A program error (P371) will occur if mirror image for facing tool posts is applied on the axis during external mirror image or parameter mirror image.
- (2) The "M01 operation error 1036" alarm will occur if external mirror image or parameter mirror image is applied on the axis during mirror image for facing tool posts.

Manual interruption

- (1) When manual absolute is OFF
If manual interruption is applied on an axis for which mirror image for facing tool posts is valid, the mirror image will not be applied on the interrupt amount. The interrupt movement amount will not be added to the workpiece coordinate values.
- (2) When manual absolute is ON
If manual interruption is applied on an axis for which mirror image for facing tool posts is valid, the mirror image will not be applied on the interrupt amount. The interrupt movement amount will be added to the workpiece coordinate values.



Precautions

- (1) The balance cut command (G68, G69) cannot be issued when the mirror image for facing tool posts option is valid in the G code list 6, 7 (#1037 cmdtyp = 7, 8).
- (2) If mirror image for facing tool posts is turned ON for an axis in a part system having axes that are exchanged with complex control or in a part system to which the axis control is transferred, the "M01 operation error 1035" will occur.
Note that mirror image for facing tool posts can be started after the axis is exchanged with complex control.
- (3) A program error (P486) will occur if polar coordinate interpolation, cylindrical interpolation or milling interpolation is commanded while mirror image for facing tool posts is ON.
- (4) A program error (P612) will occur if mirror image for facing tool posts is turned ON while the axis is moving with exponential interpolation.
- (5) A program error (P371) will occur if the axis to which mirror image for facing tool posts is commanded is a rotation axis.

13.11 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. By setting the parameters, "I_", "K_", "C_" can be set for chamfering instead of ",C_", and "R_" can be set for rounding instead of ",R_".

13.11.1 Corner Chamfering I ; G01 X_ Z_ ,C_/I_/K_/C_



Function and purpose

This chamfers a corner by connecting the both side of the hypothetical corner which would appear as if chamfering is not performed, by the amount commanded by ",C_" (or "I_", "K_", "C_").



Command format

```
N100 G01 X_ Z_ ,C_ (or I_ / K_ / C_);
N200 G01 X_ Z_ ;
```

.C / I / K / C	Length up to chamfering starting point or end point from imaginary corner
----------------	---

Chamfering is performed at the point where N100 and N200 intersect.



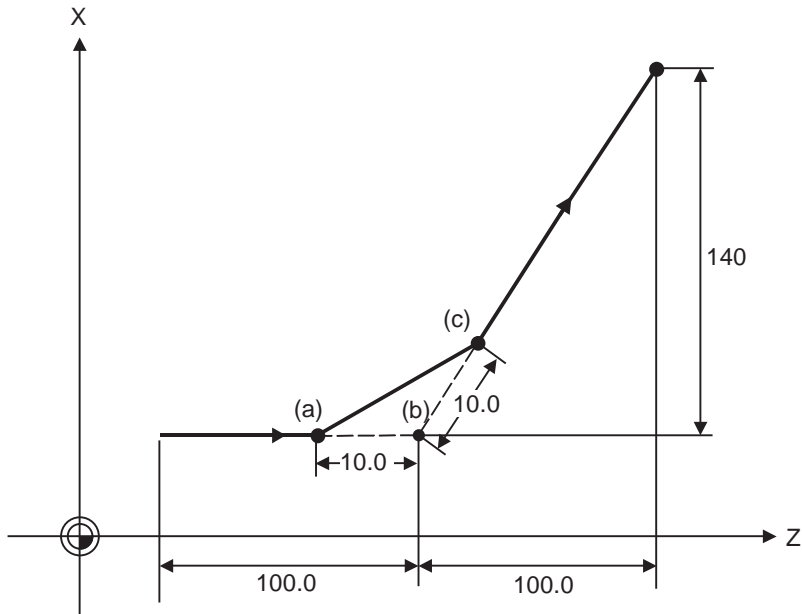
Detailed description

- (1) The start point of the block following the corner chamfering is the hypothetical corner intersection point.
- (2) If the basic specification parameter "#1272 ext08/bit6" is "0", the ",C" command will be interpreted as a C command if there is no "," (comma).
- (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/corner rounding commands exist in a 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) 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.
- (7) Program error (P383) will occur when the movement amount in the corner chamfering block is less than the chamfering amount.
- (8) Program error (P384) will occur when the movement amount is less than the chamfering amount in the block following the corner chamfering block.
- (9) Program error (P382) will occur when a movement command is not issued in the block following the corner chamfering I command.
- (10) If "C" is used as the axis name or the 2nd miscellaneous function, corner chamfering cannot be commanded with "C".
- (11) Corner chamfering cannot be commanded with "I" or "K" in a circular command block. "I" and "K" are the circular center commands.



Program example

```
G01 W100. ,C10. F100 ;
U280. W100. ;
```



- (a) Chamfering start point
- (b) Hypothetical corner intersection point
- (c) Chamfering end point



Precautions

- (1) Corner chamfering and corner rounding can be commanded with "I", "K", "R" only when the 1st block of the corner chamfering/corner rounding command is linear.
- (2) Corner chamfering with "I", "K", and corner rounding with "R" can be commanded when the 1st block of the corner chamfering/corner rounding command is linear and the 2nd block is circular. "I", "K" in the 2nd block are the circular center commands.
 N100 G01 Xx Zz li ;..... li corner chamfering length
 N200 G02 Xx Zz li Kk ;..... li, Kk circular center command
- (3) If ",C_", ",R_" or "I_", "K_", "C_", "R_" are commanded in a same block, ",C_", ",R_" will have the priority.

13.11.2 Corner Rounding I ; G01 X_ Z_ ,R_/R_



Function and purpose

This performs a corner rounding to the both side of the hypothetical corner which would appear as if chamfering is not performed, at the radius of the circular commanded with ",R_" (or "R_").



Command format

N100 G01 X__ Z__ ,R__ (or R__);
N200 G01 X__ Z__;

,R / R	Circular radius of corner rounding
--------	------------------------------------

Corner rounding is performed at the point where N100 and N200 intersect.



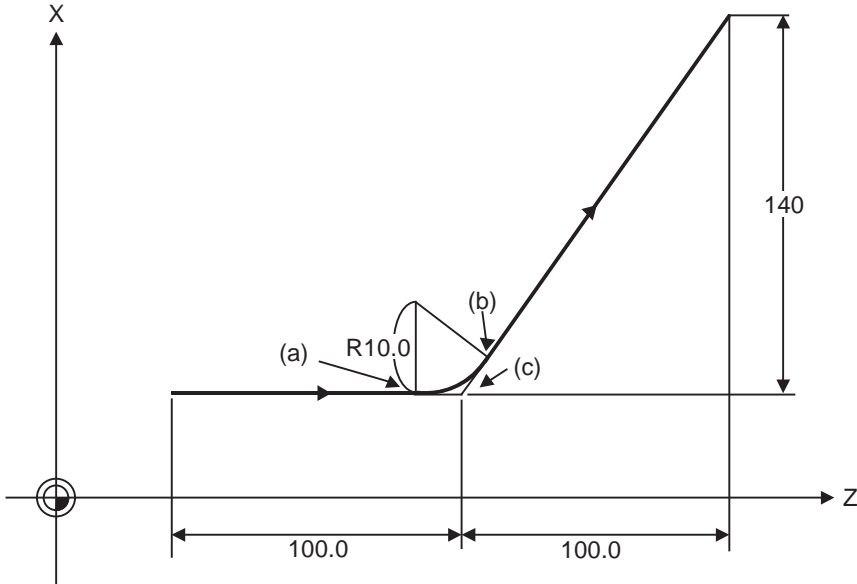
Detailed description

- (1) The start point of the block following the corner rounding is the hypothetical corner intersection point.
- (2) If the basic specification parameter "#1272 ext08/bit6" is "0", the ",R" command will be interpreted as a R command if there is no "," (comma).
- (3) When both the corner chamfering/corner rounding commands exist 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) 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.
- (6) Program error (P383) will occur when the movement amount in the corner rounding block is less than the R value.
- (7) Program error (P384) will occur when the movement amount is less than the R value in the block following the corner rounding.
- (8) Program error (P382) will occur if a movement command is not issued in the block following corner rounding.
- (9) Corner rounding cannot be commanded with "R" in a circular command block. "R" is the circular radius command.



Program example

```
G01 W100. ,R10. F100 ;
U280. W100. ;
```



- (a) Corner rounding start point
- (b) Corner rounding end point
- (c) Hypothetical corner intersection point



Precautions

- (1) Corner chamfering and corner rounding can be commanded with "I", "K", "R" only when the 1st block of the corner chamfering/corner rounding command is linear.
- (2) Corner chamfering with "I", "K", and corner rounding with "R" can be commanded when the 1st block of the corner chamfering/corner rounding command is linear and the 2nd block is circular. "I", "K" in the 2nd block are the circular center commands.
 N100 G01 Xx Zz li ;..... li corner chamfering length
 N200 G02 Xx Zz li Kk ;..... li, Kk circular center command
- (3) If ",C_", ",R_" or "I_", "K_", "C_", "R_" are commanded in a same block, ",C_", ",R_" will have the priority.

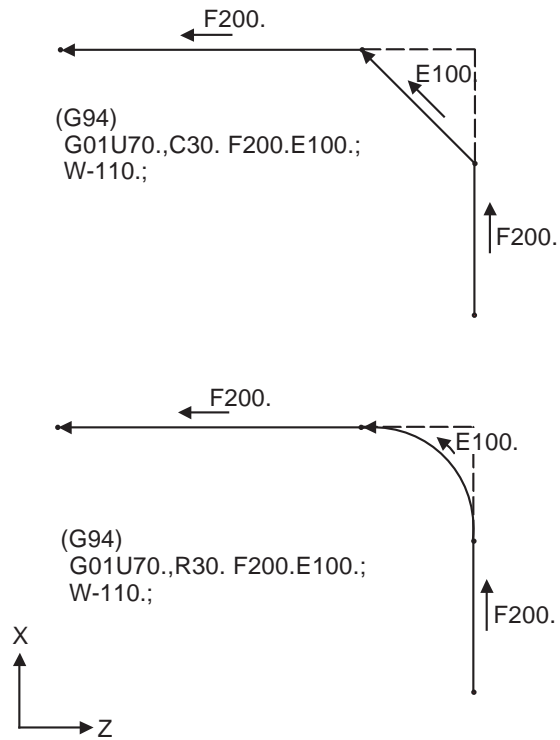
13.11.3 Corner Chamfering Expansion/Corner Rounding Expansion



Function and purpose

Using an E command, the feedrate can be designated for the corner chamfering and corner rounding section. In this way, the corner section can be cut into a correct shape.

Example

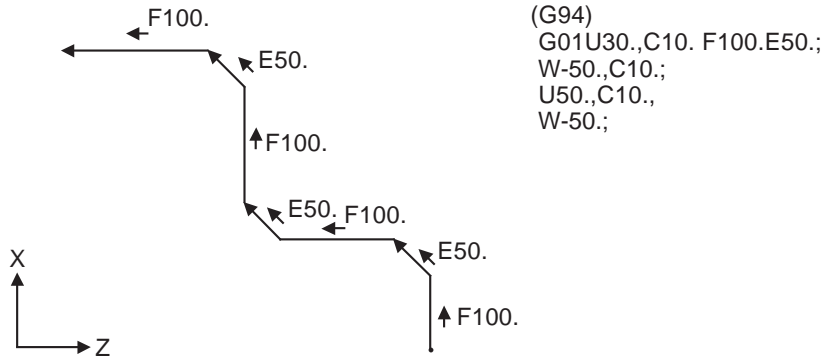




Detailed description

- (1) The E command is modal. It is also valid for the feed in the next corner chamfering/corner rounding section.

Example

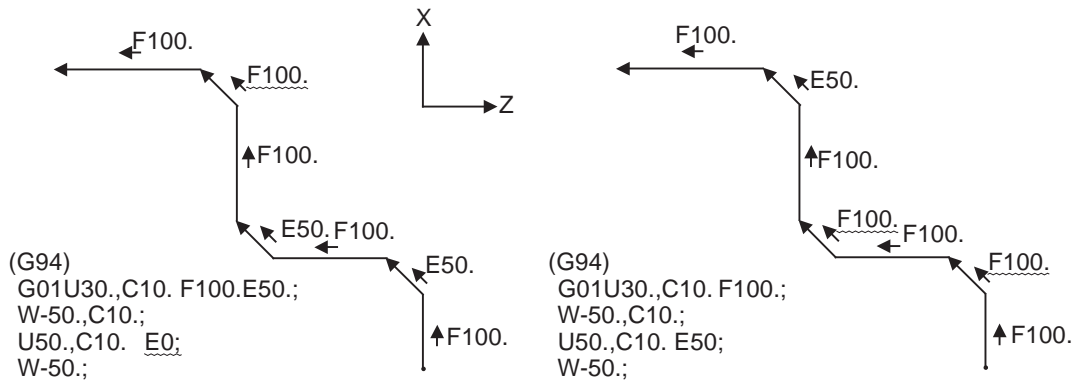


- (2) E command modal has separate asynchronous feedrate modal and synchronous feedrate modal functions.

Which one is validated depends on the asynchronous/synchronous mode (G94/G95).

- (3) When the E command is 0, or when there has not been an E command up to now, the corner chamfering/corner rounding section feedrate will be the same as the F command feedrate.

Example



- (4) E command modal is not cleared even if the reset button is pressed. It is cleared when the power is turned OFF. (In the same manner as F commands.)
- (5) All E commands except those shown below are at the corner chamfering/corner rounding section feedrate.
 - E commands during thread cutting modal
 - E commands during thread cutting cycle modal

13.11.4 Interrupt during Corner Chamfering/Interrupt during Corner Rounding

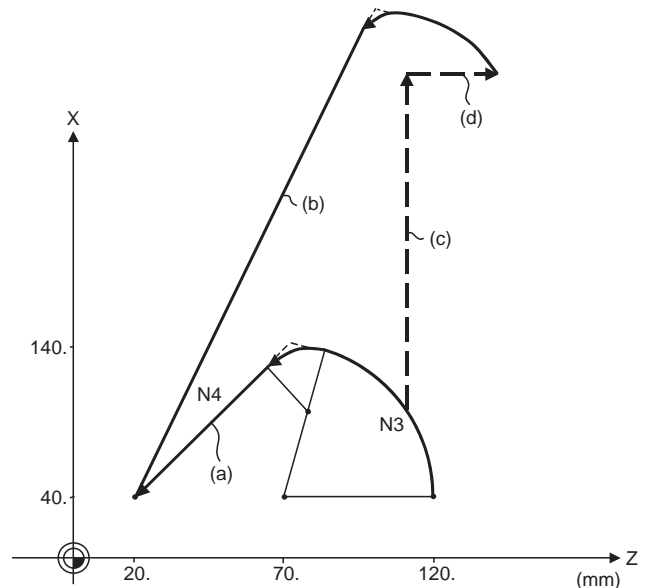


Detailed description

- (1) Shown below are the operations of manual interruption during corner chamfering or corner rounding.

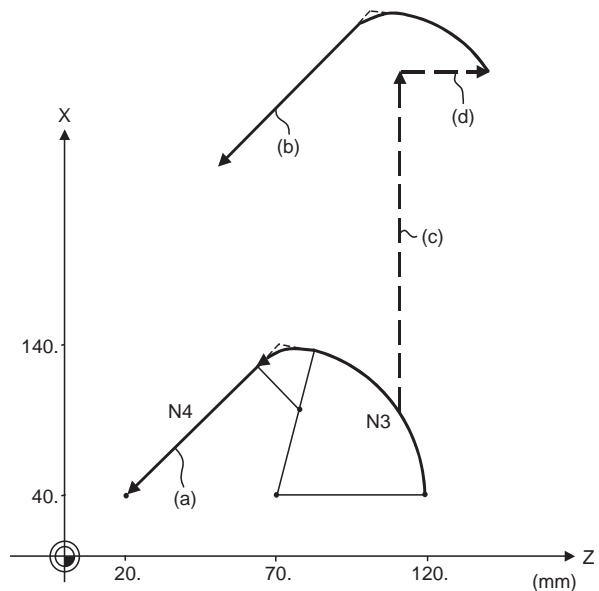
With an absolute value command and manual absolute switch ON.

```
N1 G28 XZ;
N2 G00 X40. Z120.;
N3 G03 X140.Z70. K-50. ,R20. F100 ;
N4 G01 X40. Z20. ;
:
```



With an incremental value command and manual absolute switch OFF

```
N1 G28 XZ;
N2 G00 U40. W120.;
N3 G03 U100. W-50. K-50. ,R20. F100 ;
N4 G01 U-100.W-50. ;
:
```



- (a) When interrupt is not applied
(c) X-axis interrupt

- (b) When interrupt is applied
(d) Z-axis interrupt

- (2) With a single block during corner chamfering or corner rounding, the tool stops after these operations are executed.

13.12 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. By setting the parameters, "I_", "K_", "C_" can be set for chamfering instead of ",C_", and "R_" can be set for rounding instead of ",R_". Corner chamfering and corner rounding can both be commanded with an absolute value or incremental value.

13.12.1 Corner Chamfering II ; G01/G02/G03 X_ Z_ ,C_/I_/K_/C_



Function and purpose

The corner is chamfered by commanding ",C" (or "I_", "K_", "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_ Z_ I_ K_ ,C_ (or C_);
N200 G01 X_ Z_;
```

,C / 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 corner chamfering and corner rounding options are required to use this function. A program error (P381) will occur if the function is commanded without the option.
- (2) The start point of the block following the corner chamfering is the hypothetical corner intersection point.
- (3) If the basic specification parameter "#1272 ext08/bit6" is "0", the ",C" command will be interpreted as a C command if there is no "," (comma).
- (4) If there are multiple or duplicate corner chamfering commands in a same block, the last command will be valid.
- (5) When both corner chamfering and corner rounding are commanded in the same block, the latter command will be valid.
- (6) Tool compensation is calculated for the shape which has already been subjected to corner chamfering.
- (7) Program error (P385) will occur when positioning or thread cutting is commanded in the corner chamfering command block or in the next block.
- (8) Program error (P382) will occur when the block following corner chamfering contains a G command other than group 01 or another command.
- (9) Program error (P383) will occur when the movement amount in the block, commanding corner chamfering, is less than the chamfering amount.
- (10) Program error (P384) will occur when the movement amount is less than the chamfering amount in the block following the block commanding corner chamfering.
- (11) Even if a diameter is commanded, it will be handled as a radial command value during corner chamfering.
- (12) Program error (P382) will occur when a movement command is not issued in the block following the corner chamfering II command.
- (13) If "C" is used as the axis name or the 2nd miscellaneous function, corner chamfering cannot be commanded with "C".

- (14) Corner chamfering cannot be commanded with "I" or "K" in a circular command block. "I" and "K" are the circular center commands.

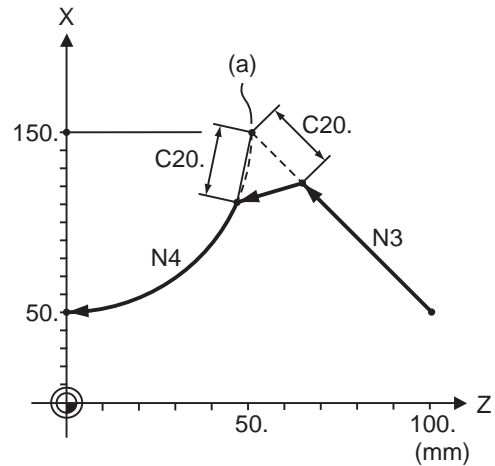


Program example

(1) Linear - arc

Absolute value command
 N1 G28 X Z ;
 N2 G00 X50. Z100. ;
 N3 G01 X150. Z50. ,C20. F100 ;
 N4 G02 X50. Z0 I0 K-50. ;
 :

Relative value command
 N1 G28 X Z ;
 N2 G00 U25. W100. ;
 N3 G01 U50. W-50. ,C20. F100 ;
 N4 G02 U-50. W-50. I0 K-50. ;
 :

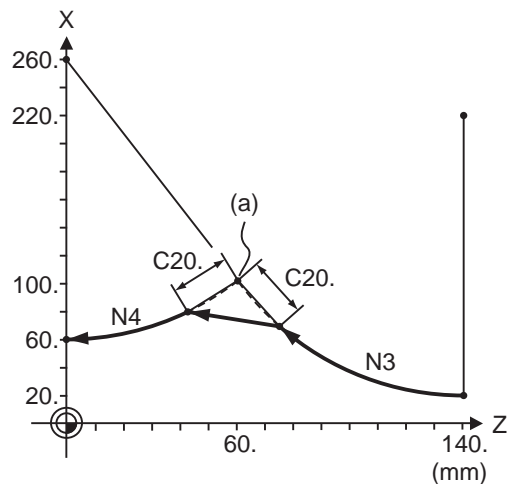


(a) Hypothetical corner intersection point

(2) Arc - arc

Absolute value command
 N1 G28 X Z ;
 N2 G00 X20. Z140. ;
 N3 G02 X100. Z60. I100. K0. ,C20. F100 ;
 N4 X60. Z0 I80. K-60. ;
 :

Relative value command
 N1 G28 X Z ;
 N2 G00 U10. W140. ;
 N3 G02 U40. W-80. R100. ,C20. F100 ;
 N4 U-20. W-60. I80. K-60. ;
 :



(a) Hypothetical corner intersection point



Precautions

- Corner chamfering and corner rounding can be commanded with "I", "K", "R" only when the 1st block of the corner chamfering/corner rounding command is linear.
- Corner chamfering with "I", "K", and corner rounding with "R" can be commanded when the 1st block of the corner chamfering/corner rounding command is linear and the 2nd block is circular. "I", "K" in the 2nd block are the circular center commands.
 N100 G01 X__ Z__ I__ ; I corner chamfering length
 N200 G02 X__ Z__ I__ K__ ; I,K circular center command
- If ",C_", ",R_" or "I_", "K_", "C_", "R_" are commanded in a same block, ",C_", ",R_" will have the priority.

13.12.2 Corner Rounding II ; G01/G02/G03 X_ Z_ ,R_/R_



Function and purpose

The corner is rounded by commanding ",R_" (or "R_") in the 1st block of the two continuous blocks containing an arc.



Command format

```
N100 G03 X__ Z__ I__ K__ ,R__ (or R__);
N200 G01 X__ Z__ ;
```

,R / R	Arc radius of corner rounding
--------	-------------------------------

Corner rounding is performed at the point where N100 and N200 intersect.



Detailed description

- (1) The corner chamfering and corner rounding options are required to use this function. Program error (P381) will occur if the function is commanded without the option.
- (2) The start point of the block following the corner rounding is the hypothetical corner intersection point.
- (3) If the basic specification parameter "#1272 ext08/bit6" is "0", the ",R" command will be interpreted as a R command if there is no "," (comma).
- (4) When both corner chamfering and corner rounding are commanded in a same block, the latter command will be valid.
- (5) Tool compensation is calculated for the shape which has already been subjected to corner rounding.
- (6) Program error (P385) will occur when positioning or thread cutting is commanded in the corner chamfering command block or in the next block.
- (7) Program error (P382) will occur when the block following corner chamfering contains a G command other than group 01 or another command.
- (8) Program error (P383) will occur when the movement amount in the corner rounding block is less than the R value.
- (9) Program error (P384) will occur when the movement amount is less than the R value in the block following the corner rounding.
- (10) Even if a diameter is commanded, it will be handled as a radial command value during corner chamfering.
- (11) A program error (P382) will occur if a movement command is not issued in the block following corner rounding.
- (12) Corner rounding cannot be commanded with "R" in a circular command block. "R" is the circular radius command.

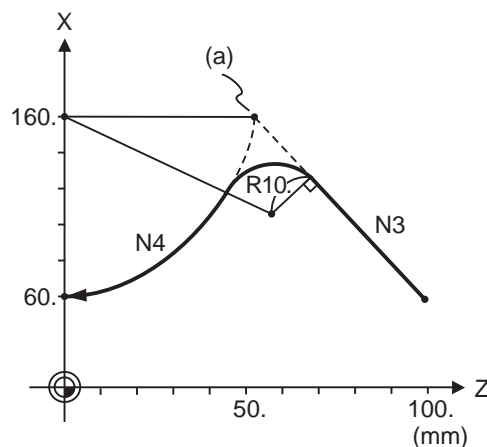


Program example

(1) Linear - arc

Absolute value command
 N1 G28 X Z ;
 N2 G00 X60. Z100. ;
 N3 G01 X160. Z50. ,R10. F100 ;
 N4 G02 X60. Z0 I0 K-50. ;
 :

Relative value command
 N1 G28 X Z ;
 N2 G00 U30. W100. ;
 N3 G01 U50. W-50. ,R10. F100 ;
 N4 G02 U-50. W-50. I0 K-50. ;
 :

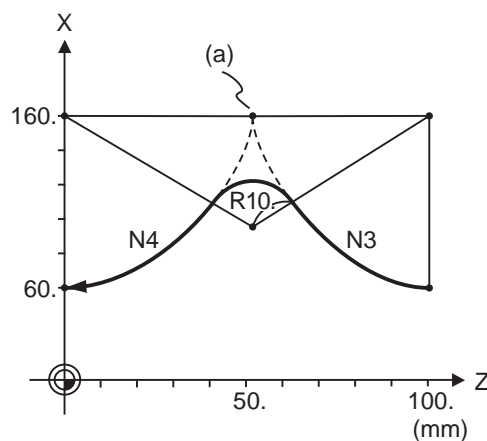


(a) Hypothetical corner intersection point

(2) Arc - arc

Absolute value command
 N1 G28 X Z ;
 N2 G00 X60. Z100. ;
 N3 G02 X160. Z50. R60 ,R10. F100 ;
 N4 X60. Z0 R50. ;
 :

Relative value command
 N1 G28 X Z ;
 N2 G00 U30. W100. ;
 N3 G02 U50. W-50. I50. K0 ,R10. F100 ;
 N4 U-50. W-50. I0. K-50. ;
 :



(a) Hypothetical corner intersection point



Precautions

- Corner chamfering and corner rounding can be commanded with "I", "K", "R" only when the 1st block of the corner chamfering/corner rounding command is linear.
- Corner chamfering with "I", "K", and corner rounding with "R" can be commanded when the 1st block of the corner chamfering/corner rounding command is linear and the 2nd block is circular. "I", "K" in the 2nd block are the circular center commands.
 N100 G01 X__ Z__ I__ ; I corner chamfering length
 N200 G02 X__ Z__ I__ K__ ; I,K circular center command
- If ",C_", ",R_" or "I_", "K_", "C_", "R_" are commanded in a same block, ",C_", ",R_" will have the priority.



13.12.3 Corner Chamfering Expansion/Corner Rounding Expansion

For details, refer to "Corner Chamfering I / Corner Rounding I: Corner Chamfering Expansion / Corner Rounding Expansion".

13.12.4 Interrupt during Corner Chamfering/Interrupt during Corner Rounding

For details, refer to "Corner Chamfering I / Corner Rounding I: Interrupt during Corner Chamfering Interrupt during / Corner Rounding".

13.13 Linear Angle Command ; G01 X_/Z_ 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.



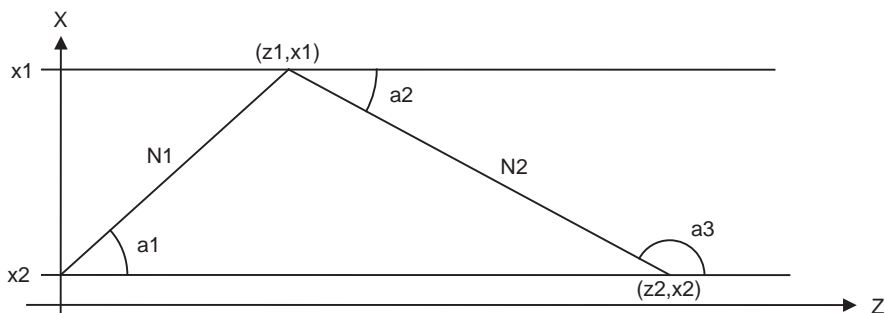
Command format

```
N1 G01 Xx1(Zz1) Aa1;
N2 G01 Xx2(Zz2) A-a2; (A-a2 can also be set as Aa 3.)
```

```
N1 G01 Xx1(Zz1) ,Aa1;
N2 G01 Xx2(Zz2) ,A-a2;
```

This designates the angle and the X or Z axis coordinates.

Select the command plane with G17 to G19.



Detailed description

- (1) As seen from the + direction of the horizontal axis of the selected plane, the counterclockwise (CCW) direction is considered to be + and the clockwise direction (CW) -.
- (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.

13.14 Geometric

13.14.1 Geometric I ; G01 A_



Function and purpose

When it is difficult to calculate the intersection point of two straight lines in a continuous linear interpolation command, the end point of the first straight line will be automatically calculated inside the CNC and the movement command will be controlled, provided that the slope of the first straight line as well as the end point coordinates and slope of the second straight line are commanded.

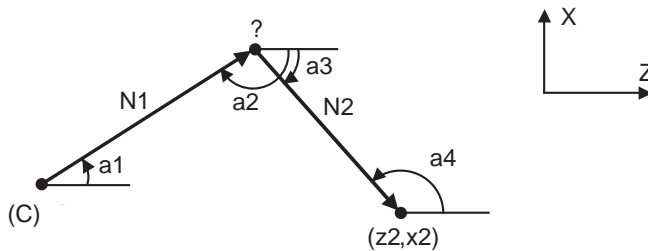
(Note) If the parameter "#1082 Geomet" is set to 0, geometric I will not function.



Command format

N1 G01 Aa1 (A-a2) Ff1;
N2 Xx2 Zz2 Aa4 (A-a3) Ff2;

Aa1, A-a2, A-a3, Aa4	Angle
Ff1, Ff2	Feedrate
Xx2, Zz2	Next block end point coordinates



(C) Present position



Detailed description

- (1) Program error (P396) will occur when the geometric command is not on the selected plane.
- (2) As seen from the + direction of the horizontal axis of the selected plane, the counterclockwise (CCW) direction is considered to be + and the clockwise direction (CW) -.
- (3) The range of slope "a" is between -360.000 and 360.000.
When a value outside this range is commanded, it will be divided by 360 (degrees) and the remainder will be commanded.
(Example) If 400 is commanded, 40°(remainder of 400/360) will become the command angle.
- (4) The slope of the line can be commanded on either the start or end point side. Whether the commanded slope is on the start or end point side is identified automatically inside the NC unit.
- (5) The end point coordinates of the second block should be commanded with absolute values. If incremental values are used, program error (P393) will occur.
- (6) The feedrate can be commanded for each block.
- (7) When the angle where the two straight lines intersect is less than 1°, program error (P392) will occur.
- (8) Program error (P396) will occur when the plane is changed in the 1st block and 2nd block.
- (9) This function is ignored when address A is used for the axis name or as the 2nd miscellaneous function.
- (10) Single block stop is possible at the end point of the 1st block.
- (11) Program error (P394) will occur when the 1st and 2nd blocks do not contain the G01 or G33 command.



Relation with other functions

(1) Corner chamfering and corner rounding can be commanded after the angle command in the 1st block.

<p>(Example 1) N1 Aa1 ,Cc1 ; N2 Xx2 Zz2 Aa2 ;</p>	
<p>(Example 2) N1 Aa1 ,Rr1 ; N2 Xx2 Zz2 Aa2 ;</p>	

(2) The geometric command I can be issued after the corner chamfering or corner rounding command.

<p>(Example 3) N1 Xx2 Zz2 ,Cc1 ; N2 Aa1 ; N3 Xx3 Zz3 Aa2 ;</p>	
--	--

(3) The geometric command I can be issued after the linear angle command.

<p>(Example 4) N1 Xx2 Aa1 ; N2 Aa2 ; N3 Xx3 Zz3 Aa3 ;</p>	
---	--

13.14.2 Geometric IB

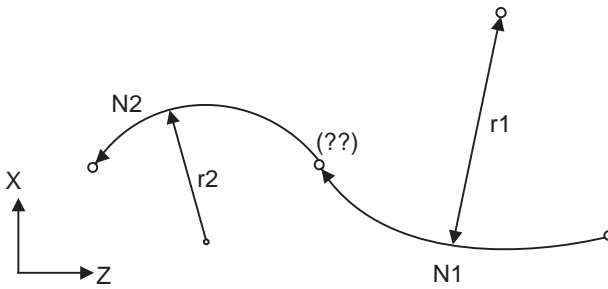


Function and purpose

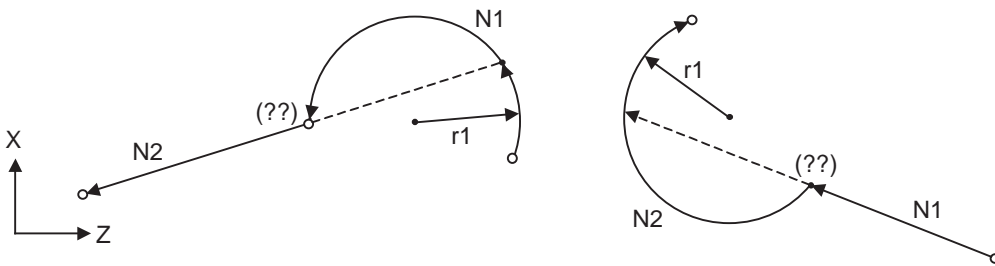
With the geometric IB function, the contact and intersection are calculated by commanding a arc center point or linear angle in the movement commands of two continuous blocks (only blocks with arc commands), instead of commanding the first block end point.

(Note) If the parameter (#1082 Geomet) is not set to 2, geometric IB will not function.

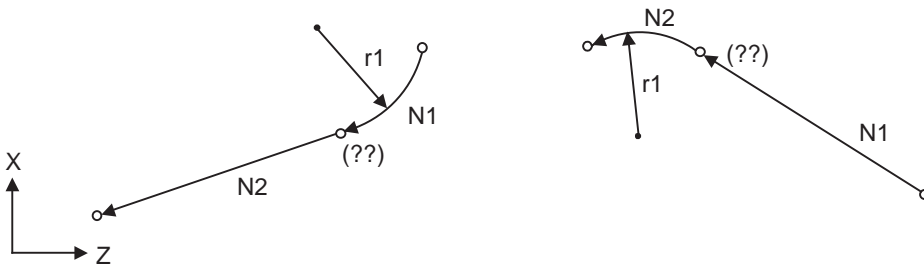
Two-arc contact



Linear - arc (arc - linear) intersection



Linear - arc (arc - linear) contact



13.14.2.1 Geometric IB (Automatic calculation of two-arc contact) ; G02/G03 P_Q_/R_



Function and purpose

When the contact of two continuous contacting arcs is not indicated in the drawing, it can be automatically calculated by commanding the 1st circular center coordinate value or radius, and the 2nd arc end point absolute value and center coordinate value or radius.



Command format

```
N1 G02(G03) Pp1 Qq1 Ff1;
N2 G03(G02) Xx2 Zz2 Pp2 Qq2 Ff2;
```

```
N1 G02(G03) Pp1 Qq1 Ff1;
N2 G03(G02) Xx2 Zz2 Rr2 Ff2;
```

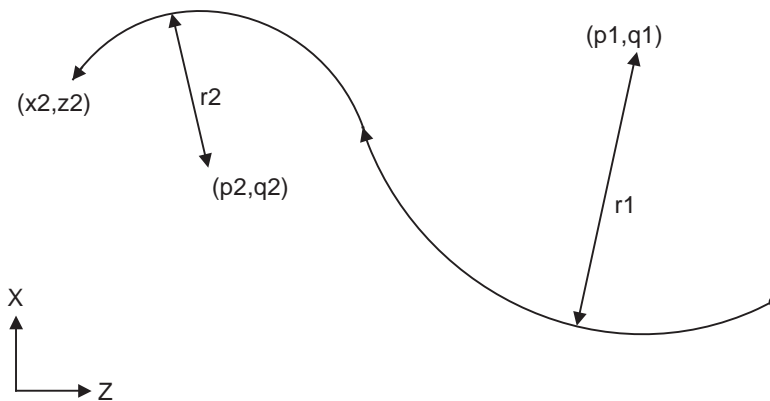
```
N1 G02(G03) Rr1 Ff1;
N2 G03(G02) Xx2 Zz2 Pp2 Qq2 Ff2;
```

P,Q	X and Z axes circular center coordinate absolute value (diameter/radius value command) The center address for the 3rd axis is commanded with A.
R	Arc radius (when a (-) sign is attached, the arc is judged to be 180° or more)

* I and K (X and Z axes arc center coordinate incremental value) commands can be issued instead of P and Q.

1st block arc : Radius command incremental amount from the start point to the center

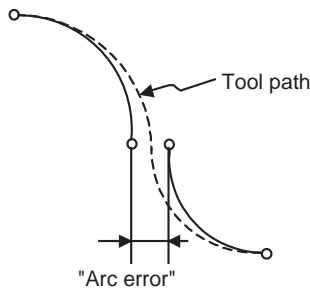
2nd block arc : Radius command incremental amount from the end point to the center



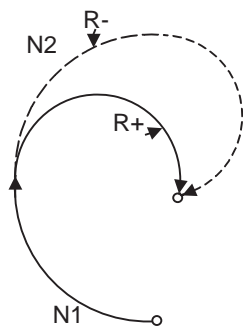


Detailed description

- (1) Program error (P393) will occur before the 1st block if the 2nd block is not a coordinate absolute value command.
- (2) Program error (P398) will occur before the 1st block if there is no geometric IB specification.
- (3) Program error (P395) will occur before the 1st block if there is no R (here, the 1st block is designated with P, Q (I, K)) or P, Q (I, K) designation in the 2nd block.
- (4) Program error (P396) will occur before the 1st block if another plane selection command (G17 to G19) is issued in the 2nd block.
- (5) Program error (P397) will occur before the 1st block if two arcs that do not contact are commanded.
- (6) The contact calculation accuracy is $\pm 1 \mu\text{m}$ (fractions rounded up).
- (7) Single block operation stops at the 1st block.
- (8) When I or K is omitted, the values are regarded as I0 and K0. P and Q cannot be omitted.
- (9) The error range in which the contact is obtained is set in parameter "#1084 RadErr".



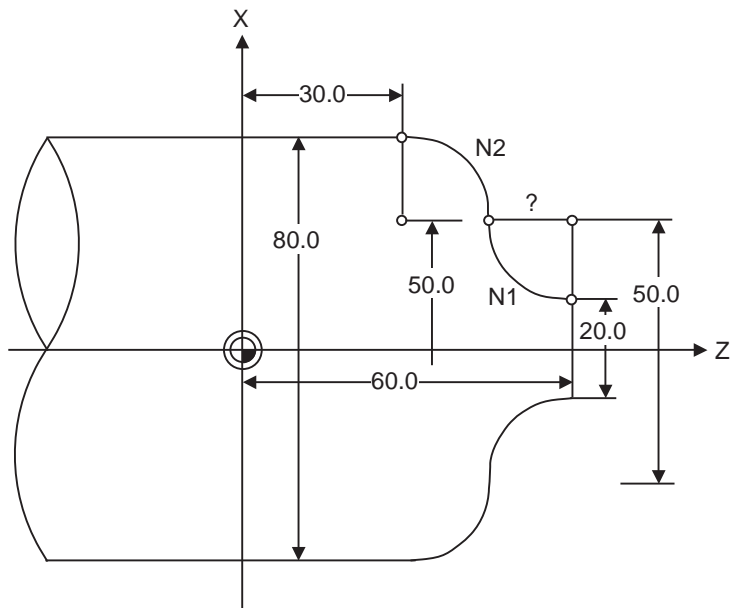
- (10) For an arc block perfect circle command (arc block start point = arc block end point), the R designation arc command finishes immediately, and there is no operation. Thus, use a PQ (IK) designation arc command.
- (11) G codes of the G modal group 1 in the 1st/2nd block can be omitted.
- (12) Addresses being used as axis names cannot be used as command addresses for arc center coordinates or arc radius.
- (13) When the 2nd block arc inscribes the 1st block arc and the 2nd block is an R designation arc, the R+ sign becomes the inward turning arc command, and the R- sign becomes the outward turning arc command.





Program example

(1) PQ, PQ command

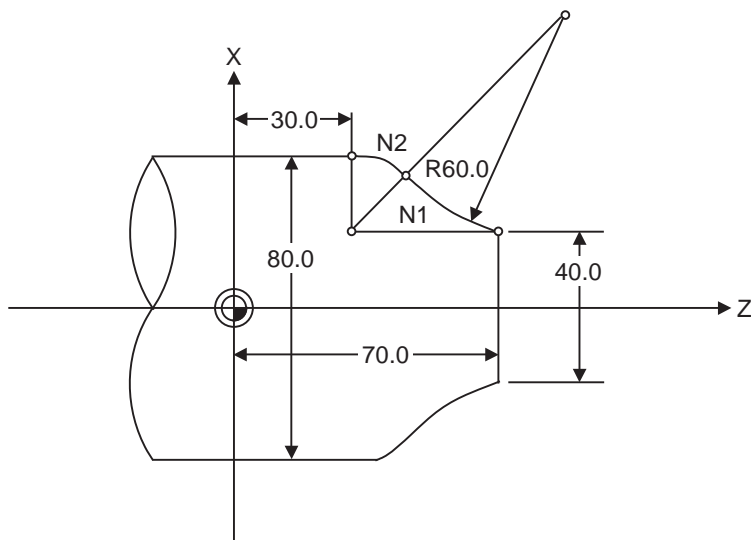


```
G01 X20.0 Z60.0;
N1 G02 P50.0 Q60.0 F100;
N2 G03 X80.0 Z30.0 P50.0 Q30.0;

```

(mm)

(2) PQ, R command



```
G01 X40.0 Z70.0 F100;
N1 G02 R60.0;
N2 G03 X80.0 Z30.0 P40.0 Q30.0;

```

(mm)



Relation with other functions

Command	Tool path
<p>Geometric IB + corner chamfering II</p> <p>N1 G09 P_ Q_ ; N2 G02 X_ Z_ R_ ,C_ ; G02 X_ Z_ R_ ;</p>	
<p>Geometric IB + corner rounding II</p> <p>N1 G03 P_ Q_ ; N2 G02 X_ Z_ R_ ,R_ ; G02 X_ Z_ R_ ;</p>	
<p>Geometric IB + corner chamfering II</p> <p>N1 G03 P_ Q_ ; N2 G02 X_ Z_ R_ ,C_ ; G01 X_ Z_ ;</p>	
<p>Geometric IB + corner rounding II</p> <p>N1 G03 P_ Q_ ; N2 G02 X_ Z_ R_ ,R_ ; G01 X_ Z_ ;</p>	

13.14.2.2 Geometric IB (Automatic calculation of linear - arc intersection) ; G01 A_ , G02/G03

P_Q_H_



Function and purpose

When the contact point of a shape in which a line and arc contact is not indicated in the drawing, it can be automatically calculated by commanding the following program.



Command format

```
N1 G01 Aa1(A-a2) Ff1 ;
N2 G02(G03) Xx2 Zz2 Pp2 Qq2 Hh2 Ff2 ;
```

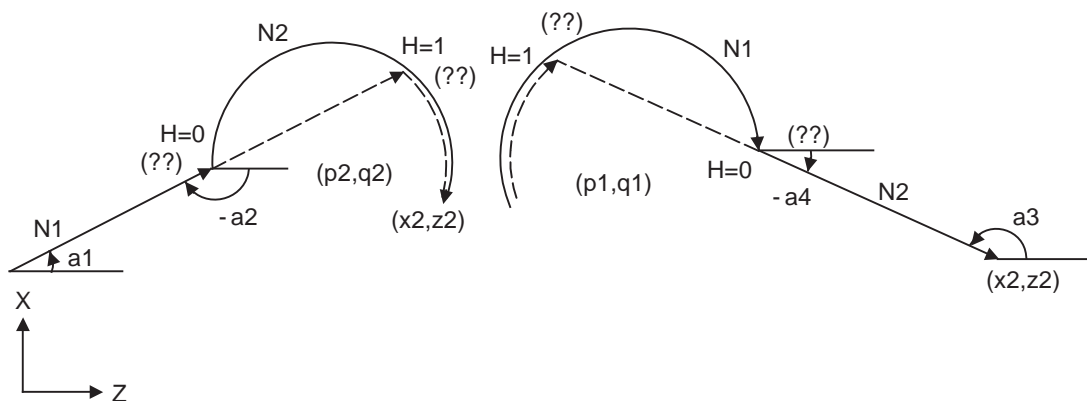
```
N1 G02(G03) Pp1 Qq1 Hh1 (,Hh1) Ff1 ;
N2 G1 Xx2 Zz2 Aa3 (A-a4) Ff2 ;
```

A	Linear angle (-360.000° to 360.000°)
P,Q	X and Z axes circular center coordinate absolute value (diameter/radius value command) The center address for the 3rd axis is commanded with A.
H (,H)	Selection of linear - arc intersection 0: Intersection of the shorter line 1: Intersection of the longer line

* I and K (X and Z axes arc center coordinate incremental value) commands can be issued instead of P and Q.

1st block arc : Radius command incremental amount from the start point to the center

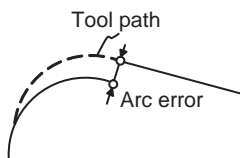
2nd block arc : Radius command incremental amount from the end point to the center



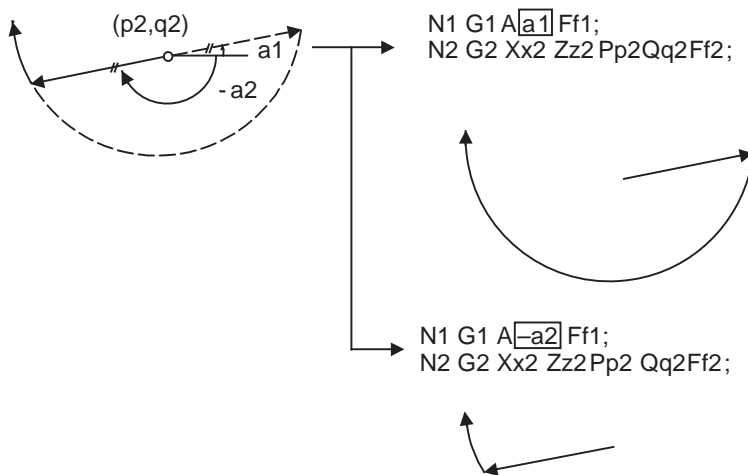


Detailed description

- (1) When the 2nd miscellaneous function address is A, the 2nd miscellaneous function is validated and this function is invalidated.
- (2) Program error (P393) will occur before the 1st block if the 2nd block is not a coordinate absolute value command.
- (3) Program error (P398) will occur before the 1st block if there is no geometric IB specification.
- (4) In case of the 2nd block arc, a program error (P395) will occur before the 1st block if there is no P, Q (I, K) designation. A program error (P395) will also occur if there is no A designation for the line.
- (5) Program error (P396) will occur before the 1st block if another plane selection command (G17 to G19) is issued in the 2nd block.
- (6) Program error (P397) will occur before the 1st block if a straight line and arc that do not contact or intersect are commanded.



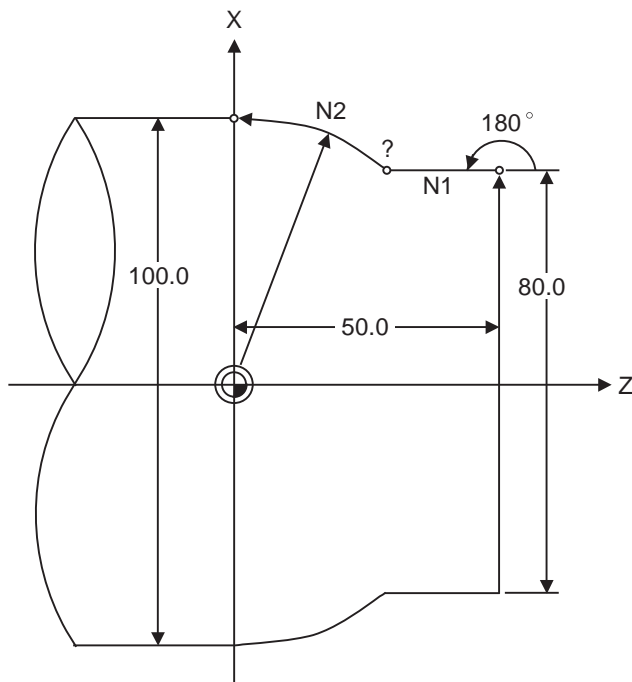
- (7) Single block operation stops at the 1st block.
- (8) When I or K is omitted, the values are regarded as I0 and K0. P and Q cannot be omitted.
- (9) When H is omitted, the value is regarded as H0.
- (10) The linear - arc contact is automatically calculated by designating R instead of P, Q (I, K).
- (11) The error range in which the intersect is obtained is set in parameter "#1084 RadErr".
- (12) As seen from the + direction of the horizontal axis of the selected plane, the counterclockwise (CCW) direction is considered to be + and the clockwise direction (CW) -.
- (13) The slope of the line can be commanded on either the start or end point side. Whether designated slope is the starting point or the end point will be automatically identified.
- (14) When the distance to the intersection from the line and arc is same (as in the figure below), the control by address H (short/long distance selection) is invalidated. In this case, the judgment is carried out based on the angle of the line.



- (15) The intersect calculation accuracy is $\pm 1 \mu\text{m}$ (fractions rounded up).
- (16) In linear - arc intersections, the arc command can only be PQ (IK) command. When the arc block start point and arc block end point are the same point, the arc is a perfect circle.
- (17) G codes of the G modal group in the 1st/2nd block can be omitted.
- (18) Addresses being used as axis names cannot be used as command addresses for angles, arc center coordinates or intersection selections.
- (19) When geometric IB is commanded, two blocks are pre-read.



Program example



```
G01 X80.0 Z50.0 F100;
N1 G01 A180.0;
N2 G03 X100.0 Z0 P0 Q0;          (mm)
```



Relation with other functions

Command	Tool path
Geometric IB + corner chamfering II N1 G01 A_ ,C_ ; N2 G03 X_ Z_ P_ Q_ H_ ;	
Geometric IB + corner rounding II N1 G01 A_ ,R_ ; N2 G03 X_ Z_ P_ Q_ H_ ;	
Geometric IB + corner chamfering II N1 G01 A_ ; N2 G03 X_ Z_ P_ Q_ H_ ; G01 X_ Z_ ;	
Geometric IB + corner rounding II N1 G01 A_ ; N2 G03 X_ Z_ P_ Q_ H_ ; G01 X_ Z_ ;	
Geometric IB + corner chamfering N1 G02 P_ Q_ H_ ; N2 G01 X_ Z_ A_ ,C_ ; G01 X_ Z_ ;	
Geometric IB + corner rounding II N1 G02 P_ Q_ H_ ; N2 G01 X_ Z_ A_ ,R_ ; G01 X_ Z_ ;	

13.14.2.3 Geometric IB (Automatic calculation of linear - arc intersection) ; G01 A_ , G02/G03 R_H_



Function and purpose

When the intersection of a shape in which a line and arc intersect is not indicated in the drawing, it can be automatically calculated by commanding the following program.

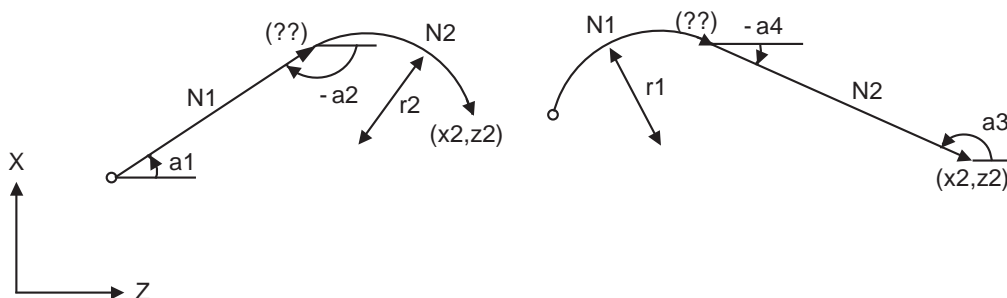


Command format (For G18 plane)

```
N1 G01 Aa1(A-a2) Ff1;
N2 G03(G02) Xx2 Zz2 Rr2 Ff2;
```

```
N1 G03(G02) Rr1 Ff1;
N2 G01 Xx2 Zz2 Aa3(A-a4)Ff2;
```

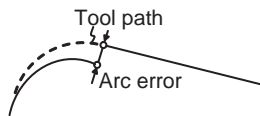
A	Linear angle (-360.000° to 360.000°)
R	Arc radius





Detailed description

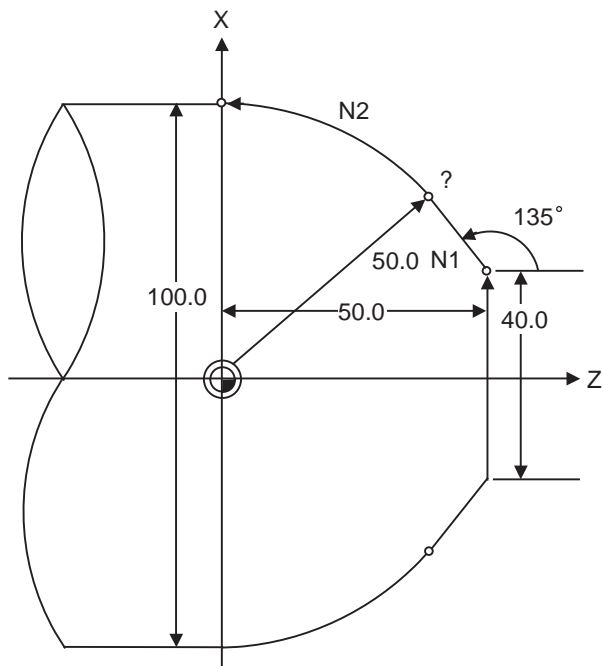
- (1) When the 2nd miscellaneous function address is A, the 2nd miscellaneous function is validated and this function is invalidated.
- (2) Program error (P393) will occur before the 1st block if the 2nd block is not a coordinate absolute value command.
- (3) Program error (P398) will occur before the 1st block if there is no geometric IB specification.
- (4) Program error (P396) will occur before the 1st block if another plane selection command (G17 to G19) is issued in the 2nd block.
- (5) A program error (P397) will occur before the 1st block if a straight line and arc that do not contact are commanded.
- (6) In case of the 2nd block arc, a program error (P395) will occur before the 1st block if there is no R designation. A program error (P395) will also occur if there is no A designation for the line.
- (7) Single block operation stops at the 1st block.
- (8) The linear - arc contact is automatically calculated by designating R instead of P, Q (I, K).



- (9) The error range in which the contact is obtained is set in parameter "#1084 RadErr".
- (10) The line slope is the angle to the positive (+) direction of its horizontal axis. Counterclockwise (CCW) is positive (+). Clockwise (CW) is negative (-).
- (11) The slope of the line can be commanded on either the start or end point side. Whether the commanded slope is on the start or end point side is identified automatically inside the NC unit.
- (12) The intersect calculation accuracy is $\pm 1 \mu\text{m}$ (fractions rounded up).
- (13) In linear - arc contact, the arc command can only be an R command. Thus, when the arc block start point = arc block end point, the arc command finishes immediately, and there will be no operation. (Perfect circle command is impossible.)
- (14) G codes of the G modal group 1 in the 1st block can be omitted.
- (15) Addresses being used as axis names cannot be used as command addresses for angles or arc radius.
- (16) When geometric IB is commanded, two blocks are pre-read.



Program example



