SEIKI - SEICOS
∑10M/16M/18M
INSTRUCTION MANUAL

6 PROGRAMMING

Edition 1.01 NP-0000-1-0221-E-1-01

Hitachi Seiki Deutschland
Werkzeugmaschinen GmbH

CONTENTS

1. G CODE 1-1
1-1 List of G Code Group(SEICOSΣ10M/16M/18M) 1-1
1-2 List of G Codes (SEICOS Σ10M/16M/18M) 1-2
2. INTERPOLATION FUNCTION ..... 2 - 1
2-1 Positioning (G00) ..... 2 - 1
2-2 Linear Interpolation (G01) ..... 2 - 2
2-3 Circular Interpolation (G02, G03) ..... 2 - 3
2-3-1 Radius Designation on Arc ..... 2 - 5
2-4 Helical Interpolation (G02, G03) ..... 2 - 7
2-5 Virtual Axis Interpolation (G07) ..... 2 - 10
2-5-1 SIN Interpolation (G02, G03, G07) ..... 2 - 12
2-6 Single Direction positioning (G60) ..... 2 - 12
2-7 Involute Interpolation (G222, G223) ..... 2 - 15
2-8 Cylindrical Interpolation (G271) ..... 2 - 19
2-8-1 Command Format ..... 2 - 19
2-8-2 Feed Rate ..... 2 - 19
2-8-3 Cylindrical Interpolation Applied Axes ..... 2 - 20
2-8-4 Cylindrical Interpolation Applied Plane ..... 2 - 20
2-8-5 Sample Program ..... 2 - 20
2-8-6 Cautions ..... 2 - 21
2-8-7 Associated Parameters ..... 2 - 22
2-8-8 Associated Alarms ..... 2 - 22
2-9 Polar Coordinate Interpolation (G120, G121) ..... 2 - 23
2-9-1 G Codes ..... 2 - 23
2-9-2 Command Format ..... 2 - 23
2-9-3 Polar Coordinate Interpolation Axes ..... 2 - 23
2-9-4 Polar Coordinate Interpolation plane ..... 2 - 24
2-9-5 Program Command ..... 2 - 24
2-9-6 sample Program (X-axis: Linear Axis, C-axis: Rotary Axis) ..... 2 - 25
2-9-7 Feed Rate Clamp ..... 2 - 26
2-9-8 Rapid Traverse(G00) Operation ..... 2 - 26
2-9-9 Precautions ..... 2 - 26
2-9-10 Associated parameters ..... 2 - 27
2-9-11 Associated Alarms ..... 2 - 28
3. FEED FUNCTION 3-1
3-1 Feed per Minute (G94) 3-1
3-2 Feed per Rotary (G95) 3-2
3-3 Inverse Time (G93) 3-3
3-4 Exact Stop (G09) 3-4

3-5 Exact Stop Mode (G61) 3-4
3-6 Automatic Corner Override (G62) 3-5
3-6-1 Automatic Override in Inner Corner Area 3-6
3-6-2 Inner Arc Cutting Speed Change 3-7
3-7 Tapping Mode (G63) 3-9
3-8 Cutting Mode (G64) 3-10
3-9 Automatic Acceleration/Deceleration 3-11
3-10 Dwell (G04) 3-11
4. REFERENCE POINT 4-1
4-1 Automatic Reference Point Return (G28) 4-1
4-2 Reference Point Return Check (G27) 4-2
4-3 Return from Reference Point (G29) 4-4
4-4 2nd-4th Reference Point Return (G30) 4-5
4-5 Reset of Floating Reference point (G301) 4-7
5. COORDINATE SYSTEM 5-1
5-1 Machine Coordinate System Selection (G53) 5-1
5-2 Work Coordinate system selection (G54 - G59) 5-2
5-3 Addition of Work Coordinate system Pairs (G540 - G599) 5-3
5-4 Local Coordinate System Setting (G52) 5-5
5-5 Work Coordinate System Change (G92) 5-7
5-6 Work Coordinate System Preset (G921) 5-9
5-7 Work Coordinate System Shift (External Work Zero Point Offset Amount) 5-11
5-8 Plane Selection (G17, G18, G19) 5-12
5-9 Rotary Table Dynamic Fixture Offset 5-14
6. COORDINATE 6-1
6-1 Absolute/Incremental Programming (G90,G91) 6-1
6-2 Polar Coordinate Input (G15,G16) 6-2
6-3 Inch/Metric Input (G20, G21) 6-5
7. SPINDLE FUNCTION (S FUNCTION) 7-1
8. TOOL FUNCTION (T FUNCTION) 8-1
9. Miscellaneous Function (M FUNCTION) 9-1
9-1 Miscellaneous Function (M Function) 9-1
9-2 2nd Miscellaneous Function (B Function) 9-2
10. Canned Cycle 10-1
10-1 Canned Cycle (G73, G74, G76, G80 - G89) 10-1
10-2 Direct Tap (G741, G841) 10-16
10-3 Drilling Pattern Cycle (G70, G71, G72, G77) 10-22
10-4 True Circular Cutting (G302 ~ G305) 10-25
10-5 Square Outside Cutting (G322,323) 10-34
10-6 Plane Cutting Cycle (G324, G325, G326) 10-38
10-7 Poketing (G327 ~ G333) 10-50
10-7-1 Circular Poketing (G327) 10-55

10-7-2 Square Poketing (G328) 10-59
10-7-3 Track Inside (G329) 10-64
10-7-4 Circle outside Pocketing (G330) 10-67
10-7-5 Square Outside Cutting (G331) 10-70
10-7-6 Track Outside (G332) 10-73
10-7-7 Circle (G333) 10-76
10-7-8 Special Fixed Cycles (G322 ~ G333) Type 2 10-78
10-8 ATC Canned Cycle (MO6) 10-79
10-8-1 ATC Canned Cycle, Type A (VK, VKC, VG, VkII) 10-82
10-8-2 ATC CANNED CYCLE TYPE E (VM40III) 10-84
10-8-3 ATC CANNED CYCLE TYPE F (HG) 10-86
10-8-4 ATC Canned Cycle, Type G (HK) 10-87
10-8-5 ATC Canned Cycle, Type I (Initial HS500) 10-89
10-8-6 ATC Canned Cycle, Type J (VS) 10-91
10-8-7 ATC Canned Cycle, Type K(HS630) 10-93
10-8-8 ATC Canned Cycle, Type L (New HS500) 10-95
10-8-9 ATC Canned Cycle, Type M (VS 16-tool) 10-96
10-8-10 ATC Canned Cycle, Type N (MS400H) 10-98
10-9 High-Speed Machining Cycle 10-99
10-9-1 Trochoid Cycle (G334) 10-99
10-9-2 Helical Drilling Cycle (G812, G813) 10-103
10-9-3 High Speed Side Face Cutting Cycle (G335) 10-107
10-9-4 Z Feed Fluting Cycle (G336) 10-111
10-9-5 Corner Pocket Cycle (G337) 10-113
10-9-6 Square Pocket Cycle (G338) 10-116
11. COMPENSATION FUNCTION 11-1
11-1 Tool Length Compensation (G43, G44, G49) 11-1
11-2 Tool Offset (G45 - G48) 11-6
11-3 Tool Diameter Compensation (G38 - G42) 11-9
11-3-1 Detailed Description of Tool Diameter Compensation 11-15
11-4 3-D Tool Offset (G40 - G41) 11-27
11-5 H and D Functions 11-31
11-6 Tool Offset by Tool Number 11-33
12. CONVERTING FUNCTION 12-1
12-1 Programmable Mirror Image (G501, G511) 12-1
12-2 Setting Mirror Image 12-4
12-3 Scaling (G50, G51) 12-7
12-4 Coordinate Ratation (G68, G69) 12-10
12-5 Optional Angle Chamfering/Corner R (, C, R) 12-15
13. MEASURMENT 13-1
13-1 Skip Function (G31) 13-1

13-2 Automatic Measurement of Tool Length (G37) 13-3
13-3 Safety Guard (Tool Length) 13-5
13-4 Safety Guard (Comparison) 13-9
14. DATA SETTING 14-1
14-1 Data Setting (G10) 14-1
14-1-1 Tool offset amount setting 14-1
14-1-2 Work coordinate system offset amount setting 14-1
14-2 Programmable Parameter Input (G10) 14-3
14-3 Plotting Parameter Setting 14-5
15. SOFT OT 15-1
15-1 Soft OT (Stored Stroke Limit 1) 15-1
15-2 Stored Stroke Limits 2 and 3 (G22 and G23) 15-3
15-3 Soft-OT before Move 15-6
16. AXIS CONTROL 16-1
16-1 Rotary Axis Controlling Function 16-1
16-2 Oscillation Function (G113, G114) 16-4
16-3 Normal Direction Control (G411, G421, G401) 16-8
17. HIGH-SPEED MACHINING 17-1
17-1 Multibuffer (G251) 17-1
17-2 Feed Rate Clamp by Circular Arc Radius 17-3
17-3 PRECONTROLLING 17-4
17-3-1 PRE-INTERPOLATION LINEAR ACCELERATION/DECELERATION 17-5
17-3-2 AUTO CORNER DECELERATION 17-6
17-4 HIGH PRECISION PROFILE CONTROL 17-8
17-5 Smooth Interpolation 17-10
17-6 NURBS Interpolation 17-12
17-7 SHG Machining 17-14
18. FIVE-FACE MACHINING 18-1
18-1 Selecting the Machining Plane (G240-G245) 18-1
18-2 Common Work Origin Offset 18-2
18-3 Axis Changeover 18-5
18-3-1 Axis Changeover (Type A) 18-5
18-3-2 Axis Changeover (Type B) 18-6
19. AUTOMATIC OPERATION 19-1
19-1 Program Restart 19-1
19-2 Block Restart 19-6
19-3 Machining Break Point Return (G206) 19-11
19-4 Reverse Movement 19-14
19-5 Sequence Number Comparison and Stop 19-16
19-6 Reset (Reset Associated with Automatic Operation) 19-18
19-6-1 Details of Reset (Reset Associated with Automatic Operation) 19-18

20. MANUAL OPERATION 21-1
20-1 Manual Absolute ON/OFF 21-1
21. TEST RUN 21-1
21-1 Miscellaneous Function Lock 21-1
21-2 Miscellaneous Function Finish 21-1
22. CUSTOM MACROS 22-1
22-1 Outline 22-1
22-2 Call Commands and Return Command 22-2
22-2-1 Types of Command 22-2
22-2-2 Multi-call 22-8
22-2-3 Argument Designation 22-13
22-3 Variables 22-18
22-4 Representation of Variables 22-31
22-5 Citation of Variables 22-31
22-6 Undefined Variables 22-32
22-7 Expression and Computation 22-33
22-8 Substitution Command 22-36
22-9 Branch Command 22-37
22-10 Repeat Command 22-38
22-11 Naming Command 22-39
22-12 IF Command 22-40
22-13 External Output Commands 22-42
23. Interrupt Type Custom Macro 23-1
23-1 Custom Macro Interrupt Operation 23-1
23-2 How to Specify 23-2
23-3 Interrupt Type Custom Macro Proper 23-2
23-4 Status Trigger Method and Edge Trigger Method (Parameters) 23-2
23-5 Reversion and Modal Information 23-3
23-6 System Variable in Interrupt Program 23-3
23-7 Custom Macro Interrupt and Custom Macro Modal Call 23-4
23-8 Interrupt Timing and Return Position in Each Mode 23-4
23-9 Associated Parameters 23-7
24. MEMORY OPERATION IN OTHER COMPANIES' FORMATS 24-1
24-1 Memory Operation in FS15 Format 24-1
24-2 Memory Operation in i80M Format 24-2

1. G CODE

1-1 List of G Code Group(SEICOSΣ10M/16M/18M)

Group	Function	Remarks
00	Non-modal	
01	Positioning/liner interpolation/circular interpolation	
02	Plane designation	
03	Absolute programming/incremental programming	
04	Stored stroke check	
05	Inverse time/feed per minute/feed per revolution	
06	Inch/metric conversion	
07	Tool diameter compensation	
08	Tool length compensation	
09	Canned cycle	
10	Initial point return/R point return	
11	Scaling	
12	Work coordinate system Selection	
13	Cutting mode/exact stop mode/automatic corner override mode	
14	Macro modal call	
15	Programming mirror image	
16	Coordinator rotation	
17	Constant surface speed control	*1
18	Tool life management	
19	Normal direction control	*1
20	Polar coordinate command	
21	Oscillation function	
22	Polar coordinate interpolation	*1
23	Spindle speed fluctuation detection	*1
24	Machining plane selection	*1
25	Tool nose interference check	*1
26	Axis switching/3-D coordinate conversion	*1
27		*2
28		*2
29		*2
30		*2
31		*2

(Note) *1 Reserved G code and not available for the moment

*2 Spare G code group for function improvement.

1-2 List of G Codes (SEICOS Σ10M/16M/18M)

Code	Group	Function	Remarks	
G00	01	Positioning		
G01		Linear interpolation		
G02		Circular interpolation/helical interpolation CW		
G03		Circular interpolation/helical interpolation CCW		
G04	00	Dwell		
G05		High-precision profile control		
G07		Virtual axis interpolation		
G08		Antecedent control		
G09		Exact stop		
G10		Data setting		
G11		Data setting node cancel		
G15	20	Polar coordinate command cancel		
G16		Polar coordinate command		
G17	02	$X_P Y_P$ plane	where: X_P : x axis or its parallel axis	
G18		$Z_P X_P$ plane	Y_P : Y axis or its parallel axis	
G19		$Y_P Z_P$ plane	Z_P : Z axis or its parallel axis	
G20	06	Inch input		
G21		Metric input		
G22	04	Store stroke check ON		
G23		Stored stroke check OFF		
G25	23	Spindle speed fluctuation detection OFF		*1
G26		Spindle speed fluctuation detection ON		*1
G27	00	Reference point return check		
G28		Reference point return		
G29		Return from reference point		
G30		2nd, 3rd, 4th reference point return		
G31		Skip function		
G33	01	Thread cutting		*1
G34		Variable lead thread cutting		*1
G37	00	Tool length automatic measurement		
G38		Tool diameter compensation vector hold		
G39		Tool diameter compensation corner arc		
G40	07	Tool diameter compensation left/3-D tool offset cancel		
G41		Tool diameter left/3-D tool offset		
G42		Tool diameter compensation right		

Code	Group	Function	Remarks
G43	08	Tool length compensation +	
G44		Tool length compensation -	
G45	00	Tool offset extension	
G46		Tool offset contraction	
G47		Tool offset double extension	
G48		Tool offset double contraction	
G49	08	Tool length compensation cancel	
G50	11	Scaling cancel	
G51		Scaling	
G52	00	Local coordinate system Setting	
G53		Machine coordinate system selection	
C54	12	Work coordinate system 1 selection	
G55		Work coordinate system 2 selection	
G56		Work coordinate system 3 selection	
G57		Work coordinate system 4 selection	
G58		Work coordinate system 5 selection	
G59		Work coordinate system 6 selection	
G60	00	Single direction positioning	Group 01 by parameter change
G61	13	Exact stop mode	
G62		Automatic corner override mode	
G63		Tapping mode	
G64		Cutting mode	
G65	00	Macro call	
G66	14	Macro modal call	
G67		Macro modal call cancel	
G68	16	Coordinate rotation	
G69		Coordinate rotation cancel	
G70	00	Bolt hole cycle	
G71		Arc	
G72		Arc	
G73	09	Peck drilling cycle	
G74		Counter tapping cycle	
G76		Fine boring cycle	
G77	00	Grid cycle	

Code	Group	Function	Remarks
G80	09	Canned cycle cancel	
G81		Drilling cycle, spot boring	
G82		Drilling cycle, counter boring	
G83		Peck drilling cycle	
G84		Tapping cycle	
C85		Boring cycle	
G86		Boring cycle	
G87		Back boring cycle	
G88		Boring cycle	
G89		Boring cycle	
G90	03	Absolute programming	
G91		Incremental programming	
G92	00	Work coordinate system change/maximun spindle speed setting	
G93	05	Inverse tine feed	
G94		Feed per minute	
G95		Feed per revolution	
G96	17	Constant surface speed control	*1
G97		Constant surface speed control	*1
G98	10	Canned cycle initial level point return	
G99		Canned cycle R point level return	
G113	21	Oscillation node ON	
G114		Oscillation node OFF	
G120	22	polar coordinate interpolation mode cancel	
G121		Polar coordinate interpolation mode	
G130	18	Tool life management OFF	
G131		Tool life management ON	
G201	00	PMC data setting	*1
G203		High-speed machining program registration start	*1
G204		High-speed machining pro-gram registration end	*1
G206		Tool retract amount setting	
G212	01	Circular thread cutting CW	*1
G213		Circular thread cutting CCW	*1
G216		Spline interpolation	*1

Code	Group	Function	Remarks
G222	01	Involute interpolation CW	
G223		Involute interpolation CCW	
G232		Exponential function interpolation CW	*1
G233		Exponential function interpolation CCW	*1
G240	24	Machining plane 0 selection (Machining plane selection cancel)	
G241		Machining plane 1 selection	
G242		Machining plane 2 selection	
G243		Machining plane 3 selection	
G244		Machining plane 4 selection	
G245		Machining plane 5 selection (Available for an optional horizontal/vertical angle)	
G248	26	Axis switching/3-D coordinate conversion ON	
G249		Axis switching/3-D coordinate conversion canel	
G251	00	Multi-buffer	
G264	25	Tool nose interference check ON	*1
G265		Tool nose interference check OFF	*1
G271	00	Cylindrical interpolation	
G301		Floating reference point return	
G302		Circular cutting inner diameter CW	
G303		Circular cutting inner diameter CCW	
G304		Circular cutting outer diameter CW	
G305		Circular cutting outer diameter CCW	
G311		Multi-step skip function 1	*1
G312		Multi-step skip function 2	*1
G313		Multi-step skip function 3	*1
G314		Multi-step skip function 4	*1
G322		Square outside cutting CW	
G323		Square outside cutting CCW	
G324		Square plane	
G325		Square plane 1-directional	
G326		Square plane 2-directional	
G327		Circle inside (pocketing)	
G328		Square inside (pocketing)	
G329		Track inside (pocketing)	
G330		circle outside (pocketing)	

Code	Group	Function	Remarks
G331	00	Square outside (pocketing)	
G332		Track outside (pocketing)	
G333		Circle (pocketing)	
G334		Trochoid cycle	
G335		High-speed side cutting cycle	
G336		Z feed fluting cycle	
G337		Corner pocket cycle	
G338		Square pocket cycle	
◤G401	19	Normal direction control cancel node	
G411		Normal direction control left side ON	
G421		Normal direction control right side ON	
C431	08	Tool axis direction tool length compensation	*1
◤G501	15	Programmable mirror image Cancel	
G511		Programmable mirror image	
G540~ G599	12	Additional work coordinate system selection (60 pairs)	
G611	00	Pre-interpolation acceleration/deceleration	*1
C653	00	Position check (Note) for Maintenance	*1
G661	14	Macro modal call B	*1
G721	00	Rotary copy	*1
G722		Parallel copy	*1
G741	09	Counter direct tap cycle	
G841		Direct tap cycle	
G812		Helical drilling cycle CW	
G813		Helical drilling cycle CCW	
G921	00	Work coordinate system preset	

(Note 1) *1 Reserved G code and not available for the noment

(Note 2) The G code marked ◤ in each group is selected in the reset state.

2. INTERPOLATION FUNCTION

2-1 Positioning (G00)

Each axis moves to a Program-specified position at an independent rapid traverse rate to perform positioning.

(1) command format

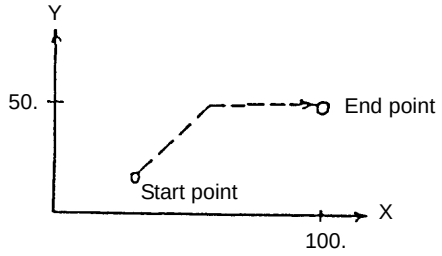
$\begin{bmatrix} G90 \\ G91 \end{bmatrix}$ G00 X_ Y_ Z_ ;

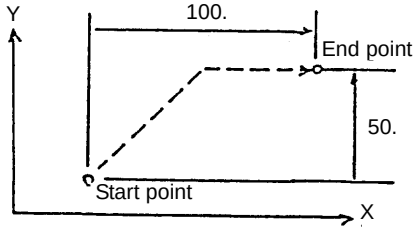
(2) Sample program

(a) Absolute programming
G90 G00 X100. Y50.

:

(b) Incremental programming
G91 G00 X100. Y50.





(3) Cautions

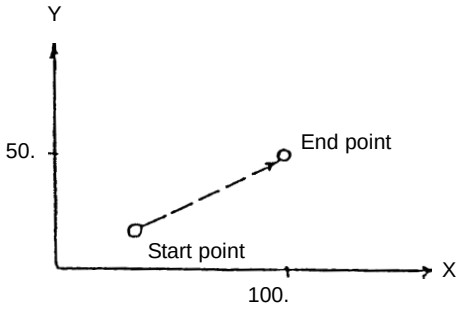
(a) The rapid traverse rate has been set independently for each axis.

(b) The tool path is non-linear. See to it that the tool does not interfere with the workpiece.

(c) Linear acceleration/deceleration is applied. Confirm imposition (an accumulated amount due to servo delay is within tolerance) at the end of the block, and then, proceed to the next block.

(d) G00, G90 are G91 ale modal G codes. Once they are specified, they remain effective until the next associated G code is specified.

(e) The tool path can be made linear by altering the parameter.



(f) You can set with the parameter whether the reset state is to be the G00 or G01 mode.

(4) Associated parameters

No.1401, # 6 = 0 Dry run made invalid for rapid traverse command.
1 Dry run made valid for rapid traverse command.
No.1401, # 1= 0 Non-linear interpolation as positioning interpolation system
1 Linear interpolation as positioning interpolation system
No.3402, # 0= 0 G00 mode in reset state
1 G01 mode in reset state

2-2 Linear Interpolation (G01)

The tool moves linearly to a program-specified position at the cutting feed rate specified with an F code.

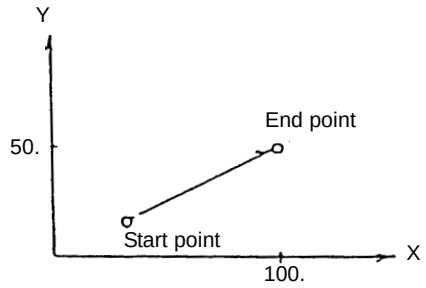
(1) Command format

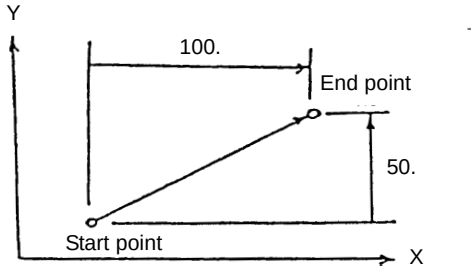
$\left\{ \begin{matrix} G90 \\ G91 \end{matrix} \right\}$ G01 X_ Y_ Z_...F_ ;

(2) Sample program

(a) Absolute programming.
G90 G01 X100. Y50. F200;

(b) Incremental programming
G90 G01 X100. Y50. F200;





(3) Cutting feed rate

The cutting feed rate specified with an F code is the speed at which the tool moves linearly. In this case, the cutting feed rate is a composite speed of all the specified axes; the cutting feed rate of each axis is as follows.

G01 G91 Xa Yb Zc Ff:

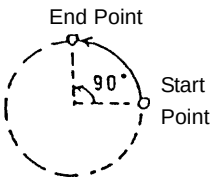
X-axis cutting feed rate: $Fx = \frac{af}{L}$, where; $L = \sqrt{a^2 + b^2 + c^2}$

Y-axis cutting feed rate: $Fy = \frac{bf}{L}$

z-axis cutting feed rate, $Fz = \frac{cf}{L}$

When the rotary axis is specified in the identical block, linear interpolation is performed taking it as a linear axis in the units of degree.

G01 G91 X100. C90. F200 :



Cutting feed rate in the rotary axis (C axis) direction:

$$Fc = \frac{90. \times 200}{L} \quad \text{(deg/min)}$$

where; $L = \sqrt{100.^2 + 90^2}$ (mm)

(4) Cautions

(a) An alarm results when no F code has been specified in the G01 block or before.

(b) Exponential type acceleration/deceleration is applied.

(c) Set with the parameter whether the reset state is to be the G00 or G01 mode.

(5) Associated parameters

No.3402, #0 = 0 G00 mode in reset state

1 G01 mode in reset state

(6) Associated alarm

No. 102 F0 was specified in the G01/G02/G03 mode.

2-3 Circular Interpolation (G02, G03)

The tool moves to a program-specified position along an arc within the plane selected with a plane selection G code (G17, G18, G19) at the cutting feed rate specified with an F code.

(1) Command format

(a) $X_P\ Y_P$ plane

$$G17\begin{bmatrix}G02\\G03\end{bmatrix}\ X_P_\ Y_P_\begin{bmatrix}R_\\I_\ \ J_\end{bmatrix}\quad F_\ ;$$

(b) $Z_P\ -X_P$ plane

$$G18\begin{bmatrix}G02\\G03\end{bmatrix}\ Z_P_\ Y_P_\begin{bmatrix}R_\\K_\ \ I_\end{bmatrix}\quad F_\ ;$$

(c) $X_P\ -Z_P$ plane

$$G19\begin{bmatrix}G02\\G03\end{bmatrix}\ Y_P_Z_P_\begin{bmatrix}R_\\J_\ \ K_\end{bmatrix}\quad F_\ ;$$

Where; X_P : X axis or its parallel axis

Y_P : Y axis or its parallel axis

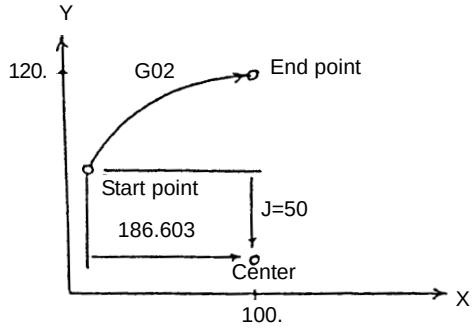
Z_P : Z axis or its parallel axis

(Note) To specify with R instead of I, J or K is called radius designation on arc.

(2) Sample program

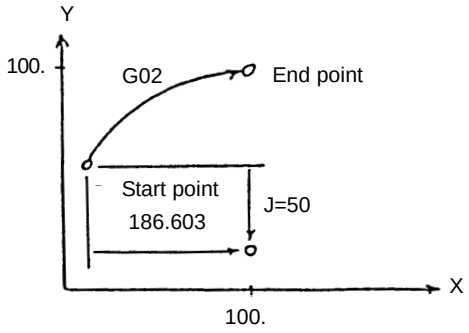
(a) Absolute programming

G17 G90 G00 X13.397 Y70. F200:
G02 X100. Y120. I86.603 J-50.:



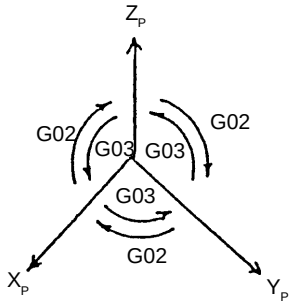
(b) Incremental programming

G17 C91 G02 X86.603 Y50
I86.603 J-50. F200:



(3) Arc rotating direction

G02 : Clockwise (CW)
G03 : Counterclockwise direction (CCW)



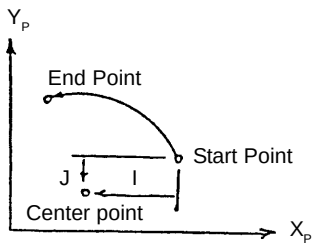
(4) Arc plane

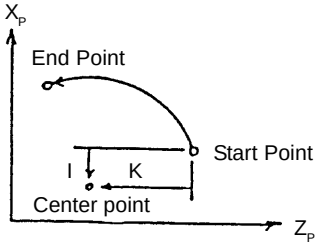
The arc plane is specified with C17, G18 or G19.

G17 : X_P -Y_P plane
G18 : Z_P -X_P plane
G19 : Y_P -Z_P plane

(5) Arc center

The arc center is specified with I, J or K corresponding to X_P ,Y_P and Z_P respectively. In this case, I, J and K are the vector components when viewing the arc center from its start point.





(6) Cutting feed rate

The cutting feed rate specified with an F code is the speed at which the tool moves on the arc.

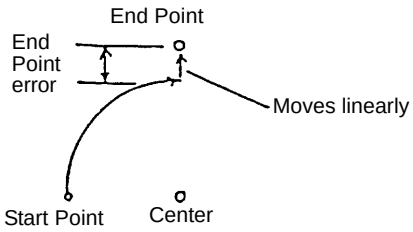
(7) Cautions

(a) An alarm results when no F code has been specified in the G02/G03 block or before.

(b) An alarm results if an arc radius = 0 is specified.

(c) I0, J0 and K0 are omissible.

(d) When there is no end point on the arc, the tool moves linearly the rest after moving along an arc if the end point error of circular interpolation is within the parameter set value. Also, an alarm results if it is other than the parameter set value.



(e) An alarm results if the axis not for the arc plane is specified.

(f) When R is specified in the same block as I, J and K, R is given priority.

(g) When the rotary axis is specified in the same block, circular interpolation is performed taking it as a linear axis in the units of degree.

(h) Exponential type acceleration/deceleration is applied.

(8) Associated parameters

No.3459 Arc end tolerance

(9) Associated alarm

No.102	F0 has been command with G01, G02 and G03.
No.132	Arc interpolation error
(#001)	Center command, although available, is 0 in value.
(#002)	Difference in radius value between of start and end
(#003)	pints has exceeded parameter set value.

2-3-1 Radius Designation on Arc

In case of circular interpolation, an arc radius can be directly specified with R instead of specifying the arc center with I, J or K.

(1) Command format.

(a) X_P-Y_P plane

$$G17 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} X_P_\ Y_P_\ R \pm_\ F_ :$$

(b) Z_P-X_P plane

$$G18 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} Z_P_\ Y_P_\ R \pm_\ F_ :$$

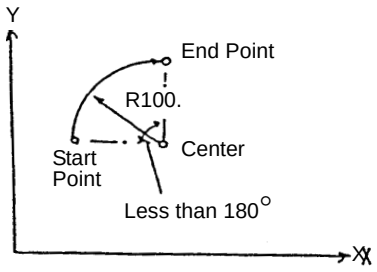
(c) Y_P-Z_P plane

$$G19 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} Y_P\text{-}Z_P_\ R \pm_\ F_ :$$

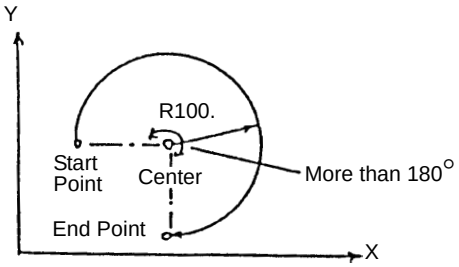
where; X_P: X axis or its parallel axis
Y_P : Y axis or its parallel axis
Z_P : Z axis or its parallel axis
R+ : Arc of less than 180°
R- : Arc of over 180°

(2) Sample program

(a) For the arc of less than 180°
G17 G91 G02 X100. Y100.
R-100. F200:



(b) For the arc of over 180°
G17 G91 G02 X100. Y-100.
R-100. F200 :



(3) Cautions

(a) When I, J, K and R are specified in the same blocks, R is given priority.

(b) When the arc center is not calculated, an alarm results.

(c) When G91 $\left[\begin{matrix} G02 \\ G03 \end{matrix} \right] R_ :$ is specified, it is taken as a block without axial move.

(4) Associated parameters

(5) Associated alarms

No. 131 An arc radius R with which arc center position cannot be calculated has been commanded.

2-4 Helical Interpolation (G02, G03)

If an arc command and any one axis for other than arc are specified, helical interpolation is enabled by control which performs linear interpolation synchronously with arc movement.

(1) Command format

$$G17 \quad \begin{bmatrix} G02 \\ G03 \end{bmatrix} X_P_ Y_P_ \begin{bmatrix} R_ \\ I_ \ J_ \end{bmatrix} \alpha_ F_ ;$$

$$G18 \quad \begin{bmatrix} G02 \\ G03 \end{bmatrix} Z_P_ Y_P_ \begin{bmatrix} R_ \\ I_ \ J_ \end{bmatrix} \alpha_ F_ ;$$

$$G19 \quad \begin{bmatrix} G02 \\ G03 \end{bmatrix} Y_P_ Z_P_ \begin{bmatrix} R_ \\ I_ \ J_ \end{bmatrix} \alpha_ F_ ;$$

where; X_P : X axis or its parallel axis

Y_P : Y axis or its parallel axis

Z_P : Z axis or its parallel axis

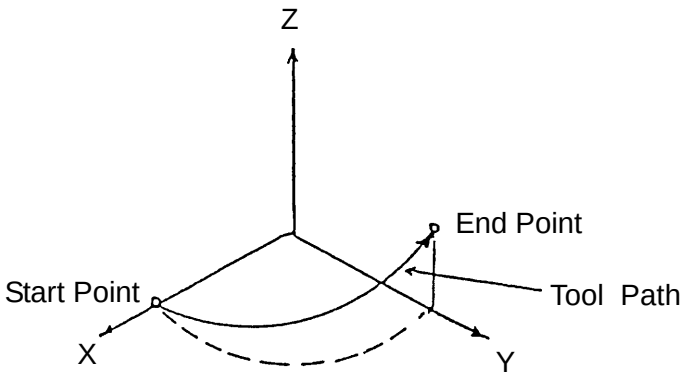
α : Any optional linear axis for other than circulate interpolation (up to 2 axes)

F : Arc speed

$$\text{Linear axis speed} = F \times \frac{\text{Linear arc length}}{\text{Arc length}}$$

(2) Sample program

G17 G91 G03 X-100. Y-100. R100. Z50. F200 :

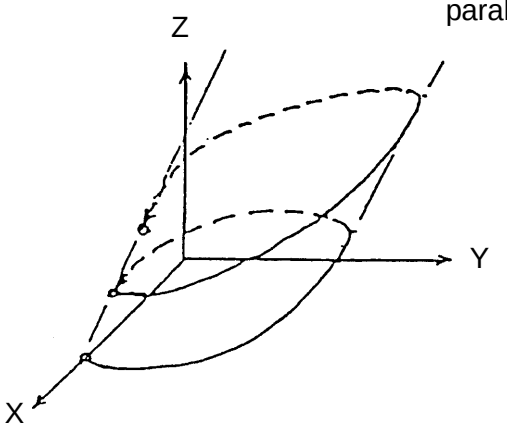


(3) The axes for other than circular interpolation can be specified up to 2 axes in the same block.

(Example) G17 G91 G03 I-100. Z100. V50. F200 :

I-100. Z100.V50.;

The V axis must be parallel with the Y axis.



(4) Cautions

(a) See to it that the linear axis speed does not exceed the maximum value.

(b) Tool diameter compensation is applied to circular interpolation.

(c) An alarm results if 3 or more linear axes are specified.

(5) Associated parameters

(6) Associated alarms

2-5 Virtual Axis Interpolation (G07)

If the axis is specified as a virtual axis, it does not move.
Interpolation can be perfomed with this axis and other one.

(1) Command format

G07 α 0 ; Sets the α axis as the virtual axis.
: } The α axis is the virtual axis
: } in this section.
G07 α 1 ; Cancels the a axis as the virtual axis.

where ; α : Any one axis

(2) Sample program

```
G07  X0 ;                                  Sets the X axis as the virtual axis.
G17  G91  G02  X0  Y0  I50.  Z-1. 0  F200 ; SIN interpolation
G01  X5. F100;                             Dwell state for the move time of the X axis
G07  X1 ;                                  Cancels the X axis as the virtual aixs.
```

(3) Cautions

(a) SIN interpolation results if one arc axis for helical interpolation is set as the virtual axis.
(b) The virtual axis is any one axis.
An alarm results if 2 or more axes are set as virtual axes.
(c) If a command is given as to only one axis specified as the virtual axis, the axis move time valve is placed in the dwell state.
(d) Program virtual axis interpolation in the incremental manner.
Since the virtual axis does not move at any time, it is necessary to be careful when programming in the absolute manner.

(4) Associated parameters

(5) Associated alarms

No.139 Two or more virtual axes have been specified.

2-5-1 SIN Interpolation (G02, G03, G07)

SIN interpolation can be performed by assuming one of axes for an arc command as a virtual axis in helical interpolation.

(1) Command format

G87 α0 ; Sets the virtual axis.

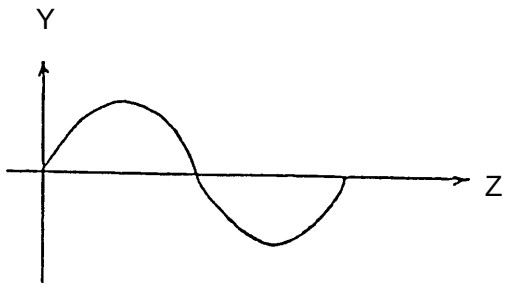
specify helical interpolation.

G07 α1 ; Cancels the virtual axis.

α is any one axis for the arc command.

(2) Sample program

```
G07  X0;
G17  G91  G03  X0  Y0  I-50.  Z100.  F200 ;
G07  X1;
```



(3) Cautions

(a) Effective only for automatic operation of the virtual axis.

(b) Program the virtual axis in the incremental manner.

(c) An alarm results if 2 or more virtual axes are specified.

2-6 Single Direction positioning (G60)

Performs final positioning always from a specified single direction.
Using this function allows you to perform high-accuracy positioning.

(1) Command format

(a) For the one-shot G code

G60 X_Y_Z_... ; Effective only in the G60 specified block

(b) For the modal G code

```
G60  X_ Y_ Z_... ;
X_ Y_ Z_...;
      :
      :
G00 ;
```

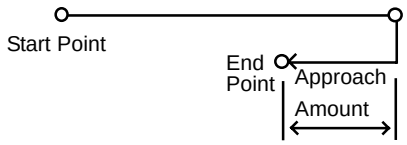
} Single direction positioning

Cancels G60 if any G code in Group 01 other than G60 is given.

(2) Sample program

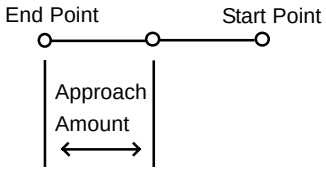
(a) When moving in the + direction

G60 G91 X100.;



(b) When moving in the - direction

G60 G91 X-100.;



(3) Final positioning direction

Approach amount > 0 : The positioning direction is the + direction.
Approach amount < 0 : The positioning direction is the - direction.
Approach amount = 0 : Positioning is not performed in the - direction.

(4) Cautions

(a) Whether G610 is to be one-shot or modal is set with the parameter.
(b) In the canned cycle, hole positioning is performed with G60.
However, single positioning is disabled for the shift amount of G76 and G87.
(c) During the mirror image, it is disabled for the approach amount of single direction positioning.

(5) Associated parameters

(5) Associated parameters

No. 3458		Single positioning direction and approach amount of each axis
No. 3400, #2 =	0	G60 is the G code of Group 00 (one-shot).
	1	G60 is the G code of Group 01 (modal).

2-7 Involute Interpolation (G222, G223)

This function allows machining along an involute curve. It also provides cutter compensation.

(1) Involute curve

The involute curve in the X-Y plane is defined as following.

$X(\theta) = R [\cos\theta + (\theta - \theta 0) \sin\theta] + X0$

$Y(\theta) = R [\sin\theta + (\theta - \theta 0) \cos\theta] + Y0$

where;

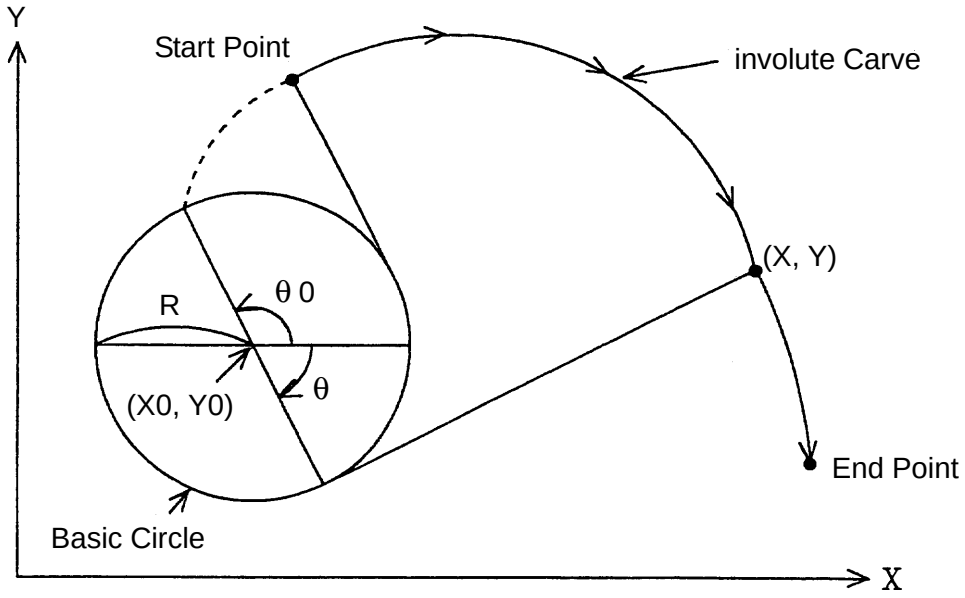
X0, Y0:Central coordinate of the basic circle

R : Radius of the basic circle

θ 0 : Angle of the point where the involute curve starts

θ : Angle of the contact of the tangent from the current position to the basic circle

X(θ), Y(θ) :Current position of the X-and Y-axis



The involute curves in the Z-X and Y-Z planes are defined in the same manner as that in the X-Y plane.

(2) Command format

(a) X_P-Y_P_ plane

G17 $\begin{bmatrix} G222 \\ G223 \end{bmatrix}$ X_P_Y_P_ I_ J_ R_ F_ ;

(b) Z_P-X_P Plane

G18 $\begin{bmatrix} G222 \\ G223 \end{bmatrix}$ Z_P_ Y_P_ K_ J_ R_ F_;

(c) Y_P-Z_P plane

G19 $\begin{bmatrix} G222 \\ G223 \end{bmatrix}$ Y_P_ Z_P J_ K_ R_ F_ ;

where;

G222	: Clockwise involute interpolation
G223	: Counterclockwise involute interpolation
X_P, Y_P, Z_P	: Coordinate value of the end point
X_P	: X-axis or its parallel axis
Y_P	: Y-axis or its parallel axis
Z_P	: Z-axis or its parallel axis
I, J, K	: Central position of the basic circle for the involute curve viewed from the start point
R	: Radius of the basic circle
F	: Cutting feed rate

(3) Start point and end point

The end point of the involute curve is specified with the address X, Y, or Z and expressed with an absolute or incremental value depending on G90 or G91. If the incremental value is used, specify the coordinate of the end point viewed from the start point of the involute curve.

When a start or end point command is within the basic circle, an alarm results. The same results when an offset vector is brought into the basic circle by cutter compensation.

Care should be taken when an offset is applied, in particular, to the inside of the involute curve.

(4) Basic circle command

The center of the basic circle is specified with I, J, and K, corresponding to X, Y, and Z, respectively. However, the numerals following I, J, and K are vector components, when the center of the basic center is viewed from the start point of the involute curve.

They should be always programmed with incremental values, regardless of G90 or G91.

Add a sign to I, J, and K as required.

(5) Feed rate

A feed rate for involute interpolation assumes a cutting feed rate specified with an F-code, and a speed along the involute curve (speed in the tangent direction of the involute curve) is controlled to be specified feed rate.

(6) Cutter compensation

Cutter compensation can be applied to the involute curve for perform machining. Cutter compensation is specified with G40, G41, or G42, like straight line or circular arc.

G40 : Cancels cutter compensation

G41 : Offsets to the left of the tool advance direction

G42 : Offsets to the right of the tool advance direction

Cutter compensation for the involute curve is achieved as follows.

First, obtain a circular arc close to the curvature of the involute curve near the start point of the involute curve. Similarly, obtain the an intersecting point at offset time near the end point, and assume the involute curve passing those two points to be a tool center path.

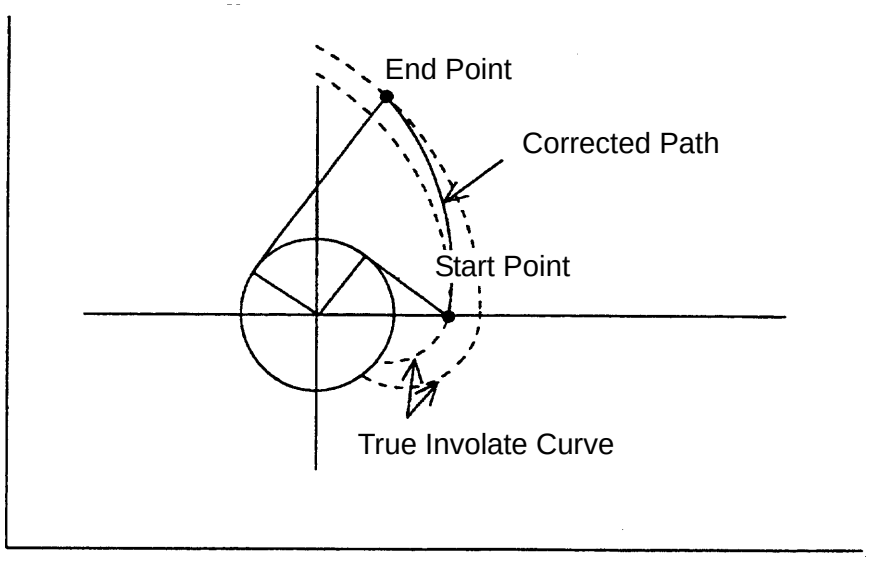
You cannot specify cutter compensation start-up or cancel during the involute interpolation mode.

(7) End point error

When the end point is not located on involute curve passing the start point, the below-mentioned curve is assumed.

An alarm will result if a shift amount between the involute curve passing the start point and that passing the end point is greater than a value set in a parameter.

When there is an end point error, a feed rate cannot be assured.



(8) Modes available for involute interpolation

Involute interpolation is allowed even during the following G-code modes.

G41 : Cutter compensation to the left
G42 : Cutter compensation to the right
G511 : Programmable mirror image
G68 : Coordinate rotation

(9) Limitations

(a) Rpm of the involute curve

Both start point and end point must be within 100 revolutions from an involute curve start point. The involute curve revolving one or more time can be also specified in one block.

(b) Unspecifiable function

Optional angle chamfering corner R cannot be specified during the involute interpolation mode. Helical interpolation and axis scaling are disallowed.

(c) Modes unavailable for involute interpolation

G411: Normal direction control to the left
G412: Normal direction control to the right
G121: Polar coordinate interpolation
G16 : Polar coordinate command

(10) Associated parameters

No. 3465 Initial angle error limit value for involute interpolation

(11) Associated alarms

No. F242 A command value has an error.
① The start or end point has been specified within the basic circle.
② I, J, K, or R has been specified with 0.
③ The start or end point is beyond 100 revolutions from the involute curve start point.

No. F243 The end point is not on the involute curve.

2-8 Cylindrical Interpolation (G271)

If the move amount of the linear axis and the angle of the rotary axis are given by a program command, the move amount of the rotary axis given in terms of angle will be internally converted into a distance on the circumference. Since the distance on the circumference can be considered the move amount of the linear axis on the circumference, linear or circular interpolation can be performed with other linear axis. After interpolation, it will be put back into the angle of the rotary axis again.

2-8-1 Command Format

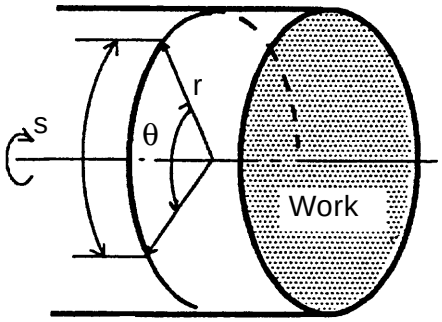
G271 A Cylinder radius ; Cylindrical interpolation ON
: Cylindrical interpolation mode
G271 A0 ; Cylindrical interpolation cancel

The A-axis is a rotary axis.

The rotating angle of the rotary axis is calculated back from the move amount on the circumference. For example; when you want to move by 100.0 on the circumference of the cylinder whose radius is 50.0, obtain the rotating angle of the rotary axis from the following formula:

r : Cylinder radius

θ : Rotating angle
s: Move amount on the cylinder circumference



$$\text{Rotating angle } \theta = \frac{360 \times S \text{ (Move amount on circumference)}}{2\pi \times r \text{ (Cylinder radius)}}$$

$$= \frac{360 \times 10.00}{2\pi \times 50.0} = 114.591$$

2-8-2 Feed Rate

During the cylindrical interpolation mode, a feed rate command F is the speed at which the tool moves on the outer circumference of the cylinder.

2-8-3 Cylindrical Interpolation Applied Axes

set in the parameters(no. 3426 for the linear axis, and no. 3427 for the rotary axis) the linear axis and rotary axis to which you want to apply cylindrical interpolation. A setting range for both parameters is 1 to the number of controlled axes; they must not have the same value.

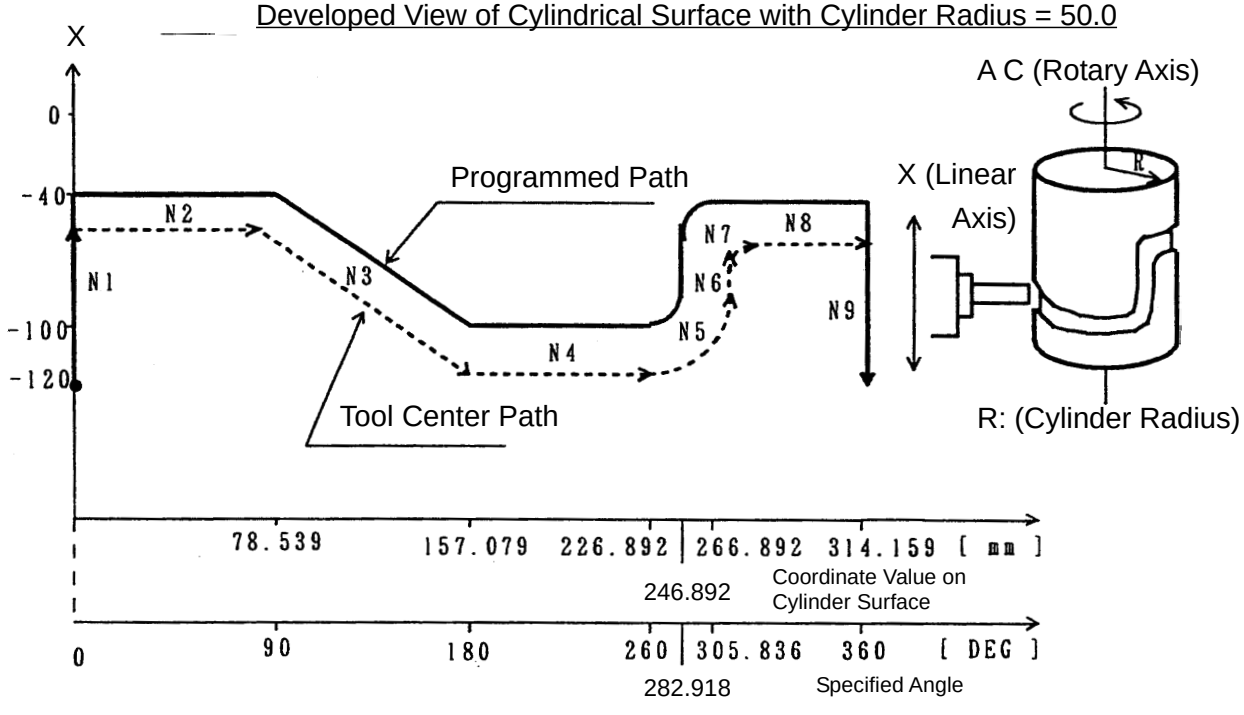
2-8-4 Cylindrical Interpolation Applied Plane

During the cylindrical interpolation mode, the plane is automatically selected, which assumes the rotary axis to be the plane first axis, and the linear axis td be the plane second axis.

2-8-5 Sample Program

```
(Select the A-Y plane with the parameters 3426 and 3427)
G40 G80 ,.
G00 X-1204.0 Y0 A0 ,.
G01 Z100.0 F100
G271 A50.0 ;                          Cylindrical interpolation mode ON
N1 G42 G01 X-40.0 F500 ;              (Cylinder radius = 50.0)
N2 A90.0 F500 ;
N3 X-100.0 A180.0
N4 A260.0 ;                           In Cylindrical interpolation mode
N5 G03 X-80.0 A282.918 R20.0 ;
N6 G01 X-60.0 ;
N7 G02 X-40.0 A305.836 R20.0 ;
N8 G01 A360.0 ;
N9 G40 G01X-120.0 ;
G271 AO ;                             Cylindrical interpolation mode OFF
G00 Z120.0 A0 ;
```

Developed View of Cylindrical Surface with Cylinder Radius = 50.0



2-8-6 Cautions

(1) When specifying cutter compensation, start up/cancel during the cylindrical interpolation mode.

(2) The plane (selected by G17 to G19) existing prior to entering the cylindrical interpolation mode is canceled once during the cylindrical interpolation mode and revived at the end of the cylindrical interpolation mode.

(3) When performing circular interpolation during the cylindrical interpolation mode, the method to specify the circular arc radius (which ones of I, J, and K addresses are to be used) is determined as follows, depending on the axis of the basic coordinate system to which the 2nd axis (linear axis) of the cylindrical interpolation plane corresponds (setting of the parameter no. 1011).

- When the linear axis is the X-axis or its parallel axis, specify with K and I, considering it the Z_p-X_p plane.
- When the linear axis is the Y-axis or its parallel axis, specify with I and J, considering it the X_p-Y_p plme.
- When the linear axis is the Z-axis or its parallel axis, specify with J and K, considering it the Y_p-Z_p plane.

It is also possible to specify a circular arc radius with an R-command.

(4) Specify a G271 command (G271 Axx;) independently in a block.
If there is no axis command (G271;) after the G271 command, an alarm will result.

(5) If the axis not set in the parameter no. 3427 is specified with the G271 command, an alarm will result.

(6) The follwing functions are not available:

- Return to the machining break point
- Manual intervention by manual absolute ON

(7) If the follwing comands are given during the cylindrical interpolation mode, an alarm will result:

- G17, G18, G19 ---------- Plane designation
- G28, G30, G53 ---------- Machine coordinate system
- G54-59, G540-G599, G92, G52 --- Work coordinate system
- G73, G74, G76, G81-G89 ---------- Canned cycle
- G302-G305, G322-G333 ---------- Special canned cycle
- G00 (when including the cylindrical interpolation applied rotary axis)

(8) In the cylindrical interpolation mode, the angle of the rotary axis is converted into the distance on the circumference and converted back into the angle after interpolation. When this is done, a slight conversion error results.

(9) If circular interpolation with small circular arc radius is executed during the cylindrical interpolation mode, a circular interpolation alarm may be caused by the above-mentioned conversion error. Care should be taken when using it. Even at the time of cutter compensation, a cutter compensation alarm may result due to the above-mentioned reason.

(10) If cylindrical interpolation mode ON (G271 Axx; (A≠ 0)) is specified again during the cylindrical interpolation mode an alarm will result.

(11) The residual move amount indicates the value when moving on the outer circumference of the cylinder.

2-8-7 Associated Parameters

No. 3426 Number of the linear axes(1 to number of controlled axes) to which cylindrical interpolation is to be applied

No. 3427 Number of the linear axes(1 to number of controlled axes) to which cylindrical interpolation is to be applied

2-8-8 Associated Alarms

No. 126 A cylindrical interpolation command has an error.

(#001) There is no rotary axis command.

(#002) This is not an independent command.

(#003) The modal G-code is not correct.

No. 127 A wrong command was given during the cylindrical interpolation mode.

(#001) A wrong G-code was given.

(#003) A rotary axis move command was given in the G00 mode.

(#004) Cylindrical interpolation was specified again during the cylindrical interpolation mode.

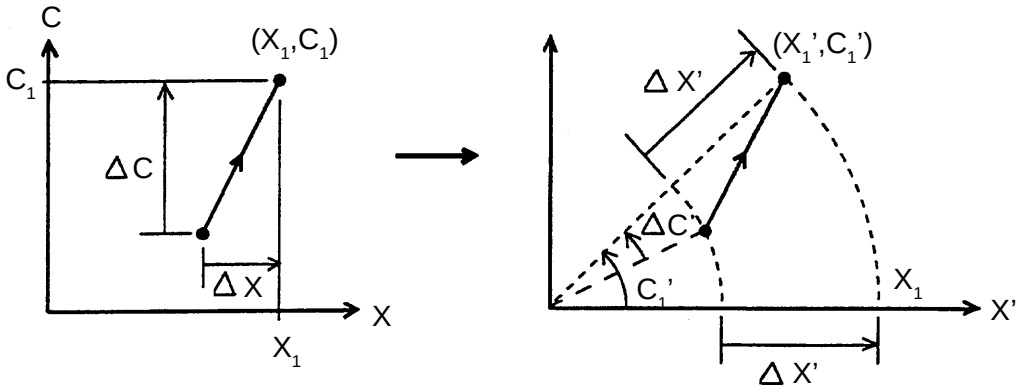
No. 189 (G271) The parameter has not been set properly.

- The parameter no. 3426 or no. 3427 is not "1" to the number of controlled axes, or they have the same value.
- The linear axis set in the parameter no. 3426 is none of the three basic axes, X, Y, and Z, or not parallelto awy.

2-9 Polar Coordinate Interpolation (G120, G121)

polar coordinate interpolation is a function to provide contour control by converting a command programmed in the orthogonal coordinate system into a linear axis move(tool move) and rotary axis move(work rotation).

<Orthogonal Coordinate system> <Polar Coordinate System>



The following shows the relations between (X_1 , C_1) and. (X_1 ' , C_1 ').

2-9-1 G Codes

G 121 : Polar coordinate interpolation mode

G 120 : Cancel polar coordinate interpolation mode

2-9-2 Command Format

G121 ; Polar coordinate interpolation mode ON

:

: Polar coordinate interpolation mode ON

:

G120 ; Cancels polar coordinate interpolation mode

2-9-3 Polar Coordinate Interpolation Axes

The linear axis and rotary axis to which you want to apply polar coordinate interpolation should be set in the parameters (no. 3418 for the linear axis and no. 3419 for the rotary axis) in advance. For both parameters, set within a range of 1 to the number of controlled axes, and they must not be of identical value.

2-9-4 Polar Coordinate Interpolation plane

A G121 command effectuates the polar coordinate interpolation mode, assumes the zero pint of the work coordinate system to be that of the coordinate system, and selects the plane(polar coordinate interpolation plane) which assumes the linear axis to the first axis of the plane and a virtual axis(rotary axis) orthogonal to the linear axis to be the second axis of the plane. Polar coordinate interpolation is performed on this plane.

Since polar coordinate interpolation is initiated, assuming the G121 specified position's angle to be zero (0), regardless of an actual position, prepare a program, assuming the work coordinate value of the rotary axis to be zero (0) when G121 is specified..

2-9-5 Program Command

(1) In the polar coordinate interpolation mode, specify a program command in terms of orthogonal coordinate value in the polar coordinate interpolation plane. Use the axial address of the rotary axis as a command address for the second axis of the plane. The unit for the command is not degrees; specify in the same unit (rpm or in.) as the linear axis, the first axis of the plane.

(2) The polar coordinate interpolation mode allows you to specify linear interpolation (G01) and circular interpolation (G02, G03). Both absolute programming and incremental programming are allowed.

(3) When performing circular interpolation in the polar coordinate interpolation plane, the circular arc radius specifying method (whether I, J, or K should be used) is determined as follow, depending on which axis in the basic coordinate system the first axis (linear axis) of the plane corresponds to(parameter setting: no. 1011).

- When the linear axis is either X-axis or its parallel axis, I and J are used, assuming the X_P -Y_P plane.
- When the linear axis is either Y-axis or its parallel axis, J and K are used, assuming the Y_P -Z_P plane.
- When the linear axis is either Z-axis or its parallel axis, K and I are used, assuming the Z_P -X_P plane.

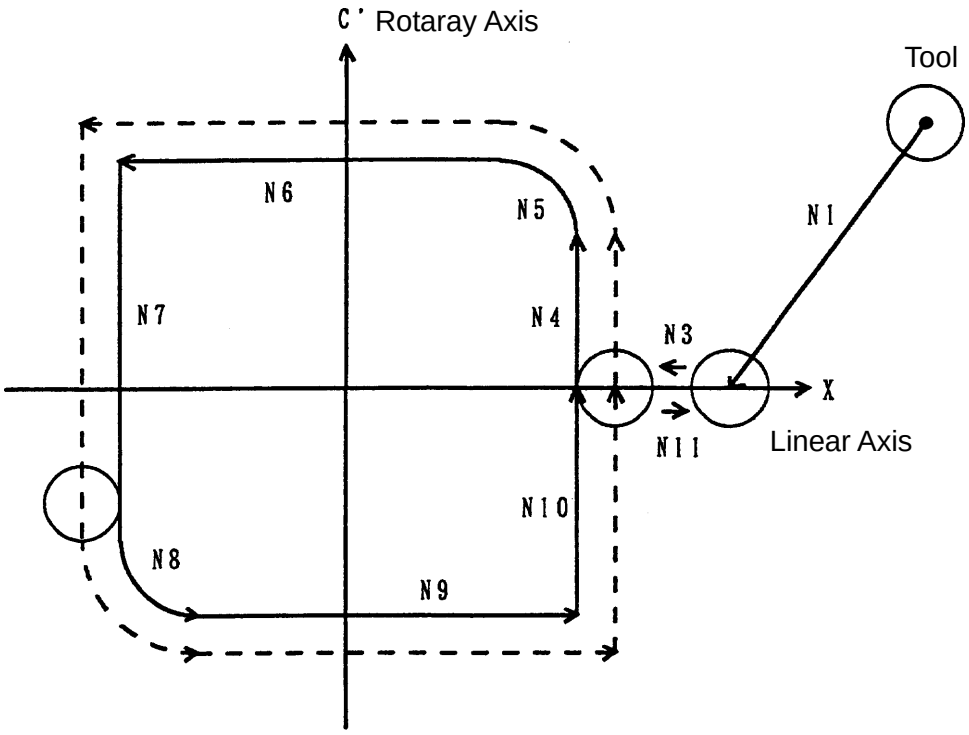
It is also possible to specify the circular arc radius with an R-command.

(4) Cutter compensation can be applied to the program command. Polar coordinate interpolation is performed on the profile after cutter compensation. However, specify G121 and G120 in the cutter compensation cancellation mode (G40).

(5) For an feed rate, use F-code to specify a tool move rate in the polar coordinate interpolation plane(orthogonal coordinate system). Normally, it is specified in feed per minute (G94); the unit for the F-code will be mm/min. or in./min.

2-9-6 sample Program (X-axis: Linear Axis, C-axis: Rotary Axis)

```
N1      G00  X50.0  C0 ;        Positions to the start position
N2      G121 ;                  Starts polar coordinate interpolation
N3      G42  G01  X30.0  F100 ;
N4      C20.0 ;
N5      G03  X20.0  C30.0  R10.0 ;   Profile program(program based on
N6      G01  X-30.0                  the orthogonal coordinate value
N7      C-20.0 ;                     In the X-C' plane)
N8      G03  X-20.0  C-30.0
            I0.0 J0;
N9      G01 X30.0;
N10       C0 ;
N11       G40 X50.0 ;           Cancels polar coordinate interpolation
N12       G120 ;
```



2-9-7 Feed Rate Clamp

The maximum cutting feed rate at polar coordinate interplation can be set in a parameter (no; 3464). If any feed rate higher than this one is specified during polar coordinate interpolation, it will be clamped to this rate. If a set value is 0, it will be clamped by the normal maximum cutting feed rate (parameter no. 1422) even during polar coordinate interpolation.

As polar coordinate interpolation converts the profile programmed in the orthogonal coordinate system into the movements of the linear axis and rotary axis, a move near the work center (coordinate zero pint) may increase the speed of the rotary axis and exceed the limit speed of the machine.

If this is the case, the allowable speed of the rotary axis can be set with the parameters as follows.

Rotary axis allowable speed = Maximum cutting feed rate (parameter no. 3464) x Allowable ratio(parameter no. 3420)

If the speed component of the rotary axis during polar coordinate interpolation exceeds the allowable speed, an override calculated by the formula below will be automatically applied. (Automatic override)

$$\text{Override} = \frac{\text{Allowable speed of the rotary axis}}{\text{Speed component of the rotary axis}} \times 100\ (\%)$$

If the speed component of the rotary axis stills exceeds the allowable speed even after a override has been applied, the feed rate will be clamped in such a manner that the speed component of the rotary axis will not exceed the maximum cutting feed rate. (Automatic speed clamp)

2-9-8 Rapid Traverse(G00) Operation

A G00 command can be executed during the polar coordinate interpolation mode. to the G00 block, however, only the end point is converted into a position in the polar coordinate system; a path remains unconverted.

(Example)"AG00 X100.0 C50.0;" in the G121 mode assumes the same path as "G01 X141.421 C45.0;" in the G120 mode.

With G00, the rotary axis takes a short-cut. (move within +/-l80°) to reduce the machining time.

2-9-9 Precautions

(1) Specify G120 and G121 in an independent block, respectively.

(2) Prior to specifying G121, the work coordinate system must have been set, where the center of the rotary axis can be the zero point of the coordinate system.

(3) The plane prior to G121 (the plane selected with G17, G18,- or- G19) is cancelled once by specifying G121 and restored by specifying G120.

(4) The following lists the G-codes which cam be specified during the G121 mode.
G00, G01, G02, G03, G04, G09, G40, G41, G42,
G65, G66, G67, G98, G99

(5) Any axis outside the plane in the G121 mode moves independent of polar coordinate interpolation.

(6) A display of "MOVE AMT." during the G121 mode refers to a remaining move amount in the polar coordinate plane (orthogonal coordinates). However, the G00 block assumes the same unit as usual.

(7) The following functions are not available for the block in the G121 mode. Program restart, Return to machining break point, Manual intervention by manual absolute ON

(8) Upon completion of polar coordinate interpolation (G120) , the work coordinate value of the rotary axis can be rounded to 360° through parameter setting. (0° ≦ 360°)

(9) When the work coordinate value of the linear Axis is negative at G121 command. To the first move block, only the end point is converted into a position in the polar coordinate system; a path remains unconverted. (Same as linear G00 in the G121 mode)
Starting from the next move block, polar coordinate conversion is applied to the move path. Only G00 and G01 can be specified in the first move block; G02 and G03 cannot be.

2-9-10 Associated parameters

No. 3410,	#1 = 0	Does not Perform automatic override and automatic speed clamp during the polar coordinate interpolation mode.
	= 1	Performs automatic override and automatic speed clamp during the polar coordinate interpolation mode.
No. 3418		Axis number of the linear axis to which you want to apply polar coordinate interpolation
No. 3419		Axis number of the rotary axis to which you want to apply polar coordinate interpolation
No. 3420		Automatic override allowable ratio during the polar coordinate interpolation mode
No. 3464		Maximum cutting feed rate during the polar coordinate interpolation mode.

2-9-11 Associated Alarms

No. 113 A polar coordinate interpolation command has an error.

(#001) G120 or G121 has not been independently specified.

(#002) When G120 or G121 was specified, cutter compensation had not been cancelled.

(#003) When the work coordinate value of the linear axis was negative at G121, a G-code other than G00 and G01 was specified in the first move block.

No. 114 A wrong Command was specified during the polar coordinate interpolation mode.

(#001) An unavailable G-code was specified.

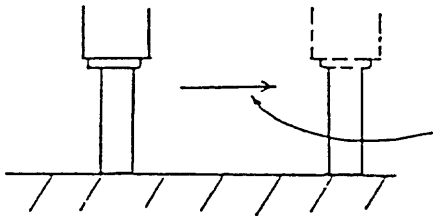
No. 189 (G121) Parameter Setting is erroneous.

- The setting of the parameters nos. 3418 and 3419 is not within a range of 1 to the number of controlled axes, or the setting of the parameter no. 3418 is equal to that of no. 3419.
- The linear axis set in the parameter no. 3418 is none of the three basic axes(X, Y, and Z) or not parallel to any of them.

3. FEED FUNCTION

3-1 Feed per Minute (G94)

Until G95 is specified after G94 was specified, the stroke per minute (mm/min., inch/min.) is directly specified with a numerical value following F.



Feed rate F is the stroke per minute
Metric system : mm/min
Inch system : inch/nin.

(1) Command format
G94 ;

(2) Sample program
F100 ; The feed rate is a move of 100 mm per minute.

(3) The F code programmable range is as follows.
(a) Metric system
F60 : 1 ~ 999999 (mm/min.)
F61 : 0.1~999999.9 (mm/min.)
(b) Inch system
F51 : 0.I ~ 99999.9 (inch/min.)
F52 : 0.01 ~ 99999.99(inch/min.)

(5) Cautions
(a) One of F60, F61, F51 and F52 is selected by parameter setting.
(b) then F61 in the metric system is specified without a decimal point, the input increment of 1 mm/min. or 0.1 m/min. is selected by parameter setting.
(c) In case of thread cutting (G33), feed per revolution is selected, disabling feed per minute.

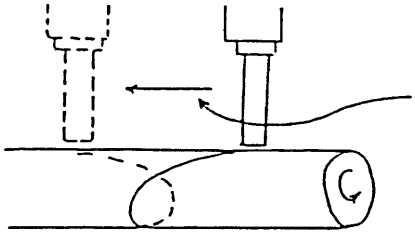
(6) Associated parameters
No. 3400, #1= 0 The least input increment for F61 in the metric system without a decimal point is 1 mm/min.
1 The least input increment for F61 of the metric system without a decimal point is 0.1 mm/min.

No. 3401, #2= 0 F51 for feed per minute in the inch system (inch/min.)
1 F52 for feed per minute in the inch system (inch/min.)
No. 3401, #3= 0 F60 for feed per minute in the metric system (mm/min.)
1 F61 for feed per minute in the metric system (mm/min.)

(7) Associated alarms

3-2 Feed per Rotary (G95)

Until G94 is specified after G95 was specified, the stroke per spindle revolution (mm/rev. inch/ rev.) is directly specified with a numerical value following F.



Feed rate F is the stroke per spindle revolution
Metric system : mm/rev.
Inch system : inch/rev.

(1) Command format
G95 ;

(2) Sample program
F1, 2 ; The feed rate is 1.2 mm per revolution.

(3) The F code programmable range is as follows.
(a) Metric system
F32 : 0.01 ~999.99 (mm/rev.)
F33 : 0.001 ~ 999.999(mm/rev.)
(b) Inch system
F23 : 0.001 ~ 99.999 (inch/rev.)
F24 : 0.0001 ~ 99.9999 (inch/rev.)

(4) Cautions
(a) One of F32, F33, F23 and F24 is selected by parameter setting.

(5) Associated parameters

No. 3401, #0=	0	F23 for feed per revolution in the inch system (inch/rev.)
	1	F24 for feed per revolution in the inch system (inch/rev.)
No. 3401, #1=	0	F32 for feed per revolution in the metric system (mm/rev.)
	1	F33 for feed per revolution in the metric system (mm/rev.)

3-3 Inverse Time (G93)

when G93 is instructed, the program is put in the INVERSE TIME mode until G94 and G95 are instructed. Inverse time (FRM) is instructed directly with a value following F.

(1) Command format:

G93 ;

(2) Inverse time (FRN)

(a) For linear interpolation (G01)

$$FRN = \frac{Speed}{Distance} \text{ (1/min)}$$

(b) For circular interpolation (G02, G03):

$$FRN = \frac{Speed}{Distance} \text{ (1/min)}$$

where; Speed : Metric system (mm/min)
Inch system (inch/min)
Distance : Metric system (mm/min)
Inch system (inch)

(3) F can be instructed within the following range:

F43 : 0.001 ~ 9999.999

(4) Sample program

```
G93 ;
G91 G01 X100. F5. ;
Y50. F10.
```

} Spped: 500mm/min

(5) Cautions

(a) In G93 mode, F code must be instructed per block.
When F code is omitted, a previously instructed F code becomes valid.

3-4 Exact Stop (G09)

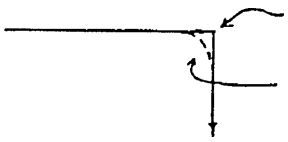
If a G09 command is specified in the same block as a move block, it decelerates and stops the machine upon completion of one block, and after confirming that the machine position is within the range in which a command position was specified, proceeds to the next block.

(1) Command format

G09 ------ ;

(2) Sample program

N1 G09 G91 G01 X100. F500 ;
N2 G01 Y-50. ;



The corner has an edge when G09 is specified.

The corner is rounded when G09 is not specified.

(3) Associated parameters

No.1826 Imposition width of each axis
No.1827 Imposition width of each axis on cutting feed

3-5 Exact Stop Mode (G61)

Until G62, G63 or G64 is specified after G61 was specified, this function decelerates and stops the machine, confirms that the machine position is within the specified range, and then, proceeds to the next block.

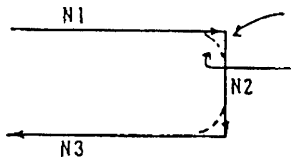
(1) Command format

G61 ;

(2) Sample program

```
N1 G61 G91 G01 X100. F500 ;
N2 Y-50. ;
N3 X - 100. ;
N4 G64;
```

} Exact stop effective blocks



The corner has an edge when the G61 mode is effective.
The corner is rounded when the G61 is ineffective.

(3) Cautions

3-6 Automatic Corner Override (G62)

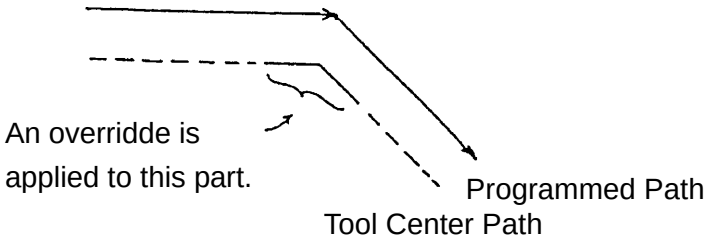
When tool diameter compensation is applied, since the tool center path is located inside the program-specified path in the inner corner and inner arc area and a cutting load increases, an override is applied automatically to the cutting feed rate to reduce the cutting load.
Once G62 is specified, it remains effective until G61, G63 or G64 is specified.

(1) Command format

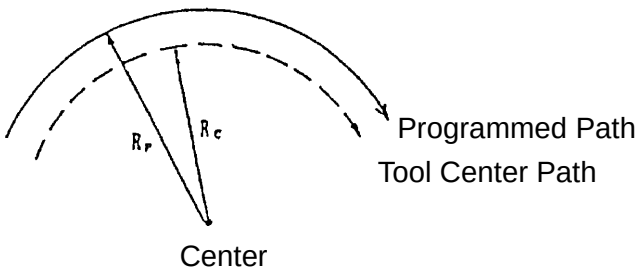
```
G62 ;
```

(2) Types of automatic corner override

(a) Automatic override in the inner corner area.



(b) Inner arc cutting speed change



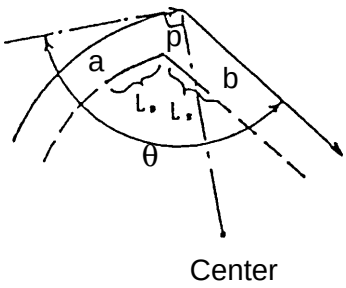
(3) Cautions

3-6-1 Automatic Override in Inner Corner Area

When the following conditions are met in the corner during the tool diameter condensation mode, an override is applied to cutting feed automatically.

[The conditions are as follows for the blocks having the corner between them.]

(a) When the G code of Group 01 is either G01, G02 or G03.
(b) When an offset amount is other than 0 in the tool diameter offset mode.
(c) When an offset is inside in the corner.
(d) When there is an axis in the offset plane.
(e) When the G41 or G42 is not contained.
(f) When an inner corner angle is smaller than a parameter set value. (θ)
(g) When a corner start distance and a corner end distance are other than 0 and internal. (Le, Ls)



Center

where; θ : Inner criterion angle
Le : Start distance
Ls : End distance

An override is applied to cutting feed from the point a through point b.

(1) An override amount for cutting feed is set with the parameter.

$1 \leqq$ (Automantic corner override amount) $\leqq 100$ (%)

An actual cutting feed rate is as follows.

F × (Automatic corner override amount) × (Feed rate override)

(2) Sample program

(D10 = 10.)

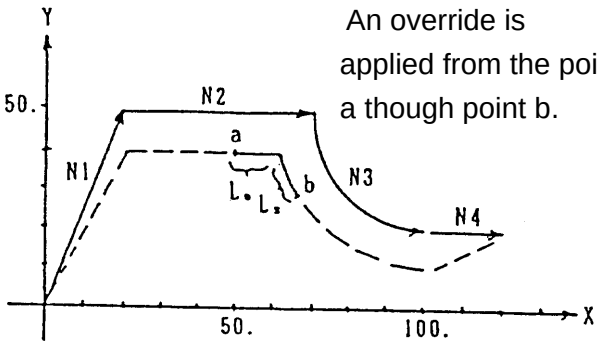
N1 G62 G42 G91 G00 X20. Y50. D10;

N2 G01 X50. F200;

N3 G03 X30. Y-30. R30.;

N4 G64 G40 G00 X20.;

(3) Cautions



An override is applied from the point a though point b.

(4) Associated parameters

No. 1711	Inner criterion angle of automatic corner override
No. 1712	Override amount of Automatic corner override.
No. 1713	Start distance of automatic corner override
No. 1714	End distance of automatic corner override

(5) Associated alarm

3-6-2 Inner Arc Cutting Speed Change

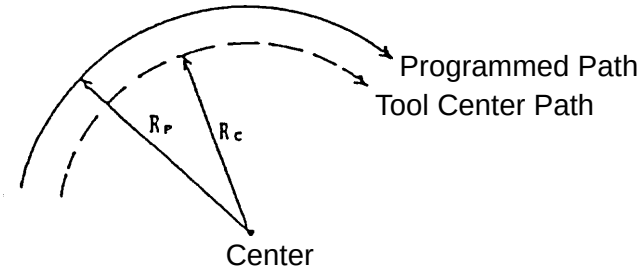
When arc cutting is offset inside during the tool diameter compensation node, an override of the ratio of radius of the tool diameter center path to that of the programmed path is applied to the specified cutting feed rate.

The cutting speed in the tool diameter center path is as follows.

$$Fx \frac{Rc}{Rp}$$

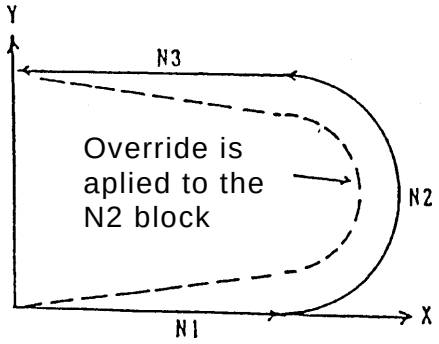
where; Rc : Radius of the tool center path

Rp : Radius of the programmed path



(1) Sample program

N1 G62 G41 G91 G00 X50. D10;
N2 G03 Y50. J25. F200;
N3 C64 G40 G00 X-50.;



(2) Cutting feed rate when automatic corner override overlaps inner arc cutting

$$F \times \frac{Rc}{Rp} \times (\text{Automatic corner override}) \times (\text{Feed rate override})$$

(3) Cautions

(a) By parameter setting, inner arc cutting speed change can be enabled regardless of the G62 mode.

(b) When the value of Rc/Rp is smaller than the parameter set value, the minimum deceleration ratio of the parameter set value is used.

(4) Associated parameters

No. 3406,	#0 =	0	Inner arc cutting speed change is regardless of the G62 mode.
		1	Inner arc cutting speed change is enabled in the G62 mode.
No. 1710			Minimum deceleration ratio of the inner arc cutting speed

(5) Associated alarms

3-7 Tapping Mode (G63)

The control state of the NC unit is as follows until G61, G62 or G64 is specified after G63 is specified.

(a) Cutting feed rate override fixed at 100 %.
(b) Feed hold disabled
(c) Spindle override fixed at 100 %
(a) Single block disabled
(e) Decelerated stop disabled at the joint of the blocks (imposition check disabled)
(f) Tapping mode signal output
(g) Dry run disabled

(1) Command format

G63 ;

(2) Sample program

```
N1 G63 G91 G01 X10. F200 ;
N2 Z -30. ;
N3 Z30. ;,
N4 X10. ;
N5 Z - 30. .;
N6 Z30. ;
N7 G64 X10 ;
```

} Tapping mode effective blocks

(3) Cautions

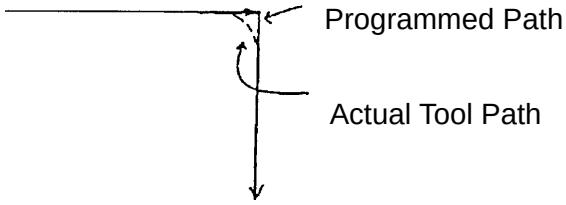
(a) By Parameter setting, you can invalidate dry run disabled.

(4) Associated parameters

No.1401,	#5=	0	Disables dry run for thread cutting and tapping commands.
		1	Enables dry run for thread cutting and tapping commands.

3-8 Cutting Mode (G64)

Until G61, G62 or G63 is specified after G64 was specified, the program makes the next block executed continuously without decleration to a stop between the blocks.
When cutting is performed in the G64 mode, the corner may be rounded at the time of cutting feed.



(1) Associated G code

G61	Exact stop
G62	Automatic corner override
G63	Tapping mode
G64	Cutting mode

(2) Cautions

(a) The G64 mode is set in the reset state.

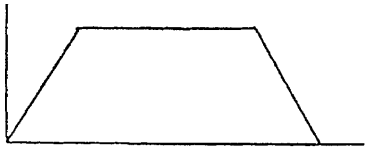
(b) A G00 command decelerates the machine to a stop regardless of the G64 mode.

3-9 Automatic Acceleration/Deceleration

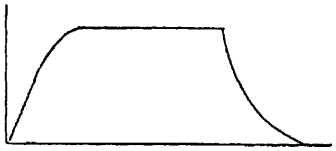
As post-interpolation acceleration/deceleration apply automatically to all commands, no special consideration is necessary.

Rapid traverse (G00,) : Linear type acceleration/deceleration
Cutting feed (G01, G02, G03,) : Exponential type acceleration/deceleration

[Linear type acceleration/deceleration]



[Exponential type acceleration/deceleration]



(a) For rapid traverse, acceleration/deceleration is always pefformed every one block.
(b) For cutting feed, acceleration/deceleration is performed continuously instead of every one block.

(2) Cautions
(a) Optimal values for parameters relating to acceleration/deceleration control have been set for each machine
(b) For cutting feed, acceleration/deceleration is performed continuously instead of every one block.

3-10 Dwell (G04)

When specifying by time: A G04 command causes the machine to wait for the time specified with the address P or X and delays switching to the next block by this time.
When specifying by revolutions: A G04 command causes the machine to wait for the spindle rpm specified with the address P or X And delays switching to the next block for this time.

(1) Command format
(a) When specifying by time

G94 G04 $\left\{ \begin{matrix} P & - \\ X & - \end{matrix} \right\}$;

P_or X_ : Dwell time (0.001 to 99999.999 sec.)

(b) When specifying by revolutions

G95 G04 $\left\{ \begin{matrix} P & - \\ X & - \end{matrix} \right\}$;

P_or X_ :Spindle rotation angle (0.001 to 9999.999 rev.)

(2) Sample program

```
G94 G04 P2000 ;  Dwell time 2 seconds
G04 X2.       ;  Dwell time 2 seconds
```

(3) Cautions

(a) By parameter setting, you can specify by time even during the feed per revolution mode.

(4) Associated parameters

Not 3400, #5 = 0 G04 always specifies by time.
1 G04 follows G94 or G95.

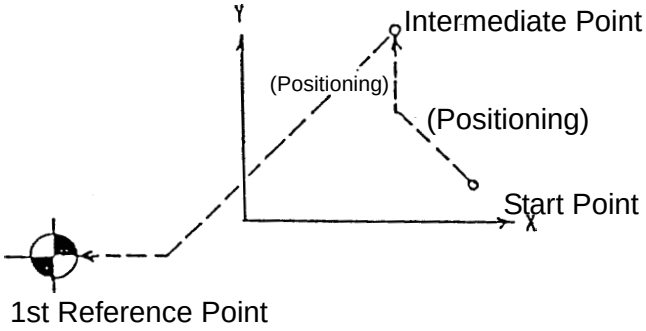
4. REFERENCE POINT

4-1 Automatic Reference Point Return (G28)

After positioning the axes specified by the program to the intermediate point, a G28 command can automatically return them to the 1st reference point.

(1) Command format
G28 X_Y_Z_... ;

(2) Sample program
G28 G91 X-50.Y100. ;



(3) When G28 is specified without completing manual reference point return after turning on the power.
After positioning the program-specified axes to the intermediate point, reference point return is performed automatically in the same sequence as manual reference point return. When OT(overtravel) occurs before completing reference point return, the axes retract by the amount set with the parameters and perform reference point return again.

(4) Cautions
(a) The G28 command onward is high-speed return after completion of reference point return.
(b) The G28 command makes a reference point return check upon completion of high-speed return, and if the axes have not returned, an alarm results.
(c) If G90 G28 G53 X_Y_Z ..., is specified, the axes are positioned to the position specified with G53, and then, they return to the reference point.
(d) The G28 command confirms imposition upon return to the reference point.

(5) Associated parameters

(6) Associated alarms

4-2 Reference Point Return Check (G27)

After positioning the axes to the program -specified position, a G27 command checks whether they have returned to the 1st reference point, and when they have not returned to the 1st reference point, an alarm results.

(1) Command format
G27 X_ Y_ Z_ ... ;

(2) Sample program
G27 G91 X100. Y-50. ;
After the X axis and Y axis moved by 100 and - 50, respectively, it is checked whether they have returned to the 1st reference point.

(3) Cautions
(a) The position reached with a G27 command has tool length compensation, tool offset and tool diameter compensation amounts added to it.
(b) Only the axis specified in the G27 block is checked whether it has returned.
(c) The 2nd to 4th reference point return check is possible.
G27 P_ X_ Y_ Z_ ... ;
Where; P2 - P4 : Specify corresponding to the 2nd through 4th reference points.
(d) The axis for which the machine lock function is turned on has no position check conducted on the reference point.
(e) The G27 command confirms imposition upon return to the reference point.

(4) Associated parameters

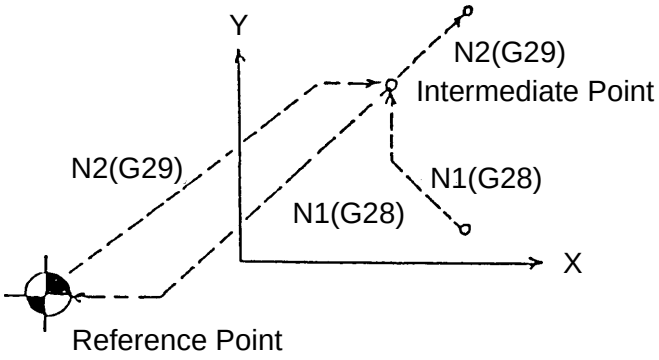
(5) Associated alarms

4-3 Return from Reference Point (G29)

A G29 command positions the program-specified axes from the reference point to the intermediate point of G28 or G30 specified just before, and then, positions them to the specified position.

(1) Command format
G29 X_Y_Z_... ;

(2) Sample program
N1 G28 G91 X-50. Y100. ;
N2 G29 G91 X50. Y50.;



(3) Cautions
(a) If G29 is specified with no G28 or G30 specified once, the intermediate point becomes zero.

4-4 2nd-4th Reference Point Return (G30)

A G30 command can automatically return the axes specified in the program to the 2nd to 4th reference point after positioning them to the intermediate point. The 2nd, 3rd, and 4th reference points are the positions specific to the machine and set with the parameters.

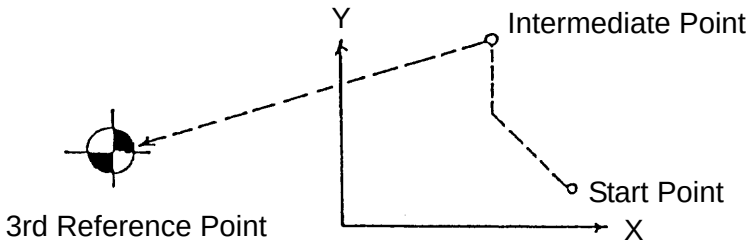
(1) Command format

$$G30 \left\{ \begin{matrix} P2 \\ P3 \\ P4 \end{matrix} \right\} X_ Y_ Z_ ...;$$

where; P2 : 2nd reference point return
P3 : 3rd reference point return
P4 : 4th reference point return
Default : 2nd reference point return

(2) Sample program

G30 G91 P3 X-50. Y100.;



(3) Cautions

(a) The G30 command confirms imposition upon return to the reference point.

(b) If G90 G30 G53 P_X_Y_Z_...; is specified, the axes are positioned to the position specified with G53, and then, returned to the 2nd - 4th reference point.

(4) Associated parameters

No. 1226 2nd reference point of each axis

No. 1227 3rd reference point of each axis

No. 1228 4th reference point of each axis

(5) Associated alarms

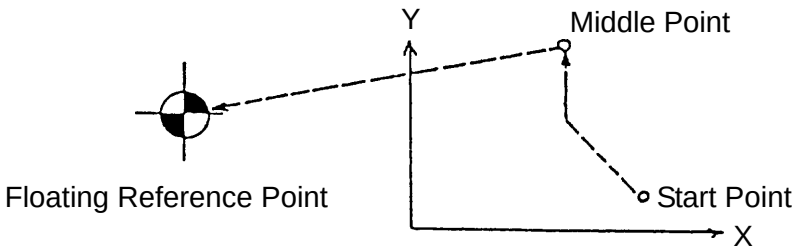
4-5 Reset of Floating Reference point (G301)

G301 instruction may be automatically reset to the following reference point after an axis instructed by the program is positioned at a middle point.
The floating reference point is a selected point on the machine.
The floating reference point can be set in accordance with the following procedure:

① Pressing the [POSITION] key selects the position screen.
② Pressing [F6] and [Y] keys set a present position of the polar coordinate system as a floating reference point.

(1) Command format
G301 X_Y_Z... ;

(2) Sample program
G301 G91 X-50. Y100 ;



(3) Cautions
(a) G301 instruction checks In-Position when reset to the floating reference point.
(b) G90, G301, G53 and X_Y_Z resets the floating reference point after positioning them to G53 instruction.

(4) Associated parameters
No. 1229 Floating reference point of each axis.

5. COORDINATE SYSTEM

5-1 Machine Coordinate System Selection (G53)

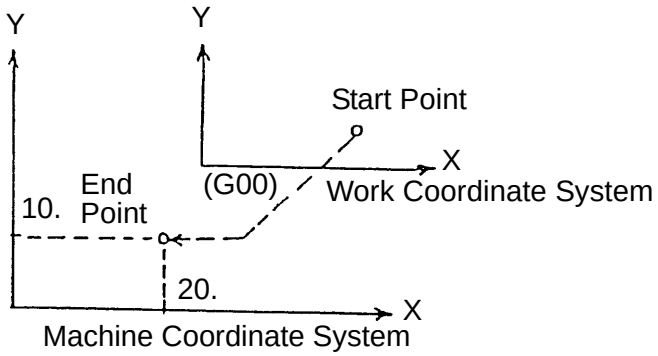
When a G53 Command, the axes are positioned to the position of the machine coordinate system specified by the program.

(1) Command format

G90 G53 X_ Y_ Z_... ;

(2) Sample program

G90 G53 X20. Y10. ;



(3) Cautions

(a) An alarm results if this is specified in the G91 mode.

(b) Specify this after cancelling tool diameter compensation.

(c) The G53 command is effective only in the block where it is specified.

(d) The G53 command confirms imposition at the end point of the block.

(4) Associated parameters

(5) Associated alarms

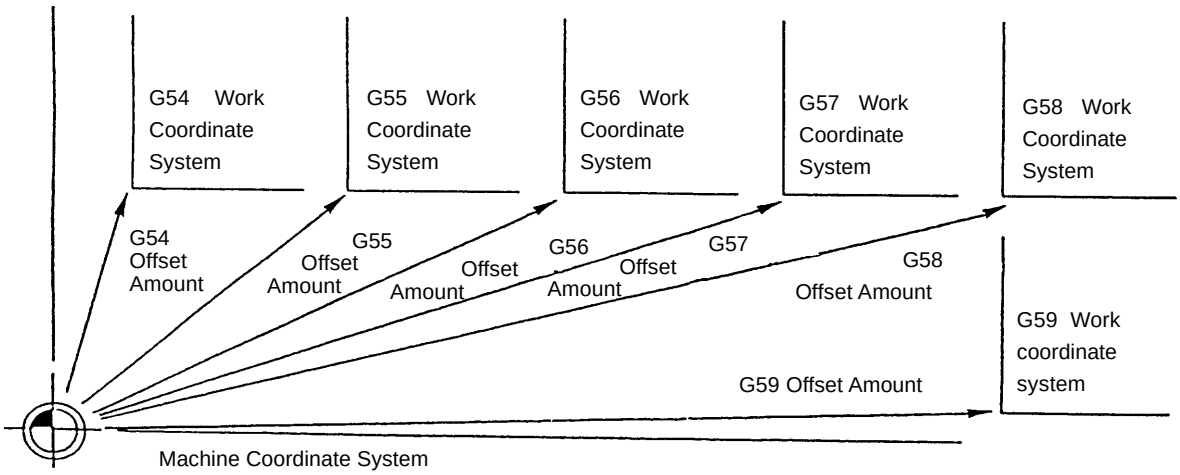
No. 146 Machine coordinate system selection was specified in the G91 mode.

5-2 Work Coordinate system selection (G54 - G59)

Six peculiar coordinate systems can be set by specifying G54 - G59, respectively. Before specifying G54 - G59, set the offset amount (machine coordinate system position when the tool nose is positioned to the zero point of the work coordinate system) from the machine zero point to the zero point of each work coordinate system.

(1) Command format

G54 ; Work coordinate system 1 selection
C55 ; Work coordinate system 2 selection
C56 ; Work coordinate system 3 selection
G57 ; Work coordinate system 4 selection
G58 ; Work coordinate system 5 selection
G59 ; Work coordinate system 6 selection



(2) Sample program

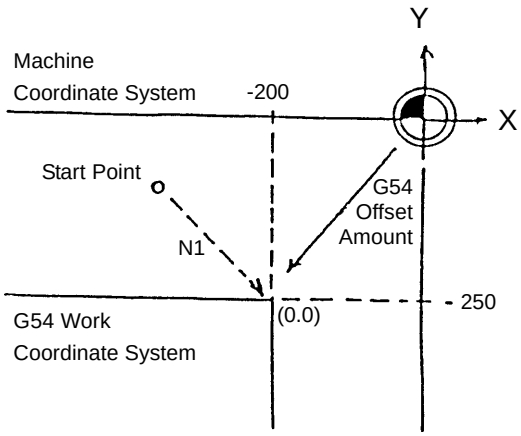
N1 G54 G90 G00 X0 Y0 ;

where; G54 offset amount is;

X - 200.

Y - 250.

This Command positions the axes to (0, 0) of the work coordinate system; the then position of the machine coordinate system is (-200., -250.).



(3) Cautions

(a) The G54 work coordinate system is selected in the reset state.

(b) The G54 coordinate system is set upon completion of zero point return.

(c) When the offset amount of the work coordinate system is changed, the new work coordinate system is set when corresponding G54 - G59 is specified next time.

(d) When G54 - G59 is specified after setting the new coordinate system with a G92 command, the zero point of the machine coordinate system and the offset amount of the G54 - G59 work coordinate system are changed relatively by the amount set newly with G92.

(e) When establishing the G54 - G59 work coordinate system after specifying G92, it is necessarily to specify manual zero point return or G921 (work coordinate system preset) in the reset state.

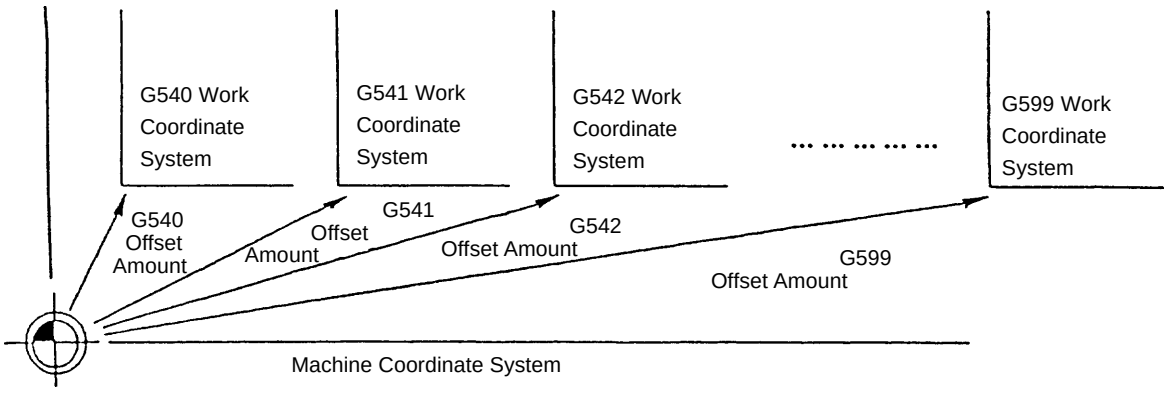
(f) Set the G54 through G59 offset amounts in the work coordinate screen.

5-3 Addition of Work Coordinate system Pairs (G540 - G599)

Sixty peculiar coordinate systems can be set by specifying G540 - G599, respectively. Before specifying G540 - G599, set the offset amount (machine coordinate system position when the tool nose is positioned to the zero point of the work coordinate system) from the machine zero point to the zero point of each work coordinate system.

(1) Commad format

G540 ; G540 work coordinate system selection
G541 ; G541 work coordinate system selection
C542 ; G542 work coordinate system selection
: :
: :
G599 ; G599 work coordinate system selection



(2) Sample program

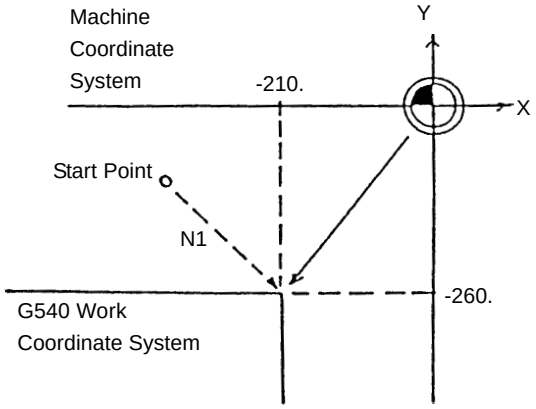
NI G540 G90 G00 X0 Y0;

where; G540 offset amount is;

X - 210.

Y - 260.

This command positions the axes to (0, 0) of the work coordinate system; then the position of the machine coordinate system will be (-210., -260.).



(3) Cautions

(a) G540 through G599 aid G54 through G59 are the G codes of the same group.

(b) When G540 - G599 is specified after setting the new coordinate system with a G92 command, the zero point of the machine coordinate system and the offset amount of the G540 - G599 work coordinate system are changed relatively by the amount set newly with G92.

(c) When establishing the G540 - G599 work coordinate system after specifying G92, it is necessary to specify manual zero point return or G921 (work coordinate system preset) in the reset state.

(f) Set the G540 through G599 offset amounts in the Work Coordinate screen.

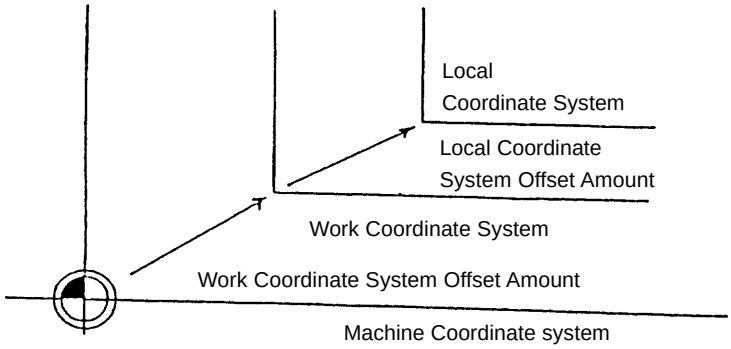
5-4 Local Coordinate System Setting (G52)

One additional coordinate system can be set in the selected work coordinate system by specifying G52.

(1) Command format

G54 X _ Y _ Z _... ; Local coordinate system setting

Offset Amount of Local Coordinate System Setting



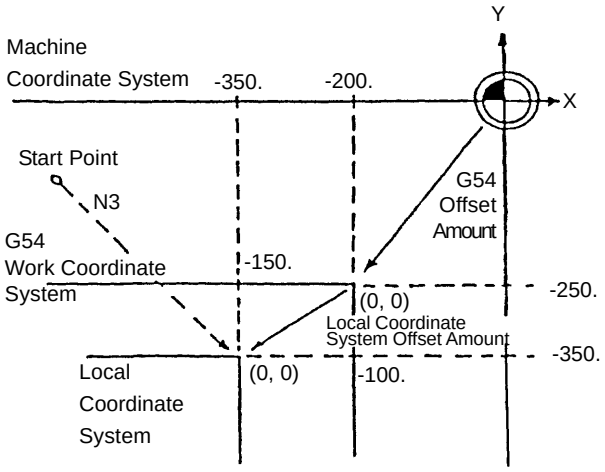
G52 X0 Y0 Z0...; Local Coordinate System Setting Cancel

(2) Sample program

N1 G54 ;
N2 G52 X-150. Y-100.;
N3 G90 G00 X0. Y0.;

where; G54 offset amount;
X - 200.
Y - 250.

Executing the N1 block sets the G54 work coordinate system.
Executing the N2 block sets the local coordinate system.
Executing the N3 block positions the axes to (0, 0) of the local coordinate system; the then position of the machine coordinate system will be (-350., -350.).