```

G01 X40.0 Z50.0 F100;
N1 G01 A135.0;
N2 G03 X100.0 Z0.0 R50.0;

```

(mm)



Relation with other functions

Command	Tool path
Geometric IB + corner chamfering N1 G03 R_ N2 G01 X_ Z_ A_ ,C_ G01 X_ Z_ 	
Geometric IB + corner rounding N1 G03 R_ N2 G01 X_ Z_ A_ ,R_ G01 X_ Z_ 	
Geometric IB + corner chamfering II N1 G01 A_ N2 G02 X_ Z_ R_ ,C_ G01 X_ Z_ 	
Geometric IB + corner rounding II N1 G01 A_ N2 G02 X_ Z_ R_ ,R_ G01 X_ Z_ 	

13.15 Parameter Input by Program ; G10 L70/L100, G11



Function and purpose

The parameters set from the setting and display unit can be changed in the machining programs.

G10 L70 ... For commanding data with decimal point, and character string data.

The data's command range conforms to the parameter setting range described in Setup Manual.



Command format

G10 L70 ;...Data setting start command

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	Data
D	Data
character string	Data

G11 ; ... Data setting end command

- (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.
- (Note 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.
- (Note 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.
- (Note 4) Address H is commanded with the combination of setting data (0 or 1) and the bit designation □ (0 to 7).
- (Note 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.
- (Note 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.
- (Note 7) Command G10 L70, G11 in independent blocks. A program error (P33, P421) will occur if not commanded in independent blocks.
- (Note 8) The following data cannot be changed with the G10 L70 command:
Tool compensation data, workpiece coordinate data, PLC switch, and PLC axis parameter.
- (Note 9) 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.



Program example

(1)When G10 L70 command

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 ;	

13.16 Macro Interruption ; M96,M97



Function and purpose

A user macro interrupt signal (UIT) is input from the machine to interrupt the program being currently executed and instead call another program and execute it. 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 with 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 while one user macro interrupt is being in service. It is also ignored in a user macro interrupt disable state such as after an M97 command is issued or the system is reset.
- (3) M96 and M97 are processed internally as user macro interrupt control M codes.
- (4) If calling a subprogram numbered with O is enabled, a program number starting with O and specified by P command value is called with a priority.
However, when P command value is less than the digit number set with parameter "#8129 subprogram number selection", increase the digit number of command value by adding leading zeros.
(Example)
When parameter "#8129 subprogram number selection"="1", call the subprogram "O0012" with "M96 P12" command.
- (5) In the following cases, a subprogram of P command value without O No. is called even with a setting to call a subprogram with O No.
 - The digit number of P command value is over the digit number of the program number set with parameter "#8129 subprogram number selection".
 - A subprogram starting with commanded O No. does not exist.

Interrupt enable conditions

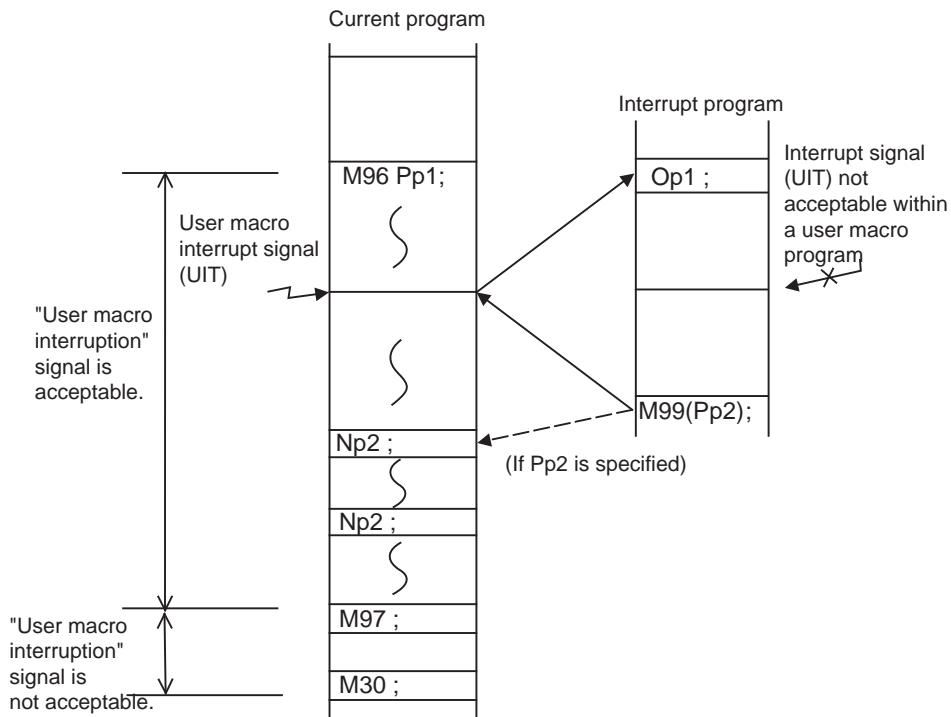
A user macro interruption is enabled only during execution of a program.
The requirements for the user macro interrupt are as follows:

- (1) An automatic operation mode or MDI has been selected.
- (2) The system is running in automatic mode.
- (3) No user macro interruption is being processed.

(Note 1) A macro interruption is disabled in manual operation mode (JOG, STEP, HANDLE, etc.)

Outline of operation

- (1) When a user macro interrupt signal (UIT) is input after an M96Pp1 ; command is issued by the current program, interrupt program Op1 is executed. When an M99; command is issued by the interrupt program, control returns to the main program.
- (2) If M99Pp2 ; is specified, the blocks from the one next to the interrupted block to the last one are searched for the block with sequence number Np2 ;. Control thus returns to the block with sequence number Np2 that is found first in the above search.



Interrupt type

Interrupt types 1 and 2 can be selected by the parameter "#1113 INT_2".

[Type 1]

- (1) When an interrupt signal (UIT) is input, the system immediately stops moving the tool and interrupts dwell, then permits the interrupt program to run.
- (2) If the interrupt program contains a move or miscellaneous function (MSTB) command, the commands in the interrupted block are lost. After the interrupt program completes, the main program resumes operation from the block next to the interrupted one.
- (3) If the interrupted program contains no move and miscellaneous (MSTB) commands, it resumes operation, after completion of the interrupt program, from the point in the block where the interrupt was caused.

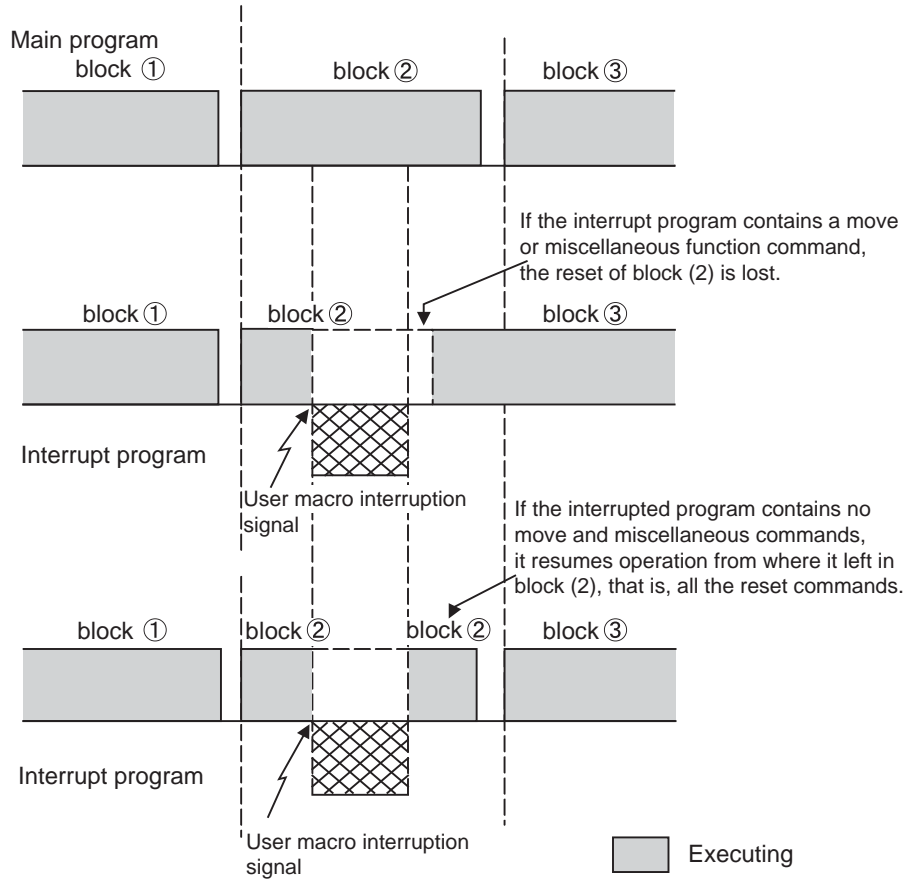
If an interrupt signal (UIT) is input during execution of a miscellaneous function (MSTB) command, the NC system waits for a completion signal (FIN). The system thus executes a move or miscellaneous function command (MSTB) in the interrupt program only after input of FIN.

[Type 2]

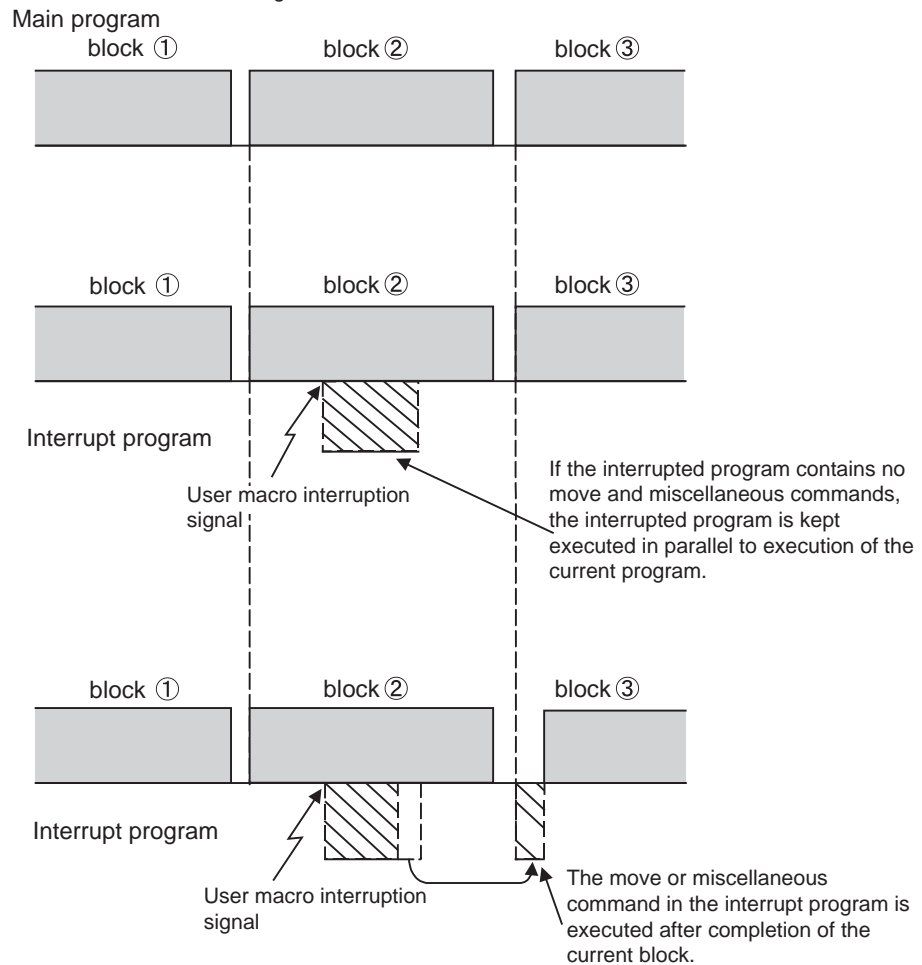
- (1) When an interrupt signal (UIT) is input, the interrupt program will be executed without interrupting execution of the current block in parallel with the executing block.
- (2) If the interrupt program contains a move or miscellaneous function (MSTB) command, the commands in the interrupted block are completed, then, these commands will be executed.
- (3) If the interrupt program contains no move and miscellaneous function (MSTB) commands, the interrupt program is executed without interrupting execution of the current block.

However, if the interrupt program has not ended even after the execution of the original block is completed, the system may stop machining temporarily.

[Type 1]



[Type 2]



Calling method

User macro interruption is classified into the following two types depending on the way an interrupt program is called. These two types of interrupt are selected by parameter "#8155 Sub-pro interrupt".

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.

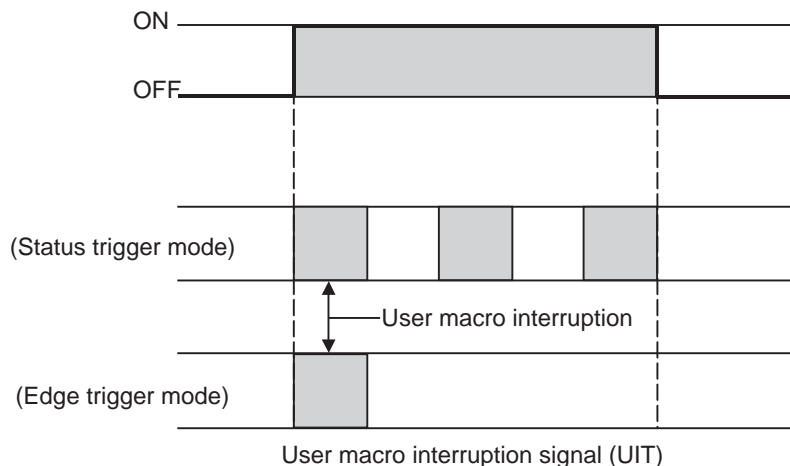
Subprogram type interrupt	The user macro interruption program is called as a subprogram. As with calling by M98, the local variable level remains unchanged before and after an interrupt.
Macro type interruption	The user macro interpretation program is called as a user macro. As with calling by G65, the 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 modes: These two modes are selected by a parameter "#1112 S_TRG".

Status trigger mode	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 mode	The user macro interrupt signal (UIT) is accepted as valid at its rising edge, that is, at the instance it turns ON. This mode is useful to execute an interrupt program once.

User macro interruption signal (UIT)



Returning from user macro interruption

M99 (P__);

An M99 command is issued in the interrupt program to return to the main program.

Address P is used to specify the sequence number of the return destination in the main program.

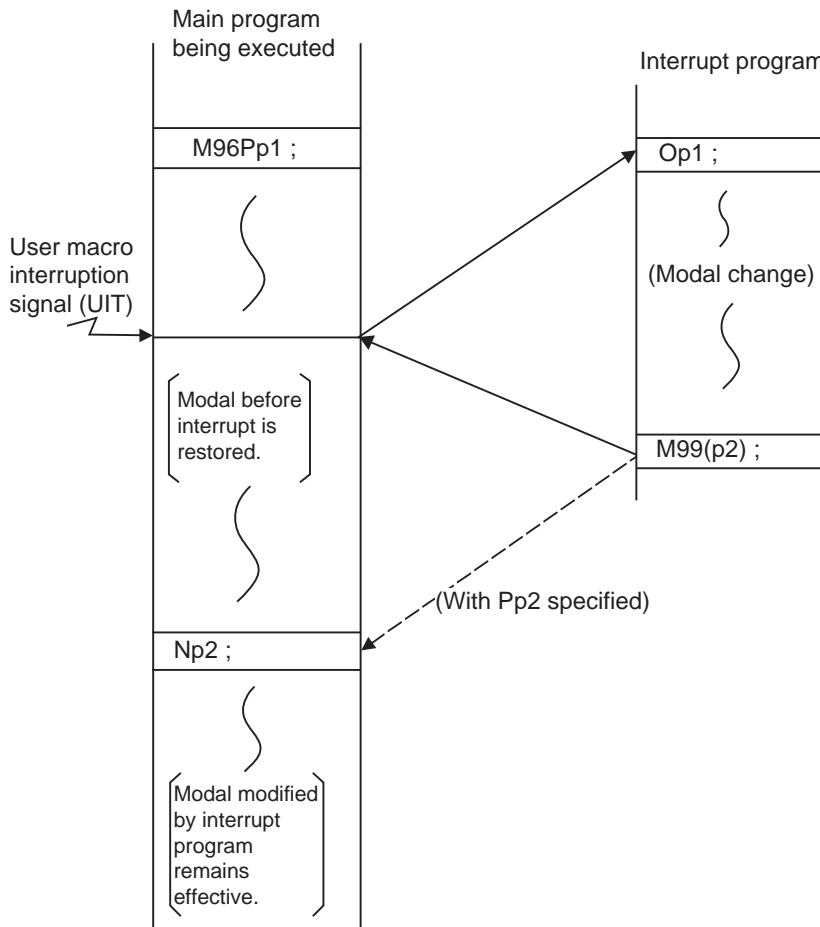
The blocks from the one next to the interrupted block to the last one in the main program are first searched for the block with designated sequence No. If it is not found, all the blocks before the interrupted one are then searched. Control thus returns to the block with sequence No. that is found first in the above search.

(This is equivalent to M99P__ used after M98 calling.)

Modal information affected by user macro interruption

If modal information is changed by the interrupt program, it is handled as follows after control returns from the interrupt program to the main program.

Returning with M99;	The change of modal information by the interrupt program is invalidated and the original modal information is restored. With interrupt type 1, however, if the interrupt program contains a move 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

Modal information variables (#4401 to #4520)

Modal information when control passes to the user macro interruption program can be known by reading system variables #4401 to #4520.

The unit specified with a command applies.

System variable	Modal information	
#4401 : #4421	G code (group01) : G code (group21)	Some groups are not used.
#4507	D code	
#4509	F code	
#4511	H code	
#4513	M code	
#4514	Sequence No.	
#4515	Program No. (Note 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.

(Note 1) The 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".

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 operation. To be prepared for such case, these command functions can be assigned to other M codes.

(This invalidates program compatibility.)

User macro interrupt control with alternate M codes is possible by setting the alternate M code in parameters "#1110 M96_M" and "#1111 M97_M" and by validating the setting by selecting parameter "#1109 subs_M".

(M codes 03 to 97 except 30 are available for this purpose.)

If the parameter "#1109 subs_M" used to enable the alternate 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"
 - 1: Subprogram type user macro interruption
 - 0: Macro type user macro interruption
- (2) Status trigger mode validity "#1112 S_TRG"
 - 1: Status trigger mode
 - 0: Edge trigger mode
- (3) Interrupt type 2 validity "#1113 INT_2"
 - 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 alternate M code for user macro interruption control "#1109 subs_M"
 - 1: Valid
 - 0: Invalid
- (5) Alternate M codes for user macro interruption

Interrupt enable M code (equivalent to M96) "#1110 M96_M"

Interrupt disable M code (equivalent to M97) "#1111 M97_M"
- (6) Subprogram number selection "#8129 subprogram number selection"

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.



Precautions

- (1) If the user macro interruption program uses system variables #5001 and after (position information) to read coordinates, the coordinates pre-read in the buffer are used.
- (2) If an interrupt is caused during execution of the nose R compensation, a sequence No. (M99P__;) must be specified with a command to return from the user macro interrupt program. If no sequence No. is specified, control cannot return to the main program normally.

13.17 Tool Change Position Return ; G30.1 - G30.5



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

G30.n ; ... Tool change position return

n = 1 to 5: 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.

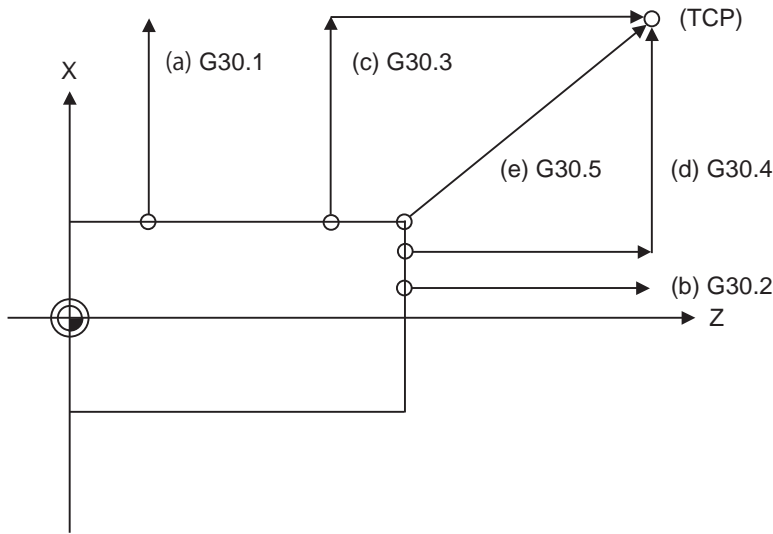
Command	Return order
G30.1	X axis only (-> additional axis)
G30.2	Z axis only (-> additional axis)
G30.3	X axis -> Z axis (-> additional axis)
G30.4	Z axis -> X axis (-> additional axis)
G30.5	X axis - Z axis (-> additional axis)

(Note 1) An arrow (->) indicates the order of axes that begin to return. An period (-) indicates that the axes begin to return simultaneously. (Example: "Z axis -> X axis" indicates that the Z axis returns to the tool change position, then the X axis does.)

- (1) The tool change position return on/off for the additional axis can be set with parameter "#1092 Tchg_A" for the additional axis.
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). For specifications having two additional axes, the two additional axes simultaneously return to the tool change position after the standard axis has finished its return to the tool change position.
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.



Operation example

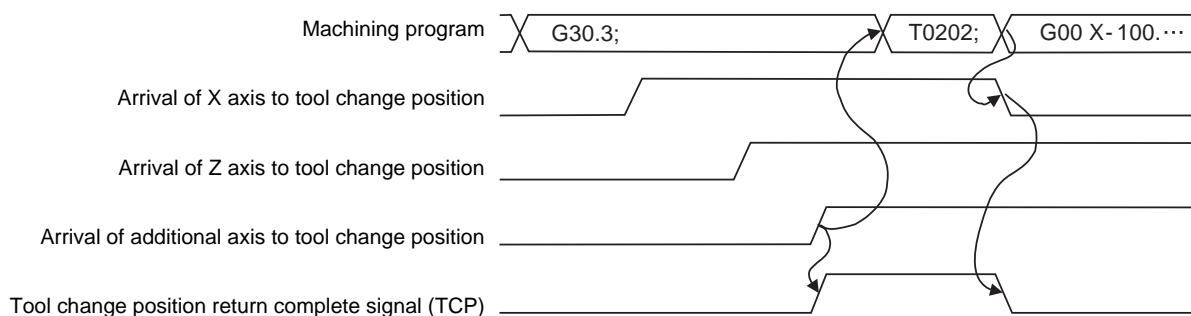


TCP : Tool change position

- (1) G30.1 command: X axis only returns to the tool change position. (If the tool change position return is validated for the additional axis, the additional axis also returns to the tool change position after the X axis reaches the tool change position.)
- (2) G30.2 command: Z axis only returns to the tool change position. (If the tool change position return is validated for the additional axis, the additional axis also returns to the tool change position after the Z axis reaches the tool change position.)
- (3) G30.3 command : X axis returns to the tool change position, then Z axis does the same thing. (If the tool change position return is validated for the additional axis, the additional axis also returns to the tool change position after the X and Z axes reach the tool change position.)
- (4) G30.4 command : Z axis returns to the tool change position, then X axis does the same thing. (If the tool change position return is validated for the additional axis, the additional axis also returns to the tool change position after the Z and X axes reach the tool change position.)
- (5) G30.5 command : X and Z axes return to the tool change position simultaneously. (If the tool change position return is validated for the additional axis, the additional axis also returns to the tool change position after the Z and X axes reach the tool change position.)

- (6) 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 G30.3 command, for example, the TCP signal is turned ON when the Z axis has reached the tool change position after the X axis did (after the additional axis did if additional axis tool change position return is valid)). The TCP signal is then turned OFF when the X or Z axis leaves the position. If tool change position return for additional axes is ON with parameter "#1092 Tchg_A", the TCP signal is turned ON when the additional axis or axes have reached the tool change position after the standard axes did. It is then turned OFF when one of the X, Z, and additional axes leaves the position.)

[TCP signal output timing chart] (G30.3 command with tool change position return for additional axes set ON)



- (7) Tool compensation data such as tool length offset and tool nose wear compensation are temporarily canceled by the tool change position return command. The machine moves to the tool change position set in the parameters, but because the tool compensation amount is stored in the memory, it moves by the next movement command to a position with the tool compensation applied.
- (8) 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.

13.18 Coordinate Rotation by Program ; G68.1/G69.1



Function and purpose

When machining a complicated shape located in a rotated position in respect to the coordinate system, this function enables to machine the rotated shape with the program for the shape before rotation on the local coordinate system and with the rotation angle designated by the program coordinate rotation command.



Command format

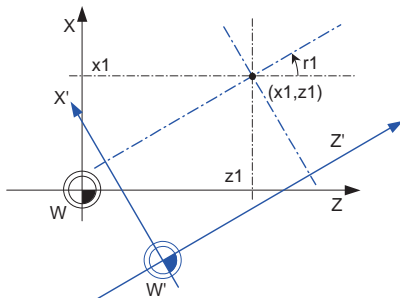
G68.1 X__ Z__ R__ ; ... Coordinate rotation ON

X,Z	Rotation center coordinates An axis corresponding to the plane selected from the rotation center coordinates X, Y and Z.
R	Rotation angle Designate the angle from -360° to 360° by the minimum setting unit. The counterclockwise direction on the selected plane is + direction.

G69.1 ; ... Coordinate rotation cancel

Select the command plane with G17 to G19.

Command) G68.1 Xx1 Zz1 Rr1 ;



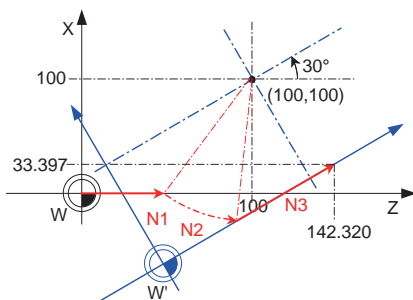
- (1) Command the rotation center coordinates (x1, z1) with an absolute value.
- (2) Rotate the coordinate counterclockwise by the angle designated in rotation angle r1.
- (3) When the minimum setting unit for r1 is 0.001deg, the setting range of rotation angle is -360.000 to 360.000. The setting range of rotation angle or r1 when the minimum setting unit is 0.001 deg is -360.000 to 360.000. When a value out of the range is commanded, a remainder of the value divided by 360° is commanded.
(Ex.) When 400 is commanded:
400 divided by 360 is 1, and the remainder is 40. Thus, 40 is the commanded angle.
- (4) The counter is indicated as the point on the coordinate system prior to rotation.

W: Local coordinate system before rotation
r1: Rotation angle

W': Local coordinate system after the rotation
(x1, z1) Rotation center

The following is the example of relationship of program command position and the displayed position.

Program example) N1 G00 Z50.
N2 G68.1 X100. Z100. R30.;
N3 G00 Z120. ;



- (1) The program command performs positioning on the local coordinates after the rotation.
- (2) The counter display shows the point after the coordinate rotation on the coordinate system before rotation. In this example, the position display when the N3 block is finished is:
X 33.397
Z 142.320
- (3) G68.1 command does not carry out the actual movement. Therefore, in this example, it moves linearly from the end point of N1 to the end point of N3.

W: Local coordinate system before rotation

W': Local coordinate system after rotation



Detailed description

- (1) G68.1 and G69.1 are the G code of group 16.
- (2) Command the rotation center coordinate (x1, z1) with an absolute value. Even if commanded with an incremental address, it will not be handled as an incremental value.
- (3) If the rotation center coordinates (x1, z1) are omitted, the position where the G68 command was executed will be the rotation center.
- (4) The rotation angle R is commanded with an absolute value. However, it can be commanded with an incremental value if the parameter "#8082 G68.1 R INC" is set.
- (5) The performance when the rotation angle R is omitted depends on the setting of parameter "#1270 ext06/bit5":

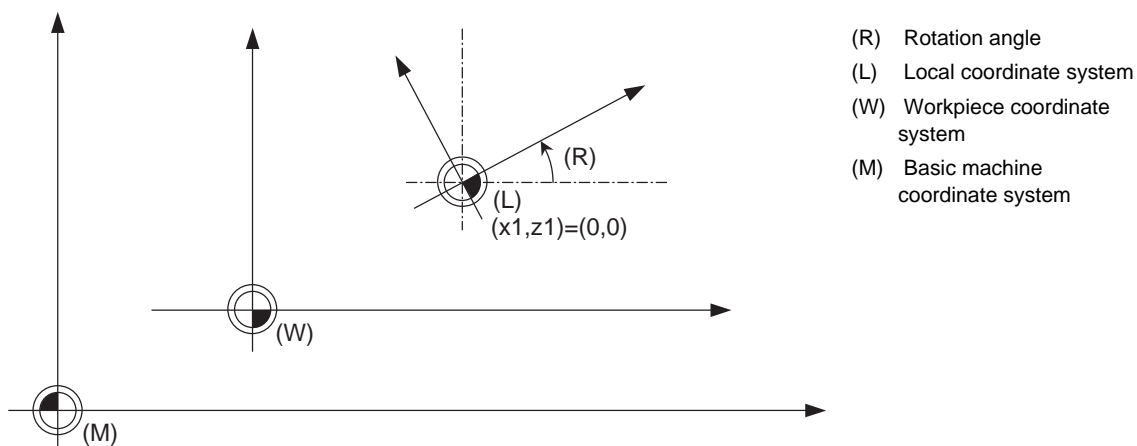
0: Use the previously commanded value (modal value).

1: Use the set value in "#8081 Gcode Rotat".

If the coordinate rotation mode is canceled by G69.1 command, the modal value will be cleared. If G68.1 is commanded after G69.1 was commanded, the rotation angle becomes 0° by omitting R.

The setting value of the parameter is an absolute value regardless of the setting of the parameter "#8082 G68.1 R INC".

- (6) The program coordinate rotation is a function used on the local coordinate system. The relation of the rotated coordinate system, workpiece coordinate system and basic machine coordinate system is shown below.



- (7) The coordinate rotation command during coordinate rotation is processed as the changes of center coordinates and rotation angle.
- (8) If commanding G68.1 without the coordinate rotation specification, a program error (P260) will occur. If commanding G69.1 without the coordinate rotation specification, a program error (P39) will occur.
- (9) If commanding the plane selection code during the coordinate rotation mode, a program error (P111) will occur.
- (10) The program coordinate rotation function is valid only in the automatic operation mode.
- (11) G68.1 is displayed on the modal information screen during the coordinate rotation mode. When the mode is canceled, the display changes to G69.1. (The modal value is not displayed for the rotation angle command R.)

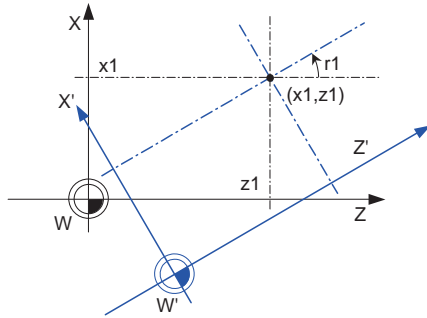
Coordinate rotation command during coordinate rotation

The coordinate rotation command during coordinate rotation is processed as the changes of center coordinates and rotation angle.

(1) For absolute command

Command)
 G68.1 Xx1 Zz1 Rr1 ;
 G68.1 Xx2 Zz2 Rr2 ;

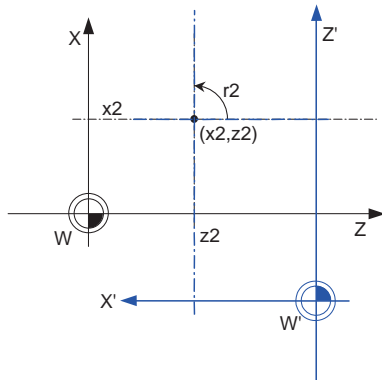
1) G68.1 Xx1 Zz1 Rr1 ;



With spinning around on the center coordinate of the rotation (x1, z1), the rotation takes place in the counterclockwise direction by the angle designated in rotation angle r1.

W: Local coordinate system before the rotation
 W': Local coordinate system after the rotation
 r1: Rotation angle
 (x1, z1) Rotation center

2) G68.1 Xx2 Zz2 Rr2 ;



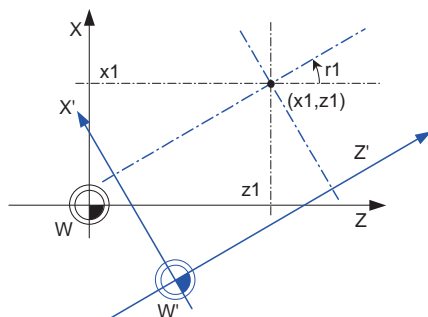
The center coordinate of the rotation switches from (x1, z1) to (x2, z2), and the rotation angle is cleared once. Then the rotation takes place in the counterclockwise direction by the angle designated with r2

W: Local coordinate system before the rotation
 W': Local coordinate system after the rotation
 r2: Rotation angle
 (x2, z2) Rotation center

(2) For incremental command

Command)
G68.1 Xx1 Zz1 Rr1 ;
G68.1 Ux2 Uz2 Rr2 ;

1) G68.1 Xx1 Zz1 Rr1 ;



With spinning around on the center coordinate of the rotation (x_1, z_1), the rotation takes place in the counterclockwise direction by the angle designated in rotation angle r_1 .

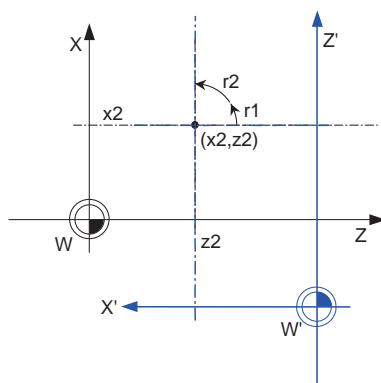
W: Local coordinate system before the rotation

W': Local coordinate system after the rotation

r_1 : Rotation angle

(x_1, z_1) Rotation center

2) G68.1 Ux2 Uz2 Rr2 ;



The center coordinate of the rotation switches from (x_1, z_1) to (x_2, z_2) . Even if the rotation center coordinate command is the incremental value command, it is handled as the absolute value.

The rotation takes place in the counterclockwise direction by the angle rotated at r_1 and another angle commanded at r_2 .

W: Local coordinate system before the rotation

W': Local coordinate system after the rotation

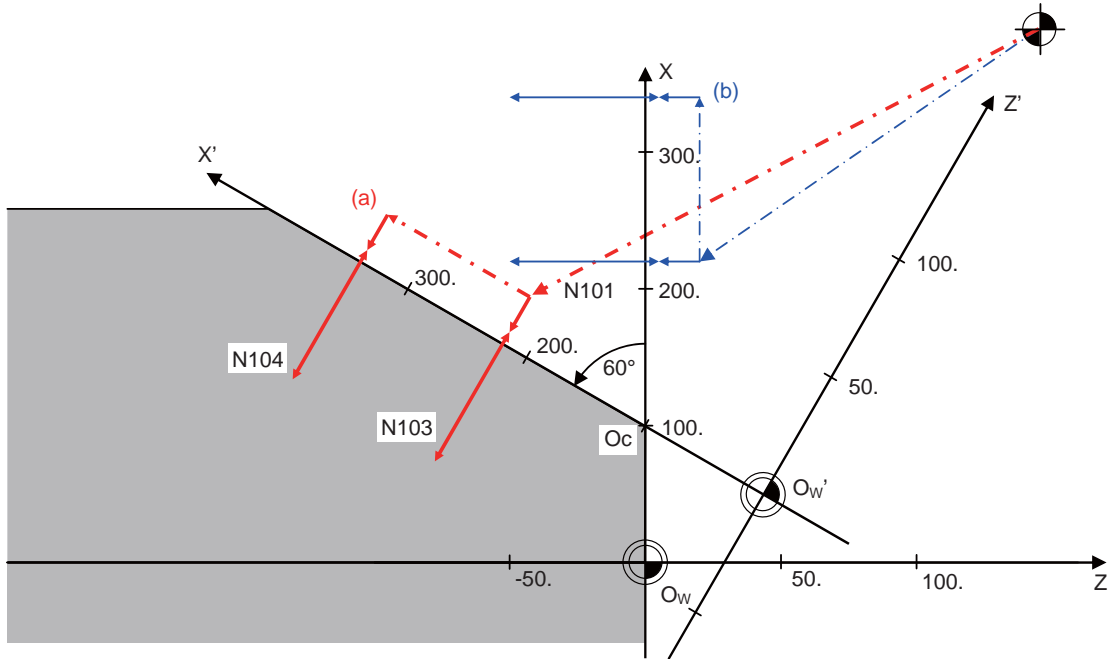
r_2 : Rotation angle

(x_2, z_2) Rotation center



Program example

Program coordinate rotation by absolute command



- (Oc) Rotation center
- (Ow) Workpiece coordinate zero point before rotation
- (Ow') Workpiece coordinate zero point after rotation
- (a) Subprogram path after rotation
- (b) Subprogram path before rotation

Main program

```

N01 G97 G18;           Z-X Plane selection
N02 G91 G28 X0. Z0.;
N03 G54;
N04 G90 T1010;
N05 G68.1 X100. Z0 R60.;   Coordinate rotation ON
N06 M98 H101;           Subprogram execution
N07 G69.1;             Coordinate rotation cancel
N08 M02 ;              End
    
```

Subprogram (Shape programmed with coordinate system before rotation)

```

N101 G00 X220. Z20.;
N102 G94 S2=1000 M3;       2nd spindle (tool spindle) forward
N103 G98 G83 Z-50. R-15. Q-10. F100;
N104 X340. ;
N105 G80;
N106 S2=0 M5;             2nd spindle stop
N107 M99;
    
```

Operation when only one axis was commanded by the first movement command

Command basically two axes in the rotation plane by an absolute value immediately after the coordinate rotation command.

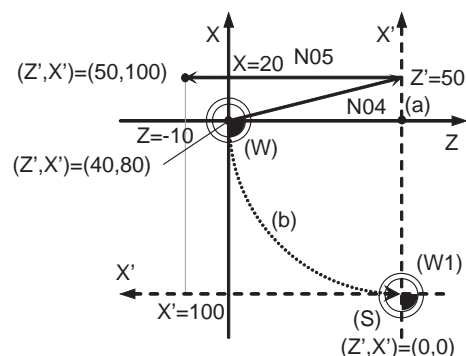
When commanding one axis only, the following two kinds of operations can be selected by the parameter "#19003 PRG coord rot type".

- (1) When "#19003 PRG coord rot type" is "1", the operation is the same as when "N04" is "X50.Z0.". The end point is calculated on the assumption that the start point rotates along with the coordinates' rotation.

```

N01 G18 G91 G28 X0.
Z0.;
N02 G90 G92 G53 X0.
Z0.;
N03 G68.1 X0. Z40. R90.; Coordinate rotation ON
N04 Z50.;
N05 X100.;
N06 G69.1;           Coordinate rotation cancel
N07 M02 ;           End

```



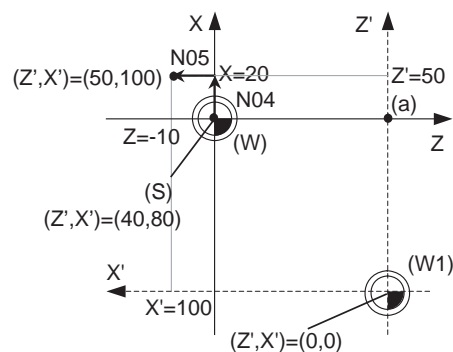
- Machine movement path
 (a) Rotation center
 (W) Local coordinate system before rotation
 (S) Start point
 (b) The start point is rotated virtually
 (W1) Local coordinate system after rotation

- (2) When "#19003 PRG coord rot type" is "0", only axis commanded in N04 (Z' Axis) is moved. The start point does not rotate along with the coordinate rotation; therefore the end position is calculated based on the current position on local coordinate system before rotation.

```

N01 G18 G91 G28 X0. Z0.;
N02 G90 G92 G53 G0 X0. Z0.;
N03 G68.1 X0. Z40. R90.; Coordinate rotation ON
N04 Z50.;
N05 X100.;
N06 G69.1;           Coordinate rotation cancel
N07 M02;           End

```



- Machine movement path
 (a) Rotation center
 (W) Local coordinate system before rotation
 (S) Start point
 (W1) Local coordinate system after rotation

Local coordinate setting during program coordinate rotation

- (1) When "#19003 PRG coord rot type" is "0", the position commanded on the rotated coordinate system is set as the local coordinate zero point.
- (2) When "#19003 PRG coord rot type" is "1", the position commanded on the coordinate system before it is rotated, is set as the local coordinate zero point and the local coordinate will be rotated.

```

N01 G18 G91 G28 X0. Z0.;
N02 G90 G92 G53 G0 X0. Z0.;
N03 G68.1 X0. Z20. R90.;      Coordinate rotation ON
N04 G52 X20. Z10.;           Local coordinate designation
N05 Z20.;
N06 X20.;
N07 G69.1;                   Coordinate rotation cancel
N08 M02 ;                     End
    
```

W: Workpiece coordinate system
L: Local coordinate system

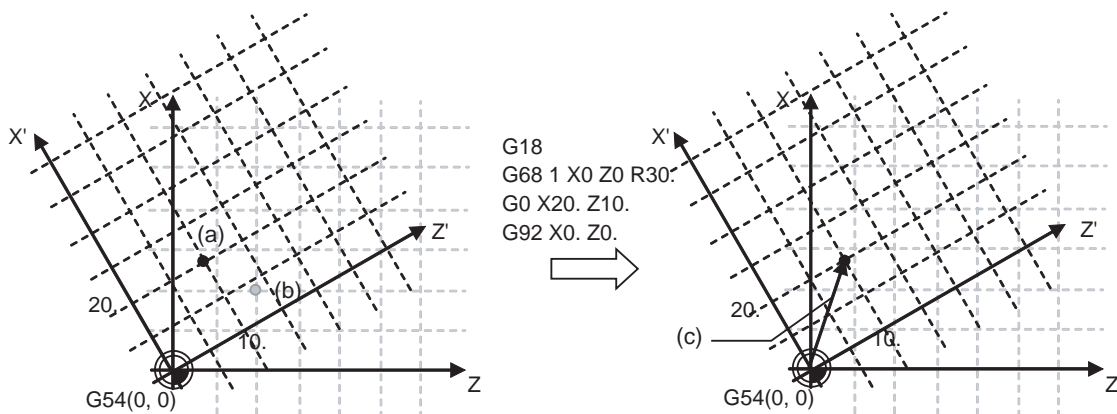
	(1) Operation of #19003 = 0	(2) Operation of #19003 = 1
N03	<p>(Workpiece coordinate system is rotated virtually.)</p>	
	Workpiece coordinate system is rotated virtually.	Workpiece coordinate system is not rotated.
N04	<p>(Local coordinate designation)</p>	<p>(Local coordinate designation)</p>
	The workpiece coordinate zero point after rotation is considered as (Z,X)=(0,0). The position after shifted by 10 in the X and 20 in the Y direction is set as the local coordinate zero point. * The direction of the shift is not the direction of Z' and X'.	Designate the local coordinate system on the workpiece coordinate system.
N05	<p>(Z'', X'')=(20,60) Start point: (Z'', X'')=(10,60)</p>	<p>(Z'', X'')=(20,60) Start point: (Z'', X'')=(-10,-20)</p> <p>(The start point is rotated virtually.)</p>
	The commanded axis moves on the rotation coordinate system. * Axis without movement command does not move.	The commanded axis moves on the rotation coordinate system. * Axis without movement command moves to the position on rotation coordinate system.
N06	<p>(Z'', X'')=(20,60)</p>	<p>(Z'', X'')=(20,20) Start point: (Z'', X'')=(20,60)</p>

Coordinate system designation during program coordinate rotation

When the coordinate system setting (G92) is executed during program coordinate rotation (G68.1), this program operates same as "Local coordinate designation during program coordinate rotation".

- (1) When "#19003 PRG coord rot type" is "0", the position is preset to the current position commanded on the rotated coordinate system.

(Ex.) Setting on the coordinate system (Z' -X') after rotation



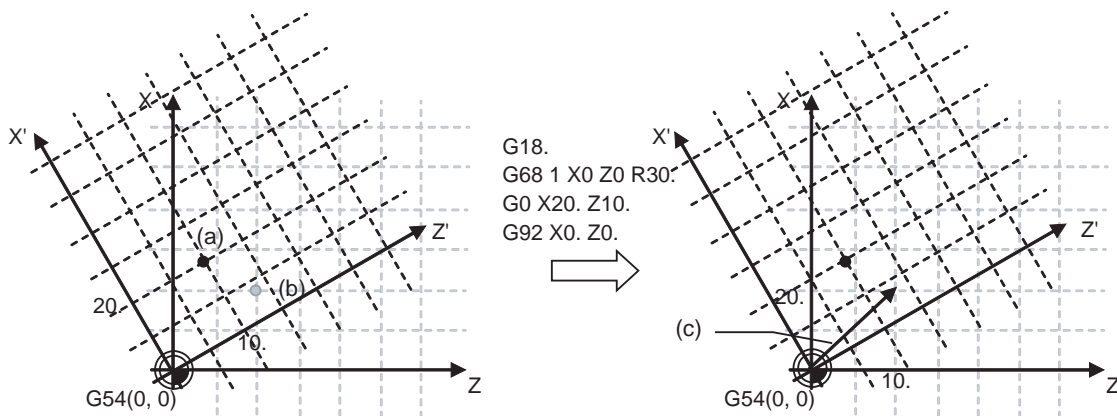
(a) Position after rotation

(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.

(Ex.) Setting on the coordinate system (Z-X) 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 mode, the rotation center of program coordinate rotation is not shifted. (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 value 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.

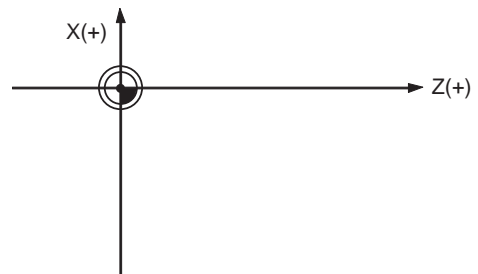
When the circular interpolation is commanded straight after the coordinate rotation command, the following operation takes place depending on the setting of the parameter "#19003 PRG coord rot type".

- (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.

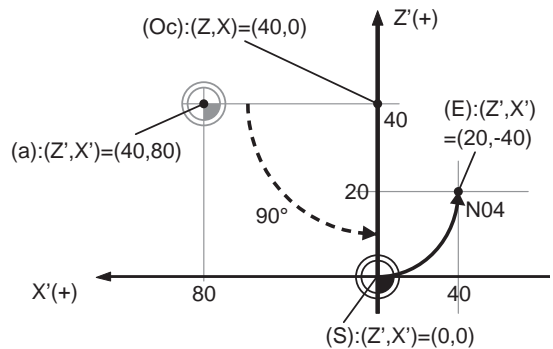
```

N01 G18 G91 G28 X0. Z0.;
N02 G90 G92 G53 G0 X0. Z0.;
N03 G68.1 X0. Z40. R90.;      Coordinate rotation ON
N04 G03 X-40. Z20. R20. F500;
N05 G00 X50.;
N06 G69.1;                    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 which was assumed to be rotated with the coordinate rotation
- (E) End point

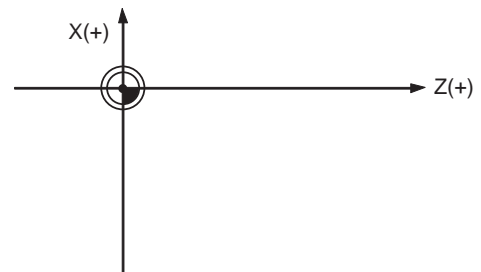
- (2) When "#19003 PRG coord rot type" is "0", the circular interpolation start point does not rotate with the coordinate rotation. It remains in the same position before the coordinate rotation in respect to the basic machine coordinate system. Consequently, the axis operates the circular interpolation from this start point to the end point.

```

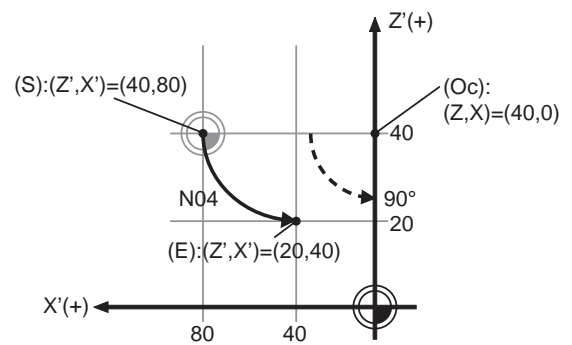
N 01 G18 G91 G28 X0. Z0.;
N 02 G90 G92 G53 G0 X0. Z0.;
N 03 G68.1 X0. Z40. R90.;   Coordinate rotation ON
N 04 G03 X40. Z20. R20. F500;
N 05 G00 X50.;
N06 G69.1;                 Coordinate rotation cancel
N07 M02 ;                  End

```

[Local coordinate system before rotation]



[Local coordinate system after rotation]



- (a) Actual axis position
- (Oc) Rotation center
- (S) Start point
- (E) End point



Relation with other functions

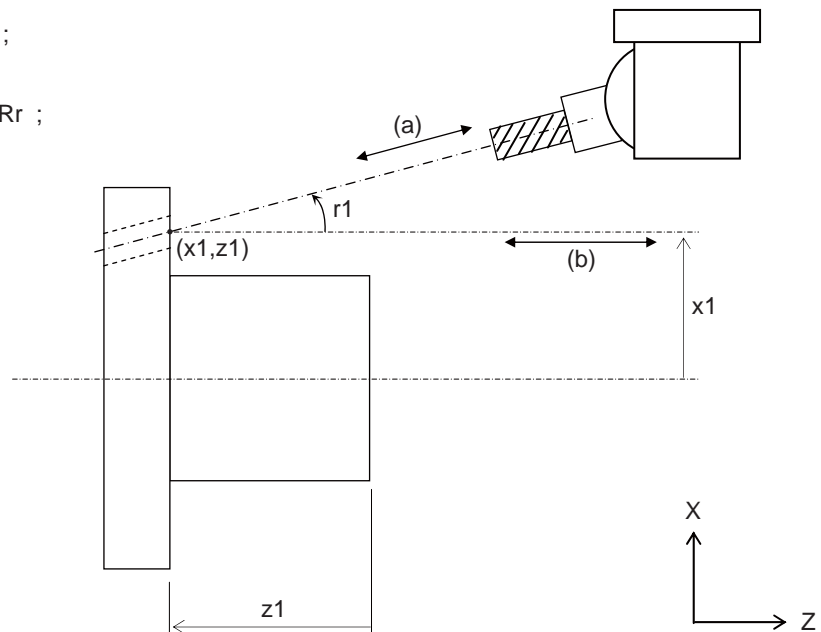
- (1) The tool compensation during the coordinate rotation mode is carried out in the local coordinate system after the coordinate rotation.
- (2) During the coordinate rotation mode, the mirror image performs reverse processing of the point after the coordinate rotation is inverted in the coordinate system before the coordinate rotation.
- (3) For all position display, points after the coordinate rotation are displayed in the coordinate system before the rotation.
- (4) Similarly for the system variable of the coordinate value, points after the coordinate rotation are displayed in the coordinate system before the rotation.
- (5) The coordinates can also be rotated for the parallel axis. Select the plane that contains the parallel axis before issuing the G68.1 command. (The plane which contains the parallel axis cannot be selected in the same block as the G68.1 command.)
- (6) Program error (P485) will occur if pole coordinate interpolation/milling interpolation is commanded during the coordinate rotation mode.
- (7) Program error (P481) will occur if the coordinate rotation is commanded during the polar interpolation/milling interpolation mode.
- (8) Program error (P485) will occur if the cylindrical interpolation is commanded during the coordinate rotation mode.
- (9) Program error (P481) will occur if the coordinate rotation mode is commanded during the cylindrical interpolation mode.
- (10) Program error (P34) will occur if the workpiece coordinate system preset (G92.1) is commanded during the coordinate rotation mode.
- (11) A system variable can read the skip coordinate value when the skip command is issued during the coordinate rotation. The system variable reads the workpiece coordinate value after the coordinate conversion.
When the multiple axes move for one axis movement command, the skip coordinate value is read by the multiple axes.
- (12) Program error (P261) will occur if the coordinate rotation and other G code are commanded in the same block. However, the type of program error differs depending on the combination of the coordinate rotation command and the G code listed below.

G code commanded with the coordinate rotation command in the same block	
- Control axis synchronization across part systems start/end - Control axis superimposition start/end	Program error (P33)
- Mixed control (Cross axis control)	Program error (P503)

- (13) Program error will occur if the following functions are commanded during coordinate rotation :
 - Mixed control (Cross axis control)
 - Axis name switch
 - Control axis synchronization across part systems
 - Control axis superimposition start/end
 - Thread cutting
 - Variable lead thread cutting
 - Circular thread cutting
 - Fixed cycles for turning machining
 - Compound type fixed cycle for turning machining
 - 2-part system simultaneous thread cutting cycle
 - User macro modal call B
 - The facing turret mirror image
 - Balance cut

- (14) Program error (P111) will occur if the following functions are commanded during the coordinate rotation:
- Plane selection (X-Y, Z-X, Y-Z)
 - Milling interpolation plane selection Y-Z cylindrical plane
- (15) Program error (P262) will occur if the coordinate rotation is commanded during the following G code modal:
- Thread cutting
 - Variable lead thread cutting
 - Circular thread cutting
 - Fixed cycles for turning machining
 - Compound type fixed cycle for turning machining
 - 2-part system simultaneous thread cutting cycle
 - Fixed cycle for drilling
 - User macro
 - User macro modal call A, B
 - The facing turret mirror image
 - Balance cut
- (16) Tapping cycle can be carried out during the coordinate rotation mode. This enables the tapping diagonally.
- Refer to "Face tapping cycle (longitudinal tapping cycle);G84(G88)" for details on tapping cycle.

```
G68.1 Xx1 Zz1 Rr1 ;
.
.
G84 Zz2 Rr2 Ff Ss ,Rr ;
.
.
```



- (a): Actual movement direction
 (b): Movement direction with program command
 r1: Rotation angle
 (x1, z1): Rotation center

- (16-1) Feedrate/pitch command (F command)
 - F command value specified with the machining program is as follows:
 - Asynchronous tap: Feedrate toward the tap cutting direction (diagonally)
 - Synchronous tap: Pitch toward the tap cutting direction (diagonally)
- (16-2) Programmable in-position check
 - In-position check takes place on two axes as two axes move during tapping diagonally.
 - Each axis is checked, and the in-position check is complete when both of two axes come into the commanded in-position width.
- (16-3) Tapping return
 - Tapping return can be carried out by the tapping return signal (1st part system: YC5C, 2nd part system: YD9C) for diagonal cutting.
 - Tap cutting axes (two axes moved during tap cutting) move toward the initial point.
 - For multiple spindle control II ("#1300 ext36/bit0" is set to "1"), select the spindle when the tapping cycle is interrupted before turning ON the tapping return signal.
 - The error "M01 operation error 1032" will occur if the tapping return is carried out while a different spindle is being selected.
- (16-4) Servo gain during the synchronous tapping cycle
 - When diagonal synchronous tapping is carried out, the setting value of "#2017 tap_g" is the servo gain of two axes which move during tap cutting.
- (16-5) Specify boring axis (#1080 Dril_Z)
 - When "#1080 Dril_Z" is set to "1", the tapping cycle during coordinate rotation mode can be carried out only when G17 plane is being selected.
 - Program error (P111) will occur if the tapping cycle is commanded while G18 or G19 is being selected.
 - (Note) "#1080 Dril_Z" is valid when MITSUBISHI CNC special format ("#1265 ext01/bit2" is set to "1") is applied.
- (16-6) Precautions for synchronous tapping cycle
 - Set the same value for the servo gain ("#2017 tap_g") of two axes which move during tap cutting.
 - Do not issue a command which moves the inclined axis during tap cutting.
- (17) Program error (P612) will occur when the exponential interpolation is commanded during program coordinate rotation.



Precautions

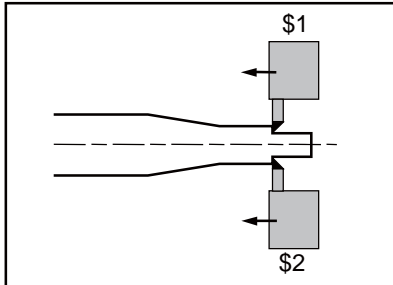
- (1) Command G68.1 in independent blocks. If not, a program error (P261) will occur.
- (2) If an axis which is not in the selected plane is commanded to the coordinate value of the rotation center, axes in other than the selected plane moves to the position which is specified by the last G1 modal.
- (3) Always command an absolute value for the movement command immediately after G68.1 and G69.1. If an increment value is commanded, it may not move to the intended position.
Also command it together with the axis address on the selected plane (for G18 plane, Z-X). If it is omitted, that axis is handled as "no movement command".
- (4) If the manual absolute is ON and interrupted the coordinate rotation axis, then, do not use automatic operation for the following absolute value command.
- (5) The intermediate point during reference point return is the position after the coordinates are rotated.
- (6) If the workpiece coordinate system offset amount is changed during the coordinate rotation mode, the rotation center for the program coordinate rotation will be shifted. (The center will follow the coordinate system.)
- (7) If workpiece coordinate system setting (G92) is executed during the coordinate rotation mode, the rotation center for program coordinate rotation will not shift. (The center position will be the same looking from the basic machine coordinate system.)
- (8) If the workpiece coordinates are changed during the coordinate rotation mode (ex. from G54 to G55), the rotation center of the program coordinate rotation will be the position on the coordinate system which the command was issued. (The same position in respect to the basic machine coordinate system)
- (9) If coordinate rotation is executed to the G00 command for only one axis during the coordinate rotation mode, two axes will move. If the parameter "#1086 G0Intp" is set to "1", the interpolation is carried out.
- (10) If a synchronous tapping is commanded during the coordinate rotation mode in inclined axis control, a normal synchronous tapping is applied even the high-speed synchronous tapping option is validated.
- (11) During the coordinate rotation by program and machine lock on a axis of vertical/horizontal axes of selected plane, normal tapping is set even high-speed synchronous tapping option is enabled.