(3) Rotation of the offset amount

When the local coordinate system is set during the coordinate rotation (type A) mode, the offset amount rotates. However, rotation of the offset amount is not cancelled even if coordinate rotation is cancelled by specifying G69.
When the local coordinate system is set during coordinate rotation, cancel the local coordinates first and then, cancel coordinate rotation.

(4) Cautions

(a) In the reset state, the local coordinate system A is cancelled.

(b) The local coordinate system of the axis specified with G92 is cancelled.

(c) An alarm results if G52 is specified during the tool diameter compensation mode.

(d) G52 is a one-shot command. The local coordinate system created with G52 remains effective until next G52 is specified.

(5) Related parameters

No. 1200, #0 = 0 The offset Amount of the local coordinate system rotates.

1 The offset amount of the local coordinate system does not rotate.

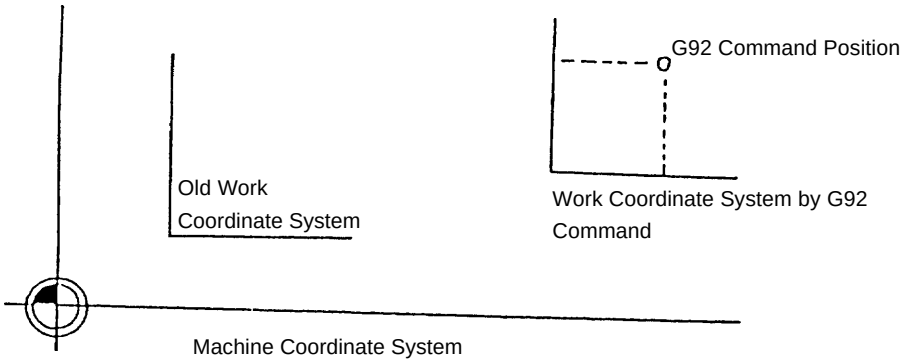
5-5 Work Coordinate System Change (G92)

By specifying G92, you can create the coordinate system on the program without using the G54 -G59 of G540 - G569 work coordinate system.

(1) Command format

G92 X_Y_Z...;

Position to Specify Current Position in G92 Work Coordinate System.

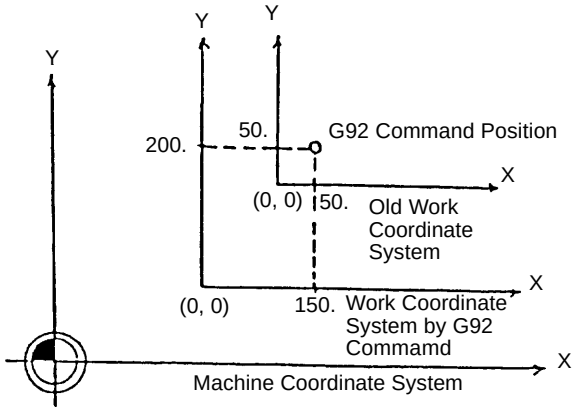


(2) Sample program

N1 G90 G00 X50. Y50. ;

N2 G92 X150. Y200. ;

Executing the N2 block creates the work coordinate system in which the current position will be (150., 200.).



(3) When G92 is specified with the G54 - G59 or G540 - G599 work coordinate system established;

The G92 command shifts all of the G54 -G59 and G540 - G599 work coordinate systems which are based on the machine coordinate system.

(4) Cautions

(a) G92 is a one-shot command.

(b) G92 has nothing to do with absolute (G90)/incremental(G91) programming.

(c) If G92 is specified while tool diameter compensation, tool length compensation or tool offset is applied, the coordinate system is set as if G92 is specified at the position prior to compensation.

(d) The local coordinate system of the axis specified with G92 is cancelled.

(e) There are two methods available for invalidating the work coordinate system with the G92 command.

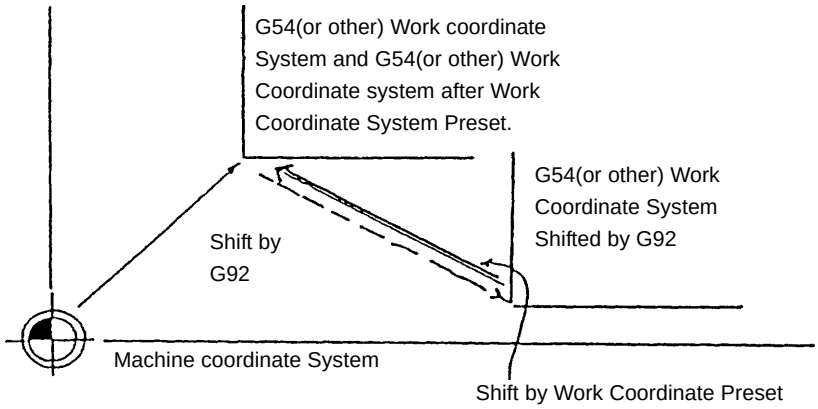
(i) Perform manual zero point return inthe reset state.

(ii) Specify G921 X0 Y0 Z0... ; (work coordinate system preset).

5-6 Work Coordinate System Preset (G921)

If you perform first manual reference point return after turning on the NC unit, the machine coordinate system is set, and then, the work coordinate system is set.
Manual reference point return in the reset state sets the work coordinate system. The work coordinate system is shifted from the machine coordinate system by the following command or operation. In this case, it is called "work coordinate system preset" to set the work coordinate system again based on the machine coordinate system.

(a) When a manual absolute signal is turned off and manual operation intervenes.
(b) When an axis move command is specified in the machine lock mode.
(c) When moving the axes by handle interrupt or manual automatic simultaneous operation.
(d) Work coordinate system shift with the G92 command.
(e) Work coordinate system origin by MDI operation.



(1) Command format/operation

(a) By the G code

G921 X0 Y0 Z0 ...;

Work Coordinate System Preset Axes

(b) By operation

Perform manual reference point return in the reset state (OP signal OFF).

(2) Cautions

(a) Using a G921 Command cancels tool diameter compensation, tool length compensation and tool offset.

(b) By parameter setting, the work coordinate system can be preset just before executing the first block in which the machine is switched over from the reset state to automatic operation.

(3) Associated parameters

No. 1202, #0 Presetting of work coordinate system on recovery of manual reference point is:

=0 Carried out in reset state.

=1 Carried out in any circumstance.

No. 1202, #2 Presetting of work coordinate system on recovery of manual reference pint:

=0 Follows #0 of Parameter No. 1202.

=1 No presetting takes place for any axis.

No. 1202, #3 Work coordinate system, on starting of auto operation following reset state :

=0 Does not perform presetting.

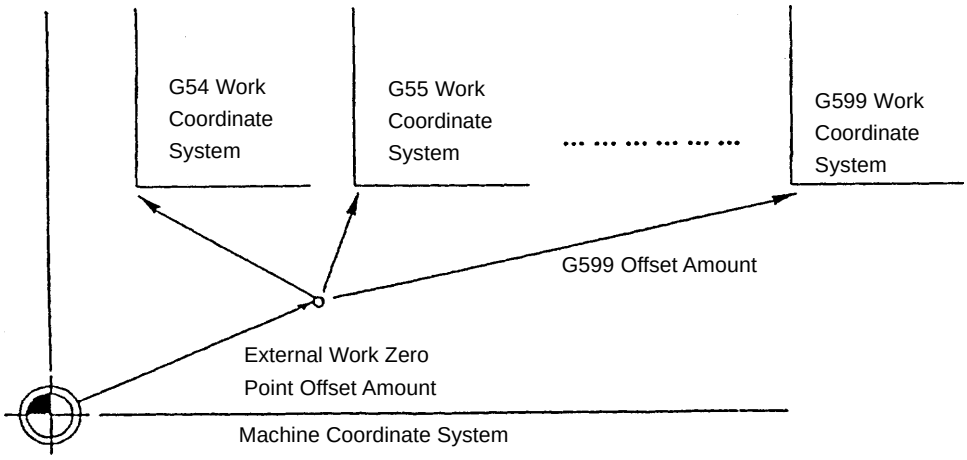
=1 Performs presetting.

(Note 1) When "presetting takes place in any circumstance" (No.2403, #0=1) has been set, processing is carried out only when no pre-read block exists and a execution block has been completed.

(4) Associated alarms

5-7 Work Coordinate System Shift (External Work Zero Point Offset Amount)

The entire G54 or other work coordinate system can be shifted by the specified amount (external work zero point offset amount) by setting an offset amount in the external offset of the work coordinate system in the NC screen.



(Note 1) When the external work zero point offset amount is changed, it becomes effective from the block where the G54 or other work coordinate system is specified.

(Note 2) If set to parameter No. 1201, #0.= 1, enabled as soon as the external work origin offset amount is changed.

5-8 Plane Selection (G17, G18, G19)

With a G17, G18. or G19 Command, this function specifies the plane in which an arc command, tool diameter compensation, coordinate rotation, etc. are performed.

(1) Command format

G17 X_P_ Y_P _ ; X_P-Y_P plane
G18 Z_P_ X_P_ ; Z_P-X_P plane
G19 Y_P_ Z_P_ ; Y_P-Z_P plane
G17 ; X-Y plane
G18 , Z-X plane
G19 ; Y-Z plane

where; X_P : X axis or its parallel axis
Y_P : Y axis or its parallel axis
Z_P : Z axis or its parallel axis

(2) Sample program

```
G00 G91 ;
G17 U0 Y0;                    Selects the U-Y plane.
    :                        }
    :                        } The U-Y plane is effective.
    :                        }
G00 G91 ;
G18 W0 X0;                    Selects the W-X plane
    :                        }
    :                        } The W-X plane is effective.
```

(3) Cautions

(a) Set with the parameter with which one of the X, Y and Z axes the additional axis is parallel.
(b) Set with the parameter whether the reset state is to be the G17 or G18 mode.
(c) An alarm results when the plane is not determined in the block where G17, G18 or G19 was specified.
(Example) G17 X_ U_ Y_ ; It is uncertain whether the X-Y or U-Y plane is selected.
(d) A move command is irrelevant to plane selection.

(4) Associate parameters

No. 3402, #5 =0 The reset state is the G17 mode

1 The reset state is the G18 mode

(5) Associate alarms

No. 106 A plane seleection (G17 - G 19) commend has an error.

5-9 Rotary Table Dynamic Fixture Offset

When loading a workpiece on the rotary table and set a work coordinate system after measuring a position of workpiece if the rotary table has rotated before starting cutting, the work coordinate system should be set again by measurement of a position of workpiece once more.

However, if a position of workpiece at a certain position of the rotary table is set at the standard fixture offset amount, even if the rotary table rotate, calculate a fixture offset amount at this time automatically from an angle of the rotary table and make a work coordinate system according to this value. In view of these facts, if a position is set once at the standard fixture offset amount, the work coordinate system can be held dynamically corresponding to a position even if The rotary table rotates any position.

A position adding a work reference point offset amount on the fixture offset amount on the fixture offset amount becomes the work coordinate reference point.

(1) Setting of data.

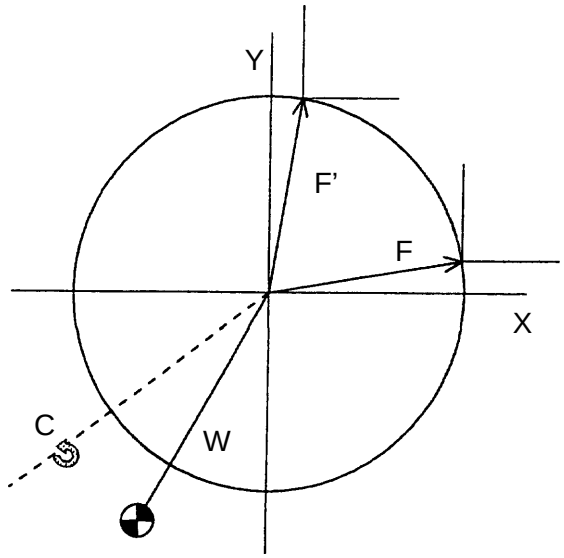
① Three-piece set of parameter designated a rotary axis and two linear axes constructed a rotary plane.

In each set, the 1st parameter is for a rotary axis number, and the 2nd and 3rd ones for linear axis numbers. Set the two linear axis numbers so that rotation from the positive direction of the 2nd linear axis to that of the 3rd linear axis will be positive directional rotation of the 1st rotary axis.

To calculate a fixture offset amount, calculate it for the rotary axis of the 1st set, and then, calculate it for the 2nd and 3rd sets with respect to that result. If you do not need as many as 3 sets, set 0 for the rotary axis.

When there are two or more rotary axes and the rotating plane is changed by other rotation, set the rotating plane when the position of the rotary axis is 0 degree.

② Standard fixture offset amount at the standard angle of rotary axis. Set a certain position of rotary axis (standard angle) and a fixture offset amount at the time. Setting is done by the fixture offset screen. Number of sets are 8 sets.



W : Work Shift Amount
F : Fixture Offset Amount at Reference Angle

③ Parameter for effective/ineffective of fixture offset for each axis (No. 1208, #0)
Set 1 for the axis to be effective of fixture offset.

④ Type of fixture offset (No.1200, #1)
When a vector of fixture offset has changed (command of G522 or when a rotary axis is moved during a command of G522),set whether a changing amount of vector is moved or not moved.
When it is 0, it moves. (Work coordinate is not changed and machine coordinate is changed.)
When it is 1, it does not move. (Work coordinate is changed and machine coordinate is not changed.)

(2) Command form
G522 Pn ;
n : No. of fixture offset amount (1-8)
By a command of G522 Pn ;,calculate a fixture offset amount by a present rotary angle and a data designated by n and make the fixture offset amount effective.
By a command of G522 P0 ;,make the fixture offset amount ineffective.
Also, if a rotary axis which is related a fixture offset during G522 mode is moved, execute a calculation of vector again.

(3) Program example and notion

parameter No.1285 4 (C axis)

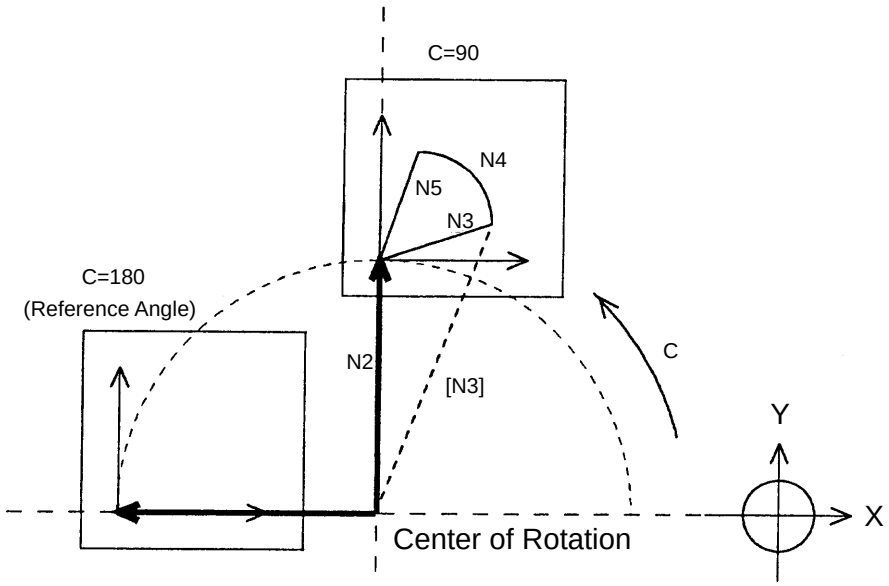
No.1286 1 (X axis) No.1208, #0 (X) =1 (X axis is effective)

No.1287 2 (Y axis) No.1208, #0 (Y) =1 (Y axis is effective)

Data of n= 1 X = -10.0

Y = 0

C = 180.0 (standard angle)



	Work coordinate value			Machine coordinate value			Fixture offset amount		
Program	X	Y	C	X	Y	C	X	Y	C
N1 G90 G00 X0 Y0 C90.;	0	0	90.	0	0	90.	0	0	0
N2G 522 P1	0	0	90.	0	10.	90.	0	10.	0
[	0	-10.	90.	0	0	90.	0	10.	0]
N3 G01 X10. Y2. F100;	10.	2.	90.	10.	12.	90.	0	10.	0
N4 G02 X2. Y10. R10.;	2.	10.	90.	2.	20.	90.	0	10.	0
N5 G01 X0 Y0,.	0	0	90.	0	10.	90.	0	10.	0

In the bracket show #1 = 1 of parameter 1200

(4) Motion at reset

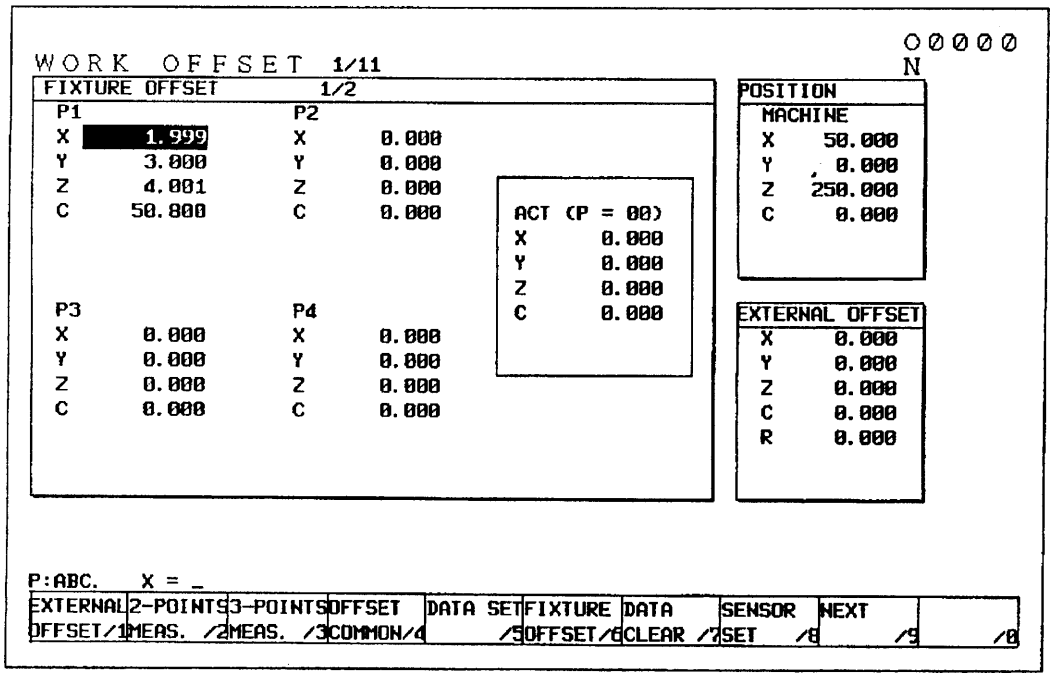
At the time of reset, whether the fixture offset is cancelled or not is according to the #2 of parameter 1200.

When it is 1, a vector before reset is held.

When it is 0, a vector is cleared. However, the machine does not move for a vector cleared.

(5) Screen of fixture offset

To select the Fixture offset screen, use F6/FIXTURE OFFSET or F4/WORK COORDINATE.



ACT at the upper left of the screen displays the currently selected fixture offset number (P) and fixture offset vector.

(6) Input/output of fixture offset amount

Setting and input/output of data in a program with outside is available.

(a) Setting of standard fixture offset amount by G10

Standard fixture offset amount or standard angle in a program can be set.

G10 L22 P_ X_ Y_ Z ... ;

P1 - P8 : Fixture offset No.

X, Y, Z... : Standard fixture offset amount or standard angle of each axis.

When it is executed in the G90 mode, a value itself is set.

When it is executed in the G91 mode, a value add on the previous value is set.

(7) Precautions

(a) A command of G522 and rotary axis related a fixture offset during a command of G522 is restrained a buffering.

(b) If a parameter or standard fixture offset amount are changed during the G522 mode, it becomes effective after a command of next G522.

(c) If the following command is executed for a rotary axis during the G522 mode, a calculation for vector of fixture offset is not executed.
Also, command the first designated rotary axis by the G90 mode.
Command by the machine coordinated system (G53)
Change of the work coordinate system (G92, G52, G54 to G59, G540 to G599)
Reference point return related (G27, G28, G30, G301)

(d) A movement by changing a vector of fixture offset is according to a G code of group 01 at that time. A command of G522 should be done in the G00 or G01 mode.

(e) When stopping an automatic operation by the single block during the G522 mode etc. and move a rotary axis by hand, a vector of fixture offset is not changed. Execute a calculation of vector when a commanding a rotary axis at automatic or MDI operation or commanding the G522.
However, after a rotary axis is moved by hand, execute a calculation of vector by using a coordinate value which is nor reflected an intervenient value by hand when a rotary axis is command in the G91 mode.

(f) The coordinate values of the rotary axis used in fixture offset calculation are those in the work coordinate system. If an offset is applied such as a tool offset, use the coordinate values available prior to applying an offset. Use also the pre-offset coordinate values as to mirror image and scaling.

(g) High-accuracy contour control cannot be command in the G522 mode.
Even if high-accuracy contour control is provided in the G522 node and the rotary axis is moved, the vector will hot be calculated.

(h) When using the rollover function for the rotary axis, be sure to assume a move amount per revolution of the rotary axis to be 360 degrees.

(8) Related parameters

No.1200,	#1	= 0	Moves when the fixture offset vector changes.
		= 1	Does not move when the fixture offset vector changes.
No.1200,	#2	= 0	Clears the modal information by reset.(clears the vector)
		= 1	Does not clear the modal information by reset.(Holds the vector)

No.1200,	#3	= 0	Enables only the fixture offset axis setting parameters in the 1st set.
		= 1	Enables the fixture offset axis setting parameters in the three sets.
No.1208,	#0	= 0	Disables fixture offset (for each axis).
		= 1	Enables fixture offset (for each axis).
No.1285			Fixture offset target rotary axis number (1st set)
No.1286			Axis number of the linear axis-1 which consists the fixture offset plane (1st set)
No.1287			Axis number of the linear axis-2 which consists the fixture offset plane (1st set)
No.1288			Fixture offset target rotary axis number (2nd set)
No.1289			Axis number of the linear axis-1 which consists the fixture offset plane (2nd set)
No.1290			Axis number of the linear axis-2 which consists the fixture offset plane (2nd set)
No.1291			Fixture offset target rotary axis number (3rd set)
No.1292			Axis number of the linear axis-1 which consists the fixture offset plane (3rd set)
No.1293			Axis number of the linear axis-2 which consists the fixture offset plane (3rd set)

(9) Related alarms

No.155	Fixture offset command error
(#001)	No offset number
(#002)	Offset number beyond the range
(#003)	Erroneous parameter setting (No. 1285 to No. 1293)
(#004)	Not in the G00 or G01mode

6. COORDINATE

6-1 Absolute/Incremental Programming (G90,G91)

In programming, you can select either absolute progaramming which causes an axial move following the axial address to move to the specified position of the coordinate system, or incremental programming which causes it to move to the incremental position of the current position.

(1) Command format

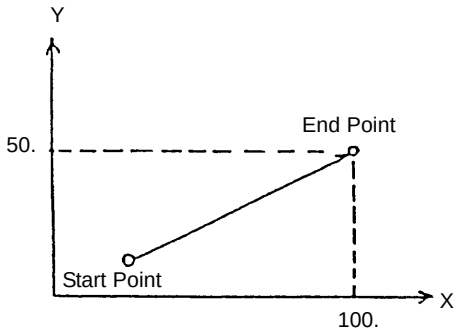
G90 ; Absolute programming

G91 ; Incremental programming

(2) Sample program

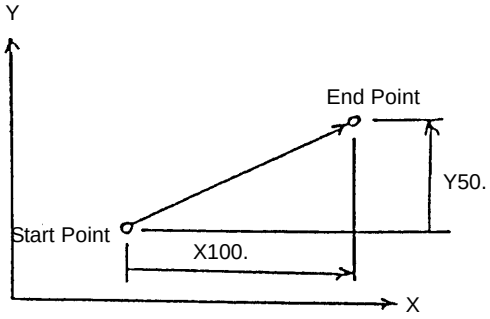
(a) Absolute programming

G90 G01 X100. Y50. F200 ;



(b) Incremental programming

G90 G01 X100. Y50. F200 ;



(3) Cutions

(a) The G90 and G91 command are modal G codes.
Once G90 is specified, it remains effective until G91 is specified.

(b) Set with the parameter whether the reset state is to be the G90 or G91 mode.

(4) Associated parameters

No. 3402, #1 = 0 The reset state is the G91 mode.

1 The reset state is the G90 mode.

6-2 Polar Coordinate Input (G15,G16)

This command allows you to specify the end point coordinate value of the mochining program in terms of radius and angle.

(1) G code

G15 : Polar coordinate command cancel

G16 : Polar coordinate command ON

(2) Command format

G16 ; Polar coordinate command ON

: }

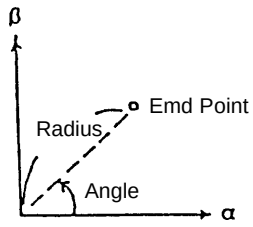
: } Polar coordinate command cancel

: }

G15 ; Polar coordinate command cancel

The following shows the command format in the polar coordinate command mode.

$\left\{\begin{matrix}G17\\G18\\G19\end{matrix}\right\}$ $\left\{\begin{matrix}G90\\G91\end{matrix}\right\}$ $\alpha_\beta_$;



where; α : Radius (1st axis in the plane)

β : Angle (2nd axis in the plane)

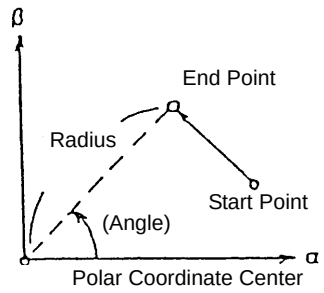
(a) Specify the plane for a polar coordinate command with plane selection (G17, G18, G19).

(b) The radius is given as follows.

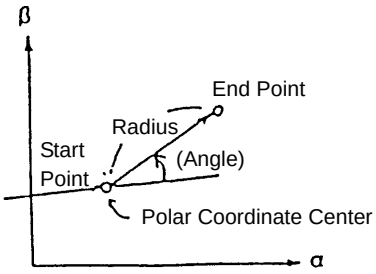
(i) When a radius value is given by absolute programming, the zero point of the local coordinate system (G52)
(zero point of the G54 or other work coordinate system when the local coordinate system has not been set) will be the polar coordinate center.

(ii) When the radius value is given by incremental programming, the current position will be the polar coordinate center.

<u>When the Radius Value Is Absolute</u>



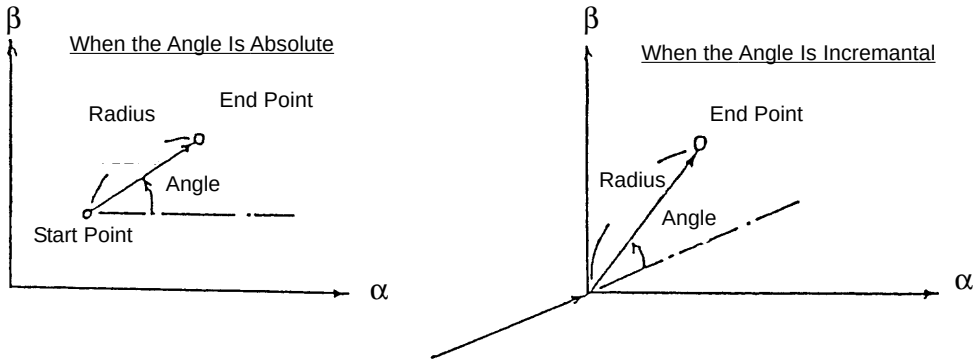
<u>When the Radius Value Is Incremental</u>



(c) The angle is given as follows

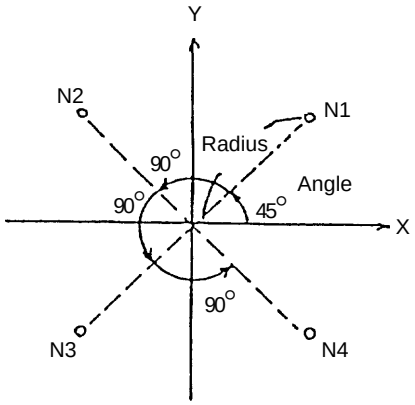
(i) When the angle is given by absolute programming, it will be exactly the angle specified in the block.

(ii) When the angle is given by incremental programming, it will be added to the angle set in the previous block.



(3) Sample program

```
G17  ;
G54 G90 G00 X0 Y0  ;
G43 Z0 H01  ;
G16  ;                                  Polar coordinate command ON
G81 G90 X100. Z-15. R-3 F100 L0  ;
N1 G91 Y45. ;                           Rabius value: Absolute command
N2 Y90. ;
N3 Y90. ;                               Angle: Incremental command
N4 Y90. ;
N5 G15 G80 ;                            Polar coordinate command cancel
```



(4) Cautions

(a) The following G codes are invalid in the polar coordinate command mode.

G04, G10, G52, G92, G53,
G22, G68, G511, G501, G51

(b) The radius for circular interpolation and helical cutting in the polar coordinate command mode should be specified by radius designation on arc.

6-3 Inch/Metric Input (G20, G21)

With a G20 or G21 command, either inch or metric system can be selected as the increment system of program command.

(1) Command format
G20 ; Inch system
G21 ; Metric sysytem
(Note) Specify this in an independent block at the beginning of the program.

(2) The following systems of units are changed with the G20 and G21 commands.
(a) Feed rate command with an F code
(b) Commands related to the position
(c) Work zero point offset amount, external work zero point offset amount, common work zero point offset amount
(d) Tool offset amount
(e) Part of parameters
(f) Unit of one graduation of the manual pulse generator

(3) Cautions
(a) Specify the G20/G21 command in an unit block.
(b) When the G20/G21 command is executed, preset the work coordinate system.
(c) Do not switch the G20/G21 command halfway the program.
(d) Switching inch/metric input can switch by parameter (setting). For switching used by parameter, turn off the power and then turn on.

(4) Associated parameters

No. 1000, #0 = 0 The increment system is metric.

1 The increment system is inch.

(5) Associated alarms

7. SPINDLE FUNCTION (S FUNCTION)

With the number of revolution of the main spindle (rpm) being commanded in a numerical value of max. 5 digits following Address S, binary code signals, strobe signals (SF), analog signals corresponding to the spindle motor rpm, gear signals, etc. are sent out to Machine side.

(1) Specify a set of S command in one block.

(2) Program example:

```
S1000;          Spindle speed 1000 rpm
M03;            Spindle rotation
 :
 :
M05;            Spindle stop
```

(3) Cautions

(a) When a negative value has been specified in an S command, the sign always becomes positive.

8. TOOL FUNCTION (T FUNCTION)

With a value of max. 8 digits following Address T being command, code signals of BCD 8 digits and strobe signals (TF) are output in Machine side.

(1) Specify a set of T command in one block.

(2) Program example:

```
T01;                    Tool No. 01 is set to Standby
 :
G30 G90 X0 Y0 Z0 M19;
T01 M06;                Tool change (Tool No. 01 is set to the main spi ndle.)
```

(3) Cautions

(a) When a negative value has been specified in a T command, the sign always becomes positive.

9. Miscellaneous Function (M FUNCTION)

9-1 Miscellaneous Function (M Function)

If the address M followed by an up to 8-digit numerical value is specified, the BCD 8-digit code signal and strobe signal (MF) are output to the machine side.

(1) Specify a set of M command in one block.

(2) The following M commands are used as special functions.

(a) M00 (Program stop)
If M00 is specified during automatic operation , automatic operation stops after executing that block.
Pressing the CYCLE START switch restarts automatic operation.

(b) M01 (Operational stop)
If the OPERATIONAL STOP switch of the machine operation panel is turned on and M01 is specified, automatic operation stops after executing that block.
Pressing the CYCLE START switch restarts automatic operation.

(c) M30 (Program end)
Specify M02 at the end of the program. If M02 is specified during automatic operation, automatic operations tops after executing that block, and the NC unit is reset.

(d) M30 (Program end)
Specify M30 at the end of the program.
If M30 is specified during automatic operation, automatic operations tops after executing that block, the NC unit is reset, and you are returned to the beginning of the program.

(e) M98 (Subprogram call)
Call a subprogram with the command ; M98 P_ ;

(f) M99 (subprogram end)
Specify M99 at the end of the subprogram.

(3) Sample program

```
G30 G91 X0 Y0 Y0 M19      ;
T01 M06 ;                          Tool change
M03  ;                             Spindle forward rotation
G54 G90 G00 X0 Y0   ;
G43 Z0 H01;
  :
  :
M05 ;                              Spindle stop
M30 ;                              Program end
 %
```

(4) Cautions

(a) If M00, M01, M02 or M30 is specified, the NC unit stops prereading.

(b) When M98 or M99 is specified, the code signal and strobe signal are not sent out.

(c) When a negative value is specified in an M command, the sign always become positive.

9-2 2nd Miscellaneous Function (B Function)

If you specify any one of the parameter-set address U, V, W, A, B and C followed by an up to 8-digit numerical value (-79999999 ~ +79999999), the BCD 8-digit code signal and strobe signal (BF) are output to the machine side.

(1) Specify a set of 2nd miscellaneous function in one block.

(2) Sample program

```
B180;                Indexes the index table by 180°.
  :
  :
B270;                Indexes the index table by 270°.
```

(3) Caution

(a) The 2nd miscellaneous function is output to the machine side as 32-bit signal data. Therefore, its programmable range is -79999999 ~ +79999999.

(b) Set the command address (U, V, W, A, B, C) of the 2nd miscellaneous function in the parameter No. 1020.

(4) Associated parameters

No.1020	Command address of the 2nd miscellaneous function

(5) Associated alarms

10. Canned Cycle

10-1 Canned Cycle (G73, G74, G76, G80 - G89)

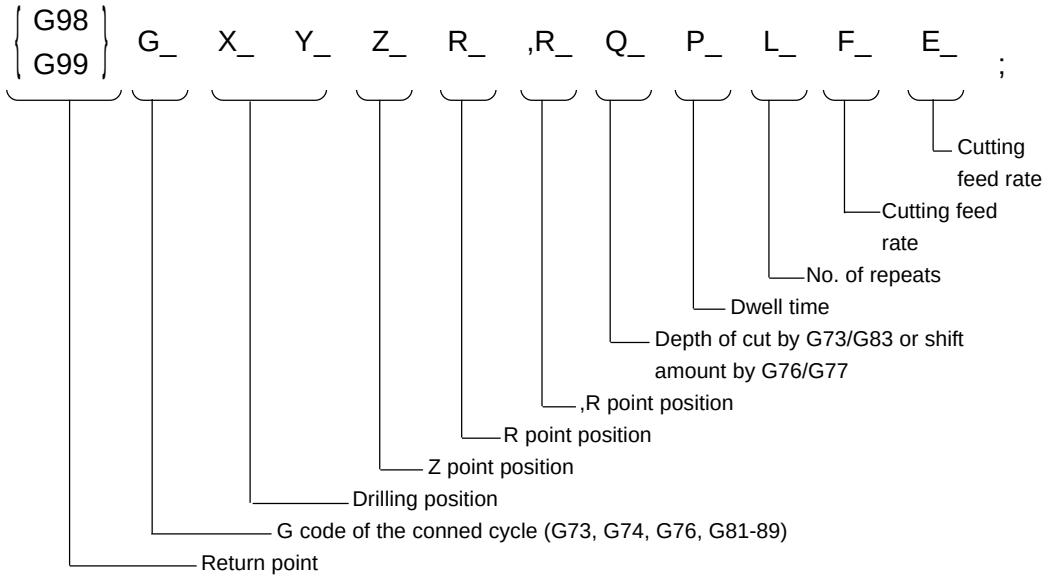
This function allows you to specify the machining cycle such as drilling, tapping, boring, etc. in one block.

The canned cycle is cancelled if you specify G80 or the G code of Group 01 (G00, G01, G02, G03, etc.) during the canned cycle mode.

(1) G code

G73 : High-speed peck drilling cycle
G74 : Counter tapping
G76 : Fine boring
G80 : Canned cycle cancel
G81 : Drilling, spot drilling
G82 : Drilling, counter boring
G83 : Peck drilling cycle
G84 : Tapping
G85 : Boring
G86 : Boring
G87 : Boring, back boring
G88 : Boring
G89 : Boring

(2) Command format



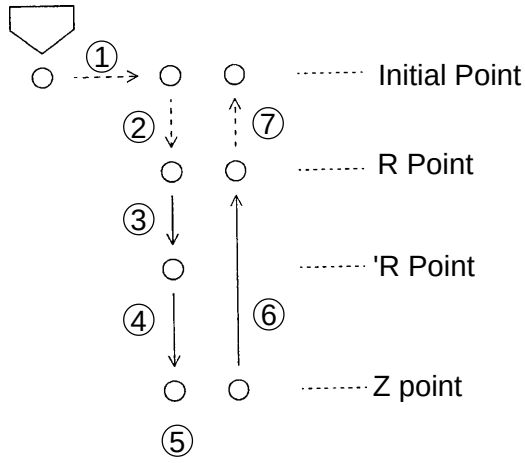
(Note 1) men the number of repeats (L) is omitted, L = 1 is assumed.
If L = 0 is specified, the tool moves to the drilling position to invalidate drilling.

(Note 2) I, J and K may be used in the G73, G76, G83 and G87 commands.

(Note 3) The R point, z point, P, Q, I, J and K are modal in the canned cycle mode.

(3) Machining cycle

The canned machining cycle generally consists of the following movements ① through⑥.



① : Positioning to the drilling position
② : Rapid traverse to the R point
③ : Cutting feed to the ,R point (Feed rate E)
④ : Drilling to the Z point (Feed rate F)
⑤ : Movement at the Z point
⑥ : Return to the R point
⑦ : Rapid traverse to the initial point

Where; - - - → Rapid traverse
——→ Cutting feed

(Note 1) Single block operation is prohibited in the movements ④ ,⑤ and ⑥ .

(Note 2) The ,R-Point command is allowed in the following cycles:

- G81, G82 (Drilling cycle)
- G73, G83 (Peck drilling cycle)

If the ,R-command is omitted, the movement ④ (feed rate F) will be executed after the R-Point, skipping the movement ③.

If the E-command is omitted, the movements designated at the feed rats E will run at the feed rate F.

(Note 3) The E-command in the tapping cycle (G74, G84) will result in the feed rate for the movement ⑥ . If the E-command is omitted, the movement ⑥ will also run at the feed rate F.

(4) Return point

The return point of the canned cycle is specified with the following G codes.

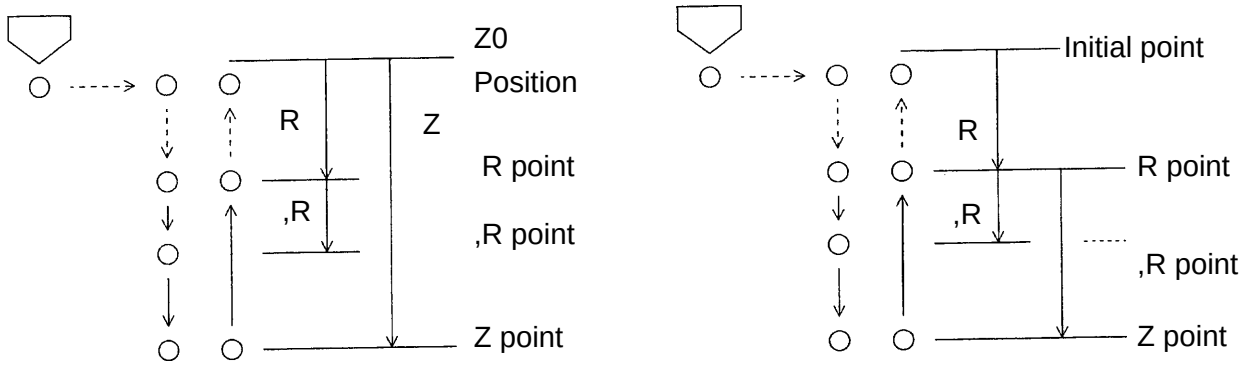
G98 : Initial point level return
G99 : R point level return.

(Note 1) The initial point is the position of the drilling axis when the canned cycle cancellation state is switched to the canned cycle mode.

(5) "R-point," "Z-point," and ",R-point"

The R-point and Z-Pint will be as follows in the G90 and G91 commands, but the ,R-point will be always of incremental command.

[G90] [G91]



(Note 1) The ,R-point is at a position incremental from the R-point.

(6) Plane of the drilling position and drilling axis

The plane of the drilling position is determined with a G17, G18 or G 19 command. The drilling axis is the X, Y or Z axis which does not constitute the plane of the drilling position, or its parallel axis.

Whether the drilling axis is the basic axis of parallel axis depends on the drilling axis address Specified in the block containing G73, G 74, G76 or G81 - G89.

G17 G_ X_P_ Y_P_ Z_P_ R_ Q_ P_ L_ F_ :

Mode: G_ | Drilling position: X_P_ Y_P_ | Drilling Axis: Z_P_

G18 G_ Z_P_ X_P_ Y_P_ R_ Q_ P_ L_ F_ :

Mode: G_ | Drilling position: Z_P_ X_P_ | Drilling Axis: Y_P_

G19 G_ Y_P_ Z_P_ X_P_ R_ Q_ P_ L_ F_ :

Mode: G_ | Drilling position: Y_P_ Z_P_ | Drilling Axis: X_P_

where : X_P : X axis or its parallel axis

Y_P : Y axis or its parallel axis

Z_P : Z axis or its parallel axis

(Note 1) When Changing over the plane of the drilling position, cancel the canned cycle first.

(Note 2) By Parameter setting, the drilling axis can be limited to the Z axis regardless of the G17 - G19 commands.

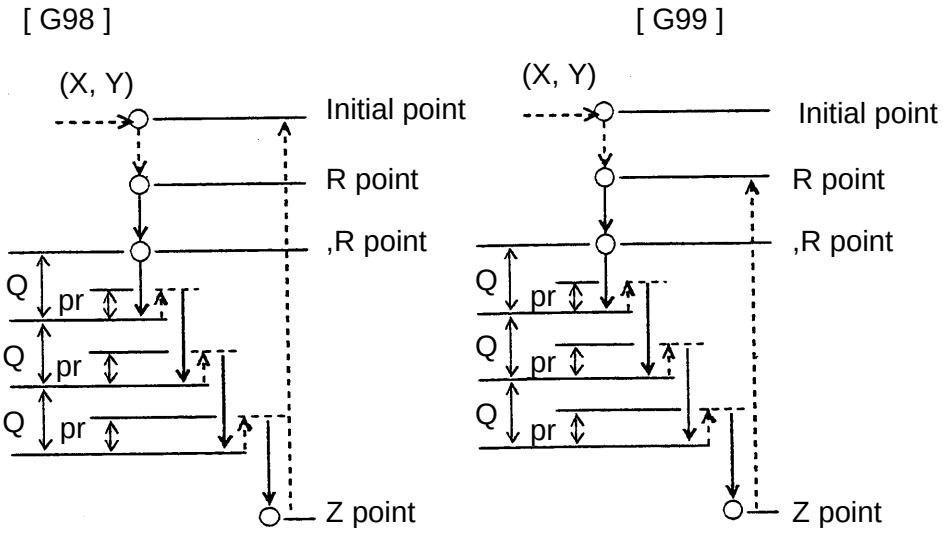
(7) Description of the movements in the canned cycle

The following. description of the movements in the canned cycle assumes the drilling position to be on the X-Y Plane, and the drilling axis to be the Z axis.

(a) G73 (High-speed peck drilling drill)

[Fixed pitch]

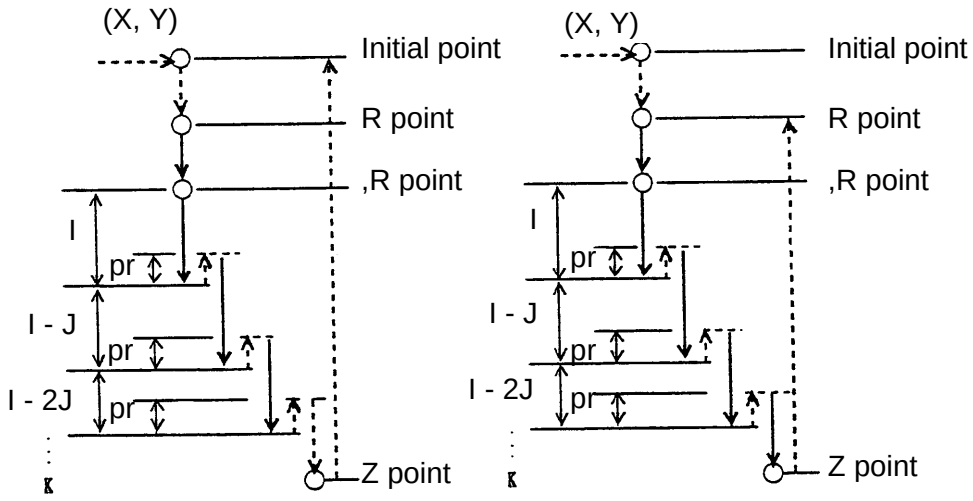
$\left\{\begin{matrix}G98\\G99\end{matrix}\right\}$ G73 X_ Y_ Z_ R_ ,R_ Q_ P_ L_ F_ E_ ;



pr : Parameter setting

[Variable pitch]

$\left\{\begin{matrix}G98\\G99\end{matrix}\right\}$ G73 X_ Y_ Z_ R_ ,R_ I_ J_ K_ P_ L_ F_ E_ ;



pr : Parameter setting

where ; I : Initial value of the depth of cut (positive value)

J : Decremental value in 2nd cut onward (positive value)

K : Final value of the depth of cut (positive value)

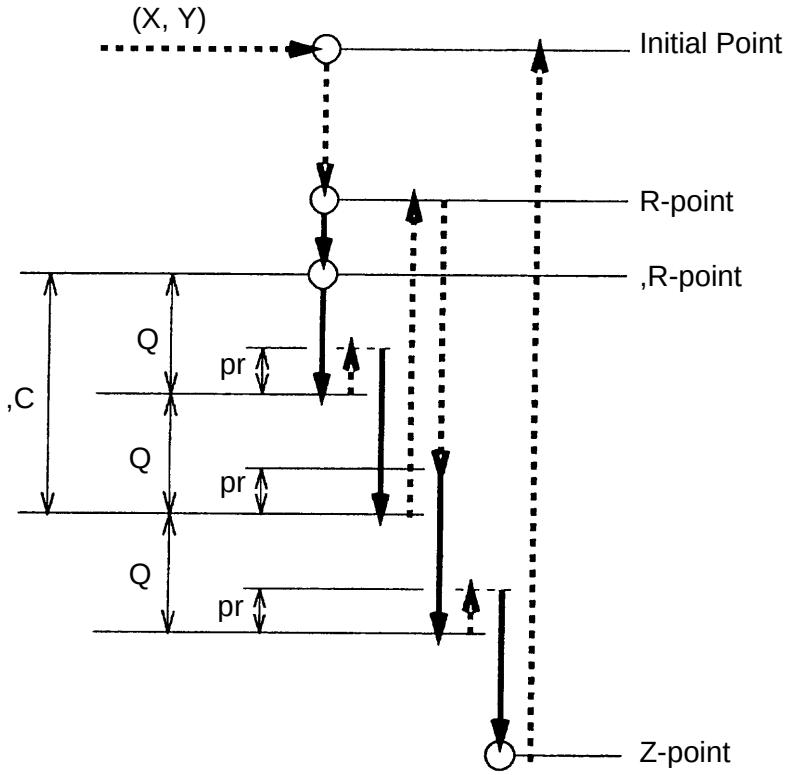
(Note 1) When there is a Q command before specifying the variable pitch with I, J and K, specify Q.

(Note 2) Dwell operation is enabled at the hole bottom by parameter setting.

(Note 3) Giving a ",C" command withdraws the tool halfway drilling.

G73 X_ Y_ Z_ R_ , R_Q_P_L_F_E_, C_;

The following figure shows an operational example of initial point return.

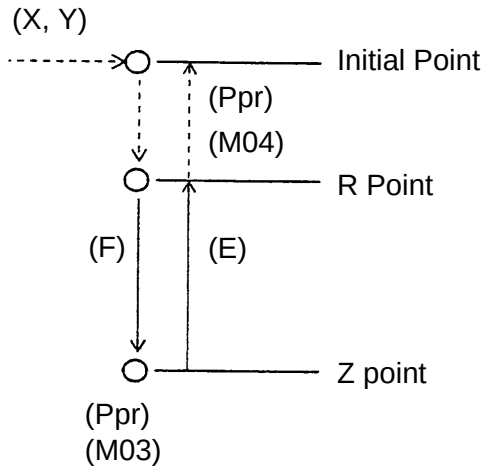


",C" denotes an incremental amount from the R-point top the Z-point.
A sign is invalid.
The tool stops at the R-point during single block operation.
The tool is withdrawn by the value given by ",C." For example, if ",C50" is given, a withdrawal position will be 50., 100., 150., and so on.
When passing through the withdrawal position in drilling operation(move by the Q or I/J/K value), the withdrawal position will be the end point of that operation.
The address ",C" is valid only if it is given in the same block as G73.

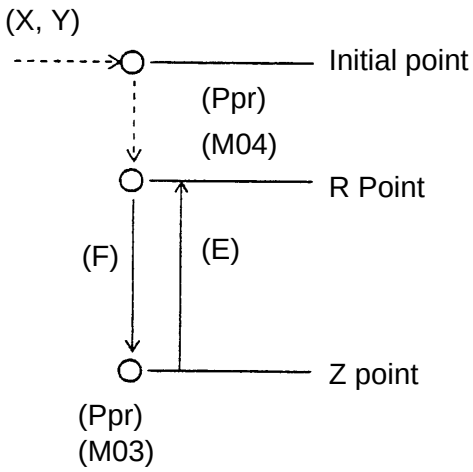
(b) G74 (counter tapping)

$\left\{\begin{matrix}G98\\G99\end{matrix}\right\}$ G74 X_ Y_ Z_ R_ P_ L_ F_ E_;

[G98]



[G99.]



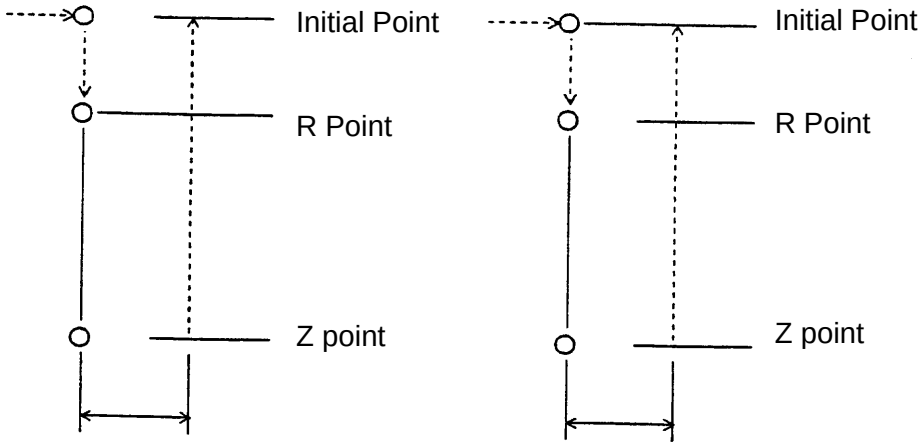
(M03) : Spindle forward
(M04) : Spindle reverse
(Ppr) : Dwell (by parameter setting)

(Note 1) By parameter setting, (Ppr) is invalidated.
(Note 2) Feed hold And override are prohibited during cutting.

(c) G76 (Fine boring)

$\left\{ \begin{matrix} G98 \\ G99 \end{matrix} \right\}$ G76 X_ Y_ Z_ R_ P_ Q_ L_ F_ ;

[G98] [G99]



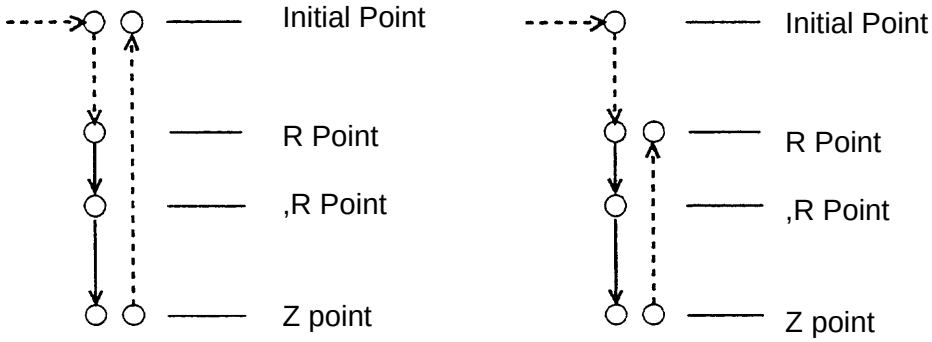
(M03) : Spindle forward
(p) : Dwell
(M19) : Spindle index stop
: shift (rabid traverse linear interpolation)

(Note 1) By parameter setting, specify the shift amount with I, J and K instead of Q.
G17 command : I, J (X_P - Y_P plane)
G18 command : K, I (Z_P - X_P plane)
G19 command : J, K (Y_P - Z_P plane)

(d) G81 (Drilling)

$\left\{ \begin{matrix} G98 \\ G99 \end{matrix} \right\}$ G81 X_ Y_ Z_ R_,R_ L_ F_ E_;

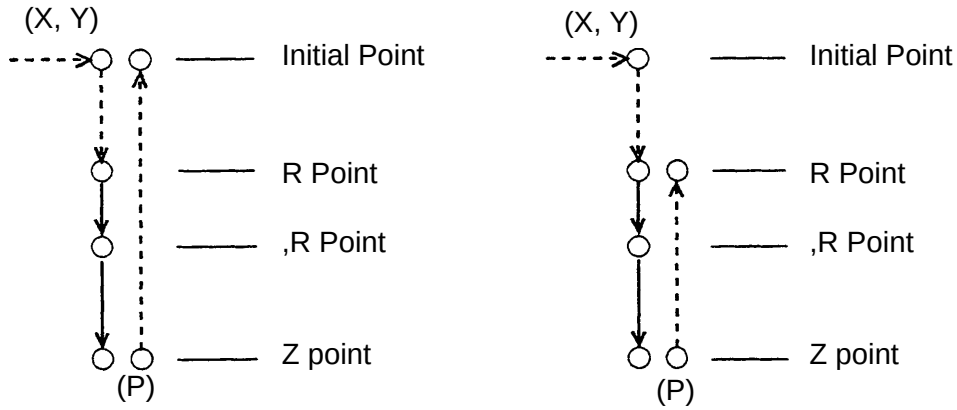
[G98] [G99]



(e) G82 (Drilling)

$\left\{ \begin{matrix} G98 \\ G99 \end{matrix} \right\}$ G82 X_ Y_ Z_ R_ ,R_ P_ L_ F_ E_ ;

[G98] [G99]



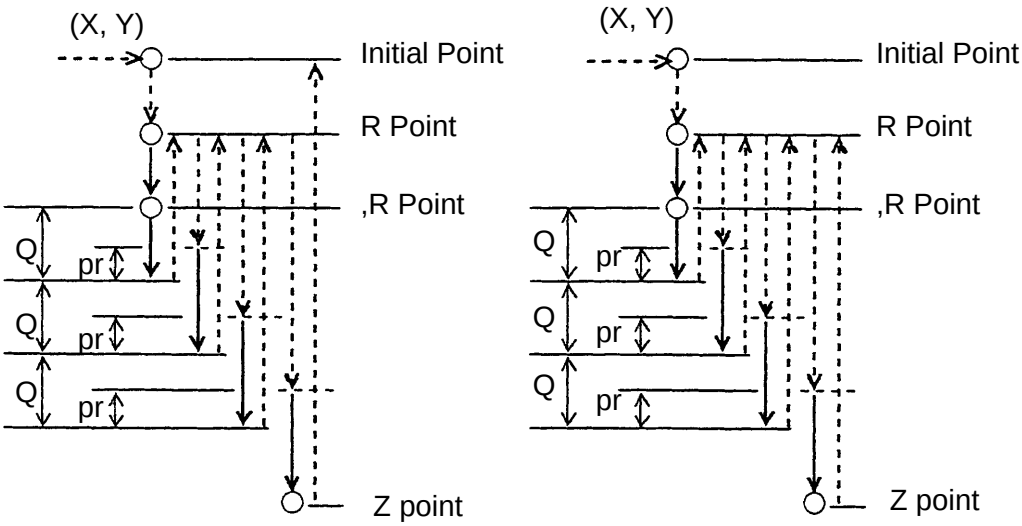
(P) : Dwell

(f) G83 (Peck drilling)

[Fixed pitch]

$\left\{ \begin{matrix} G98 \\ G99 \end{matrix} \right\}$ G83 X_ Y_ Z_ R_ ,R_ I_ J_ K_ P_ L_ F_ E_ ;

[G98] [G99]

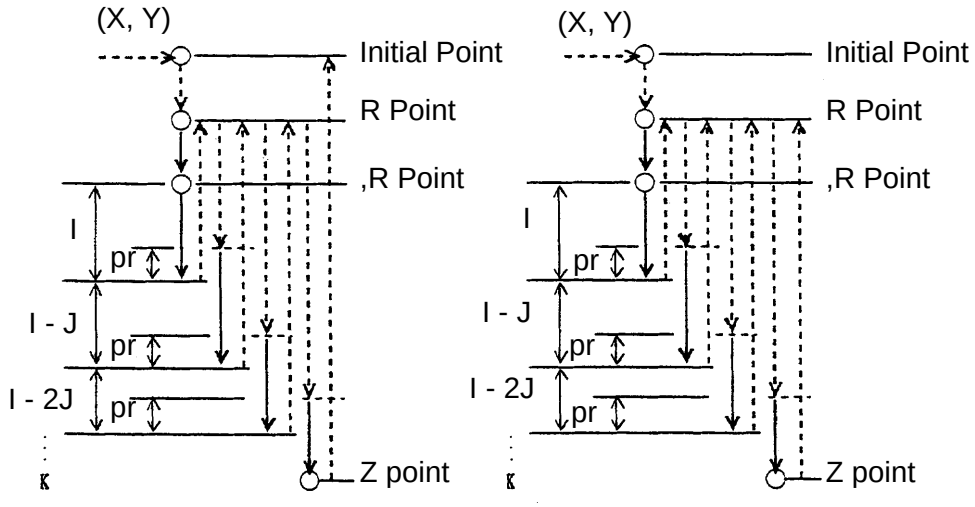


Pr: Parameter setting

[Variable pitch]

$\left\{ \begin{matrix} G98 \\ G99 \end{matrix} \right\}$ G83 X_ Y_ Z_ R_ ,R_ I_ J_ K_ P_ L_ F_ E_ ;

[G98] [G99]



pr : parameter setting

Where; I : Initial value of the depth of cut (positive value)

J : Decremental value in 2nd cut onward (positive value)

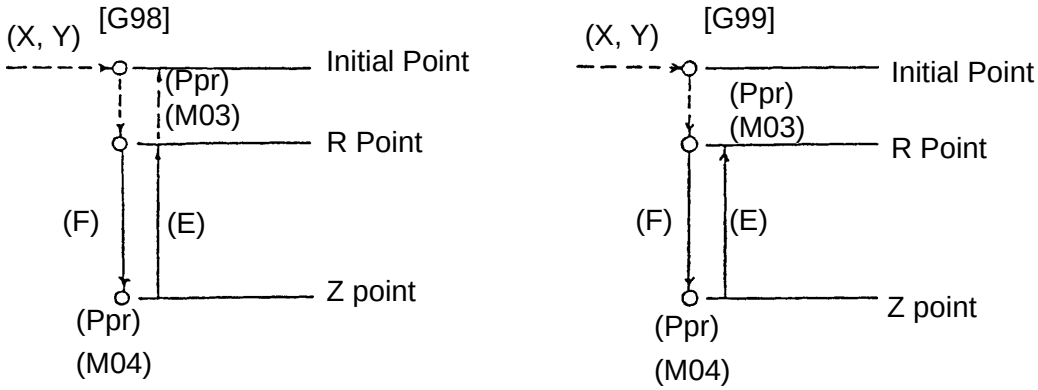
K : Final value of the depth of cut (positive value)

(Note 1) When there is a Q command before specifying the variable pitch with I, J and K, specify Q0.

(Note 2) Dwell at the hole bottom is enabled through parameter setting.

(g) G84 (Tapping)

$\left\{ \begin{matrix} G98 \\ G99 \end{matrix} \right\}$ G84 X_ Y_ Z_ R_ P_ L_ F_ E_ ;



(M03) : Spindle forward

(M04) : Spindle reverse

(Ppr) : Dwell (by parameter setting)

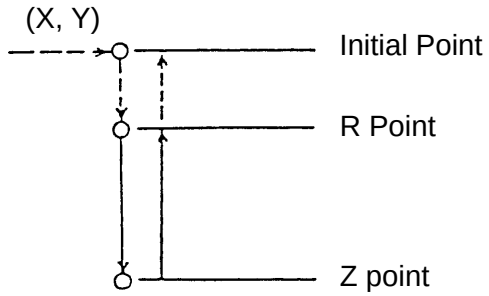
(Note 1) By parameter setting, (Ppr) is invalidated.

(Note 2) Feed hold and override are prohibited during cutting.

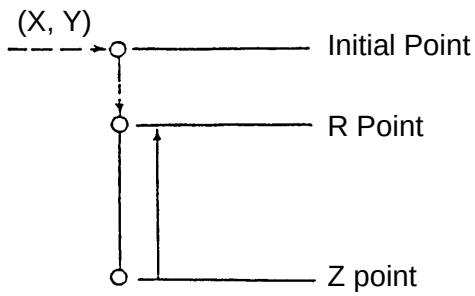
(h) G85 (Boring)

$\left\{ \begin{matrix} G98 \\ G99 \end{matrix} \right\}$ G85 X_ Y_ Z_ R_ L_ F_;

[G98]



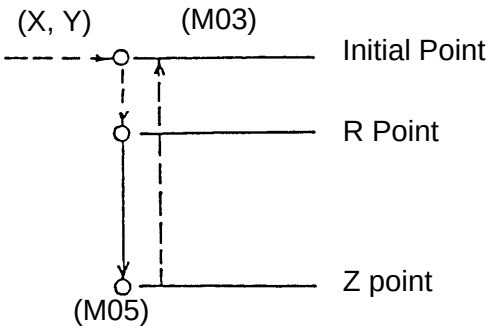
[G99]



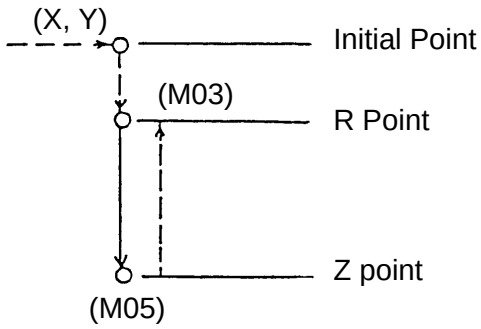
(i) G86 (Boring)

$\left\{ \begin{matrix} G98 \\ G99 \end{matrix} \right\}$ G86 X_ Y_ Z_ R_ L_ F_;

[G98]



[G99]

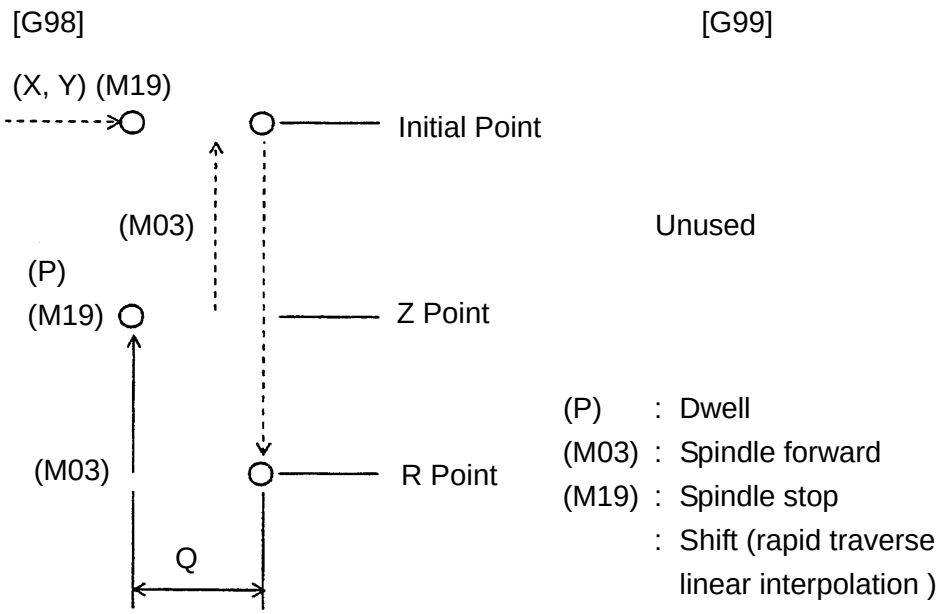


(M03) : Spindle forward

(M04) : Spindle stop

(j) G87 (Back boring)

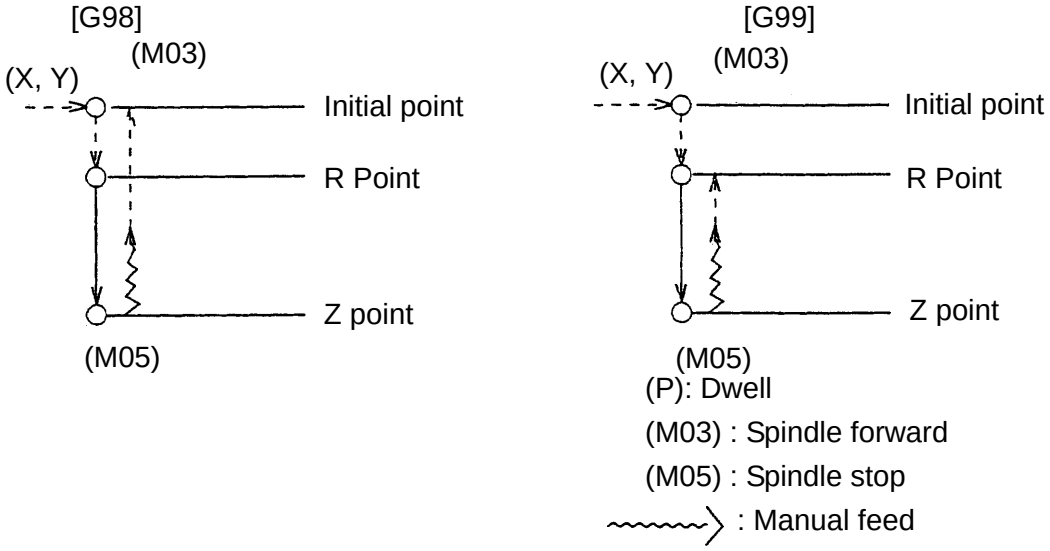
{G98 / G99} G87 X_ Y_ Z_ R_ P_ Q_ L_ F_ ;



(Note 1) By parameter setting, you can specify the shift amount with I, J and K instead of Q.
G17 command : I, J (X_P -Y_P plane)
G18 command : K, I (Z_P -X_P plane)
G19 command : J, K (Y_P -Z_P plane)

(k) G88 (Boring)

{G98 / G99} G88 X _ Y _ Z _ R _ P _ L _ F _ ;

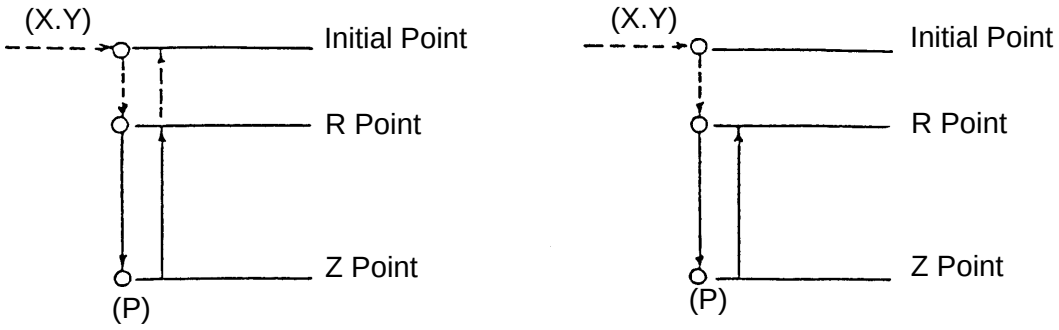


(Note 1) If the Z axis is reached and the spindle stops after swell, the machine results in the single block stop state. Manual feed is allowed by selecting the manual mode. Operation can be continued by selecting the automatic node and pressing the CYCLE START button.

(l) G89 (Boring)

$\left\{\begin{matrix} G98 \\ G99 \end{matrix}\right\}$ G89 X_ Y_ Z_ R _ P_ L_ F_ ;

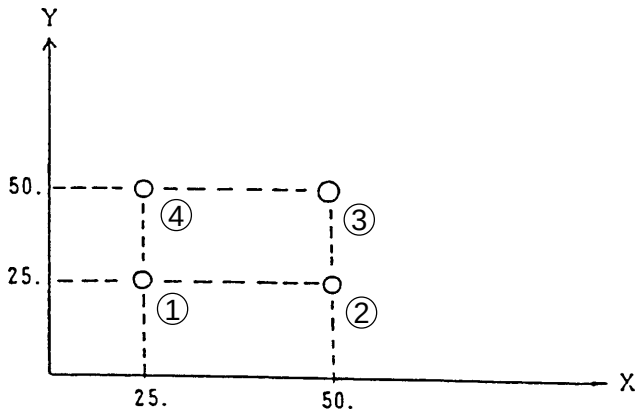
[G98] [G99]



(P) : Dwell

(8) Sample program

```
G17 G54 G90 G00 X0 Y0 ;
G43 Z0 H01 ;
M03 S1000 ;                    Spindle forward
G73 Z-50. R-5. Q5. L0 F200 ;   Stores the machining data of the canned cycle.
G99 X25. Y25. ;                Peck drilling cycle ①.
X50. ;                         Peck drilling cycle ② .
Y50. ;                         Peck drilling cycle ③ .
G98 X25. ;                     Peck drilling cycle ④ .
G80 ;                          Cancels the canned cycle.
```



(9) Caution

(a) When the SINGLE BLOCK button is turned on, the tool stops at each end point of the movements ①, ②, ③, ⑥ and ⑦.
In this case, the FEED HOLD lamp is turned on at each end point of the movement ①, ②, ③, and ⑥ , and the movement ⑦ when the number of repeats is left incomplete.
By parameter setting, a single block stop is allowed between the movements ④ and ⑥ in the cycle other than the tapping cycle (G74, G84).

(b) If the FEEDHOLD is pressed in the movements ④ to ⑥ in the G74 or G84 mode, the FEEDHOLD lamp will be illuminated immediately and the machine will stop after continuing its operation to the end point of the movement ⑥.

(c) The feed rate override for the movements ④ ~ ⑥ of G74/G84 is fixed at 100%. Dry run can be enabled/disabled by parameter setting.

(d) When the drilling axis is limited to the Z axis by parameter setting, drilling is performed in the block where any one of the X, Y, Z, R and additional axes is specified.
When L0 has been specified, however, the tool moves to the drilling position and cancels drilling.

(e) Specify Q, P, I, J in K in the block where any one of the X, Y, Z, R and additional axis has been specified. In the other blocks, they are invalid as the data of the canned cycle.
Also, Q, P, I, J and K in the block where the G code of group 00 has been specified (except G09 and G 601) are invalid as the data of the canned cycle.

(f) The canned cycle is cancelled by specifying the G code of Group 01 (G00, G01, G02, G03, etc.) instead of G80.

(g) If M, S or T is specified in the block where the canned cycle has been specified, it is sent out at the time of the first movement ① (positioning to the drilling position).
When the number of repeats (L) has been specified, M, S and T are sent out in the first time only.

(h) The spindle can be switched to the high-speed gear in the movement ① of G74/G84. The S value to switch to is set with the parameter.

(i) The numerical values of P. Q. I. J. K. L and F should be given in positive values.

(10) Associated parameters

No.1401, #5=	0	Dry run is valid for tapping command.
	1	Dry run is invalid for tapping command.
No.3407, #0=	0	Does not make single block stop for each canned cycle for drilling.
	1	Makes single block stop for each canned cycle for drilling.
No.5100, #0=	0	Drilling axis of canned cycle is always z.
	1	Drilling axis of canned cycle is the axis selected by program.
No.5100, #2=	0	G76/G87 shift direction is specified with parameter, while shift amount is specified with Q.
	1	G76/G87 shift direction and shift amount are specified with I, J, and K.
No.5100, #3=	0	M05 output while in M03↔M04 reversal is invalid.
=	1	M05 output while in M03↔M04 reversal is valid.
No.5100, #5=	0	Dwell addition with P command in G73/G83 is invalid.
	1	Dwell addition with P command in G73/G83 is valid.
No.5127		Tool relief direction after G76/G87 orientation
No.5150		Retract amount for canned cycle G73
No.5151		Clearance amount for canned cycle G83

(11) Associated alarms

No.133 A canned cycle (G73-G89) command has an error.

(#000) Z point (hole bottom position)has not been specified in the first block of canned cycle.

(#001) Back boring has been specified in R-Point Recovery mode.

(#100) When shift amount is specified with Q in back boring (G87)/ fine boring (G76), relief axis and direction following orientation has not been set for Parameter 5127 boring axis. (5127 boring axis = 0)

(#101) When shift amount is specified with Q in back boring (G87)/ fine boring (G76), The same axis has been set for Parameter 5127 boring axis. (5127 boring axis = boring axis)

10-2 Direct Tap (G741, G841)

This function synchronizes the; spindle with the feed axes and allows high-speed high-accuracy tapping.
Conventional tappers are unnecessary.

(1) Command format

[G741 / G841] [G98 / G99] [G94 / G95] X_ Y_ Z_ R_ P_ Q_ L_ S_ F_ E_ ;

G841 : Forward direct tap (Note 6)
G741 : Rivers direct tap (Note 6)
G98/G99 : Initial point return/R-point return
G94/G95 : Feed per minute/feed per revolution
X_ /Y_ : Drilling position (Note 2)
Z_ : Z-point position (Note 2)
R_ : R-point position
P_ : Dwell time
Q_ : Depth of cut ((4) Pecking cycle function)
L_ : No. of repeats (Note 1)
S_ : Spindle speed (Note 5)
F_ : G94 mode --- Drilling axis going feed rate (R-point to Z-point)
G95 mode --- Tap pitch
E_ :
G95 mode --- No. of threads per inch

(Note 1) When the number of repeats (L) is omitted, it is assumed to be L = 1.
When L = 0 is specified, the tool moves to the drilling position, but does not drill.
(Note 2) The Z-point position is specified with the address of the drilling axis.
The drilling position is specified with the address of the non-drilling axis.
(Note 3) The R-point, Z-point, P, and Q are modal during a canned cycle.
(Note 4) The "return point," "R-point" and "Z-point," and "drilling axis" and "positioning axis to the drilling position" are the same as those for the "canned cycle for drilling."
Refer to 11-1.

(Note 5) S/F commands are made valid in a block where G841 (G741) has been specified, serving to determine feed rate and pitches.

Example :

```
G841      Z_ R_ F_ S_ ;
X_ Y_ ;                  · When specifying feed rata, F/S=pitch
X_ Y_ ;                  · When specifying pitches, F x S=feed rate
G80 ;
```

(Note 6) Direct tap G-codes (G84l, G741) can be made compatible with the Σ10L by parameter setting.

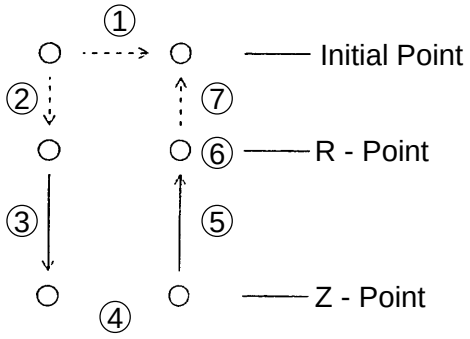
G741 replaced by G843

G841 replaced by G842

(2) Machining cycle

A general machining cycle for the direct tap consists of the movements ① through ⑦.

[G841] - - - - Rapid Traverse ——— cutting Feed



① Positioning to the drilling position
② Rapid traverse to the R-point
③ Tool forward rotation and drilling to the Z-point
④ Dwell operation by parameter setting
⑤ Tool reverse rotation and return to the R-point, and tool rotation stop
⑥ Dwell operation by parameter setting
⑦ Rapid traverse to the initial point

(Note) With G741, tool rotating directions will be opposite to G841

(3) Feed rate sitting and pitch setting (F-command)

The direct tap has different meanings of F-command between the feed per minute mode (G94) and feed per revolution mode (G95).

- G94 mode : F represents a drilling axis feed rate.
  (mm/min., in./min.)

- G95 mode: F represents tap pitches. (mm, in.)
  E represents the number of threads per inch.
  (threads/in.)

The number of effective digits for the F-command differs as follows, depending on parameter settings.

• G94 mode:	Metric input:	F60: 1 to 999.999 (mm/min.)
		F61: 0.1 to 999,999.9 (mm/min.)
	Inch input :	F51: 0.1 to 99,999.9 (mm./min.)
		F52: 0.01 to 99,999.99 (in./min.)
• G95 mode:	Metric input:	F32: 0.01 to 999,99 (mm/rev)
		F33: 0.001 to 999,999 (mm/rev)
	Inch input :	F23: 0.001 to 99,999 (in./rev)
		F24: 0.0001 to 99.9999 (in./rev)

(Note 1) The G95 mode also assumes feed per minute.

(Note 2) The number of effective digits for F and E is the same.

(Note 3) When F and E are specified in the same block, F becomes valid.

(Note 4) In the G94 mode, the pitches are determined by F and S (tool's rpm).

$$\text{pitch (mm, in.)} = \frac{\text{F(mm/min, inch/min)}}{\text{S(r.p.m)}}$$

(4) Pecking cycle function

When performing deep tapping in direct tapping, it may be difficult due to entangled cutting chips or increased cutting resistance.

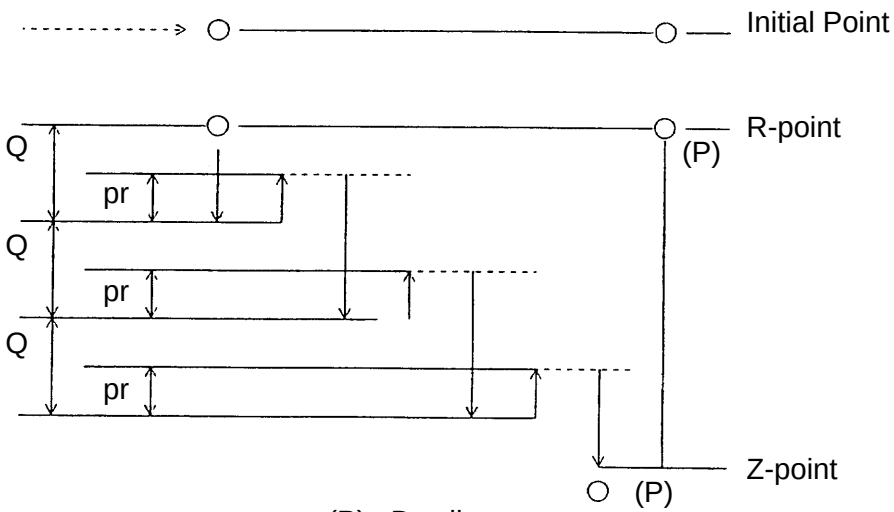
In that case, this function allows you to perform cutting, dividing between the R-point and Z-point into several sections.

When parameter No. 5105, #4 = 0 is set, the tool returns to the R-point through pecking operation. When it is set to 1, the tool returns by the amount specified by a parameter(No. 5157) through pecking operation.

The following figure shows an example of Parameter No. 5105, #4 = 1.

[G741 / G841] [G98 / G99] G94 X_ Y_ Z_ R_ P_ Q_ L_ S_ F_;

[G741 / G841] [G98 / G99] G95 X_ Y_ Z_ R_ P_ Q_ L_ S_ F_ E_;



(P) : Dwell

(Note 1) A return amount pr is set with a parameter. (No.5157)

(Note 2) In the G94 mode, do not specify E.

(5) Notes

(a) The description in this section assumes a drilling position to be on the XY plane, and a drilling axis to be the Z-axis.

(b) Dwell operation can be enabled/disabled by parameter setting.

(c) During tapping, feed rate override and spindle override are fixed at 100%. Dry run can be enabled/disabled by parameter setting.

(d) In the single block mode, the tool stops at the initial point and R-point.

(e) When FEED HOLD is pressed during tapping, the FEED HOLD lamp goes on immediately and the tool stops after continuing to move to the R-point.

(f) To cancel direct tapping, specify G80 or any G-code in the G01 group (G00, G01, G02,.....). However, do not specify other canned cycle (G70, G71, etc.) in the same block as a cancel command.

(6) Related parameters

No.1401, #5 =	0	Dry run is enabled for a tapping command
	= 1	Dry run is disabled for a tapping command
No.3401, #0 =	0	F23 is used for feed per revolution for the inch system
	= 1	F24 is used for feed per revolution for the inch system
No.340l, #1 =	0	F32 is based for feed per revolution for the metric system
	= 1	F33 is used for feed for revolution for the metric system
No.3401, #2 =	0	F51 is used for feed per minute ford the inch system
	= 1	F52 is used for feed per minute for the inch system
No.3401, #3 =	0	F60 is used for feed per minute for the metric system
	= 1	F61 is used for feed pr minute for the metric system
No.5100, #1 =	0	In direct tapping, a dwell command by P is disabled
	= 1	In direct tapping, a dwell command by P is enabled
No.5101, #4 =	0	Direct tapping G codes do not inc1ude M codes:
	= 1	Direct tapping G codes include M21 and G84(G74).
No.5101, #5 =	0	Direct tapping G codes include G841/G741.
	= 1	Direct tapping G codes include G842/G843.

No.5105, #4 =	0	Returns to the R-point through pecking operation.
=	1	Returns by the amount specified by a parameter(No. 5157) through pecking operation.
No.5200, #4 =	0	In direct tapping, override on returning operation is invalid.
=	1	In direct tapping, override on returning operation is valid.
No.5200, #6 =	0	In direct tapping, Feed Hold and Single Block are invalid.
=	1	In direct tapping, Feed Hold and Single Block are valid.
No. 5157		Return volume of pecking cycle
No. 5211		Override value on returning operation in direct

(7) Associated Alarms

No.138	A direct tap command has an error.
(#0000)	Z point (hole bottom position) has not been specified in the first block of canned cycle.
(#001)	S=0
(#002)	F=0
(#005)	Any other canned cycle has been specified in a block together with a cancel command (G80, G00...).
(#006)	E=0
(#010)	G33 or G34 mode

10-3 Drilling Pattern Cycle (G70, G71, G72, G77)

[Purpose]

When drilling the holes at equal intervals on the circumference, this function automatically calculates the orthogonal coordinate value with the radius and angle and positions the tool to that position.

(1) Command format

G70 X_ Y_ I_ J_ L_ ;

G70	:	Bolt hole cycle
X , Y	:	Enters based on the circle center coordinates G90/G91.
I	:	Circle radius. Must be a positive number. Can be specified in the least input increment.
J	:	Setting of the angle of the point to which the tool is positioned initially. Unit : 0.001° Counterclockwise is positive.
L	:	No. of divisions of the circle. Counterclockwise is positive.

(2) Sample program

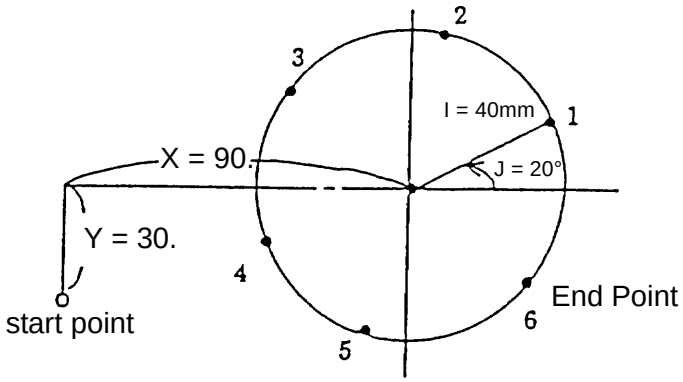
G54 G90 G00 X0. Y0. S1000 T15 ;	Move to the start point
G43. Z30. H14 ;	Z-axis approach
M03 ;	Spindle forward
G98 G81 Z-30. R3. F150 L0 ;	Canned cycle for drilling specified, No move.
G70 X90. Y30. I40. J20. L6 ;	Bolt hole cycle
G80 M0S ;	Canned cycle cancelled
M98 P98 ;	Migration to the subprogram 098.

(3) Description of the movements in the canned cycle

(a) G70 : Bolt hole cycle

G70 X_ Y_ I_ J_ L_ ;

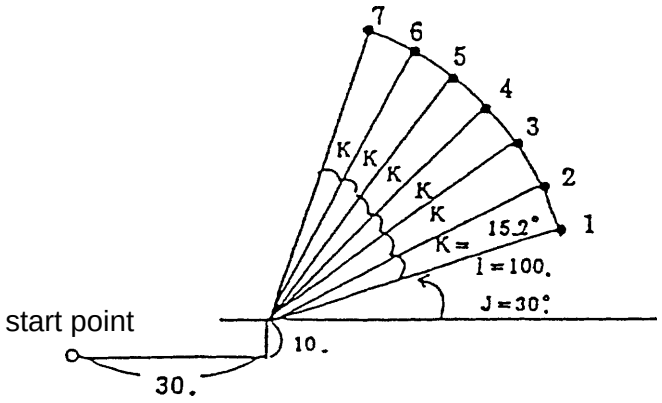
Example) G70 G91 X90. Y30. I40. J20. L6 ;



(b) G71; Arc cycle

G72 X_ Y_ I_ J_ K_ L_ ;

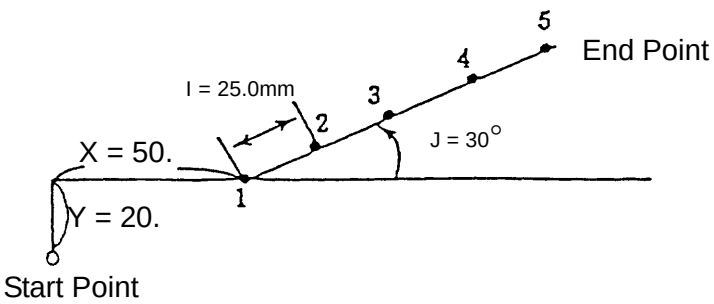
Example) G71 G91 X30. Y10. I100. J30. K15. 2 L7 ;



(c) G72 : Line at angle cycle

G71 X_ Y_ I_ J_ L_ ;

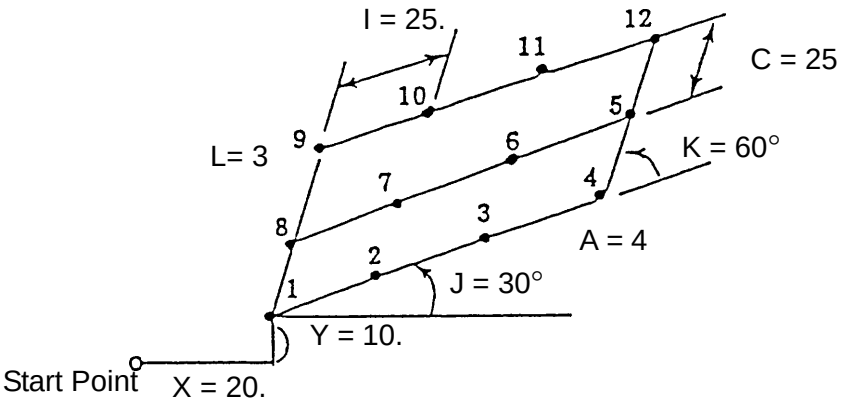
Example) G72 G91 X50. Y20. I25. J30. L5.;



(d) G77 : Grid cycle

G77 X_ Y_ I_ J_ K_ C_ A_ L_ ;

Example) G77 G91 X20. Y10. I25. J30. K60. C25. A4 L3 ;



(4) Cautions

(a) G70, G71, G72 and G77 are non-made G codes.

(b) Be sure to specify G70, G71, G72 and G77 in the canned cycle mode.

(c) Be sure to cancel G70, G71, G72 And G77 by specifying G80 in the next block to completion of machining.

(d) The plane in the drilling pattern cycle is decided in the canned cycle mode and becomes the same as the plane at the drilling position. G17- G19 must not be specified.

(5) Associated alarms

No. 134 Drilling Pattern Cycle command has an error.

(#000) L=0 has been specified, or L has not been specified.

(#001) I has not been specified in G70.

(#002) When L≠1 in G71, K or I has not been specified.

(#003) When L=1 in G71, I has not been specified.

(#004) In G72, I has not been specified.

(#005) In G77, A=0 has been specified, or A has not been specified.

(#006) In G77, K, C, or I has not been specified.

(#007) In G80 mode, G70, G71, G72, or G77 has been specified.

10-4 True Circular Cutting (G302 ~ G305)

In one block, you can specify a series of actions to cut the inside or outside of a true circle.

(1) G code

G302 : True circular cutting inside CW (clockwise)

G303 : True circular cutting inside CCW (counterclockwise)

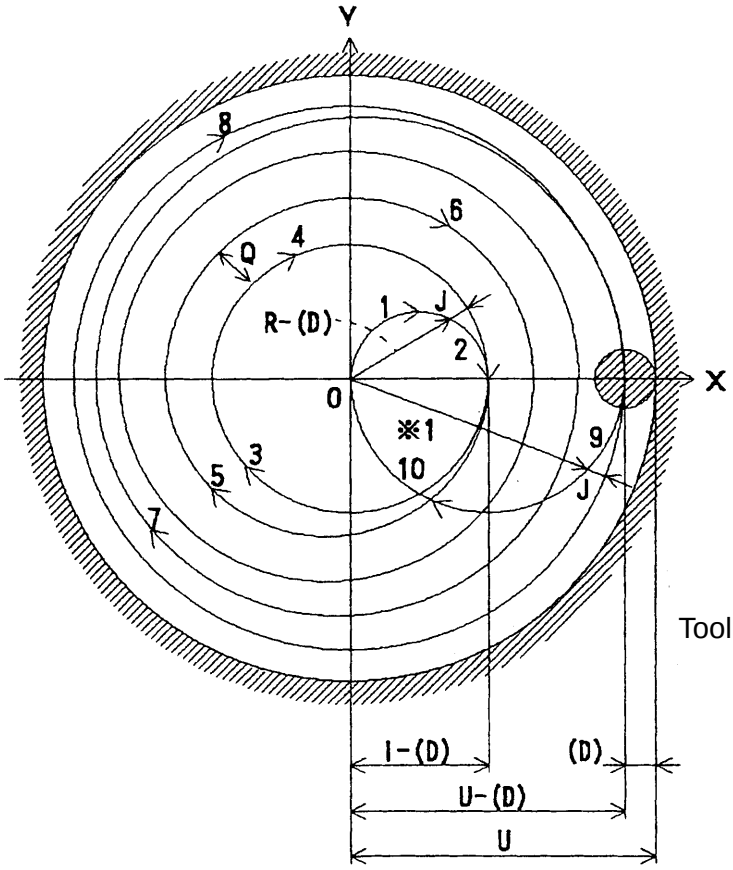
C304 : True circular cutting inside CW (clockwise)

G305 : True circular cutting inside CCW (counterclockwise)

(2) Command format

(a) True circular cutting ID (G302, G303)

$$\begin{Bmatrix} G302 \\ G303 \end{Bmatrix} I_ \begin{Bmatrix} R_ \\ J_ \end{Bmatrix} U_\ Q_\ L_\ D_\ F_\ ;$$



Tool center path
: 0→1→2→3→4→
5→6→7→8→9→
10→0

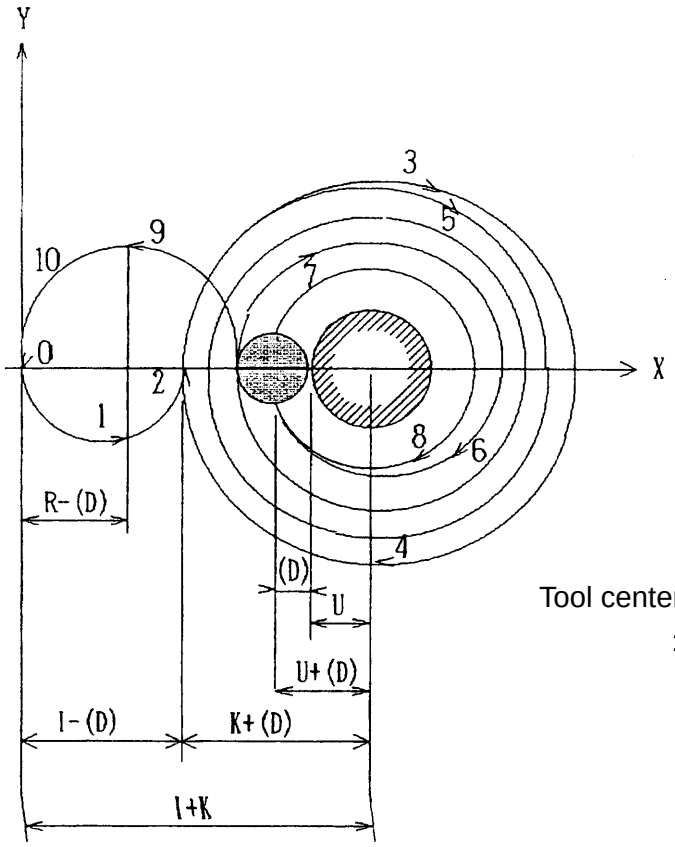
*1=R - (D) + U -I

where;
- I : Radius of the finished circle.
  - I + = Approach in the plus direction,
  - I - = Approach in the minus direction
- R : R designation for the high-speed feed section
- J : J designation for the high-speed feed section
- U : Radius of the final finished circle in spiral true circular cutting
- Q : Arc increment in spiral true circular cutting
- L : No. of repeats for the true circular area
- D : Tool diameter compensation number
- F : Cutting feed rate

(Example) G302 Spiral true circular cutting

(b) True circular cutting 0D (G304,G305)

$$\left\{\begin{matrix}G304\\G305\end{matrix}\right\} I_ \quad \left\{\begin{matrix}R_\\J_\end{matrix}\right\} K_\ U_\ Q_\ L_\ D_\ F_\ ;$$



Tool center path
: 0→1→2→3→4
5→6→7→8→9→0

where;
- I : Diameter of the approach circle.
  I + = Approach in the plus direction,
  I - = Approach in the minus direction
- R : R designation for the high-speed feed section
- J : J designation for the high-speed feed section
- K : Radius of the finished circle
- U : Radius of the final finished circle in spiral true circular cutting
- Q : Arc decrement in spiral true circular cutting
- L : No. of repeats for the true circular area
- D : Tool diameter compensation number
- F : Cutting feed rate

(3) True circular cutting plane

Specify the true circular cutting plane with G17,G18, or G19.

G17 : X_P-Y_P plane

G18 : Z_P-X_P plane

G19 : Y_P-Z_P plane

where; X_P : X axis or its parallel axis

Y_P : Y axis or its parallel axis

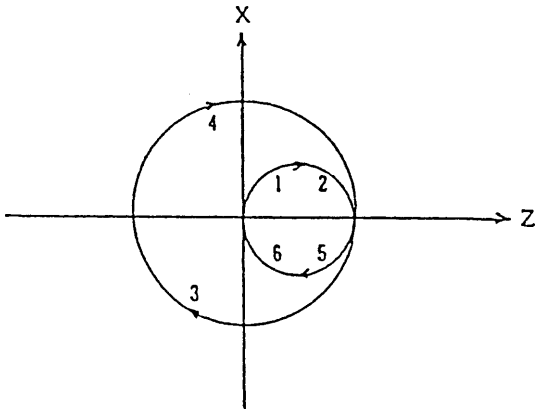
Z_P : Z axis or its parallel axis

(Note) It is impossible to specify the U axis for the true circular cutting plane.

(Example)

G18: Z-X plane

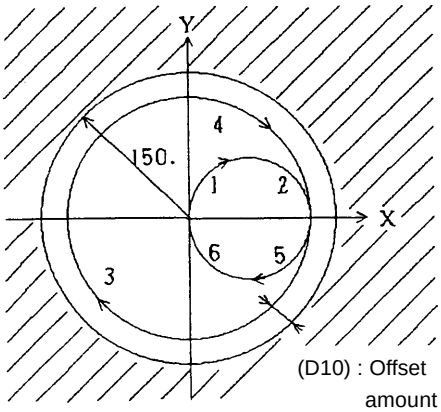
G302 I50. D10 F500 ; True circular cutting inside CW

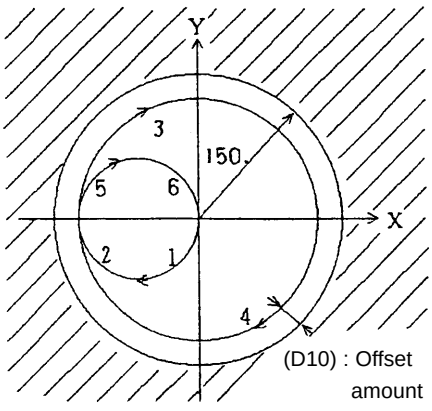


(4) Sample program

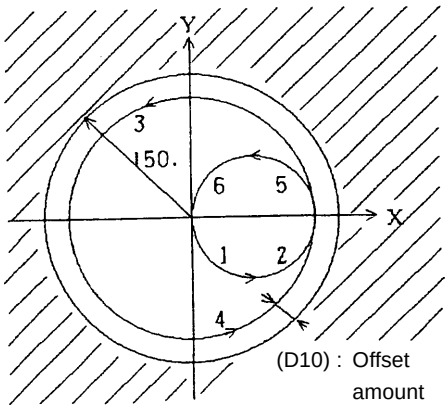
(a) Basic format

G302 I50. D10 F500 ; G302 I-50. D10 F500 ;

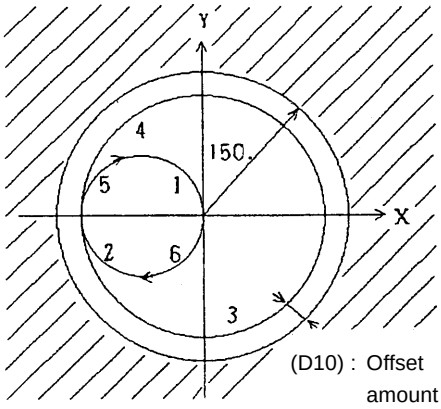




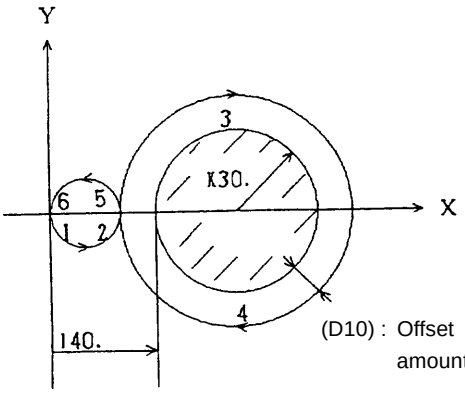
G303 I50. D10 F500 ;



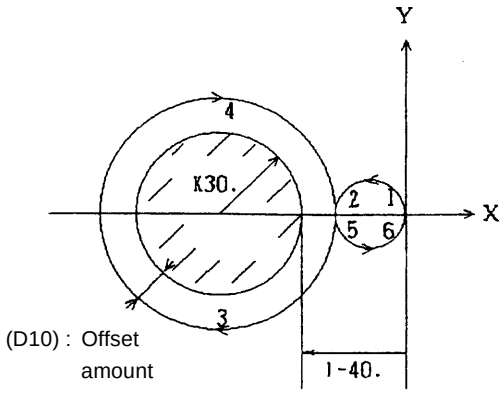
G303 I-50. D10 F500;



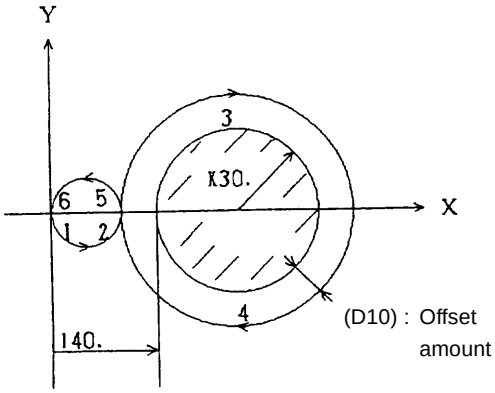
G304 I40. K30. D10 F500 ;



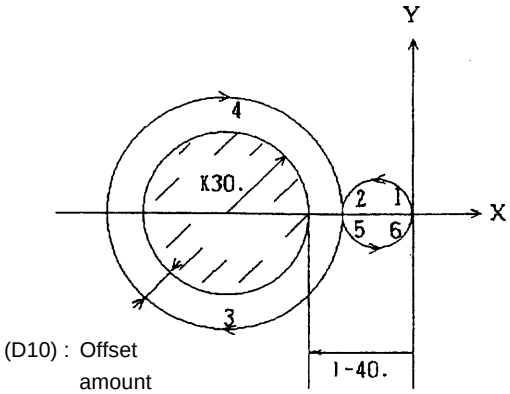
G304 I-40. K30. D10 F500;



G305 I40. K30. D10 F500 ;

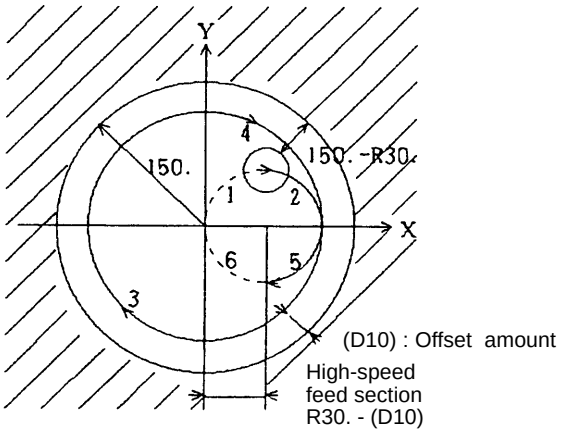


G305 I-40. K30. D10 F500;

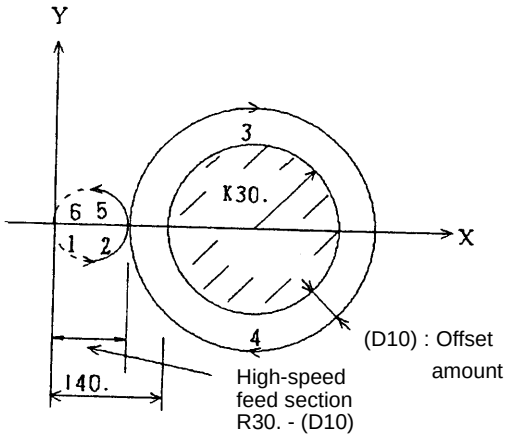


(b) R designation for the high-speed feed section

G302 I50. R30 D10 F500 ;

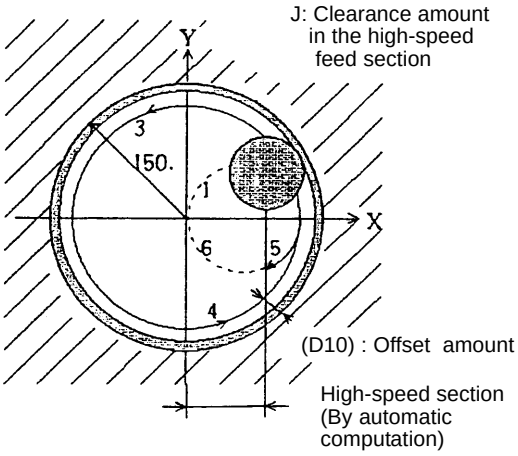


G304 I40. R30. K30. D10 F500;

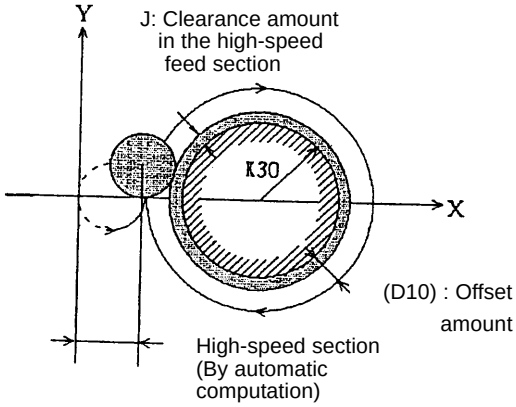


(c) J designation for the high-speed feed section

G302 I50. J5 D10 F500 ;

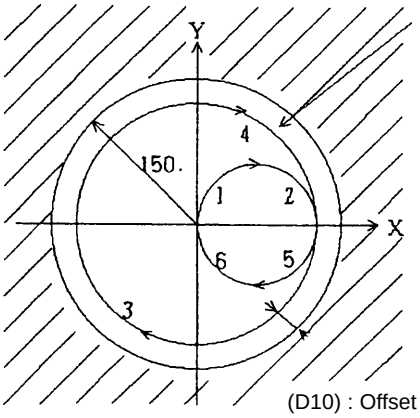


G304 I40. J5. K30. D10 F500;



(d) Designation of repeat for the true circular area (L)

G302 I50. L2 D10 F500 ;

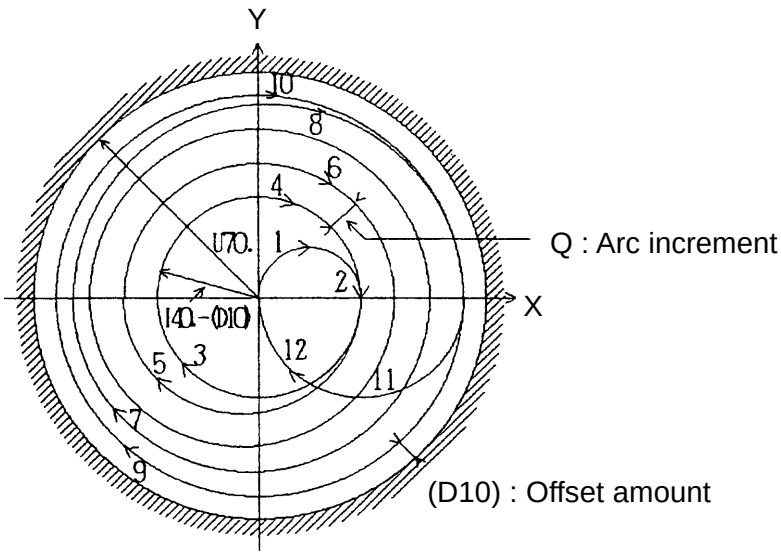


Repeats the finished circle twice.