13.19 Balance Cut ; G15,G14



Function and purpose

The timing for starting the operation of the 1st part system turret and 2nd part system turret can be synchronized.



When machining a relatively thin and long workpiece with a lathe, the workpiece could slack, and highly accurate machining may not be possible.

In this case, if the cutters are applied simultaneously from both sides of the workpiece and the workpiece is machined while synchronizing these (balance cut), the slack can be suppressed. Furthermore, the machining time can be shortened by machining with two cutters.

With this function, the movement of two turrets belonging to different part systems can be completely synchronized, so the following type of machining can be done easily.



Command format

G15 ; ... Balance cut command ON

G14 ; ... Balance cut command OFF



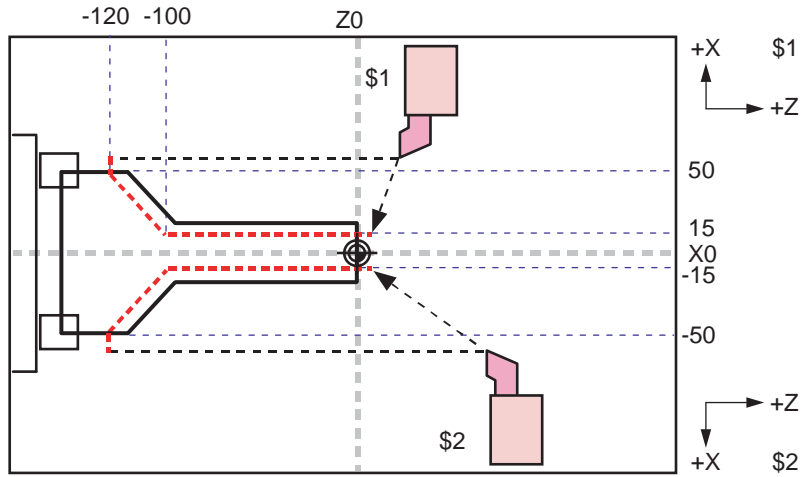
Detailed description

- (1) Command G15 and G14 in independent blocks.
- (2) G15 and G14 commands are modals. In the CNC's initial state, the G14 balance cut command is OFF.
- (3) When G15 is commanded, movement will standby at all cutting feed command blocks until G14 is commanded or until the modal information is cleared by the reset signal.
- (4) If G15 or G14 is commanded in one part system, movement will not advance until the same G code is commanded in the other part system.
- (5) When subprogram call, macro call or PLC interrupt is carried out, the cutting feed command in a block that configures the subprogram is handled as one block and will be waited.
- (6) After G14 is commanded in both part systems, 1st part system and 2nd part system will operate independently.



Program example

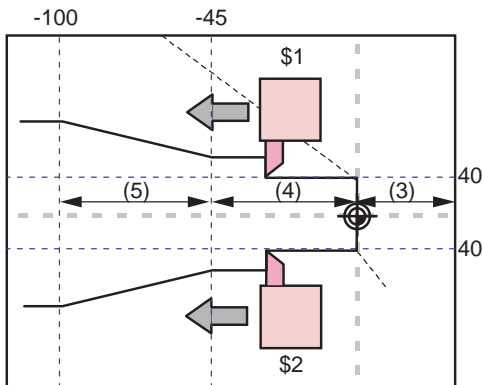
An example of a program for machining with a 1-spindle 2-turret CNC lathe while simultaneously applying the cutters from the top and bottom of a thin long workpiece using balance cut is shown below.



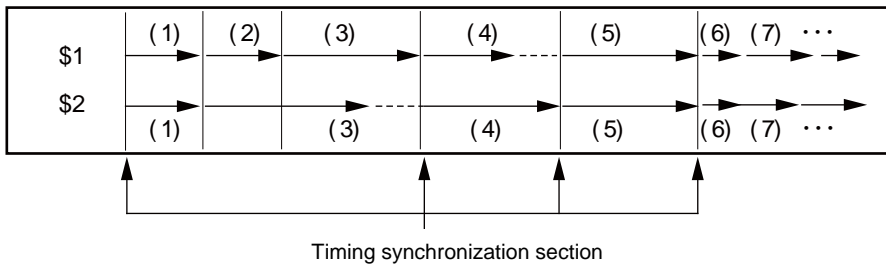
1st part system program (\$1)	2nd part system program (\$2)
G28 XZ ;	G28 XZ ;
S100 T0101 ;	T0101 ;
G15 ;	G15 ;
G00 X15 Z3 ;	G00 X15 Z3 ;
G01 Z-100 F0.2 ;	G01 Z-100 F0.2 ;
X50 Z-120 ;	X50 Z-120 ;
X52 ;	X52 ;
G14 ;	G14 ;
G28 XZ ;	G28 XZ ;
M30 ;	M30 ;



Operation example



<1st part system>	<2nd part system>	
:	:	
G15	G15	...(1)
S200		...(2)
G00 X40. Z-2.	G00 X40. Z-2.	...(3)
G01 W47. F10.	G01 W47. F5.	...(4)
G01 U40. W55.	G01 X80. Z100. F10.	...(5)
G14	G14	...(6)
G00 X100.	G00 X100.	...(7)
:	:	



- (1) Balance cut is turned ON with the G15 command.
- (2)(3) The S command and rapid traverse command are not waited, so the operation waits at the head of (4).
- (4) 1st part system will finish first, but since the next block is a cutting feed command, the operation will wait at the head of (5).
- (5) Cutting will start with 1st system and 2nd system together.
- (6) Balance cut is turned OFF with the G14 command.
- (7) Each part system will operate independently after this.



Precautions

Synchronization during movement

This function simultaneously starts cutting feed for both part systems. The following synchronization will change according to the movement amount and feedrate, etc., and thus cannot be guaranteed. To move in complete synchronization, the movement amount and feedrate must be set to the same values. Note that even if the previous conditions match, there may be cases when accurate machining, such as thread cutting, is not possible.

Number of cutting feed blocks in balance cut mode

IF G14 is commanded in one part system first and the other part system is in cutting feed, the first part system will enter the timing synchronization state. Operation cannot be advanced to the next blocks in this case.

When commanding balance cut, make sure that the same number of cutting feed blocks are set between the 1st part system and 2nd part system mode ON and mode OFF states.

<1st part system>	<2nd part system>	
:	:	
N20 G15	N20 G15	
N30 G00 X40. Z0.	N30 G00 X-40. Z250.	
N40 G01 W-30. F1000	N40 G01 W-130. F500	
N50 G01 U40. W-70.	N50 G01 X-80. Z50. F1000	
N60 G01 W-20.	N60 G14	← The balance cut mode is canceled with G14 on the 2nd part system side first, so 1st part system enters the timing synchronization state.
N70 G14 ;	N70 S200	The timing synchronization state will also be entered when only one part system has been reset.
N80 G01 X120. Z30.	N80 G00 X-100.	

Use with timing synchronization operation between part systems

If one part system is standby for synchronization with the timing synchronization operation between part systems and the other part system enters the synchronization standby state with the G15 command, both part systems will be in the standby state, and will not shift to the next block. Command so that standby for waiting for G15 and standby for waiting with the timing synchronization operation between part systems do not occur simultaneously.

Timing synchronization operation between part systems during balance cut mode

When the timing synchronization operation between part systems is issued during the balance cut mode, which is handled as a command of one block without movement and will not be waited.

Conditions for alarm with G15 and G14

- (1) For one part system
A program error (P34) will occur if G15 or G14 is commanded in one part system.
- (2) Command during milling
A program error (P481) will occur if G15 or G14 is commanded during the milling mode.

Conditions for ignoring G15 and G14

If G14 is commanded when G15 is not commanded (when balance cut is OFF), the G14 block will be handled as one that has no process.

13.20 Timing Synchronization Operation

⚠ CAUTION

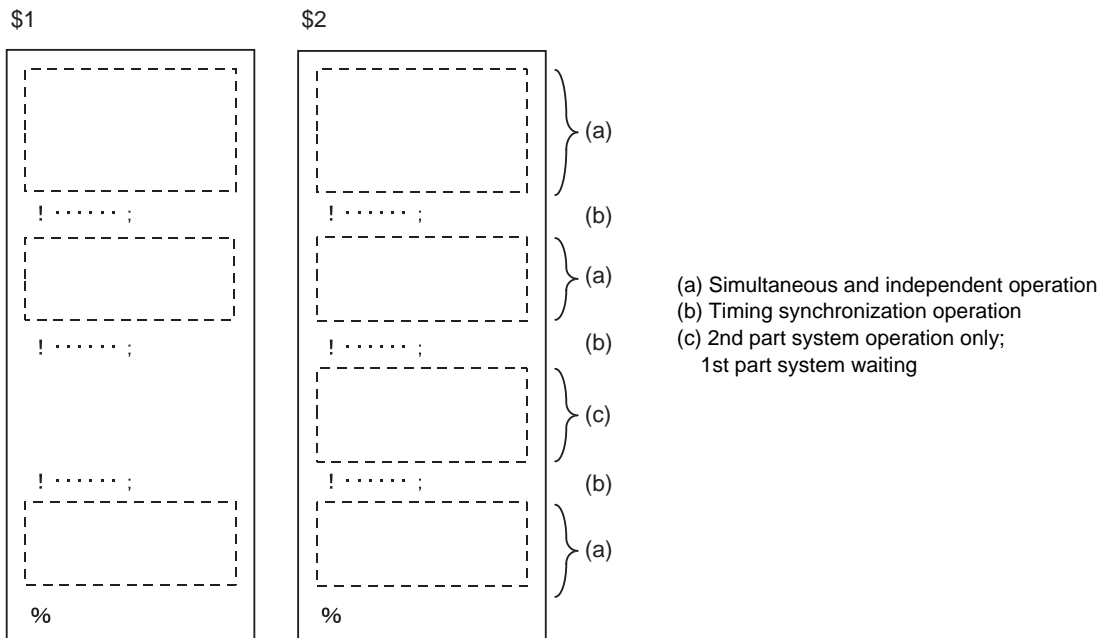
1. When programming a multi-part system, carefully observe the movements caused by other part systems' programs.

13.20.1 Timing Synchronization Operation (! code) ; !L



Function and purpose

The multi-axis, multi-part system complex control NC system can simultaneously run multiple machining programs independently. The synchronization-between-part systems function is used in cases when, at some particular point during operation, the operations of 1st and 2nd part systems are to be synchronized or in cases when the operation of only one part system is required.



Command format

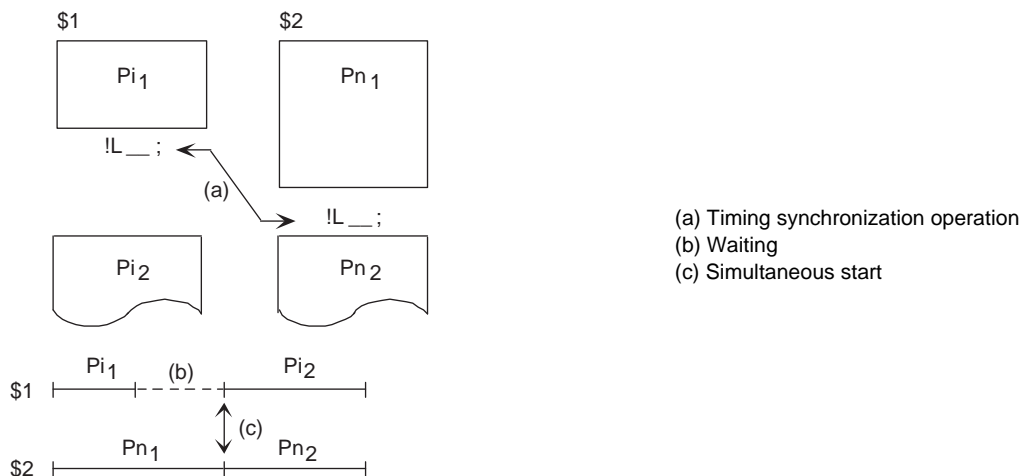
!L_ ;

L Timing Synchronization Operation No. 1 to 9999



Detailed description

- (1) If !L1__ is commanded from one part system, operation of the first part system's program will wait until !L1__ is commanded from the other part system's program.
When !L__ is commanded, the programs for the two part systems will start simultaneously.

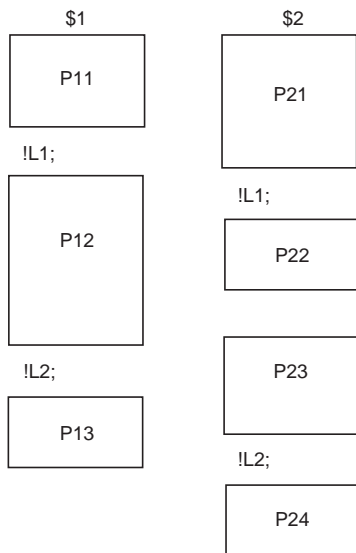


- (2) The timing synchronization command is normally issued in a single block. However, if a movement command or M, S or T command is issued in the same block, whether to synchronize after the movement command or M, S or T command or to execute the movement command or M, S or T command after synchronization will depend on the parameter (#1093 Wmvfin).
- #1093 Wmvfin
0 : Wait before executing movement command.
1 : Wait after executing movement command.
- (3) If there is no movement command in the same block as the timing synchronization operation, when the next block movement starts, synchronization may not be secured between the part systems. To synchronize the part systems when movement starts after waiting, issue the movement command in the same block as the timing synchronization operation.
- (4) Timing synchronization operation is done only while the part system to be waited is operating automatically. If this is not possible, the timing synchronization operation will be ignored and operation will advance to the next block.
- (5) 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.
- (6) "SYN" will appear in the operation status section during timing synchronization operation. The timing synchronization operation signal will be output to the PLC I/F.

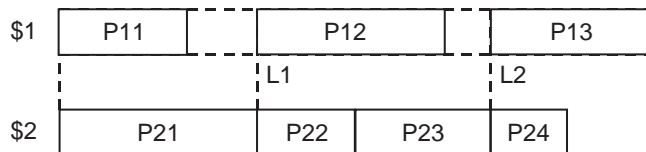


Operation example

Example of timing synchronization operation between part systems



The above programs are executed as follows:



13.20.2 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

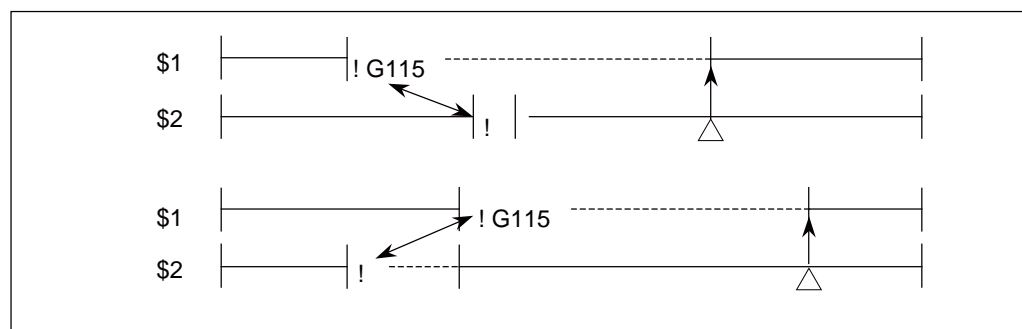
```
!L__ G115 X__ Z__ C__ ;
```

!L	Timing synchronization operation
G115	G command
X Z C	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 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.

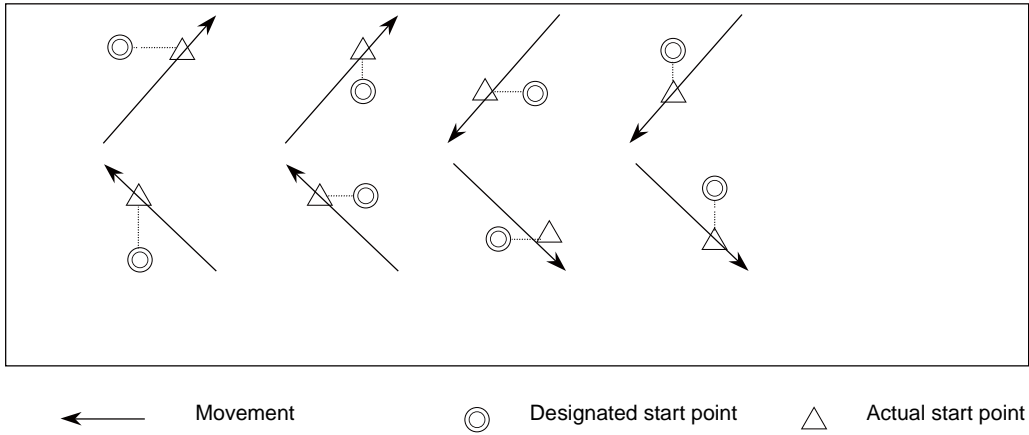


Timing synchronization operation

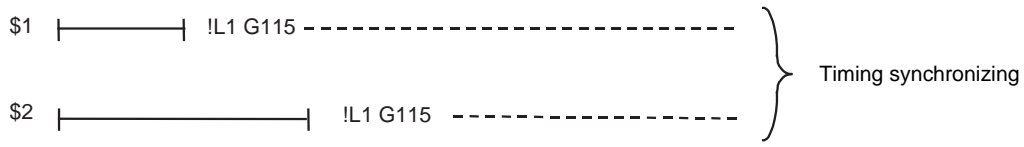


Designated start point

- (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.



- (6) The timing synchronization status continues when the G115 command has been duplicated between part systems.



- (7) The single block stop function does not apply for the G115 block.
- (8) A program error (P32) will occur if an address other than an axis is designated in G115 command block.

13.20.3 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

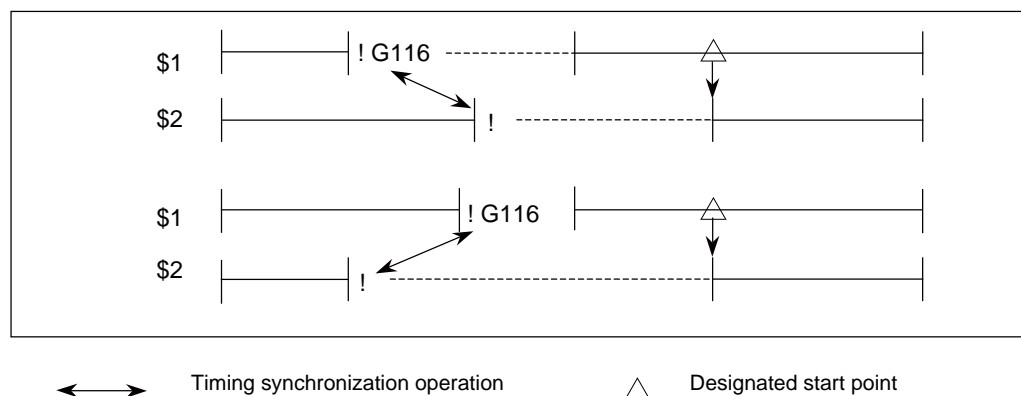
```
!L__ G116 X__ Z__ C__ ;
```

!L	Timing synchronization operation
G116	G command
X Z C	Start point (Command by axis and workpiece coordinate value.)

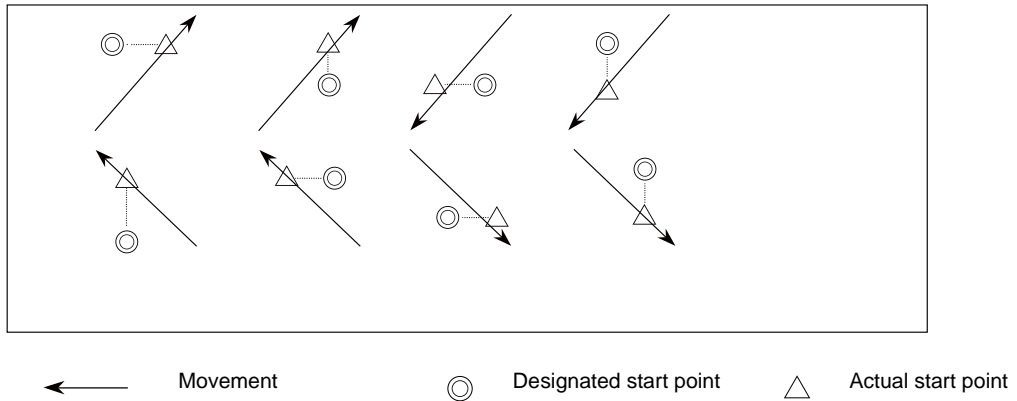


Detailed description

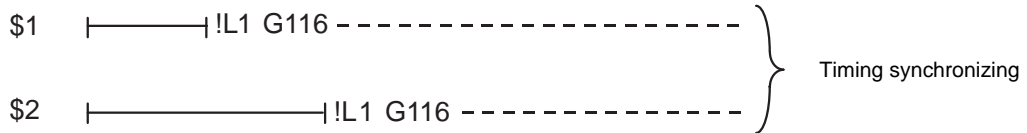
- (1) Designate the start point using the workpiece coordinates of the own part system (ex. \$1).
- (2) The start point check is executed only for the axis designated by G116.
(Example) !L1 G116 X100. ;
Once the own part system reaches X100., the other part system (ex. \$2) will start. The other axes 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.



- (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.



- (6) The timing synchronization status continues when the G116 command has been duplicated between part systems.



- (7) The single block stop function does not apply for the G116 block.
- (8) A program error (P32) will occur if an address other than an axis is designated in G116 command block.

13.20.4 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 is determined with the parameter settings.



Command format

M*** ;

***	Timing synchronization operation M code
-----	---

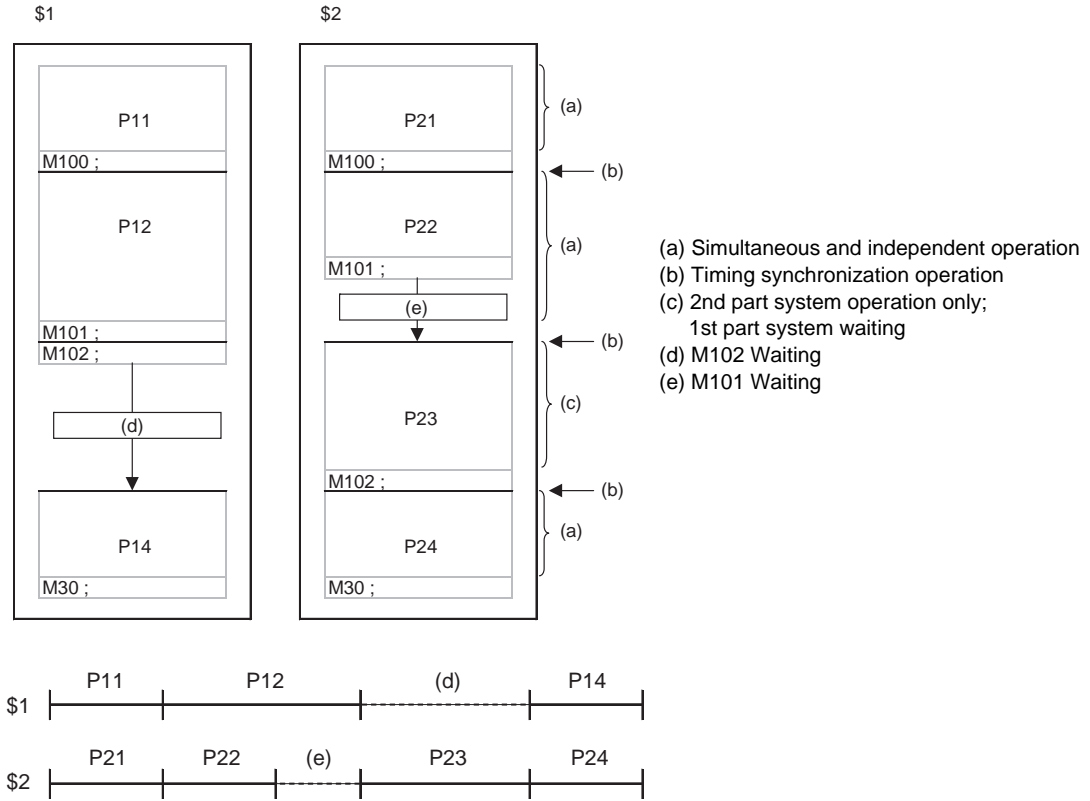
Set the timing synchronization operation M code within the setting range of the parameter.

The minimum value is set in the parameter "#1310 WtMmin", and the maximum value is set in the parameter "#1311 WtMmax".

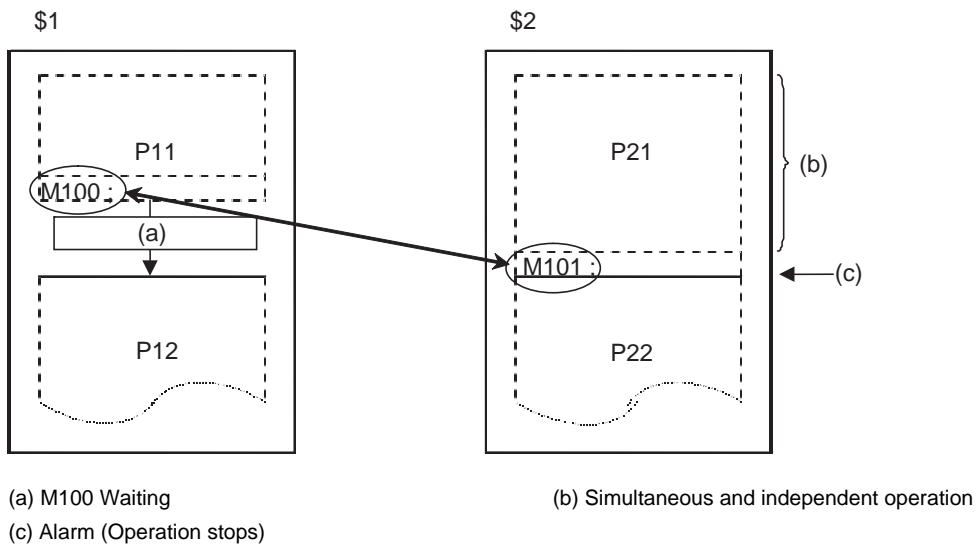


Detailed description

- (1) When the timing synchronization operation M code is commanded in the machining program, the two part systems will be waited and operation will start in the commanded block. If the timing synchronization operation M code is commanded in either part system during automatic operation, the system will wait for the same M code to be commanded in the other part system before executing the next block.



- (2) When the timing synchronization operation M code has been commanded in one part system, and the part system is standing by for waiting, an alarm will occur if a different M code is commanded in the other part system.



(3) The part systems are waited with the M code following the parameters below.

(a) M code range designation parameter (M code minimum value \leq M code \leq M code maximum value)

#	Item		Details	Setting range
1310	WtMmin	Minimum value for timing synchronization operation M code	The minimum value of the M code is set. If the setting value is "0", the the timing synchronization operation M code will be ignored.	0, 100 to 99999999
1311	WtMmax	Maximum value for timing synchronization operation M code	The maximum value of the M code is set. If the setting value is "0", the the timing synchronization operation M code will be ignored.	0, 100 to 99999999

Timing synchronization operation between part systems using the M code is valid when both of the above parameters are set to a value other than "0". (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	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: Waiting before executing movement command 1: Waiting after executing movement command	0 / 1



Precautions

- (1) When timing synchronization operation with the M code, always command the M code in an independent block.
- (2) When standing by after commanding the timing synchronization operation M code in one part system, an alarm will occur if a different M code is commanded in the other part system. Operation will stop in both part systems.
- (3) The M code in the machining program can be ignored with the timing synchronization operation ignore signal. Operation with a single part system is possible without deleting the M code in the machining program.
- (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, an alarm will occur 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, an alarm will occur 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 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.

13.21 Mixed Control (Cross Axis Control) I ; G110



Function and purpose

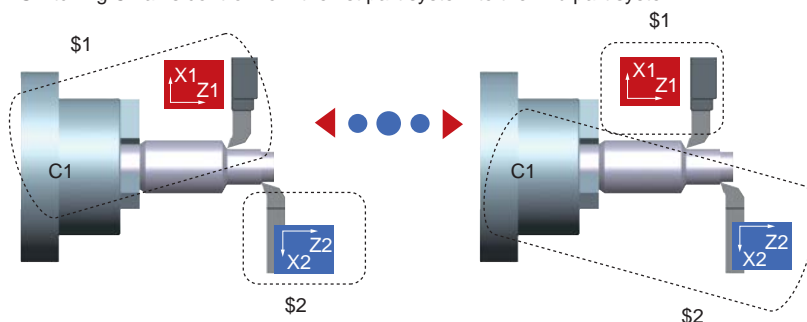
This function enables any axis to be replaced by another axis between part systems.

This makes it possible to perform operations which are not possible with regular axis configurations: for instance, tools which are provided only on part system 1 can be used for machining on part system 2.

This manual contains descriptions where part system Nos. have been added after the axis names in order to identify which part system the axes belong to (such as X1). In terms of designating the program commands, however, a single letter is used to designate the axis address just as before.

There are two types of mixed control (cross axis control) depending on the command method: Mixed control (cross axis control) I (G command), and mixed control (cross axis control) II (PLC signal). Set the parameter "#1280 ext16/bit4" for which method to use.

Switching C1 axis control from the 1st part system to the 2nd part system



Command format

G110 Axis name1 Axis name2 Axis name3 ... ; ... Mixed Control (Cross Axis Control) I

Axis name1 Axis name2 Axis name3 ...	All axes which exist in the part system after commanded (axis name set in the parameter "#1022 axname2")
---	---

G110 needs to be commanded for a part system which occurs the existing axis switching (or move) by other part system's G110 command.



Detailed description

(1) 2-digit axis name

For two or more part systems, the same axis name may exist in each part system. To distinguish the name, display 2-digit axis name set by the parameter "#1022 axname2".

(2) Displaying the switched coordinate value

The basic specification parameter "#1280 ext16/bit2,bit6" can switch the axis name and coordinate value (such as the workpiece coordinate position, and the machine coordinate position).

This parameter can also set the command to move the axis (such as moving C axis of 1st part system to 2nd part system).

Following items are switched for display

Operation screen (except restart search)	Counter	Each counter(*), machine status animation
	Trace	Each counter(*), display mode, drawing area
	Coordinate system	Each coordinate system offset
Setup screen	Counter	Each counter(*)
	Coordinate system	Each coordinate system offset

(*) The setting of the basic specification parameter "#1280 ext16/bit6" prevents the displays of machine position counters from being interchanged.

The following is displayed when X axis of 1st part system and X axis of 2nd part system are switched.



- (3) G110 command carries out timing synchronization operation until it becomes mixing valid status by other part system's G110 command. On this occasion, "CRS" appears in the operation status of the operation screen to indicate that the machine is in timing synchronization operation by mixed control. After that, mixing completes when the part system to control against all axes to be mixed which relates to that part system, then "CRS" is cleared, and the machining program continues.



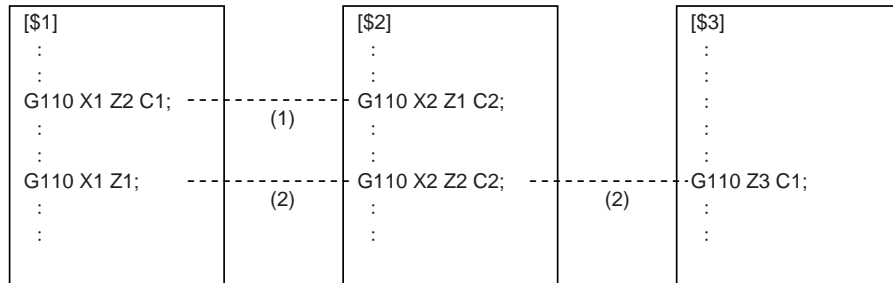
Program example

Initial axis configuration

1st part system : X1, Z1, C1

2nd part system : X2, Z2, C2

3rd part system : Z3



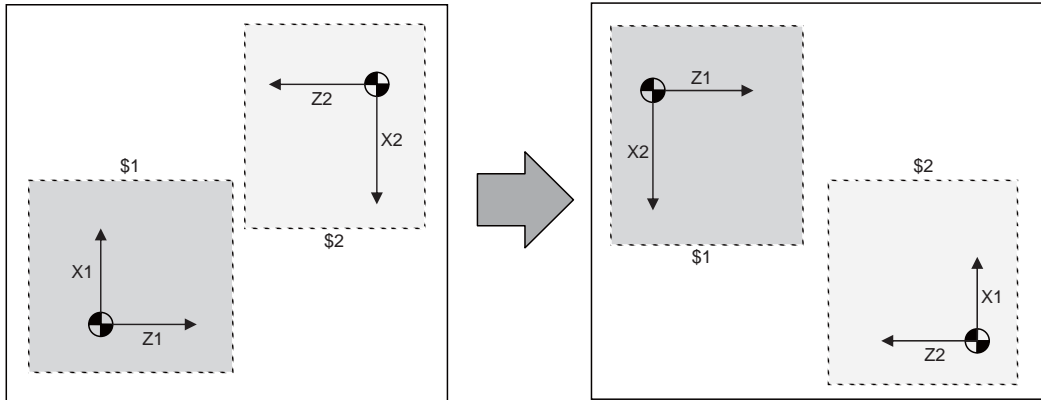
- (1) 1st part system and 2nd part system execute timing synchronization operation with each other, and Z1 axis is switched with Z2 axis. Timing synchronization operation is not executed for 3rd part system as all axes of 1st part system and 2nd part system can mix due to 1st part system and 2nd part system's G commands.
- (2) "1st part system and 2nd part system" and "1st part system and 3rd part system" execute timing synchronization operation with each other. Z1 axis and Z2 axis are replaced to restore the initial state, and move C1 axis to 3rd part system. On this occasion, if G110 command of 2nd part system is executed lastly, 3rd part system does not wait for 2nd part system's G command, and mix 1st part system and C1 axis first and then the program continues.



Relation with other functions

(1) Coordinate system

Because the coordinate information, such as a reference position, machine coordinate zero point and workpiece coordinate zero point, is decided for each axis, the coordinate systems are also switched when the axes are switched by the mixed control.



(2) Plane selection

For the part system where the axes are switched by mixed control, the plane is configured with the switched axis. On this occasion, an axis under the mixed control configures the plane with the axis address set by the parameter "#1023 crsadr".

(Example)

Suppose the following is set:

- 1st part system axis configuration (X1, Z1, C1, Y1)
- 2nd part system axis configuration (X2, Z2, C2)
- "#1023 crsadr" X of Y1 axis

Plane configuration of 2nd part system

- "#1026 base_I" X
- "#1027 base_J" X
- "#1027 base_K" Z

If mixing X2 axis and Y1 axis by mixed control, 2nd part system's plane configuration is as follows:

- I-J plane: Y1 axis - Y1 axis (this is insignificant as a plane like X2 axis)
- J-K plane: Y1 axis - Z1 axis
- K-I plane: Z1 axis - Y1 axis

When the other part system's axis is moved by the mixed control or when using the plane which includes the axis name never existed before, set the name of "#1023 crsadr" to the parallel axis parameter "#1028 aux_I" to "#1030 aux_K".

(Example)

- 1st part system axis configuration (X1, Z1, C1, Y1)
- 2nd part system axis configuration (X2, Z2, C2)
- Y axis of "#1023 crsadr" Y

In the above example, when the mixing, such as moving Y axis to 2nd part system, is executed, to configure the plane with Y axis as J direction axis, set the 2nd part system plane configuration parameter "#1029 aux_J" to "Y".

(3) Tool compensation

When the tool compensation is carried out after the axis is switched by the mixed control, the tool compensation is applied to the mixed axis configuration. On this occasion, when two axes whose axes addresses are different are switched is different are switched, the compensation is carried out for the switched axis address.

For example, when Y axis (Y1) in 1st part system is mixed with X axis (X2) in 2nd part system, the compensation amount for X2 axis is applied to Y axis.

(4) Soft limit

Soft limit is carried out using axis parameter setting value. Soft limit I and II define the movement range of the axis, and it is valid even during the mixed control. Soft limit IB, IIB, and IC are the range set with two or more axes' parameters setting values, and the range during mixed control is set by the parameter of the then axis configuration.

Therefore, in across part systems where the zero points differ, the setting values of soft limit IB, IIB, and IC need to match for the mixed configuration axis to validate the same area as before mixed even during the mixed control.

(5) Chuck barrier/tail stock barrier

Chuck barrier/tail stock barrier are the tool nose point entry prohibited area of the tool which is configured with the axis parameter setting value of the 1st axis and 2nd axis. In across part systems where the zero points differ, each parameter needs to be set again to validate the chuck barrier/tail stock barrier after the mixed control.



Precautions

Precautions when commanding

- (1) Command G110 in an independent block. If not, a program error (P33) will occur.
- (2) All axes which had existed before switching in the G110 command need to be commanded by G110 in some part system.
- (3) Pairing G110 command performs timing synchronization operation between the part systems.
- (4) Mixed control by G110 command is only valid when the parameter "#1022 ext16/bit4" is set to "1". If G110 is commanded when "ext16/bit4" is set to "0", a program error (P610) will occur.
- (5) Command the axis name set with the parameter "#1022 axname2". If not, a program error (P503) will occur.
- (6) The axis name commanded with G110 needs to be 2-digits. Set the parameter "#1022 axname2" with 2-digits.
- (7) "In mixed control" signal is ON even during the mixed control by G110.
- (8) When G110 is commanded and the command part system is in a state where the mixed control is disabled, a program error (P501) will occur.
Under the following circumstances, the mixed control is disabled:
 - In tool nose mode
 - In polar coordinate interpolation mode
 - In cylindrical interpolation mode
 - In balance cut mode
 - In fixed cycle machining mode
 - In mirror image for facing tool posts
 - In constant surface speed control mode
 - In hobbing mode
 - In axis name switch
- (9) The part system without axis cannot be created with the mixed control. If G110 command without axis name is given, a program error (P33) will occur.
- (10) If the number of axes commanded exceeds the maximum control axis for the part system, a program error (P503) will occur.
- (11) If G110 is commanded to the part system whose parameter "#1501 polyax" is set other than "1", a program error (P503) will occur.
- (12) If switching is commanded with G110 to the axis whose parameter "#2071 s_axis" is set to other than "0", a program error (P503) will occur.
- (13) If switching is commanded with G110 to the axis whose parameter "#1205 G0bdcc" is set to other than "0", a program error (P503) will occur.
- (14) If switching is commanded with G110 to the axis whose parameter "#1072 chop_ax" is set to other than "0", a program error (P503) will occur.
- (15) If switching is commanded to the parameter which is set as a master axis or slave axis for the synchronous control, a program error (P503) will occur.
- (16) Mixed control by G110 command is invalid for a graphic check.
- (17) If a program restart is performed for G110 command, a program error (P49) will occur.

Other precautions

- (1) A command which duplicates the existing axis and the command address by the mixed control cannot be executed.
- (2) Tool compensation amount holds the value before the mixed control even after the mixed control was carried out. Tool compensation command during mixing compensates the tool with the axis configuration of the part system of mixing destination.
- (3) 1st axis and 2nd axis of the tool compensation and the tool nose wear compensation are fixed to 1st axis and 2nd axis of each part system respectively. Thus, if the mixed control which moves 1st axis and 2nd axis to the other part system, the target axes for the tool length compensation and the tool nose wear compensation are changed so that the normal compensation cannot be carried out.
- (4) For mixing the diameter axis and radius axis, if the tool compensation is commanded to the radius axis during mixing, only half the value of tool compensation amount which is set to the mixing destination part system is applied. On the contrary, for the diameter axis, the double of the tool compensation amount which is set to the mixing destination part system is applied.
- (5) Mixed control is canceled by emergency stop.
- (6) Mixed control is canceled by reset. However, if the parameter "#1280 ext16/bit1" is set to "1", the mixed control state will continue when it is reset.
- (7) PLC I/F per axis during switching the axes with the mixed control uses the same device before switching. However, if the parameter "#1280 ext16/bit0" is set to "1", mirror image, machine lock and interlock signal use the device in the command part system side. For moving the axis not switching, PLC signal of the axis No. in the part system is used by following terms:
 - Set the last axis No. of the source part system for the moved axis.
 - Set the axis No. in ascending order of destination part system No. at the end of the destination part system for the axes moved from several part systems.
 - Set the axis No. in axis order of the source part system at the end of the destination part system for the several axes moved from the same part system.
- (8) Mixed control is only valid during the automatic operation. If switching 1st part system axis and 2nd part system axis or restore the axes to their original state, both part systems must be in the automatic operation. If an attempt is made to mixing axes of the part systems not in the automatic operation, the error "M01 Mixed sync ctrl disable modal 1035" will occur.

13.22 Control Axis Superimposition I ; G126



Function and purpose

This function enables superimposition on and control of an axis in a selected part system with an axis in another part system.

This is effective when machining by the 1st part system (X1, Z1) and the 2nd part system (X2, Z2) are executed simultaneously for the machine configuration where a workpiece moves by a movement command to the Z axis direction. This function eliminates the need of fixing the workpiece position for machining in the superimposed part system (the 2nd part system) and allows the simultaneous machining of the reference axis part system (the 1st part system) and the superimposed axis part system (the 2nd part system).

There are two types of control axis superimposition: control axis superimposition I (G code) and control axis superimposition II (PLC signal). The difference between the two is command method.

Select the method with the basic parameter "#1280 ext16/bit7".

Reference axis and superimposition axis are defined as follows.

Reference axis	The basic in the control axis superimposition function (moves only by its own axis command)
Superimposed axis	The axis which moves including the reference axis movement in the control axis superimposition function (moves by reference axis or/and own axis command).

While using the control axis superimposition function, the workpiece zero point of the superimposed axis moves according to the movement amount of the reference axis. Because the superimposed axis tries to maintain the position in the workpiece coordinate system, it moves as much as the movement amount of the reference axis.

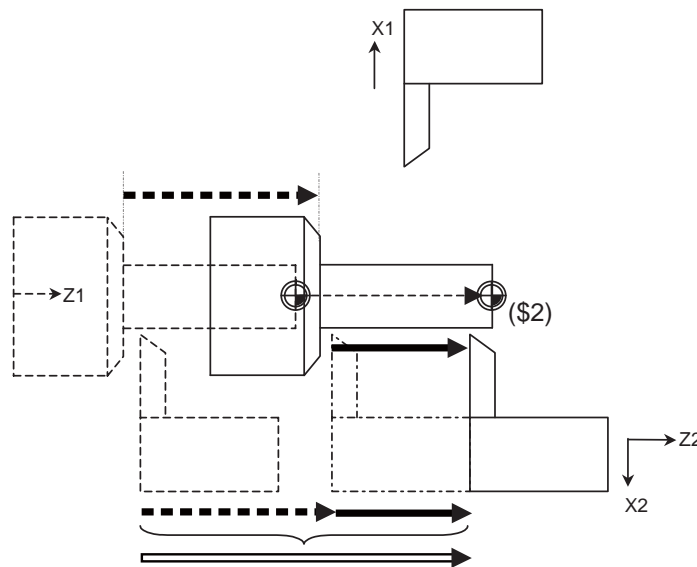
As long as the reference axis and the superimposed axis do not overlap, the number of control axis superimposition set has no limitation.

(Example)

Reference axis: Z1

Superimposed axis: Z2

The zero point in the figure indicates the 2nd part system workpiece coordinate.



---➔ Z1 actual movement amount = Z1 commanded movement amount

—➔ Z2 commanded movement amount

==➔ Z2 actual movement amount = Z1 actual movement amount + Z2 commanded movement amount



Command format

Superimposition start

G126 Superimposed axis name = Reference axis name (,P__);

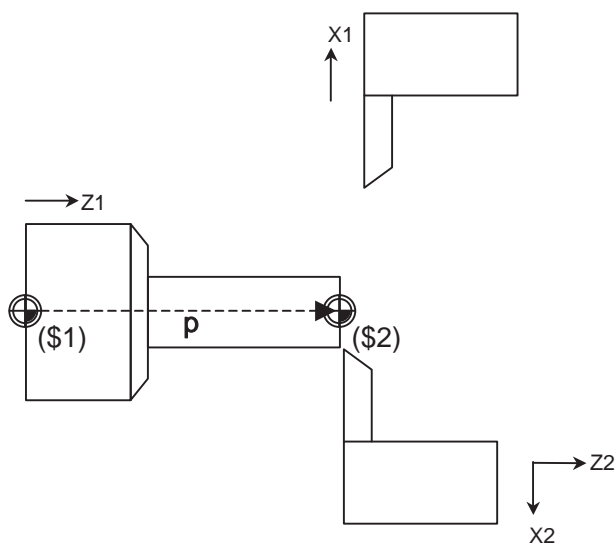
Superimposed axis name	The axis to be operated as superimposed axis (The axis name set in the parameter "#1022 axname2")
Reference axis name	The axis to be operated as reference axis (The axis name set in the parameter "#1022 axname2")
,P	Designation of the superimposed axis workpiece coordinate system (Set the workpiece coordinate zero point of the superimposed axis in the workpiece coordinate system of the basic axis) (Radius value)

Superimposition end

G126 Superimposed axis name = Reference axis name (,P__);

Superimposed axis name	The axis operating as superimposed axis (The axis name set in the parameter "#1022 axname2")
------------------------	--

Command: G126 Z2=Z1 ,Pp ;



**Detailed description**

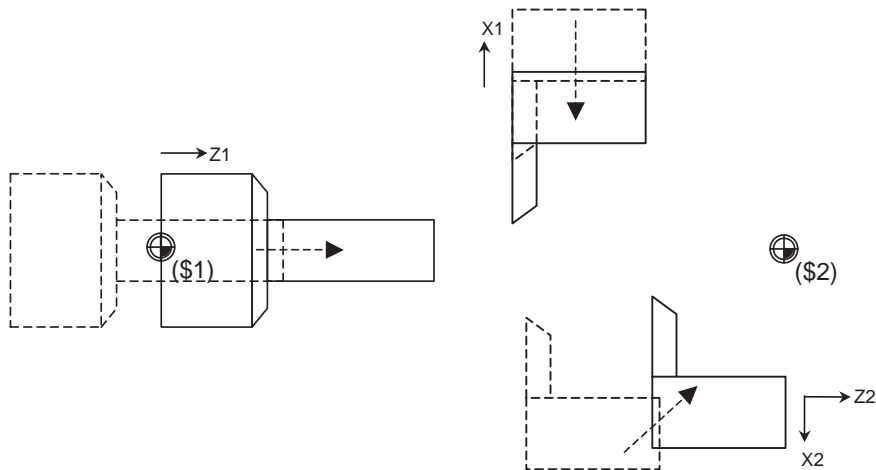
The machine configuration in the figure below describes each operation.

In the 1st part system (X1,Z1), the X axis command moves the tool and the Z axis command moves the workpiece.

In the 2nd part system (X2,Z2), the X and Z axes command moves the tool.

The workpiece coordinate zero point (\$1) in the figure indicates the Z axis workpiece coordinate zero point.

Reference axis: Z1, superimposed axis: Z2.



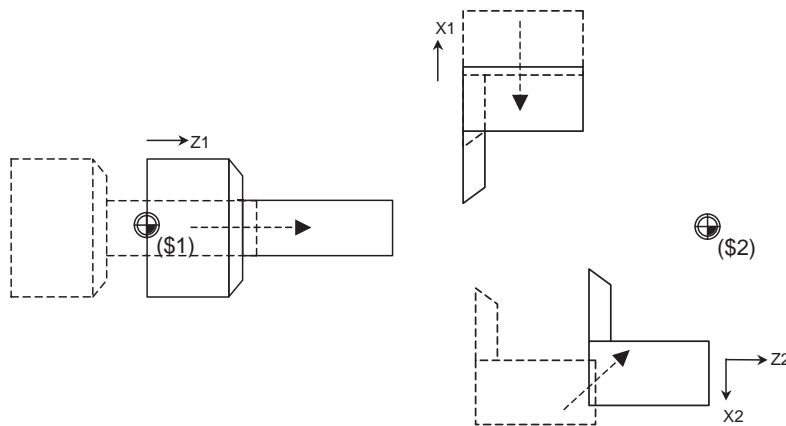
Superimposition starting operation

The following operation example explains the start command to superimpose Z2 axis onto Z1 axis.

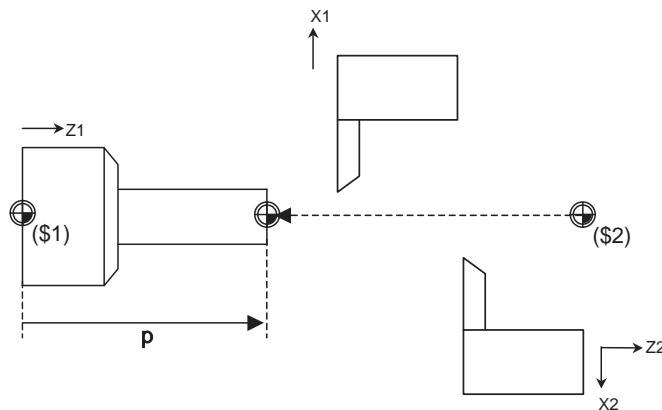
Command: G126 Z2 = Z1 ,Pp ;

The superimposition start command automatically executes the following operation.

- (1) Superimposition starting operation 1 (Wait for the deceleration completion)
After completing the currently processing block, the part system containing the reference axis and superimposed axis decelerates and stops.



- (2) Superimposition starting operation 2 (Movement of the superimposed axis workpiece zero point)
Set the superimposed axis workpiece zero point by a P command. The setting of the workpiece zero point can be carried out by shifting the G92 coordinate system. (The superimposed axis will not move.)

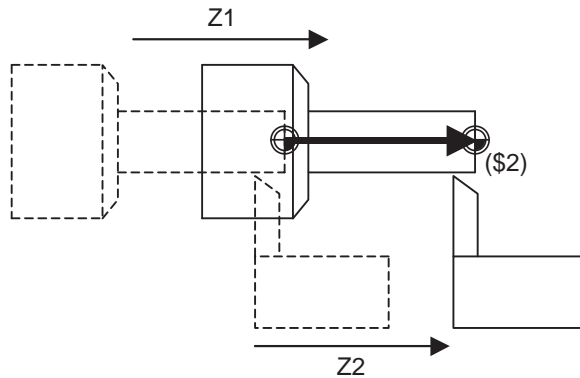


- (3) Superimposition starting operation 3
Switch the time constant of all axes of the part system containing the reference axis and superimposed axis, into the time constant for the superimposition set in the parameter "#2092 - #2095".
- (4) Superimposition starting operation 4
Switch the rapid traverse rate and clamp rate of all axes of the part system containing the reference axis and superimposed axis, into these for the superimposition set in the parameter "#2090 - #2091".

Operation during superimposition

[Workpiece coordinate system of the superimposed axis]

When the movement of reference axis is executed, the superimposed axis workpiece coordinate zero point moves according to the movement of the superimposed axis. The superimposed axis moves as much as the reference axis to maintain the workpiece position.



[The feedrate of the reference axis and superimposed axis]

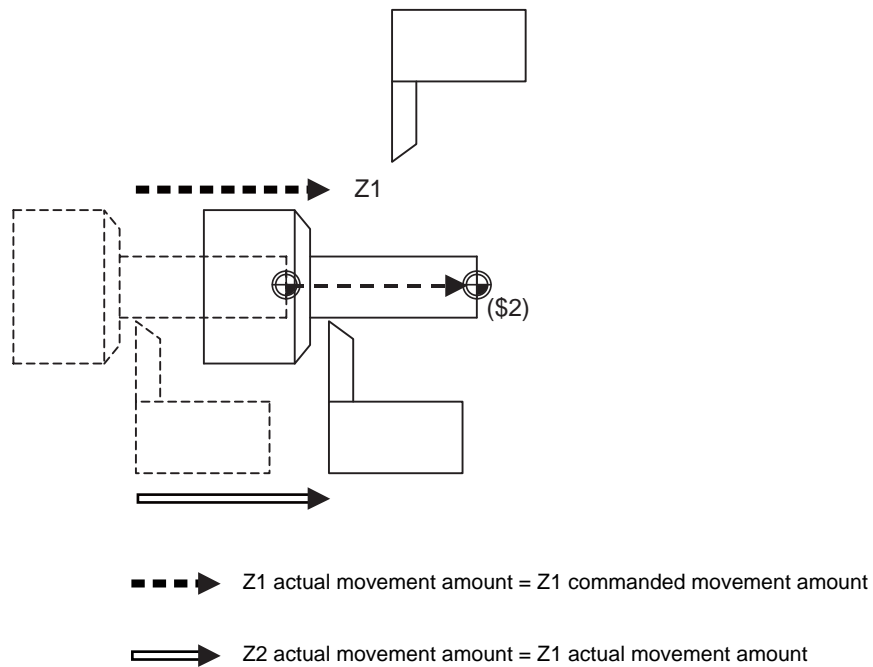
If movement command is issued to both the reference axis and superimposed axis, movement rate of the reference axis will be added to that of the superimposed axis, making it faster than when the movement is commanded only by superimposed axis. In this case, the feedrate may exceed the motor capacity under the normal speed clamp processing. Switch the rapid traverse rate and clamp rate of the part system containing the reference axis and superimposed axis to the following parameter setting values during the superimposition control.

- Rapid traverse rate during the superimposition control #2090 plrapid
- Clamp rate during the superimposition control #2091 plclamp

[Axis movement during the superimposition control]

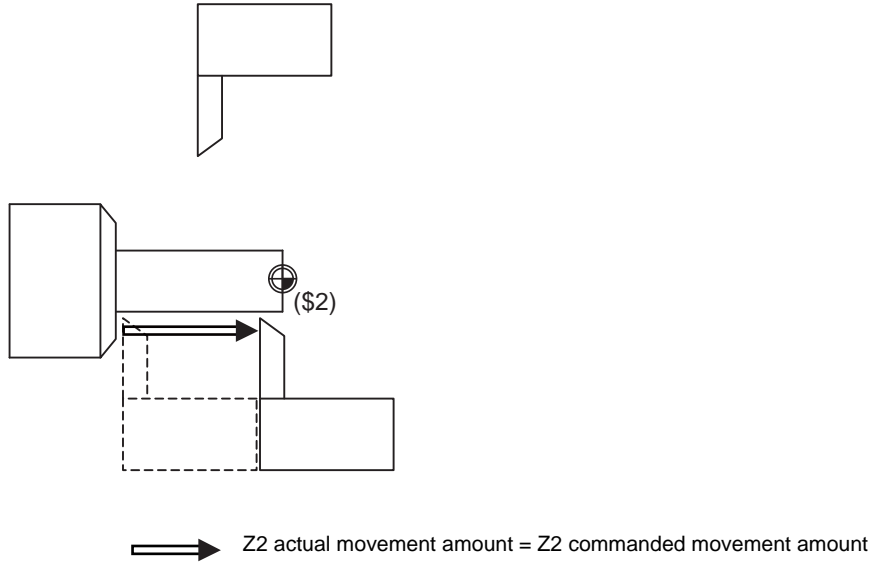
- (1) When commanding movement only to the reference axis

If a movement command is issued only to the reference axis and not to the superimposition axis when the control axis is superimposed, the superimposition axis moves as same as the reference axis. And the workpiece zero point of the superimposed axis moves as much as the movement of the reference axis.