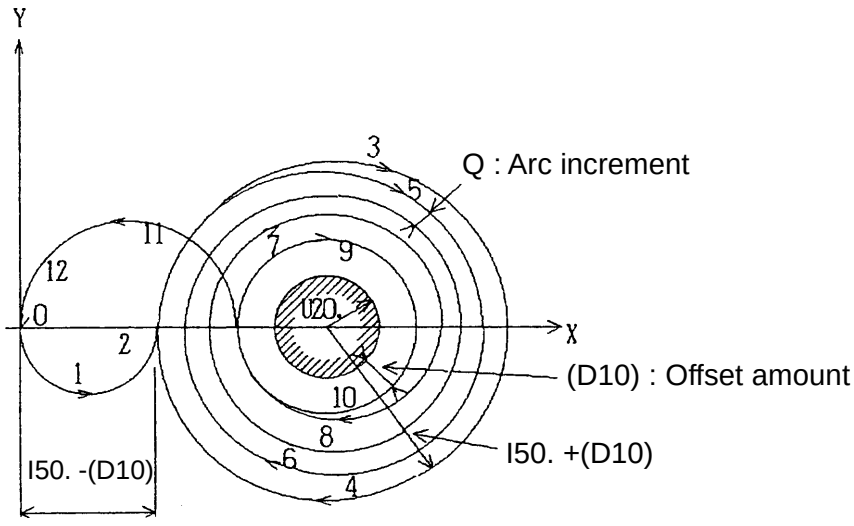
(Note) The final finished circle is repeated in case of spiral true circular cutting.

(e) Designation of spiral true circular cutting (U, Q)

G302 I40. U70. Q10. D10 F200 ;



G304 I50. K50. U20 Q10. D10 F200 ;



(5) Cautions

(a) Specify the G302 ~ G305 commands in the tool diameter compensation cancel mode (G40).

(b) G302 through G305 are non-modal G codes
The numerical values of the addresses other than D and F specified in the same block are valid only in the block where they are specified.

(c) The numerical values of R, J, K, U and Q should be always specified in positive values.

(d) When R and J are omitted, the high-speed feed section is invalidated.

(e) When U and Q are omitted, spiral true circular cutting is invalidated.

(f) When D and F are omitted, already specified D and F are validated.

(g) When D and F are omitted, the number of repeats for the true circulate area is always 1.

(h) When R and J are specified in the same block, J is given priority.

(i) The following commands result in an alarm.

(i) When the "radius of the finished circle (I) - offset amount" is 0 or negative in the G302/G303 command.

(ii) When the high-speed feed section cannot be obtained in R designation of the high-speed feed section.

(iii) When the high-speed feed section cannot be obtained by automatic computation in J designation of the high-speed feed section.

(iv) When the "radius of the finished circle (U) - offset amount" is 0 or negative in spiral cutting by the G302/G303 command.

(v) When the "diameter of approach (I) - offset amount" is 0 or negative in the G302/G303 command.

(vi) When there is no more approach circle existing after changing the offset amount.

(j) By parameter setting, the X-Y plane can be selected regardless of the G17 ~ G19 commands.

(k) When a negative value is set for the offset amount, it is converted into a positive value.

(l) For the R specification in the high-speed feed section, note that cutting occurs while moving in the high-speed section depending on the value of R since it is impossible to specify the cutting width.

(6) Associated parameters

No.5101, #0 = 0	The true circular cutting plane is always the X-Y plane.
No.5159	True circular cutting speed in the high-speed feed section

(7) Associated alarms

No.135	True circular cutting error
(#002)	I command not available (G302 ~ G305)
(#003)	K command not available (G304, G305)
(#004)	U command not available (G304, G305)
(#005)	I command = 0 (G304,305)
(#006)	K command = 0 (G304,305)
(#007)	U command = 0 (G304,305)
(#008)	Q command = 0 (G302~305)
(#009)	Tool diameter ≧ | I command | (G302 ~ 305)
(#012)	| R command | > | I command | (G302 ~ G305)
(#013)	| I command | - | J command | ≦ tool diameter (G304, G305)
(#014)	| J command | > | I command | (G302 ~ G305)
(#015)	| U command | > | K command | (G304, G305)

10-5 Square Outside Cutting (G322,323)

A series of square outside cutting actions can be specified in one block.

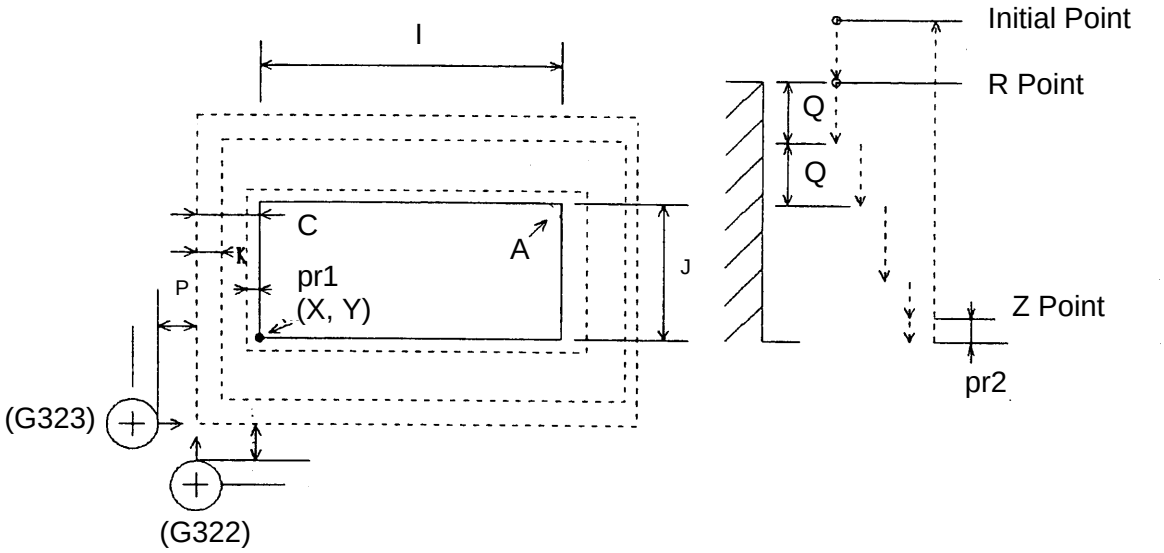
(1) G code

G322 : Square outside cutting CW (clockwise)

G232 : Square outside cutting CCW (counterclockwise)

(2) Command format

$\left\{\begin{matrix}G322\\G323\end{matrix}\right\}$ X_ Y_ Z_ R_ Q_ I_ J_ K_ P_ A_ C_ D_ F_ ;



(X, Y) : X-axis and Y-axis reference points

Z : Z point

R : R point

Q : Each depth of cut of the Z axis

I : X-axis length of the finished square

J : Y-axis length of the finished square

K : Each cutting width of the X and U axes

P : Approach amount

A : Cornet R radius

C : Removal amount

D : Tool offset number

F : Cutting feed rate

pr1 : Finish allowance (parameter setting)

pr2 : Clearance amount (parameter setting)

(Note 1) When cutting the finish allowance, an override is applied to the cutting feed rate at the rate set with the parameter.

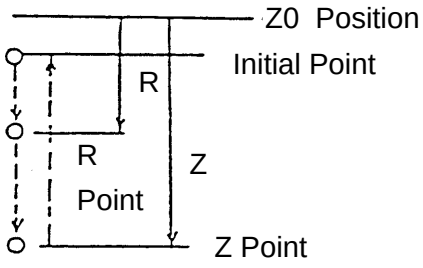
(3) Initial point

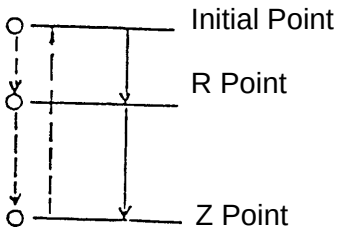
Machining start point of the G322/G323 command. When a series of actions is completed, the X, Y and Z axes return to the start point.

(4) R point and Z point

The R and Z points are as follows by the G90/G91 command.

[G90] [G91]





(5) Square outside cutting plane

The square outside cutting plane is determined by the G17/G18/G19 command

G17 G_ X_ Y_ Z_ R_ Q_ I_ .... ;
X-Y Plane, Cutting Axis

G17 G_ Z_ X_ Y_ R_ Q_ I_ .... ;
Z-X Plane, Cutting Axis

G17 G_ Y_ Z_ X_ R_ Q_ I_ .... ;
Y-Z Plane, Cutting Axis

(Note 1) You cannot select the plane including the additional axis.

(Note 2) By parameter setting, you can select the X-Y plane regardless of the G17 ~ G19 commands.

(Note 3) With the following command, when the program has the same parallel axis as the basic axis, specify G17 ~ G19 in another block.

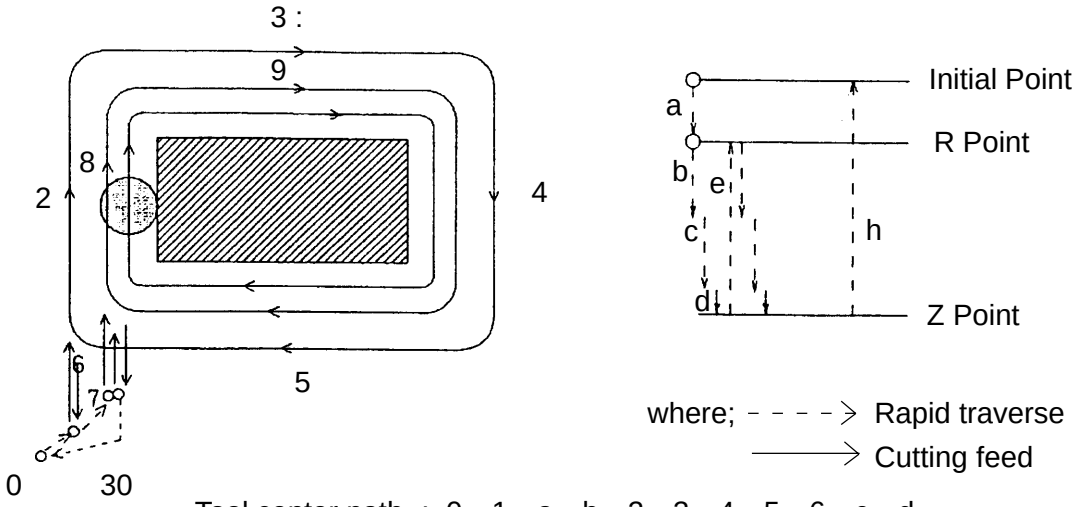
{G17 / G18 / G19} {G322 / G323} .....;

(6) Sample program

G17:

G90 G322 X50. Y-100. Z-50. R-10. Q20. I80.

J40. K8. P30. A2. C15. D10 F200 ;



Tool center path : 0→1→a→b→2→3→4→5→6→c→d→
2→3→4→5→6→e→7- - - - - - - - -
h→30→0

(7) Cautions

(a) G322/G323 applies tool diameter compensation regardless of the tool diameter compensation commands (G41, G42). Therefore, specify them in the tool diameter compensation cancel (G40) mode.

(b) G322 and G323 are Non-modal G codes.

(c) When A is omitted in the G322/G323 block, corner R is invalidated.

(d) When the numerical value of K is negative in the G322/G323 block, finishing is invalidated, and when the numerical value of Q is negative, clearance amount is invalidated.

(e) Specify all the numerical values of the addresses I, J, P, A, C, etc. in positive values. A negative value results in an alarm.

(f) An alarm results when the tool radius is larger than the approach amount. (P<(D))

(g) When D and F are omitted, already specified D and F are validated.

(8) Associated parameters

No.5101, #0= 0 The square outside cutting plane is always the X-Y plane.

1 The square outside cutting plane depends on the specified one of the G17~ G19 commands.

No.5115 Finish speed override value (1 to 100 %)

No.5152 Finish allowance

No.5153 Clearance amount

(9) Associated alarms

No.136 Square cutting command error

(#002) I command not available (G322~G326)

(#003) J command not available (G322~G326)

(#004) K command not available (G322~G326)

(#005) P command not available (G322~G326)

(#006) Q command not available (G322~G326)

(#010) I command = 0 (G324~G326)

(#011) J command = 0 (G324~G326)

(#012) K command = 0 (G324~G326)

(#013) Q command = 0 (G324~G326)

(#014) I command ≦ 0 (G322,G323)

(#015) J command ≦ 0 (G322,G323)

(#017) P command < 0 (G322,G323)

(#018) A command < 0 (G322,G323)

(#019) C command < 0 (G322,G323)

(#020) C command < | K command | (G322, G323)

(#021) A command > [I command/2] (G322, G323)

(#022) A command > [J command/2] (G322, G323)

(#023) | K command | > | J command | (G324~G326)

(#024) Finish allowance> | J command | - | K command | (G325, G326)

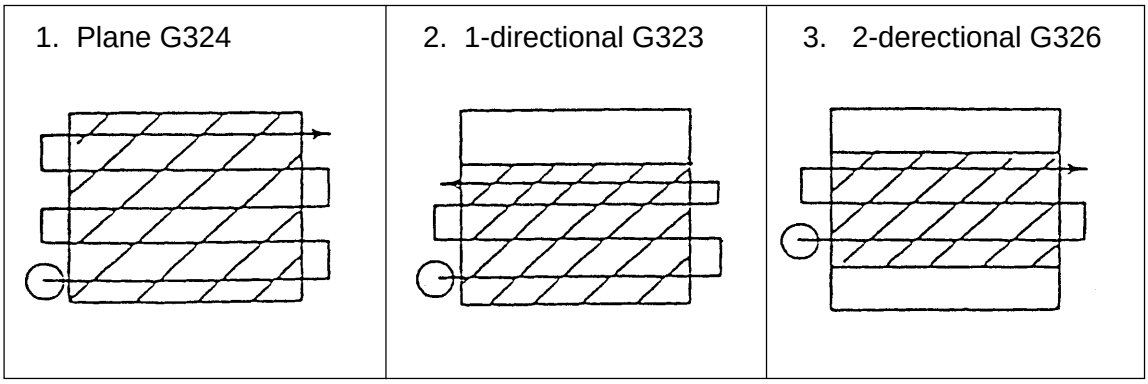
(#025) C command is not 1~4. (G325, G326)

10-6 Plane Cutting Cycle (G324, G325, G326)

There are 3 kinds of canned circle for plane cutting ; square plane cutting (G324), square plane 1-directional (G325) and square plane 2-directioal(G326)

These canned cycles are convenient when cutting the plane or groove, using a face mill or an end mill.

In these cycles, the work piece is cyclically machined by the depth of cut (Q) of one plane, assuming the coordinate value specified with X1 and Y2 as the start point, and the range limited by the length in the spindle direction(I) and that in the countershaft direction(J) as the work plane.



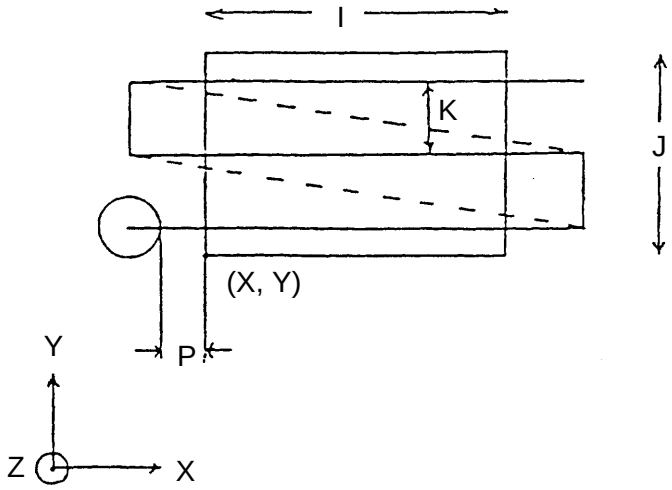
(1) Square plane cutting (G324)

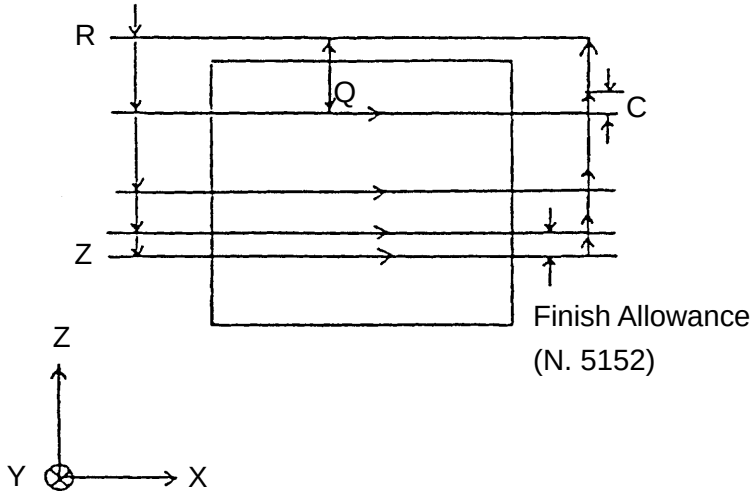
[Purpose] This function collectively machines the square plane by single/double directional cutting.

(a) command format

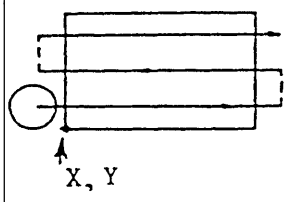
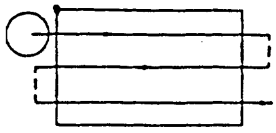
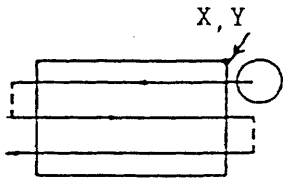
G324 X_ Y_ Z_ R_ I_ J_ K_ Q_ P_ C_ D_ E_ U_ F_;

G324	: Square plane cutting
X, Y	: Start point coordinate value of the plane. Enter it based on G90/G91
Z	: Z axis coordinate value of the finish surface. Enter it based on G90/G91.
R	: R point coordinate value in the 3rd axis
I	: Length in X axis direction
J	: Length in Y axis direction
K	: Plane cutting width (cutting method specified with the sigh "+/-")
Q	: Each depth of cut in the Z axis ("-" : No finishing)
P	: Approach amount in the Z axis direction (absolute value)
C	: Relief amount in the Z axis direction (absolute value)
D	: Tool diameter compensation number (radius registration)
E	: Finish feed rate (Default feed rate : F) (mm/min.)
U	: Finish spindle speed (Default spindle speed : S) (rpm)
F	: Cutting feed rate (F specified before default) (mm/min.)





(b) Movements

			
I = ⊕ J = ⊕	I = ⊖ J = ⊕	I = ⊕ J = ⊖	I = ⊖ J = ⊖

The start point and cutting direction can be changed by changing the sign of I and J. When the cutting width K is a negative value, the cutter center is projected outside by the approach amount

I

J

Y P

Approach Amount

P

Approach Amount

Z X

R

1st Cut + Q

2nd Cut + Q

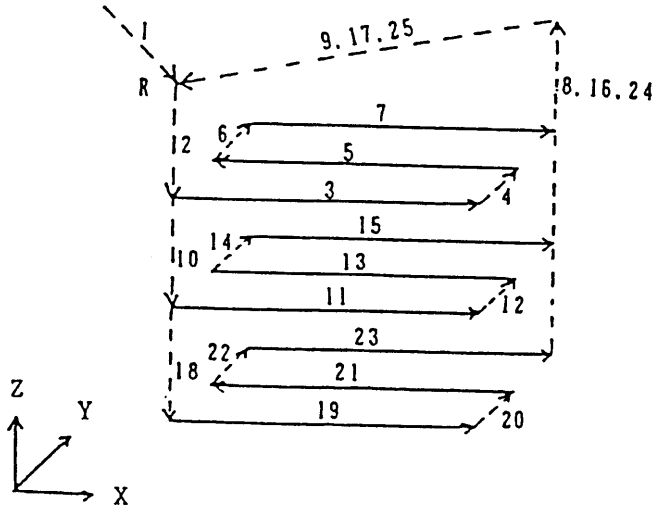
3rd Cut (Z - Finish Allowance)

Z

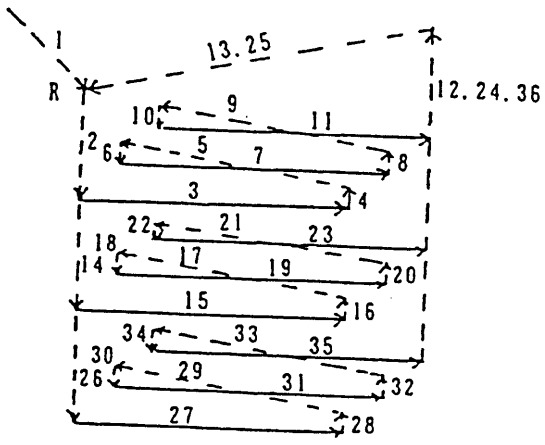
Y X

Full line : Cutting feed

Dotted line : Rapid traverse

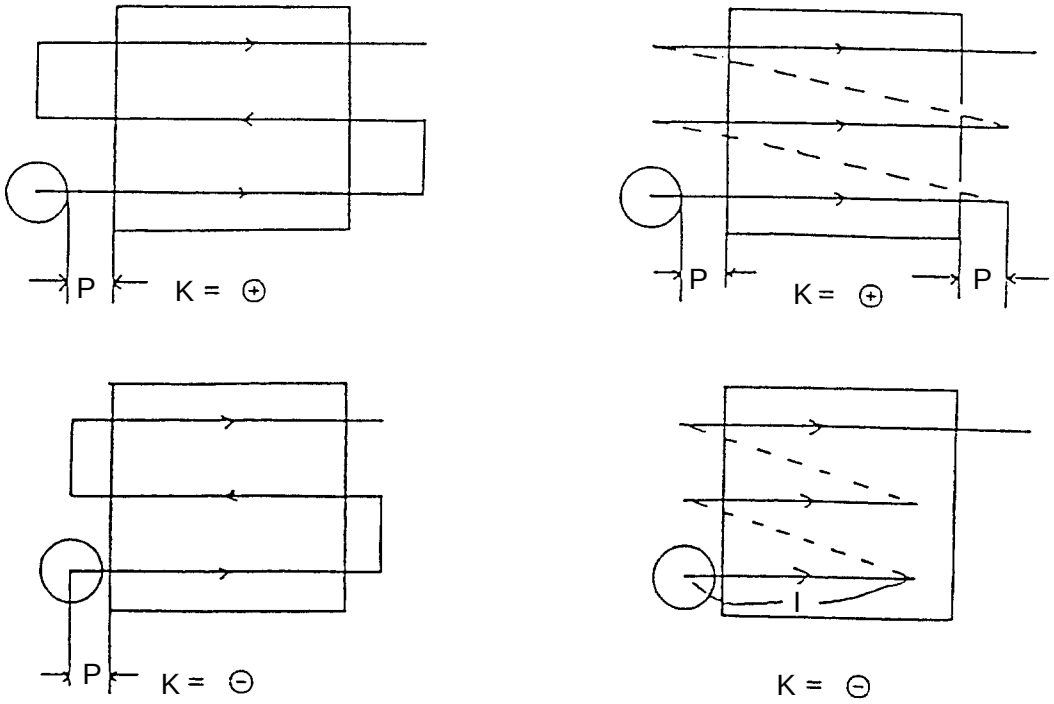


Double Directional Cutting (|Q| ≧ C)



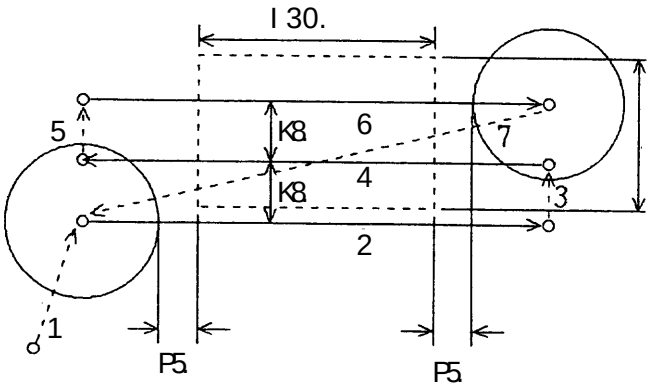
Single Directional Cutting (|Q| < C)

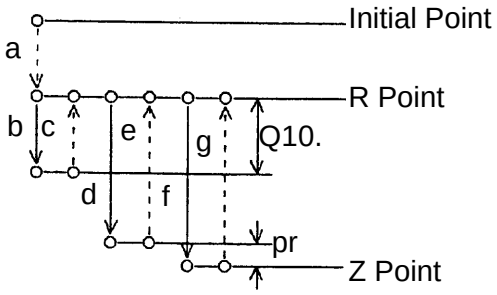
Double Directional Cutting (|Q| ≧ C) Single Directional Cutting (|Q| < C)



(c) Sample program

G324 X-15. Y-10. Z-30. R-10. 130. J20. K8. Q10. P5. P10 F200





⟶ Cutting feed

- - > Rapid traverse

Tool center path 1→a→b→2→3→4→5→6→c→7
→d→2→3→4→5→6→e→7
→f→2→3→4→5→6→g→7

(2) Square plane l-directional (G325)

[purpose] Capable of performing multi-directional cutting and specifying the end surface.

(a) Command format

G325 X_ Y_ Z_ R_ I_ J_ K_ Q_ P_ C_ D_ E_ U_ F_ ;

G325 : Square plane l-directional

X,Y : Start point coordinate value of the plane.
Enter it based on G90/G91.

Z : Z axis coordinate value of the finish surface.
Enter it based on G90/G91.

R : R point coordinate value in the Z axis direction.

I : Length in the I axis direction

J : Length in the J axis direction

K : Plane cutting width (cutting method specified with the sigh "+/-")

Q : Each depth of cut in the Z axis direction ("-" : NO finishing)

P : Approach amount (absolute value)

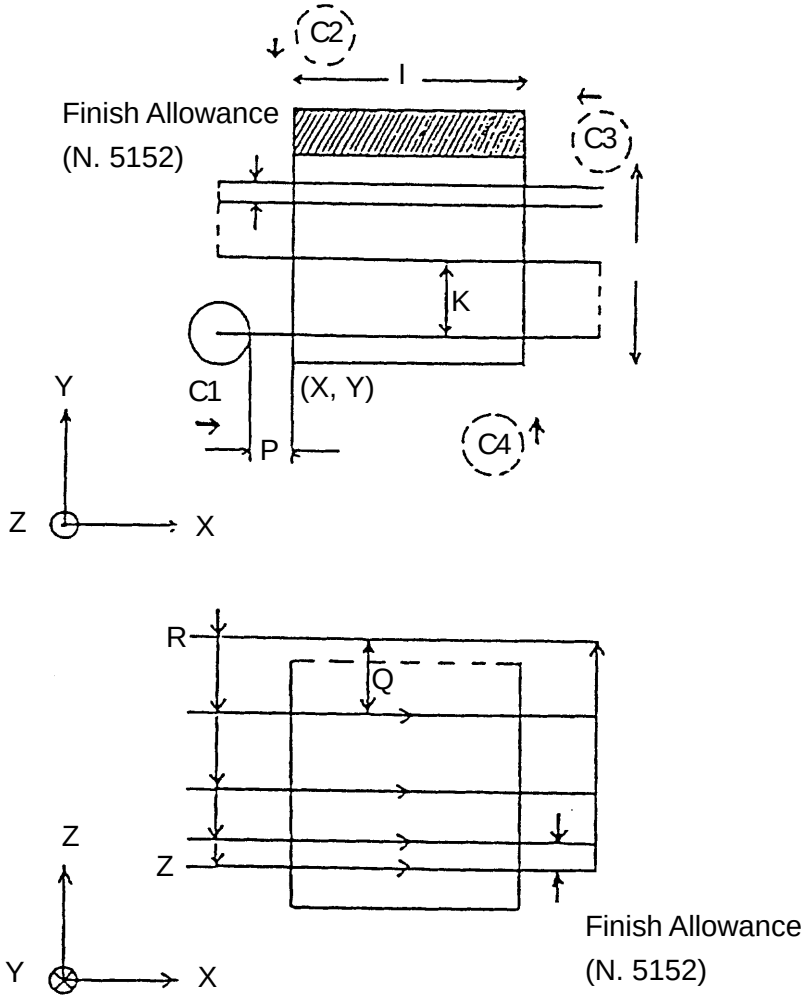
C : Designation of the cutting direction ($1 \leqq C \leqq 4$)

D : Tool diameter compensation number (radius registration)

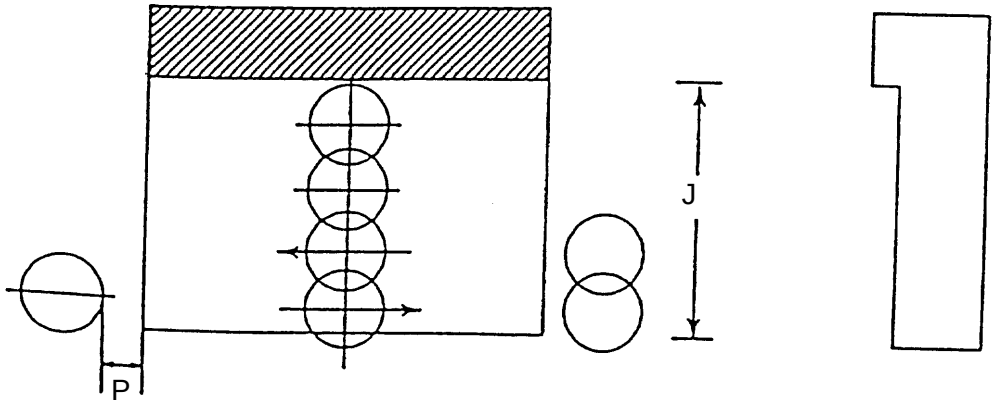
E : Finish feed rate (Default feed rate :F) (mm/min.)

U : Finish spindle speed (Default feed rate : S) (rpm)

F : Cutting feed rate (F specified before defaulted) (mm/min.)

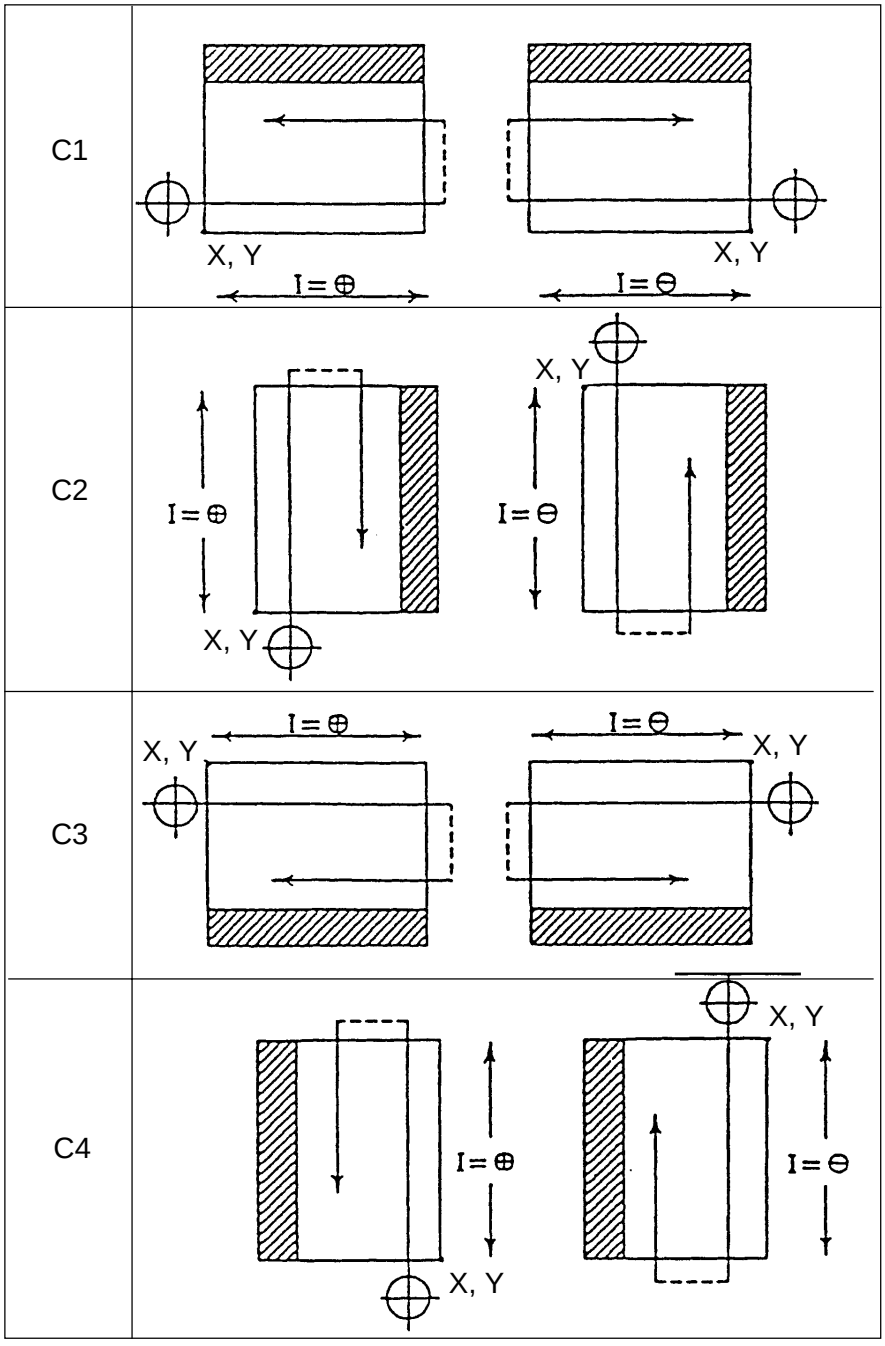


(b) Movement



Approach Amount

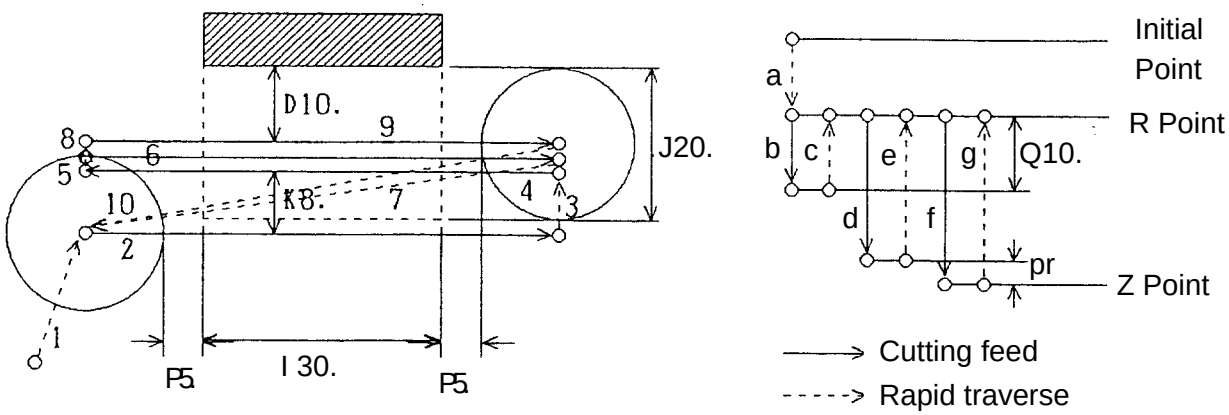
1) X,Y approach point, rapid traverse to the R point
↓
2) Rapid traverse to the Z axis cut-in height
↓
3) Machining in the I-specified axis direction
↓
4) Machining in the J-specified axis direction, K ="+" : rapid
↓ traverse, K = "-" : machining
5) I-J plane, 3) and 4) repeated until the end of machining
↓
6) Return to the R point by rapid traverse
↓
7) 1) through 6) repeated as far as the Z axis coordinate value. Bottom and plane finishing in the final step.



The start point and cutting direction are specified with the sign of C and I.

(C) Sample program

G324 X-15. Y-10. Z-30. R-10. I30. J20. K8. Q10. P5. C1 D10 F200



Tool center path 1 →a→b→2→3→4→5→6→c→7
→d→2→3→4→5→6→e→7
→f→2→3→4→8→9→g→10

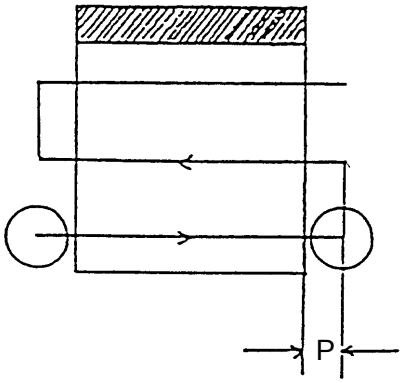
(d) Cautions

(d. 1) When each depth of cut(Q) is a negative value(-), no finishing is performed. (plane included) Machining is performed only once in case of (|R - Z|) ≦ |Q|.

(d. 2) The tool cuts in by each depth of cut(Q) from the R point.

(d. 3) The length in the J direction is always positive regardless of the sign "+/-".

(d. 4) When K is a negative value, the cutter center is projected by the approach amount. The cutting width is cut at the cutting feed rate.



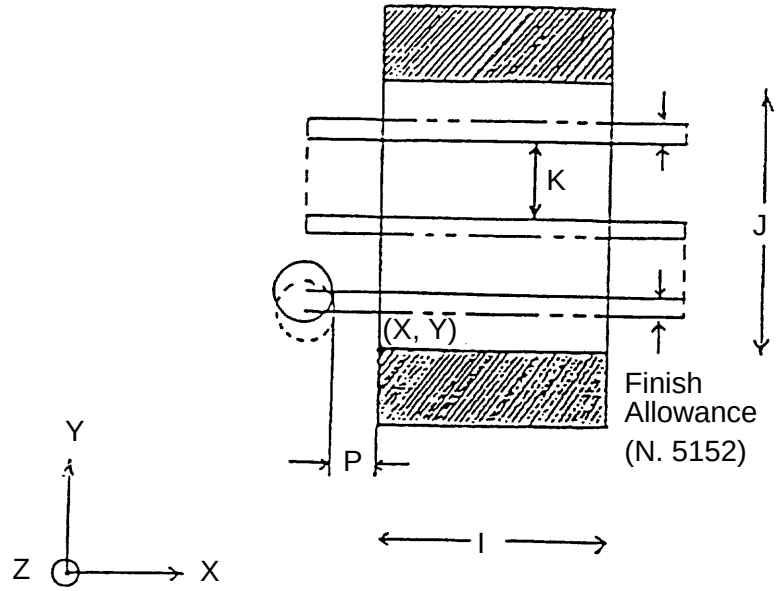
(3) Square plane 2-directional

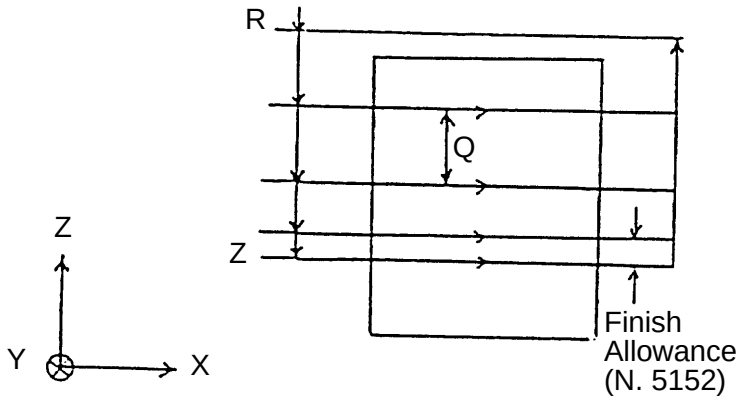
[Purpose] There are both sides and this is used for inside plane cutting.

(a) Command format

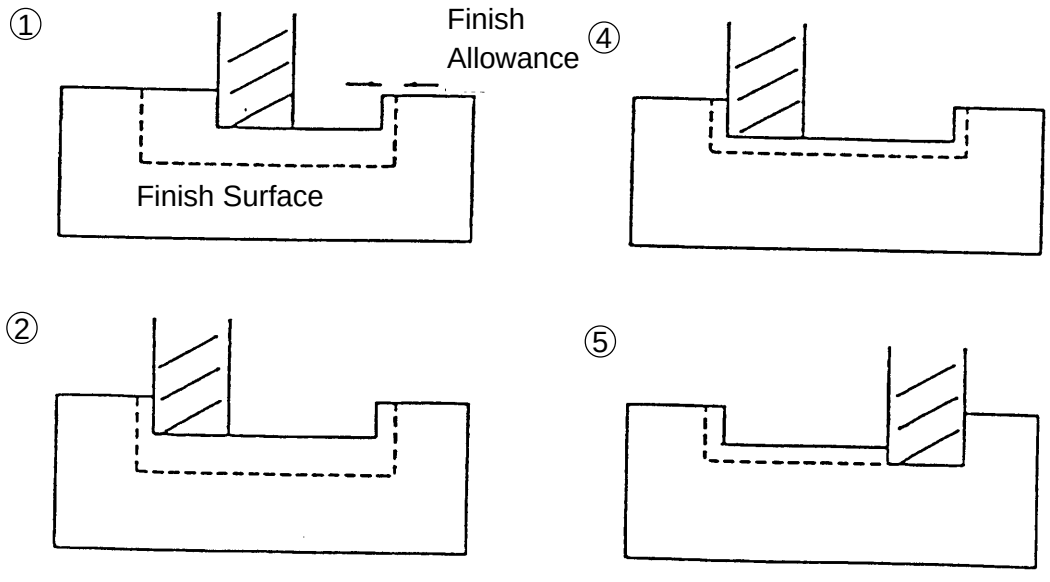
G326 X_ Y_ Z_ R_ I_ J_ K_ Q-P_ C_ D_ E_ U_ F_ ;

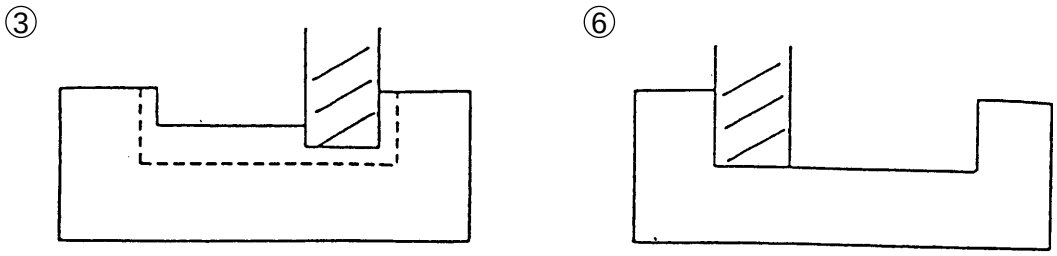
G326 : Square plane 2-directional
X, Y : Start point coordinate value of the plane.
Enter it based on G90/G91.
Z : Z axis corrdinate value of the finish surface.
Enter it based on G90/G91.
R : R point coordinate value in the Z axis direction.
I : Length in the I direction
J : Length in the J direction
K : Plane cutting width (cutting method specified with the sigh "+/-")
Q : Each depth of cut in the Z axis direction
("-" : No finishing)
P : Approach amount (absolute value)
C : Designation of the cutting direction (1 = or 2)
D : Tool diameter compensation number (radius registration)
E : Finish feed rate (Default feed rate: F) (mm/min.)
U : Finish spindle speed (Default feed rate: S) (rpm)
F : Cutting feed rate (F specified before defaulted) (mm/min.)


K
J
(X, Y)
Finish
Allowance
(N. 5152)
P
I
Y
Z
X


R
Q
Z
Z
Finish
Allowance
(N. 5152)
Y
X

(b) Movements


①
Finish
Allowance
Finish Surface
④
②
⑤



① Cuts on the start point side, leaving the side finish allowance.

↓

② Leaves the side finish allowance on the end poing side.

↓

③ Cuts in on the start point side, from the position where the side finish allowance was left.

↓

③ Leaves the side finish allowance on the end point side.

↓

④ Finishes the side and bottom on the start point side.

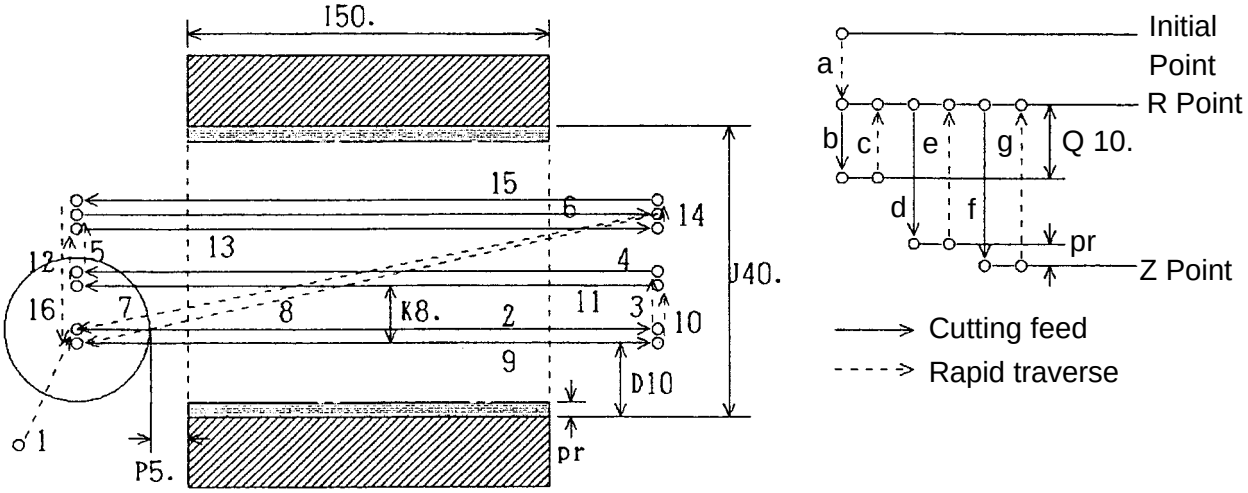
↓

⑤ Finishes the side and bottom on the end poing side.