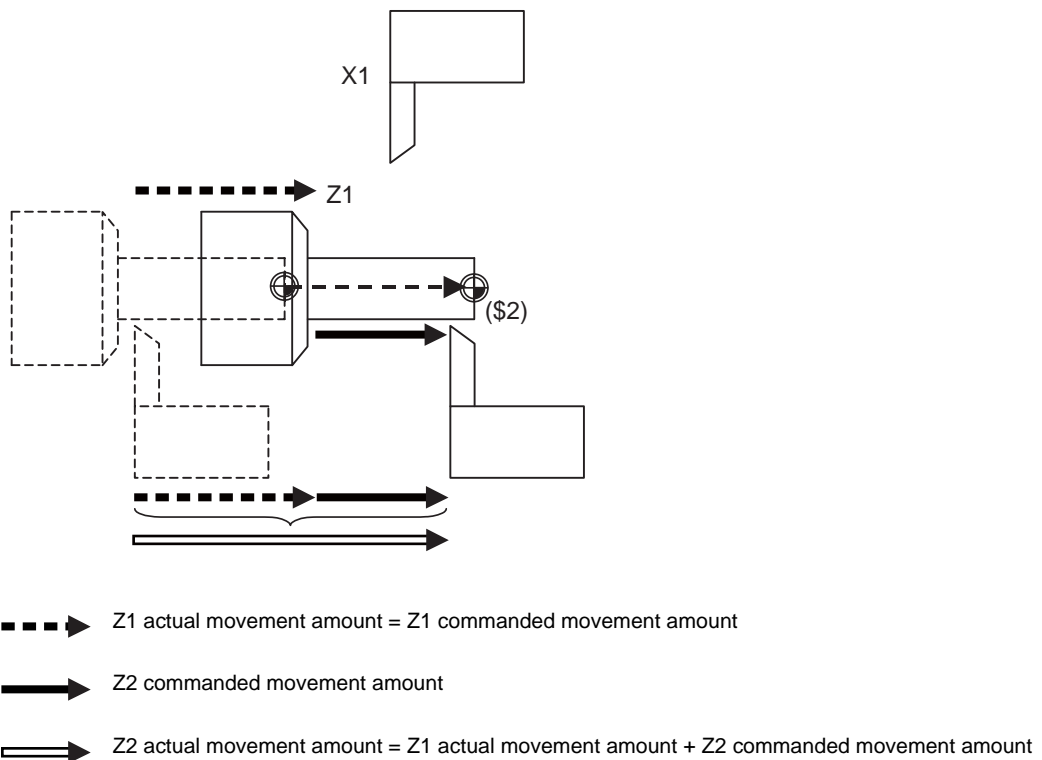
(2) When commanding movement only to the superimposition axis

If a movement command is issued only to the superimposition axis and not to the reference axis when the control axis is superimposed, the superimposition axis actual movement amount equals to that of the reference axis. And the workpiece zero point of the superimposed axis will not move because the reference axis will not move.



(3) When commanding movement to both the reference axis and superimposed axis

If a movement command is issued to both the reference axis and superimposition axis at the same time when the control axis is superimposed, the superimposition axis actual movement amount equals to "reference axis movement amount + superimposition axis commanded movement amount. And the relative movement amount of the superimposition axis to the workpiece equals the commanded movement amount of the superimposition axis. And the workpiece zero point of the superimposed axis moves as much as the movement amount of the reference axis.



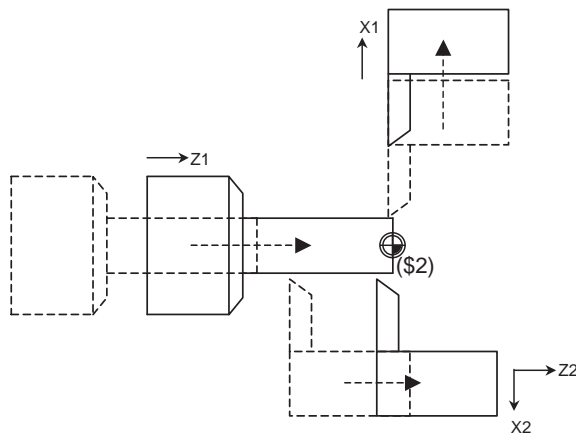
Superimposition end

This section shows the examples of executing an ending operation when the Z2 axis is superimposed onto Z1 axis.

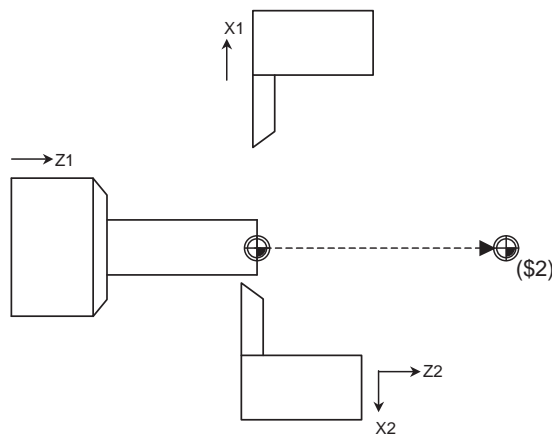
Command: G126 Z2 ;

The superimposition end command automatically executes the following operation.

- (1) Superimposition ending operation 1 (Wait for the deceleration completion)
After completing the currently processing block, the part system containing the reference axis and superimposed axis decelerates and stops.



- (2) Superimposition ending operation 2
Return to the normal the rapid traverse rate and clamp rate of the part system containing the reference axis and superimposed axis.
- (3) Superimposition ending operation 3
Return to the normal the acceleration/deceleration time constant of the part system containing the reference axis and superimposed axis.
- (4) Superimposition ending operation 4 (Movement of the superimposed axis workpiece zero point)
Return the superimposed axis workpiece coordinate zero point to where it was before the superimposition start command was issued. (The superimposed axis will not move.)





Program example

(1) When commanding from the part system containing the reference axis

[1st part system]		[2nd part system]
:		:
:		:
!L1 ;	Waits for the axis to stop by the timing synchronization operation command.	!L1 ;
G126 Z2=Z1 ,P0 ;	Superimposes Z2 axis onto Z1 axis	
!L2 ;	Waits for the axis to stop by the timing synchronization operation command.	!L2 ;
G00 X46. Z2. ;	Superimposing operation	G00 X54. Z-70. ;
G01 Z-50. F0.1 ;		G01 X40. F0.1 ;
X54. ;		:
:		:
:		:
!L3 ;	Waits for the axis to stop by the timing synchronization operation command.	!L3 ;
G126 Z2 ;	Ends superimposition of Z2 axis	
!L4 ;	Waits for the axis to stop by the timing synchronization operation command.	!L4 ;
:		:

(2) When commanding from a part system containing neither the superimposed axis nor reference axis

[1st part system]	[2nd part system]		[3rd part system]
:	:		:
:	:		:
!2!3 L1 ;	!1!3 L1 ;	Waits for the axis to stop by the timing synchronization operation command.	!1!2 L1 ;
		Superimposes Z2 axis onto Z1 axis	G126 Z2=Z1 ,P10. ;
!2!3 L2 ;	!1!3 L2 ;	Waits for the axis to stop by the timing synchronization operation command.	!1!2 L2 ;
G00 X46. Z2. ;	G00 X54. Z-80. ;	Superimposing operation	G00 X46. Z2. ;
G01 Z-50. F0.1 ;	G01 X40. F0.1 ;		G01 Z-50. F0.1 ;
X54. ;	:		X54. ;
:	:		:
:	:		:
!2!3 L3 ;	!1!3 L3 ;	Waits for the axis to stop by the timing synchronization operation command.	!1!2 L3 ;
		Ends superimposition of Z2 axis	G126 Z2 ;
!2!3 L4 ;	!1!3 L4 ;	Waits for the axis to stop by the timing synchronization operation command.	!1!2 L4 ;



Relation with other functions

Functions which cannot be used simultaneously with the control axis superimposition

- (1) An alarm will occur if the following commands are issued to the superimposed axis and reference axis during the control axis superimposition.

<Reference axis>

Function	G code	Alarm
Dog-type reference position return	G28	Operation error 1003
Skip command	G31	

<Superimposed axis>

Function	G code	Alarm
Automatic reference point return	G28 - G30 G31 G53	Operation error 1003
Skip command		
Machine coordinate system selection command		

- (2) An alarm will occur if the following functions are issued to the superimposed axis and reference axis during the control axis superimposition.

Function	Alarm
Synchronization control	Operation error 1036
Control axis synchronization across part systems (Note)	Operation error 1037

- (Note) The alarm (operation error 1037) will also occur if the control axis synchronization across part systems command is issued to the axis whose part system includes the reference axis and the superimposed axis during the control axis superimposition.

- (3) An alarm will occur if the control axis superimposition start/end command is issued to an axis executing the following functions.

Function	Alarm
Control axis synchronization across part systems (Note) Synchronization control Milling Interpolation	Operation error 1004

- (Note) The alarm (operation error 1004) will also occur if the superimpose start/end command is issued to the axis whose part system includes the reference axis and synchronous axis during the control axis superimposition.

As the control axis superimposition automatically switches the acceleration/deceleration time constant, it cannot be used with the function whose acceleration/deceleration time constant must be the same in the two related axes, such as the control axis synchronization across part systems.

Stroke end/Stored stroke limit

In an automatic operation mode, the part system containing the superimposed axis/reference axis will stop when the axis of the part system reaches a stroke end (over travel)/stored stroke limit during the control axis superimposition.

In a manual operation mode, both the superimposed axis/reference axis will stop when either axis reaches a stroke end (over travel)/stored stroke limit during the control axis superimposition.

Chuck Barrier/Tailstock Barrier check

In an automatic operation mode, the part system containing the superimposed axis/reference axis will stop when the axis of the part system reaches the barrier boundary with a chuck barrier/tailstock barrier check during the control axis superimposition.

In a manual operation mode, both the superimposed axis/reference axis will stop when either axis reaches the barrier boundary with a chuck barrier/tailstock barrier check during the control axis superimposition.



Precautions

Precautions for superimposition start command

- (1) The superimposition start command can be issued from a part system which does not contain the superimposed axis/reference axis.
- (2) The axis name set in the parameter "#1022 axname2" must be used to issue the superimposition start command. A program error (P520) will occur if an axis which is not set in the parameter is commanded.
- (3) The axis name needs to be 2 digits. Set the parameter "#1022 axname2" with 2 digits.
- (4) To conduct a superimposition, the superimposition axis No. must be bigger than that of the reference axis. A program error (P520) will occur if the superimposition axis No. is smaller than that of the reference axis. The axis No. increases towards the right side of the reference axis-specific parameter setting screen.
- (5) Program error (P520) will occur if a same axis is designated as the superimposed axis and reference axis at the superimposition start command.
- (6) Program error (P33) will occur if P address is written without comma at the superimposition start command.
- (7) Program error (P33) will occur if G126 alone is issued.
- (8) Program error (P520) will occur if a rotary axis is selected as the superimposed axis/reference axis.
- (9) In G126 block, a superimposition command can be issued only to one set of axes. Program error (P241) will occur if more than 2 sets are commanded.
- (10) A program error (P520) will occur if the superimposition start command is executed to an axis whose parameter "#2071 s_axis Inclined axis selection" is not "0".
- (11) Command this in an independent block.
- (12) Program error (P520) will occur if the superimposition start command is executed to an axis whose parameter "#1205 G0bcc Acceleration and deceleration before G0 interpolation" is not "0".
- (13) When ,P command is P0, the superimposed axis workpiece coordinate zero point overlaps with that of the reference axis.
- (14) When there is no ,P command, the superimposed axis workpiece coordinate zero point will be same as before the superimposition start command.
- (15) Program error (P35) will occur if a value exceeding the command range is commanded by ,P.
- (16) If G126 is commanded without this option, the program error (P39) occurs.
- (17) G126 command is enabled with the G code lists 2, 3, 4 and 5. If G126 is commanded with the G code 6 or 7, the program error (P39) will occur.

Precautions for superimposition end command

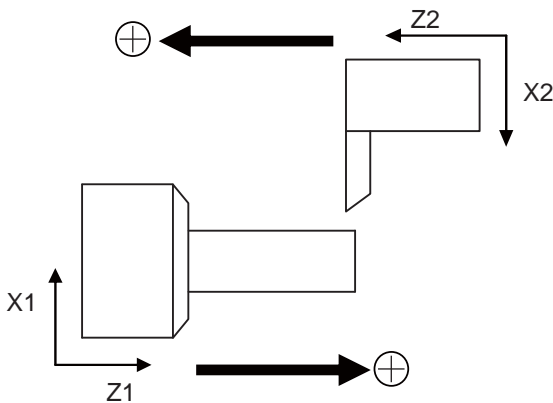
- (1) The superimposition end command can be issued from a part system which does not contain the superimposed axis/reference axis.
- (2) The axis name set in the parameter "#1022 axname2" must be used to issue the superimposition end command. A program error (P520) will occur if an axis which is not set in the parameter is commanded.
- (3) The axis name needs to be 2 digits. Set the parameter "#1022 axname2" with 2 digits.
- (4) The superimposition end command will be ignored, if an axis other than superimposed axes is commanded.
- (5) A program error (P33) will occur if G126 alone is issued.
- (6) In G126 block, a superimposition end command can be issued to only one superimposed axis. A program error (P33) will occur if more than one axis is commanded.
- (7) Command this in an independent block.

Other precautions

- (1) When being reset during the superimposition, the operation will depend on by the parameter "#1280 ext16/bit3".
- (2) A timing synchronization operation must be conducted in the block just before the superimposition start/end command, in order to stop the superimposed axis/reference axis and maintain timing between the superimposed axis/reference axis.
- (3) When the coordinate axis direction differs between the superimposed axis and reference axis, set the axis specification parameter "#2143 polar" as follows. Set "0" to the reference axis and set "1" to the superimposed axis.

For example, when the reference axis Z1 and the superimposed axis Z2 are in a relation as shown in the figure below, the parameter should be set as follows;

- Z1 axis polar 0 (+)
- Z2 axis polar 1 (-)



- (4) An alarm (operation error 1004) will occur if a control axis superimposition command, which includes a reference axis and synchronous axis under the synchronous control, is executed.
- (5) An alarm (operation error 1004) will occur if the control axis superimposition command is issued to the axis whose part system includes the reference axis and synchronous axis during the control axis superimposition during the control axis synchronization across part systems.
- (6) An alarm (operation error 1037) will occur if the control axis synchronization across part systems is conducted to the axes of the part system which includes the superimposed axis/reference axis under a control axis superimposition.
- (7) An alarm (operation error 1036) will occur if a synchronous control is conducted to the superimposed axis/reference axis under a control axis superimposition.
- (8) An alarm (operation error 1004) will occur if a control axis superimposition command is executed during a milling interpolation.
- (9) A program error (P520) will occur if the control axis superimposition command is executed to an axis whose parameter "#2071 s_axis Inclined axis selection" is not "0".
- (10) An alarm (operation error 1004) will occur if a superimposition start command is issued to a superimposed axis or reference axis with other axis before performing a superimposition end command.
- (11) Do not use restart search function to commands in the control axis superimposition because the restart position cannot be guaranteed depending on the superimposition movement amount.
- (12) The graphic trace in the control axis superimposition may draw a path different from the machining program path.
- (13) When conducting an interlock to the superimposition axis, only the commands towards the superimposed axis will be interlocked. Movements accompanying the movement of the reference axis will not be interlocked.

- (14) The superimposition will be canceled if it falls into an emergency stop.
- (15) The superimposition will be canceled if a servo OFF signal is entered to the superimposed axis/reference axis in the control axis superimposition. Make sure that the superimposed axis/reference axis is stopped before entering a servo OFF signal. To confirm the axis stop, check if axis movement + / - signals are turned off.
- (16) Start/end of control axis superimposition toward a part system containing the superimposed axis/reference axis must be issued after all the axes are stopped. To stop all the axes, note that a manual command, chopping, and other factors which is not a machining program may cause an axis to move.
- (17) A command deceleration check will be performed to a superimposed axis command even when an in-position check is valid.
- (18) All axes in-position signal of the part system containing a superimposed axis will be turned on if the movements commanded by the part system are completed, even when the superimposed axis is moving according to the reference axis.
- (19) The reset signals for the part systems containing the superimposed axis and reference axis must be input at the same time.
- (20) If the address ,P of G126 is commanded from an axis which does not contain the superimposed axis, the setting of workpiece zero point by the address ,P will become valid from the block which is next to the block being executed in the part system containing the superimposed axis.

[1st part system]		[2nd part system]	
:		:	
:		:	
!L1 ;		!L1 ;	
G126 Z2=Z1 ,P20. ;	Superimposing operation		
!L2 ;		!L2 ;	Execute G126 while timing synchronization operation
G00 X46. Z2. ;	Superimposing operation	G01 X-10 F100 ;	From this block, the setting of the workpiece zero point becomes valid
G01 Z-50. F0.1 ;		Z60. ;	
X54. ;		:	
:		:	
:		:	
:		:	
!L3 ;		!L3 ;	
G126 Z2 ;	Superimposing operation		
!L4 ;		!L4 ;	Execute G126 while timing synchronization operation
:		G01 X-10 F100 ;	From this block, the workpiece zero point returns to the original point.
:		:	

- (21) To cancel the superimposition, reset the G92 compensation amount of the superimposed axis to the amount before the superimposition start. The G92 compensation amount set by the G50 (G92) command during the superimposition will be invalidated when canceling the superimposition.
- (22) When conducting a machine lock to the superimposition axis, only the commands towards the superimposed axis will be machine locked. Movements accompanying the movement of the reference axis will not be machine locked.
- (23) A program error (P520) will occur if the superimposition start command is executed to an axis whose parameter "#1205 G0bdcc Acceleration and deceleration before G0 interpolation" is not "0".
- (24) The deceleration check will be commanded for the superimposed axis even if the programmable imposition check is commanded.

- (25) When the synchronous tapping is performed with the superimposed axis, set the following values to the gain for the synchronous tapping and the normal gain.
- #2203 SV003(PGN) = #2017 tap_g (#2249 SV049(PGN1sp))
 - #2204 SV004(PGN2) = #2250 SV050(PGN2sp)
 - #2257 SV057(SHGC) = #2258 SV058(SHGCsp)
- (26) When the high-speed synchronous tapping is performed with the superimposition axis, an in-position check is performed at R point as in-position check cannot be canceled for traveling from the initial point to R point because of the data communication specification between NC and the drive unit.

13.23 Control Axis Synchronization Across Part System I ; G125



Function and purpose

Synchronization control enables an arbitrary control axis in the other part system to move in synchronization with the movement command assigned to an arbitrary control axis.

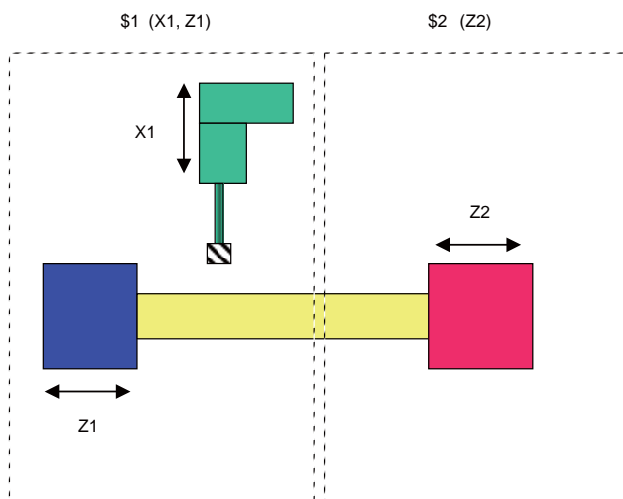
The direction in which the axis is to move synchronously can also be reversed using a parameter.

Reference axis	Axis to be commanded to move as the reference for synchronous movement.
Synchronization axis	Axis whose movement is synchronized with base axis.

There are two types of control axis synchronization across part systems depending on the command method: Control axis synchronization across part system I (G command) and control axis synchronization across part system II (PLC signal).

Set the basic specification the parameter "#1280 ext1602/bit5" to choose which method to use.

The following is the example of machining while synchronizing Z1 axis with Z2 axis in 1st part system (X1, Z1) and 2nd part system (Z2).



While in synchronization, synchronous axis synchronizes and moves against the reference axis regardless of conducting automatic operation or manual operation.



Command format

Synchronization start

G125 Synchronous axis name = Reference axis name ;

Synchronous axis name	Synchronous 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 Synchronous axis name ;

Synchronous axis name	Name of the synchronus axis to be ended (Axis name set in the parameter "#1022 axname2")
-----------------------	--



Detailed description

- (1) When G125 is commanded, two part systems which include the reference axis and the synchronous axis wait for the completion of currently executing block. After confirming the smoothing for all axes reached zero for both part systems, start/end the synchronization.
In synchronous/superimposition control signal turns ON for the reference axis and the synchronous axis during synchronization.
- (2) Set the synchronous direction with the parameter "#2087 syncnt/bit0". In synchronization start command, if a code "-" is added to the beginning of the reference axis name, the synchronous direction will be reversed.
- (3) G125 can be commanded from any part system.
- (4) By re-commanding the reference axis in synchronization as the reference axis, multiple synchronous axes are given to per reference axis.

Synchronization deviation detection

The synchronization deviation is detected during the control axis synchronization across part systems. The deviation of the synchronous 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 synchronous axis. If the synchronization deviation amount exceeds the value of parameter "#2024 synerr", the error "M01 operation error 0051" will occur. Cancel the control axis synchronization across 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

(1) When the number of part systems is two.

<Axis configuration>

1st part system: X1, Z1

2nd part system: X2, Z2

Below is the example of machining while executing the control axis synchronization across part systems.

Reference axis: Z1, synchronous axis: Z2

[1st part system]		[2nd part system]	
G28XZ;		G28XZ;	
G00 X100. Z100.;		G00 X60. Z50.;	
!L1;	(timing synchronization operation)	!L1;	(timing synchronization operation)
G125 Z2 = Z1;	(Start synchronizing Z2 to Z1)		
!L2;	(timing synchronization operation)	!L2;	(timing synchronization operation)
G01 X200. F500;		G01 X40. F300;	
G01 Z300.;		:	
G01 X100.;		:	
!L3;	(timing synchronization operation)	!L3;	(timing synchronization operation)
G125 Z2;	(End the synchronization between Z1 and Z2)		
!L4	(timing synchronization operation)	!L4	(timing synchronization operation)

(2) When number of part systems is three.

<Axis configuration>

1st part system: X1,Z1

2nd part system: X2,Z2

3rd part system: X3,Z3

Below is the example of machining while executing the control axis synchronization across part systems.

Reference axis: Z1, synchronous axis: Z2

[1st part system]		[2nd part system]		[3rd part system]	
:		:		:	
:		:		:	
!2!3 L1;	(timing synchronization operation)	!1!3 L1;	(timing synchronization operation)	!1!2 L1;	(timing synchronization operation)
				G125 Z2=Z1;	(Start synchronizing Z2 to Z1)
!2!3 L2;	(timing synchronization operation)	!1!3 L2;	(timing synchronization operation)	!1!2 L2;	(timing synchronization operation)
G01 Z50. F100;		G01 X-10. F100;		G01 Z50. F100;	
X4. Z4.;		:		G00 X10. Z20.;	
:		:		:	
:		:		:	
!2!3 L3;	(timing synchronization operation)	!1!3 L3;	(timing synchronization operation)	!1!2 L3;	(timing synchronization operation)
				G125 Z2;	(End the synchronization between Z1 and Z2)
!2!3 L4;	(timing synchronization operation)	!1!3 L4;	(timing synchronization operation)	!1!2 L4;	(timing synchronization operation)



Relation with other functions

- (1) Reference position return
When the reference position return is commanded to the reference axis, the synchronous axis only synchronizes with the reference axis movement and moves, and the reference position return is not carried out.
- (2) Emergency stop/reset
When the emergency stop occurs during the control axis synchronization across part systems, the control axis synchronization across part system will be canceled.
When the system is reset during the control axis synchronization across part systems, select whether to cancel the control axis synchronization across part systems by the parameter "#1280 ext16/bit3".
Input reset to all part systems which are related to the synchronization at a time.
- (3) Stroke end, stored stroke limit, chuck barrier/tail stock barrier
When the reference axis enters stroke end (over travel), stored stroke limit, or chuck barrier/tail stock barrier, the synchronous axis synchronizes with the reference axis and stops. On this occasion, the movement command is enabled for other axes, including the synchronous axis of the part system.
When the synchronous axis enters stroke end (over travel), stored stroke limit, or chuck barrier/tail stock barrier, the reference axis will be stopped to keep the synchronization state. Furthermore, if the automatic operation is conducted, all axes, including the reference axis, will stop.
- (4) Machine lock
When the reference axis is in automatic machine lock or manual machine lock, the synchronous axis is also in machine lock.
- (5) Servo OFF
When the state of the reference axis is servo OFF, that of the synchronous axis is also servo OFF.
When the parameter "#1064 svof" is set to "1" (correct the error), do not make them servo OFF during synchronization. If they are servo OFF, the synchronous relationship between the reference axis and synchronous axis cannot be maintained by the tolerable correction operation.
- (6) Inclined axis control
The control axis synchronization across part systems of the basic axis in the inclined axis control and the inclined axis are listed below.

The control axis synchronization across part systems for inclined axis ("#2071 s_axis=1")	The control axis synchronization across part systems for the reference axis ("#2071 s_axis=2") in the inclined axis control
× (Note2)	○ (Note3)

○ : Available, ×: Not available

- (Note 1) Above operations are determined only by the setting value of the parameter "#2071 s_axis" and the control axis synchronization across part systems command regardless of "inclined axis control valid" signal (YC35).
- (Note 2) When the synchronization start command is given while the control axis synchronization across part systems is disabled, the operation differs depending on the control axis synchronization across part systems I or the control axis synchronization across part systems II. Refer to "Precautions" for details.
- (Note 3) While the basic axis of the inclined axis control is in the control axis synchronization control, the manual movement command and the automatic movement command cannot be given to the corresponding inclined axis. If the manual movement command or the automatic movement command is given to the inclined axis, the error "M01 operation error 0005" will occur.



Precautions

Precautions for starting/ending the synchronization

- (1) Command G125 in an independent block. If not, a program error (P33) will occur.
- (2) The control axis synchronization across part systems by G125 command is valid only when the parameter "#1280 ext16/bit5" is set to "1". If G125 is commanded when "ext16/bit5" is set to "0", a program error (P610) will occur.
- (3) Specify the axis name set with the parameter "#1022 axname2". If the axis name which is not set with "axname2" is specified, a program error (P521) will occur.
- (4) The axis name specified in G125 needs to be 2 digits. Set the parameter "#1022 axname2" with two digits.
- (5) If an axis other than synchronous axis is specified at the synchronization end command, that command will be ignored.
- (6) If the same axis name is given to the reference axis and the synchronous axis, a program error (P521) will occur.
- (7) Two axes in the same part system cannot be commanded as a reference axis and synchronous axis. If this type of command is given, a program error (P521) will occur.
- (8) Synchronization start/cancel by G125 command is invalid for graphic check and program restart.
- (9) When an axis is set as "inclined axis (setting value: 1)" by the parameter "#2071 s_axis", it cannot be specified as a reference axis or synchronous axis. If the synchronization start command (G125 command) is given like this, a program error (P521) will occur.
- (10) If G125 is commanded without the option, a program error (P39) will occur.

Other precautions

- (1) When the control axis synchronization across 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 the smoothing for all axes reached zero for both part systems.
- (2) As the state of two target part systems affects the control axis synchronization across part systems, be aware of the timing between the part systems. Therefore, command the timing synchronization operation before and after commanding the synchronization start/cancel.
- (3) Do not change the related parameters during the control axis synchronization across part systems.
- (4) The following parameters of two target axes for the control axis synchronization across part systems or two part systems which includes target axes need to match.
 - Control unit (#1004 ctrl_unit)
 - Input setting unit (#1003 iunit)
 - Diameter specification axis (#1919 dia)
 - Designation of the linear axis/linear type rotary axis/rotating type rotary axis (#1017 rot, #8213 Rotation axis type)
 - Rapid traverse rate (#2001 rapid)
 - Cutting clamp speed ("#2002 clamp)
 - Acceleration/deceleration time constant (#2004 G0tL, #2005 G0t1, #2007 G1tL, #2008 G1t1)
 - Acceleration and deceleration types (#2003 smgst)

As the acceleration/deceleration time constants between two part systems related to the synchronization need to match, the acceleration/deceleration time constant of the constant inclination acceleration/deceleration or the control axis superimposition cannot be used together with the function which calculates or switches automatically and this function.
- (5) For the part systems which relate to the control axis synchronization across part systems, set "#1200 G0_acc", "#1201 G1_acc" and "#1205 G0bdcc" to "0".
- (6) Regardless of conducting the automatic operation or the manual operation, the movement command cannot be issued for the synchronous axis during the control axis synchronization across part systems. If this type of command is given, the error "M01 operation error 1038" will occur.
- (7) The slave axis during the synchronization control cannot be commanded as a reference or synchronous axis for the control axis synchronization across part systems. If this type of command is given, the error "M01 operation error 1037" will occur.
- (8) The synchronous axis during the control axis synchronization across part systems cannot be commanded as a reference axis or synchronous axis. If this type of command is given, the error "M01 operation error 1037" will occur.
- (9) The reference axis during the control axis synchronization across part systems cannot be commanded as a synchronous axis. If this type of command is given, the error "M01 operation error 1037" will occur. However, a command which has multiple synchronous axes to one reference axis is possible.
- (10) The axis of the part system which includes the reference axis of the control axis superimposition or superimposition axis cannot be commanded as a reference axis or synchronous axis of the control axis synchronization across part systems. If this type of command is given, the error "M01 operation error 1037" will occur.
- (11) The target control axis at the start of the control axis synchronization across part systems should be in a state where the reference position return after tuning the power ON is complete or the absolute position is established. If the axis whose zero point is not established is commanded, the error "M01 operation error 1037" will occur.
- (12) The spindle/C axis whose servo is OFF cannot be commanded as a reference axis or synchronous axis. If this type of command is given, the error "M01 operation error 1037" will occur.

13.24 2-part System Simultaneous Thread Cutting Cycle



Function and purpose

The 2-part system simultaneous thread cutting cycle function allows 1st part system and 2nd part system to perform thread cutting simultaneously for the same spindle.

The 2-part system simultaneous thread cutting cycle has two commands; the command (G76.1) for simultaneously cutting threads in two places, which is known as the "2-part system simultaneous thread cutting cycle I" and the command (G76.2) for simultaneously cutting a thread by two part systems, which is known as the "2-part system simultaneous thread cutting cycle II".

2-part system synchronous thread-cutting cycle I and II enable chamfering.

Refer to "13.1.2 Thread Cutting Cycle ; G78" for details of chamfering.

13.24.1 2-part System Simultaneous Thread Cutting Cycle Parameter Setting Command ; G76



Command format

G76 Pmra QΔdmin Rd ;

Address		Significance
P	m	Number of cutting passes for finishing ... Reversible parameter "#8058 G76 TIMES" is also available for setting.
	r	Chamfering amount ... Reversible parameter "#8014 CDZ-VALE (L system only)" is also available for setting. This sets the chamfering width based on the thread lead 1 across a range from 0.0 to 9.9 with a 2-digit integer with the decimal point omitted. (00 to 99)
	a	Tool nose angle (thread angle) ... Reversible parameter "#8059 G76 ANGLE" is also available for setting. This selects the angle from 0° to 99° and commands the value in two digits.
Q	Δdmin	Minimum cut amount If the calculated cut amount is smaller than Δdmin, it is clamped by Δdmin. (When omitted, operations differ depending on #1222/bit4 setting.)
R	d	Finishing allowance ... Reversible parameter "#8057 G76 LAST-D" is also available for setting.

Parameters for thread cutting are set with this command.

Command in the block just before 2-part system simultaneous thread cutting cycle I/II are commanded.

(Note) A reversible parameter enables to use parameter setting value without issuing a program command and also, the value can be changed by the program command.



Detailed description

- (1) The data is set in machining parameters m: #8058, r: #8014, a: #8059 and d: #8057 for each part system.
- (2) Issue the command for each part system.
- (3) The parameter setting command can be omitted when "#1222 aux06/bit5" is set to "1".
- (4) When "#1265 ext01/bit0" is "1", MITSUBISHI CNC special format will be applied. Therefore, program error (P33) will occur when the parameter setting commands.
- (5) If the parameter setting command is omitted, the parameter setting values are used from #8014, #8057, #8058, #8059 settings. The minimum cut-in amount (Δdmin) follows the #1222/bit4 setting.

13.24.2 2-part System Simultaneous Thread Cutting Cycle I ; G76.1



Command format

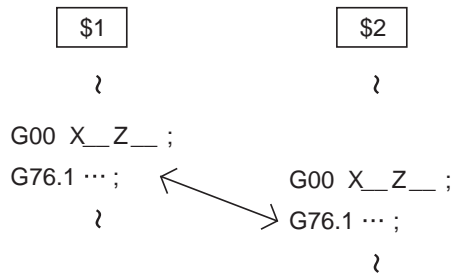
G76.1 X/U__ Z/W__ R__ P__ Q__ F__ ;

X/U	X-axis end point coordinates of thread section The X-axis coordinates of the end point at the thread section are commanded with absolute or incremental values.
Z/W	Z-axis end point coordinates of thread section The Z-axis coordinates of the end point at the thread section are commanded with absolute or incremental values.
R	Taper height component (radius value) at thread section ... A straight thread is created when R0 = 0.
P	Thread height ... This thread height is commanded with a positive radius value.
Q	Cut amount ... The cut amount for the first cutting pass is commanded with a positive radius value.
F	Thread lead

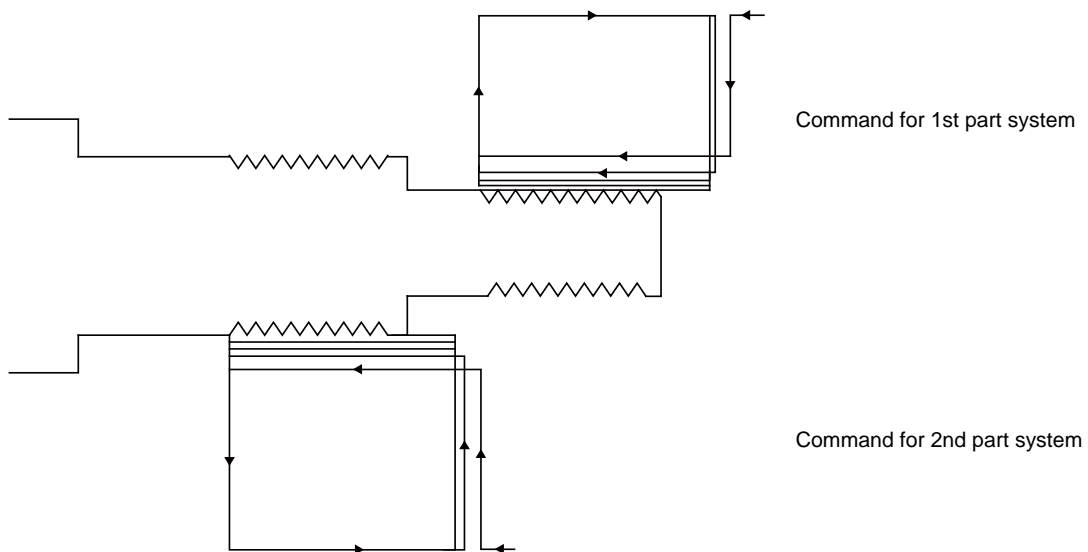


Detailed description

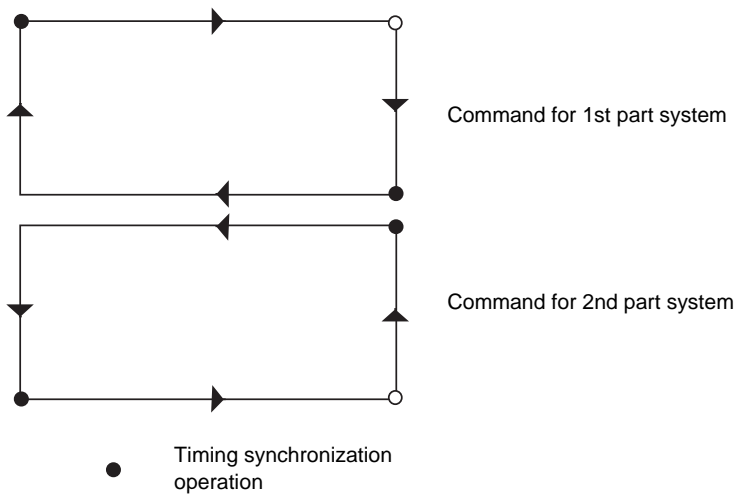
- (1) When G76.1 is issued by 1st part system and 2nd part system, waiting is done until the command is issued to another part system. The thread cutting cycle starts when the commands are aligned properly.



- (2) In the G76.1 cycle, G76.1 is issued simultaneously by 1st part system and 2nd part system, and the thread is cut in waiting at the start and end of thread cutting.



- (3) In one cycle, waiting is done at the start and end of the thread cutting.



- (4) The same precautions for thread cutting command (G33), thread cutting cycle (G78) and compound thread cutting cycle (G76) apply to this cycle.
- (5) As the threads are cut in two places by the G76.1 command, the various commands do not need to be the same. Each of them can be issued independently.
- (6) Thread cutting controls the Z axis position while tracking the spindle encoder rotation. Thus, the relative relation of the spindle position detected by the spindle encoder and the Z axis will change with the following elements.
- Z axis feedrate (spindle rotation speed * screw pitch)
 - Cutting feed acceleration/deceleration time constant
 - Position loop gain
- Thus, when cutting a multi-thread screw, the above conditions must be the same from the start to end of machining.
- (7) This function is unmodal. So it needs to be commanded each time.
- (8) G group 1 modal (G00, G01, G02, G03, G02.3, G03.3 etc.) is canceled temporarily during 2-part system simultaneous thread cutting.
- (9) Thread cutting start shift angle is not available in G76.1. A program error (P32) will occur if commanded.

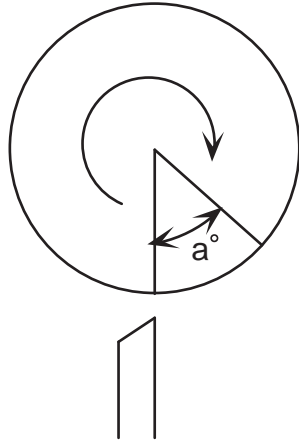
13.24.3 2-part System Simultaneous Thread Cutting Cycle II ; G76.2



Command format

G76.2 X/U__ Z/W__ R__ P__ Q__ Aa F__ ;

(1) Thread cutting start shift angle



The thread cutting command starts movement after waiting for the spindle encoder's one rotation synchronization signal. However, the start point can be delayed by a degree amount.

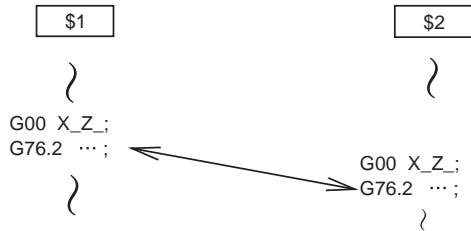
a : Thread cutting start angle

The meanings of the addresses other than A are the same as the 2-part system simultaneous thread cutting cycle I.

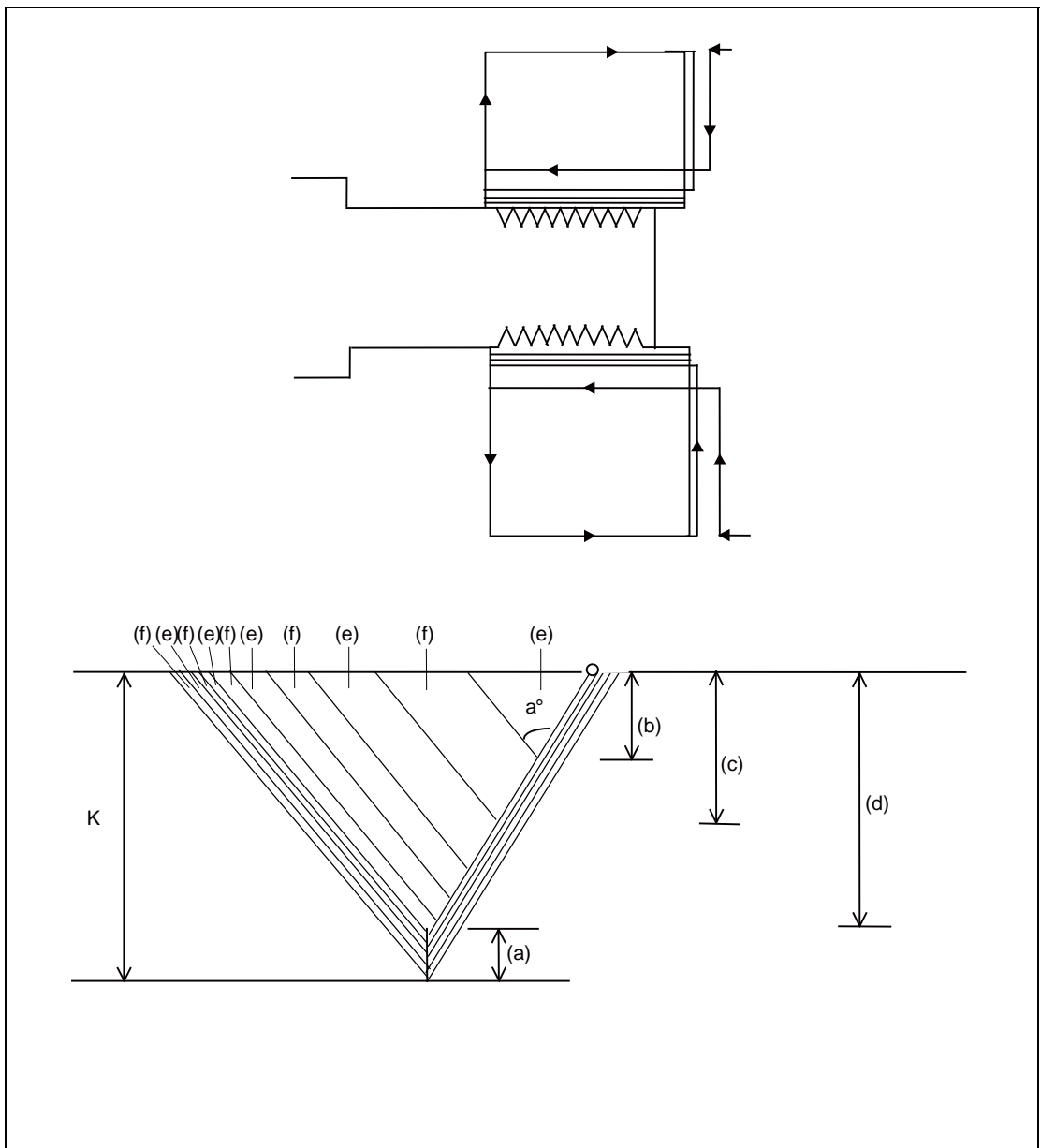


Detailed description

- (1) When G76.2 is issued by 1st part system and 2nd part system, waiting is done until the command is issued to another part system. The thread cutting cycle starts when the commands are aligned properly.

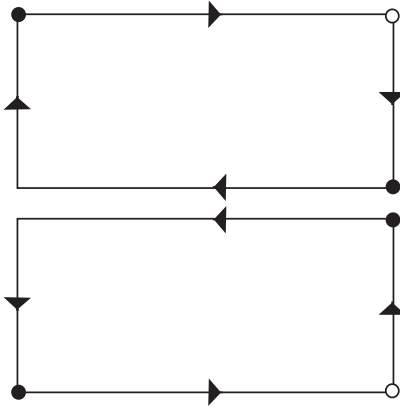


- (2) G76.2 assumes the same thread cutting, and deeply cuts in with the cutting amount using 1st part system and 2nd part system alternately.



- (a) Finishing allowance d (b) Δd (c) $\Delta d \times \sqrt{2}$
 (d) $\Delta d \times \sqrt{n}$ (e) Cutting with 1st part system (f) Cutting with 2nd part system

(3) In one cycle, waiting is done at the start and end of the thread cutting.



● Timing synchronization operation

(4) The same precautions for thread cutting command (G33), thread cutting cycle (G78) and compound thread cutting cycle (G76) apply to this cycle.

(5) G76.2 is the same thread cutting, so the various parameters, thread section, taper height, screw thread height, cutting amount and thread lead must be commanded to the same values for 1st part system and 2nd part system.

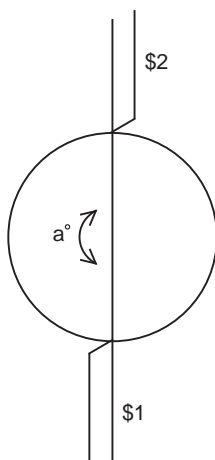
Note that the start shift angle can be commanded to match the thread cutting state.

(6) Thread cutting controls the Z axis position while tracking the spindle encoder rotation. Thus, the relative relation of the spindle position detected by the spindle encoder and the Z axis will change with the following elements.

- (a) Z axis feedrate (spindle rotation speed * screw pitch)
- (b) Cutting feed acceleration/deceleration time constant
- (c) Position loop gain

Thus, with G76.2 which is same thread cutting, the parameters must be set so that the conditions are the same for 1st part system and 2nd part system.

(7) Thread cutting start shift angle command



As shown on the left, if the 1st part system and 2nd part system blades oppose at 180 degree, the 1st part system and 2nd part system thread cutting start shift angle difference is set to 180 degree.

(Example)

```
$1
.
.
G76.2 X_ Z_ A0.;
.
.
```

```
$2
.
.
G76.2 X_ Z_ A180.;
.
.
```

\$1 : 1st part system

\$2 : 2nd part system

- (8) When G76.2 and G76.1 are commanded
The part systems, in which each are commanded, will carry out the G76.1 and G76.2 movements.
However, the part system in which G76.2 is commanded will assume that the other part system is using G76.2 when cutting the threads, so the thread grooves will not be guaranteed.
- (9) This function is unmodal. So it needs to be commanded each time.
- (10) G group 1 modal (G00, G01, G02, G03, G02.3, G03.3 etc.) is canceled temporarily during 2-part system simultaneous thread cutting.
- (11) Reset simultaneously for 1st part system and 2nd part system during 2-part system simultaneous thread cutting cycle II (G76.2).
If reset is individually executed for each part system, the part system is not in automatic operation. Thus, the other part system keeps moving according to #1279 setting.

13.25 2-part System Simultaneous Thread Cutting Cycle (MITSUBISHI special format) ; G76.1,G76.2



Function and purpose

The 2-part system simultaneous thread cutting cycle function allows 1st part system and 2nd part system to perform thread cutting simultaneously for the same spindle.

The 2-part system simultaneous thread cutting cycle has two commands; the command (G76.1) for simultaneously machining threads in two places, which is known as the "2-part system simultaneous thread cutting cycle I" and the command (G76.2) for simultaneously cutting a thread by two part systems, which is known as the "2-part system simultaneous thread cutting cycle II".

When using the MITSUBISHI CNC special format (#1265 ext01/bit0 ON), some addresses differ from the normal format. This section gives details on the formats different from the normal ones.

Refer to section "2-part System Simultaneous Thread Cutting Cycle" for details on the 2-part system simultaneous thread cutting cycle.



Command format

G76.1 X/U__ Z/W__ I__ K__ D__ F__ A__ ;

G76.2 X/U__ Z/W__ I__ K__ D__ Q__ F__ A__ ;

X/U	X-axis end point coordinates of thread section The X-axis coordinates of the end point at the thread section are commanded with absolute or incremental values.
Z/W	Z-axis end point coordinates of thread section The Z-axis coordinates of the end point at the thread section are commanded with absolute or incremental values.
I	Taper height component (radius value) at thread section A straight thread is created when I0 = 0.
K	Thread height This is commanded by a positive radius value.
D	Cut amount The cut amount for the first cutting pass is commanded with a positive radius value.
Q	Thread cutting start shift angle
F	Thread lead
A	Tool nose angle (thread angle)

(Note 1) This is 1 block command. It is not necessary to command "G76P__Q_R_" immediately before.



Detailed description

Comparison of normal format and MITSUBISHI CNC special format

Normal format	MITSUBISHI CNC special format	Remarks
(1)G76 Pmra Q_ R_; (2)G76.1/G76.2 X_ Z_ R_ P_ Q_ A_ F_;	G76.1 /G76.2 X_ Z_ I_ K_ D_ Q_ F_ A_;	Command format
(2)X/U	X/U	X-axis end point coordinates of thread section
(2)Z/W	Z/W	Z-axis end point coordinates of thread section
(2)R	I	Taper height component (radius value) at thread section
(2)P	K	Thread height
(2)Q	D	Cut amount
(2)A	Q	Thread cutting start shift angle
(2)F	F	Thread lead
(1) Q [When omitting : #1222/bit4]	None ["0" fixed]	Minimum cut amount
(1) Pa [Reversible parameter #8059]	A [Reversible parameter #8059]	Tool nose angle (thread angle)
(1) Pm [Reversible parameter #8058]	Parameter #8058	Number of cutting passes for finishing
(1) Pr [Reversible parameter #8014]	Parameter #8014	Chamfering amount
(1) R [Reversible parameter #8057]	Parameter #8057	Finishing allowance

(Note) A reversible parameter enables to use parameter setting value without issuing a program command and also, the value can be changed by the program command.



Relation with other functions

The modals must be set as shown below when commanding 2-part system simultaneous thread cutting.

Function	G code
Cylindrical interpolation cancel	G07.1
Polar coordinate interpolation cancel	G13.1
Balance cut OFF	G14
Tool nose radius compensation cancel	G40
Mirror image by parameter setting	cancel
Mirror image by external input	cancel
Polygon machining mode cancel	G50.2
Constant surface speed control mode cancel	G97



Precautions

- (1) The switch of MITSUBISHI CNC special format and normal format can be selected by setting "#1265/bit0". (0: normal format, 1: MITSUBISHI CNC special format)
- (2) A program error (P33) will occur if a G76 P__Q_R_ command or a normal format is commanded when MITSUBISHI CNC special format is selected.

13.26 Axis Name Switch ; G111



Function and purpose

This function switches the commanded axis and the control axis.

When using a function, such as the hole drilling cycle (G88), that can be commanded to the limited axis this function can be used to give commands to axes that cannot be commanded with the normal command methods.



Command format

G111 Axis name 1 Axis name 2 ; ... Switching start command

Axis name 1/Axis name 2 | The axis which operates after switching (The axis name set in the parameter "#1013 axname")

G111 ; ... Switching end command



Detailed description

- (1) Axis name switch can be commanded simultaneously to several systems.
While changing the axis name, G111 cannot be re-commanded.
If re-commanded, a program error (P411) will occur.
Command G111 in an independent block. If it is commanded in the same block with other G code, a program error (P33) will occur.
- (2) G111 changes the command axis in the same part system.
During changing, "In axis name switch" signal turns ON.
If there is no commanded axis name, a program error (P32) will occur.
- (3) The plane selection command modal (G17, G18, G19) will not change.
The plane which corresponds to the modal command during G111 command will be selected automatically.

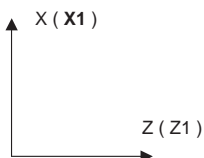
(Ex.)

Basic system parameter			Basic axis specification parameter			
#1026	Basis axis I	X	<1>	<2>	<3>	
#1027	Basis axis J	Y	#1013 axname	X	Y	Z
#1028	Basis axis K	Z	#1022 axname2	X1	Y1	Z1
#1029	Parallel axis I					
#1030	Parallel axis J					
#1031	Parallel axis K					

(G18)

G18 plane

⋮
⋮
⋮
⋮
⋮
⋮

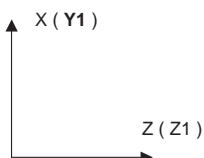


N10 G111 X Y ;

⋮
⋮

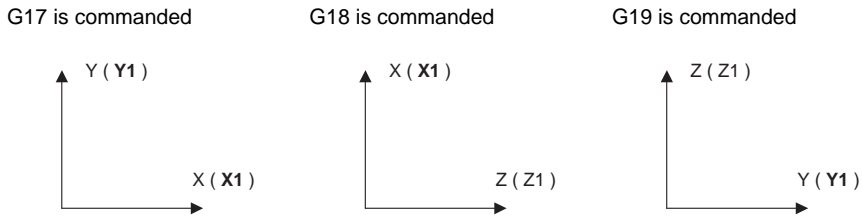
X-Y axis switch

G18 plane. However, X command is applied to 2nd axis by G111 command.

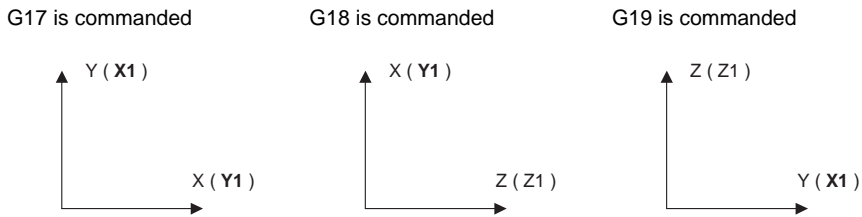


- (4) Plane selection can be commanded during the axis name switching.

(Ex.1) Plane where no axis switching is carried out

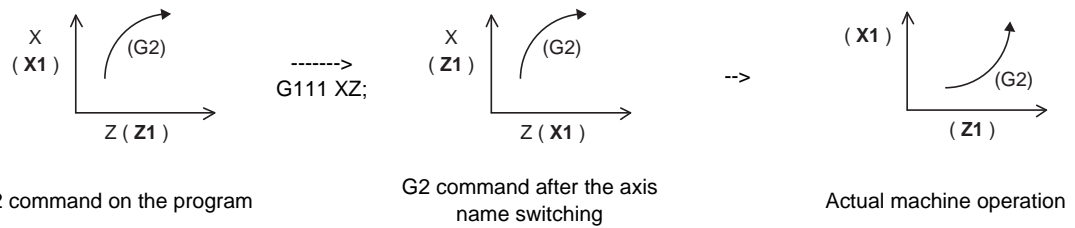


(Ex.2) Plane where axis name switching "G111 X Y ;" is carried out



- (5) Circular interpolation, nose R and fixed cycle are carried out on the selected plane.

(Ex.) Circular interpolation after switching the axis name



- (6) The plane selection will return to the state before G111 is commanded by canceling the axis name switching.

- (7) Tool length compensation and tool nose wear compensation are applied to the axis before axis name switching.

When X and Y are switched and then Y axis is moved by the X command, the compensation amount for Y axis is applied to Y axis and the compensation amount for X axis is not applied.

(Example)

[Compensation amount]		X	Y	Z
T1103 -> Tool nose wear compensation No.	"03"	0.01	0.02	0.03
Tool length compensation No.	"11"	0.1	0.2	0.3

[Program]

[Machine coordinate value]

[Program]	[Machine coordinate value]
(G111)	<X1> <Y1> <Z1>
:	
N10 T1103 ;	
N20 G90 G00 X10.0 Y20.0 Z30.0 ;	10.11 20.22 30.33
:	
N30 T0 ;	
N40 G00 X0 Y0 Z0 ;	
:	
N50 G111 X Y ;	
N60 T1103 ;	
N70 G00 X10.0 Y20.0 Z30.0 ;	20.11 10.22 30.33

- (8) For the machine specification whose absolute/incremental is switched by address, when the axis name switching is carried out, likewise the absolute address will be switched.

(Example)

	Control X axis	Control Y axis
G111 ;	X , U , I	Y , V , J
G111 X Y ;	Y , V , J	X , U , I

However, an address which does not affect the axis movement, such as X address of dwell, does not switch.

(Example)G4 X2. ;

G111 X Y ;

G4 X2. ; <- X address of dwell does not switch to the command by Y.

- (9) When an axis for which the axis name switch is carried out is a constant surface speed axis, the constant surface speed axis will be switched automatically.

When "G96 P_" is commanded, the axis No. is that of after the axis name is switched.

When the P command is omitted, the constant surface speed axis will follow the basic specifications parameter "#1181 G96_ax". However, when the constant surface speed axis whose P command is omitted is commanded after the axis name switching, the axis No. which is set by the parameter becomes that of after the axis name is switched.

(Example)

: <- Command "G96 P1" before "G111 X Y;" -> X (X1) is the constant surface speed axis

G111 X Y ;

: <- Command "G96 P1" after "G111 X Y;" -> X (y1) is the constant surface speed axis

- (10) The axis coordinate value and the tool length value which are read by variables are fixed regardless of the axis name switching.

Axis allocation by variables is not switched even during the axis name switching.

(Example)

	<1>	<2>	<3>
#1013 axname	X	Y	Z
#1022 axname2	X1	Y1	Z1

G111;

N10 G00 X100. Z200. Y300. ;

N20 #500 = #5021 ; <- Machine position coordinate value (100.) of <X1> is substituted into #500.

N30 #501 = #5022 ; <- Machine position coordinate value (200.) of <Z1> is substituted into #501.

N40 #502 = #5023 ; <- Machine position coordinate value (300.) of <Y1> is substituted into #502.