↓

(c) Sample program

G326 X-15. Y-10. Z-30. R-10. 130. J20. K8. Q10. P5. C1 D10 F200



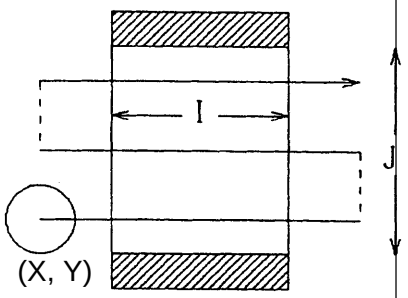
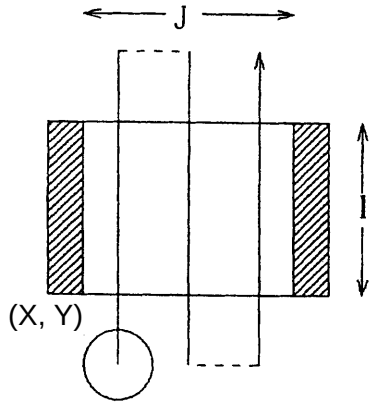
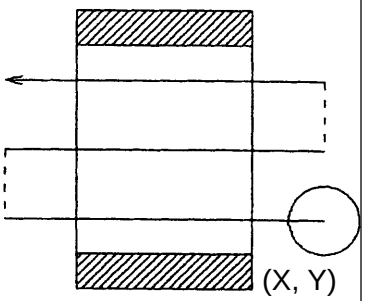
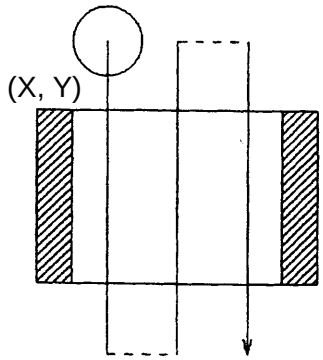
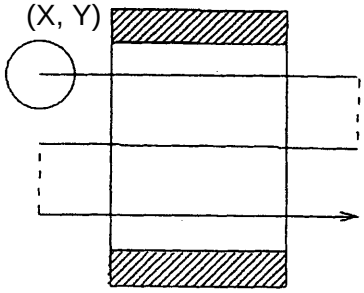
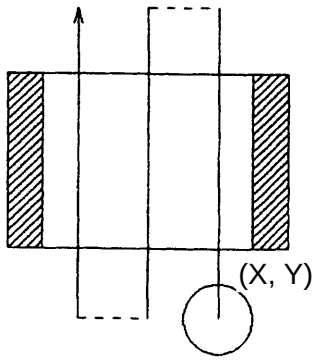
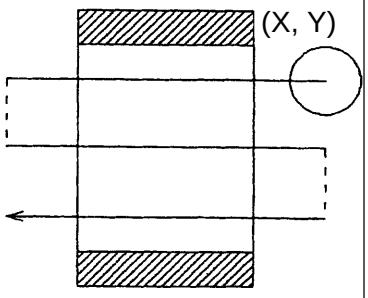
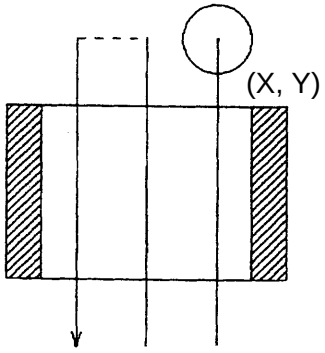
Tool center path 1→a→b→2→3→4→5→6→c→7
→d→2→3→4→5→6→e→8
→f→9→10→11→12→13→14→15→g→16

(d) Cautions

(d. 1) No finishing is performed in case of (|R - Z|) ≦ |Q|.

(d. 2) If each depth of cut (Q) is a negative value, no finishing is performed (plane included).

(d. 3) When the cutting width(K) is a negative value, the cutter center is prokected by the approach amount.

I J	C1	C2
+ -		
- +		
- +		
- -		

10-7 Poketing (G327 ~ G333)

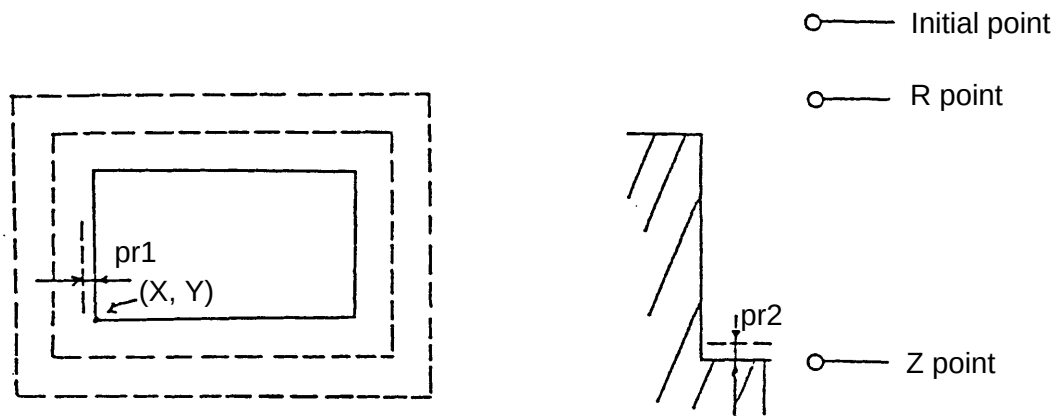
A series of inside/outside cutting actions for circle, track, square, etc. can be specified in one block.

(1) G code

G327 : Circle inside
G328 : Square inside
G329 : Track inside
G330 : Circle outside
G331 : Square outside
G332 : Track outside
G333 : Circle

(2) Command format

G_ X_ Y_ Z_ R_ Q_ I_ J_ K_ P_ A_ C_ U_ V_ W_ E_ D_ F_ ;



G_ : Mode
(X, Y) : Reference point of the X and Y axes
Z : Z point
R : R point
Q : Each depth of cut in the Z axis

I, J, K, P, A,
C, U, V, W, E } Refer to the description of each function.

D : Tool offset number
F : Cutting feed rate
pr1 : Finish allowance (parameter setting)
pr2 : Clearance amount (parameter setting)

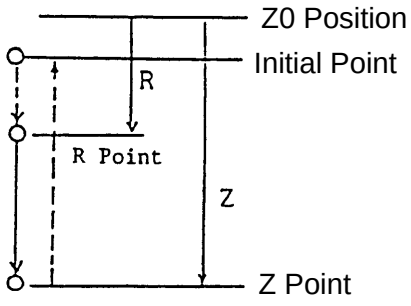
(3) Initial point

Machining start point of the G327 ~ G333 commands. All of the X, Y and Z axes return to the start point when a series of actions is completed.

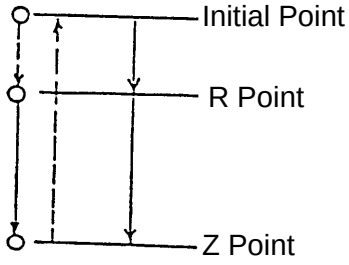
(4) R and Z points

The R and Z points are set as follows by the G90/G91 command.

[G90]



[G91]



(5) Pocketing plane

The pocketing plane is determined by G17, G18 or G19.

G17 G_ X_ Y_ Z_ R_ Q....;
X-Y Plane, Cutting Axis

G18 G_ Z_ X_ Y_ R_ Q....;
Z-X Plane, Cutting Axis

G19 G_ Y_ Z_ X_ R_ Q....;
Y-Z Plane, Cutting Axis

(Note 1) The plane including the additional axis cannot be selected.

(Note 2) By parameter setting, the X-Y plane can be selected regardless of theG17~G19 commands.

(Note 3) When specifying the program address of the same parallel axis as the basic axis in the block which contains

{ G17 / G18 / G19 } G_ ...; specify the G17/G18 command in another block.

(6) Cautions

(a) Give the G327 to G333 commands in the cutter compensation cancel (G40) mode.

(b) G327 through G333 are non-modal G codes.

(c) When D and F are omitted, already specified D and F are validated.

(Note 1) For detailed description of each function, refer to separate document.

(7) Associated parameters

No. 5101, #0 =	0	The X-Y plane is always selected as the pocketing plane.
	1	Selection of the pocketing plane follows the G17 ~ G19 command.
No. 5115		Finish speed override value (1 to 100%)
No. 5116		Corner speed override value (1 to 100%)
No. 5152		Finish allowance
No. 5153		Clearance amount

(8) Associated Alarms

No. 137 Pocket cutting command error

(#009) I command not available (G327~G333)
(#010) J command not available (G328~G332)
(#011) K command not available (G327~G333)
(#012) P command not available (G330~G332)
(#013) Q command not available (G327~G333)
(#014) A command not available (G329, G332)
(#015) Z command not available (G327, G328, G330, G331)
(#017) I command = 0 (G333)
(#018) K command = 0 (G327~G332)
(#019) Q command = 0 (G327~G332)
(#021) J command < 0 (G330)
(#022) C command < 0 (G329)
(#023) C command ≦ 0 (G332)
(#024) P command ≦ 0 (G332)
(#025) P command < 0 (G330, G331)
(#026) A command - tool diameter ≦ 0 (G328)
A command + tool diameter ≦ 0 (G331)
(#027) | I command | < tool diameter (G327)
(#028) I command < C command (G333)
(#029) I command < |K command | (G333)
(#030) | I comand | < finish allowance (G327)
(#031) [I command/2] < tool diameter (G328)
(#032) [I command/2] < finish allowance (G328)
(#033) [I command/2] < [A command - tool diameter] (G328)
[I command/2] < [A command + tool diameter] (G331)
(#035) | I command | - | J command | < [finish allowance x 2] (G327)
(#037) Finish allowance > [I command - tool diameter] (G333)
(#038) Tool diameter > [| I command | - finish allowance] (G327)
(#039) [(I command/2) - finish allowance]< tool compensation (G328)
(#040) Tool diameter > [I command - J command - (finish allowance x 2)] (G327)
(#042) Tool diameter > [J command/2] (G328)
(#043) Finish allowance > [J command/2] (G328)

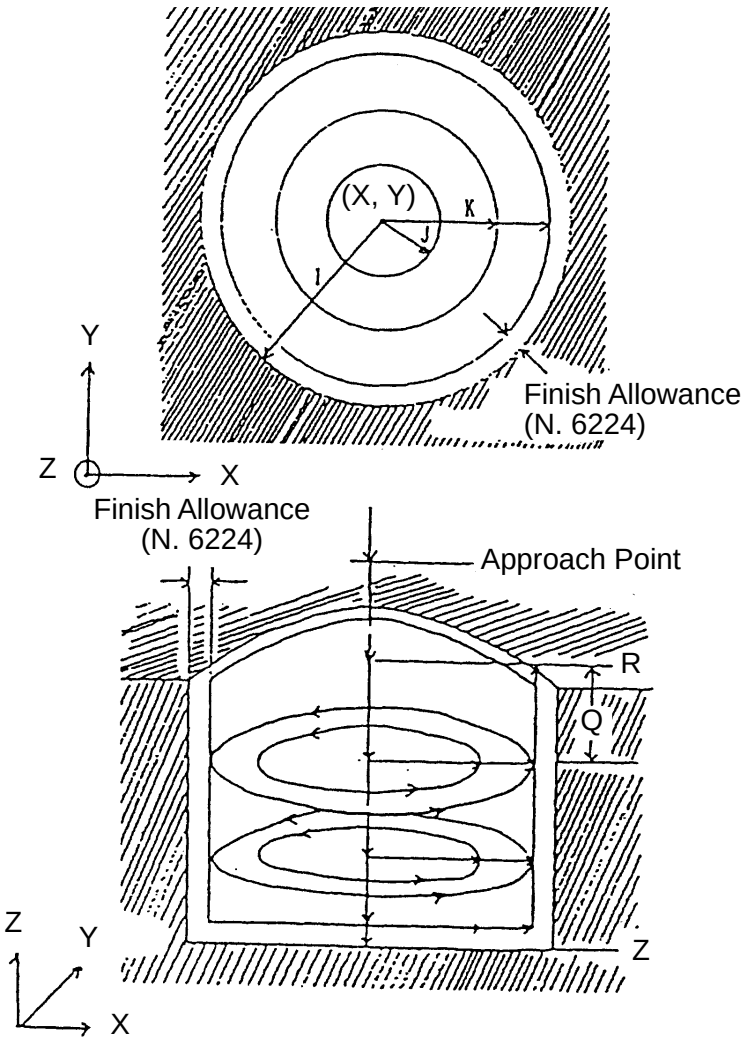
(#044)	[A command - tool diameter] > [J command/2] (G328)
	[A command + tool diameter] > [J command/2] (G331)
(#043)	Finish allowance > [J command/2] (G328)
(#044)	[A command - tool diameter] > [J command/2] (G328)
	[A command + tool diameter] > [J command/2] (G331)
(#045)	Tool diameter > [(J command/2) - finish allowance] (G328)
(#046)	Finish allowance > Q command (G329, G332, G333)
(#047)	Finish allowance > K command (G329, G332, G333)
(#048)	Finish allowance > C command (G329, G332)
(#049)	C command > | A command | (G329)
(#050)	C command < finish allowance (G328, G331)
(#051)	Tool diameter > [| A command | - finish allowance] (G329)
(#052)	| R command -Z command | < finish allowance (G327, G328)
(#053)	Start radius < finish radius (G330)
	Start width < finish width (G331)
(#055)	Track left and right sides have the same arc center. (G329, G3329)

10-7-1 Circular Poketing (G327)

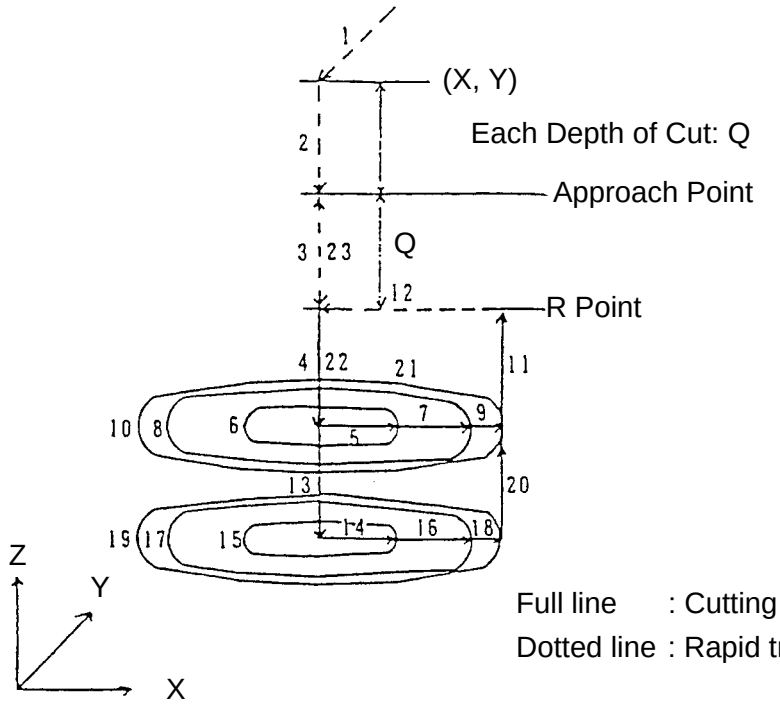
[Purpose] Used for pocketing inside the circle with an end mill.
For detailed description in case of G17 (X, Y plane), as follows.

(1) Command format
G327 X_ Y_ Z_ R_ I_ J_ K_ Q_ D_ E_ U_ V_ F_ ;

G327	:	Circular pocket
X, Y	:	Center coordinate value of the center. Enter it based on G90/G91.
Z	:	Coordinate value of the pocket finishing Z axis.. Enter it based on G90/G91.
R	:	R point coordinate value in the Z axis direction
I	:	Radius of the circle to be cut (cutting direction selected depending on the sign "+/-")
J	:	Radius of the premachined hole (cutting method specified by the sign "+/-")
K	:	Cutting width (no finishing in the side direction when "-" is given)
Q	:	Each depth of cut in the Z axis direction ("-": no finishing in the Z axis direction)
D	:	Tool diameter compensation number (radius registration)
E	:	Finish feed rate (Default feed rate: F) (mm/min.)
U	:	Finish spindle speed (Default feed rate: S) (rpm)
V	:	Z axis cut-in speed (Default: F specified before) (mm/min.)
F	:	Cutting feed rate (Default: F specified before) (mm/min.)



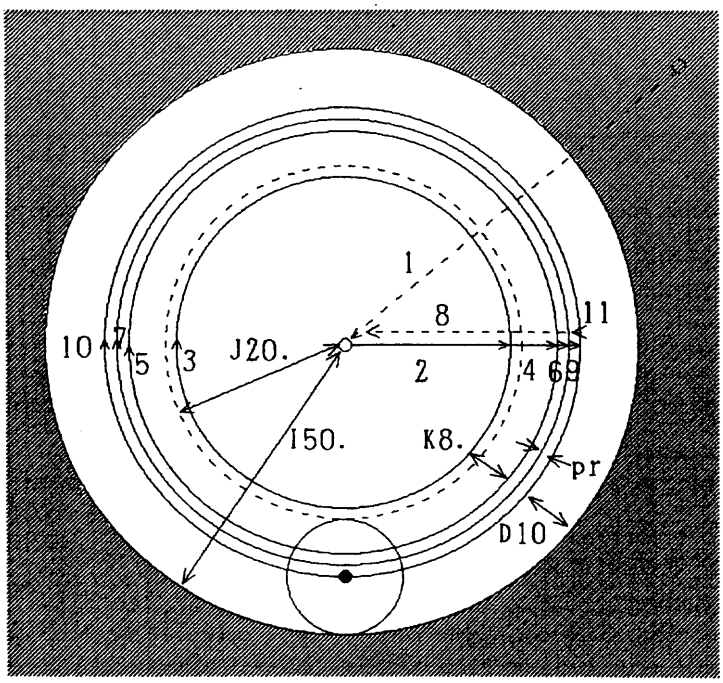
(2) Movements

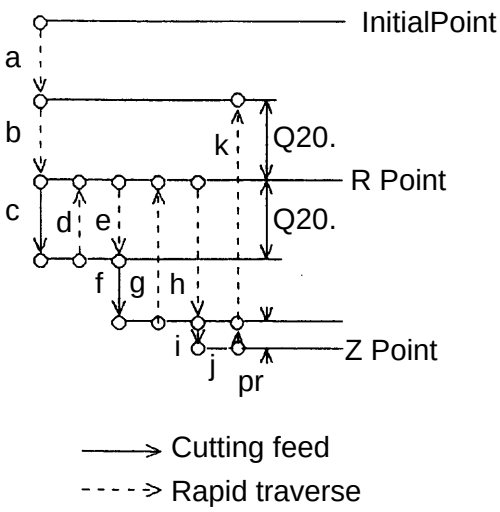


1. Moves to the X, Y point at the rapid traverse rate.
↓
2. Moves to the Z axis approach point at the rapid traverse
↓ (R Point + each depth of cut(Q) in the Z axis direction)
3. Rapid traverse to the R point
↓
4. Cuts in by each depth of cut Q in the Z axis direction at the cutting speed of V
↓ (when abbreviated, F)
5. Cuts the cutting width(K) at the cutting speed(F), considering the premachined hole(J) in X direction.
↓
6. Arc cutting Circular radius command: I “+” (clockwise rotation)
Circular radius command: I “-” (counterclockwise rotation)
↓
7 ~ 10. Repeats the steps 5 and 6.
↓
11. Retracts by each depth of cut Q in the Z axis direction at the cutting speed of V
↓ (when abbreviated, F)
12. Returns to the spindle center at the rapid traverse rate.
↓
13 ~ 21. Repeats the steps 4 through 12.
↓
o Performs finish cutting by the specified dimension in the X and Z axis direction.
↓
22. Returns to the R point at the rapid traverse rate.
↓
23. Returns to the approach point in the Z axis direction at the rapid traverse rate and terminates.

(3) Sample program

G327 X50. Y50. Z-50. R-10. I50. J20. K8. Q20. D10 F200





Tool center path 1→a→b→c→2→3→4→5→6→7→d→8
→e→f→2→3→4→5→6→7→g→8
→h→i→2→3→4→5→6→7→9→10→11→8
→j→k→8

(4) Cautions

(a) When the cutting width(K) is a negative value, finishing is not performed in the X axis and Y axis directions.
(Fig. 7.1)

(b) If the radius (J) of the lower hole is -/, a profile (Figure 7.2) leaving the radius (J) is obtained.

(c) When each depth of cut(Q) is a negative value in the Z axis direction, finishing is not performed in the Z axis direction.

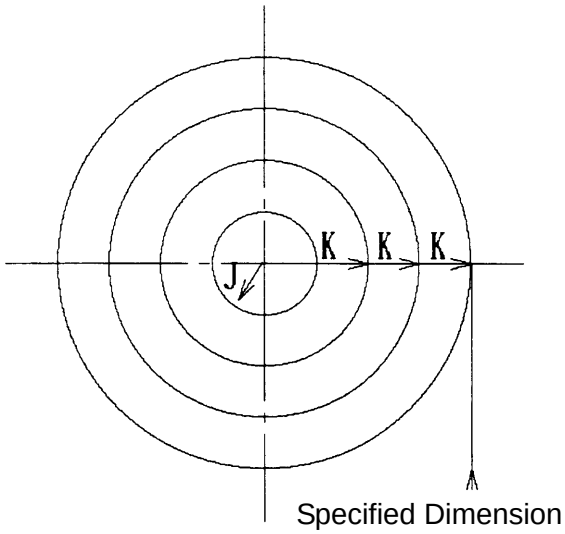


Fig. 7.1 K-**

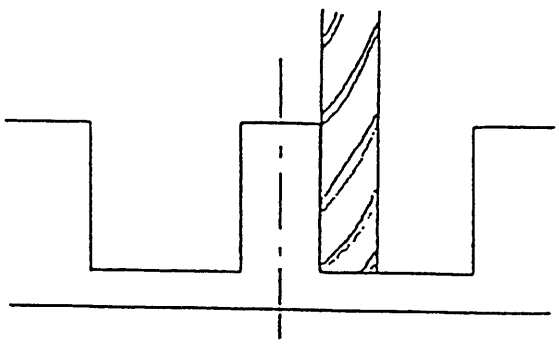


Fig. 7.2 J-**

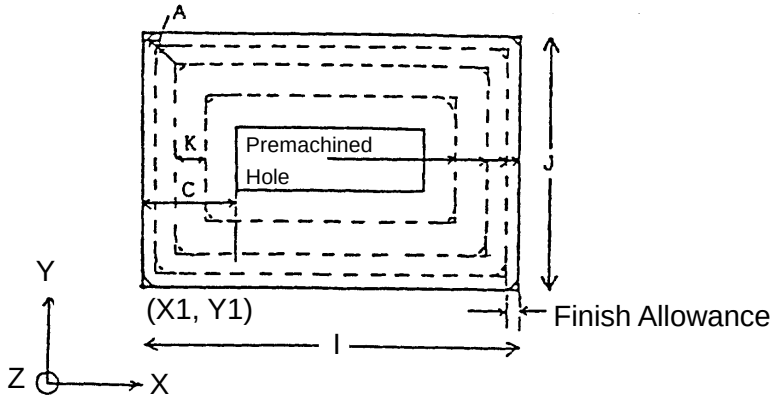
10-7-2 Square Poketing (G328)

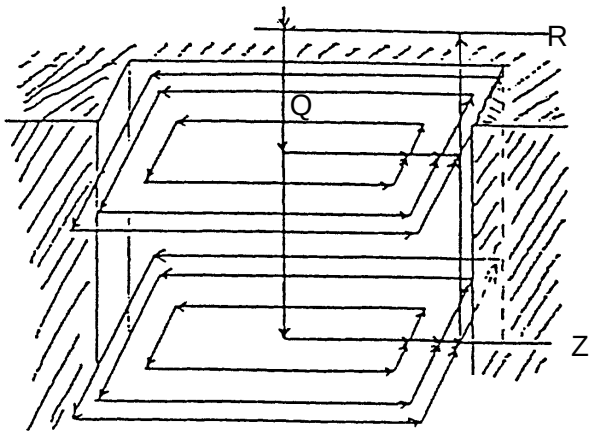
[Purpose] Used when machining inside the square bar with an end mill. The corner R can be also specified.

(1) Command format

G328 X_ Y_ Z_ R_ I_ J_ K_ Q_ C_ A_ D_ E_ U_ V_ F _ ;

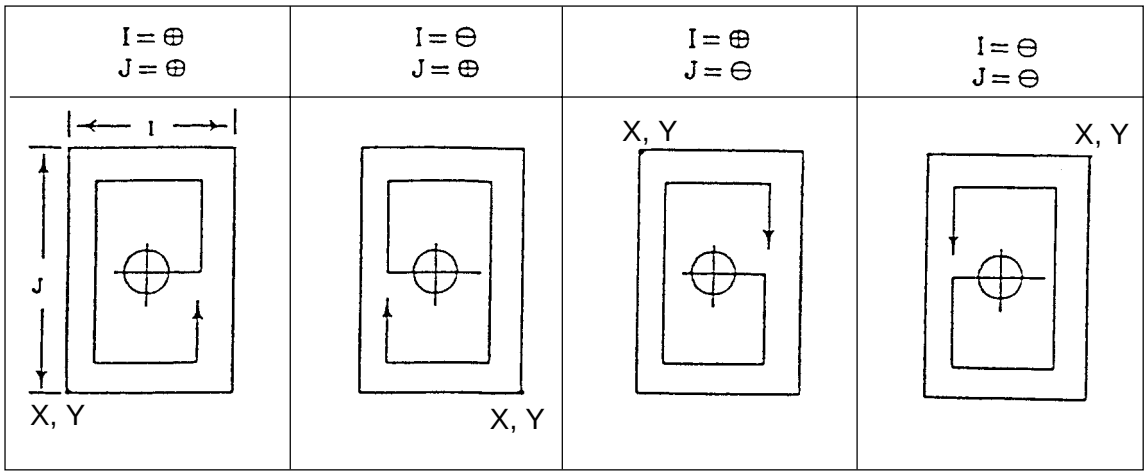
G328 : Square pocket
X, Y : Start point coordinate value of the plane.
Enter it based on G90/G91.
Z : Coordinate value of the pocket finishing Z axis.
Enter it based on G90/G91.
R : R point coordinate value in the Z axis direction
I : Length of one side in the X direction
J : Length of one side of Y axis direction
K : Cutting width ("-": no finishing in the side direction)
Q : Each depth of cut in the Z axis direction
("-": no finishing in the Z axis direction)
C : Removal amount of the single wall in the side direction (absolute value)
A : Radius when specifying the corner R (absolute value)
D : Tool diameter compensation number (radius registration)
E : Finish feed rate (Default feed rate: F) (mm/min.)
U : Finish spindle speed (Default spindle speed: S) (rpm)
V : Z axis cutting-in speed (Default feed rate: F) (mm/min.)
F : Cutting feed rate (Default: F specified before) (mm/min.)



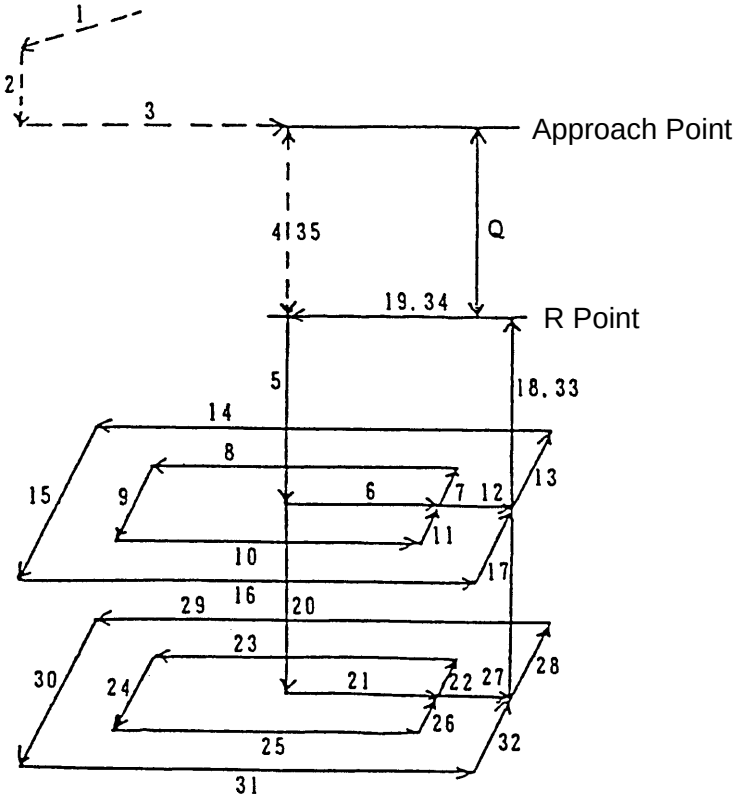


(2) Movements

Cutting Pattern



The cutting direction can be changed with the sign of I and J.



1. Moves to the X, Y point at the rapid traverse rate.
↓
2. Moves to the Z axis approach point.
↓ (R Point + each depth of cut(Q) in the Z axis direction)
3. Moves to the workpiece center in the side direction at the rapid traverse rate.
↓
4. Rapid traverse to the R point
↓
5. Cuts in by each depth of cut Q in the Z axis direction at the cutting speed of V
↓ (when abbreviated, F)

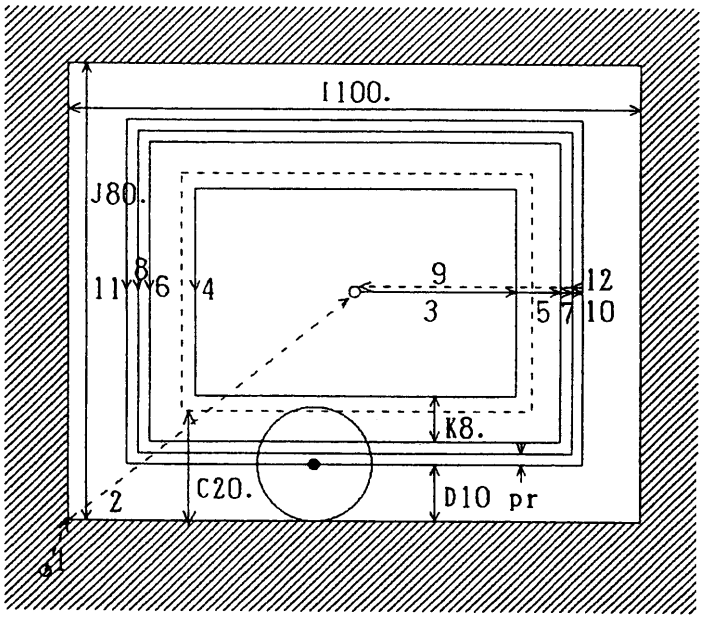
6. The X direction width of cut K at the cutting speed F.
↓
7. Cuts in the Y direction at the cutting speed of nJmax.
↓
8. Cuts in the X direction at the cutting speed of nJmin.
↓
9. Cuts in the Y direction at the cutting speed of Jmin.
↓
10. Cuts in the X direction at the cutting speed of nImax.
↓
11. Cuts as far as the workpiece center in the Y direction at the cutting speed.
↓
12~17. Repeats the steps 6 through 11.
↓
18. Retracts by each depth of cut(Q) in the Z direction at the cutting speed of V
↓ (when abbreviated, F)

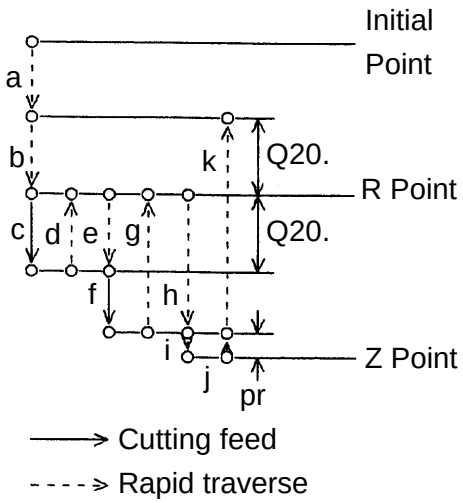
19. Moves to the workpiece center in the side direction at rapid traverse rate.
↓
20~32. Repeats the steps 5 through 17.
↓
o When only the finish allowance is left in the step 32, the tool returns to the R
↓ point, cuts by the specified dimension in the Z axis direction under the finish conditions, and also, cuts by the specified dimension in the side direction.

33. Returns to the R point in the Z axis direction at the cutting speed.
↓
34. Returns to the workpiece center in the side direction at rapid traverse rate.
↓
35. Returns to the approach point in the Z axis direction at the rapid traverse rate and terminates.

(3) Sample program

G328 X-50. Y-25. Z-50. R-10. I100. J50. K8. Q20. C15. D10





Tool center path 1→a→2→b→c→3→4→5→6→7→8→d→9
→e→f→3→4→5→6→7→8→g→9
h→i→3→4→5→6→10→11→12→j→k→9

(4) Cautions

(a) When the workpiece has a premachined hole, specify the removal amount (C)of single wall.
When (C) is not specified, the tool cuts from the center, assuming that there is no premachined hole. (Fig. 7.3)

(b) When the cutting width (K) is a negative value, no finishing is performedin the side direction.

(c) When each depth of cut (Q) in the Z axis direction is a negative value, no finishing is performed in the Z direction.

(d) When the corner R is specified (radius A), an override is applied to the corner based on the parameter (N5116).
When A is larger than the half of the side I or J, it is not machined.

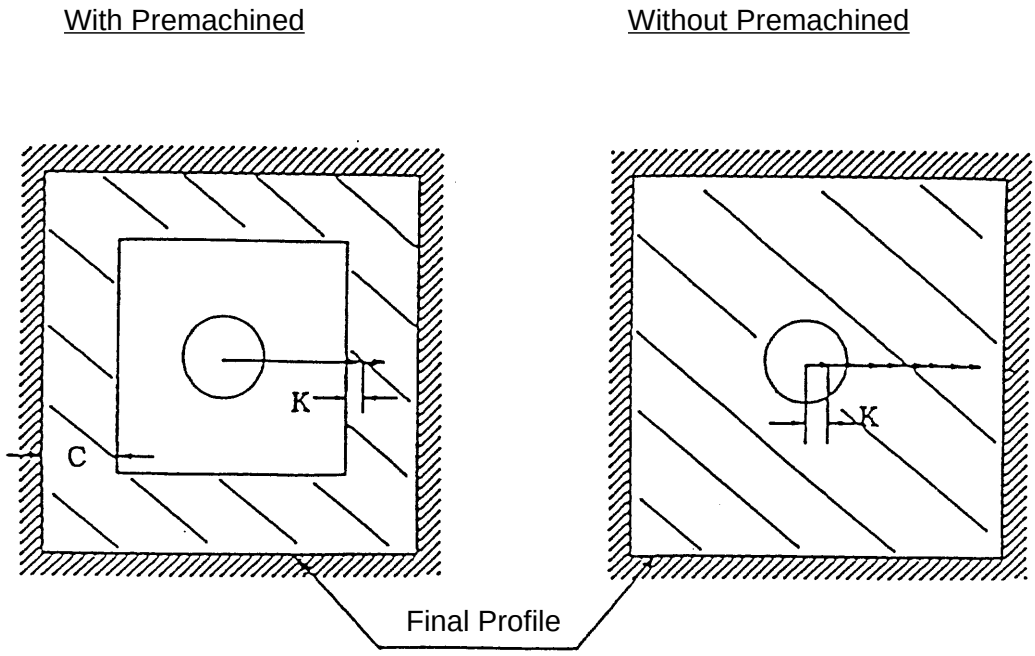


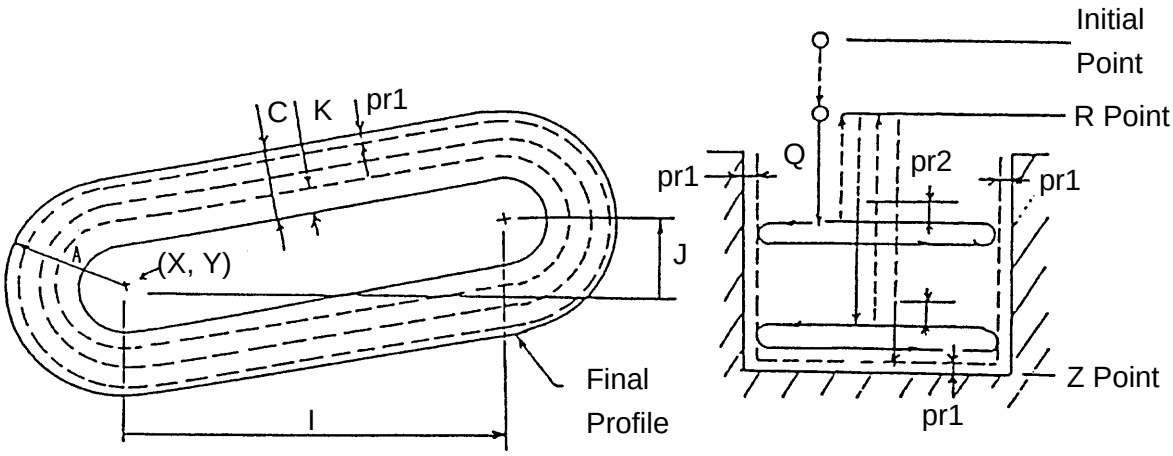
Fig. 7.3

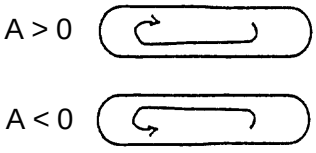
10-7-3 Track Inside (G329)

You can specify in one block a series of actions which cuts the inside of the track, using an end mill. The following describes the case of G17 (X-Y plane).

(1) Command format

G329 X_ Y_ Z_ R_ I_ J_ A_ C_ K_ Q_ D_ V_ E_ U_ F_ ;





(X, Y) : Reference point of the X and Y axes
Z : Z point
R : R point
I : Arc center component on the opposite side (X axis)
J : Arc center component on the opposite side (Y axis)
A : Arc radius
C : Removal amount
K : Each cutting width (X-Y plane)
Q : Each depth of cut of the Z axis
D : Tool offset number
V : Z-axis cutting-in speed (Default: Cutting feed rate F)
E : Finish feed rate (Default: Cutting feed rate F)
U : Finish spindle speed (Default: Already specified S)
F : Cutting feed rate
pr1 : Finish allowance (set in the parameter No. 5152)
pr2 : Clearance amount (set in the parameter No. 5153)

(Note 1) When E is omitted, the finish feed rate is the FX finish override (parameter No. 6207).

(Note 2) When the numerical value of A is positive, an arc becomes clockwise, and when the value is negative, the arc becomes counterclockwise. When A is 0, an alarm results.

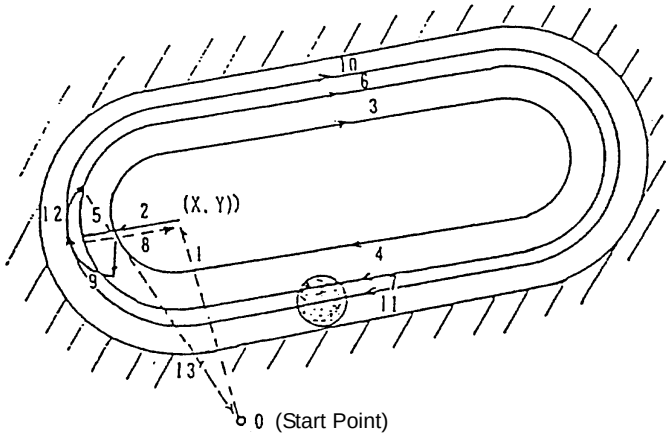
(Note 3) When the numerical value of K is negative, side finishing is invalidated.

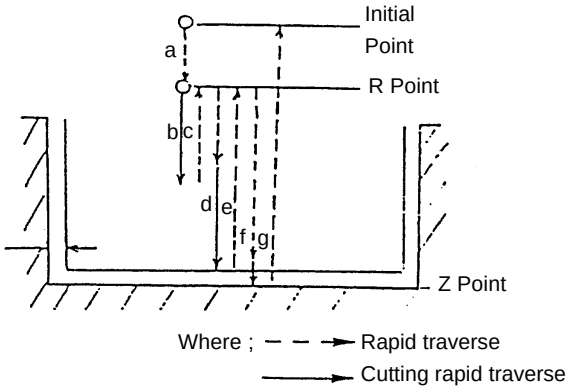
(Note 4) When the numerical value of Q is negative, bottom finishing is invalidated.

(2) Sample program

G17 ;

G90 G329 X50. Y-100. Z-50. R-10. Q20. I50. J20. A50. C15. K8. D10 F200;





Tool center path: 0→1→a→b→2→3→4→5→6→7→c→8→
d→2→3→4→5→6→7→e→8→
f→2→3→4→9→10→11→12→g→
13→0

Movements:

Start point

↓

Rapid traverse to (X, Y)

↓

Rapid traverse to the R point

↓ ↓

Z-axis cut-in

↓ ↓

1st inside cutting of the track

↓ ↓

2nd inside cutting of the track

↓ ↓

Rapid traverse of the R point

↓ ↓

Rapid traverse to (X, Y)

↓

Rapid traverse and cut-in to the Z point

↓

Bottom finishing

↓

Side finishing

↓

Return to the Z-axis initial point (start point)

↓

Return to the X- and Y-axis start point

Roughing

Finishing

(3) Cautions

(a) Specify the numerical values of the addresses V, E and U without a decimal point. (in case of the metric system)

(b) An alarm results in case of tool offset amount ((D)) > arc radius (A).

(c) An alarm results if you specify I = 0 and J = 0.

(d) When the address C is omitted, the arc radius is taken as the removal amount.

(e) For the numerical values of the addresses A, C, K and Q, and the tool offset amount ((D)), specify those enabling track inside cutting.

(4) Differences from SEICOS II

(a) The addresses I and K are relative positions viewed from (X, Y).

(b) Upon start of machining, the Z axis moves directly to the R-point without being positioned to the approach Z point.

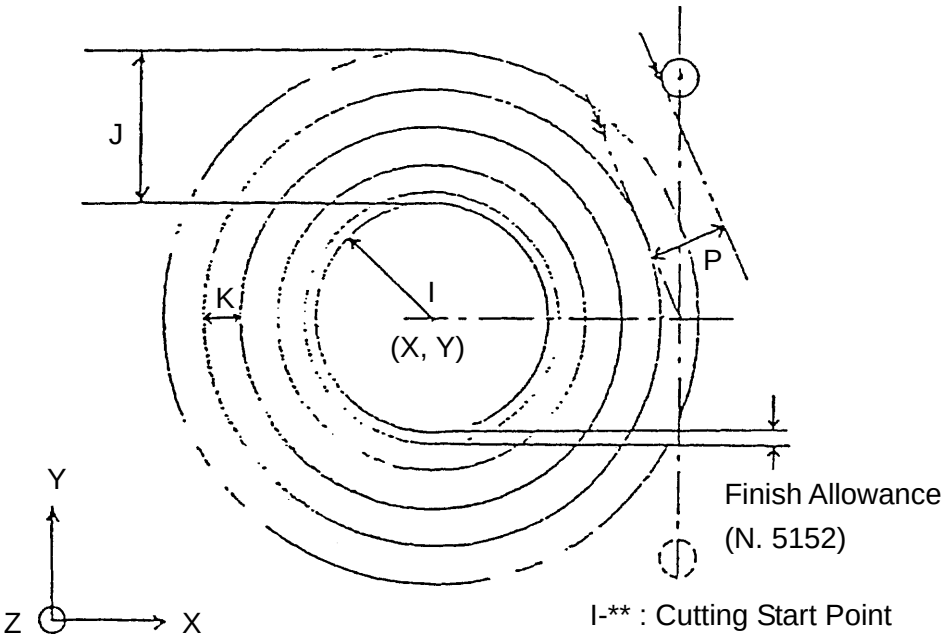
10-7-4 Circle outside Pocketing (G330)

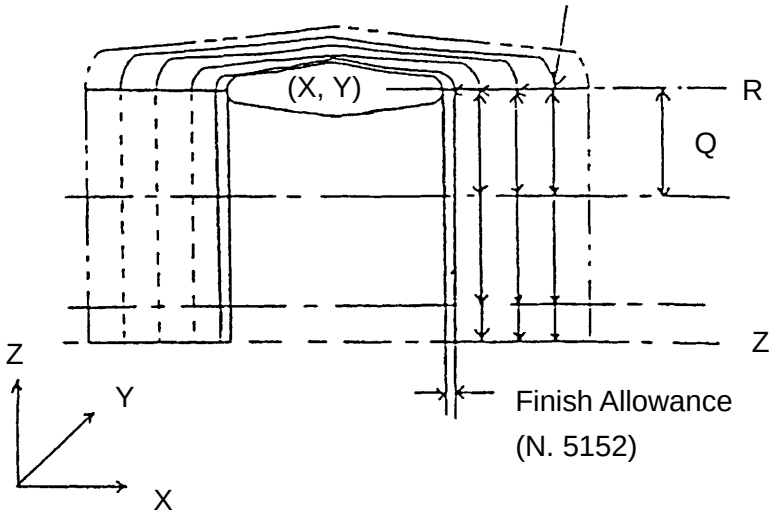
[Purpose] Used when cutting the outside of the circle with an end will. For detailed description in case of G17 (X, Y plane), as follows.

(1) Command format

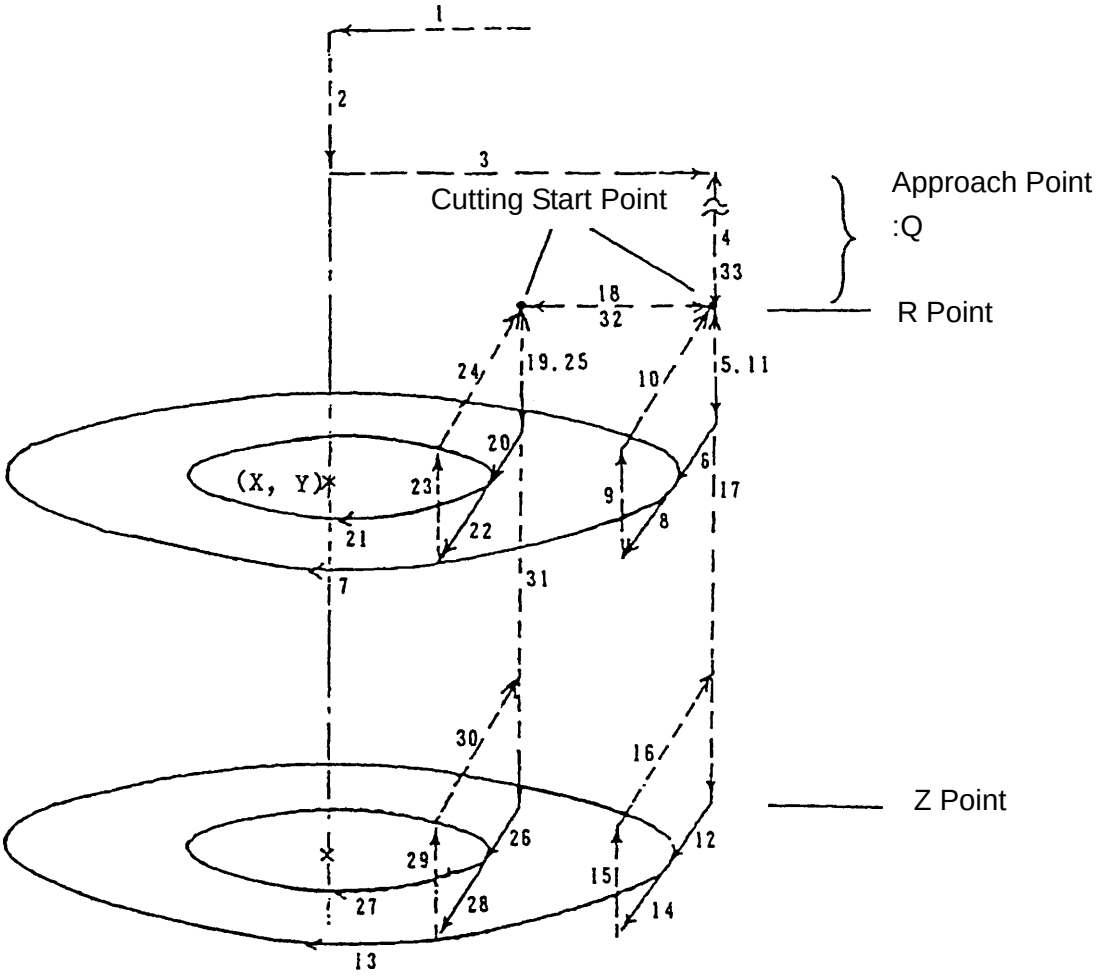
G330 X_ Y_ Z_ R_ I_ J_ K_ Q_ P_ D_ E_ U_ V_ F_ ;

G330	: Circle outside
X, Y	: Coordinate value of the circle center.
Z	: Z coordinate value. Enter it based on G90/G91.
R	: R point coordinate value in teh Z axis direction
I	: Radius of the circle to be cut (cutting direction selected depending on the sign "+/-")
J	: Removal amount of the circle to be cut (absolute value)
K	: Cutting width ("-" : no finishing in the side direction)
Q	: Each depth of cut in the Z axis direction
P	: Approach amount (absolute value)
D	: Tool diameter compensation number (radius registration)
E	: Finish feed rate (Default: Feed rate F) (mm/min.)
U	: Finish spindle speed (Default: Spindle speed S) (rpm)
V	: Z axis cutting-in speed (Default: F specified before) (mm/min.)
F	: Cutting feed rate (Default: F specified before) (mm/min.)




(X, Y)
R
Q
Z
Z
Y
X
Finish Allowance
(N. 5152)

(2) Movements


1
2
3
Cutting Start Point
4
33
18
32
Approach Point
:Q
R Point
24
19.25
10
5.11
20
6
(X, Y)
23
9
17
21
22
8
7
31
30
16
Z Point
26
12
29
15
27
28
14
13

1. Moves to the X, Y point at the rapid traverse rate.
↓
2. Moves to the Z axis approach point at the rapid traverse rate. (R Point + each
↓ depth of cut(Q) in the Z axis direction)

3. Moves to the approach point at the rapid traverse rate, considering the cutting
↓ width of the removal amount.

4. Rapid traverse to the R point
↓
5. Moves by each depth of cut (Q) in the Z axis direction (When this is done, the tools
↓ moves by the clearance amount (No. 5153) to the near side at the rapid traverse rate, and after reaching, it moves by cutting feed)
6. Cuts in as far as the Y point in the Y direction by cutting feed.
↓
7. Arc cutting (outside)
↓ Citcular radius command : I “+” (clockwise rotation)
Circular radius command : I “-” (counterclockwise rotation)
8. Retracts by the cutting width (K) in the tangent direction of the outer circle by cutting feed.
↓
9. Moves to the cutting start point in the Z axis direction at the rapid traverse rate.
↓
10. Moves to the cutting start point on the X/Y direction at the rapid traverse rate.
↓
11~16. Repeats the steps 5 through 10.
↓
17. Returns to the R point at the rapid traverse rate.
↓
18. Moves by the cutting width (K) in the X direction at the rapid traverse rate.
↓
19~31. Repeats the steps 5 through 17.
↓
32. Returns to the initial cutting start point in the X direction at the rapid traverse rate.
↓
33. Returns to the approach point in the Z axis direction and terminates.
↓

(3) Cautions

(a) When the cutting width (K) is a negative value, no finishing is performed in the X and Y (side) directions.