N50 G111 X Y ;

N60 #504 = #5021 ; <- Machine position coordinate value (100.) of <X1> is substituted into #504.

N70 #505 = #5022 ; <- Machine position coordinate value (200.) of <Z1> is substituted into #505.

N80 #506 = #5023 ; <- Machine position coordinate value (300.) of <Y1> is substituted into #506.



Program example

```
(Ex.)
G90 G00 ;
G111 X Y ;
G01 X100. ;          -> Y axis moves to 100.
G01 Y100. ;          -> X axis moves to 100.
G111 ;
G01 X0. ;            -> X axis moves to 0.
G01 Y0. ;            -> Y axis moves to 0.
```



Relation with other functions

Relation with fixed cycle

(1) Fixed cycle for drilling

Do not command the axis name switching during the fixed cycle for drilling. If G111 is commanded during the fixed cycle for drilling, a program error (P411) will occur.

The following operation is carried out during the longitudinal hole drilling fixed cycle.

When "Longitudinal hole drilling axis selection" (YCD4) is turned ON, axis switching including Y axis becomes temporarily invalid during the longitudinal hole drilling fixed cycle (G87, G88, G89, G88.1).

When the longitudinal hole drilling fixed cycle is canceled, the switching becomes valid.

If the switching does not include Y axis, the switching is valid even during the longitudinal hole drilling fixed cycle.

"In axis name switch" signal will not turn OFF even when the switching is temporarily invalid.

(Ex.) Axis configuration X (1st axis), Z (2nd axis), Y (3rd axis), and C (4th axis)

```
G28 X Z Y C ;
G111 X Y ;          (Axis name switching which includes the hole drilling axis)
G87 R-5. X10. Z-30. Y-40. F1000 ; ... Switching X axis and Y axis becomes invalid during the longitudinal hole
Z-40. Y-50. ;      drilling fixed cycle
:                  X: 1st axis
:                  Z: 2nd axis
:                  Y: 3rd axis
G80 ;
G01 X15. Z25. Y35. ; ... Switching X axis and Y axis becomes valid after canceling the longitudinal
:                  hole drilling fixed cycle
:                  X: 3rd axis
:                  Z: 2nd axis
:                  Y: 1st axis
```

(2) Fixed cycle for turning machining

If G111 is commanded during the fixed cycle for turning machining (G77, G78, G79), the fixed cycle is canceled.

(3) Compound type fixed cycle for turning machining

Do not command the axis name switching for the finished shape program designated with G71, G72 or G73 block. If G111 is commanded to the finished shape program, a program error (P411) will occur.

Relation with other functions

- (1) Milling interpolation (G12.1/G13.1)
Do not command the axis name switching during the milling interpolation mode.
If G111 is commanded during the milling mode, a program error (P411) will occur.
- (2) Constant surface speed control (G96, G97) (including clamp)
Do not command the axis name switching during the constant surface speed control mode.
If G111 is commanded during the constant surface speed control mode, a program error (P411) will occur.
(Example) Part system of axis no. (1) X, (2) Z, (3) Y and (4) C
G96 S300 P1
G111 X Y
G01 U-30.
The constant surface speed control is possible after the axis name switching.
- (3) Tool nose radius compensation (G41/G42/G40), tool nose radius compensation automatic direction identification (G46/G40)
Do not command the axis name switching during the tool nose radius compensation mode.
If G111 is commanded during the tool nose radius compensation mode, a program error (P411) will occur.
- (4) Workpiece coordinate selection (6 sets) (G54 to G59), external workpiece coordinate offset
Coordinate system offset amount is applied to the axis which is specified by the parameter regardless of the axis name switching.
- (5) Plane selection
Axis name switching command does not change the plane selection's modal.
It automatically selects the plane which corresponds to the modal command when G111 is commanded.
- (6) NC reset (Reset 1/2, reset & rewind)
When NC is reset, the axis name switching is automatically canceled.
- (7) Mixed control (Cross axis control)
Do not command the axis name switching during the mixed control (cross axis control) mode.
If G111 is commanded during the mixed control (cross axis control) mode, a program error (P411) will occur.
If the mixed control (cross axis control) is commanded during axis name switch, the error "M01 operation error 1035" will occur.
- (8) Polar coordinate interpolation
Do not command the axis name switching during the polar coordinate interpolation mode.
If G111 is commanded during polar interpolation mode, a program error (P411) will occur.
- (9) Cylindrical interpolation
Do not command the axis name switching during the cylindrical interpolation mode.
If G111 is commanded during the cylindrical interpolation mode, a program error (P411) will occur.

- (10) Workpiece offset input (G10 L2)
Do not command the workpiece offset input during G111 modal.
If the workpiece offset input is commanded during G111 modal, a program error (P421) will occur.
- (11) Tool compensation input (G10 L10)
Do not command the tool compensation input during G111 modal.
If the tool compensation input is commanded during G111 modal, a program error (P421) will occur.
- (12) Tool nose wear compensation input (G10 L11)
Do not command the tool nose wear compensation input during G111 modal.
If the tool nose wear compensation input is commanded during G111 modal, a program error (P421) will occur.
- (13) Workpiece coordinate offset amount setting (G10 L20)
Do not set the workpiece coordinate offset amount during G111 modal.
If the workpiece coordinate offset amount is set during G111 modal, a program error (P421) will occur.
- (14) Programmable current limitation (G10 L14)
Do not command the programmable current limitation during G111 modal.
If the programmable current limitation is commanded during G111 modal, a program error (P421) will occur.
- (15) Workpiece coordinate preset
This function presets the workpiece coordinate system shifted with the program command or manual operation to the workpiece coordinate system which is offset by the workpiece coordinate offset amount from the machine zero point by the program command (G92.1/G50.3).
- (16) Start point designation synchronization
When the axis name switching is carried out at the other part system after start point designation synchronization is commanded, the part systems can be waited at the axis before switching.
Command start point designation synchronization after the axis name switching completed.



Precautions

- (1) Only two axes can be combined for the commanded axes when using the axis name switch command. A program error (P33) will occur if one independent axis or three or more axes are commanded.
- (2) The settings such as the diameter value and radius value of the two axes used for axis name switch cannot be interchanged. Only the axis name is interchanged.
- (3) Machining program, including the axis name switch, cannot perform program restart search.
If the axis name switch is commanded (G111) during the program restart search, a program error (P49) will occur.
- (4) When an address check is validated (set "#1227 aux11/bit4" to "1"), do not command the axis name switch which has no command value.
If the axis name switch is commanded without the command value, a program error (P33) will occur.

Coordinate System Setting Functions

14.1 Coordinate Words and Control Axes



Function and purpose

In the case of a lathe, axis names (coordinate words) and directions are defined as follows.

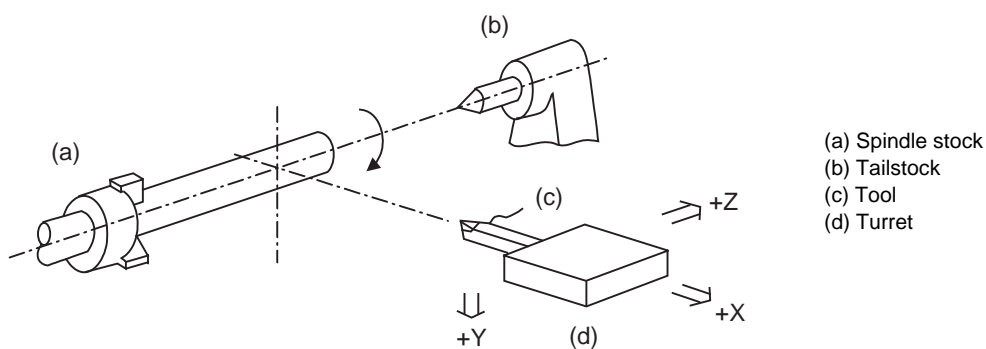
The axis at right angles to the spindle

Axis name: X axis

The axis parallel to the spindle

Axis name: Z axis

Coordinate axes and polarities

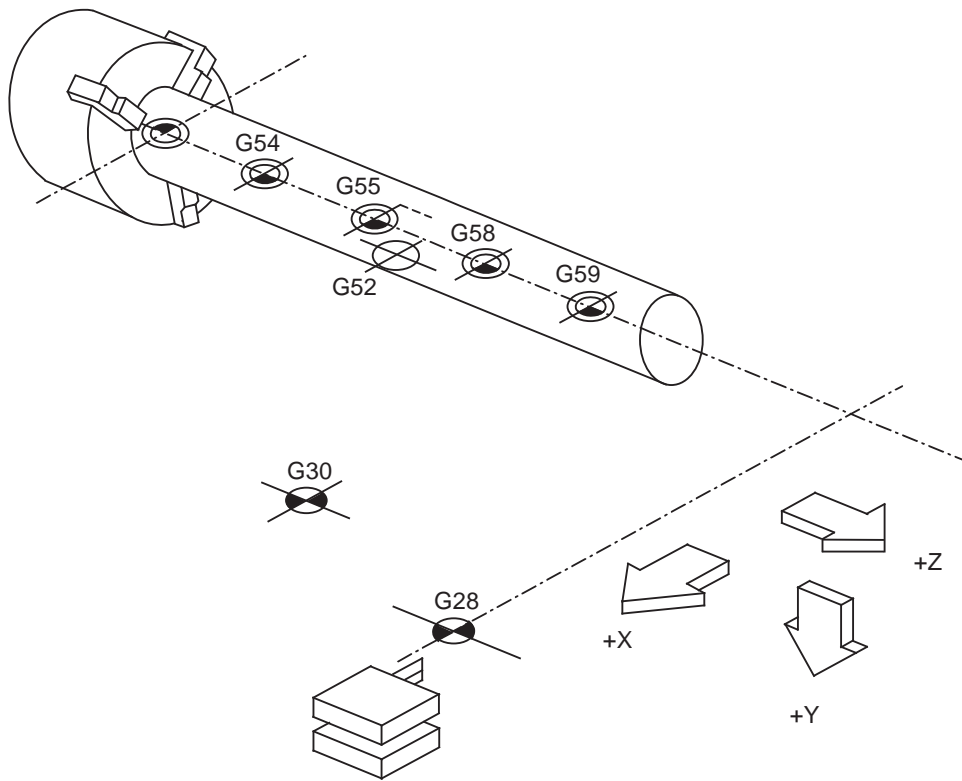






Since coordinates based on the right hand rule are used with a lathe, in the above figure, the positive direction of the Y axis which is at right angles to the X-Z plane is downward.

Note that a circular on the X-Z plane is expressed as clockwise or counterclockwise as seen from the forward direction of the Y axis.

(Refer to the section on circular interpolation.)

Relationship between coordinates



-  Reference position
-  Basic machine coordinate
-  Workpiece coordinate zero points
-  Local coordinate zero point

14.2 Basic Machine, Workpiece and Local Coordinate Systems



Function and purpose

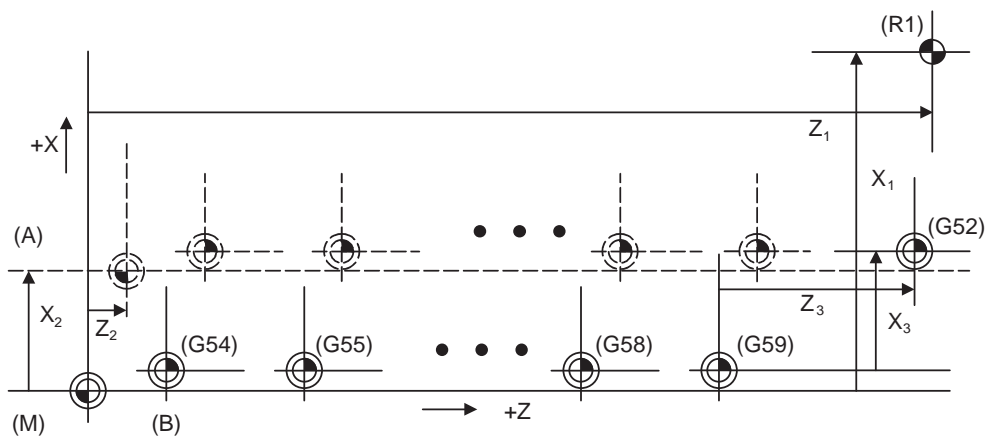
The basic machine coordinate system is fixed in the machine and it denotes that position which is determined inherently by the machine.

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).



- (A) Hypothetical machine coordinate system (G92 shift)
- (B) Machine zero point
- (G54) Workpiece coordinate system 1
- (G55) Workpiece coordinate system 2
- (G58) Workpiece coordinate system 5
- (G59) Workpiece coordinate system 6
- (G52) Local coordinate system
- (R1) 1st reference position
- (M) Basic machine 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".

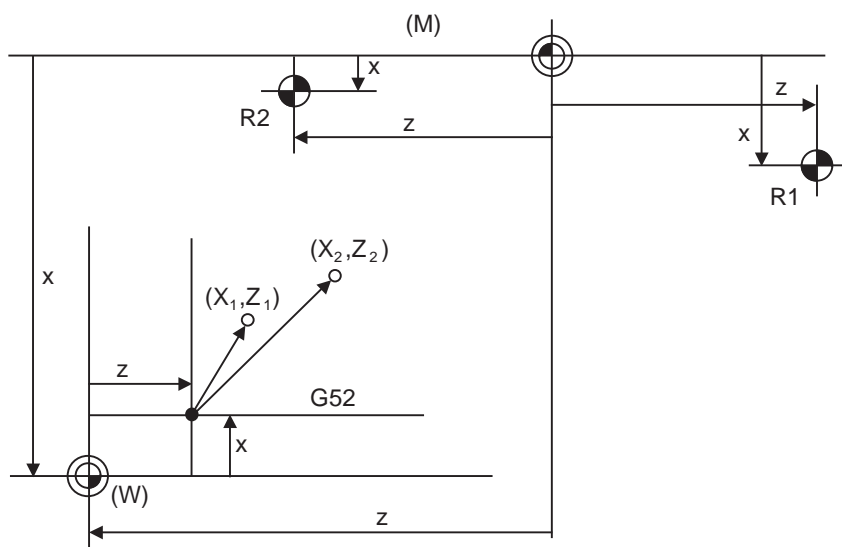
14.3 Machine Zero Point and 2nd Reference Position (Zero point)



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) point return.

2nd reference position (zero point) relates to the position of the coordinates which 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

(R1) 1st reference position

(R2) 2nd reference position

(W) Workpiece coordinate systems (G54 to G59)

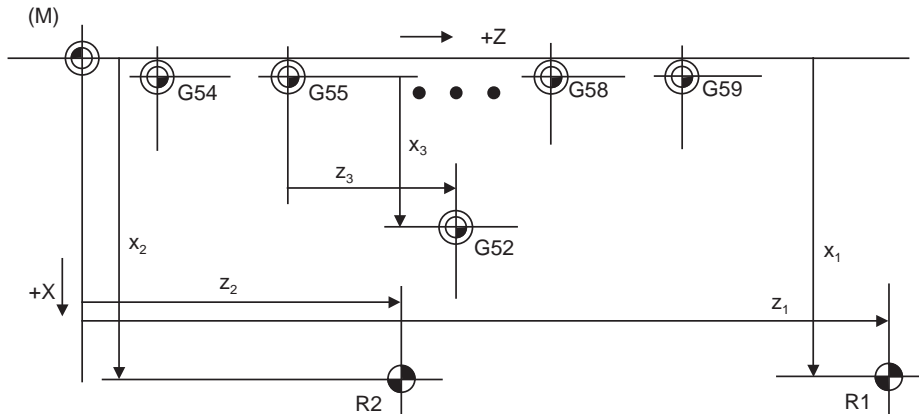
14.4 Automatic Coordinate System Setting



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 which have been set above.



(M) Basic machine coordinate system

(R1) 1st reference position

(R2) 2nd reference position

(G54) Workpiece coordinate system 1

(G55) Workpiece coordinate system 2

(G58) Workpiece coordinate system 5

(G59) Workpiece coordinate system 6

(G52) Local coordinate system



Detailed description

- (1) The coordinate systems created by this function are as follow:
 - (a) Basic machine coordinate system
 - (b) 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

1. 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.

14.5 Basic Machine Coordinate System Selection ; G53



Function and purpose

The tool is moved to the position commanded on the basic machine coordinate system, using the G53 command, the feed mode command (G01 or G00) and the coordinate command that follow them.



Command format

G53 G00 X__ Z__ α __;

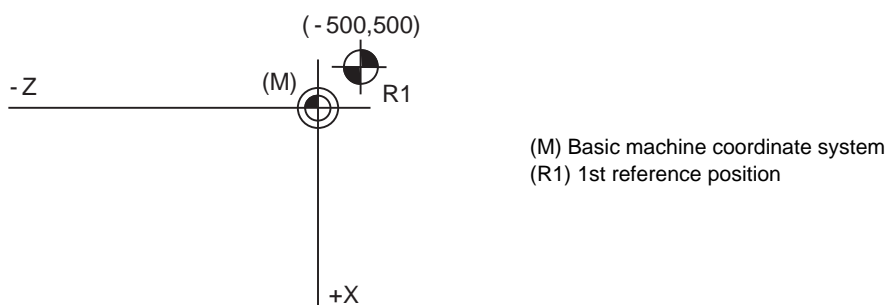
G53 G00 U__ W__ β __;

α	Additional axis
β	Incremental command of the 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) When an incremental value is issued (U, W, β), the axis will move with the incremental value within the selected coordinate system.
- (5) The 1st reference position coordinate value indicates the distance from the basic machine coordinate system zero point to the reference position (zero point) return position.
- (6) The G53 command will move with cutting feedrate or rapid traverse following command modal.



1st reference position coordinate value: X = -500 and Z=+500

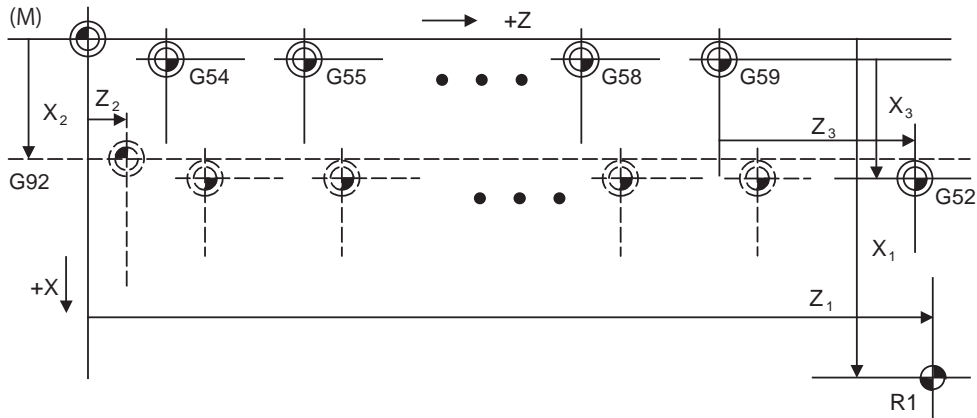
14.6 Coordinate System Setting ; G92



Function and purpose

This function places the tool at the desired position, and the coordinate system is set by assigning the coordinate system setting command G92 at that position.

This system can be set as desired though normally the X and Y axes are set so that the workpiece center serves as the zero point and the Z axis is set so that the workpiece end serves as the zero point.



- (M) Basic machine coordinate system
- (G92) Hypothetical machine coordinate system (shifted by G92)
- (G54) Workpiece coordinate system 1
- (G55) Workpiece coordinate system 2
- (G58) Workpiece coordinate system 5
- (G59) Workpiece coordinate system 6
- (G52) Local coordinate system
- (R1) 1st reference position



Command format

G92 Xx2 Zz2 α α 2;

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



Detailed description

- (1) The basic machine coordinate system is shifted by the G92 command, the hypothetical machine coordinate system is created, and at the same time all workpiece coordinate systems 1 to 6 are also shifted.
- (2) When G92 and S or Q are assigned, the spindle clamp rotation speed is set. (Refer to the section on setting the spindle clamp rotation speed.)



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.

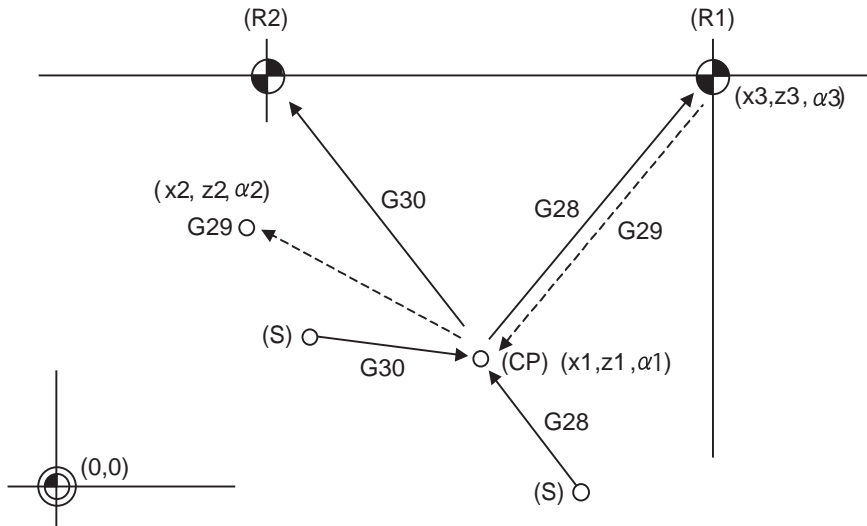
14.7 Reference Position (Zero point) Return ; G28,G29



Function and purpose

After the commanded axes have been positioned by G0, 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 G0 at the commanded position.



- (CP) Intermediate point
- (R1) 1st reference position
- (R2) 2nd reference position
- (S) Start point



Command format

G28 Xx1 Zz1 α α 1; ... Automatic reference position return

X, Z, α	Coordinate value of the intermediate point (α is an additional axis)
---------	--

G29 Xx2 Zz2 α α 2; ... Start point return

X, Z, α	Coordinate value of the end point (α is an additional axis)
---------	---



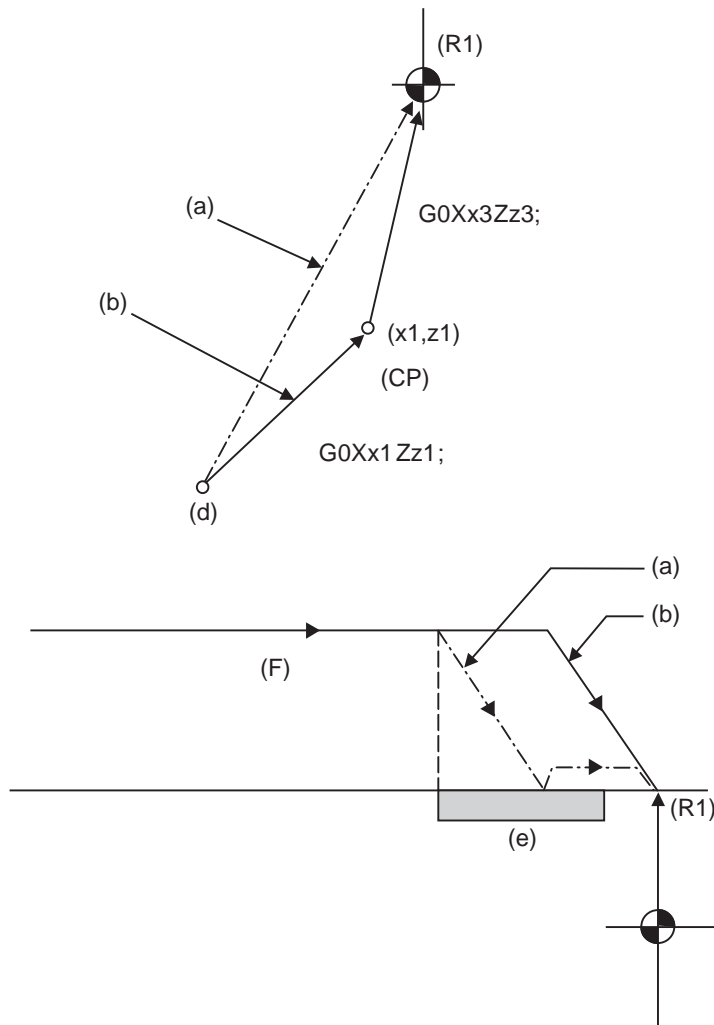
Detailed description

- (1) The G28 command is equivalent to the following:
 $G00 Xx1 Zz1 \alpha \alpha 1 ;$
 $G00 Xx3 Zz3 \alpha \alpha 3 ;$
 In this case, $x3$, $z3$ and $\alpha 3$ are the reference position coordinates and they are set by parameters "#2037 G53ofs" as the distance from the basic machine coordinate system zero point.
- (2) After the power has been switched on, the axes which 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. For the second and subsequent returns, the return is made at high speed to the reference position which was stored at the first time.
- (3) When reference position return is completed, the zero point arrival output signal is output and also #1 appears at the axis name line on the setting and display unit screen.
- (4) The G29 command is equivalent to the following:
 $G00 Xx1 Zz1 \alpha \alpha 1 ;$
 $G00 Xx2 Zz2 \alpha \alpha 2 ;$
 The rapid traverse (non-interpolation type) independent for each axis takes place.
 In this case, $x1$, $z1$ and $\alpha 1$ are the coordinate value of the G28 or G30 intermediate point.
- (5) Program error (P430) occurs when G29 is executed without executing automatic reference position (zero point) return (G28) after the power has been turned ON.
- (6) The intermediate point coordinate values ($x1$, $z1$, $\alpha 1$) of the positioning point are assigned by absolute/incremental value commands.
- (7) G29 is valid for either G28 or G30 but the commanded axes are positioned after a return has been made to the latest intermediate point.
- (8) The tool offset will be temporarily canceled during reference position return unless it is already canceled, and the intermediate point will be the compensated position.
- (9) The intermediate point can be ignored by parameter "#1091 Ignore middle point" setting.
- (10) Control from the intermediate point to the reference position is ignored for reference position return in the machine lock status. The next block is executed when the commanded axis reaches as far as the intermediate point.
- (11) Mirror image is valid from the start point to the intermediate point during reference position return in the mirror image mode and the tool will move in the opposite direction to that of the command. However, mirror image is ignored from the intermediate point to the reference position and the tool will move to the reference position.
- (12) Do not issue T commands in a block where G29 exists. Program error (P29) occurs if the workpiece compensation amounts differ between the current and the previous blocks.
- (13) When G28/G29/G30 is commanded in single block mode, if "#1279 ext15/bit6 Enable single block stop at middle point" is set to "1", single block stop at middle point will be performed; single block stop at middle point will not be performed if set to "0".
- (14) If the mode is switched to MDI mode or reference point return mode while in a single block stop at middle point, "M01 Operation Error 0013" occurs.
- (15) If the NC is reset while in a single block stop at middle point, the interim position for G29 start position return will not be updated.
- (16) If a miscellaneous function is commanded in the same block as G28/G29, the miscellaneous function completion waiting point will be the end of commanded movement, instead of the interim position.
- (17) If the PLC interrupt operation is operated while in a single block stop at intermediate point, "M01 Operation Error 0129" occurs.



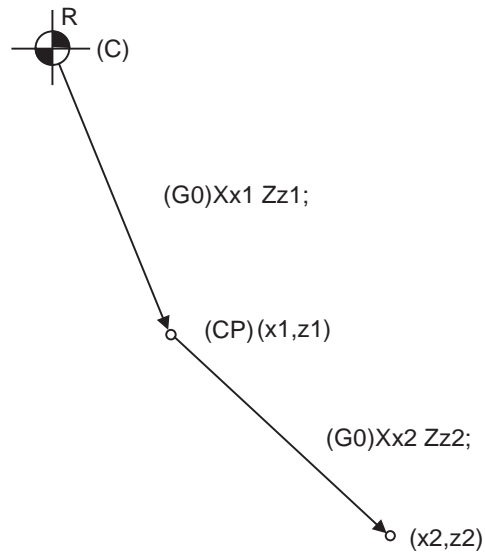
Program example

(Example 1) G28 Xx1 Zz1 ;



- (a) 1st operation after power has been turned ON
- (b) 2nd and subsequent operations
- (d) Return start position
- (e) Near-point dog
- (F) Rapid traverse rate
- (CP) Intermediate point
- (R1) Reference position (#1)

(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)

:

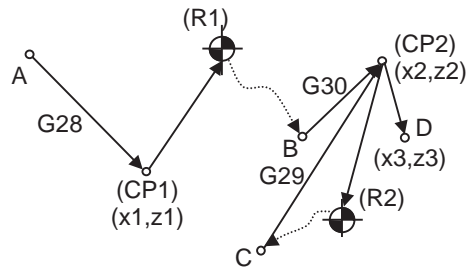
G30 Xx2 Zz2 ;

: (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)

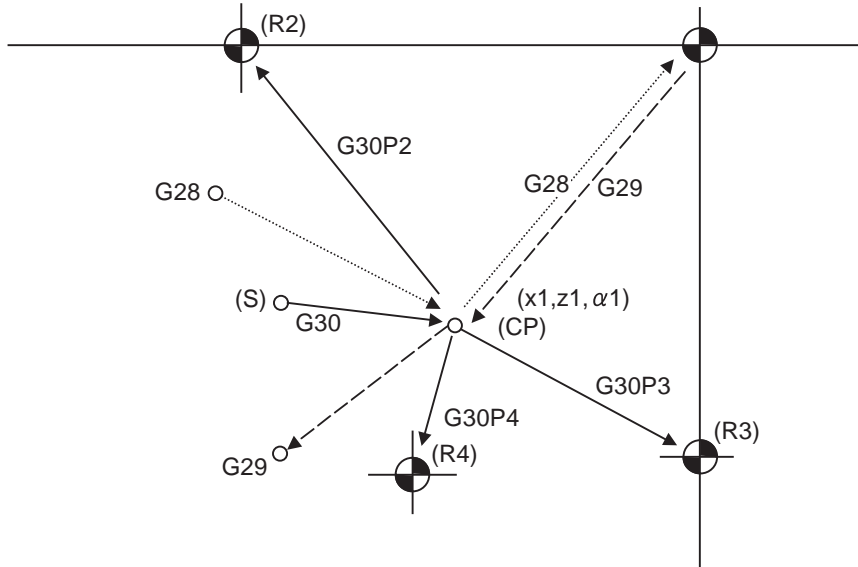
(R2) 2nd reference position (#2)

14.8 2nd, 3rd, and 4th Reference Position (Zero point) Return ; G30



Function and purpose

The tool can return to the second, third, or fourth reference position by specifying G30 P2 (P3 or P4).



- (S) Start point
- (CP) Intermediate point
- (R2) 2nd reference position
- (R3) 3rd reference position
- (R4) 4th reference position



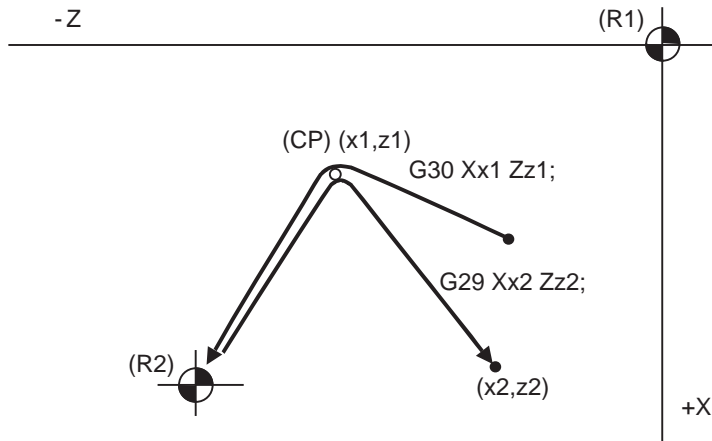
Command format

G30 P2(P3,P4)Xx1 Zz1 α α1;	
X, Y, Z, α	Coordinate value of the intermediate point (α is an additional axis)
P	Reference position No. P2: 2nd reference position return P3: 3rd reference position return P4: 4th reference position return



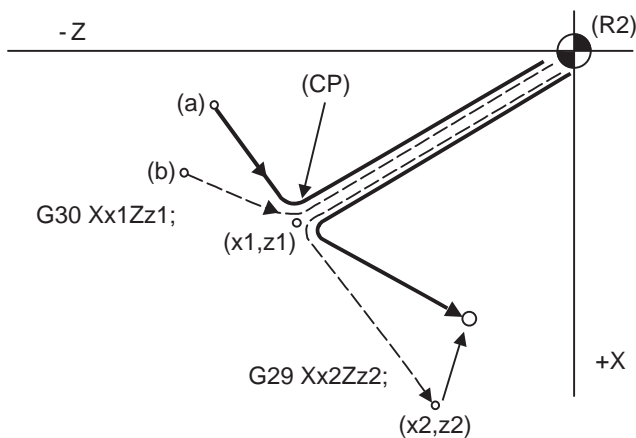
Detailed description

- (1) The 2nd, 3rd, or 4th reference position return is specified by P2, P3, or P4.
 A command without P or with other designation method will return the tool to the 2nd reference position.
- (2) In the 2nd, 3rd, or 4th reference position return mode, as in the 1st reference position return mode, the tool returns to the 2nd, 3rd, or 4th reference position via the intermediate point specified by G30.
- (3) The 2nd, 3rd, and 4th reference position coordinates refer to the positions specific to the machine, and these can be checked with the setting and display unit.
- (4) If G29 is commanded after completion of returning to the 2nd, 3rd, and 4th reference position, the intermediate position used last is used as the intermediate position for returning by G29.



(CP) Intermediate point
 (R1) 1st reference position
 (R2) 2nd reference position

- (5) With reference position return on a plane during compensation, the tool moves without tool nose radius compensation (zero compensation) from the intermediate point as far as the reference position. With a subsequent G29 command, the tool move without tool nose radius compensation from the reference position to the intermediate point and it moves with such compensation until the G29 command from the intermediate point.



(CP) Intermediate point
 (a) Tool nose center path
 (R1) 1st reference position
 (b) Program path
 (R2) 2nd reference position

14.9 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__ Z__ α __ P__ ; ... Check command

X Z α	Return control axis
P	Check No. P1: 1st reference position check P2: 2nd reference position check P3: 3rd reference position check P4: 4th reference position check



Detailed description

- (1) If the P command has been omitted, the 1st reference position will be checked.
- (2) The number of axes whose reference positions can be checked simultaneously depends on the number of axes which can be controlled simultaneously.
- (3) An alarm will occur if the reference position is not reached after the command is completed.

14.10 Workpiece Coordinate System Setting and Offset ; G54 to G59 (G54.1)



Function and purpose

- (1) The workpiece coordinate systems facilitate the programming on the workpiece, serving the reference position of the machining workpiece as the zero point.
- (2) These commands enable the tool to move to the positions in the workpiece coordinate system. There are 48 sets of added workpiece coordinate systems, as well as 6 workpiece coordinate systems, which are used by the programmer for programming (G54 to G59). (The 48 sets are options.)
- (3) Among the workpiece coordinate systems currently selected by these commands, any workpiece coordinate system with coordinates which have been commanded by the current position of the tool is reset. (The "current position of the tool" includes the compensation amounts for tool nose R, tool length compensation.)
- (4) A hypothetical machine coordinate system with coordinates which have been commanded by the current position of the tool is set by this command.
(The "current position of the tool" includes the compensation amounts for tool nose R, tool length and compensation.) (G54,G92)



Command format

G54 to G59 ... Workpiece coordinate system selection

(G54 to G59) G92 X__ Z__ α __ ; ... Workpiece coordinate system setting

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

G54.1 Pn ; ... Workpiece coordinate system selection (P1 to P48)

**G54.1 Pn ;
G92 X__ Z__ ; ... Workpiece coordinate system setting (P1 to P48)**

G10 L20 Pn X__ Z__ ; ... Workpiece coordinate system offset amount setting (P1 to P48)

When the designated extended workpiece coordinate system offset amount is rewritten

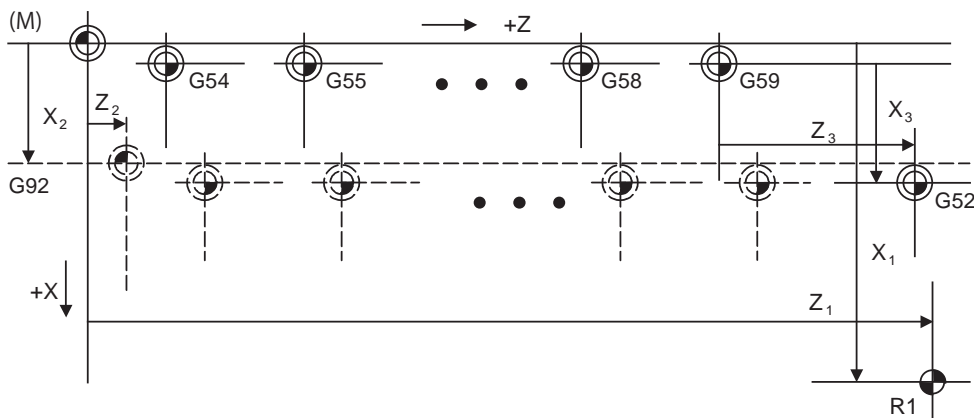
G10 G54.1 Pn X__ Z__ ; ... Workpiece coordinate system offset amount setting (P1 to P48)

When the extended workpiece coordinate system is selected, and the offset amount is rewritten



Detailed description

- (1) With any of the G54 through G59 commands, the nose radius offset amounts for the commanded axes will not be canceled even if workpiece coordinate system selection is commanded.
- (2) The G54 workpiece coordinate system is selected when the power is turned ON.
- (3) Commands G54 through G59 are modal commands (group 12).
- (4) The coordinate system will move with G92 in a workpiece coordinate system.
- (5) The offset setting amount in a workpiece coordinate system denotes the distance from the basic machine coordinate system zero point.



- (M) Basic machine coordinate system
 (G92) Hypothetical machine coordinate system (shifted by G92)
 (G54) Workpiece coordinate system 1
 (G55) Workpiece coordinate system 2
 (G58) Workpiece coordinate system 5
 (G59) Workpiece coordinate system 6
 (G52) Local coordinate system
 (R1) 1st reference position

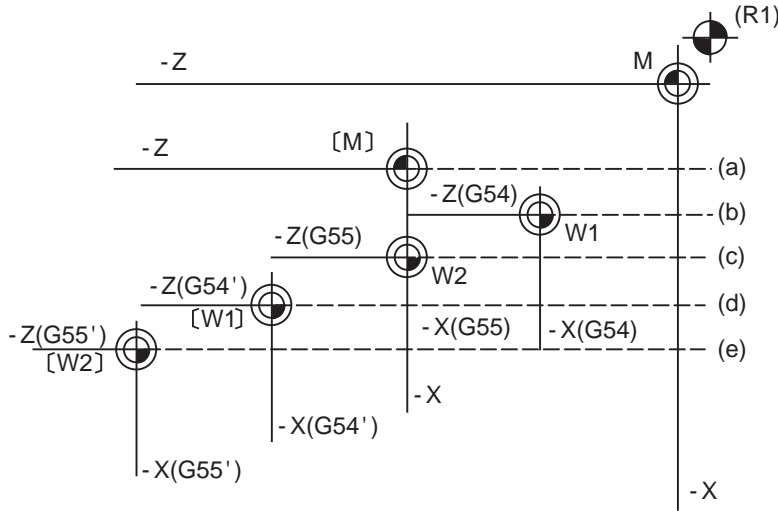
- (6) The offset settings of workpiece coordinate systems can be changed any number of times. (They can also be changed by G10 L2 Pp1 Xx1 Zz1.)

[Handling when L or P is omitted]

G10 L2 Pn Xx Zz ;	n=0 : Set the offset amount in the external workpiece coordinate system. n=1 to 6: Set the offset amount in the designated workpiece coordinate system. Others : The program error (P35) will occur.
G10 L2 Xx Zz ;	Set the offset amount in the currently selected workpiece coordinate system. When in G54.1 modal, the program error (P33) will occur.
G10 L20 Pn Xx Zz ;	n=1 to 48 : Set the offset amount in the designated workpiece coordinate system. Others : The program error (P35) will occur.
G10 L20 Xx Zz ;	Set the offset amount in the currently selected workpiece coordinate system. When in G54 to G59 modal, the program error (P33) will occur.
G10 Pn Xx Zz ; G10 Xx Zz ; G10 G54.1 Xx Yy Zz ;	L10 (tool offset) will be judged if there is no L value.

- (7) A new workpiece coordinate system 1 is set by issuing the G92 command in the G54 (workpiece coordinate system 1) mode. At the same time, the other workpiece coordinate systems 2 to 6 (G55 to G59) will move in parallel and new workpiece coordinate systems 2 to 6 will be set.

- (8) A hypothetical machine coordinate system is formed at the position which deviates from the new workpiece reference position (zero point) by an amount equivalent to the workpiece coordinate system offset amount.



After the power has been switched on, the hypothetical machine coordinate system is matched with the basic machine coordinate system by the first automatic (G28) or manual reference position (zero point) return.

- (R1) Reference position 1
- (a) Hypothetical machine coordinate system based on G92
- (b) Old workpiece 1 (G54) coordinate system
- (c) Old workpiece 2 (G55) coordinate system
- (d) New workpiece 1 (G54) coordinate system
- (e) New workpiece 2 (G55) coordinate system

- (9) By setting the hypothetical machine coordinate system, the new workpiece coordinate system will be set at a position which deviates from that hypothetical machine coordinate system by an amount equivalent to the workpiece coordinate system offset amount.
- (10) When the first automatic (G28) or manual reference position (zero point) return is completed after the power has been turned ON, the basic machine coordinate system and workpiece coordinate systems are set automatically in accordance with the parameter settings.
- (11) If G54 X- ; is commanded after the reference position return (both automatic or manual) executed after the power is turned ON, the program error (P62) will occur. (A speed command is required as the movement will be controlled with the G01 speed.)
- (12) Do not command a G code for which a P code is used in the same block as G54.1 or G10L20. If a G code is commanded, a P code is used for a prior G command or the program error occurs(P33).
- (13) When number of workpiece offset sets additional specifications is not added, the program error (P35) will occur when the G54.1 command is executed.
- (14) When number of workpiece offset sets additional specifications is not added, the program error (P172) will occur when the G10 L20 command is executed.
- (15) The local coordinate system cannot be used during G54.1 modal. The program error (P438) will occur when the G52 command is executed during G54.1 modal.
- (16) A new workpiece coordinate system P1 can be set by commanding G92 in the G54.1 P1 mode. However, the workpiece coordinate system of the other workpiece coordinate systems G54 to G59, G54.1, and P2 to P48 will move in parallel with it, and a new workpiece coordinate system will be set.

- (17) The offset amount of the extended workpiece coordinate system is assigned to the variable number as shown in Table 1.

[Variable Nos. of the extended workpiece coordinate offset system]

	1st axis to nth axis		1st axis to nth axis
P1	#7001 to #700n	P25	#7481 to #748n
P2	#7021 to #702n	P26	#7501 to #750n
P3	#7041 to #704n	P27	#7521 to #752n
P4	#7061 to #706n	P28	#7541 to #754n
P5	#7081 to #708n	P29	#7561 to #756n
P6	#7101 to #710n	P30	#7581 to #758n
P7	#7121 to #712n	P31	#7601 to #760n
P8	#7141 to #714n	P32	#7621 to #762n
P9	#7161 to #716n	P33	#7641 to #764n
P10	#7181 to #718n	P34	#7661 to #766n
P11	#7201 to #720n	P35	#7681 to #768n
P12	#7221 to #722n	P36	#7701 to #770n
P13	#7241 to #724n	P37	#7721 to #772n
P14	#7261 to #726n	P38	#7741 to #774n
P15	#7281 to #728n	P39	#7761 to #776n
P16	#7301 to #730n	P40	#7781 to #778n
P17	#7321 to #732n	P41	#7801 to #780n
P18	#7341 to #734n	P42	#7821 to #782n
P19	#7361 to #736n	P43	#7841 to #784n
P20	#7381 to #738n	P44	#7861 to #786n
P21	#7401 to #740n	P45	#7881 to #788n
P22	#7421 to #742n	P46	#7901 to #790n
P23	#7441 to #744n	P47	#7921 to #792n
P24	#7461 to #746n	P48	#7941 to #794n

 **CAUTION**

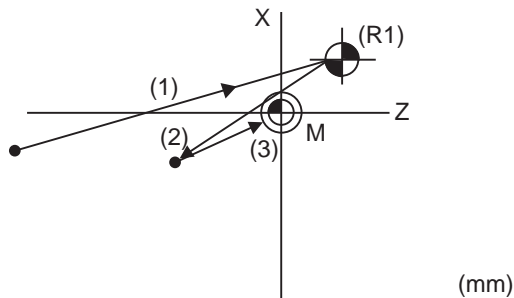
1. If the workpiece coordinate system offset amount is changed during single block stop, the new setting will be valid from the next block.



Program example

(Example 1)

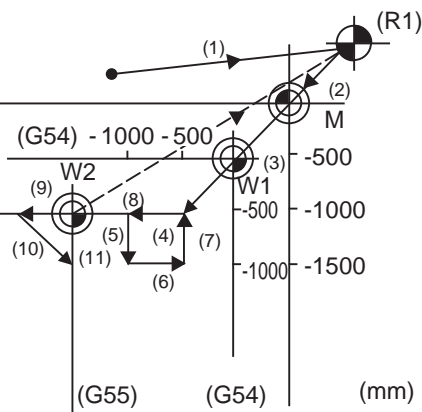
- (1) G28 X0Z0;
- (2) G53 X-500 Z-1000;
- (3) G53 X0Z0;



When the 1st reference position coordinate position is zero, the basic machine coordinate system zero point and reference position (zero point) return position (#1) will coincide.

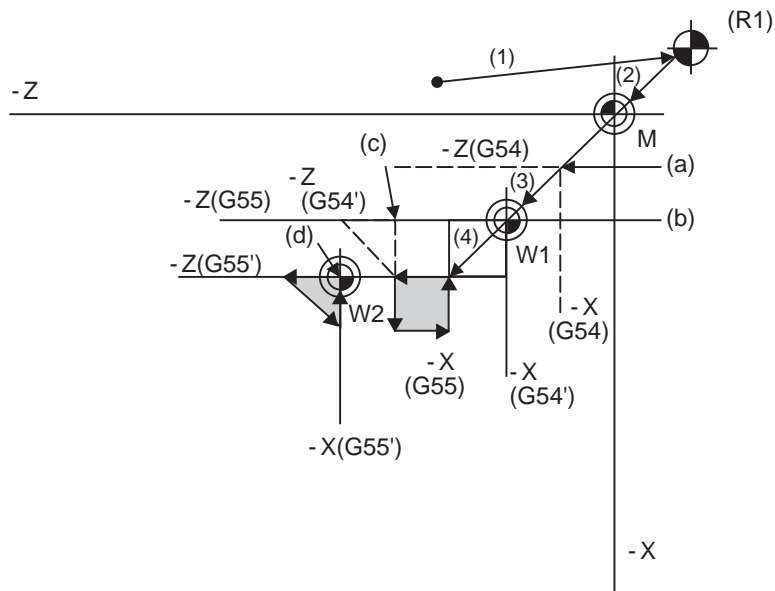
(Example 2)

- (1) G28X0Z0;
- (2) G00G53X0Z0;
- (3) G54X - 500Z - 500;
- (4) G01W - 500F 100;
- (5) U - 500;
- (6) W+500;
- (7) U+500;
- (8) G00G55X0Z0;
- (9) G01Z - 500 F200;
- (10) X - 500 Z0;
- (11) G28X0Z0;



(Example 3) When workpiece coordinate system G54 (-500, -500) has deviated in Example 2. (It is assumed that (3) to (10) in Example 2 have been entered in subprogram 1111.)

(1) G28 X0 Z0 ;	
(2) G00 G53 X0 Z0 ;	(Not required when there is no basic machine coordinate system offset.)
(3) G54 X-500 Z-500 ;	Amount by which workpiece coordinate system deviates
(4) G92 X0 Z0 ;	New workpiece coordinate system is set.
(5) M98 P1111 ;	



- (a) Old G54 coordinate system
- (b) New G54 coordinate system
- (c) Old G55 coordinate system
- (d) New G55 coordinate system
- (R1) Reference position return position

(Note) The workpiece coordinate system will deviate each time when steps (3) to (5) are repeated. The reference position return (G28) command should therefore be issued upon completion of the program.

14.11 Local Coordinate System Setting ; G52



Function and purpose

The local coordinate systems can be set on the G54 through G59 workpiece coordinate systems using the G52 command so that the commanded position serves as the programmed zero point.

The G52 command can also be used instead of the G92 command to change the deviation between the zero point in the machining program and the machining workpiece zero point.



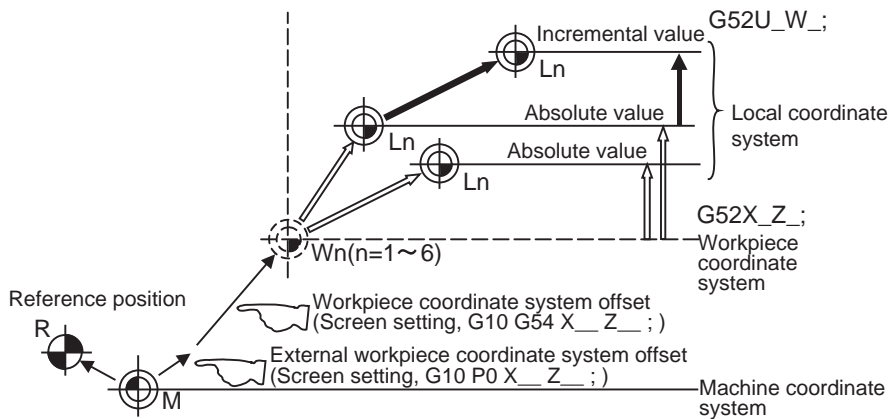
Command format

G54(G54 to G59) G52 X__ Z__ ;



Detailed description

- (1) The G52 command is valid until a new G52 command is issued, and the tool does not move. This command comes in handy for employing another coordinate system without changing the zero point positions of the workpiece coordinate systems (G54 to G59).
- (2) The local coordinate system offset will be cleared by the dog-type manual reference (zero) point return or reference (zero) point return performed after the power has been switched ON.
- (3) The local coordinate system is canceled by (G54 to G59) G52 X0 Z0;.
- (4) Coordinate commands in the absolute value cause the tool to move to the local coordinate system position.



- (Note) If the program is executed repeatedly, the workpiece coordinate system will deviate each time. Thus, when the program is completed, the reference position return operation must be commanded.

14.12 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 coordinates or not can be selected with the parameters.



Command format

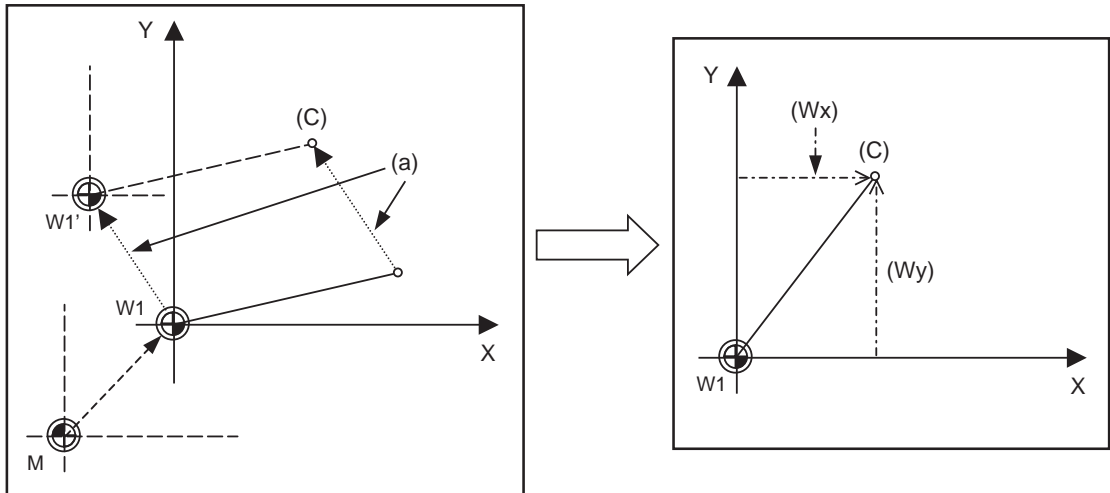
G92.1 X0 Y0 Z0 α0; (G50.3)

α 0	Additional axis
------------	-----------------



Detailed description

- (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) Depending on the G code list, the G code will be "G50.3".
- (4) When manual operation is carried out when manual absolute is set to OFF, or if the axis is moved with handle interrupt.

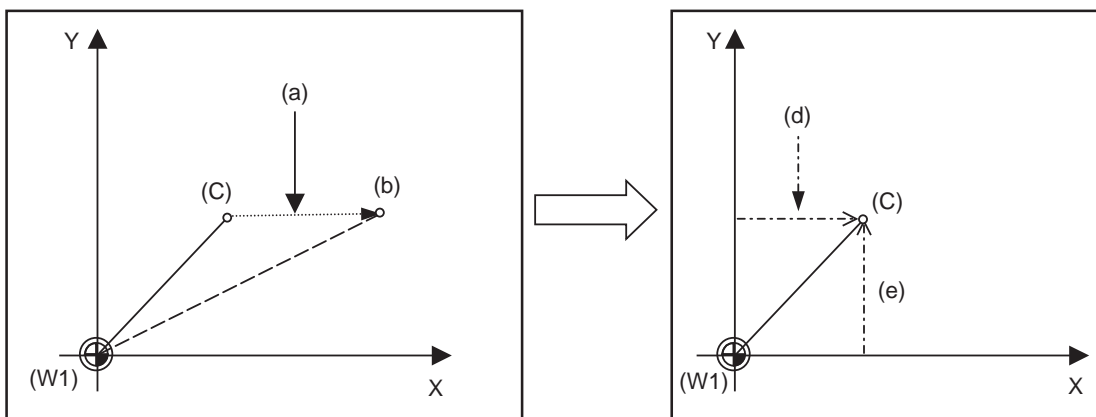


- (a) Manual movement amount
- (Wx) Workpiece coordinate x after preset
- (Wy) Workpiece coordinate y after preset

(C) Current position

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.

(5) When movement command is issued in machine lock state

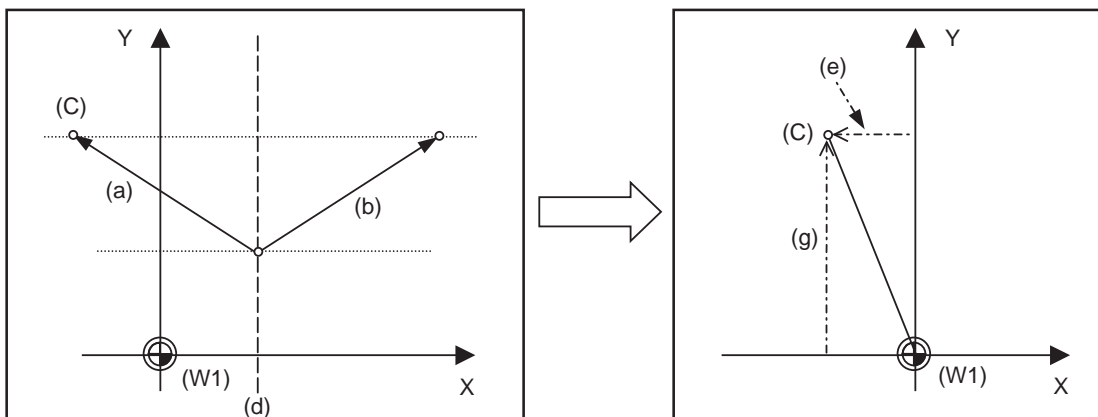


- (a) Movement amount during machine lock
- (b) Workpiece coordinate system coordinate value
- (C) Current position
- (d) Workpiece coordinate x after preset
- (e) Workpiece coordinate y after preset
- (W1) Workpiece coordinate zero point

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.