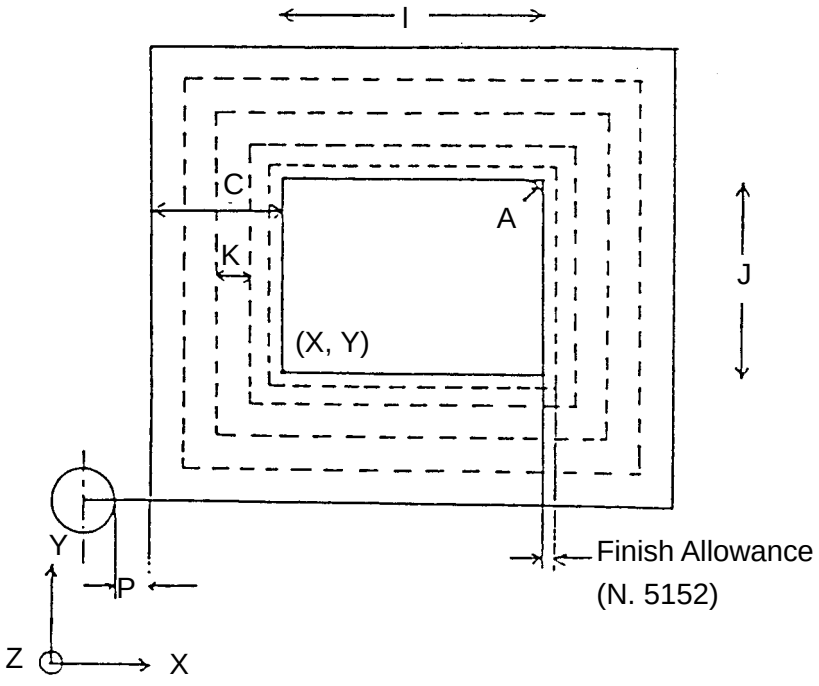
10-7-5 Square Outside Cutting (G331)

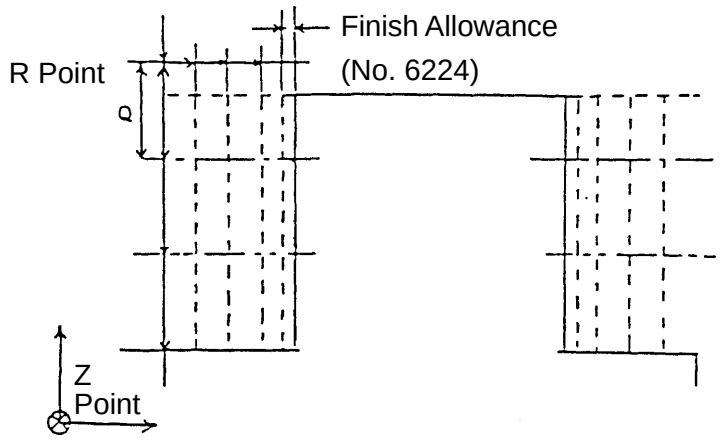
[Purpose] Used when cutting the outside of the square with an end mill.
It is also possible to specify the corner R.
For detailed description in case of G17 (X, Y plane), as follows.

(1) Command format

G331 X_ Y_ Z_ R_ I_ J_ K_ Q_ P_ C_ A_ D_ E_ U_ F_ ;

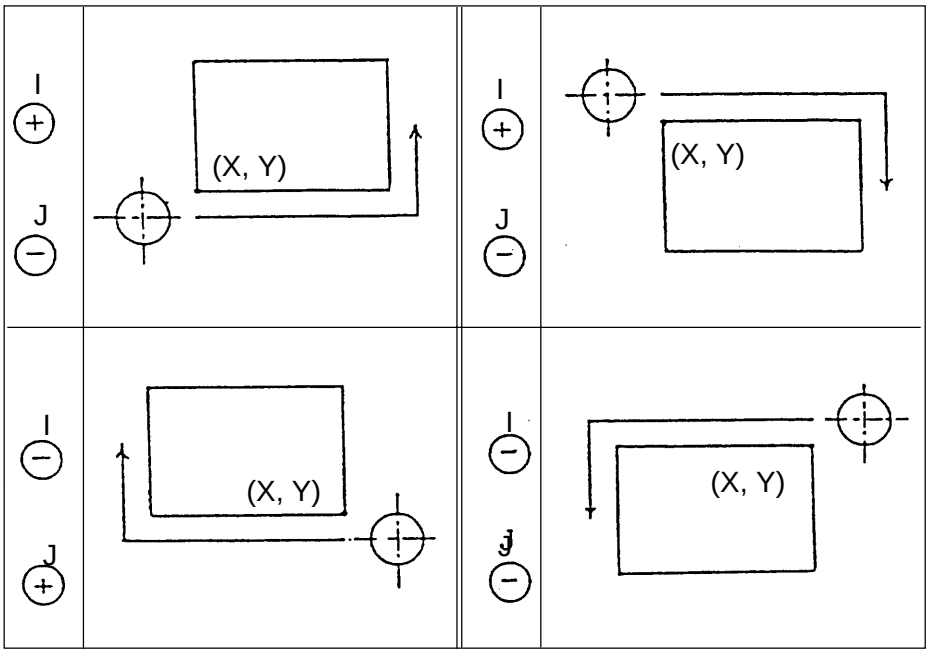
G331	: Square outside
X, Y	: Start point coordinate value of the plane. Enter it based on G90/G91.
Z	: Z axis coordinate value. Enter it based on G90/G91.
R	: R point coordinate value in the Z axis direction
I	: Length of one side in the X direction
J	: Length of one side in the Y direction
K	: Cutting width (“-” : no finishing in the side direction)
Q	: Each depth of cut in the Z axis direction
P	: Approach amount (absolute value)
C	: Removal amount of single wall in the side direction (absolute value)
A	: Radius when the corner R is specified (absolute value)
D	: Tool diameter compensation number (radius registration)
E	: Finish feed rate (Default: Feed rate F) (mm/min.)
U	: Finish spindle speed (Default: Spindle speed S) (rpm)
F	: Cutting feed rate (Default: F specified before) (mm/min.)

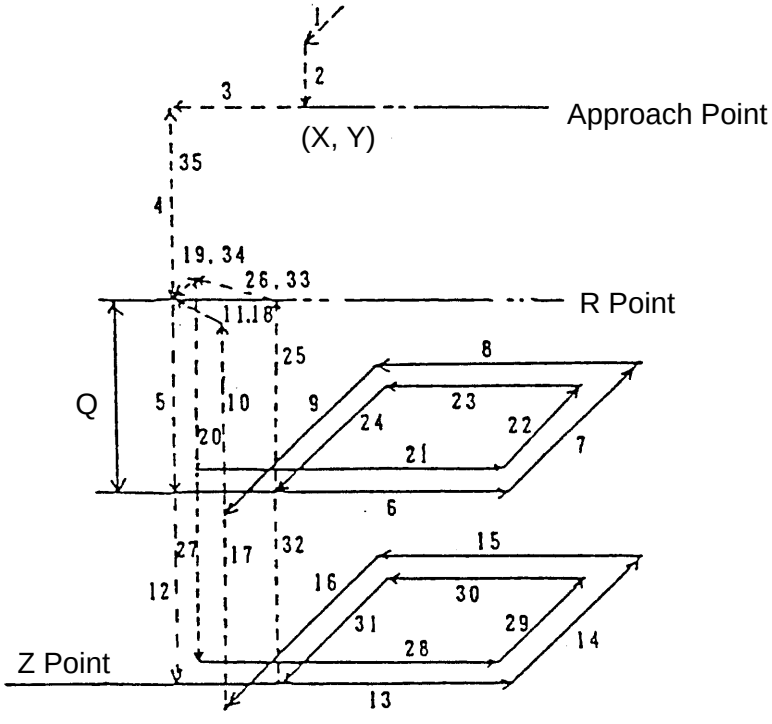




(2) Movements

Cutting Pattern





1. Moves to the X, Y point at the rapid traverse rate.
↓
2. Moves to the Z axis approach point at the rapid traverse rate.
↓ (R Point + each depth of cut (Q) in the Z axis direction)
3. Moves to the 1st cutting-in position in the side direction at the rapid traverse rate.
↓
4. Rapid traverse to the R point.
↓
5. Moves by each depth of cut (Q) in the Z axis direction
↓ (When this is done, the tools moves to the near side by the clearance amount (No. 5153) at the rapid traverse rate, and after reaching, it moves by cutting feed)
6. Cuts in the X direction at the cutting speed of nImax.
↓
7. Cuts in Y direction at the cutting speed of nJmax.
↓
8. Cuts in the X direction at the cutting speed of nImin.
↓
9. Cuts by one cutting width (K) of nJmin. In the Y direction at the cutting speed.
↓
10. Retracts to the R point in the Z axis direction at the rapid traverse rate.
↓
11. Returns to the cutting start point at the rapid traverse rate.
↓
12~18. Repeats the steps 5 through 11.
↓
19. Moves by the cutting width (K) from the cutting start point in the Y direction at the rapid traverse rate.
20~33. Repeats the steps 5 through 18.
↓
o When only the finish alowance is left in the step 33, the tool cuts the workpiece by the
↓ specified dimension in the side and Z axis directions under the finish conditions.

34. Returns to the initial cutting start point in the side direction.
↓
35. Returns to the approach point in the Z axis direction and terminates.

(3) Cautions

(a) When the cutting width (K) is a negative value, no finishing is performed in the side direction.

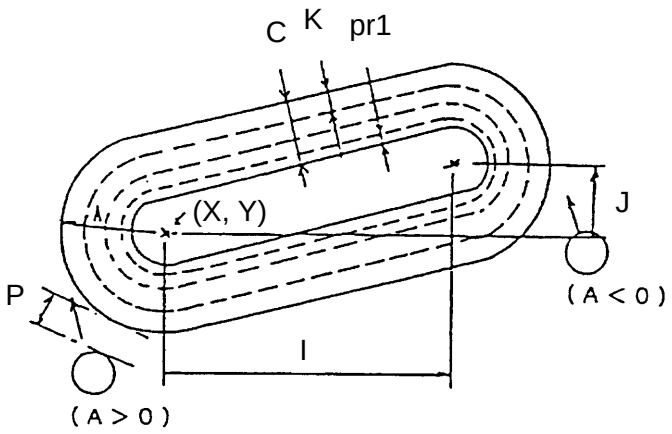
(b) When the corner R is specified (radius A), an override is applied at the corner based on the parameter (N5116), and the retract direction by K after nJmin. in the Y direction results in the X direction. When A is larger than half of the side I or J, no machining is done.

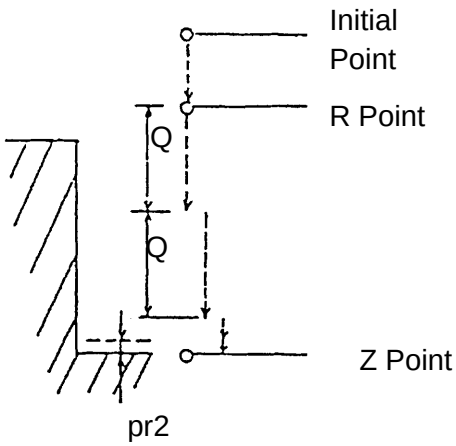
10-7-6 Track Outside (G332)

You can specify a series of action which cuts the outside of the track with an end mill. The following describes the case of G17 (X-Y plane).

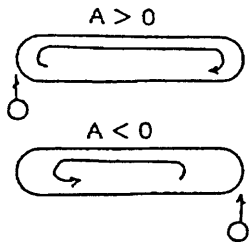
(1) Command format

G332 X_ Y_ Z_ R_ I_ J_ A_ C_ K_ Q_ P_ D_ E_ U_ F_ ;





(X, Y) : Reference point of the X and Y axes
Z : Z point
R : R point
I : Arc center component on the opposite side (X axis)
J : Arc center component on the opposite side (Y axis)
A : Arc radius
C : Removal amount
K : Each cutting width (X-Y plane)
Q : Each depth of cut of the Z axis
P : Approach amount
D : Tool offset number
E : Finish feed rate (Default: Cutting feed rate F)
U : Finish spindle speed (Default: S already specified)
F : Cutting feed rate
pr1 : Finish allowance (set in the parameter No. 5152)
pr2 : Clearance amount (set in the parameter No. 5153)



(Note 1) When E is specified, the finish feed rate is the FX finish override (parameter No. 5115).

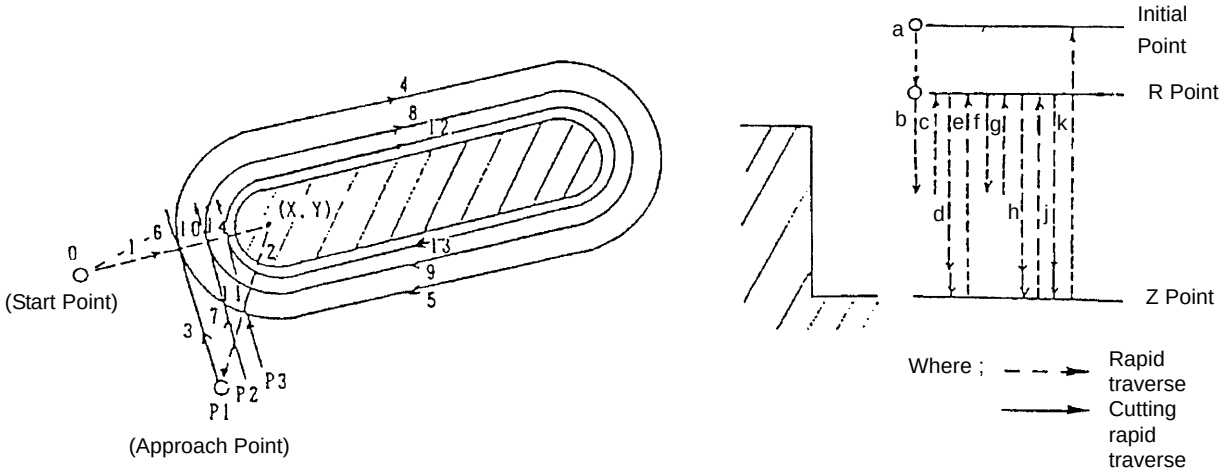
(Note 2) When the numerical value of A is positive, the arc is clockwise, and when the value is negative, the arc is counterclockwise. An alarm results in case of A = 0.

(Note 3) When the numerical value of K is negative, side finishing is invalidated.

(2) Sample program

G17 ;

G90 G332 X50. Y-100. Z-50. R-10. Q20. I50. J20. A50. C15. K8. P5. D10 F200 ;



Tool center path: 0→1→2→a→b→3→4→5→6→c→P1→d
→3→4→5→6→e→P2→f
→7→8→9→10→g→P2→h
→7→8→9→10→i→P3→j
→11→12→13→14→k→0

Movements : Start point

Rapid traverse to (X, Y)

↓

Rapid traverse to the approach point

↓

Rapid traverse to the R point

↓

Rapid traverse of the Z-axis of the Z-axis depth of cut

↓

Track outside cutting

↓

Rapid traverse to the R point

↓

Rapid traverse to the approach point

↓

Rapid traverse and cut-in to the Z point

↓

Track outside cutting

↓

Rapid traverse to the initial point

↓

Return to the X- and Y-axis start point

Roughing

Finishing

(3) Cautions

(a) Specify the numerical values of the addresses E and U without a decimal point. (in case of the metric system)

(b) An alarm results if you specify I = O and J = O.

(c) An alarm results if P = O is specified.

(d) When the address C is omitted, the arc radius is taken as the removal amount.

(e) For the numerical values of the addresses A, C, K and Q, and the tool offset amount ((D)), specify those enabling track outside cutting.

(4) Differences from SEICOS II

(a) The addresses I and J are relative positions viewed from (X, Y).

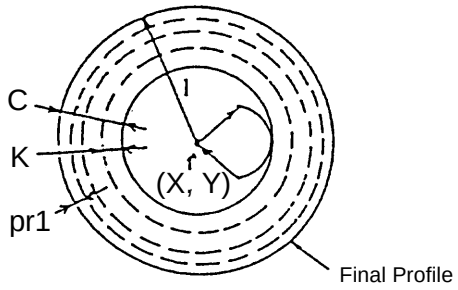
(b) Upon start of machining, the Z axis moves directly to the R-point without being positioned to the approach Z point.

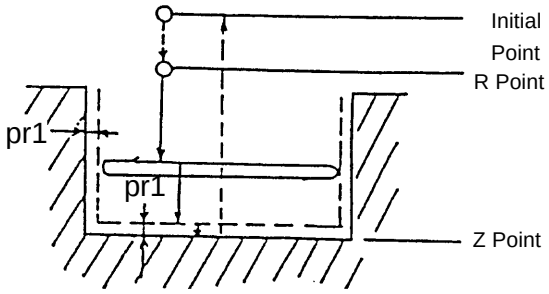
10-7-7 Circle (G333)

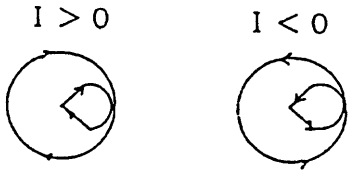
You can specify in one block a series of actions which cuts the inside of the true circle using an end mill. The following describes the case of G17 (X-Y plane).

(1) Command format

G333 X_ Y_ Z_ R_ I_ Q_ C_ K_ D_ U_ V_ W_ E_ F_ ;







(X, Y)	: Reference point of the X and Y axes
Z	: Z point
R	: R point
I	: Radius of the circle of the final profile
Q	: Each depth of cut of the Z axis
C	: Removal amount
K	: Each cutting width (X-Y plane)
D	: Tool offset number
U	: Finish spindle speed (Default: Already specified S)
V	: Z-axis cutting-in speed (Default: Cutting feed rate F)
W	: Approach circle relief speed (Default: Cutting feed rate F)
E	: Finish feed rate (Default: Cutting feed rate)
F	: Cutting feed rate
pr1	: Finish allowance (set in the parameter No. 5152)

(Note 1) When E is specified, the finish feed rate is the F x finish override (parameter No. 5115).

(Note 2) When the numerical value of I is positive, the arc is clockwise, and when the value is negative, the arc is counterclockwise. An alarm results in case of I = 0.

(Note 3) When the numerical value of K is negative, side finishing is invalidated.

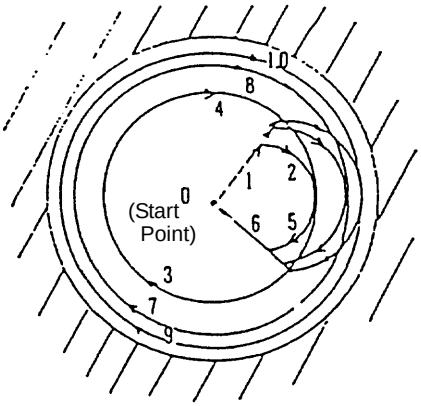
(Note 4) When the numerical value of Q is negative, bottom finishing is invalidated.

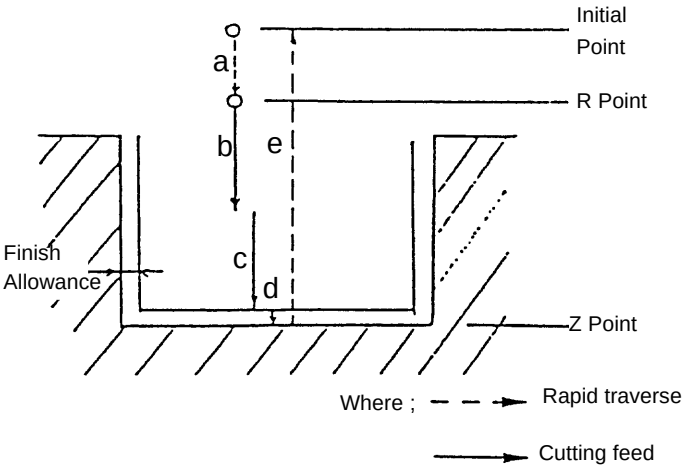
(Note 5) For special canned cycle, P indicates approach relief speed. In this case, W indicates finish allowance. (See Item 11-7-8.)

(2) Sample program

G17 ;

G90 G333 X50. Y-100. Z-50. R-10. Q20. I50. C15. K8. D10 F200 ;





Tool center path : 0→a→b→1→2→3→4→5→6→c
→1'→2'→7→8→5'→6'→d
→Bottom finishing (Starts from the circle which takes the cutting width as the radius)
→1'→2'→9→10→5'→6'→e

Movements:

Start point
↓
Rapid traverse to the R point
↓ ↓
Cutting feed of the Z-axis depth of cut
↓ ↓ ↓
X- and Y-axis approach
↓ ↓ ↓ ↓
True circle inside cutting
↓
X- and Y-axis approach return
↓
Cutting feed to the Z axis
↓
X- and Y-axis approach
↓
True circle bottom and inside cutting
(Starts from the circle taking the cutting width as the radius)
↓
X- and Y-axis approach return
↓
Rapid traverse to the initial point

Roughing

Finishing

(3) Cautions

(a) In the rapid traverse area of approach, if rapid traverse is specified for the cutting area by setting the cutting width (K) and offset amount ((D)), cutting feed is applied automatically.

10-7-8 Special Fixed Cycles (G322 ~ G333) Type 2

There are the following two types of special fixed cycles (G322~G333):

Parameter No. 5101, #1 = 0 ... Type 1 (operation as mentioned before)

= 1 ... Type 2 (unified specifications)

For type 2, all operations at special fixed cycles are performed according to the same rules.

(1) Basic operation

① Moving to X- and Y-axes and the machining start position

② Going down to the Z-axis and point R

③ Machining operation

④ Going up to the Z-axis and start position

When parameter No. 5101, #2 = 1, the operation ⑤ is finally performed.

⑤ Moving to X- and Y- axes and the command position

* Start position ... Position prior to operation at fixed cycles

* Machining start position ... Position for approach to square out-of-plane cutting
Position for approach to plane cutting
Pocket center for inside pocket cutting Position for approach to outside pocketcutting

* Command position ... Position where a command is given at addresses of X and Y

(2) Feed speed

Feed speed ... F command

• When F command omitted →use a modal F

Feed speed at Z-axis ... V command

• When V command omitted → feed speed

Feed speed at corner R ... feed speed x parameter No. 5116(%)

Finishing feed speed ... E command

• When E command omitted → feed speed x parameter No. 5115(%)

Finishing feed speed at Z-axis ... V command x parameter No. 5115(%)

• When V command omitted → finishing feed speed

Finishing feed speed at corner R ... finishing feed speed x parameter No. 5116(%)

* Parameter No. 5115 = 0 → 100(%)

* Parameter No. 5116 = 0 →100(%)

(3) Finishing stock

Finishing stock is assigned with W command. With this being omitted, Parameter No. 5152 is taken as the finishing stock.

When finishing is made ineffective by the program command (KQ minus command) → Rough cutting is performed until the programmed shape is taken without leaving any finishing stock.

When finishing = 0 → Finishing operation is not performed irrespective of any program command.

10-8 ATC Canned Cycle (MO6)

A series of ATC operations (tool change) can be specified in one block.

(1) Command format

{ T_ } M06 { B_ } { P_ } { X_ } ; { ~ } → Omissible

T_ : Tool number to be called.

When omitted, a stand by tool is called.

T00 delivers the spindle tool.

B_ : 2nd auxiliary function.

P_ : Operation parameter.

Specify with P or A. When the 2nd auxiliary function is used or A is used for an additional axis, A is not available.

Depending on the machine models, the operation parameter is not available.

X_ : Table axis ATC position (machine coordinates).

Specify with X or Y.

When omitted, the table moves in accordance with the settings of the parametersNo. 5103 and No. 5161. Refer to (3).

※For the additional axis, refer to (4).

(2) Types

There are the following types of ATC canned cycles:

Type A --- Models VK, VKC, VG, VKII
Type E --- Model VM40III, VM50
Type F --- Model HG
Type G --- Model HK
Type I --- Model HS500(old)
Type J --- Model VS(standard)
Type K --- Model HS630
Type L --- Model HS500(new)
Type M --- Model VS40(16-tool)
Type N --- Model MS400H
Type B, C, D were used in SEICOSMIII.

※For detailed operations and parameters, refer to the description of each type (11-8-1 onward).

(3) ATC position of the table

• At ATC time, the table can be moved to a specified position.

Method ① : In the same block as M06, specify a table position in terms of machine coordinates.

Method ② : With 1 being set in table axis #0 of Parameter No. 5109, the table is shifted to place set in No. 5161. (Machine coordinates)

When two methods above are used simitaneously, the method ① is given priority.

(4) ATC position of the additional axis

• At ATC time, the additional axis can be returned to the reference point. Set the axis you to return and the desired reference point to the parameter No. 5109.

No. 5109, #0 = 0	Does not return to the reference point
= 1	Returns to the reference point
No. 5109, #2: #1	
= 0:0	Position set in No. 5161
= 0:1	1st reference point
= 1:0	2nd reference point

• When the additional axis is a rotary axis, select the return method out of the following.

No. 5109, #4: #3	
= 0:0	Same as the linear axis
= 0:1	Short-cut (return within one revolution)
= 1:0	“+” direction (return within one revolution)
= 1:1	“-” direction (return within one revolution)

• The parameter No. 5109 is available for 8 axes. Set a parameter corresponding to the additional axis. (Setting to the parameters for other than the additional axis is invalid)

• When the additional axis is a rotary axis, setting to #3 or #4 is invalid.
(The return method complies with rotary control)

(5) Related alarms

No. 162(#???) An M-code command for canned cycle has an error.

???	Cause and Countermeasure
001	ATC canned cycle has been executed while in canned cycle. Cancel canned cycle and execute M06.
002	ATC canned cycle has been executed while in normal alignment. After executing G4501, execute M06.
003	ATC canned cycle has been executed while in diameter compendation. After executing G40, execute M06.
004	G code and M06 exist in the same block. Delete the G code.
005	Erroneous address exists in M06 block. Delete the address.
006	Erroneous axial command exists in M06 block. Command inhibited axes has been specified. Delete it before execution. Command inhibited axes: Type A ... Y, Z Type D ..... Y, Z Type E ... X, Z Type F, G .. X, Y, Z
007	Parameter error Confirm parameters.
008	The specified tool is the same as the spindle tool. Issued only when PRA5103 #3 = 1 with the HK630
009	An additional axis was specified with the HG. The additional axis cannot be specified with the HG. This is also true when set with a parameter.

(6) Notes

- { T_ } M6 { B_ } { P_ } { X_ } ; command should be specified in a singleblock.
- If the ATC canned cycle is specified in the canned cycle mode (when a G-code in the G09 group is other then G80), an alarm will result.

10-8-1 ATC Canned Cycle, Type A (VK, VKC, VG, VkII)

(1) Command format

{ T_ } M06 { B_ } { X_ } ; { ~ } → Omissible

T_ : Tool number to be called.

When omitted, a standby tool is called.

T00 delivers the spindle tool.

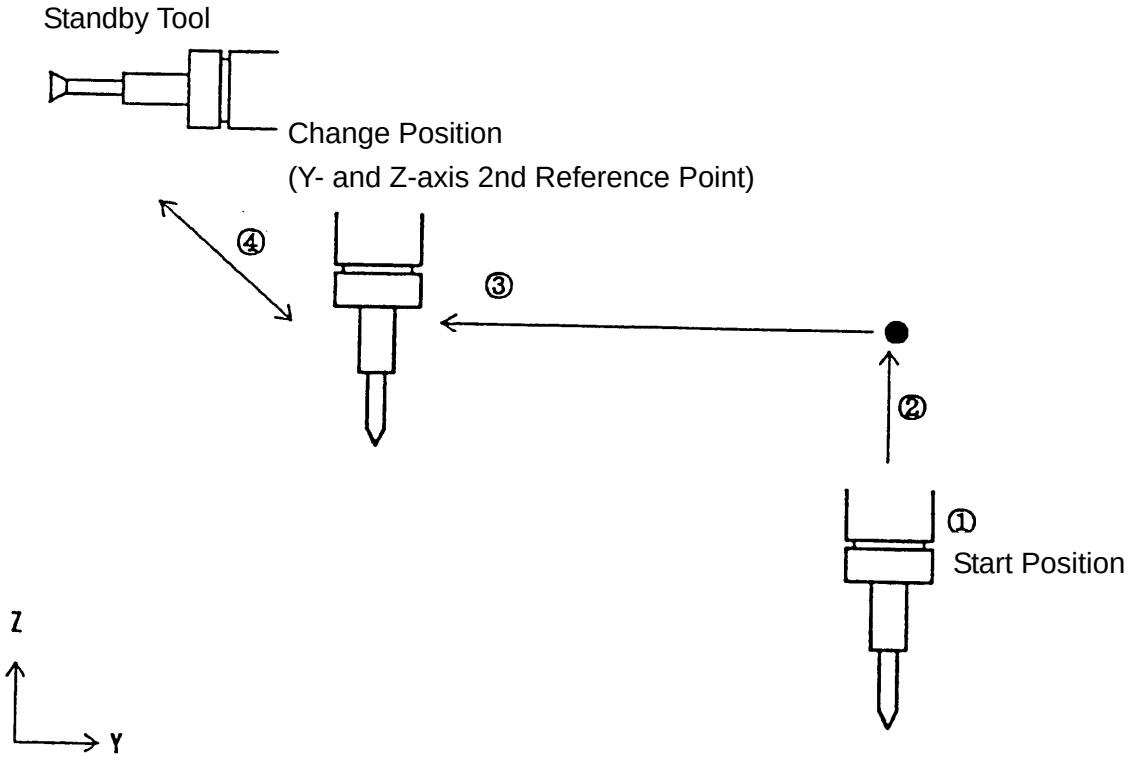
B_: 2nd auxiliary function.

Refer to Page 13-4.

X_: X-axis (table) ATC position.

When omitted, the table moves in accordance with the settings of the parametersNo. 5109 and No. 5161.

(2) Movements



	Command	Movement	
①	M15;	Spindle + coolant stop	
②	G30 G91 Z0;	Z-axis change position	
③	G30 G91 Y0 M19;	Y-axis change position + spindle orientation	*1
④	T_ M06;	Tool change	

*1 When the table (X), 2nd auxiliary function (B), and ATC position of the additional axis are ordered or specified with parameters, the machine moves in the movement ③ .

(3) Related parameters

No.		Description
1226	X	--
	Y	ATC change position (Machine coordinated[mm])
	Z	ATC change position (Machine coordinated[mm])
5103	#1	Set 0.
	#2	Set 0.
5109	#0 0	No shifting of table/additional axes.
5161		Shifted to position set with Parameter 5161.
5161		X-axis ATC position (Machine coordinated[mm/inch])

※ For the parameters related to the additional axis, refer to 7-8 (4).

10-8-2 ATC CANNED CYCLE TYPE E (VM40III)

(1) Command Format

{ T_ } M06 { B_ } { Y_ } ; { ~ } → Omissible

T_ : A tool No. called out.

When omitted, call a standby tool.

For T00, a spindle tool is discharged.

B_: 2nd auxiliary function

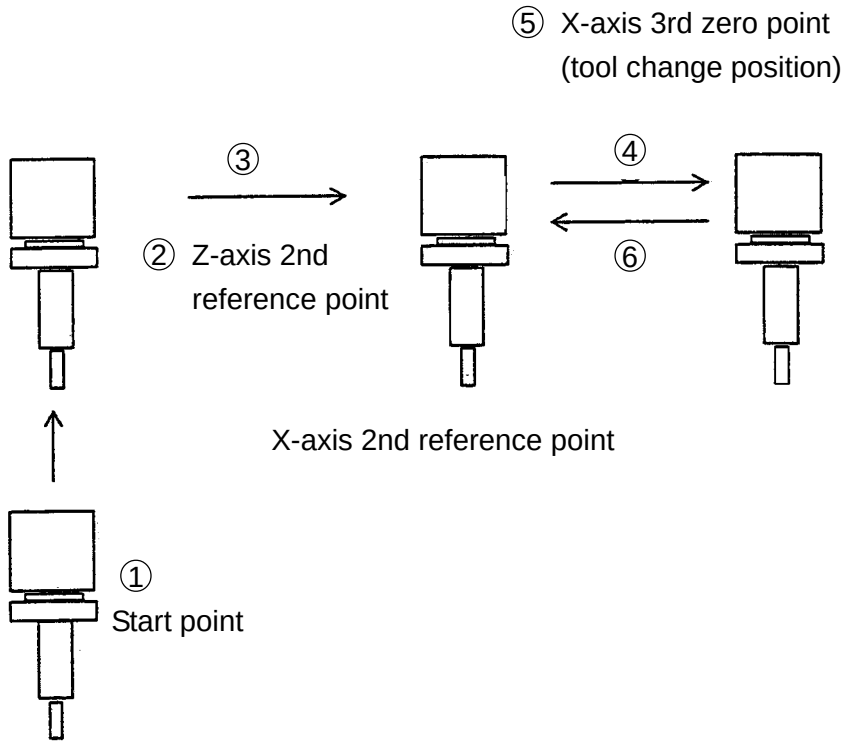
See Item 13-4.

Y_: Y-axis (table) ATC position

When omitted, table shifting follows Parameter Nos. 5109/5161.

* For additional axes, see Item 11-8 (4).

(2) Operation



	Command	Operation	
①	T_ ;	Tool call	
②	G91 G30 Z0 M15;	Z-axis 2nd reference point, Spindle stop	*1
③	G91 G30 P2 X0 M106;	X-axis 2nd reference point + arm swing-out	*2
④	G91 G30 P3 X0 M06 { Y_ B2 } ;	X-axis 3rd reference point + arm turn	
⑤	G91 G30 P2 X0 M107;	X-axis 2nd reference point + arm return	

(*1) If the tool called and spindle tool are the same, processing will end in ③ . (Does not output the T-code in ① and M106 in ③)

(*2) When Table (Y), 2nd auxiliary function (B), and ATC position of additional axis have been specified with a command or a parameter, shifting takes place in Operation ④.

(3) Associated Parameters

No.			Details	
1226	X		ATC change position	(Machine coordinates[mm])
	Y		--	
	Z		ATC change position	(Machine coordinates[mm])
1227	X		ATC change position	(Machine coordinates[mm])
	Y		--	
	Z		--	
5130	#1		Set 0.	
	#2		Set 0.	
5109	#0	=0	Y-axis not shifted in ATC canned cycle.	Y-axis command, if any, is preceded.
		=1	Y-axis shifted to No. 5161 position.	
6226			Y-axis ATC position	(Machine coordinates[mm/inch])

* As for parameters relating to additional axes, see Item 7-8 (4).

10-8-3 ATC CANNED CYCLE TYPE F (HG)

(1) Command Format

{ M_ } M06 { B_ } ; { ~ } → Omissible

T_ : A tool No. called out.

When omitted, a standby tool is called out.

With T00, a spindle tool is discharged.

B_: 2nd auxiliary function

See Item 13-4.

* For additional axes, see Item 7-8(4).

(2) Operation

	Command	Operation
①	G91 G30 P2 Z0 M15;	Z-axis 2nd reference point, Spindle, Coolant stop
②	G91 G30 P2 X0 Y0 M06 (T_) (B2_);	X- and Z-axis 2nd reference point, Spindle orientation; 2nd auxiliary function B is output only when specified
③	(S_);	An S-code is output only when specified

(3) Associated Parameters

No.		Details	
1226	X	ATC change position	(Machine coordinates[mm])
	Y	ATC change position	(Machine coordinates[mm])
	Z	ATC change position	(Machine coordinates[mm])
5103	#1	Set 0.	
	#2	Set 0.	

10-8-4 ATC Canned Cycle, Type G (HK)

(1) Command format

{ T_ } M06 { B_ } ; { ~ } → Omissible

T_ : Tool number to be called

- When omitted, a standby tool is called.
- TOO Delivers the spindle tool.

B_ : 2nd auxiliary function.
See Section 10-4.

* For an additional axis, see Section 11-8, (4).

Due to a reduced cycle time, the 3rd reference point changes depending on the tool used.

- When both spindle and standby tools are ordinary ones;
  3rd reference point = 2nd reference + Parameter 5171
- When either spindle or standby tool or both of them are large-diameter tools;
  3rd reference point = 2nd reference + Parameter 5172

The following tools are large-diameter tools.

- Large-diameter, U-axis, angular, and special boring tools

(2) Movements

	Command	Operation
①	G90 G53 G00 Z_ ;	The Z-axis moves to the 2nd reference point.
②	G90 G53 G00 X_ Y_ M106 { T_ } {B2_ } ;	The X-axis moves to the 3rd reference point. The Y-axis moves to the 2nd reference point. If the specified tool is the same as the spindle tool; • Jumps to the final step • Alarm if PRA5103 #3 = 1 • Does not output M106 2nd auxiliary function: Output when specified
③	G03 P2 X0 Y0 Z0 M06 {Additional axis} ;	X-, Y-, and Z-axis to the 2nd reference point. The additional axis follows the parameter 5109. Tool change.
④	G90 G53 G00 X_ M107 ;	X-axis to the 3rd reference point

(*1) If the tool called and spindle tool are the same, processing will end in ② .(Does not output the T-code and M106 in ②)

(3) Associated parameters

No.			Description
1226	X		ATC approach position (Machine coordinates [mm])
	Y		ATC change position (Machine coordinates [mm])
	Z		ATC Change position (Machine coordinates [mm])
5103	#1		Set 0.
	#2		Set 0.
	#3	=1	Results in an alarm if the specified tool and the spindle tool are the same.
		=0	Does not result in an alarm.
5171			ATC position of the standard too, Distance from the 2nd reference point
5172			ATC position of the large-diameter tool, Distance from the 2nd reference point

10-8-5 ATC Canned Cycle, Type I (Initial HS500)

(1) Command format

{ T_ } M06 { B_ } ; { ~ } → Omissible

T_ : Tool number to be called

- When omitted, the standby tool is called.
- T00 Delivers the spindle tool.

B_ : 2nd auxiliary function.

See Section 10-4.

※ For an additional axis, see Section 11-8, (4).

(2) Movements (Conventional type)

	Command	Movement
①	{ T_ }	Output only when a T-code is specified. Not output if same as the spindle tool.
②	G91 G00 Z_ M106;	The Z-axis moves to the 3rd reference point. M106 is not output if the specified tool is the same as the spindle.
③	G91 G00 Y_ ;	The Y-axis moves to an approach point to the 2nd reference point. *1
④	G91 G00 X_ Y_ { B2 }; {Additional axis} ;	X- and Y-axis to the 2nd reference point. The additional axis follows the parameter 5109. 2nd auxiliary function output. If the called tool is the same as the spindle tool, processing will end.
⑤	G91 G00 X_ Y_ ;	X- and Y-axis to the 3rd reference point
⑥	G91 G30 X0 Y0 M06 ;	X-, Y-, and Z-axis to the 3rd reference point, Return signal ON, M06 output
⑦	G91 X_ Y_ M107;	X- and Y-axis to the 2nd reference point, M107 output

(*1) Step ③ may not be executed depending on the X and Y coordinate values at start.
When the MAS is available (PRA5103 #7 = 1)

- Step ③ is not executed.
- Each axis in Steps ② and ④ move at the same time. Fully note that they may interfere depending on the shape of the work or jig.

(3) Associated parameters

No.			Description	
1226	X		ATC approach position	(Machine coordinates [mm])
	Y		ATC approach position	(Machine coordinates [mm])
	Z		Unused	(Machine coordinates [mm])
1227	X		Tool change position	(Machine coordinates [mm])
	Y		Tool change position	(Machine coordinates [mm])
	Z		Tool change position	(Machine coordinates [mm])
5103	#1		Set 0.	
	#2		Set 0.	
	#7	=1	MAS available type; X-, Y-, and Z-axis simultaneous move	
		=0	Conventional type	

10-8-6 ATC Canned Cycle, Type J (VS)

(1) Command format

{ T_ } M06 { B_ } { Y_ } ; { ~ } → Omissible

T_ : Tool number to be called

- When omitted, the standby tool is called.
- T00 Delivers the spindle tool.

B_ : 2nd auxiliary function.

See Section 10-4.

※ For an additional axis, see Section 11-8, (4).

(2) Movement

	Command	Movement
①	{ T_ } ;	Output only when a T-code is specified. Output only when a T-code is specified. Not output if same as the spindle tool.
②	G91 G00 Z_ M106;	The Z-axis moves to the 2nd reference point. M106 is not output if the specified tool is the same as the spindle.
③	G91 G00 X_ { Y_ }; { B2} {Additional axis} ;	X- and Y-axis to the 2nd reference point. The additional axis follows the parameter 5109. 2nd auxiliary function output. If the called tool is the same as the spindle tool, processing will end.
④	G91 G00 X_	X-axis to the 3rd reference point
⑤	G91 G30 X0 Z0 M06 ;	X-axis to the 3rd reference point, Return signal ON, Z-axis to the 2nd reference point, Return signal ON, M06 output
⑦	G91 X_ M107;	X-axis to the 2nd reference point, M107 output

If the MAS is made available (PRA5103 #7 = 1), each axis in Steps ② and ③ moves simultaneously. Fully note that they may interfere depending on the shape of the work or jig.

(3) Associated parameters

No.			Description	
1226	X		ATC approach position	(Machine coordinates [mm])
	Y		ATC approach position	(Machine coordinates [mm])
	Z		Unused	(Machine coordinates [mm])
1227	X		Tool change position	(Machine coordinates [mm])
	Y		Tool change position	(Machine coordinates [mm])
	Z		Tool change position	(Machine coordinates [mm])
5103	#1		Set 0.	
	#2		Set 0.	
	#7	=1	MAS available type; X-, Y-, and Z-axis simultaneous move	
		=0	Conventional type	

10-8-7 ATC Canned Cycle, Type K(HS630)

(1) Command format

M06 { T_ } { B_ } ; { ~ } → Omissible

T_ : Tool number to be called

- When omitted, the standby tool is called.
- T00 delivers the spindle tool.

B_ : 2nd auxiliary function. See 10-4.

※For an additional axis, see Section 11-8, (4).

(2) Movements

	Command	Movement	
①	G91 G00 Z_ { T_ }	Moves the Z-axis to the 2nd reference point.	*1
②	G91 G00 Y_ ;	Retracts the Y-axis. *2	*2
③	G91G00 X_ Y_ M106 {B2_};	Moves the X-axis to the 2nd reference point. (2nd + Parameter 5172 position in case of a large-diameter tool) Moves the Y-axis to the 2nd reference point. If the specified tool and the spindle one are the same; • Jumps to the last step. • Issues an alarm if PRA5103 #3 = 1. • Does not output M106. 2nd auxiliary function: Output when specified.	
④	G91 G00 X_ { additional axis } ;	Moves the X-axis to the 3rd reference point. The additional axis abides by Parameter 5109.	
⑤	G30 P3 X0 P2 Y0 M06;	Turns on the X-, Y-, and Z-axis lamps. Changes tools.	
⑥	G91 G00 X_ M107;	Moves the X-axis to the 2nd reference point. (2nd + Parameter 5172 position in case of a large-diameter tool)	

(*1) Movement ② may not be executed depending on the path.
When accommodating the MAS (PRA5103 #7 = 1);

- Movement ② is not executed.
- Each axis in ① and ③ moves simultaneously. **Note fully that they may interfere depending on the shape of the workpiece or jig.**

(*2) If the tool called and spindle tool are the same, processing will end in ③ . (Does not output the T-code in ① and M106 in ③)

(3) Related parameters

No.			Description
1226	X		ATC approach position (Machine coordinate [mm])
	Y		ATC approach position (Machine coordinate [mm])
	Z		Unused (Machine coordinate [mm])
1227	X		Tool change position (Machine coordinate [mm])
	Y		Tool change position (Machine coordinate [mm])
	Z		Tool change position (Machine coordinate [mm])
5013	#1		Set 0.
	#2		Set 0.
	#3	=1	Issues Alarm 162 (#008) when the specified tool and spindle tool are the same.
		=0	Does not issue the alarm.
	#7	=1	MAS accommodating type: Moves the X-, Y-, and Z-axis simultaneously.
		=0	Conventional type

10-8-8 ATC Canned Cycle, Type L (New HS500)

(1) Command format

M06 { T_ } { B_ } ; { ~ } → Omissible

T_ :Tool number to be called

- When omitted, the standby tool is called.
- T00 delivers the spindle tool.

B_ : 2nd auxiliary function. See 10-4.

※ For the additional axis, see Section 11-8, (4).

(2) Movements

	Command	Movement
①	G91 G00 Z_ M106 { T_ }	Moves the Z-axis to the 2nd reference point. If spindle tool = specified tool; • Does not execute Steps 4-6. • Issues an alarm if PRA5103 #3 = 1. Output a T-code only when specified.
②	G91 G00 Y_ ;	Retracts the Y-axis.
③	G91G00 X_ Y_ {B2_} { additional axis }	Moves the X-axis to the 2ndreference point. Moves the Y-axis to the 2nd reference point. 2nd auxiliary function: Output when specified. The additional axis abides by Parameter 5109.
④	G91 G00 X_	Moves the X-axis to the 3rd reference point.
⑤	G30 P3 X0 P2 Y0 M06;	Moves the X-axis to the 2nd reference point. Moves the Y- and Z-axis to the 2nd reference point. Changes tools.
⑥	G91 G00 X_ M107;	Moves the X-axis to the 2nd reference point. Closes the shutter.

(*1) Movement ② may not be executed depending on the path.
When accommodating the MAS (PRA5103 #7 = 1);

- Movement ② is not executed.
- Each axis in ① and ③ moves simultaneously. **Note fully that they may interfere depending on the shape of the workpiece or jig.**

(*2) If the tool called and spindle tool are the same, processing will end in ③. (Does not output the T-code and M106 in ③)

10-8-9 ATC Canned Cycle, Type M (VS 16-tool)

(1) Command format

M06 { T_ } ; { ~ } → Omissible

(2) Movements

Note: A number enclosed by the brackets (<>) represents an origin number.
T-code command = Spindle T-code or Spindle T-code = For standby T-code

	Command	Movement
①	{ T_ } ;	Output only when a T-code command is given
②	M06	Outputs M06.

When the Spindle T-code Is Not 0;

	Command	Movement
①	{ T_ } ;	Output only when a T-code command is given
②	M06	Output M06.
③	M123	Outputs M123.
④	Z<3>	Moves the Z-axis to the 3rd reference point. Executes the movement ⑤ simultaneously when Parameter 5103, #7 is “1”.
⑤	X<1> M106	Moves the X-axis to the 1st reference point. Outputs M106. Moves to the point which assumes this value to be the coordinates, when Parameter 5173 is not “0”. Outputs M106 after moving the X-axis, when the X-axis coordinate value is greater than the value of Parameter 5173 at the time of M06 execution.
⑥	M124	Outputs M124.
⑦	X<3>	Moves the X-axis to the 3rd reference point.
⑧	M129	Outputs M129.
⑨	Z<1>	Moves the Z-axis to the 1st reference point. Jumps to ⑬ when a T-code command is “0” or a standby T-code is “0”.
⑩	M125	Outputs M125.
⑪	M126	Outputs M126.
⑫	Z<3>	Moves the Z-axis to the 3rd reference point.
⑬	M128	Outputs M128.

⑭	M127	Outputs M127.
⑮	X<1> M107	Moves the X-axis to the 1st reference point. Outputs M107. Moves to the point which assumes this value to be the coordinates, when Parameter 5173 is not “0”.

When the Spindle T-code Is “0”;

	Command	Movement
①	{ T_ } ;	Output only when a T-code command is given
②	M06	Outputs M06.
③	M125	Outputs M125.
④	Z<1>	Moves the Z-axis to the 1st reference point.
⑤	X<1> M106	Moves the X-axis to the 1st reference point. Moves to the point which assumes this value to be the coordinates, when Parameter 5173 is not “0”. Outputs M106 after moving the X-axis, when the X-axis coordinate value is greater than the value of Parameter 5173 at the time of M06 execution.
⑥	X <3>	Moves the X-axis to the 3rd reference point.
⑦	M129	Outputs M129.
⑧	M126	Outputs M126.
⑨	Z<3>	Moves the Z-axis to the 3rd reference point.
⑩	M128	Outputs M128.
⑪	M127	Outputs M127.
⑫	X<1> M107	Moves the X-axis to the 1st reference point. Outputs M107. Moves to the point which assumes this value to be the coordinates, when Parameter 5173 is not “0”.

10-8-10 ATC Canned Cycle, Type N (MS400H)

(1) Command format

{ T_ } M06 { B_ } { S_ } ; { ~ } → Omissible

T_ : Tool number to be called

- When omitted, the standby tool is called.
- T00 delivers the spindle tool.

B_ : 2nd auxiliary function. See 10-2.

※ For the additional axis, see 11-8 (4).

An alarm is issued when an axis command is given is the same block as an M06 command.

(2) Movements

When the Spindle Tool Differs from the Specified Tool;

	Command	Movement
①	(T_) m108;	ATC shutter open, Positioning, Coolant stop one-shot. Outputsa T-code command only when it is specified
②	G91 G30 P2 Z0 ;	Moves the Z-axis to the 2nd reference point.
③	G91 G30 P2 Z0 Y0 M06 (B_) ;	Moves the X- and Y-axis to the 2nd reference point and outputs M06. Outputs the 2nd auxiliary function only when it is specified.;
④	(S_);	Outputs an S-code command only when it is specified.

When Accommodating the MAS (PRA5103, #7 = 1);

- Each axis in ② and ③ moves simultaneously. **Note fully that they may interfere depending on the shape of the workpiece or jig.**

When the Spindle Tool Equals the Specified Tool;

	Command	Movement
①	G91 G30 P2 Z0 M15 ;	Moves the Z-axis to the 2nd reference point. Stops the spindle and coolant.
②	(B_) (S_) ;	Outputs B- and S-code commands only when they are specified.

(3) Related parameters

No.		Description
1226	X	ATC change position (Machine coordinates [mm])
	Y	ATC change position (Machine coordinates [mm])
	Z	ATC change position (Machine coordinates [mm])
5103	#1	Set “0”.
	#2	Set “0”.

10-9 High-Speed Machining Cycle

10-9-1 Trochoid Cycle (G334)

To perform fluting in circular cutting through use of an end mill.

(1) Command Format

G334 X_ Y_ Z_ I_ J_ K_ A_ W_ R_ C_ P_ Q_ D_ F_ V_ ;

X, Y : Coordinate value of the reference point (When in default, current position)

Z : Z-axis coordinate value (When in default, current position)

I, J : Work end position (vector volume from X/Y)

K : Cutting width per time

A : Circular radius (Circular rotating direction is assigned by A code.)