(6) When operation is carried out with mirror image

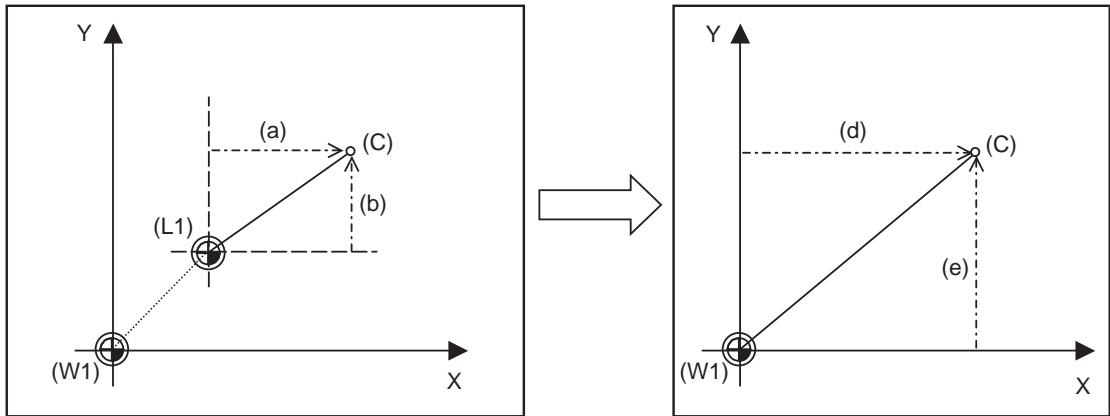


- (a) Actual operation
- (b) Program command
- (C) Current position
- (d) Mirror image center
- (e) Workpiece coordinate x after preset
- (g) Workpiece coordinate y after preset
- (W1) Workpiece coordinate zero point

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.

(7) Setting local coordinate system with G52

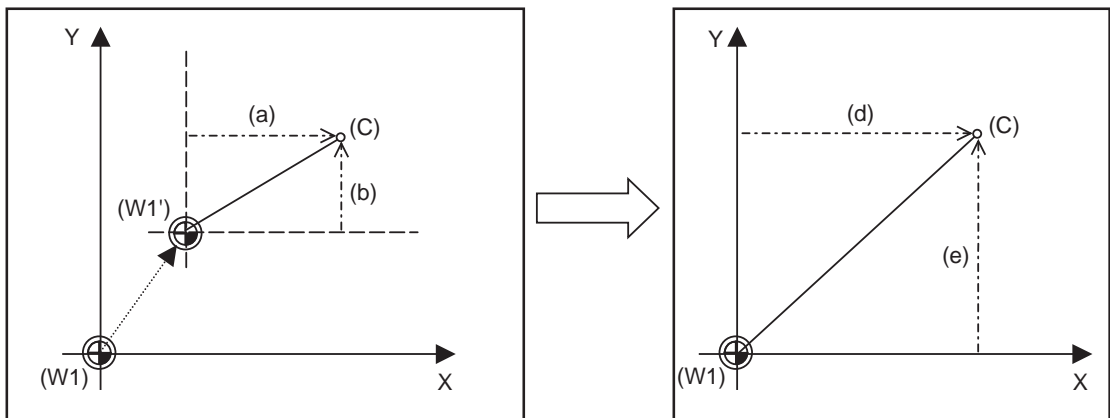


- (a) Local coordinates x
- (C) Current position
- (d) Workpiece coordinate x after preset
- (e) Workpiece coordinate y after preset
- (L1) Local coordinate zero point
- (W1) Workpiece coordinate zero point

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.

(8) Shifting the workpiece coordinate system with G92



- (a) Local coordinates x
- (C) Current position
- (d) Workpiece coordinate x after preset
- (e) Workpiece coordinate y after preset
- (W1) Workpiece coordinate zero point
- (W1') Workpiece zero point after G92 command

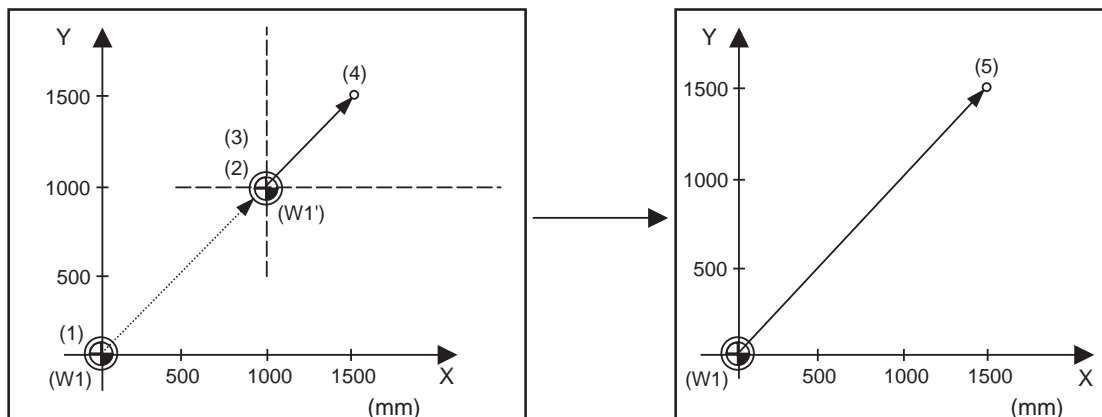
The workpiece coordinate system shifts with the G92 command, and the distance between W1' and the current position is set as the current position of the workpiece coordinate system.

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)
```



Precautions

- (1) Cancel nose R compensation, tool length compensation and before 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.
- (3) Do not command this function during the scaling, coordinate rotation or program mirror image modes. A program error (P34) will occur if commanded.

14.13 Coordinate System for Rotary Axis



Function and purpose

The axis designated as the rotary axis with the parameters is controlled with the rotary axis' coordinate system.

The rotary axis includes the rotating type (short-cut valid/invalid) and linear type (workpiece coordinate position linear type and all coordinate position linear type).

The workpiece coordinate position range is 0 to 359.999° for the rotating type, and 0 to ± 99999.999° for the linear type.

The machine coordinate value and relative position differ according to the parameters.

The rotary axis is commanded with a degree (°) unit regardless of the inch or metric designation.

The rotary axis type can be set with the parameter "#8213 rotation axis type" for each axis.

	Rotary axis				Linear axis
	Rotating type rotary axis		Linear type rotary axis		
	Short-cut invalid	Short-cut valid	Workpiece coordinate position linear type	All-coordinate position linear type	
#8213 setting value	0	1	2	3	-
Workpiece coordinate position	Displayed in the range of 0° to 359.999° .		Displayed in the range of 0° to ± 99999.999° .		
Machine coordinate position/relative position	Displayed in the range of 0° to 359.999° .			Displayed in the range of 0° to ± 99999.999° .	
ABS command	The incremental amount from the end point to the current position is divided by 360 degrees, and the axis moves by the remainder amount according to the sign.	Moves with a short-cut to the end point.	In the same manner as the normal linear axis, it moves according to the sign by the amount obtained by subtracting the current position from the end point (without rounding up to 360 degrees).		
INC command	Moves in the direction of the commanded sign by the commanded incremental amount starting at the current position.				
Reference position return	Depends on the absolute command or the incremental command during the movement to the intermediate point.				
	Returns with movement within 360 degrees.			Moves and returns in the R point direction for the difference from the current position to the R point.	



Operation example

Examples of differences in the operation and counter displays according to the type of rotation coordinate are given below.

(The workpiece offset is set as 0°.)

Rotary type (short-cut invalid)

- (1) The machine coordinate position, workpiece coordinate position and relative position are displayed in the range of 0° to 359.999°.
- (2) For the absolute position command, the axis moves according to the sign by the remainder amount obtained by dividing by 360°.

	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

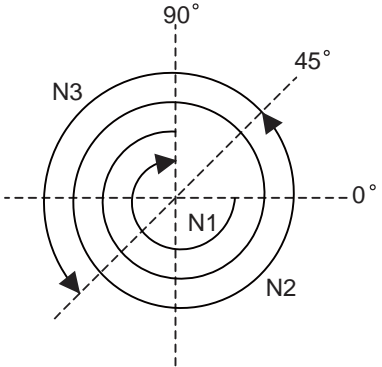
Rotary type (short-cut valid)

- (1) The machine coordinate position, workpiece coordinate position and relative position are displayed in the range of 0° to 359.999°.
- (2) For the absolute position command, the axis rotates to the direction having less amount of movement to the end point.

	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

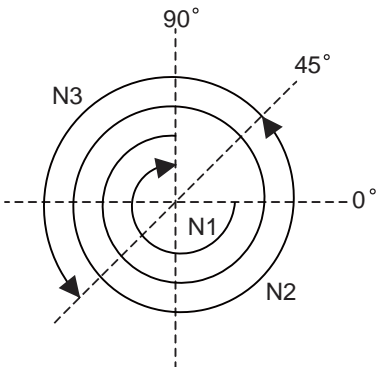
Linear type (workpiece coordinate position linear type)

- (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 0 to ±99999.999°.
- (2) The movement is the same as the linear axis.
- (3) During reference position return, the axis moves in the same manner as the linear axis until the intermediate point is reached. The axis returns with a rotation within 360° from the intermediate point to the reference position.
- (4) During absolute position detection, even if the workpiece coordinate position is not within the range of 0 to 359.999°, the system will start up in the range of 0 to 359.999° when the power is turned ON again.

	Program	Workpiece	Machine	Relative position
	G28 C0.			
	N1 G90 C-270.	-270.000	90.000	90.000
	N2 C405.	405.000	45.000	45.000
	N3 G91 C180.	585.000	225.000	225.000
	After the power is turned ON again			
	Workpiece	Machine		
	225.000	225.000		

Linear type (all coordinate position linear type)

- (1) The workpiece coordinate position counter is displayed in the range of 0 to ±99999.999°.
- (2) The movement is the same as the linear axis.
- (3) During reference position return, the axis moves in the same manner as the linear axis until the intermediate point is reached.
The axis rotates by the difference from the intermediate point to the reference position and returns to the reference position.
- (4) During absolute position detection, the system starts up at the position where the power was turned OFF when the power is turned ON again.

	Program	Workpiece	Machine	Relative position
	G28 C0.			
	N1 G90 C-270.	-270.000	-270.000	-270.000
	N2 C405.	405.000	405.000	405.000
	N3 G91 C180.	585.000	585.000	585.000
	After the power is turned ON again			
	Workpiece	Machine		
	585.000	585.000		

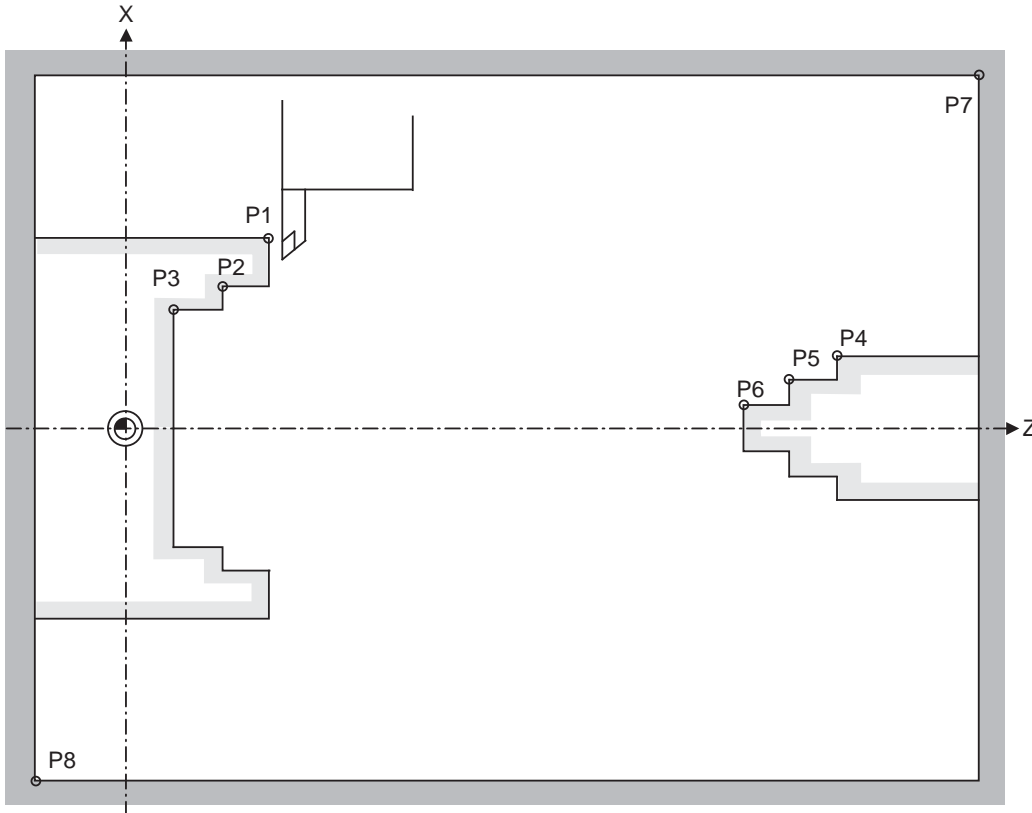
Protection Function

15.1 Chuck Barrier/Tailstock Barrier ; G22,G23



Function and purpose

By limiting the tool nose movement range, the chuck barrier and tailstock barrier prevent collision with the chuck and tailstock due to programming errors. If movement is commanded which exceeds the region set by the parameters, the tool will automatically stop at the barrier boundary.



- P1,P2,P3 : Chuck barrier
- P4,P5,P6 : Tailstock barrier
- P7,P8 : Stored stroke limit



Command format

G22 ; ... Barriers valid

G23 ; ... Barriers invalid

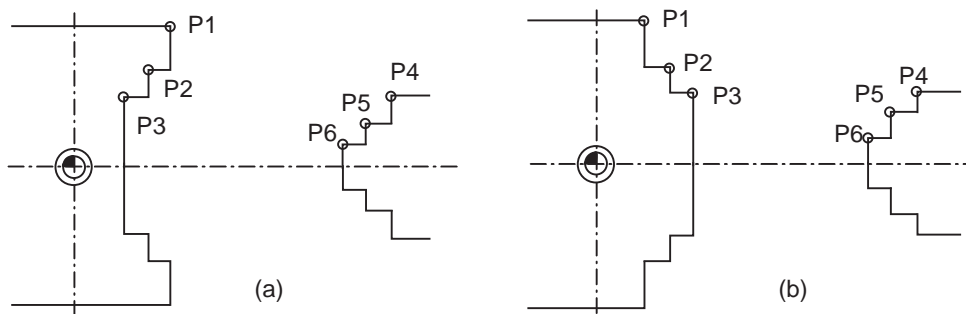
Command G22 and G23 in independent blocks.



Detailed description

- (1) An alarm will appear at the same time as the machine stops because it was about to exceed the set region.
Reset to cancel this alarm.
- (2) This function is also valid during machine lock.
- (3) This function is validated when all axes in which chuck barrier and tailstock barrier are set have finished their reference position returns.
- (4) When there is a stored stroke check function, and the stored stroke limit region is set, the chuck barrier/tailstock barrier function is validated simultaneously with the stored stroke check function.
- (5) A detailed barrier can be set on the left and right by using the PLC external signal.

Setting when using G22 and G23



- (1) Three points can be input as parameters for both the chuck barrier and tailstock barrier. Set them in the machine coordinate system.
Points P1, P2 and P3 (parameters "#8301 P1" to "#8303 P3") are for the chuck barrier. Points P4, P5 and P6 (parameters "#8304 P4" to "#8306 P6") are for the tailstock barrier.
- (2) The barrier region should be a symmetric shape regarding the Z axis. When the X axis coordinates of barrier point P_n are a negative value, reverse the sign to the positive side, then convert and check.
The absolute value of each barrier point's X axis coordinates must be set as follows.
 $P1 \geq P2 \geq P3, P4 \geq P5 \geq P6$
(Note that the Z axis coordinates do not have to follow this setting.)

Setting when using the PLC external signal input

Set the coordinate values for each point between "#8300 P0" to "#8314 P10".

P0 is the basic X coordinate of the chuck and tailstock barrier. Set the workpiece center coordinate in the basic machine coordinate system as a radius value.

The coordinate value from the workpiece center (P0) is set as a radius value for the P1 to P10 X axes. The Z axis is set as a basic machine coordinate system coordinate.

The barrier region is symmetrical in respect to P0.

The X axis coordinate of each point must be set in the following manner.

$$P1 \geq P2 \geq P3, P4 \geq P5 \geq P6,$$

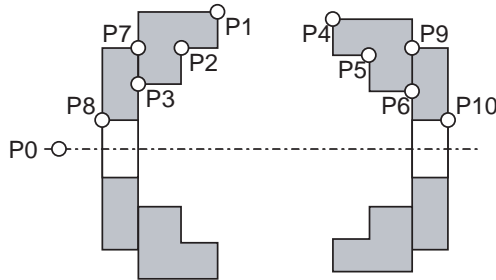
$$P7 \geq P8, P9 \geq P10$$

If the P8 Z axis coordinate must be less than P1 to P3, and the P10 Z axis coordinate must be more than P4 to P6.

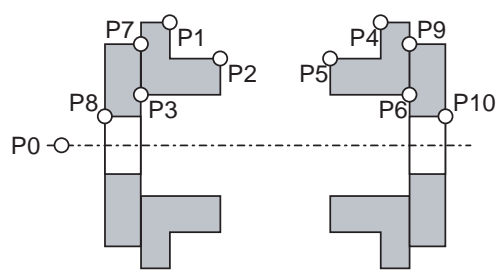
The positions of the each point between P0 and P10 in the chuck/tailstock are as follow.

(1) Setting the chuck

[For external claw]



[For internal claw]



P0 : Workpiece rotation center coordinate

P1,P2,P3 : Chuck section (left)

P7,P8 : Spindle section (left)

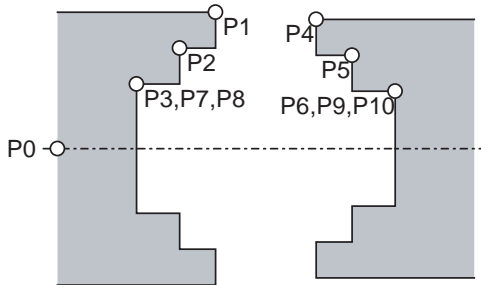
P4,P5,P6 : Chuck section (right)

P9,P10 : Spindle section (right)

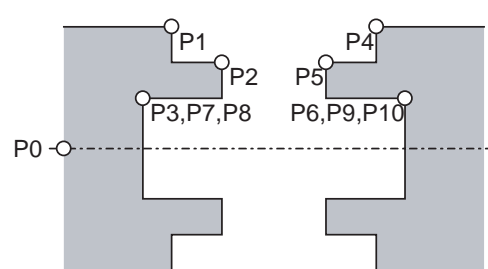
- Set the X axis coordinates so that $P1 > P2 > P3$ is established.
- The P3 and P7 in Z coordinates and the P6 and P9 in Z coordinates shall be equal.
- If there is no spindle inner diameter, set the P8 X coordinate to the same value as P0.

If a spindle section is not to be set, set the P3, P7 and P8 values to the same values, and the P6, P9 and P10 values to the same values. The barrier range will be as shown below in this case.

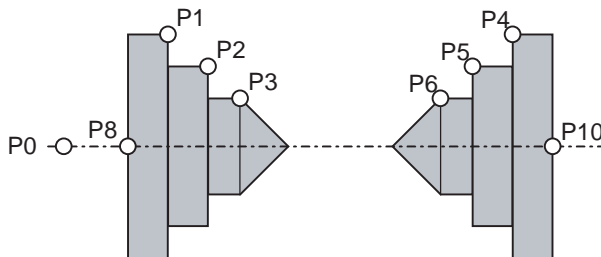
[For external claw]



[For internal claw]



(2) Setting the tailstock



- P7 and P9 are not used.
- The tailstock end section angle is set with the parameters.
 "#8318 STOCK ANGLE (L)"
 "#8319 STOCK ANGLE (R)"
- If "0" is set, the angle will be 90°. (Default)
- Set the P8 X coordinate to the same value as P0. (No spindle inner diameter)

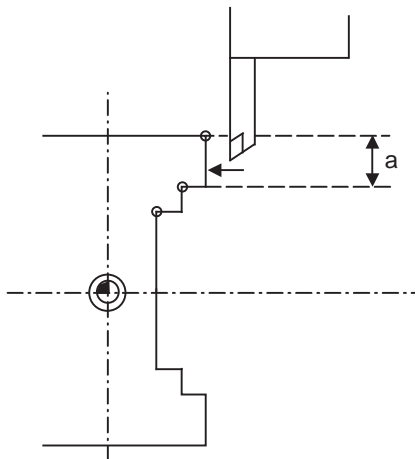


Precautions

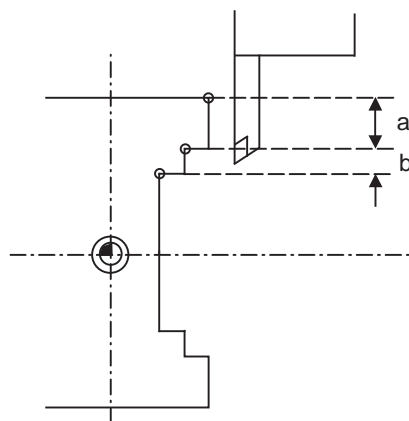
- (1) There is only one checkpoint from the tool regarding the chuck barrier/tailstock barrier. Therefore, the following cautions must be observed.

In the following examples, when the barrier points are set to be checked by the hypothetical tool nose point and the tool moves in the direction of the arrow in the drawing, the following situation may occur. In Example 1, there is a checkpoint in the range "a", so the tool will automatically stop at the barrier boundary. However, in Example 2 there is a checkpoint in the range "b", so the chuck and tool may collide in the range "a".

(Example 1)

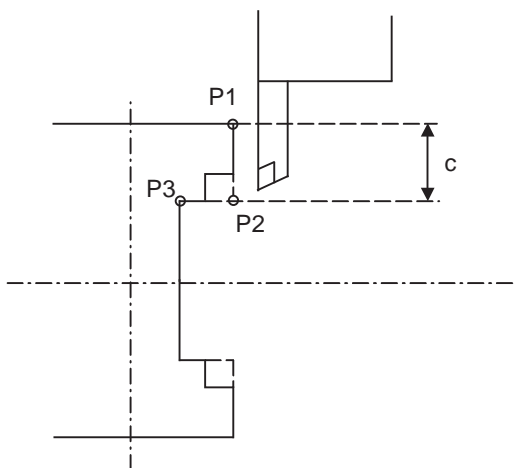


(Example 2)



To avoid this, Example 3 is given. In this example, if the barrier points P1, P2 and P3 are set and the checkpoint is set in range "c", the tool can be stopped at the barrier boundary.

(Example 3)



- (2) When the tool enters the barrier region and an alarm occurs, the tool may move in the opposite direction from which it came, once the alarm is canceled by resetting.
- (3) There is no barrier region for axes without a reference position return function. Thus, there is no barrier alarm for that axis.
- (4) When the tool enters a canceled barrier region, and that barrier is then validated, an alarm will occur immediately if the tool is moved.
In this case, after canceling the alarm with reset and then invalidate the barrier (G23) before escaping or change the value set for each barrier point.
- (5) The soft limit is valid even if the barrier is invalid (G23).

15.2 Stored Stroke Limit ; G22,G23



Function and purpose

The stored stroke limit is a function that can prevent collision of the tools by setting the tool entry prohibited ranges.

The stored stroke limit II function can change the tool entry prohibited ranges (parameter) and command its function's ON/OFF by the program.



Command format

G22 X__ Z__ C__ I__ J__ K__ ; ... Stored stroke limit II function parameter change, function ON (all axes)

G23; ... Stored stroke limit II function OFF (all axes)

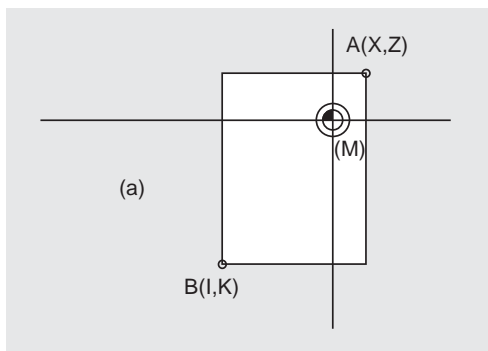
X, Z, C (1st to 3rd axis name)	Stored stroke limit + side designation
I, J, K (Plane selection axis name)	Stored stroke limit - side designation



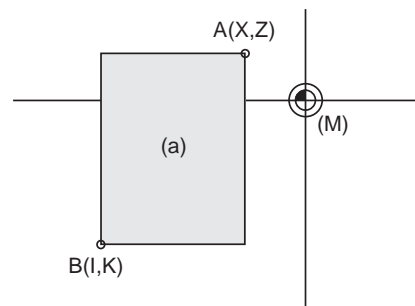
Detailed description

Command G22 and G23 in independent blocks.

Whether to set the prohibited range on the inside or outside follows the parameter settings (#8204 or #8205).



[When prohibited range is on outside]



[When prohibited range is on inside]

- (a) Prohibited range
- (M) Machine coordinate system



Precautions

- (1) If the maximum value and minimum value of the stored stroke limit's prohibited range are set to the same value, the following will occur.
 - (a) When the maximum value and minimum value are set to 0, if the outside is the prohibited range, the entire range will be prohibited. If the inside is the prohibited range, the entire range will be the moveable range.
 - (b) If data other than 0 is set for the maximum value and minimum value, the entire range will be the moveable range.
- (2) The settings (G22, G23) with the stored stroke limit II program command are valid at the G code list 6 and 7 (cmdtyp = 7, 8).
- (3) G22 and G23 are unmodal, and are valid only in the commanded block.

Measurement Support Functions

16.1 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 wear compensation amount.



Command format

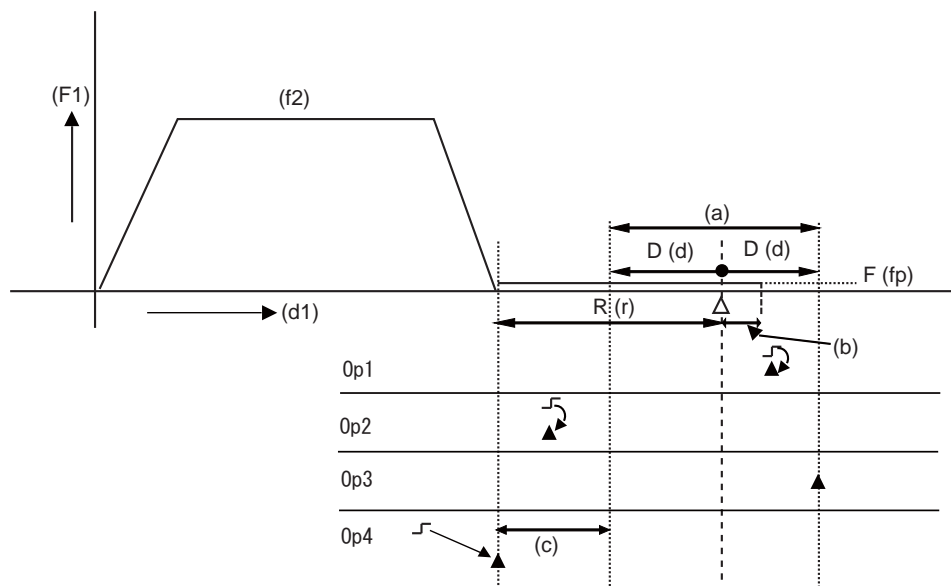
G37 α _ R _ D _ F _ ; ... Automatic tool length measurement command

α	Measuring axis address and coordinates of measurement position ----- X, Z
R	This commands the distance between the measurement position and point where the movement is to start at the measuring speed. (Radius value fixed, incremental value)
D	This commands the range within which the tool is to stop. (Radius value fixed, incremental value)
F	This commands the measuring feedrate. When R_, D_ or F_ is omitted, the value set in the parameter is used instead. <Parameter> ("AUTO TLM" on machining parameter screen) - #8004 SPEED (measuring feedrate): 0 to 60000 [mm/min] - #8005 ZONE r: 0 to 99999.999 [mm] - #8006 ZONE d: 0 to 99999.999 [mm]



Detailed description

(1) Operation with G37 command



Op1 : Normal completion as it is measurement within the allowable range.

Op2 : Alarm stop (P607) as it is outside of the measurement allowable range.

Op3 : Alarm stop (P607) as the sensor is not detected.

Op4 : Alarm stop (P607) as it is outside of the measurement allowable range. However if there is no (c) area, normal completion will occur.

(a) Measurement allowable range

(b) Compensation amount

(d1) Distance

(F1) Speed

(f2) Feedrate

(d) Measurement range

(r) Deceleration range

△ Measuring position

▲ Stop point

┌ Sensor output

(2) The sensor signal (measuring position arrival signal) is used in common with the skip signal.

(3) The feedrate will be 1mm/min if the F command and parameter measurement speed are 0.

(4) During the synchronous feed mode, the axis will move at the synchronous feedrate [mm/rev].

(5) An updated offset amount is valid unless it is assigned from the following T command of the G37 command.

(6) Excluding the delay at the PLC side, the delay and fluctuations in the sensor signal processing range from 0 to 0.2ms.

As a result, the measuring error shown below is caused.

$$\text{Maximum measuring error [mm]} = \text{Measuring speed [mm/min]} * 1/60 * 0.2 \text{ [ms]}/1000$$

(7) The machine position coordinates at that point in time are read by sensor signal detection, and the machine will overtravel and stop at a position equivalent to the servo droop.

$$\text{Maximum overtravel [mm]} = \text{Measuring speed [mm/min]} * 1/60 * 1/\text{Position loop gain [1/s]}$$

The standard position loop gain is 33 (1/s).



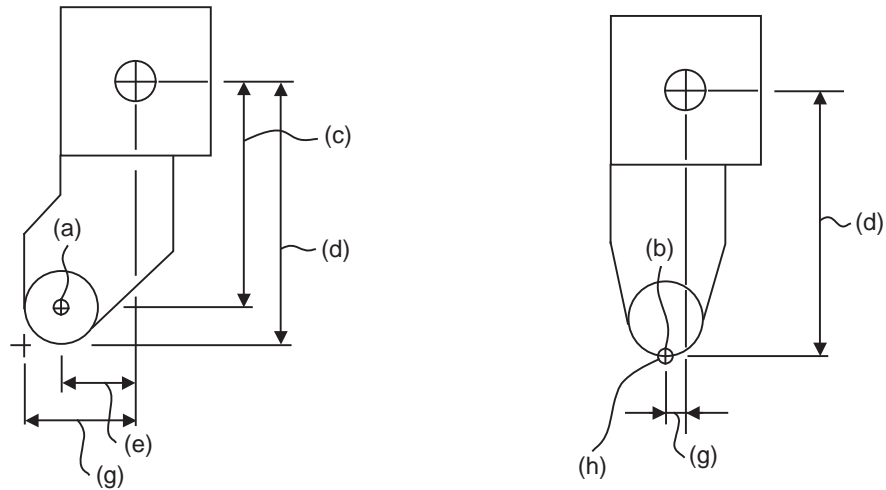
Precautions

- (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 T code is commanded in the G37 block. If the last one digit or last two digits is 0, the (4) error will occur.
- (4) Program error (P606) will occur when T code is not commanded prior to the G37 block. If the last one digit or last two digits is 0, the (P606) error will occur even if T is commanded.
- (5) Program error (P607) will occur when the sensor signal was input outside the allowable measuring range or when the sensor signal was not detected even upon arrival at the end point. However, if the sensor signal stays ON and there is no range of (b) in Op3 exemplified in Detailed description, it will be judged as normal measurement.
- (6) When a manual interrupt is applied while the tool is moving at the measuring speed, a return must be made to the position prior to the interrupt and then operation must be resumed.
- (7) The data commanded in G37 or the parameter setting data must meet the following conditions:
| Measurement point - start point | > R command or parameter r > D command or parameter d
- (8) When the D address and parameter d in (7) above are zero, the operation will be completed normally only when the commanded measurement point and sensor signal detection point coincide. Otherwise, program error (P607) will occur.
- (9) When the R and D addresses as well as parameters r and d in (7) above are all zero, program error (P607) will occur regardless of whether the sensor signal is present or not after the tool has been positioned at the commanded measurement point.
- (10) When the measurement allowable range is larger than the measurement command distance, it becomes the measurement allowable range for all axes.
- (11) When the measurement speed movement distance is larger than the measurement command distance, all axes move at the measurement speed.
- (12) When the measurement allowable range is larger than the measurement speed movement distance, the axis moves in the measurement allowable range at the measurement speed.
- (13) Always cancel nose R compensation before commanding G37.

- (14) Calculate the tool length offset amount without regard for the nose R value and tool nose point No., even if the nose R compensation option is attached.

To set the tool nose point No. to 0, subtract the nose R value from the measured tool length offset amount.

When the tool nose point No. (tool nose shape) is 5, 6, 7, or 8, measure the tool length at the tool tip.



- (a) Tool nose point 0
- (b) Tool nose point 8
- (c) X axis tool length offset value with nose R value subtracted
- (d) Measured X axis tool length offset amount
- (e) Z axis tool length offset value with nose R value subtracted
- (g) Measured Z axis tool length offset amount
- (h) Tip of tool nose

16.2 Skip Function ; G31



Function and purpose

When the skip signal is input externally during linear interpolation based on the G31 command, the machine feed is stopped immediately, the coordinate value is read, the remaining distance is discarded and the command in the following block is executed.



Command format

G31 X/U__ Z/W__ R__ F__ ;

X, Z, U, W	Axis coordinate value; they are commanded with the absolute or the incremental values.
R	Acceleration/deceleration command R0: Acceleration/deceleration time constant=0 (No automatic acceleration/deceleration after interpolation.) R1: Acceleration/deceleration time constant valid. Accelerate/decelerate with the time constant set with the parameters "#2102 skip_tL" and "#2103 skip_t1". R0 is applied when it is omitted.
F	Feedrate (mm/min)

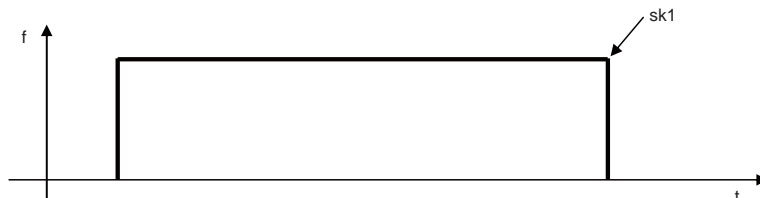


Detailed description

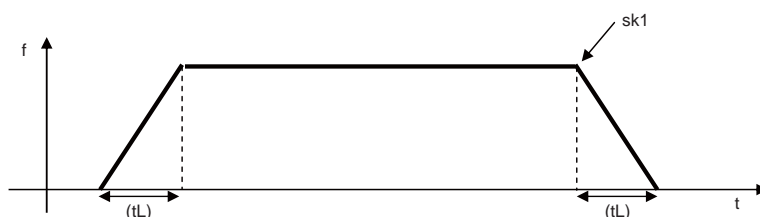
- (1) If Ff is commanded as the feedrate in the same block as the G31 command block, command feed f will apply; if not assigned, the value set in the parameter "#1174 Skip_F" will serve as the feedrate. In either case, the F modal will not be updated.
- (2) The maximum speed of G31 command is determined by the machine specification.
- (3) When R0 is commanded or the R command is omitted, the step acceleration/deceleration will be applied to G31 block after the interpolation without performing the automatic acceleration/deceleration. When R1 is commanded, the automatic acceleration/deceleration will be performed according to the cutting feed acceleration/deceleration mode set by the parameter "#2003 smgst" with the time constant set by the parameter "#2102 skip_tL" and "#2103 skip_t1". Even if G1 constant inclination acceleration/deceleration (the parameter "#1201 G1_acc" is set to "1") is valid, the time constant acceleration and deceleration will be performed.

- (4) When the R1 is commanded with the acceleration and deceleration command, the automatic acceleration and deceleration will be performed after the interpolation even if the skip single is input. Note that if the value of the parameter "#2102 skip_tL" and "#2103 skip_t1" are large, it will not stop immediately.

Acceleration/deceleration when R0 is commanded or R is omitted



Acceleration/deceleration when R1 is commanded



(sk1) Skip signal

(tL) Skip time constant

- (5) Command the acceleration/deceleration command (R0/R01) per G31 command. If not, "the acceleration/deceleration time constant=0 (R0)" will be applied and no automatic acceleration/deceleration will be performed after the interpolation.
- (6) Override is invalid with the G31 command and it is fixed at 100%. Dry run is also invalid. The stop conditions (feed hold, interlock, override zero and stroke end) are valid. External deceleration is also valid.
- (7) The G31 command is unmodal and it needs to be commanded each time.
- (8) If the skip command is input at the start of the G31 command, the G31 command will be completed immediately.
When a skip signal has not been input until the completion of the G31 block, the G31 command will also be completed upon completion of the movement commands.
- (9) When the G31 command is issued during tool nose radius compensation, the program error (P608) will occur.
- (10) When there is no F command in the G31 command and the parameter speed is also zero, the program error (P603) will occur.
- (11) With machine lock or with the Z axis cancel switch ON when only the Z axis is commanded, the skip signal will be ignored and execution will continue as far as the end of the block.

Readout of skip coordinates

The coordinate positions for which the skip signal is input are stored in the system variables #5061 (1st axis) to #506n (n-th axis), so these can be used in the user macros.

```

:
G00 X-100. ;
G31 X-200. F60 ;      (Skip command)
#101=#5061 ;        Skip signal input coordinate position (workpiece coordinate system) is readout to #101.
:

```

- (Note) When the parameter "#1366 skipExTyp (Multi-system simultaneous skip command)" is set to "1", the skip coordinate value will be "0", even if G31 command is given in the first part system or G31 command is given in only one of the multiple part system.

G31 coasting

The amount of coasting from when the skip signal is input during the G31 command until the machine stops differs according to the parameter "#1174 skip_F" or F command in G31.

The time to start deceleration to stop after responding to the skip signal is short, so the machine can be stopped precisely with a small coasting amount. The coasting amount can be calculated from the following formula.

$$\begin{aligned} \delta 0 &= \frac{F}{60} \times T_p + \frac{F}{60} \times (t_1 \pm t_2) \\ &= \underbrace{\frac{F}{60} \times (T_p + t_1)}_{\delta 1} \pm \underbrace{\frac{F}{60} \times t_2}_{\delta 2} \end{aligned}$$

$\delta 0$: Coasting amount (mm)

F : G31 skip speed (mm/min)

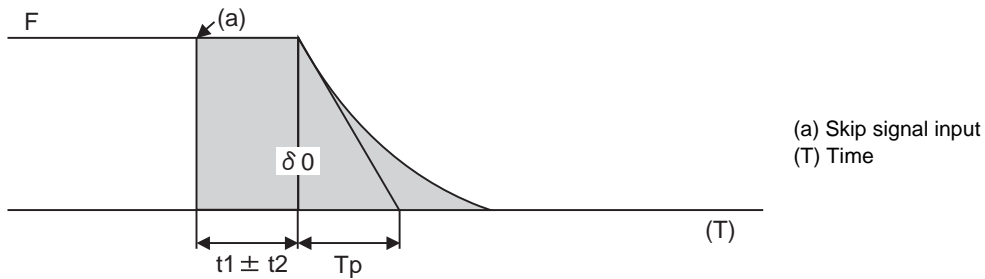
T_p : Position loop time constant (s) = (position loop gain)⁻¹

t_1 : Response delay time (s) = (time taken from the detection to the arrival of the skip signal at the controller via PC)

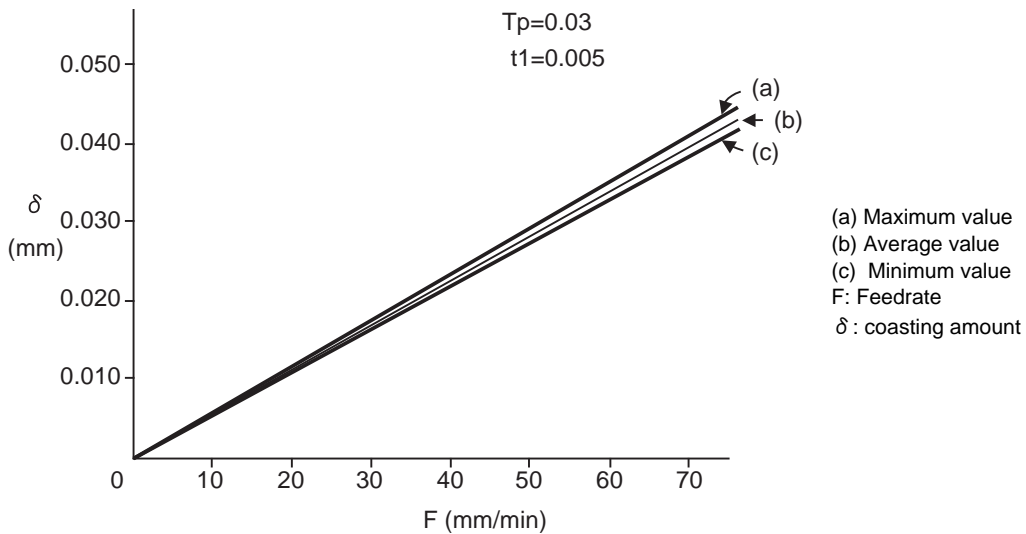
t_2 : Response error time 0.001 (s)

When G31 is used for calculation, the value calculated from the section indicated by $\delta 1$ in the above equation can be compensated, however, $\delta 2$ results in calculation error.

Stop pattern with skip signal input is shown below.



The relationship between the coasting amount and speed when T_p is 30ms and t_1 is 5ms is shown in the following figure.



Readout error of skip coordinates mm

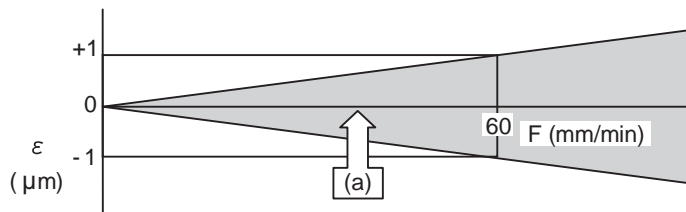
(1) Skip signal input coordinate readout

The coasting amount based on the position loop time constant T_p and cutting feed time constant T_s 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 t_1 results in a measurement error and so compensation must be provided.

$$\varepsilon = \pm (F/60) * t_2$$

ε : Readout error
 F : Feedrate
 t_2 : Response error time 0.001 (s)
 (a) Measurement value



Readout error of skip signal input coordinates

Readout error with a 60mm/min feedrate is as shown below and the measurement value is within readout error range of $\pm 1 \mu\text{m}$:

$$\varepsilon = \pm (60/60) * 0.001 = \pm 0.001 \text{ (mm)}$$

(2) Readout of other coordinates

The readout coordinate values include the coasting amount. Therefore, when coordinate values at the time of skip signal input is required, reference should be made to the section on the G31 coasting amount to compensate the coordinate value. As in the case of (1), the coasting amount based on the delay error time t_2 cannot be calculated, and this generates a measuring error.

Examples of compensating for coasting

(1) Compensating for skip signal input coordinates

```

:
G31 X100.F100 ;      Skip command
G04;                Machine stop check
#101=#5061 ;        Skip signal input coordinate readout
#102=#110*#111/60 ; Coasting based on response delay time
#105=#101-#102 ;    Skip signal input coordinates
:
#110 = Skip feedrate;
#111 = Response delay time t1;
  
```

(2) Compensating for workpiece coordinates

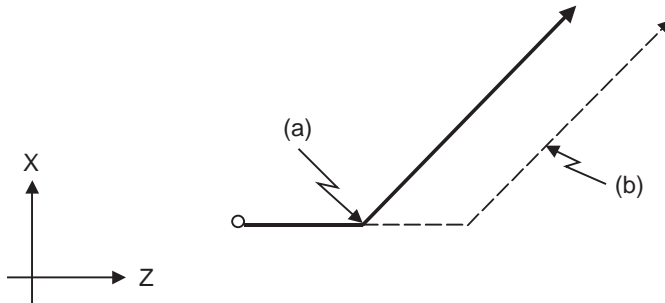
```

:
G31 X100.F100 ;      Skip command
G04;                Machine stop check
#101=#5061 ;        Skip signal input coordinate readout
#102=#110*#111/60 ; Coasting based on response delay time
#103=#110*#112/60 ; Coasting based on position loop time constant
#105=#101-#102-#103 ; Skip signal input coordinates
:
#110 = Skip feedrate;
#111 = Response delay time t1;
#112 = Position loop time constant Tp;
  
```



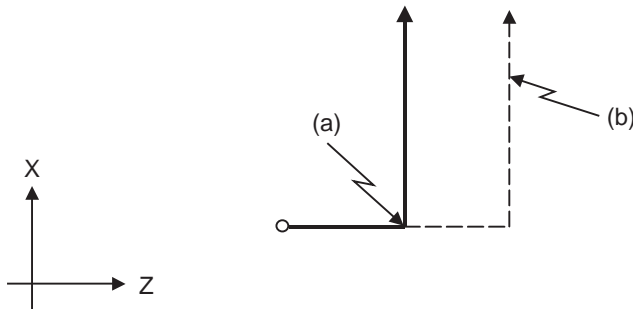
Operation example

(Example 1) When the next block is an incremental value command
 G31 Z1000 F100;
 G01 U2000 W1000;



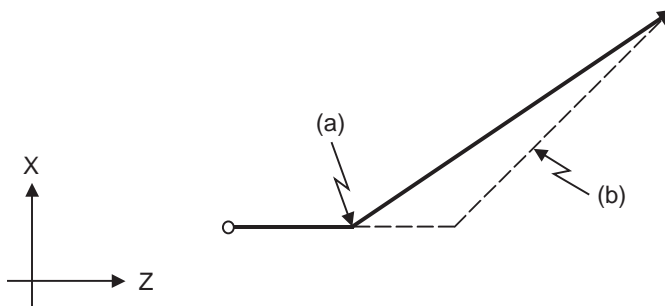
(a) Movement when external signal is entered
 (b) Movement when external signal is not entered

(Example 2) When the next block is a one axis movement command with absolute value
 G31 Z1000 F100;
 G01 X1000;



(a) Movement when external signal is entered
 (b) Movement when external signal is not entered

(Example 3) When the next block is a two axes movement command with absolute value
 G31 Z1000 F100;
 G01 X1000 Z2000;



(a) Movement when external signal is entered
 (b) Movement when external signal is not entered

16.3 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 can be set by parameters.



Command format

G31.1 X__ Z__ α__ R__ F__ ;

X 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 with 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) Command the skip speed by the program command or the parameter. Feedrate G31.1 set with the parameter corresponds to "#1176 skip1f", G31.2 corresponds to "#1178 skip2f", and G31.3 corresponds to "#1180 skip3f". However, F modal will not be 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 which 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) Other commands work the same as the G31 (skip function) command.



Operation example

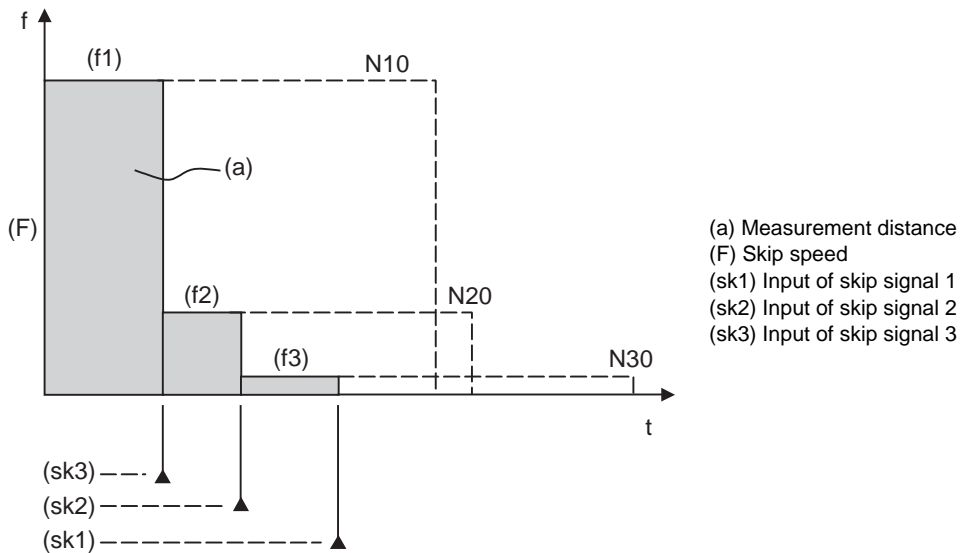
- (1) The multi-step skip function enables the following control, thereby improving measurement accuracy and shortening the time required for measurement.

[Parameter settings]

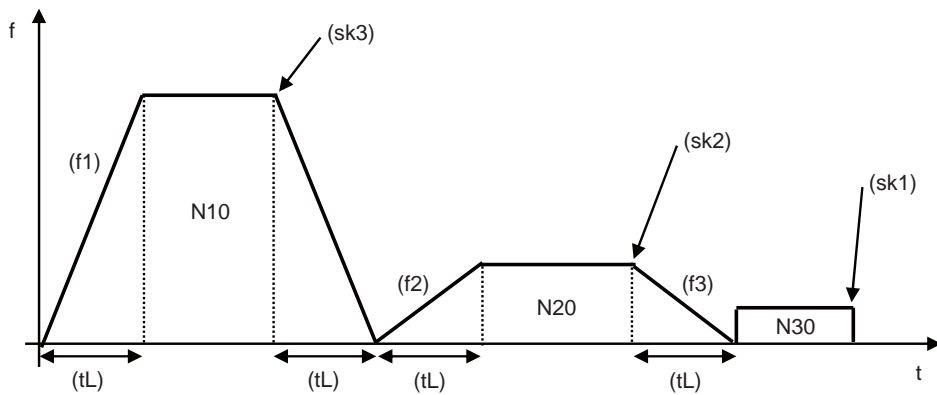
Skip condition	Skip speed
G31.1 : 7	20.0mm/min (f1)
G31.2 : 3	5.0mm/min (f2)
G31.3 : 1	1.0mm/min (f3)

[Program example]

```
N10 G31.1 X200.0 ;
N20 G31.2 X40.0 ;
N30 G31.3 X1.0 ;
```



- (Note 1) 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.

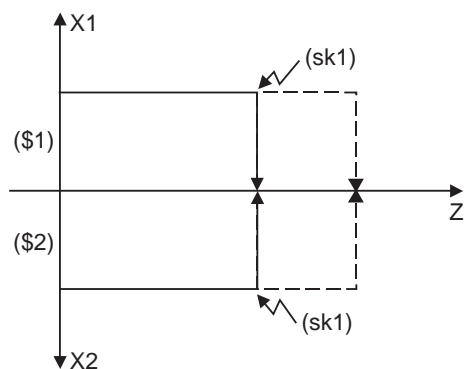
16.4 Multi-step Skip Function 2 ; G31 P



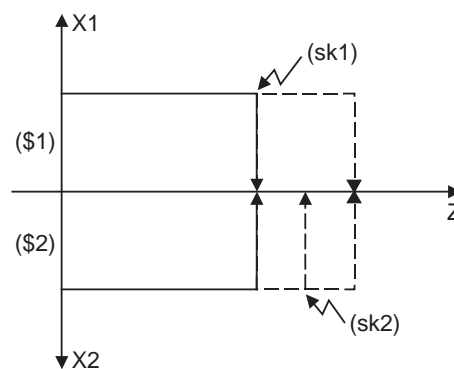
Function and purpose

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__ Z__ α__ P__ R__ F__ ;

X Z α	Target coordinates
P	Skip signal command
R	Acceleration/deceleration command R0: Acceleration/deceleration time constant=0 (No automatic acceleration/deceleration after interpolation.) R1: Acceleration/deceleration time constant valid. Accelerate/decelerate with the time constant set with the parameters "#2102 skip_tL" and "#2103 skip_t1". R0 is applied when it is omitted.
F	Feedrate (mm/min)



Detailed description

- (1) The skip speed is specified by program command or parameter. The feedrate by the parameter is set by "#1174 skip_F". Note that the F modal is not updated in each case.
- (2) The skip signal is specified by skip signal command p. The command range of "p" is from 1 to 255. If outside the range is commanded, program error (P35) will occur.

Skip signal command P	Valid skip signal							
	8	7	6	5	4	3	2	1
1								○
2							○	
3							○	○
4						○		
5						○		○
6						○	○	
7						○	○	○
8					○			
⋮								
253	○	○	○	○	○	○		○
254	○	○	○	○	○	○	○	
255	○	○	○	○	○	○	○	○

- (3) The specified skip signal command is a logical sum of the skip signals.
(Example) G31 X100. P5 F100 ;
Operation is skipped if skip signal 1 or 3 is input.
- (4) If skip signal parameter Pp is not specified, it works as a skip function (G31), not as a multi-step skip function. If speed parameter Ff is not specified, the skip speed set by the parameter "#1174 skip_F" will apply.

[Relations between skip and multi-step skip]

Skip specifications	x		○	
	Condition	Speed	Condition	Speed
G31 X100 ; (Without P and F)	Program error (P601)		Skip 1	#1174 skip_F
G31 X100 P5 ; (Without F)	Program error (P602)		Command value	#1174 skip_F
G31 X100 F100 ; (Without P)	Program error (P601)		Skip 1	Command value
G31 X100 P5 F100 ;	Program error (P602)		Command value	Command value

- (5) If skip specification is effective and P is specified as an axis address, skip signal parameter P will be given a priority. The axis address "P" will be ignored.
(Example) G31 X100. P500 F100 ;
This is regarded as a skip signal. (The program error (P35) will occur.)
- (6) Other than above, the same detailed description as "Skip function; G31" applies.

16.5 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

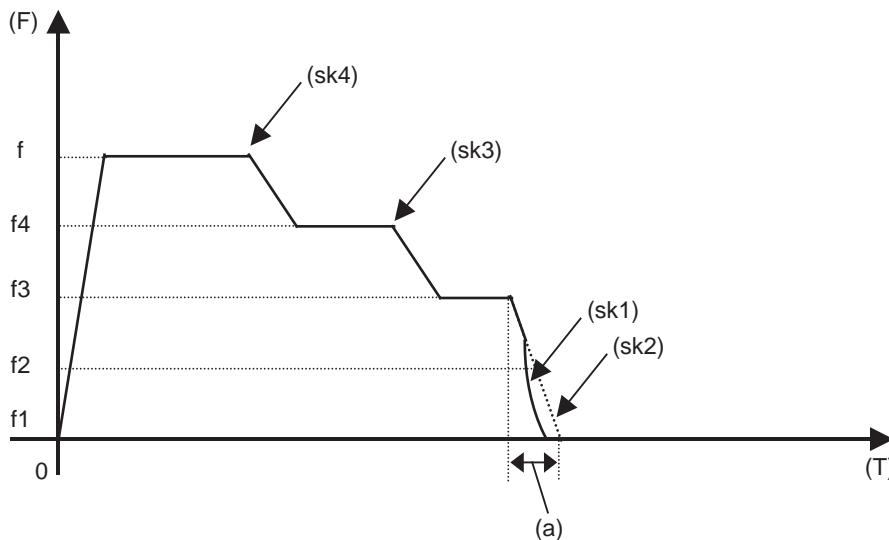
G31 X__ Z__ α__ R__ F__ F1 = __ ... Fn = __ ; ("n" is the skip signal 1 to 8) ... Skip command

X Z α	Target coordinates
F	Feedrate when starting the cutting feed (mm/min)
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 with the parameter "#2102 skip_tL" and "#2103 skip_t1". When omitted, R0 will be applied.
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



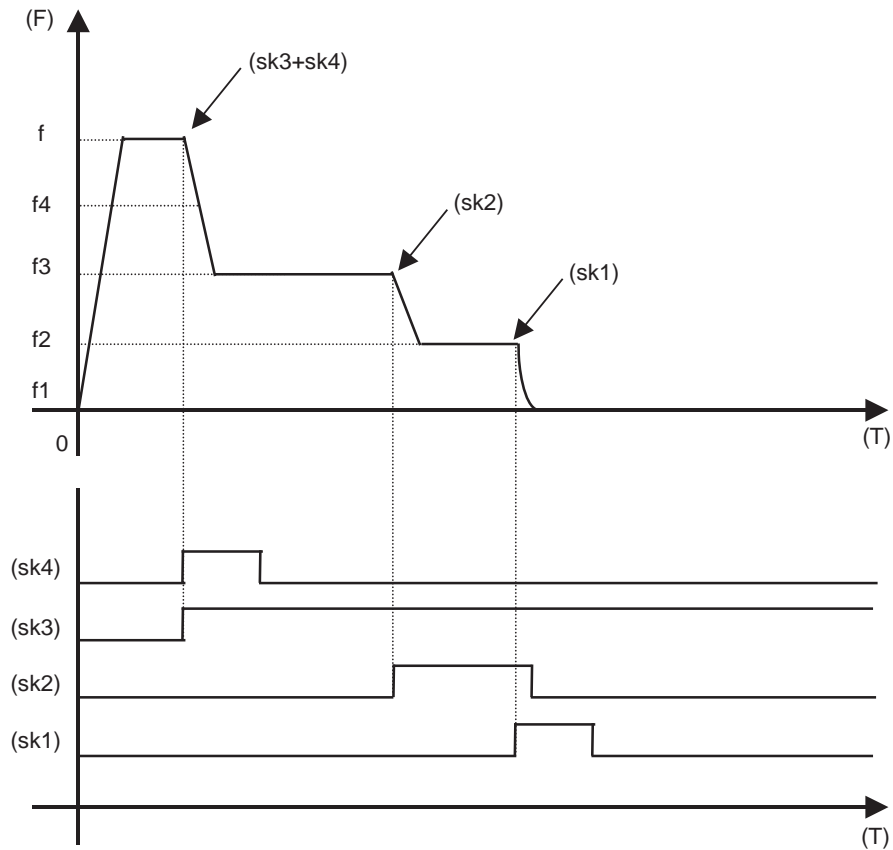
Detailed description

- (1) When the skip signal for which the feedrate $f_n \neq 0$ is commanded, the speed is changed to the command speed corresponding to the skip signal.
- (2) When the skip signal for which the feedrate $f_n=0$ is commanded, the movement is stopped.
 If R0 is commanded or R command is omitted, the skip stop will occur when the movement is stopped by the skip signal detection without performing the automatic acceleration/deceleration by the skip time constant.
 When R1 is commanded, the automatic acceleration/deceleration will be performed with the skip time constant after the interpolation even if the movement is stopped by the skip signal detection.
 Note that if the value of the parameter "#2102 skip_tL" and #2103 skip_t1" are large, it will not stop immediately.
 After the movement is stopped, the remaining movement commands are canceled and the following block will be executed.
- (3) When a skip signal has not been input until the completion of the G31 block, the G31 command will also be completed upon completion of the movement commands.
- (4) When the skip return is valid, the return operation by the skip signal detection is executed after the movement is stopped.
- (5) Even if G1 constant inclination acceleration/deceleration (#1201 G1_acc) is valid, the speed change skip will be the operation of the time constant acceleration and deceleration.
- (6) When the feedrate command ($F_n=f_n$) is not specified after detecting the skip signal, the normal G31 skip operation will be applied.
- (7) If the skip signal is input during the deceleration by the movement command completion, the speed change will be ignored.



- (a) Deceleration section by the movement command completion
- (F) Speed (T) Time
- (sk1) Skip signal 1 (Movement stop): Valid
- (sk2) Skip signal 2 (Speed change) : Invalid
- (sk3) Skip signal 3
- (sk4) Skip signal 4

- (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.



(F) Speed (T) Time
 (sk1) Skip signal 1
 (sk2) Skip signal 2
 (sk3) Skip signal 3
 (sk4) Skip signal 4
 (sk3+sk4) Skip signal 3+Skip signal 4

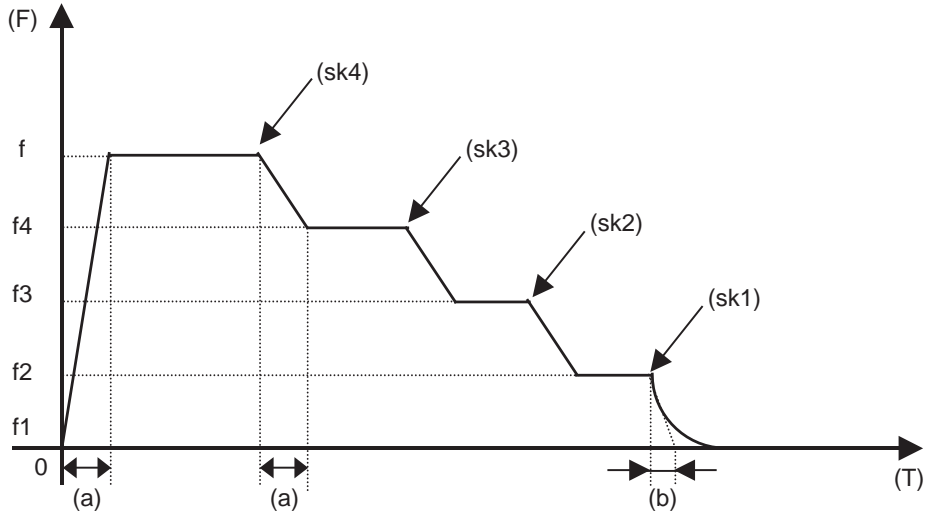
- (10) If the G31 block is started with the skip signal input, that signal is considered to rise at the same time as the block starts.
- (11) If the skip signals for changing the speed and for stopping the movement are simultaneously input, the skip signal for stopping the movement will be valid regardless of the size of the number.
- (12) If the skip time constant "#2102 skip_tL" is illegal, the "Y51 PARAMETER ERROR 15" will occur, and if the "#2103 skip_t1" is illegal, the "Y51 PARAMETER ERROR 16" will occur.
- (13) Other than above, the same detailed description as "Skip function; G31" applies.



Operation example

(1) Example of when R is not commanded

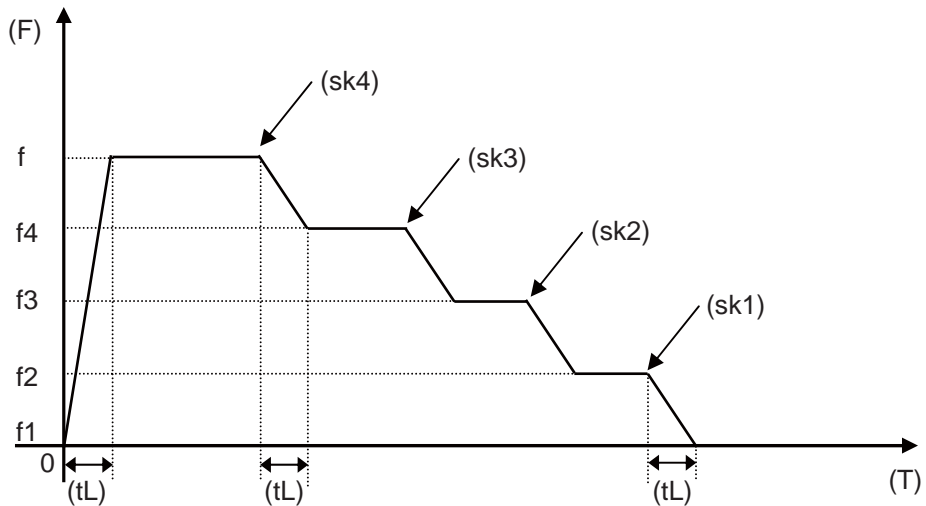
```
G31 X100. Ff F1=0 F2=f2 F3=f3 F4=f4 ;
```



- (a) Skip signal constant
- (F) Speed
- (sk1) Skip signal 1
- (sk3) Skip signal 3
- (b) Position loop time constant
- (T) Time
- (sk2) Skip signal 2
- (sk4) Skip signal 4

(2) Example of when R1 is commanded

```
G31 X100. R1 Ff F1=0 F2=f2 F3=f3 F4=f4 ;
```



- (F) Speed
- (sk1) Skip signal 1
- (sk3) Skip signal 3
- (tL) Skip time constant
- (T) Time
- (sk2) Skip signal 2
- (sk4) Skip signal 4

16.6 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".

Appendix 1

Program Errors

These alarms occur during automatic operation, and the causes of these alarms are mainly program errors which occur, for instance, when mistakes have been made in the preparation of the machining programs or when programs which conform to the specification have not been prepared.

P10 No. of simultaneous axes over

Details

The number of axis addresses commanded in a block exceeds the specifications.

Remedy

- Divide the alarm block command into two.
- Check the specifications.

P11 Illegal axis address

Details

The axis address commanded by the program does not match any of the ones set by the parameter.

Remedy

- Correct the axis names in the program.

P20 Division error

Details

The issued axis command cannot be divided by the command unit.

Remedy

- Correct the program.

P29 Not accept command

Details

The command has been issued when it is impossible.

- The normal line control command (G40.1, G41.1, G42.1) has been issued during the modal in which the normal line control is not acceptable.
- The command has been issued during the modal in which the 2-part system synchronous thread cutting is not acceptable.

Remedy

- Correct the program.

P30 Parity H error

Details

The number of holes per character on the paper tape is even for EIA code and odd for ISO code.

Remedy

- Check the paper tape.
- Check the tape puncher and tape reader.

P31 Parity V error

Details

The number of characters per block on the paper tape is odd.

Remedy

- Make the number of characters per block on the paper tape even.
- Set the parameter parity V selection OFF.

P32 Illegal address

Details

An address not listed in the specifications has been used.

Remedy

- Correct the program address.
- Correct the parameter settings.
- Check the specifications.

P33 Format error**Details**

The command format in the program is not correct.

Remedy

- Correct the program.

P34 Illegal G code**Details**

The commanded G code is not in the specifications.

An illegal G code was commanded during the coordinate rotation command.

Remedy

- Correct the G code address in the program.

Details

G51.2 or G50.2 was commanded when "#1501 polyax (Rotational tool axis number)" was set to "0".

G51.2 or G50.2 was commanded when the tool axis was set to the linear axis ("#1017 rot (Rotational axis)" is set to "0").

Remedy

- Correct the parameter settings.

P35 Setting value range over**Details**

The setting range for the addresses has been exceeded.

Remedy

- Correct the program.

P36 Program end error**Details**

"EOR" has been read during tape and memory mode.

Remedy

- Enter the M02 and M30 command at the end of the program.
- Enter the M99 command at the end of the subprogram.

P37 O, N number zero**Details**

"0" has been specified for program or sequence No.

Remedy

- Designate program Nos. within a range from 1 to 99999999.
- Designate sequence Nos. within a range from 1 to 99999.

P38 No spec: Add. Op block skip**Details**

"/n" has been issued while the optional block skip addition is not in the specifications.

Remedy

- Check the specifications.

P39 No specifications**Details**

- A non-specified G code was commanded.

- The selected operation mode is out of specifications.

Remedy

- Check the specifications.

P45 G-CODE COMB.**Details**

The combination of G codes in a block is inappropriate.
A part of unmodal G codes and modal G codes cannot be commanded in a same block.

Remedy

Correct the combination of G codes.
Separate the incompatible G codes into different blocks.

P48 Restart pos return incomplete**Details**

A travel command was issued before the execution of the block that had been restart-searched.

Remedy

- Carry out program restart again.
Travel command cannot be executed before the execution of the block that has been restart-searched.

P49 Invalid restart search**Details**

- Restart search was attempted for the 3-dimensional circular interpolation.
- Restart search was attempted for the mixed control (cross axis control) command (G110).
- Restart search was attempted during the cylindrical interpolation, polar coordinate interpolation, or tool tip center control.
- Restart search was attempted from a block (G68.2) during the inclined surface machining mode or from the inclined surface machining mode cancel command block (G69).
- Restart search was attempted to the program after direct command mode.

Remedy

- Correct the program.
- Correct the restart search position.

P50 No spec: Inch/Metric change**Details**

Inch/Metric changeover (G20/G21) command was issued while the function is out of specifications.

Remedy

- Check the specifications.

P60 Compensation length over**Details**

The commanded movement distance is excessive (over 2^{31}).

Remedy

- Correct the command range for the axis address.

P61 No spec: Unidirectional posit.**Details**

Unidirectional positioning (G60) was commanded while the function is out of specifications.

Remedy

- Check the specifications.

P62 No F command**Details**

- No feed rate command has been issued.
- There is no F command in the cylindrical interpolation or polar coordinate interpolation immediately after the G95 mode is commanded.

Remedy

- The default movement modal command at power ON is G01. This causes the machine to move without a G01 command if a movement command is issued in the program, and an alarm results. Use an F command to specify the feed rate.
- Specify F with a thread lead command.

P63 No spec: High-speed machining**Details**

High-speed machining cancel (G5P0) was commanded while the high-speed machining is out of specifications.

Remedy

- Check the specifications.

P65 No spec: High speed mode 3**Details****Remedy**

- Check whether the specifications are provided for the high-speed mode III.

P70 Arc end point deviation large**Details**

- There is an error in the arc start and end points as well as in the arc center.
- The difference of the involute curve through the start point and the end point is large.
- When arc was commanded, one of the two axes configuring the arc plane was a scaling valid axis.

Remedy

- Correct the numerical values of the addresses that specify the start and end points, arc center as well as the radius in the program.
- Correct the "+" and "-" directions of the address numerical values.
- Check for the scaling valid axis.

P71 Arc center error**Details**

- An arc center cannot be obtained in R-specified circular interpolation.
- A curvature center of the involute curve cannot be obtained.

Remedy

- Correct the numerical values of the addresses in the program.
- Correct the start and end points if they are inside of the base circle for involute interpolation. When carrying out tool radius compensation, make sure that the start and end points after compensation will not be inside of the base circle for involute interpolation.
- Correct the start and end points if they are at an even distance from the center of the base circle for involute interpolation.

P72 No spec: Helical cutting**Details**

A helical command has been issued though it is out of specifications.

Remedy

- Check whether the specifications are provided for the helical cutting.
- An Axis 3 command has been issued by the circular interpolation command. If there is no helical specification, move the linear axis to the next block.

P73 No spec: Spiral cutting**Details**

A spiral command was issued though it is out of specifications.

Remedy

- Issue the G02.1 and G03.1 commands for circular interpolation.
- Check whether the specifications are provided for the spiral cutting.

P74 Can't calculate 3DIM arc**Details**

The 3-dimension circular cannot be obtained because the end block was not specified during 3-dimension circular interpolation supplementary modal.

The 3-dimension circular cannot be obtained due to an interruption during 3-dimension circular interpolation supplementary modal.

Remedy

- Correct the program.

P75 3DIM arc illegal

Details

An illegal G code was issued during 3-dimension circular interpolation modal. Otherwise, 3-dimension circular interpolation command was issued during a modal for which a 3-dimension circular interpolation command cannot be issued.

Remedy

- Correct the program.

P76 No spec: 3DIM arc interpolat

Details

G02.4/G03.4 was commanded though there is no 3-dimension circular interpolation specification.

Remedy

- Check the specifications.

P80 No spec: Hypoth ax interpolat

Details

Hypothetical axis interpolation (G07) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P90 No spec: Thread cutting

Details

A thread cutting command was issued though it is out of specifications.

Remedy

- Check the specifications.

P91 No spec: Var lead threading

Details

Variable lead thread cutting (G34) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P93 Illegal pitch vaule

Details

An illegal thread lead (thread pitch) was specified at the thread cutting command.

Remedy

- Correct the thread lead for the thread cutting command.

P100 No spec: Cylindric interpolat

Details

A cylindrical interpolation command was issued though it is out of specifications.

Remedy

- Check the specifications.

P110 Plane select during figure rot

Details

Plane selection (G17/G18/G19) was commanded during figure rotation.

Remedy

- Correct the machining program.

P111 Plane selected while coord rot

Details

Plane selection commands (G17, G18, G19) were issued during a coordinate rotation was being commanded.

Remedy

- Always command coordinate rotation cancel after the coordinate rotation command, and then issue a plane selection command.

P112 Plane selected while R compen**Details**

- Plane selection commands (G17, G18, G19) were issued while tool radius compensation (G41, G42) and nose R compensation (G41, G42, G46) commands were being issued.
- Plane selection commands were issued after completing nose R compensation commands when there were no further axis movement commands after G40, and compensation has not been cancelled.

Remedy

- Issue plane selection commands after completing (axis movement commands issued after G40 cancel command) tool radius compensation and nose R compensation commands.

P113 Illegal plane select**Details**

The circular command axis does not correspond to the selected plane.

Remedy

- Select a correct plane before issuing a circular command.

P120 No spec: Feed per rotation**Details**

Feed per rotation (G95) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P121 F0 command during arc modal**Details**

F0 (F 1-digit feed) was commanded during the arc modal (G02/G03).

Remedy

- Correct the machining program.

P122 No spec: Auto corner override**Details**

An auto corner override command (G62) was issued though it is out of specifications.

Remedy

- Check the specifications.
- Delete the G62 command from the program.

P123 No spec: High-accuracy control**Details**

High-accuracy control command was issued though it is out of specifications.

Remedy

- Check the specifications.

P124 No spec: Inverse time feed**Details**

- The inverse time option is not provided.

Remedy

- Check the specifications.

P125 G93 mode error**Details**

- The issued G code command is illegal during G93 mode.
- G93 command was issued during a modal for which inverse time feed cannot be performed.

Remedy

- Correct the program.

P126 Invalid cmnd in high-accuracy**Details**

An illegal command was issued during the high-accuracy control mode.

- A G code group 13 command was issued during the high-accuracy control mode.
- Milling, cylindrical interpolation or pole coordinate interpolation was commanded during the high-accuracy control mode.

Remedy

- Correct the program.

P127 No spec: SSS Control**Details**

The SSS control valid parameter has been set although there is no SSS control specification.

Remedy

- Check the specifications. Set "#8090 SSS ON" to "0" when there is no SSS control specification.

P128 Machin condtn select I disable**Details**

Machining condition selection I was commanded during the mode where the selection command is unavailable.

Remedy

- Check the program and see whether any unavailable mode is included during machining condition selection I. If any, cancel that mode.

P130 2nd M function code illegal**Details**

The 2nd miscellaneous function address, commanded in the program, differs from the address set in the parameters.

Remedy

- Correct the 2nd miscellaneous function address in the program.

P131 No spec: Cnst surface ctrl G96**Details**

A constant surface speed control command (G96) was issued though it is out of specifications.

Remedy

- Check the specifications.
- Issue a rotation speed command (G97) instead of the constant surface speed control command (G96).

P132 Spindle rotation speed S=0**Details**

No spindle rotation speed command has been issued.

Remedy

- Correct the program.

P133 Illegal P-No. G96**Details**

The illegal No. was specified for the constant surface speed control axis.

Remedy

- Correct the parameter settings and program that specify the constant surface speed control axis.

P134 G96 Clamp Err.**Details**

The constant surface speed control command (G96) was issued without commanding the spindle speed clamp (G92/G50).

Remedy

Press the reset key and carry out the remedy below.

- Check the program.
- Issue the G92/G50 command before the G96 command.
- Command the constant surface speed cancel (G97) to switch to the rotation speed command.

P140 No spec: Pos compen cmd**Details**

The position compensation command (G45 to G48) is out of specifications.

Remedy

- Check the specifications.

P141 Pos compen during rotation**Details**

Position compensation was commanded during the figure rotation or coordinate rotation command.

Remedy

- Correct the program.

P142 Pos compen invalid arc**Details**

Position compensation cannot be executed with the issued arc command.

Remedy

- Correct the program.

P150 No spec: Nose R compensation**Details**

- Tool radius compensation commands (G41 and G42) were issued though they are out of specifications.
- Nose R compensation commands (G41, G42, and G46) were issued though they are out of specifications.

Remedy

- Check the specifications.

P151 Radius compen during arc mode**Details**

A compensation command (G40, G41, G42, G43, G44, or G46) has been issued in the arc modal (G02 or G03).

Remedy

- Issue the linear command (G01) or rapid traverse command (G00) in the compensation command block or cancel block.
(Set the modal to linear interpolation.)

P152 No intersection**Details**

- In interference block processing during execution of a tool radius compensation (G41 or G42) or nose R compensation (G41, G42, or G46) command, the intersection point after one block is skipped cannot be determined.
- The compensation amount cannot be calculated during the tool radius compensation for 5-axis machining (G41.2,G42.2).

Remedy

- Correct the program.

P153 Compensation interference**Details**

An interference error has occurred while the tool radius compensation command (G41 or G42) or nose R compensation command (G41, G42 or G46) was being executed.

Remedy

- Correct the program.

P154 No spec: 3D compensation**Details**

A three-dimensional compensation command was issued though it is out of specifications.

Remedy

- Check the specifications.

P155 Fixed cyc exec during compen

Details

A fixed cycle command has been issued in the radius compensation mode.

Remedy

- Issue a radius compensation cancel command (G40) to cancel the radius compensation mode that has been applied since the fixed cycle command was issued.

P156 R compen direction not defined

Details

A shift vector with undefined compensation direction was found at the start of G46 nose R compensation.

Remedy

- Change the vector to that which has the defined compensation direction.
- Change the tool to that which has a different tip point No.

P157 R compen direction changed

Details

During G46 nose R compensation, the compensation direction is reversed.

Remedy

- Change the G command to that which allows the reversed compensation direction (G00, G28, G30, G33, or G53).
- Change the tool to that which has a different tip point No.
- Enable "#8106 G46 NO REV-ERR".

P158 Illegal tip point

Details

An illegal tip point No. (other than 1 to 8) was found during G46 nose R compensation.

Remedy

- Correct the tip point No.

P159 C-CMP REMAINS

Details

One of the followings was commanded while the compensation has not been canceled (the compensation amount remained).

- (1) Automatic tool nose R compensation (G143)
- (2) Radius compensation (G145)
- (3) Plane selection (G17 to G19)
- (4) Skip (G31 or G31.1/G31.2/G31.3)
- (5) Fixed cycle for drilling (G81 to G89)
- (6) Compound type fixed cycle II (G74 to G76)

Remedy

- Cancel the compensation (with compensation amount "0") before commanding.
- Command the G00 move block before the block with a command among (1) to (6).

P160 G53 CMP. ERR

Details

- G53 was commanded during nose R compensation (by G41/G42/G46).
- G53 was commanded to the block where the nose R compensation mode is changed (with G40/G41/G42/G46).
- G53 was commanded while nose R compensation amount has not been cleared.

Remedy

- Correct the program.
- When issuing G53 after G40 command, move the compensation plane axis by G00/G01/G02/G03 command before issuing G53.

P161 No spec: 5ax tool R compensate**Details**

Tool radius compensation for 5-axis machining is not included in the specifications.

Remedy

- Check the specifications.

P162 Disable Cmd in 5ax tool R comp**Details**

A command (G or T command, etc) was issued during tool radius compensation for 5-axis machining, although it is disabled during the compensation.

Remedy

- Cancel the tool radius compensation for 5-axis machining.

P163 5 ax tool R comp is disable**Details**

Tool radius compensation for 5-axis machining was commanded in a mode where the command is disabled.

Remedy

- Cancel the mode that disables the command.

P170 No offset number**Details**

- No compensation No. (DOO, TOO or HOO) command was given when the radius compensation (G41, G42, G43 or G46) command was issued. Otherwise, the compensation No. is larger than the number of sets in the specifications.
- When setting the L system tool life management II, the tool group management program was executed with the tool life management disabled.

Remedy

- Add the compensation No. command to the compensation command block.
- Check the number of sets for the tool compensation Nos. and correct the compensation No. command to be within the number of sets.
- When setting the L system tool life management II ("#1096 T_Ltyp"="2"), enable the tool life management ("#1103 T_Life"="1") to execute a tool group management program.

P171 No spec:Comp input by prog G10**Details**

Compensation data input by program (G10) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P172 G10 L number error**Details**

An address of G10 command is not correct.

Remedy

- Correct the address L No. of the G10 command.

P173 G10 P number error**Details**

The compensation No. at the G10 command is not within the permitted number of sets in the specifications.

Remedy

- Check the number of sets for the tool compensation Nos. and correct the address P designation to be within the number of sets.

P174 No spec:Comp input by prog G11**Details**

Compensation data input by program cancel (G11) was commanded though there is no specification of compensation data input by program.

Remedy

- Check the specifications.

P177 Tool life count active**Details**

Registration of tool life management data with G10 was attempted when the "usage data count valid" signal was ON.

Remedy

- The tool life management data cannot be registered during the usage data count. Turn the "usage data count valid" signal OFF.

P178 Tool life data entry over**Details**

The number of registration groups, total number of registered tools or the number of registrations per group exceeded the range in the specifications.

Remedy

- Correct the number of registrations.

P179 Illegal group No.**Details**

- A duplicate group No. was found at the registration of the tool life management data with G10.
- A group No. that was not registered was designated during the T****99 command.
- An M code command, which must be issued as a single command, coexists in the same block as that of another M code command.
- The M code commands set in the same group exist in the same block.

Remedy

- Register the tool life data once for one group: commanding with a duplicate group No. is not allowed.
- Correct to the group No.

P180 No spec: Drilling cycle**Details**

A fixed cycle command (G72 - G89) was issued though it is out of specifications.

Remedy

- Check the specifications.
- Correct the program.

P181 No spindle command (Tap cycle)**Details**

Spindle rotation speed (S) has not been commanded in synchronous tapping.

Remedy

- Command the spindle rotation speed (S) in synchronous tapping.
- When "#8125 Check Scode in G84" is set to "1", enter the S command in the same block where the synchronous tapping command is issued.

P182 Synchronous tap error**Details**

- Connection to the main spindle unit was not established.
- The synchronous tapping was attempted with the spindle not serially connected under the multiple-spindle control I.

Remedy

- Check connection to the main spindle.
- Check that the main spindle encoder exists.
- Set 1 to the parameter #3024 (sout).

P183 No pitch/thread number**Details**

The pitch or number of threads has not been commanded in the tap cycle of a fixed cycle for drilling command.

Remedy

- Specify the pitch data and the number of threads by F or E command.

P184 Pitch/thread number error**Details**

- The pitch or the number of threads per inch is illegal in the tap cycle of the fixed cycle for drilling command.
- The pitch is too small for the spindle rotation speed.
- The thread number is too large for the spindle rotation speed.

Remedy

- Correct the pitch or the number of threads per inch.

P185 No spec: Sync tapping cycle**Details**

Synchronous tapping cycle (G84/G74) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P186 Illegal S cmd in synchro tap**Details**

S command was issued during synchronous tapping modal.

Remedy

- Cancel the synchronous tapping before issuing the S command.

P190 No spec: Turning cycle**Details**

A lathe cutting cycle command was issued though it is out of specifications.

Remedy

- Check the specification.
- Delete the lathe cutting cycle command.

P191 Taper length error**Details**

In the lathe cutting cycle, the specified length of taper section is illegal.

Remedy

- Set the smaller radius value than the axis travel amount in the lathe cycle command.

P192 Chamfering error**Details**

Chamfering in the thread cutting cycle is illegal.

Remedy

- Set a chamfering amount not exceeding the cycle.

P200 No spec: MRC cycle**Details**

The compound type fixed cycle for turning machining I (G70 to G73) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P201 Program error (MRC)**Details**

- The subprogram, called with a compound type fixed cycle for turning machining I command, has at least one of the following commands: reference position return command (G27, G28, G29, G30); thread cutting (G33, G34); fixed cycle skip-function (G31, G31.n).
- An arc command was found in the first movement block of the finished shape program in compound type fixed cycle for turning machining I.

Remedy

- Delete G27, G28, G29, G30, G31, G33, G34, and fixed cycle G codes from the subprogram called with the compound type fixed cycle for turning machining I commands (G70 to G73).
- Delete G02 and G03 from the first movement block of the finished shape program in compound type fixed cycle for turning machining I.

P202 Block over (MRC)**Details**

The number of blocks in the shape program of the compound type fixed cycle for turning machining I is over 50 or 200 (the maximum number differs according to the model).

Remedy

- Set a 50/200 or less value for the number of blocks in the shape program called by the compound type fixed cycle for turning machining I commands (G70 to G73). (The maximum number differs according to the model).

P203 D cmnd figure error (MRC)**Details**

A proper shape will not obtained by executing the shape program for the compound type fixed cycle for turning machining I (G70 to G73).

Remedy

- Correct the shape program for the compound type fixed cycle for turning machining I (G70 to G73).

P204 E cmnd fixed cycle error**Details**

A command value of the compound type fixed cycle for turning machining (G70 to G76) is illegal.

Remedy

- Correct the command value of the compound type fixed cycle for turning machining (G70 to G76).

P210 No spec: Pattern cycle**Details**

A compound type fixed cycle for turning machining II (G74 to G76) command was commanded though it is out of specifications.

Remedy

- Check the specifications.

P220 No spec: Special fixed cycle**Details**

There are no special fixed cycle specifications.

Remedy

- Check the specifications.

P221 No. of special fixed holes = 0**Details**

"0" has been specified for the number of holes in special fixed cycle mode.

Remedy

- Correct the program.

P222 G36 angle error**Details**

A G36 command specifies "0" for angle intervals.

Remedy

- Correct the program.

P223 G12/G13 radius error**Details**

The radius value specified with a G12 or G13 command is below the compensation amount.

Remedy

- Correct the program.

P224 No spec: Circular (G12/G13)**Details**

There are no circular cutting specifications.

Remedy

- Check the specifications.

P230 Subprogram nesting over**Details**

Over 8 times of subprogram calls have been done in succession from a subprogram.

- A M198 command was found in the program in the data server.
- The program in the IC card has been called more than once (the program in the IC card can be called only once during nested).

Remedy

- Correct the program so that the number of subprogram calls does not exceed 8 times.

P231 No sequence No.**Details**

The sequence No., commanded at the return from the subprogram or by GOTO in the subprogram call, was not set.

Remedy

- Specify the sequence Nos. in the call block of the subprogram.

P232 No program No.**Details**

- The machining program has not been found when the machining program is called.
- The file name of the program registered in IC card is not corresponding to O No.

Remedy

- Enter the machining program.
- Check the subprogram storage destination parameters.
- Ensure that the external device (including IC card) that contains the file is mounted.

P235 Program editing**Details**

Operation was attempted for the file under program editing.

Remedy

- Execute the program again after completion of program editing.

P240 No spec: Variable commands**Details**

A variable command (with #) was issued though it is out of specifications.

Remedy

- Check the specifications.

P241 No variable No.**Details**

The variable No. commanded is out of the range specified in the specifications.

Remedy

- Check the specifications.
- Correct the program variable No.

P242 = not defined at vrble set**Details**

The "=" sign has not been commanded when a variable is defined.

Remedy

- Designate the "=" sign in the variable definition of the program.

P243 Can't use variables**Details**

An invalid variable has been specified in the left or right side of an operation expression.

Remedy

- Correct the program.

P244 Invalid set date or time**Details**

Date or time was set earlier than current date or time in the system variables (#3011, #3012) when the system lock was valid.

Remedy

- Date or time cannot be changed.
- Correct the program.

P250 No spec: Figure rotation**Details**

Figure rotation (M98 I_J_P_H_L_) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P251 Figure rotation overlapped**Details**

Figure rotation command was issued during figure rotation.

Remedy

- Correct the program.

P252 Coord rotate in fig. rotation**Details**

A coordinate rotation related command (G68, G69) was issued during figure rotation.

Remedy

- Correct the program.

P260 No spec: Coordinates rotation**Details**

A coordinate rotation command was issued though it is out of specifications.

Remedy

- Check the specifications.

P261 G code illegal (Coord rot)**Details**

Another G code or a T command has been issued in the block of coordinate rotation command.

Remedy

- Correct the program.

P262 Illegal modal (Coord rot)**Details**

A coordinate rotation command has been issued during modal in which coordinate rotation is not allowed.

Remedy

- Correct the program.

P270 No spec: User macro**Details**

A macro specification was commanded though it is out of specifications.

Remedy

- Check the specifications.

P271 No spec: Macro interrupt**Details**

A macro interruption command has been issued though it is out of specifications.

Remedy

- Check the specifications.

P272 NC and macro texts in a block**Details**

An executable statement and a macro statement exist together in the same block.

Remedy

- Place the executable statement and macro statement in separate blocks in the program.

P273 Macro call nesting over**Details**

The number of macro call nests exceeded the limit imposed by the specifications.

Remedy

- Correct the program so that the macro calls do not exceed the limit imposed by the specifications.

P275 Macro argument over**Details**

The number of argument sets in the macro call argument type II has exceeded the limit.

Remedy

- Correct the program.

P276 Illegal G67 command**Details**

A G67 command was issued though it was not during the G66 command modal.

Remedy

- Correct the program.
- Issue G66 command before G67 command, which is a call cancel command.

P277 Macro alarm message**Details**

An alarm command has been issued in #3000.

Remedy

- Refer to the operator messages on the diagnosis screen.
- Refer to the instruction manual issued by the machine tool builder.

P280 Brackets [] nesting over**Details**

Over five times have the parentheses "[" or "]" been used in a single block.

Remedy

- Correct the program so that the number of "[" or "]" is five or less.

P281 Brackets [] not paired**Details**

A single block does not have the same number of commanded parentheses "[" as that of "]".

Remedy

- Correct the program so that "[" and "]" parentheses are paired up properly.

P282 Calculation impossible**Details**

The arithmetic formula is incorrect.

Remedy

- Correct the formula in the program.

P283 Divided by zero**Details**

The denominator of the division is zero.

Remedy

- Correct the program so that the denominator for division in the formula is not zero.

P288 IF EXCESS**Details**

Over ten times of IF statement nesting have been done.

Remedy

Correct the program so that the number of IF statement nesting does not exceed ten.

P289 IF-ENDIF MMC.**Details**

An IF statement is not ended with ENDIF.
THEN/ELSE has been commanded while there is no IF command.

Remedy

- Correct the program so that IF statements are ended with ENDIFs.
- Put IF[condition] before THEN/ELSE command.

P290 IF sentence error**Details**

There is an error in the "IF[<conditional>]GOTO(" statement.

Remedy

- Correct the program.

P291 WHILE sentence error**Details**

There is an error in the "WHILE[<conditional>]DO(-END(" statement.

Remedy

- Correct the program.

P292 SETVN sentence error**Details**

There is an error in the "SETVN(" statement when the variable name setting was made.

Remedy

- Correct the program.
- The number of characters in the variable name of the SETVN statement must be 7 or less.

P293 DO-END nesting over**Details**

The number of DO-END nesting levels in the "WHILE[<conditional>]DO(-END(" statement has exceeded 27.

Remedy

- Correct the program so that the nesting levels of the DO-END statement does not exceed 27.

P294 DO and END not paired**Details**

The DOs and ENDS are not paired off properly.

Remedy

- Correct the program so that the DOs and ENDS are paired off properly.

P295 WHILE/GOTO in tape**Details**

There is a WHILE or GOTO statement on the tape during tape operation.

Remedy

- Apply memory mode operation instead of tape mode that does not allow the execution of the program with a WHILE or GOTO statement.

P296 No address (macro)**Details**

A required address has not been specified in the user macro.

Remedy

- Correct the program.

P297 Address-A error**Details**

The user macro does not use address A as a variable.

Remedy

- Correct the program.

P298 G200-G202 cmnd in tape**Details**

User macro G200, G201, or G202 was specified during tape or MDI mode.

Remedy

- Correct the program.

P300 Variable name illegal**Details**

The variable names have not been commanded properly.

Remedy

- Correct the variable names in the program.

P301 Variable name duplicated**Details**

A duplicate variable name was found.

Remedy

- Correct the program so that no duplicate name exists.

P310 Not use GMSTB macro code**Details**

G, M, S, T, or B macro code was called during fixed cycle.

Remedy

- Correct the program.
- Correct the parameter settings.

P350 No spec: Scaling command**Details**

The scaling command (G50, G51) was issued though it is out of specifications.

Remedy

- Check the specifications.

P360 No spec: Program mirror**Details**

A mirror image (G50.1 or G51.1) command has been issued though the programmable mirror image specifications are not provided.

Remedy

- Check the specifications.

P370 No spec: Facing t-post MR**Details**

The facing turret mirror image specifications are not provided.

Remedy

- Check the specifications.

P371 Facing t-post MR illegal**Details**

- Mirror image for facing tool posts was commanded to an axis in external mirror image or parameter mirror image.
- The commanded mirror image for facing tool posts enables the mirror image for a rotary axis.

Remedy

- Correct the program.
- Correct the parameter settings.

P380 No spec: Corner R/C**Details**

The corner R/C was issued though it is out of specifications.

Remedy

- Check the specifications.
- Delete the corner chamfering/corner rounding command in the program.

P381 No spec: Arc R/C**Details**

Corner chamfering II or corner rounding II was commanded in the arc interpolation block though it is out of specifications.

Remedy

- Check the specifications.

P382 No corner movement**Details**

The block next to corner chamfering/ corner rounding is not a travel command.

Remedy

- Replace the block succeeding the corner chamfering/ corner rounding command by G01 command.

P383 Corner movement short**Details**

The travel distance in the corner chamfering/corner rounding command was shorter than the value in the corner chamfering/corner rounding command.

Remedy

- Set the smaller value for the corner chamfering/corner rounding than the travel distance.

P384 Corner next movement short**Details**

The travel distance in the following block in the corner chamfering/corner rounding command was shorter than the value in the corner chamfering/corner rounding command.

Remedy

- Set the smaller value for the corner chamfering/corner rounding than the travel distance in the following block.

P385 Corner during G00/G33**Details**

A block with corner chamfering/corner rounding was given during G00 or G33 modal.

Remedy

- Correct the program.

P390 No spec: Geometric**Details**

A geometric command was issued though it is out of specifications.

Remedy

- Check the specifications.

P391 No spec: Geometric arc**Details**

There are no geometric IB specifications.

Remedy

- Check the specifications.

P392 Angle < 1 degree (GEOMT)**Details**

The angular difference between the geometric line and line is 1° or less.

Remedy

- Correct the geometric angle.

P393 Inc value in 2nd block (GEOMT)**Details**

The second geometric block has a command with an incremental value.

Remedy

- Issue a command with an absolute value in the second geometric block.

P394 No linear move command (GEOMT)**Details**

The second geometric block contains no linear command.

Remedy

- Issue the G01 command.

P395 Illegal address (GEOMT)**Details**

The geometric format is invalid.

Remedy

- Correct the program.

P396 Plane selected in GEOMT ctrl**Details**

A plane switching command was issued during geometric command processing.

Remedy

- Complete the plane switching command before geometric command processing.

P397 Arc error (GEOMT)**Details**

In geometric IB, the circular arc end point does not contact or cross the next block start point.

Remedy

- Correct the geometric circular arc command and the preceding and following commands.

P398 No spec: Geometric1B**Details**

A geometric command was issued though the geometric IB specifications are not provided.

Remedy

- Check the specifications.

P411 Illegal modal G111

Details

- G111 was issued during milling mode.
- G111 was issued during nose R compensation mode.
- G111 was issued during constant surface speed.
- G111 was issued during mixed control (cross axis control).
- G111 was issued during fixed cycle.
- G111 was issued during polar coordinate interpolation.
- G111 was issued during cylindrical interpolation mode.

Remedy

- Before commanding G111, cancel the following commands.
- Milling mode
- Nose R compensation
- Constant surface speed
- Mixed control (cross axis control)
- Fixed cycle
- Polar coordinate interpolation
- Cylindrical interpolation

P412 No spec: Axis name switch

Details

Axis name switch (G111) was issued though it is out of specifications.

Remedy

- Check the specifications.

P420 No spec: Para input by program

Details

Parameter input by program (G10) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P421 Parameter input error

Details

- The specified parameter No. or set data is illegal.
- An illegal G command address was input in parameter input mode.
- A parameter input command was issued during fixed cycle modal or nose R compensation.
- G10L70, G11 were not commanded in independent blocks.

Remedy

- Correct the program.

P430 R-pnt return incomplete

Details

- A command was issued to move an axis, which has not returned to the reference position, away from that reference position.
- A command was issued to an axis removal axis.

Remedy

- Execute reference position return manually.
- Disable the axis removal on the axis for which the command was issued.

P431 No spec: 2,3,4th R-point ret

Details

A command for second, third or fourth reference position return was issued though there are no such command specifications.

Remedy

- Check the specifications.

P432 No spec: Start position return**Details**

Start position return (G29) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P433 No spec: R-position check**Details**

Reference position check (G27) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P434 Compare error**Details**

One of the axes did not return to the reference position when the reference position check command (G27) was executed.

Remedy

- Correct the program.

P435 G27 and M commands in a block**Details**

An M command was issued simultaneously in the G27 command block.

Remedy

- Place the M code command, which cannot be issued in a G27 command block, in separate block from G27 command block.

P436 G29 and M commands in a block**Details**

An M command was issued simultaneously in the G29 command block.

Remedy

- Place the M code command, which cannot be issued in a G29 command block, in separate block from G29 command block.

P438 G52 invalid during G54.1**Details**

A local coordinate system command was issued during execution of the G54.1 command.

Remedy

- Correct the program.

P450 No spec: Chuck barrier**Details**

The chuck barrier on command (G22) was specified although the chuck barrier is out of specifications.

Remedy

- Check the specifications.

P451 No spec: Stroke chk bef travel**Details**

Stroke check before travel (G22/G23) was commanded though it is out of specifications.

Remedy

- Check the specifications.

P452 Limit before travel exists**Details**

An illegal command, which places the axis travel start/end point in the prohibited area or moves the axis through the prohibited area, was detected when Stroke check before travel (G22) was commanded.

Remedy

- Correct the coordinate values of the axis address commanded in the program.

P460 Tape I/O error**Details**

An error has occurred in the tape reader. Otherwise an error has occurred in the printer during macro printing.

Remedy

- Check the power and cable of the connected devices.
- Correct the I/O device parameters.

P461 File I/O error**Details**

- A file of the machining program cannot be read.
- IC card has not been inserted.

Remedy

- In memory mode, the programs stored in memory may have been destroyed. Output all of the programs and tool data and then format the system.
- Ensure that the external device (including an IC card, etc) that contains the file is mounted.
- Correct the parameter settings for HD operation or IC card operation.

P462 Computer link commu error**Details**

A communication error occurred during the BTR operation.

Remedy

- "L01 Computer link error" is displayed simultaneously. Take the remedy corresponding to the error No.

P480 No spec: Milling**Details**

- Milling was commanded though it is out of specifications.
- Polar coordinate interpolation was commanded though it is out of specifications.

Remedy

- Check the specifications.

P481 Illegal G code (mill)**Details**

- An illegal G code was used during the milling mode.
- An illegal G code was used during cylindrical interpolation or polar coordinate interpolation.
- The G07.1 command was issued during the tool radius compensation.

Remedy

- Correct the program.

P482 Illegal axis (mill)**Details**

- A rotary axis was commanded during the milling mode.
- Milling was executed though an illegal value was set for the milling axis No.
- Cylindrical interpolation or polar coordinate interpolation was commanded during mirror image.
- Cylindrical interpolation or polar coordinate interpolation was commanded before the tool compensation was completed after the T command.
- G07.1 was commanded when cylindrical interpolation was not possible (there is no rotary axis, or external mirror image is ON).
- An axis other than a cylindrical coordinate system axis was commanded during cylindrical interpolation.

Remedy

- Correct the machining program, parameters and PLC interface signals.

P484 R-pnt ret incomplete (mill)**Details**

- Movement was commanded to an axis that had not completed reference position return during the milling mode.
- Movement was commanded to an axis that had not completed reference position return during cylindrical interpolation or polar coordinate interpolation.

Remedy

- Carry out manual reference position return.

P485 Illegal modal (mill)**Details**

- The milling mode was turned ON during nose R compensation or constant surface speed control.
- A T command was issued during the milling mode.
- The mode was switched from milling to cutting during tool compensation.
- Cylindrical interpolation or polar coordinate interpolation was commanded during the constant surface speed control mode (G96).
- The command unacceptable in the cylindrical interpolation was issued.
- A T command was issued during the cylindrical interpolation or polar coordinate interpolation mode.
- A movement command was issued when the plane was not selected just before or after the G07.1 command.
- A plane selection command was issued during the polar coordinate interpolation mode.
- Cylindrical interpolation or polar coordinate interpolation was commanded during tool radius compensation.
- The G16 plane in which the radius value of a cylinder is "0" was specified.
- A cylindrical interpolation or polar coordinate interpolation command was issued during coordinate rotation by program.

Remedy

- Correct the program.
- Issue G40 or G97 before issuing G12.1.
- Issue a T command before issuing G12.1.
- Issue G40 before issuing G13.1.
- Specify the radius value of a cylinder other than "0", or specify the X axis's current value other than "0" before issuing G12.1/G16.

P486 Milling error**Details**

- The milling command was issued during the mirror image (when parameter or external input is turned ON).
- Polar coordinate interpolation, cylindrical interpolation or milling interpolation was commanded during mirror image for facing tool posts.
- The start command of the cylindrical interpolation or polar coordinate interpolation was issued during the normal line control.

Remedy

- Correct the program.

P501 Cross (G110) impossible**Details**

Mixed control (cross axis control) command (G110) was issued in the following modes.

- During nose R compensation mode
- During pole coordinate interpolation mode
- During cylindrical interpolation mode
- During balance cut mode
- During fixed cycle machining mode
- During facing turret mirror image
- During constant surface speed control mode
- During hobbing mode
- During axis name switch

Remedy

- Correct the program.

P503 Illegal G110 axis

Details

- The commanded axis does not exist.
- The mixed control (cross axis control) (G110) was commanded to the axis for which the mixed control (cross axis control) is disabled.
- The number of axes included in the mixed control (cross axis control) (G110) command is exceeding the maximum number of axes per part system.

Remedy

- Correct the program.

P511 Synchronization M code error

Details

- Two or more synchronization M codes were commanded in the same block.
- The synchronization M code and "!" code were commanded in the same block.
- Synchronization with the M code was commanded in 3rd part system or more. (Synchronization with the M code is valid only in 1st part system or 2nd part system.)

Remedy

- Correct the program.

P520 Control axis superimposition/Designated axis illegal

Details

An axis which was impossible to superimpose was designated as a master axis or superimposing axis.

Remedy

Correct the program.

P521 Illegal synchronization axis

Details

The axis specified as a basic or synchronous axis of synchronization across part systems cannot be synchronized.

Remedy

- Correct the program.

P530 DEC. POINT ERR

Details

A decimal point was added to the address where the decimal point command is not allowed. ("#1274 ext10/bit0 (Type of address enabling/disabling decimal point command)")

Remedy

Do not add any decimal point to the addresses where the decimal point is not allowed.

P544 No spec: Wk instl err cmp

Details

The workpiece installation error compensation function is out of the specifications.

Remedy

- Check the specifications.

P545 Invld cmd in wk instl err cmp

Details

During workpiece installation error compensation, a command impossible to issue (such as G command) was issued.

Remedy

- Check the program. If you wish to issue a command impossible to issue (such as G command) during workpiece installation error compensation, cancel workpiece installation error compensation once.

P546 Wk instl err cmp cmd invalid**Details**

- Workpiece installation error compensation was commanded in a G modal in which commanding it is not allowed.
- An illegal G command was issued in the block that has a workpiece installation error compensation command.

Remedy

- Check the program. Also check the G modals which were issued at commanding the workpiece installation error compensation, and cancel illegal ones.
- Issue the G command in a separate block.

P547 Illegal wk instl err cmp cmd**Details**

A command in which the rotary axis's travel distance exceeds 180 degrees was issued.

Remedy

- Divide the travel command so that the rotary axis's travel distance per block is less than 180 degrees.

P550 No spec: G06.2(NURBS)**Details**

There is no NURBS interpolation option.

Remedy

- Check the specifications.

P551 G06.2 knot error**Details**

The knot (k) command value is smaller than the value for the previous block.

Remedy

- Correct the program.
- Specify the knot by monotone increment.

P552 Start point of 1st G06.2 err**Details**

The block end point immediately before the G06.2 command and the G06.2 first block command value do not match.

Remedy

- Match the G06.2 first block coordinate command value with the previous block end point.

P554 Invl manual interrupt in G6.2**Details**

Manual interruption was executed in a block that applies the G06.2 mode.

Remedy

- Execute the manual interruption in the block that does not apply the G06.2 mode.

P555 Invalid restart during G06.2**Details**

Restart was attempted from the block that applies G06.2 mode.

Remedy

- Restart from the block that does not apply the G06.2 mode.

P600 No spec: Auto TLM**Details**

An automatic tool length measurement command (G37) was issued though it is out of specifications.

Remedy

- Check the specifications.

P601 No spec: Skip**Details**

A skip command (G31) was issued though it is out of specifications.

Remedy

- Check the specifications.

P602 No spec: Multi skip**Details**

A multiple skip command (G31.1, G31.2, G31.3 or G31 Pn) was issued though it is out of specifications.

Remedy

- Check the specifications.

P603 Skip speed 0**Details**

The skip speed is "0".

Remedy

- Specify the skip speed.

P604 TLM illegal axis**Details**

No axis was specified in the automatic tool length measurement block. Otherwise, two or more axes were specified.

Remedy

- Specify only one axis.

P605 T & TLM command in a block**Details**

The T code is in the same block as the automatic tool length measurement block.

Remedy

- Specify the T code before the automatic tool length measurement block.

P606 T cmdnd not found before TLM**Details**

The T code was not yet specified in automatic tool length measurement.

Remedy

- Specify the T code before the automatic tool length measurement block.

P607 TLM illegal signal**Details**

The measurement position arrival signal turned ON before the area specified by the D command or "#8006 ZONE d". Otherwise, the signal remained OFF to the end.

Remedy

- Correct the program.

P608 Skip during radius compen**Details**

A skip command was issued during radius compensation processing.

Remedy

- Issue a radius compensation cancel (G40) command or remove the skip command.

P610 Illegal parameter**Details**

- The parameter setting is not correct.
- G114.1 was commanded while the spindle synchronization was selected with the PLC interface signal.
- G110 was commanded while the mixed control (cross axis control) was selected with the PLC interface signal.
- G125 was commanded while the control axis synchronization across part systems was selected with the PLC interface signal.
- G126 was commanded while the control axis superimposition was selected with the PLC interface signal.

Remedy

- Correct the settings of "#1514 expLinax (Exponential function interpolation linear axis)" and "#1515 expRotax (Exponential function interpolation rotary axis)".
- Correct the program.
- Correct the parameter settings.

P611 No spec: Exponential function**Details**

There is no specification for the exponential interpolation.

Remedy

- Check the specifications.

P612 Exponential function error**Details**

A travel command for exponential interpolation was issued during mirror image for facing tool posts.

Remedy

- Correct the program.

P700 Illegal command value**Details**

Spindle synchronization was commanded to a spindle that is not connected serially.

Remedy

- Correct the program.
- Correct the parameter settings.

P900 No spec: Normal line control**Details**

A normal line control command (G40.1, G41.1, or G42.1) was issued though it is out of specifications.

Remedy

- Check the specifications.

P901 Normal line control axis G92**Details**

A coordinate system preset command (G92) was issued to a normal line control axis during normal line control.

Remedy

- Correct the program.

P902 Normal line control axis error**Details**

- The normal line control axis was set to a linear axis.
- The normal line control axis was set to the linear type rotary axis II axis.
- The normal line control axis has not been set.
- The normal line control axis is the same as the plane selection axis.

Remedy

- Correct the normal line control axis setting.

P903 Plane chg in Normal line ctrl

Details

The plane selection command (G17, G18, or G19) was issued during normal line control.

Remedy

- Delete the plane selection command (G17, G18, or G19) from the program of the normal line control.

P920 No spec: 3D coord conv

Details

There is no specification for 3-dimensional coordinate conversion.

Remedy

- Check the specifications.

P921 Illegal G code at 3D coord

Details

The commanded G code cannot be performed during 3-dimensional coordinate conversion modal.

Remedy

- Refer to "MITSUBISHI CNC 700/70 Series Programming Manual (Machining Center System)" for usable G commands.
- When the parameter "#8158 Init const sur spd" is enabled, disable the parameter or issue the constant surface speed control cancel (G97) command.

P922 Illegal mode at 3D coord

Details

A 3-dimensional coordinate conversion command was issued during a modal for which 3-dimensional coordinate conversion cannot be performed.

Remedy

- Refer to "MITSUBISHI CNC 700/70 Series Programming Manual (Machining Center System)" for usable G commands.

P923 Illegal addr in 3D coord blk

Details

A G code and G68 was commanded in a block though the G code cannot be commanded with G68.

Remedy

- Refer to "MITSUBISHI CNC 700/70 Series Programming Manual (Machining Center System)" for usable G commands.

P930 No spec: Tool axis compen

Details

A tool length compensation along the tool axis command was issued though it is out of specifications.

Remedy

- Check the specifications.

P931 Executing tool axis compen

Details

There is a G code that cannot be commanded during tool length compensation along the tool axis.

Remedy

- Correct the program.

P932 Rot axis parameter error

Details

There is an illegal linear axis name or rotary axis name set in the rotary axis configuration parameters. There is an illegal setting in the parameter concerning the configuration of the inclined surface machining axis.

Remedy

- Set the correct value and turn the power ON again.

P940 No spec: Tool tip control**Details**

There is no specification for tool tip center control.

Remedy

- Check the specifications.

P941 Invalid T tip control command**Details**

A tool tip center control command was issued during a modal for which a tool tip center control command cannot be issued.

Remedy

- Correct the program.

P942 Invalid cmd during T tip ctrl**Details**

A G code that cannot be commanded was issued during tool tip center control.

Remedy

- Correct the program.

P943 Tool posture command illegal**Details**

In tool tip center control type 1, if the signs at the tool-side rotary axis or table base-side rotary axis start and finish points differ, a tool base-side rotary axis or table workpiece-side rotary axis rotation exists for the same block, and does not pass a singular point.

In tool tip center control type 2, the posture vector command is incorrect.

Remedy

- Correct the program.

P950 No spec: Tilt face machining**Details**

Inclined surface machining option is not supported.

Remedy

- Check the specifications.

P951 Ill cmd in tilt face machining**Details**

A forbidden command (G command, etc) was issued during inclined surface machining.

Remedy

- Check the program. If you want to execute a command (G command, etc) that is unavailable during inclined surface machining, cancel the inclined surface machining.

P952 Inclined face cut prohibited**Details**

Inclined surface machining was commanded during the mode where the machining is unavailable. Inclined surface machining was commanded during interruption.

Remedy

- Check the program and see whether any unavailable mode is included during inclined surface machining command. If any, cancel that mode.

P953 Tool axis dir cntrl prohibited**Details**

Tool axis direction control was commanded during the mode where the control is unavailable.

Remedy

- Check the program and see whether any unavailable mode is included during tool axis direction control. If any, cancel that mode.

P954 Inclined face command error

Details

The address to issue the inclined surface machining command is incorrect.

Remedy

- Check the program.

P955 Inclined face coord illegal

Details

Impossible to define an inclined surface with the values you specified.

Remedy

- Check the program.

P956 G68.2P10 surface not defined

Details

The coordinate system for the machining surface selected with G68.2P10 has not been defined.

Remedy

- Set the machining surface so that the coordinate system can be defined.

P957 Tool axis dir ctrl cmp amt 0

Details

When the tool axis direction control type 2 (G53.6) was commanded, a tool length compensation No. whose compensation amount is 0 was commanded.

Remedy

- Correct the program. Set the tool length compensation amount, or command a tool length compensation No. whose compensation amount is not 0.

P960 No spec: Direct command mode

Details

G05 P4 was commanded while direct command mode option is OFF.

Remedy

- Check the specifications.

P961 Invalid during dir cmnd mode

Details

- A G code other than G05 P0 was commanded in direct command mode.
- A sequence No. command, F code command, MSTB command or variable command was issued.
- A corner chamfering command or corner R command was issued.
- A travel command was issued to an axis that had not been command in the G05 P4 block.

Remedy

- Check the program.

P962 Dir cmnd mode cmnd invalid

Details

G05 P4 was commanded in a modal where direct command mode is not available.

Remedy

- Check the program.

P963 Illegal direct cmnd mode cmnd

Details

The commanded coordinate value was beyond the maximum travel distance in direct command mode.

Remedy

- Correct the coordinate value in direct command mode.

P990 PREPRO error**Details**

Combining commands that required pre-reading (nose R offset, corner chamfering/corner rounding, geometric I, geometric IB, and compound type fixed cycle for turning machining) resulted in eight or more pre-read blocks.

Remedy

- Delete some or all of the combinations of commands that require pre-reading.

Appendix 2

Order of G Function Command Priority

Appendix 2 Order of G Function Command Priority

When commanded in the same block, ○ indicates that the both commands are executed simultaneously. Command in a separate block when possible.

Commanded G code	G group			
	04 G22, G23	07 G40 to G42 G46	09 G70 to G79 G80 G81 to G89	11 G50.2, G51.2
G04 Dwell	G04 is executed. G22 and G23 are ignored. (Note)	G04 is executed. G40 to G42 are ignored. (Note)	G04 is executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are ignored. (Note)	G04 is executed. G50.2 and G51.2 are ignored. (Note)
G09 Exact stop check	○	○	G09 is executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are executed. (Note)	○
G10, G11 Program data setting	G10 and G11 are executed. G22 and G23 are ignored. (Note)	G10 and G11 are executed. G40 to G42 are ignored. (Note)	G10 and G11 are executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are ignored. (Note)	○
(System 6 and 7) G22, G23 Soft limit	○	○	G22 and G23 are executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are executed. (Note)	○
G27 to G30 Reference position check/return	G27 to G30 are executed. G22 and G23 are ignored. (Note)	G27 to G30 are executed. G40 to G42 are ignored. (Note)	G27 to G30 are executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are ignored. (Note)	G27 to G30 are executed. G50.2 and G51.2 are ignored. (Note)
G30.1 to G30.6 Tool change position return	○	○	G30.1 to G30.6 are executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are ignored. (Note)	○
G31, G31.1 to G31.3 Skip/multi-step skip	○	○	G31, G31.1 to G31.5 are executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are ignored. (Note)	○
G37 Automatic tool length measurement	G37 is executed. G22 and G23 are ignored. (Note)	G37 is executed. G40 to G42 are ignored. (Note)	G37 is executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are ignored. (Note)	G37 is executed. G50.2 and G51.2 are ignored. (Note)
G52 Local coordinate system	G52 is executed. G22 and G23 are ignored. (Note)	G52 is executed. G40 to G42 are ignored. (Note)	G52 is executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are executed. (Note)	G52 is executed. G50.2 and G51.2 are ignored. (Note)
G53 Machine coordinate system	G53 is executed. G22 and G23 are ignored. (Note)	G53 is executed. G40 to G42 are ignored. (Note)	G53 is executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are ignored. (Note)	G53 is executed. G50.2 and G51.2 are ignored. (Note)
G65 User macro calls	○	○	G65 is executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are executed. (Note)	○

Commanded G code	G group			
	04 G22, G23	07 G40 to G42 G46	09 G70 to G79 G80 G81 to G89	11 G50.2, G51.2
G92 Spindle clamp speed setting/ coordinate system setting	○	○	G92 is executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are ignored. (Note)	○
G92.1(G50.3) Workpiece coordinate system preset	○	○	G92.1 is executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are ignored. (Note)	○
G110 Mixed control (cross axis control) I	○	○	G110 is executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are executed. (Note)	○
G111 Axis name switch	○	○	G110 is executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are executed. (Note)	○
G113, G114.1 Spindle synchronization control	○	○	G113 and G114.1 are executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are executed. (Note)	○
G113, G114.2 Polygon machining	○	○	G113 and G114.2 are executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are executed. (Note)	○
G114.3 Spindle synchronization control II	○	○	G114.3 is executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are executed. (Note)	○
G115, G116 Start point designation synchronization	○	○	G115 and G116 are executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are ignored. (Note)	○
G117 M code output during axis travelling	○	○	G117 is executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are executed. (Note)	○
G125 Control axis synchronization across part systems	○	○	G125 is executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are executed. (Note)	○
G126 Control axis superimposition	○	○	G126 is executed. G70 to G79 are ignored. G80 is executed. G81 to G89 are executed. (Note)	○

(Note) A program error (P45) will occur if they are commanded in the same block.
This error can be avoided by setting the parameter "#1241 bit0 (No G-CODE COMB. Error)" to "1",
but be aware that one of the G commands is ignored.

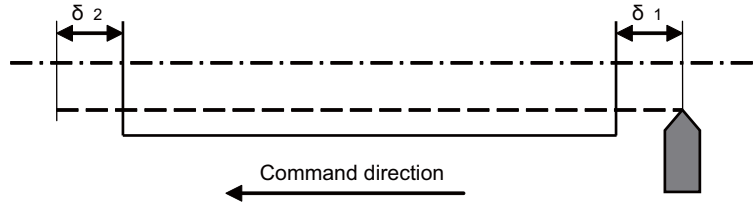
Appendix 3

Supplementary Explanation for Incomplete Thread Area of Thread Control



Function and purpose

The delay caused by the automatic acceleration/deceleration and delay caused by the position loop in the servo system create an illegal pitch near the start and end points of thread cutting. When programming steps must be taken to assign thread cutting commands which include a margin for the approach distance 1 and for the length of the area 2 where the thread is incomplete during chamfering, as shown in the figure below.



- δ 1 : Approach distance
- δ 2 : Area where thread is incomplete during chamfering



Detailed description

Approach distance [δ 1]

(1) When Ts is not equal to zero

$$\delta_1 = \frac{F}{60} t1 - \frac{F}{60} (Ts + Tp - \frac{Tp^2 e^{-\frac{t1}{Tp}} - Ts^2 e^{-\frac{t1}{Ts}}}{Tp - Ts}) \text{ (mm)}$$

- Where F : Thread cutting speed (mm/min)
- t1 : Time taken until pitch error reaches allowable limit "a" (s)
- Ts : Acceleration/deceleration time constant (s)
- Tp : Position loop time constant (s) = 1/Position loop gain(PGN1)
- During the SHG control, a position loop gain should be calculated by multiplying $\sqrt{2}$.

If "p" is the pitch and "ΔP" is the pitch error, then allowable limit "a" will be:

$$a = \frac{1}{Tp - Ts} (Tp e^{-\frac{t1}{Tp}} - Ts e^{-\frac{t1}{Ts}})$$

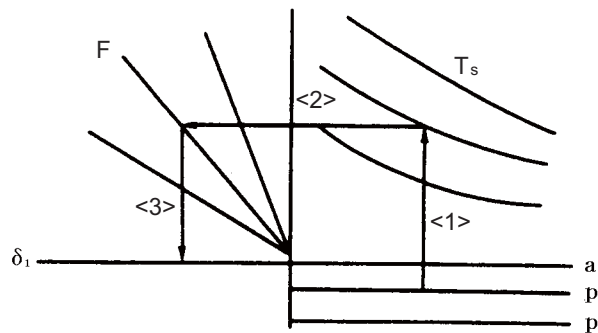
(2) When T_s is equal to zero

$$\delta_1 = \frac{F}{60} t_1 - \frac{F}{60} (T_p - T_p e^{\frac{t_1}{T_p}}) \text{ (mm)}$$

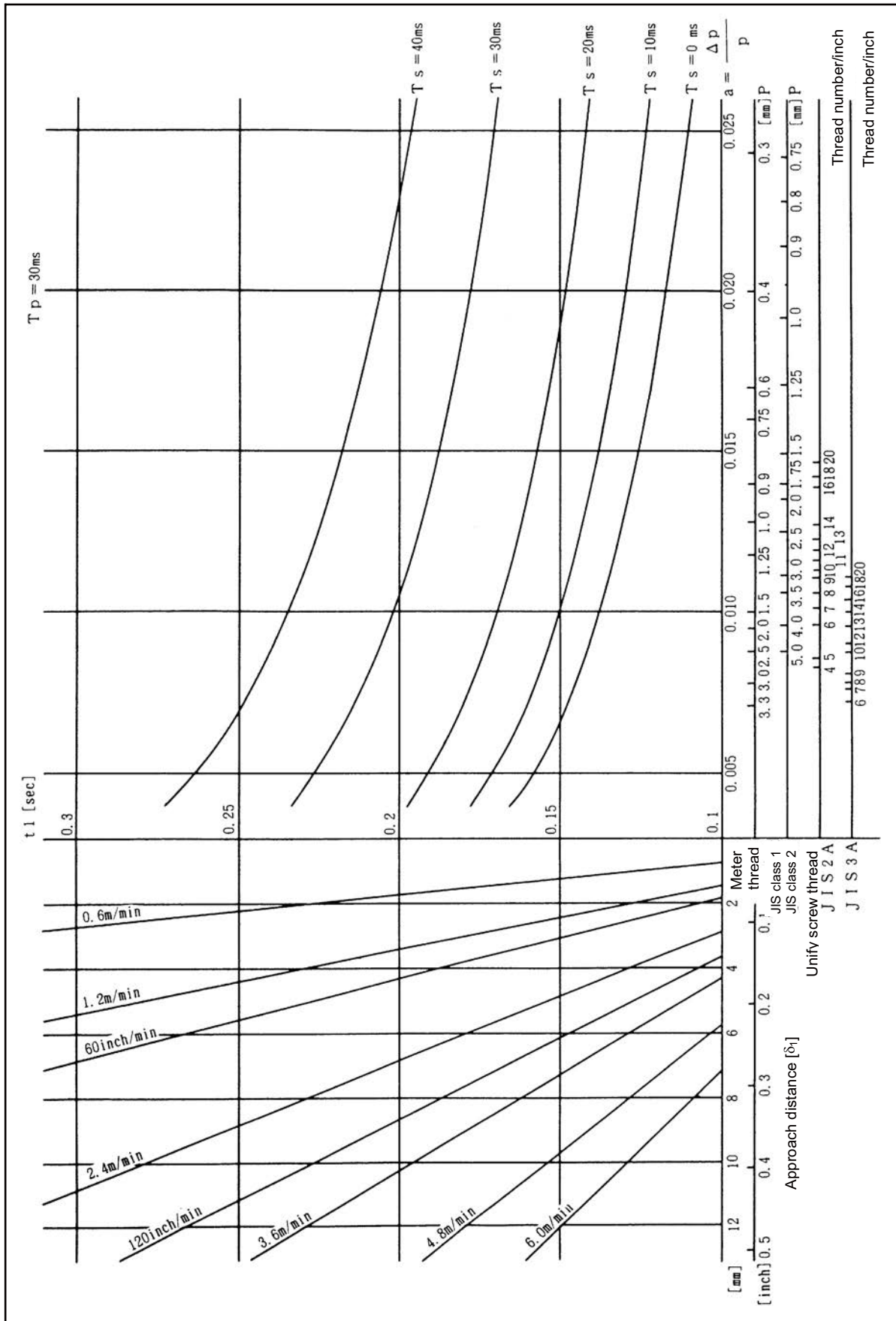
$$a = e^{-\frac{t_1}{T_p}}$$

Since the calculation of approach distance δ_1 is a complicated procedure, δ_1 is normally determined from the chart on the next page. This chart is used as follows.

- Find the position on the p axis scale represents by the thread grade and pitch [P], and follow the
- perpendicular <1> drawn upward to find the point where it intersects with the curve of acceleration/ deceleration time constant [T_s].
 - Follow horizontal line <2> and find where it intersects with the thread cutting speed [F].
 - Follow perpendicular <3> and find approach distance [δ_1] on the scale at the point where it intersects with the δ_1 axis.



Approach distance δ_1 calculation chart (when position loop gain = 33)



Length of area where thread is incomplete during chamfering [δ 2]

$$\delta_2 = (T_s + T_p) \frac{F}{60} \quad (\text{mm})$$

Where F : Thread cutting speed (mm/min)

Ts : Acceleration/deceleration time constant (s)

Tp : Position loop time constant (s) = 1/ position loop gain (PGN1)

During the SHG control, a position loop gain should be calculated by multiplying $\sqrt{2}$.

(Note) When proceeding with chamfering during a thread cutting cycle, the length of the area where the thread is incomplete is equivalent to the value produced by adding δ 2 determined by the above formula to the chamfering pitch set by parameter.

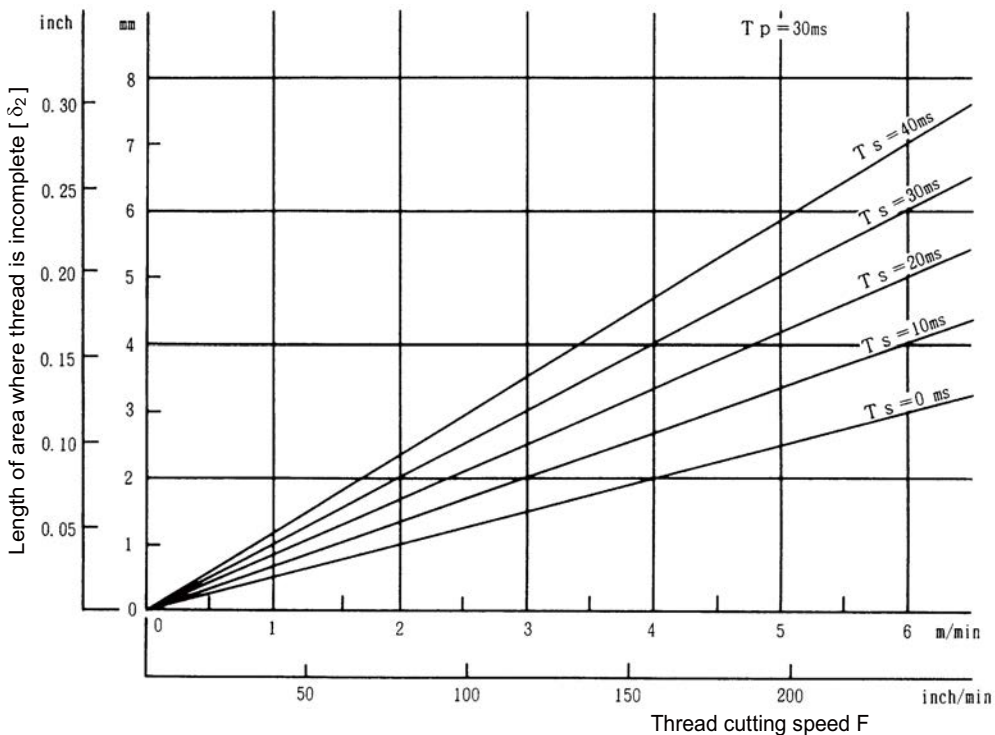


Chart to calculate length of area where thread is incomplete [δ 2]

Appendix 4

Parameter Input by Program N No. (G10 L50, G11)



Function and purpose

The parameters set from the setting and display unit can be changed in the machining programs.
 (Note) G10 L50 is for specific models only (no specification for M700V/M70V Series).



Command format

```
G10 L50;
P__N__H□__ ;
P__A__N__D__ ;
P__A__N__S__ ;
P__A axis No. N__L__ ; ... Data setting command
```

P	Major classification No.
N	Data No.
A	Axis No.
H	Bit type data
D	Byte type data
S	S word type data
L	2 word type data

G11 ; ... Data setting mode cancel (data setting completed)

```
With axis-common data
Axis-common bit-type parameter ..... P__N__H□__ ;
Axis-common byte-type parameter ..... P__N__D__ ;
Axis-common word-type parameter ..... P__N__S__ ;
Axis-common 2-word-type parameter ..... P__N__L__ ;
With axis-independent data
Axis-independent bit-type parameter ..... P__A__N__H□__ ;
Axis-independent byte-type parameter ..... P__A__N__D__ ;
Axis-independent word-type parameter ..... P__A__N__S__ ;
Axis-independent 2-word-type parameter ..... P__A__N__L__ ;
```

There are 8 types of data formats according to the type of parameter (axis-common and axis-independent) and data type, as listed below.

- (Note 1) The sequence of addresses in a block must be as shown above.
- (Note 2) For a bit type parameter, the data type will be H □ (□ is a value between 0 and 7).
- (Note 3) The axis No. is set in the following manner: 1st axis is 1, 2nd axis is 2, and so forth.
 When using multi-part systems, the 1st axis in each part system is set as 1, the 2nd axis is set as 2, and so forth.
- (Note 4) Command G10 L50, G11 in independent blocks. A program error (P33, P421) will occur if not commanded in independent blocks.



Detailed description

- (1) The units in the table indicate the minimum setting units for the parameter data.
- (2) The setting ranges given in the table are the setting ranges on the screen. Designate parameters related to the length by doubling the input setting unit.
 (Example 1) If the input setting unit in the metric system is "B" (0.001mm), set 30mm in the parameter.
 L60000
 (Example 2) If the input setting unit in the inch system is "B" (0.0001inch), set 5inch in the parameter.
 L100000
- (3) The binary type parameters must be converted into byte type data, and commanded with a decimal data following address D.
 (Example 1) Binary data
 01010101B = 55H = 85D Command 85
 (Example 2) ASCII code
 "M" = 01001101B = 4DH = 77D Command 77
 (B indicates Binary, H indicates Hexadecimal, and D indicates Decimal.)
- (4) In a system with a section No. (P) (in case of G10 L50), the section No. is assigned according to the function increment after No. 20. Thus, the No. may be duplicated with the parameters that can be set with other section Nos.

[Machining parameters]								
#	Item		P	A	N	Data type	Setting range	Unit
8007	OVERRIDE		11	-	1010	D	0 to 100	%
8008	MAX ANGLE		11	-	756	L	0 to 180	°
8009	DSC. ZONE		11	-	760	L	0 to 199999998	0.5 μm
8010	ABS. MAX.		11	-	776	L	0 to 199998	0.5 μm
8011	INC. MAX.		11	-	780	L	0 to 199998	0.5 μm
8013	G83 n		11	-	832	L	0 to 199999998	0.5 μm
8014	CDZ-VALE		11	-	1012	D	0 to 127	0.1 lead
8015	CDZ-ANGLE		11	-	1011	D	0 to 89	°
8016	G71 MINIMUM		11	-	788	L	0 to 199998	0.5 μm
8017	G71 DELTA-D		11	-	792	L	0 to 199998	0.5 μm
8051	G71 THICK		11	-	784	L	0 to 199999998	0.5 μm
8052	G71 PULL UP		11	-	796	L	0 to 199999998	0.5 μm
8053	G73 U		11	-	800	L	± 199999998	0.5 μm
8054	G73 W		11	-	804	L	± 199999998	0.5 μm
8055	G73 TIMES		11	-	808	L	0 to 99999	
8056	G74 RETRACT		11	-	820	L	0 to 199998	0.5 μm
8057	G76 LAST-D		11	-	824	L	0 to 199998	0.5 μm
8058	G76 TIMES		11	-	997	D	0 to 99	
8059	G76 ANGLE		11	-	998	D	0 to 99	

[Axis parameters]								
#	Item		P	A	N	Data type	Setting range	Unit
8202	OT-CHECK OFF		2	No.	897	H2	0 to 1	
8204	OT-CHECK-N		2	No.	916	L	± 199999998	0.5 μm
8205	OT-CHECK-P		2	No.	912	L	± 199999998	0.5 μm

[Barrier data]								
#	Item		P	A	N	Data type	Setting range	Unit
8300	(P0) X		11	-	1128	L	± 199999998	0.5 μm
8301	(P1) X		11	No.	1136	L	± 199999998	0.5 μm
	Z		11	No.	1160	L	± 199999998	0.5 μm
8302	(P2) X		11	No.	1140	L	± 199999998	0.5 μm
	Z		11	No.	1164	L	± 199999998	0.5 μm
8303	(P3) X		11	No.	1144	L	± 199999998	0.5 μm
	Z		11	No.	1168	L	± 199999998	0.5 μm
8304	(P4) X		11	No.	1148	L	± 199999998	0.5 μm
	Z		11	No.	1172	L	± 199999998	0.5 μm
8305	(P5) X		11	No.	1152	L	± 199999998	0.5 μm
	Z		11	No.	1176	L	± 199999998	0.5 μm
8306	(P6) X		11	No.	1156	L	± 199999998	0.5 μm
	Z		11	No.	1180	L	± 199999998	0.5 μm

[BASE SPEC PARAM]								
#	Item		P	A	N	Data type	Setting range	Unit
1595	hobm		11	-	595	D	1 to the number of spindles	
1596	hobs		11	-	596	D	1 to the number of the axis in the part system	

[BASE SPEC PARAM]								
#	Item		P	A	N	Data type	Setting range	Unit
2592	Extstnum		2	No.	2592	S	4101 to 5508	
2593	Extset		2	No.	2593	D	0 to 32	
2594	Extsc		2	No.	2594	D	0 to 99	



Program example

```
G10 L50 ;
P11 N1010 D30 ; parameter "#8007 OVERRIDE" to "30"
G11 ;
```

Index

Numbers

2nd, 3rd, and 4th Reference Position (Zero point) Return ; G30.....	672
2-part System Simultaneous Thread Cutting Cycle	643
2-part System Simultaneous Thread Cutting Cycle (MITSUBISHI special format) ; G76.1,G76.2.....	650
2-part System Simultaneous Thread Cutting Cycle I ; G76.1.....	644
2-part System Simultaneous Thread Cutting Cycle II ; G76.2.....	646
2-part System Simultaneous Thread Cutting Cycle Parameter Setting Command ; G76.....	643

A

Absolute/Incremental Value Commands ; G90,G91.....	32
Acceleration/Deceleration Mode Change in Hole Drilling Cycle	424
ASCII Code Macro	473
Automatic Acceleration/Deceleration	163
Automatic Coordinate System Setting	664
Automatic Corner Override ; G62.....	189
Automatic Tool Length Measurement ; G37	698
Axis Name Switch ; G111	652

B

Balance Cut ; G15,G14.....	600
Basic Machine Coordinate System Selection ; G53.....	665
Basic Machine, Workpiece and Local Coordinate Systems	662
Boring Cycle ; G85.....	444
Boring Cycle ; G89.....	445

C

Chuck Barrier/Tailstock Barrier ; G22,G23.....	692
Circular Interpolation ; G02,G03	53
Circular Thread Cutting ; G35,G36	71
Common Variables	480
Compensation Data Input by Program ; G10 L2/L10/L11, G11	339
Compound Thread Cutting Cycle ; G76.....	389
Compound Type Fixed Cycle for Turning Machining	362
Compound Type Fixed Cycle for Turning Machining (MITSUBISHI CNC special format) ; G71,G73,G74,G76	395
Constant Lead Thread Cutting ; G33	61
Constant Surface Speed Control ; G96,G97.....	211
Continuous Thread Cutting ; G33	67
Control Axis Superimposition I ; G126	622
Control Axis Synchronization Across Part System I ; G125	637
Control Commands	526
Coordinate Rotation by Program ; G68.1/G69.1 ...	586

Coordinate System for Rotary Axis.....	688
Coordinate System Setting ; G92	666
Coordinate Systems and Coordinate Zero Point Symbols	4
Coordinate Words and Control Axes	2
Coordinate Words and Control Axes	660
Corner Chamfering Expansion/Corner Rounding Expansion	549
Corner Chamfering Expansion/Corner Rounding Expansion	556
Corner Chamfering I ; G01 X_ Z_ ,C_/I_/K_/C_..	545
Corner Chamfering I/Corner Rounding I	545
Corner Chamfering II ; G01/G02/G03 X_ Z_ ,C_/I_/K_/C_	552
Corner Chamfering II / Corner Rounding II	552
Corner Rounding I ; G01 X_ Z_ ,R_/R_.....	547
Corner Rounding II ; G01/G02/G03 X_ Z_ ,R_/R_	554
Counting the Tool Life	345
Cutting Feed Constant Inclination Acceleration/Deceleration	172
Cutting Feedrate	149
Cutting Mode ; G64.....	195
Cylindrical Interpolation ; G07.1 (only 6 and 7 in G code list)	126

D

Deceleration Check	185
Decimal Point Input	37
Deep Hole Drilling Cycle ; G83.....	431
Deep Hole Drilling Cycle 2 ; G83.2.....	418
Detailed Description for Macro Call Instruction ...	471
Drilling Cycle High-Speed Retract	423
Drilling Cycle, Counter Boring Cycle ; G82.....	430
Drilling Cycle, Spot Drilling Cycle ; G81.....	429
Dwell (Time Designation) ; G04.....	198

E

Exact Stop Check ; G09	180
Exact Stop Check Mode ; G61	184
Expanded Method at Starting Tool Compensation	288
Exponential Interpolation ; G02.3,G03.3.....	141
External Output Commands ; POPEN, PCLOS, DPRNT.....	529
External Workpiece Coordinate System Compensation (#2501, #2601)	501

F

F1-digit Feed.....	150
Face Boring Cycle (Longitudinal boring cycle) ; G85 (G89)	416
Face Cut-Off Cycle ; G74	385
Face Cutting Cycle ; G79.....	357
Face Deep Hole Drilling Cycle 1 (Longitudinal deep hole drilling cycle 1) ; G83 (G87)	403
Face Rough Cutting Cycle ; G72.....	377
Face Tapping Cycle (Longitudinal tapping cycle) / Face Reverse Tapping Cycle (Longitudinal reverse tapping cycle) ; G84 (G88) / G84.1 (G88.1).....	405

Feed Functions	93
Feed Hold, Feedrate Override, G09 Valid/Invalid (#3004)	495
Feed Per Minute/Feed Per Revolution (Asynchronous Feed/Synchronous Feed) ; G94,G95.....	153
Feedrate Designation and Effects on Control Axes.....	156
File Format	16
Finishing Cycle ; G70.....	383
Fixed Cycle for Drilling	400
Fixed Cycle for Drilling (MITSUBISHI CNC special format)	426
Fixed Cycle for Drilling Cancel; G80	420
Fixed Cycles for Turning Machining	350
Fixed Cycles for Turning Machining (MITSUBISHI CNC special format) ; G77,G78,G79.....	360
Formed Material Rough Cutting Cycle ; G73.....	379

G

G code	22
G code Lists	22
G Code Macro Call	469
G Command Modals (#4001-#4021, #4201-#4221)	497
G1 -> G0 Deceleration Check.....	187
G1 -> G1 Deceleration Check.....	188
G41/G42 Commands and I, J, K Designation	324
General Precautions for Tool Nose Radius Compensation	332
Geometric	558
Geometric I ; G01 A_.....	558
Geometric IB	560
Geometric IB (Automatic calculation of linear - arc intersection) ; G01 A_ , G02/G03 R_H_.....	569
Geometric IB (Automatic calculation of linear - arc intersection) ; G01 A_ , G02/G03 P_Q_H_.....	565
Geometric IB (Automatic calculation of two-arc contact) ; G02/G03 P_Q_/R_.....	561

H

Helical Interpolation ; G17,G18,G19 and G02,G03	75
---	----

I

Inch Thread Cutting ; G33	65
Inch/Metric Conversion ; G20,G21	35
Index Table Indexing	205
Indexing Increment	9
Initial Point and R Point Level Return ; G98,G99.....	421
Input Setting Unit	8
Integrating Time (#3001, #3002)	494
Interference Check	116
Interference Check	333
Interrupt during Corner Chamfering/Interrupt during Corner Rounding	551

Interrupt during Corner Chamfering/Interrupt during Corner Rounding	556
Interrupts during Tool Nose Radius Compensation	329

L

Linear Angle Command ; G01 X_/Z_ A_/A_	557
Linear Interpolation ; G01	51
Local Coordinate System Setting ; G52	682
Local Variables (#1 to #33)	481
Longitudinal Cut-off Cycle ; G75	387
Longitudinal Cutting Cycle ; G77	351
Longitudinal Rough Cutting Cycle ; G71	363

M

M Code Output during Axis Traveling ; G117.....	207
Machine Zero Point and 2nd Reference Position (Zero point)	663
Macro Call Instruction	461
Macro Interface Inputs/Outputs (#1000 to #1035, #1100 to #1135, #1200 to #1295, #1300 to #1395)	484
Macro Interruption ; M96,M97	575
Message Display and Stop (#3006)	495
Milling Interpolation ; G12.1.....	79
Milling Interpolation Control and Command Axes ..	82
Mirror Image (#3007)	496
Mirror Image for Facing Tool Posts ; G68,G69	535
Miscellaneous Command Macro Call (for M, S, T, B Code Macro Call)	470
Miscellaneous Functions	94
Miscellaneous Functions (M8-digits)	202
Mixed Control (Cross Axis Control) I ; G110	615
Modal Call A (Movement Command Call) ; G66 ..	465
Modal Call B (for each block) ; G66.1	467
Modal, unmodal	22
Multiple-spindle Control I (spindle control command) ; S ○ =	274
Multiple-spindle Control I (spindle selection command) ; G43.1,G44.1,G47.1	275
Multiple-spindle Control.....	273
Multiple-spindle Control II.....	279
Multi-step Skip Function 1 ; G31.n, G04	707
Multi-step Skip Function 2 ; G31 P.....	709

N

NC Alarm (#3000)	493
Number of Workpiece Machining Times (#3901, #3902)	501

O

Operation Commands	521
Optional Block Skip	18
Optional Block Skip Addition ; /n	20
Optional Block Skip; /	18
Other Modals (#4101 - #4120, #4301 - #4320)	498
Other Operations during Tool Nose Radius Compensation	316

P

Parameter Input by Program ;	
G10 L70/L100, G11	573
Plane Selection ; G17,G18,G19.....	59
Polar Coordinate Interpolation ; G12.1,G13.1/ G112,G113 (Only 6, 7 in G code list)	134
Position Information (#5001 - #5140 + n)	499
Positioning (Rapid Traverse) ; G00.....	44
Precautions	533
Precautions Before Starting Machining	28
Precautions for Compound Type Fixed Cycle for Turning Machining; G70 to G76	393
Precautions for Using Fixed Cycle for Drilling (MITSUBISHI CNC special format)	446
Precautions for Using Spindle Synchronization Control	234
Precautions When Using a Fixed Cycle for Drilling	420
Preparatory Function.....	88
Pre-read Buffers	30
Program Format	12
Program Support Functions	93
Programmable Current Limitation ; G10 L14 ;	715

R

R Device Access Variables (#50000 - #50749, #51000 - #51749, #52000 - #52749)	515
R Specification Circular Interpolation ; G02,G03	57
Radius/Diameter Designation	34
Rapid Traverse Constant Inclination Acceleration/Deceleration	167
Rapid Traverse Rate	148
Reading PLC data (#100100-#100103,#100110)	510
Reading The Parameters (#100000-#100002, #100010)	506
Reference Position (Zero point) Return ; G28,G29.....	668
Reference Position Check ; G27	675

S

Secondary Miscellaneous Functions (A8-digits, B8-digits or C8-digits)	204
Selecting a Plane during the Milling Mode ; G17,G19,G16	84
Selecting Milling Mode	81
Setting Milling Coordinate System	86
Setting of Workpiece Coordinates in Fixed Cycle Mode	422
Simple Macro Calls ; G65	461
Skip Function ; G31.....	702
Speed Change Skip ; G31 Fn	711
Speed Clamp	179
Spindle Clamp Speed Setting ; G92	213
Spindle Functions.....	210
Spindle Synchronization Control I ; G114.1	219
Spindle Synchronization Control II	229
Spindle Synchronization.....	218
Spindle/C Axis Control	215
Stepping Cycle ; G83.1	433
Stored Stroke Limit ; G22,G23	696
Subprogram Call ; M198	454

Subprogram Call ; M98,M99.....	448
Subprogram Control; M98, M99, M198	448
Suppression of Single Block Stop and Miscellaneous Function Finish Signal Waiting (#3003)	494
Switching from Milling Mode to Turning Mode; G13.1	93
Synchronous Tapping Cycle ; G84.2.....	439

T

Table of G Code Lists	23
Tapping Cycle ; G84.....	435
Tapping Mode ; G63	194
Thread Cutting Cycle ; G78	354
Thread Cutting Mode	161
Thread Cutting.....	61
Time Reading Variables (#3001, #3002, #3011, #3012)	513
Timing Synchronization Operation (! code) ; !L	604
Timing Synchronization Operation Function Using M codes ; M***	611
Timing Synchronization Operation with Start Point Designated (Type 1) ; G115.....	607
Timing Synchronization Operation with Start Point Designated (Type 2) ; G116.....	609
Timing Synchronization Operation.....	604
Tool Change Position Return ; G30.1 - G30.5.....	583
Tool Compensation	491
Tool Compensation Start	287
Tool Compensation.....	286
Tool Functions (T8-digit BCD)	284
Tool Length Compensation	95
Tool Length Compensation.....	290
Tool Life Management (#60000 - #63016)	502
Tool Life Management II ; G10 L3, G11	342
Tool Nose Point and Compensation Directions ...	295
Tool Nose R Compensation ; G40,G41,G42,G46.....	293
Tool Nose Radius Compensation Operations	298
Tool Nose Wear Compensation	292
Tool Radius Compensation	98
Tool Radius Compensation Operation	99
Tool Spindle Synchronization IB (Spindle-Spindle, Polygon) ; G51.2 (only 6 and 7 in G code list)	245
Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon) ; G51.2 (only 6 and 7 in G code list)	252
Tool Spindle Synchronization II (Hobbing) ; G114.3	255
Tool Spindle Synchronization IA (Spindle-Spindle, Polygon) ; G114.2.....	236
Types of Variables	480

U

User Macro	460
------------------	-----

V

Variable.....	478
Variable Commands	455
Variable Lead Thread Cutting ; G34.....	68

W

Workpiece Coordinate System Compensation (#5201 - #532n).....	492
Workpiece Coordinate System Preset ; G92.1	683
Workpiece Coordinate System Setting and Offset ; G54 to G59 (G54.1)	676

Revision History

Date of revision	Manual No.	Revision details
Nov. 2008	IB(NA)1500924-A	First edition created.
Dec. 2009	IB(NA)1500924-B	<ul style="list-style-type: none"> - Reviewed "Precautions for Safety". - Corrected the items below. <ul style="list-style-type: none"> 10.4 Constant Surface Speed Control; G96, G97 10.5 Spindle Clamp Speed Setting; G92 - Corrected the mistakes.
Mar. 2010	IB(NA)1500924-C	<ul style="list-style-type: none"> - Conducted a thorough review. Modified the structures of the following chapters. <ul style="list-style-type: none"> 3. Program Formats <ul style="list-style-type: none"> Changed the chapter title. (Previously "Data Formats") Removed 3.1 - 3.3, 3.5, 3.6 in the previous version. Newly created 3.1 and 3.2. Changed the title of 3.4 and divided it into 3 chapters. (Newly created 3.4.1) 4. Pre-read Buffers <ul style="list-style-type: none"> Changed the chapter title. (Previously "Buffer Register") Removed 4.1 in the previous version. 6.1 Positioning (Rapid Traverse); G00 <ul style="list-style-type: none"> Made a major change to the structure. 6.8.10 Tool Length Offset <ul style="list-style-type: none"> Changed the chapter title. (Previously "Tool Offset Functions") 6.8.11 Tool Radius Compensation <ul style="list-style-type: none"> Separated the explanation on the tool radius compensation from 6.8.10 in the previous version. 6.8.11.1 Tool Radius Compensation Operation <ul style="list-style-type: none"> Separated the explanation on the tool radius compensation operation from 6.8.10 in the previous version. 6.8.11.2 Interference Check <ul style="list-style-type: none"> Changed the chapter title of 6.8.11 in the previous version. (Included in the explanation on tool radius compensation) 10. Spindle Functions <ul style="list-style-type: none"> Removed 10.1 and 10.2 in the previous version. 13.5 Fixed Cycle for Drilling <ul style="list-style-type: none"> Moved 13.6.1 in the previous version to 13.5.1 Moved 13.6.2 in the previous version to 13.5.2 Added 13.5.3. Moved 13.5.1 - 13.5.6 in the previous version to 13.5.4 - 13.5.9 13.6 Fixed Cycle for Drilling (MITSUBISHI CNC special format) <ul style="list-style-type: none"> Divided the contents of 13.6 into 13.6.1 - 13.6.9. 13.9.2 Macro Call Instruction <ul style="list-style-type: none"> Divided the contents of 13.9.2 into 13.9.2.1 - 13.9.2.6. 13.9.5 Types of Variables <ul style="list-style-type: none"> Divided the contents of 13.9.5 into 13.9.5.1 - 13.9.5.21. 13.14 Geometric <ul style="list-style-type: none"> Divided the contents of 13.14.2 into 13.14.3 - 13.14.5. - Added/Changed the following chapters in order to support the system software G0 version. <ul style="list-style-type: none"> 13.20 Control Axis Superimposition 13.9.5.21 R Device Access Variables <ul style="list-style-type: none"> (#50000 - #50749, #51000 - #51749, #52000 - #52749) 13.9.8 External Output Commands; POPEN, PCLOS, DPRNT - Corrected the mistakes.
Jun. 2010	IB(NA)1500924-D	<ul style="list-style-type: none"> - Changed the chapter order as follows. <ul style="list-style-type: none"> "13.5.1 to 13.5.3" in the previous version to "13.5.7 to 13.5.9". "13.5.4 to 13.5.9" in the previous version to "13.5.1 to 13.5.6". - Corrected the mistakes.

Date of revision	Manual No.	Revision details
Nov. 2011	IB(NA)1500924-E	<p>- The following chapters are added or changed corresponding to S/W version H0.</p> <p>[Added] Handling of our product 9.4 M Code Output during Axis Traveling ; G117 13.11 Coordinate Rotation by Program ; G68/G69 13.21 Mixed Control (Cross Axis Control I ; G110 13.23 Control Axis Synchronization Across Part Systems ; G125 13.26 Axis Name Switch ; G111 Appendix 3 Order of G Function Command Priority</p> <p>[Changed] Precautions for Safety 3.4.3 Table of G Code Lists 5.4 Decimal Point Input 6.3 Circular Interpolation ; G02, G03 6.9 Cylindrical Interpolation ; G07.1 (only 6 and 7 in G code list) 7.3 F1-digit Feed 13.5.2 Face Tapping Cycle (Longitudinal tapping cycle) / Face Reverse Tapping Cycle (Longitudinal reverse tapping cycle) ; G84 (G88) / G84.1 (G88.1) 13.7 Subprogram Control; M98, M99, M198 13.8 Variable Commands 13.22 Control Axis Superimposition ; G126 14.6 Coordinate System Setting ; G92 16.2 Skip Function ; G31 16.3 Multi-step Skip Function 1 ; G31.n, G04 16.4 Multi-step Skip Function 2 ; G31 P 16.5 Speed Change Skip ; G31 Fn</p> <p>- Corrected the mistakes.</p>
Sep. 2013	IB(NA)1500924-F	<p>- The following chapters are added or changed corresponding to S/W version J2.</p> <p>[Added] 7.9 Cutting Feed Constant Inclination Acceleration/Deceleration 13.5.10 Acceleration/Deceleration Mode Change in Hole Drilling Cycle</p> <p>[Changed] 3.2 File Format 3.3.1 Optional Block Skip; / 3.4.3 Table of G Code Lists 5.1 Absolute/Incremental Value Commands ; G90,G91 6.6.1 Constant Lead Thread Cutting ; G33 6.6.2 Inch Thread Cutting ; G33 7.2 Cutting Feedrate 7.8 Rapid Traverse Constant Inclination Acceleration/Deceleration 7.15 Cutting Mode ; G64 10.8 Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon) ; G51.2 (only 6 and 7 in G code list) 10.9 Tool Spindle Synchronization II (Hobbing) ; G114.3 10.10 Multiple-spindle Control 10.10.1 Multiple-spindle Control I (spindle control command) ; S ○ = 10.10.2 Multiple-spindle Control I (spindle selection command) ; G43.1,G44.1,G47.1 13.3.1 Longitudinal Rough Cutting Cycle ; G71 13.3.7 Compound Thread Cutting Cycle ; G76 13.4 Compound Type Fixed Cycle for Turning Machining (MITSUBISHI CNC special format) ; G71,G73,G74,G76 13.5 Fixed Cycle for Drilling 13.5.2 Face Tapping Cycle (Longitudinal tapping cycle) / Face Reverse Tapping Cycle (Longitudinal reverse tapping cycle) ; G84 (G88) / G84.1 (G88.1) 13.6 Fixed Cycle for Drilling (MITSUBISHI CNC special format) 13.6.5 Tapping Cycle ; G84 13.6.6 Synchronous Tapping Cycle ; G84.2 (Continue to the next page)</p>

Date of revision	Manual No.	Revision details
		<p>(Continued from the previous page)</p> <p>13.9.2.4 G Code Macro Call</p> <p>13.9.2.5 Miscellaneous Command Macro Call (for M, S, T, B Code Macro Call)</p> <p>13.9.5.6 NC Alarm (#3000)</p> <p>13.9.5.14 Position Information (#5001 - #5140 + n)</p> <p>13.9.5.18 Reading The Parameters (#100000-#100002, #100010)</p> <p>13.9.5.19 Reading PLC data (#100100-#100103,#100110)</p> <p>13.9.5.21 R Device Access Variables (#50000 - #50749, #51000 - #51749, #52000 - #52749)</p> <p>13.9.8 External Output Commands ; POPEN,PCLOS,DPRNT</p> <p>13.15 Programmable Parameter Input ; G10 L70/L100, G11</p> <p>13.20.2 Waiting-and-simultaneous Operation with Start Point Designated (Type 1) ; G115</p> <p>13.20.3 Waiting-and-simultaneous Operation with Start Point Designated (Type 2) ; G116</p> <p>14.7 Reference Position (Zero point) Return ; G28,G29</p> <p>14.10 Workpiece Coordinate System Setting and Offset ; G54 to G59 (G54.1)</p> <p>14.12 Workpiece Coordinate System Preset ; G92.1</p> <p>16.4 Multi-step Skip Function 2 ; G31 P</p> <p>- Changed the title of Appendix 1 to Appendix 3.</p> <p>- The other No. of Appendixes are changed due to changing the No. of Appendix 1.</p> <p>- The mistakes were corrected.</p>
Mar. 2014	IB(NA)1500924-G	<p>- The following chapters are added or changed corresponding to support the system software version K1.</p> <p>[Added]</p> <p>Appendix 3. Supplementary explanation for incomplete thread area of thread control</p> <p>[Changed the chapter titles]</p> <p>7.7 Automatic Acceleration/Deceleration</p> <p>13.22 Control Axis Superimposition I ; G126</p> <p>13.23 Control Axis Synchronization Across Part System I ; G125</p> <p>[Changed]</p> <p>3.1 Program Format</p> <p>7.2 Cutting Feedrate</p> <p>7.4 Feed Per Minute/Feed Per Revolution (Asynchronous Feed/Synchronous Feed) ; G94,G95</p> <p>7.7 Automatic Acceleration/Deceleration</p> <p>7.8 Rapid Traverse Constant Inclination Acceleration/Deceleration</p> <p>12.5 Compensation Data Input by Program;G10 L2/L10/L11,G11</p> <p>12.6 Tool Life Management II ; G10 L3,G11</p> <p>13.1.2 Thread Cutting Cycle ; G78</p> <p>13.3.1 Longitudinal Rough Cutting Cycle ; G71</p> <p>13.3.2 Face Rough Cutting Cycle ; G72</p> <p>13.5.6 Precautions When Using a Fixed Cycle for Drilling</p> <p>13.7.1 Subprogram Call ; M98,M99</p> <p>13.7.2 Subprogram Call ; M198</p> <p>13.9.2.1 Simple Macro Calls ; G65</p> <p>13.9.2.2 Modal Call A (Movement Command Call) ; G66</p> <p>13.9.2.3 Modal Call B (for each block) ; G66.1</p> <p>13.9.5.13 Other Modals(#4101-#4120,#4301-#4320)</p> <p>13.15 Parameter Input by Program ; G10 L70/L100,G11</p> <p>13.16 Macro Interruption ; M96,M97</p> <p>13.18 Coordinate Rotation by Program ; G68.1/G69.1</p> <p>13.21 Mixed Control (Cross Axis Control) I ; G110</p> <p>13.22 Control Axis Superimposition I ; G126</p> <p>13.24 2-part System Simultaneous Thread Cutting Cycle</p> <p>14.10 Workpiece Coordinate System Setting and Offset ; G54 to G59 (G54.1)</p> <p>16.6 Programmable Current Limitation ; G10 L14</p> <p>- Corrected the mistakes.</p>

Global Service Network

AMERICA

MITSUBISHI ELECTRIC AUTOMATION INC. (AMERICA FA CENTER)

Central Region Service Center
500 CORPORATE WOODS PARKWAY, VERNON HILLS, ILLINOIS 60061, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Michigan Service Satellite
ALLEGAN, MICHIGAN 49010, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Ohio Service Satellite
LIMA, OHIO 45901, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650
CINCINNATI, OHIO 45201, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Minnesota Service Satellite
ROGERS, MINNESOTA 55374, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

West Region Service Center
16900 VALLEY VIEW AVE., LAMIRADA, CALIFORNIA 90638, U.S.A.
TEL: +1-714-699-2625 / FAX: +1-847-478-2650

Northern CA Satellite
SARATOGA, CALIFORNIA 95070, U.S.A.
TEL: +1-714-699-2625 / FAX: +1-847-478-2650

Pennsylvania Service Satellite
PITTSBURG, PENNSYLVANIA 15644, U.S.A.
TEL: +1-732-560-4500 / FAX: +1-732-560-4531

Connecticut Service Satellite
TORRINGTON, CONNECTICUT 06790, U.S.A.
TEL: +1-732-560-4500 / FAX: +1-732-560-4531

South Region Service Center
1845 SATTELITE BOULEVARD STE. 450, DULUTH, GEORGIA 30097, U.S.A.
TEL: +1-678-258-4529 / FAX: +1-678-258-4519

Texas Service Satellites
GRAPEVINE, TEXAS 76051, U.S.A.
TEL: +1-678-258-4529 / FAX: +1-678-258-4519
HOUSTON, TEXAS 77001, U.S.A.
TEL: +1-678-258-4529 / FAX: +1-678-258-4519

Tennessee Service Satellite
Nashville, Tennessee, 37201, U.S.A.
TEL: +1-678-258-4529 / FAX: +1-678-258-4519

Florida Service Satellite
WEST MELBOURNE, FLORIDA 32904, U.S.A.
TEL: +1-678-258-4529 / FAX: +1-678-258-4519

Canada Region Service Center
4299 14TH AVENUE MARKHAM, ONTARIO L3R 0J2, CANADA
TEL: +1-905-475-7728 / FAX: +1-905-475-7935

Canada Service Satellite
EDMONTON, ALBERTA T5A 0A1, CANADA
TEL: +1-905-475-7728 FAX: +1-905-475-7935

Mexico Region Service Center
MARIANO ESCOBEDO 69 TLALNEPANTLA, 54030 EDO. DE MEXICO
TEL: +52-55-3067-7500 / FAX: +52-55-9171-7649

Monterrey Service Satellite
MONTERREY, N.L., 64720, MEXICO
TEL: +52-81-8365-4171

BRAZIL

MELCO CNC do Brasil Comércio e Serviços S.A

Brazil Region Service Center
ACESSO JOSE SARTORELLI, KM 2.1 CEP 18550-000, BOITUVA-SP, BRAZIL
TEL: +55-15-3363-9900 / FAX: +55-15-3363-9911

EUROPE

MITSUBISHI ELECTRIC EUROPE B.V.
GOTHAER STRASSE 10, 40880 RATINGEN, GERMANY
TEL: +49-2102-486-0 / FAX: +49-2102-486-5910

Germany Service Center
KURZE STRASSE, 40, 70794 FILDERSTADT-BONLANDEN, GERMANY
TEL: +49-711-770598-123 / FAX: +49-711-770598-141

France Service Center DEPARTEMENT CONTROLE NUMERIQUE
25, BOULEVARD DES BOUVETS, 92741 NANTERRE CEDEX FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

France (Lyon) Service Satellite DEPARTEMENT CONTROLE NUMERIQUE
120, ALLEE JACQUES MONOD 69800 SAINT PRIEST FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

Italy Service Center
VIALE COLLEONI, 7 - CENTRO DIREZIONALE COLLEONI PALAZZO SIRIO INGRESSO 1
20864 AGRATE BRIANZA (MB), ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

Italy (Padova) Service Satellite
VIA G. SAVELLI, 24 - 35129 PADOVA, ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

U.K. Branch
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +49-2102-486-0 / FAX: +49-2102-486-5910

Spain Service Center
CTRA. DE RUBI, 76-80-APDO. 420
08173 SAINT CUGAT DEL VALLES, BARCELONA SPAIN
TEL: +34-935-65-2236 / FAX: +34-935-65-1579

Poland Service Center
UL. KRAKOWSKA 50, 32-083 BALICE, POLAND
TEL: +48-12-630-4700 / FAX: +48-12-630-4701

Mitsubishi Electric Turkey A.Ş Ümraniye Şubesi
Turkey Service Center
ŞERİFALİ MAH. NUTUK SOK. NO.5 34775
ÜMRANIYE, İSTANBUL, TURKEY
TEL: +90-216-526-3990 / FAX: +90-216-526-3995

Czech Republic Service Center
KAFKOVA 1853/3, 702 00 OSTRAVA 2, CZECH REPUBLIC
TEL: +420-59-5691-185 / FAX: +420-59-5691-199

Russia Service Center
213, B.NOVOODMITROVSKAYA STR., 14/2, 127015 MOSCOW, RUSSIA
TEL: +7-495-748-0191 / FAX: +7-495-748-0192

MITSUBISHI ELECTRIC EUROPE B.V. (SCANDINAVIA)
Sweden Service Center
HAMMARBACKEN 14 191 49 SOLLENTUNA, SWEDEN
TEL: +46-8-6251000 / FAX: +46-8-966877

Bulgaria Service Center
4 A.LYAPCHEV BOUL., POB 21, BG-1756 SOFIA, BULGARIA
TEL: +359-2-8176009 / FAX: +359-2-9744061

Ukraine (Kharkov) Service Center
APTEKARSKIY LANE 9-A, OFFICE 3, 61001 KHARKOV, UKRAINE
TEL: +380-57-732-7774 / FAX: +380-57-731-8721

Ukraine (Kiev) Service Center
4-B, M. RASKOVOYI STR., 02660 KIEV, UKRAINE
TEL: +380-44-494-3355 / FAX: +380-44-494-3366

Belarus Service Center
OFFICE 9, NEZAVISIMOSTI PR.177, 220125 MINSK, BELARUS
TEL: +375-17-393-1177 / FAX: +375-17-393-0081

South Africa Service Center
5 ALBATROSS STREET, RHODESFIELD, KEMPTON PARK 1619, GAUTENG, SOUTH AFRICA
TEL: +27-11-394-8512 / FAX: +27-11-394-8513

ASEAN**MITSUBISHI ELECTRIC ASIA PTE. LTD. (ASEAN FA CENTER)**

Singapore Service Center
307 ALEXANDRA ROAD #05-01/02 MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 / FAX: +65-6476-7439

Malaysia (KL) Service Center
60, JALAN USJ 10/1B 47620 UEP SUBANG JAYA SELANGOR DARUL EHSAN, MALAYSIA
TEL: +60-3-5631-7605 / FAX: +60-3-5631-7636

Malaysia (Johor Baru) Service Center
17 & 17A, JALAN IMPIAN EMAS 5/5, TAMAN IMPIAN EMAS, 81300 SKUDAI, JOHOR MALAYSIA.
TEL: +60-7-557-8218 / FAX: +60-7-557-3404

Philippines Service Center
UNIT NO.411, ALABANG CORPORATE CENTER KM 25, WEST SERVICE ROAD
SOUTH SUPERHIGHWAY, ALABANG MUNTINLUPA METRO MANILA, PHILIPPINES 1771
TEL: +63-2-807-2416 / FAX: +63-2-807-2417

VIETNAM**MITSUBISHI ELECTRIC VIETNAM CO.,LTD**

Vietnam (Ho Chi Minh) Service Center
UNIT 01-04, 10TH FLOOR, VINCOM CENTER 72 LE THANH TON STREET, DISTRICT 1,
HO CHI MINH CITY, VIETNAM
TEL: +84-8-3910 5945 / FAX: +84-8-3910 5946

Vietnam (Hanoi) Service Satellite
SUITE 9-05, 9TH FLOOR, HANOI CENTRAL OFFICE BUILDING, 44B LY THUONG KIET STREET,
HOAN KIEM DISTRICT, HANOI CITY, VIETNAM
TEL: +84-4-3937-8075 / FAX: +84-4-3937-8076

INDONESIA**PT. MITSUBISHI ELECTRIC INDONESIA**

Indonesia Service Center
GEDUNG JAYA 11TH FLOOR, JL. MH. THAMRIN NO.12, JAKARTA PUSAT 10340, INDONESIA
TEL: +62-21-3192-6461 / FAX: +62-21-3192-3942

THAILAND**MITSUBISHI ELECTRIC FACTORY AUTOMATION (THAILAND) CO.,LTD**

Thailand Service Center
12TH FLOOR, SV.CITY BUILDING, OFFICE TOWER 1, NO. 896/19 AND 20 RAMA 3 ROAD,
KWAENG BANGPONGPANG, KHET YANNAWA, BANGKOK 10120,THAILAND
TEL: +66-2-682-6522-31 / FAX: +66-2-682-6020

INDIA**MITSUBISHI ELECTRIC INDIA PVT. LTD.**

India Service Center
2nd FLOOR, TOWER A & B, DLF CYBER GREENS, DLF CYBER CITY,
DLF PHASE-III, GURGAON 122 002, HARYANA, INDIA
TEL: +91-124-4630 300 / FAX: +91-124-4630 399

Ludhiana satellite office
Jamshedpur satellite office

India (Pune) Service Center
EMERALD HOUSE, EL-3, J-BLOCK, MIDC BHOSARI, PUNE – 411 026, MAHARASHTRA, INDIA
TEL: +91-20-2710 2000 / FAX: +91-20-2710 2100

Baroda satellite office

Mumbai satellite office

India (Bangalore) Service Center
PRESTIGE EMERALD, 6TH FLOOR, MUNICIPAL NO. 2,
LAVELLE ROAD, BANGALORE - 560 043, KAMATAKA, INDIA
TEL: +91-80-4020-1600 / FAX: +91-80-4020-1699

Chennai satellite office
Coimbatore satellite office

OCEANIA**MITSUBISHI ELECTRIC AUSTRALIA LTD.**

Australia Service Center
348 VICTORIA ROAD, RYDALMERE, N.S.W. 2116 AUSTRALIA
TEL: +61-2-9684-7269 / FAX: +61-2-9684-7245

CHINA**MITSUBISHI ELECTRIC AUTOMATION (CHINA) LTD. (CHINA FA CENTER)**

China (Shanghai) Service Center
1-3.5-10,18-23/F, NO.1386 HONG QIAO ROAD, CHANG NING QU,
SHANGHAI 200336, CHINA
TEL: +86-21-2322-3030 / FAX: +86-21-2308-3000

China (Ningbo) Service Dealer
China (Wuxi) Service Dealer
China (Jinan) Service Dealer
China (Hangzhou) Service Dealer
China (Wuhan) Service Satellite

China (Beijing) Service Center
9/F, OFFICE TOWER 1, HENDERSON CENTER, 18 JIANGUOMENNEI DAJIE,
DONGCHENG DISTRICT, BEIJING 100005, CHINA
TEL: +86-10-6518-8830 / FAX: +86-10-6518-8030

China (Beijing) Service Dealer

China (Tianjin) Service Center
UNIT 2003, TIANJIN CITY TOWER, NO 35 YOUYI ROAD, HEXI DISTRICT,
TIANJIN 300061, CHINA
TEL: +86-22-2813-1015 / FAX: +86-22-2813-1017

China (Shenyang) Service Satellite
China (Changchun) Service Satellite

China (Chengdu) Service Center
ROOM 407-408, OFFICE TOWER AT SHANGRI-LA CENTER, NO. 9 BINJIANG DONG ROAD,
JINJIANG DISTRICT, CHENGDU, SICHUAN 610021, CHINA
TEL: +86-28-8446-8030 / FAX: +86-28-8446-8630

China (Shenzhen) Service Center
ROOM 2512-2516, 25/F., GREAT CHINA INTERNATIONAL EXCHANGE SQUARE, JINTIAN RD.S.,
FUTIAN DISTRICT, SHENZHEN 518034, CHINA
TEL: +86-755-2399-8272 / FAX: +86-755-8218-4776

China (Xiamen) Service Dealer
China (Dongguan) Service Dealer

KOREA**MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD. (KOREA FA CENTER)**

Korea Service Center
1480-6, GAYANG-DONG, GANGSEO-GU, SEOUL 157-200, KOREA
TEL: +82-2-3660-9602 / FAX: +82-2-3664-8668

Korea Taegu Service Satellite
4F KT BUILDING, 1630 SANGYEOK-DONG, BUK-KU, DAEGU 702-835, KOREA
TEL: +82-53-382-7400 / FAX: +82-53-382-7411

TAIWAN**MITSUBISHI ELECTRIC TAIWAN CO., LTD. (TAIWAN FA CENTER)**

Taiwan (Taichung) Service Center (Central Area)
NO.8-1, INDUSTRIAL 16TH RD., TAICHUNG INDUSTRIAL PARK, SITUN DIST.,
TAICHUNG CITY 40768, TAIWAN R.O.C.
TEL: +886-4-2359-0688 / FAX: +886-4-2359-0689

Taiwan (Taipei) Service Center (North Area)
10F, NO.88, SEC.6, CHUNG-SHAN N. RD., SHI LIN DIST., TAIPEI CITY 11155, TAIWAN R.O.C.
TEL: +886-2-2833-5430 / FAX: +886-2-2833-5433

Taiwan (Tainan) Service Center (South Area)
11F-1., NO.30, ZHONGZHENG S. ROAD, YONGKANG DISTRICT, TAINAN CITY 71067, TAIWAN, R.O.C
TEL: +886-6-252-5030 / FAX: +886-6-252-5031

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 2008-2014 MITSUBISHI ELECTRIC CORPORATION
ALL RIGHTS RESERVED

MITSUBISHI CNC



MODEL	M700V/M70V Series
MODEL CODE	100-212
Manual No.	IB-1500924