W : Flute width (When in default, A x 2)

R : Circular flute radius (When in default, linear flute)

C : Cutting direction for circular flute (C2 : CW, C3 : CCW)

P, Q : Approach volume of X/Y axes (When in default, X and Y taken as work start position)

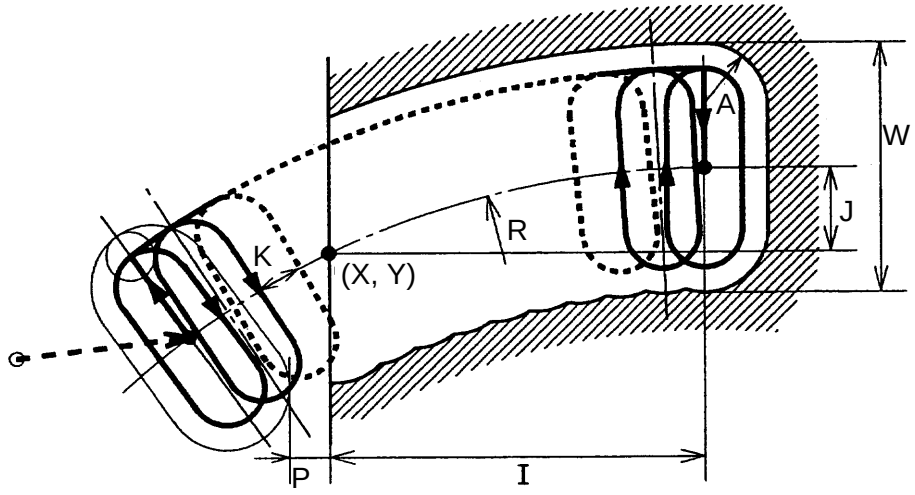
D : Cutter compensation number

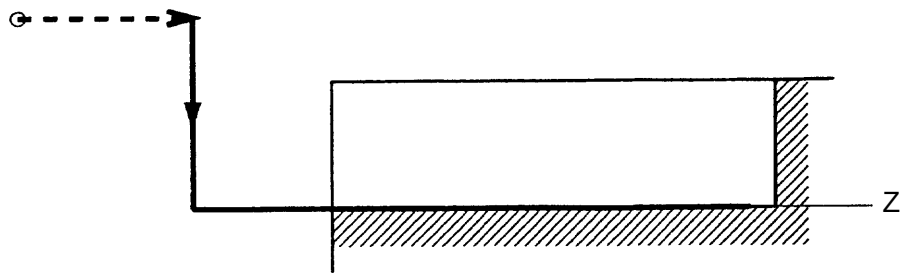
F : Cutting feed speed (When in default, previously commanded F value)

V : Z-axis lowering speed (When in default, V=F)

(Note 1) IO and J0 are omissible.

(Note 2) For a positive A value, the circular becomes CW. For a negative A value, it becomes CCW. Also, for A=0, alarm takes place.

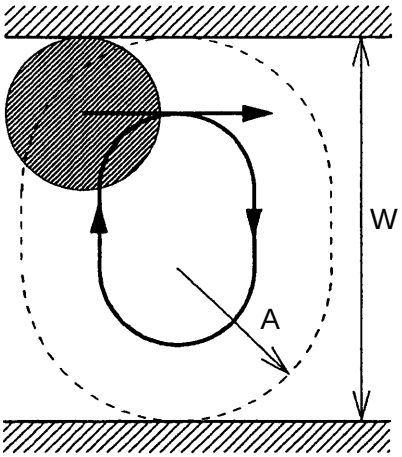




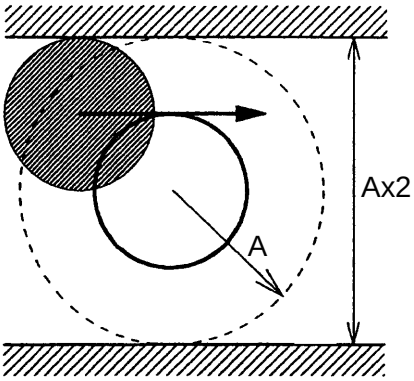
(2) Flute Width (W)

Flute width is commanded with an address W. With W not assigned, flute width gets equal to (A x 2).

(a) With W command

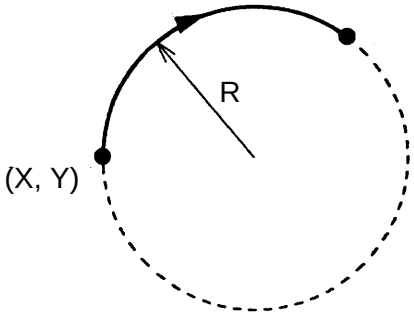
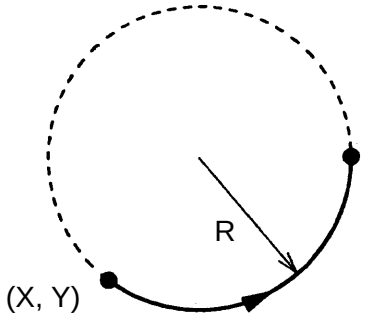
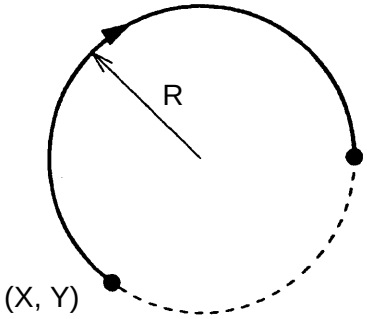
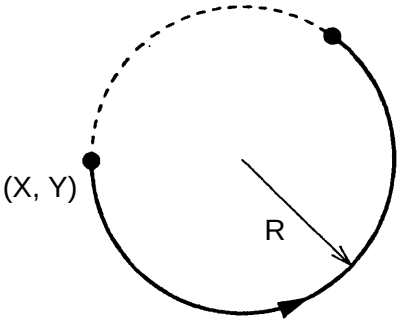


(b) Without W command



(3) Circular Flute (R, C)

When addresses R and C are commanded, a circular flute is obtained. The circular flute radius is assigned by address R and the rotating direction by address C.A positive R value results in an arc of 180° or smaller, while a negative R value in an arc of 180° or larger. The circular rotating directions are CW for C2 and CCW for C3.

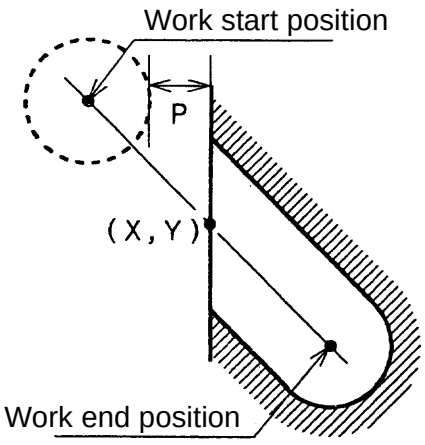
	C2	C3
R+		
R-		

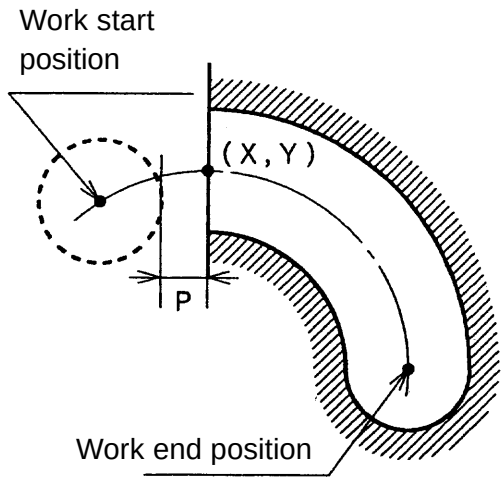
(4) Approach Volume (P, Q)

When address P or Q being commanded, work start position is automatically calculated based on the reference points on X and Y axes. Without assignment of P andQ, the work start position is taken at the reference points of X/Y axes.

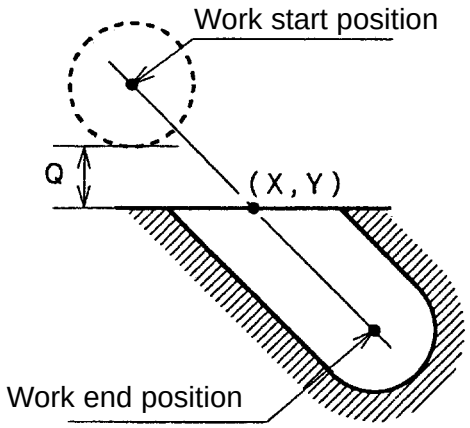
As the work start position for a circular flute is located on the circumference, it may happen depending on the P or Q command value that the work start position cannot be calculated.

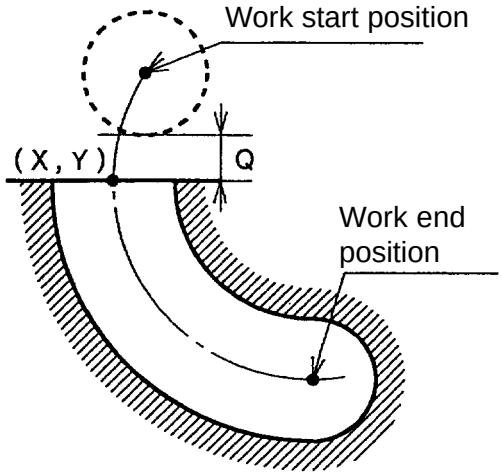
(a) With P command:



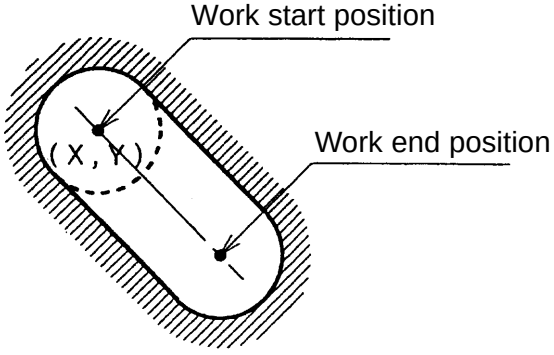


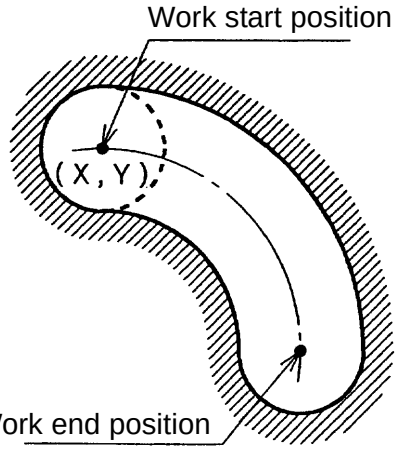
(b) With Q command:





(c) Workout P or Q





(5) Plane of Trochoid Cycle

Planes for the trochoid cycle are assigned with G17, G18, and G19:

G17 : XY plane

G18 : ZX plane

G19 : YZ plane

(6) Cautions

(a) When using G334 command, set the cutter compensation to Cancel (G40) state.

(b) G334 is a non-modal G code.

(c) Without assignment of D and F, the previously set D and F are made valid.

(7) Associated Parameters

No. 5101, #0	=	0	Plane selection always applies to XY plane.
		1	Plane selection conforms to G17 to G19 command.

(8) Associated Alarms

No. 220	Trochoid cycle command error
(#001)	I command = 0 and J command = 0
(#002)	Without K command or K command ≦ 0
(#003)	Without A command or A command = 0
(#004)	[A command] < K command
(#005)	[A command] ≦ tool radius
(#006)	Without C command, or neither C2 nor C3
(#007)	W command < A x 2
(#008)	Center position of circular flute not calculated
(#009)	Work start position not calculated
(#010)	P command < 0
(#011)	Q command < 0

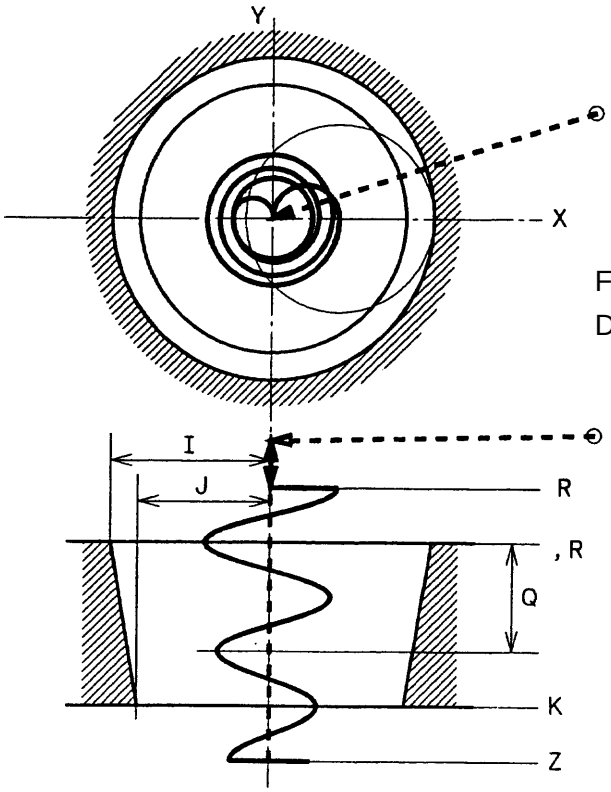
10-9-2 Helical Drilling Cycle (G812, G813)

To perform drilling in helical interpolation through use of an end mill. G812/G813 remain valid until it is cancelled with a modal G code (09 group).

(1) Command Format

$$\begin{bmatrix}G812\\G813\end{bmatrix} \quad \begin{bmatrix}G98\\G99\end{bmatrix} \quad X_\ Y_\ Z_\ R_\ ,\ R_\ I_\ J_\ K_\ Q_\ D_\ ,\ C_\ L_\ F\ ;$$

G812 : Helical drilling cycle (CW)
G813 : Helical drilling cycle (CCW)
G98 : Initial point returning
G99 : R point returning
X, Y : Drilling position (When in default, the current position)
Z : Z point position
R : R point position (When in default, the current position)
,R : ,R point position (When in default, ,R = R)
I : Circular radius in , R point position (For a negative value, raised from Z to R point)
J : Circular radius in K point position (When in default, J = I)
K : K point position (When in default, K = Z)
Q : Cut depth per helical turn (For a negative value, without full circle cutting at the hole bottom.)
D : Cutter compensation number
,C : Ball radius of ball end mill (When in default, square end mill)
L : No. of times of repetition (When in default, L = 1)
F : Cutting feed speed (When in default, the previously commanded F)



Full line: Cutting feed
Dotted line: Quick feed

(2) Movement

(a) Where I > 0 and Q > 0;

① Shifted in quick feed to the X/Y axis drilling place.
② Shifted in quick feed to R point on Z axis.
③ Circular cutting to the X/Y axis cutting start point.
④ Conical cutting to Z point. (Note 1)
⑤ X/Y axis full circle cutting.
⑥ Circular cutting to X/Y axis drilling position.
⑦ Shifted in quick feed to the Z axis return point.

(b) Where I > 0 and Q < 0;

① Shifted in quick feed to the X/Y axis drilling position.
② Shifted in quick feed to R point on Z axis.
③ Helical cutting to the X/Y axis cutting start point.
④ Conical cutting to (Z point + Q/4) (Note 1)
⑤ Helical cutting to the X/Y axis drilling position.
⑥ Shifted in quick feed to the Z axis return point.

(c) Where I < 0;

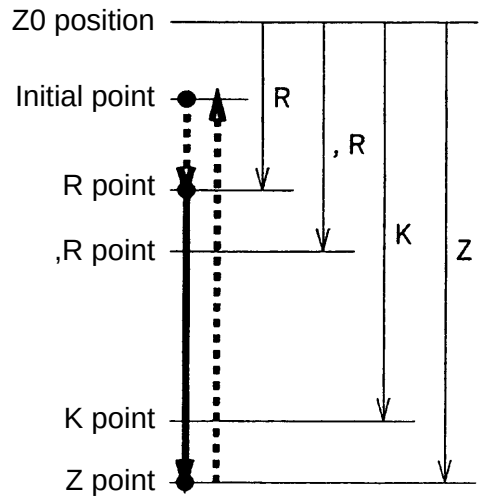
① Shifted in quick feed to the X/Y axis drilling position.
② Shifted in quick feed to R point on Z axis.
③ Shifted in quick feed to Z point on Z axis.
④ Helical cutting to the X/Y axis cutting start point.
⑤ Conical cutting to (R point -Q/4) (Note 1)
⑥ Helical cutting to the X/Y axis drilling position.
⑦ Shifted in quick feed to the Z axis return point.

(Note 1) Omission of J command results in helical cutting.

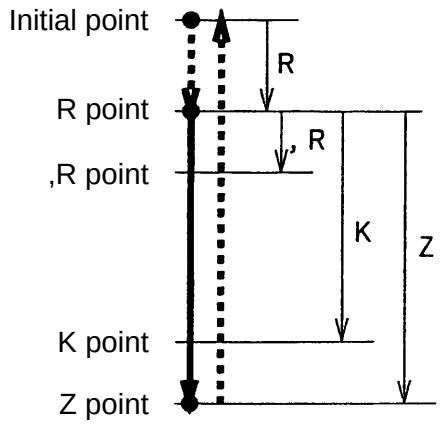
(3) R Point/ Z Point/ ,R point/ K Point

When G90 or G91 command is given, Points R, Z, ,R, and K become as follows:

(a) G90



(b) G91



(4) Return Point

The return point of the helical drilling canned cycle is commanded with the following G code:

G98: Returned to the initial point level

G99: Returned o R point level

(Note 1) The initial point indicates the drilling axis position when mode has changed into Helical Drilling Cycle mode from Cancel state.

(5) Drilling Position Planes and Drilling Axes

Drilling position planes are assigned with G17, G18 and G19 commands. The drilling axes include X, Y, and Z axes which do not constitute the drilling position plane.

G17: X/Y plane The drilling axis is Z.

G18: Z/X plane The drilling axis is Y.

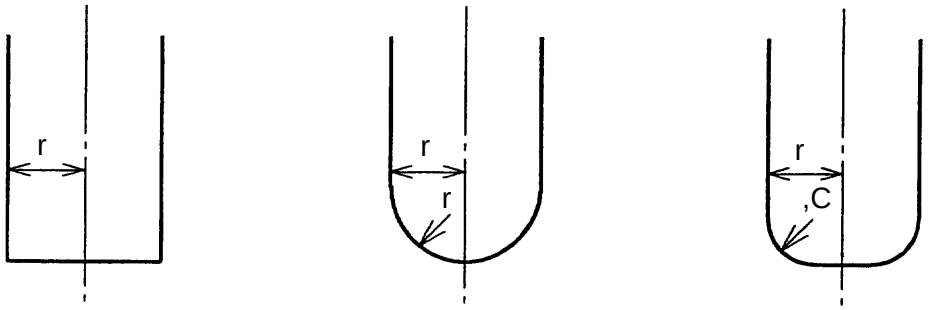
G19: Y/Z plane The drilling axis is X.

(Note 1) To change over a drilling position plane, first cancel the helical drilling cycle.

(6) Ball Radius of Ball End Mill

Ball radius of a ball end mill is assigned by address ,C. Tool radius volume r is assigned by a cutter compensation number (address D).

(a) Without ,C command (b) ,C0 (c) ,C_



(7) Cautions

(a) When using G812 or G813 command, be sure to set state to Cutter Compensation Cancel (G40).

(b) Z/ R/ ,R/ I/ J/ K/ Q/ ,C are held modal while in helical drilling cycle.
Where no assignment has been made for K, ,R, or J while Z, R, and I are assigned, these K, ,R, and J are considered to have been omitted.

(c) The number of times of repetition (L), when not assigned, is taken as L= 1.
With L= 0 assignment, shifting takes place to the drilling position and drilling is made invalid.

(d) With no assignment of D and F, the previously commanded D and F are made valid.

(e) Tangent speed of a circular gets the commanded rate. As the arc radius is reduced gradually in conical cutting, drilling axis speed may be increased. Pay attention, therefore, so that drilling axis speed exceeds the limit value.

(f) Conical cutting, for which the circular center and the radius are changed by each circular diving angle having been set with parameters, cannot achieve a perfect cone in the strict sense of the word.

(8) Associated Parameters

No. 5117 Circular dividing angles (1 to 90)

(9) Associated Alarms

No. 221	Helical drilling cycle command error
(#000)	Without Z command
(#001)	Without I command or I command = 0
(#002)	J command ≦ 0
(#003)	I command ≦ tool radius
(#004)	J command ≦ tool radius
(#005)	Without Q command or Q command = 0
(#006)	,R point is erroneous.
(#007)	K point is erroneous.
(#008)	Calculation error
(#009)	Conical cutting error

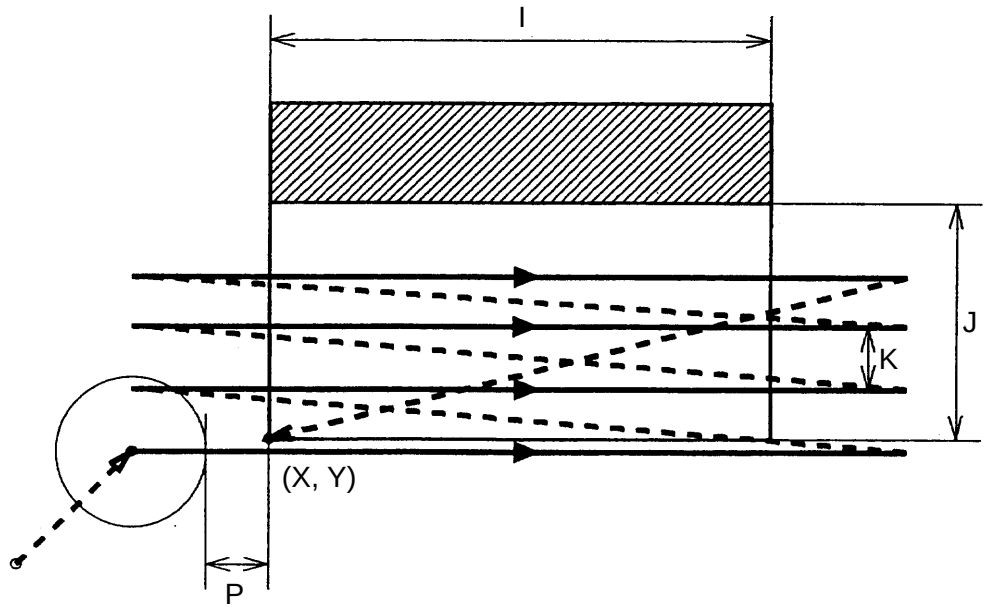
10-9-3 High Speed Side Face Cutting Cycle (G335)

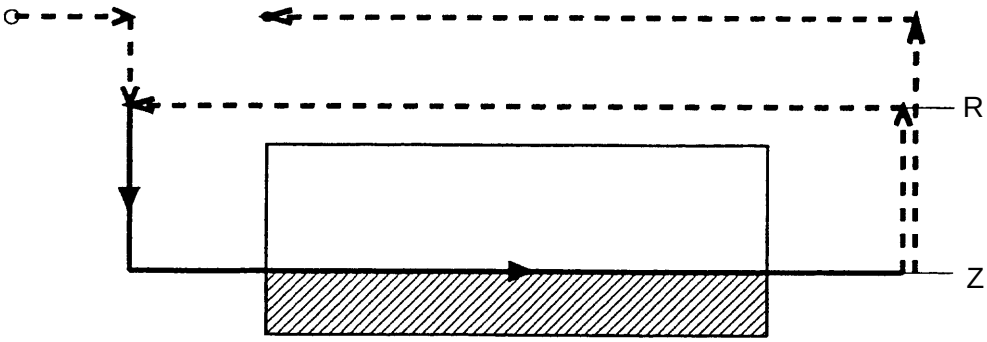
To perform side face cutting through use of an end mill.

(1) Command Format

G335 X_ Y_ Z_ R_ I_ J_ K_ P_ C_ D_ F_ V_ ;

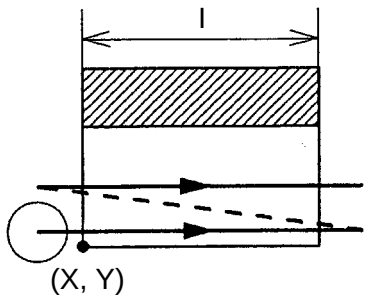
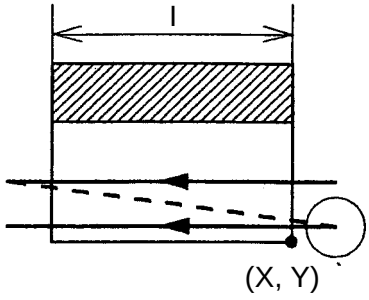
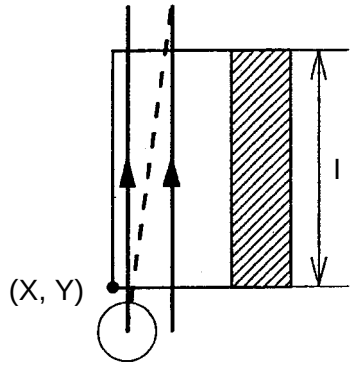
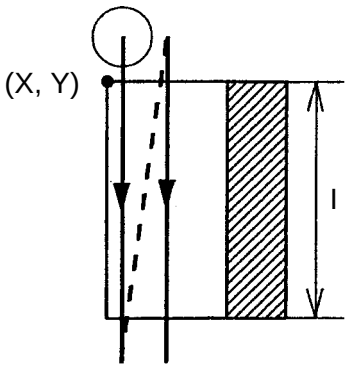
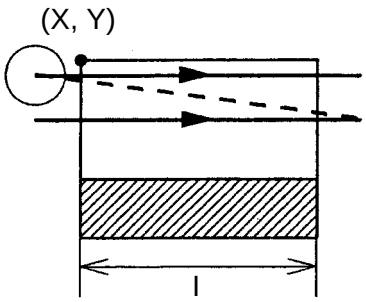
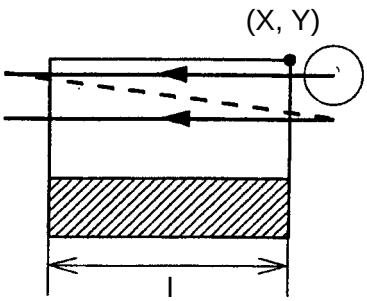
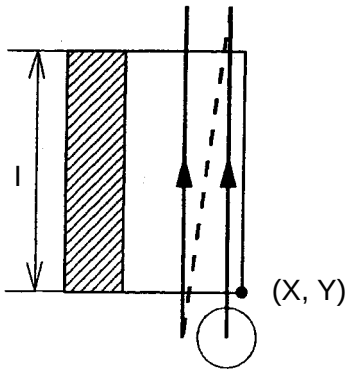
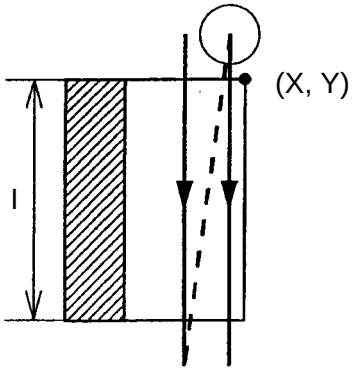
X, Y : Reference point coordinate value (When in default, the current position)
Z : Z point coordinate value
R : R point coordinate value (When in default, the current position)
I : Side face length (Cutting direction is assigned with I code.)
J : Length between a reference point and a side face
K : Plane cutting width (Approach position is assigned by K code.)
P : Approach volume
C : Cutting direction assignment ($1 \leqq C \leqq 4$)
D : Cutter compensation number (When in default, the previously commanded D.)
F : Cutting feed speed (When in default, the previously commanded F.)
V : Z axis lowering speed (When in default, V = F.)





(2) Cutting Start Position and Cutting Direction

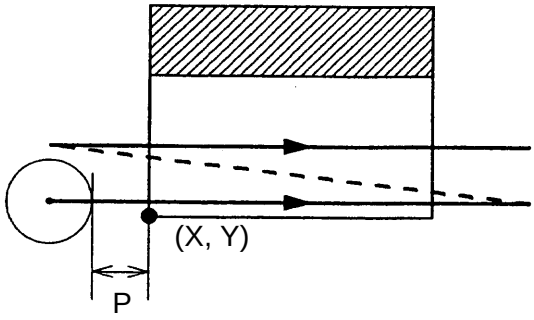
Cutting start position and the direction are assigned through use of address C and I codes.

	I +	I -
C1		
C2		
C3		
C4		

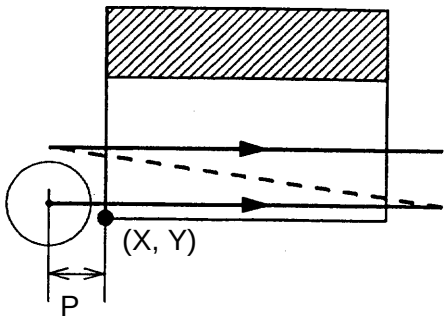
(3) Approach Position

Approach position is changed by the address K code. With it being negative, the cutter center is located outside only by the distance equal to the approach volume.

(a) K+



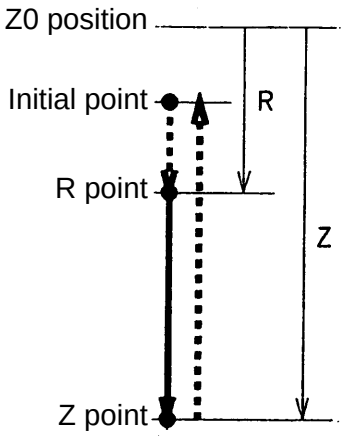
(b) k-



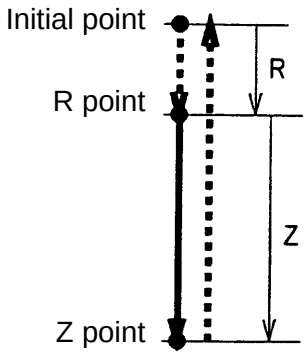
(4) R point and Z point

With G 90/ G91 command, R and Z points are made as follows:

(a) G90



(b) G91



(5) Planes for High Speed Face Cutting Cycle

The planes for the high speed side face cutting cycle are assigned with G17, G18, and G19.

G17 : XY plane
G18 : ZX plane
G19 : YZ plane

(6) Cautions

(a) Be sure to command G335 while tool length compensation is held in Cancel (G40) state.
(b) G335 is a non-modal G code.
(c) With no assignment of addresses D and F, the previously commanded D and F are made valid.
(d) The address J codes are ignored.

(7) Associated Parameters

No. 5101, #0= 0 Plane selection always applies to XY plane.

1 Plane selection conforms to G17 to 19 commands.

(8) Associated Alarms

No. 222 High speed side face cutting cycle command error

(#001) Without Z command

(#002) Without I command or I command = 0

(#003) Without J command or J command = 0

(#004) Without K command or K command = 0

(#005) Without P command

(#006) Without C command or not equal to 1 to 4.

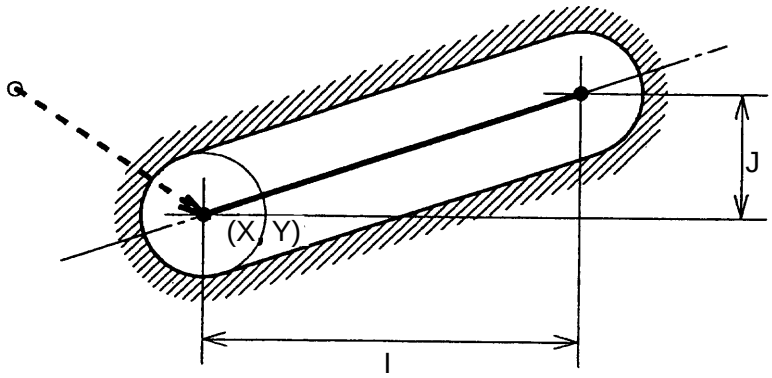
10-9-4 Z Feed Fluting Cycle (G336)

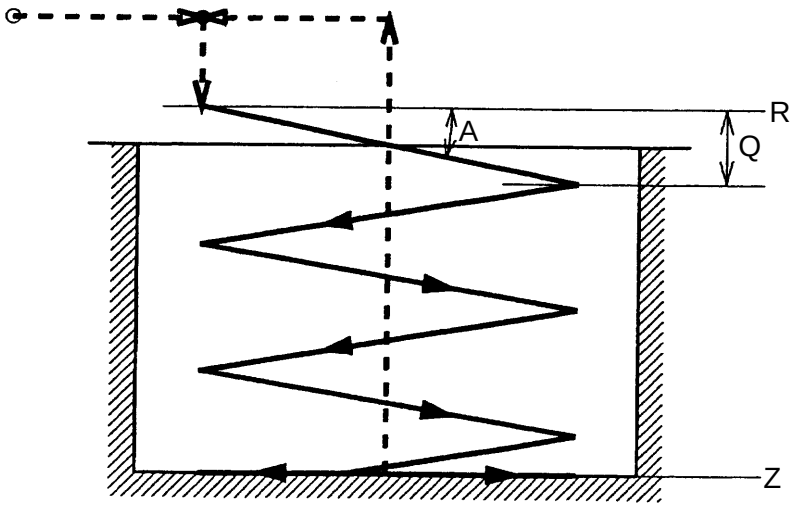
To perform fluting through use of oblique cutting.

(1) Command Format

G336 X_ Y_ Z_ R_ I_ J_ A_ Q_ F_ ;

X, Y : Cutting start point coordinate value (When in default, the current position.)
Z : Z point coordinate value
R : R point coordinate value (When in default, the current position.)
I, J : Cutting end point (Vector amount from X, Y)
A : Z axis cut depth angle (For a negative value, no bottom finishing.)
Q : Z axis cut depth volume (For a negative value, no bottom finishing.)
F : Cutting feed speed (When in default, the previously commanded F.)
(Note 1) I0 and J0 are omissible.
(Note 2) Command either A or Q. When both of them are commanded, Q is given priority.

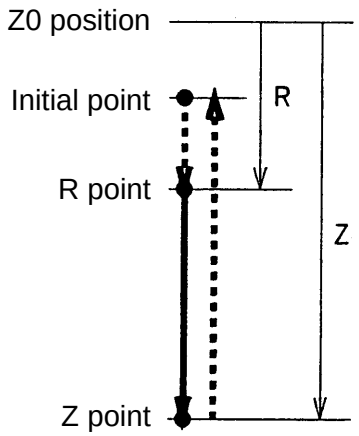




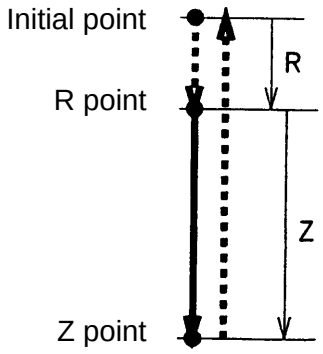
(2) R Point and Z Point

With G 90/ G91 command, R and Z points are made as follows:

(a) G90



(b) G91



(3) Planes for Z Feed Fluting Cycle

The planes for the Z feed fluting are assigned with G17, G18, and G19.

G17 : XY plane

G18 : ZX plane

G19 : YZ plane

(4) Cautions

(a) Command G336 while tool length compensation is held in Cancel (G40) state.

(b) G336 is a non-modal G code.

(c) With no assignment of address F, the previously commanded F is made valid.

(d) When “no bottom finish” is assigned, cutting ends on reaching of z point. Following shifting to the flute center, returning to the initial point takes place.

(5) Associated Parameter

No. 5101, #0 = 0 Plane selection always applies to XY plane.

1 Plane selection conforms to G17 to G19 commands.

(6) Associated Alarms

No. 223 Z feed fluting cycle command error

(#001) Without Z command

(#002) I command = 0 and J command = 0

(#003) A command = 0 and Q command = 0

10-9-5 Corner Pocket Cycle (G337)

To work corners through use of an end mill.

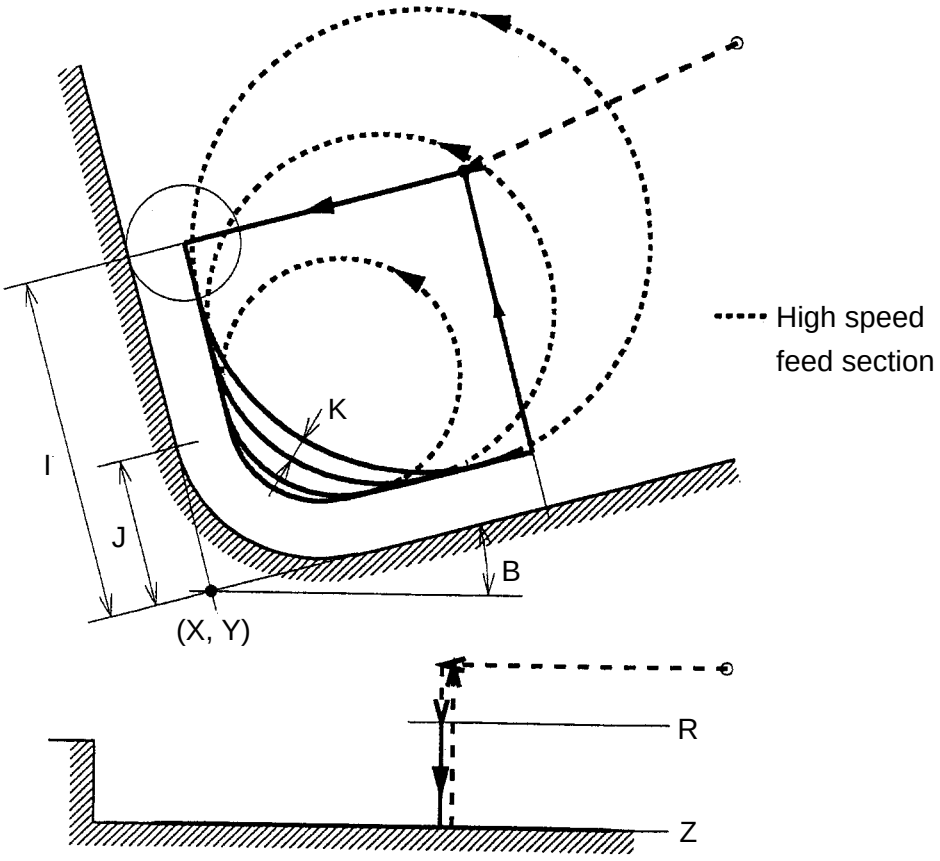
(1) Command Format

G337 X_ Y_ Z_ R_ I_ J_ K_ C_ D_ F_ V_ ;

X, Y : Corner reference point coordinate value (When in default, the current position.)
Z : Z point coordinate value
R : R point coordinate value (When in default, the current position.)
I : Initial corner R radius (Circular rotating direction is assigned by I code.)
J : Final corner R radius (When in default, without corner R.)
K : Cutting width
B : Corner inclination
C : Quadrant of corner ($1 \leqq C \leqq 4$)
D : Cutter compensation number (When in default, the previously commanded D.)
F : Cutting feed speed (When in default, the previously commanded F.)
V : Z axis depth cutting speed (When in default, V = F)

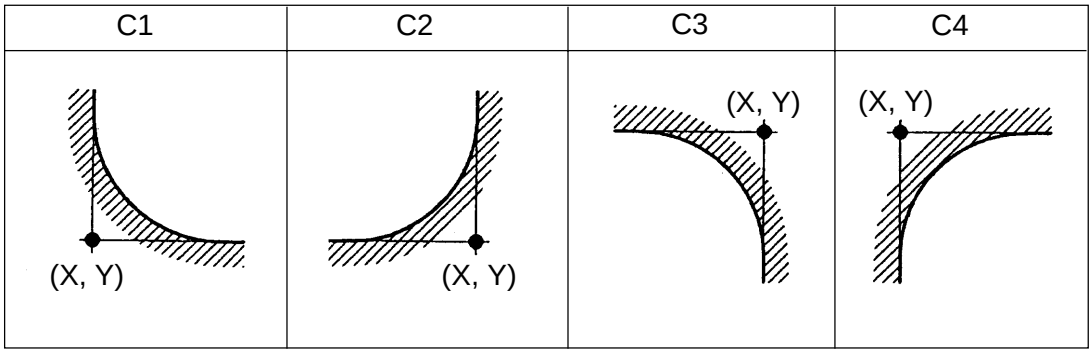
(Note 1) For a positive I value, the arc gets CW. For a negative I value, the arc becomes CCW. Also, for I = 0, alarm takes place.

(Note 2) When both B and C are specified, C has priority.



(2) Quadrant

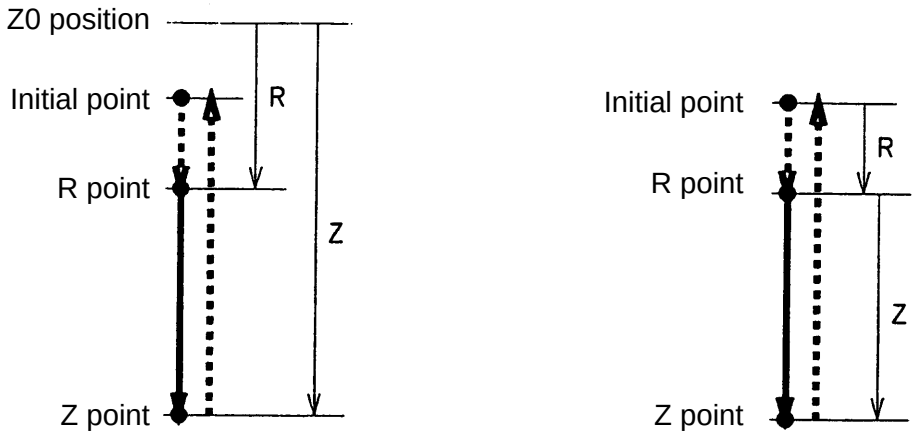
A corner quadrant is assigned through use of an address C value. When the address C is specified, the address B is ignored.



(3) R Point and Z Point

With G 90/ G91 command, R and Z points are made as follows:

(a) G90 (b) G91



(5) Planes for Corner Pocket Cycle

The planes for the corner pocket cycle are assigned with G17, G18, and G19.

G17 : XY plane

G18 : ZX plane

G19 : YZ plane

(5) Cautions

(a) Command G337 while tool length compensation is held in Cancel (G40) state.

(b) G337 is a non-modal G code.

(c) With no assignment of addresses D and F, the previously commanded D and Fare made valid.

(6) Associated Parameters

No. 5101, #0=	0	Plane selection always applies to XY plane.
	1	Plane selection conforms to G17 to G19 commands.
No. 5158		Feed speed for high-speed feed section

(7) Associated Alarms

No. 224	Corner pocket cycle command error
(#001)	Without Z command
(#002)	Without I command or I command = 0
(#003)	[I command] ≦ J command
(#004)	[I command] ≦ tool radius
(#005)	J command < tool radius
(#006)	Without K command or K command ≦ 0
(#007)	C-code command ≠1 to 4

10-9-6 Square Pocket Cycle (G338)

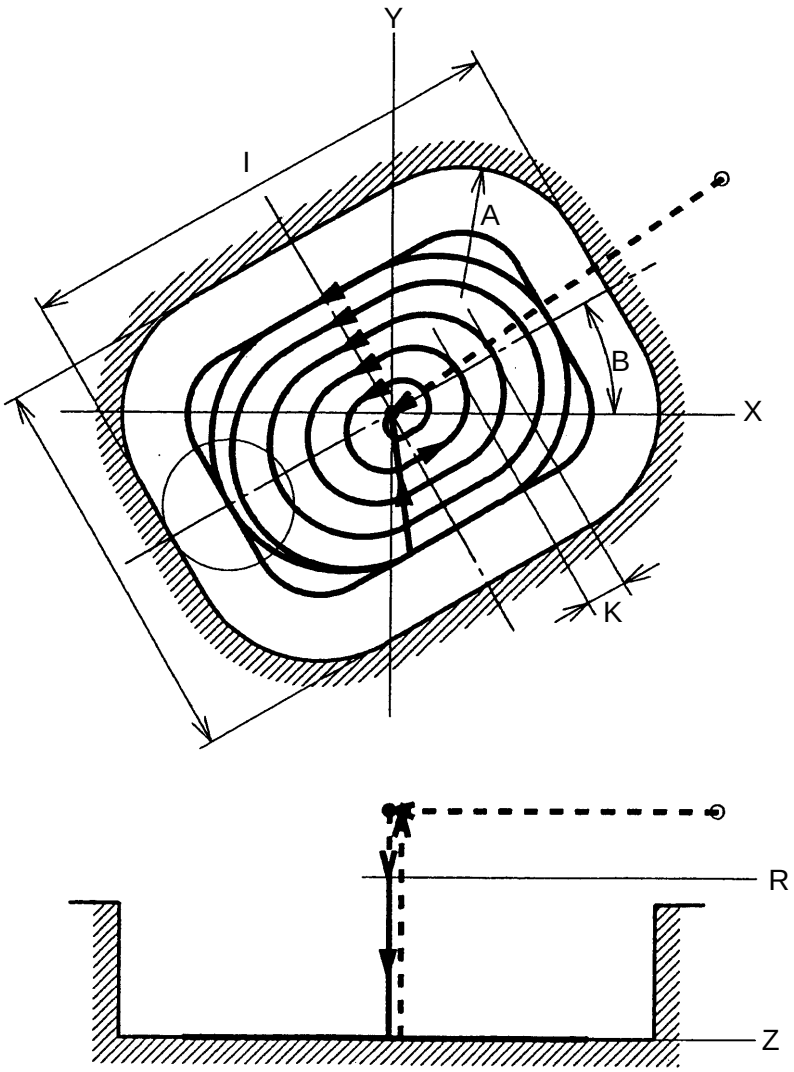
To work the square pocket through use of an end mill.

(1) Command Format

G338 X_ Y_ Z_ R_ I_ J_ A_ B_ K_ D_ F_ V_ ;

X, Y	: Pocket center coordinate value (When in default, the current position.)
Z	: Z point coordinate value
R	: R point coordinate value (When in default, the current position.)
I	: Length of a longer side (Circular rotating direction is assigned by I code.)
J	: Length of a shorter side
A	: Corner R radius (When in default, without corner R.)
B	: Longer side tilt against X axis (When in default, B=0)
K	: Cutting width
D	: Cutter compensation number (When in default, the previously commanded D.)
F	: Cutting feed speed (When in default, the previously commanded F.)
V	: Z axis depth cutting speed (When in default, V = F)

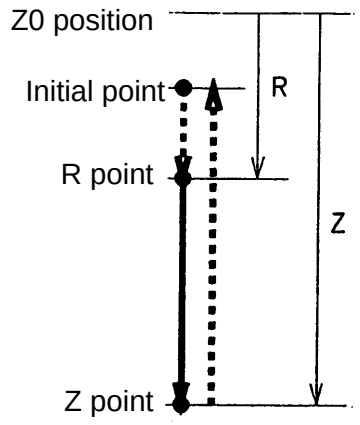
(Note 1) For a positive I value, the arc gets CW. For a negative I value, the arc gets CCW. Also, for I = 0, alarm takes place.



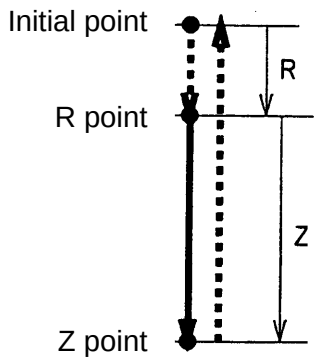
(2) R Point and Z Point

With G 90/ G91 command, R and Z points are made as follows:

(a) G90



(b) G91



(3) Planes for Square Pocket Cycle

The planes for the square pocket cycle are assigned with G17, G18, and G19.

G17 : XY plane

G18 : ZX plane

G19 : YZ plane

(4) Cautions

(a) Command G338 while tool length compensation is held in Cancel (G40) state.

(b) G338 is a non-modal G code.

(c) With no assignment of addresses D and F, the previously commanded D and Fare made valid.

(5) Associated Parameter

No. 5101, #0= 0 Plane selection always applies to XY plane.

1 Plane selection conforms to G17 to G19 commands.

No. 5158 Feed speed for the high-speed feed section

(6) Associated Alarm

No. 225 Square pocket cycle command error

(#001) Without Z command

(#002) Without I command or I command = 0

(#003) [I command] ≦ tool radius x 2

(#004) Without J command or J command = 0

(#005) J command ≦ tool radius x 2

(#006) Without K command or K command ≦ 0

(#007) A command ≦ tool radius

11. COMPENSATION FUNCTION

11-1 Tool Length Compensation (G43, G44, G49)

This command adds the offset amount specified with an H code to or subtracts it from the position of the move end point against one optional axis.

(1) G code

G43 : Tool length compensation in the "+" direction (end point position + offset amount by an H code)

G44 : Tool length compensation in the "-" direction (end point position - offset amount by an H code)

G49 : Tool length compensation cancel

(2) Command format

$\left\{ \begin{matrix} G43 \\ G44 \end{matrix} \right\}$ α _ H _ ; where; α : One optional axis

This command adds the offset amount specified with an H code to the α axis of a subsequent move command (when G43 is used), and subtracts the former from the latter (when G44 is used).

$\left\{ \begin{matrix} G49 \\ H00 \end{matrix} \right\}$;

This command cancels tool length compensation.
When this block is executed, the axes do not move by the cancel amount equivalent to tool length compensation.In the next block onward, the axes move to cancel the offset amount at the time of axis move by absolute programming.

(3) Change of the offset number (H _)

When the offset number is changed halfway the program, the tool offset amount specified with the changed offset number is assumed.

(4) Sample program

	[End Point Position]	[Tool Length Compensation]
G54 G90 G00 X0 Y0 ;		
G40 Z0 H01 ;	Z axis : 200.	+200.offset
G01 Z-30.F500 ;	Z axis : 170.	
Z-100. ;	Z axis : 100.	
G44 G00 Z0 H02 ;	Z axis :-150.	-150.offset
G01 Z-30. F500 ;	Z axis :-180.	
Z-100. ;	Z axis :-250.	
G49 ;	Z axis :-250.	Cancel
G00 Z0 ;	Z axis : 0.	Axis move to cancel : -150.

where ; H01 : 200.
H02 : 150.

(5) By parameter setting, tool length compensation can be performed, fixing it to the Z axis.

$\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$ Z _ H _ ; or $\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$ H _ :

This command adds the offset amount specified with an H code to the Z axis of a subsequent move command (when G43 is used), and subtracts the former from the latter (when G44 is used).
Even when a Z-axis command is omitted, the axes are moved by the offset amount.

$\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$;

This command cancels tool length compensation. When this block is executed, the axes do not move by the cancel amount equivalent to tool length compensation. In the next block onward, the axes move to cancel the offset amount at the time of Z-axis move by absolute programming.

[Sample Program]

	[End Point Position]	[Tool Length Compensation]
G54 G90 G00 X0 Y0 ;		
G43 Z0 H01 ;	Z axis : 200.	+200.offset
G01 Z-30.F500 ;	Z axis : 170.	
H02 ;	Z axis : 120.	+150. offset
Z-100.;	Z axis : 50.	
H00 ;	Z axis : 50.	Cancel
G00 Z0 ;	Z axis : 0.	Axis move to cancel: -150.

Where; H01 : 200.
H02 :150.

(6) Cautions

(a) If the following commands are specified in the block containing G43, G44 or H, an alarm results.

G04, G53, G92. G52. G28, G30

(b) If H is specified independently with tool length compensation applied to any two axes, an alarm results.

(c) Tool length compensation can be applied to up to the first any two axes, one by one. An alarm results if it is applied to three or more axes.

G43 Z _ H _ ; Applies tool length compensation to the Z axis.
G43 W _ H _ ; Applies tool length compensation to the W axis.

(d) When tool length compensation is applied to any one axis, an alarm results if that axis cannot be determined in the G43 or G44 specified block.

(e) When the offset amount is changed, the block containing G43, G44 or H becomes valid first.

(f) The G43, G44 or G49 mode can be set as the reset state by parameter setting.

(g) When the G43 or G44 mode is set as the reset state by parameter setting, the proper offset state results when G43, G44 or H is specified and executed.

(h) If G245 is specified in the G43/G44 mode, an alarm results.

(i) When the RESET button is pressed, note the following points as to the tool length compensation vector.

- In case of the parameter No.5002, #5 = l
  (clear the tool length compensation vector by reset), the tool length compensation vector is cleared by pressing the RESET button.
  Whether the reset state is the G43 or G44 mode, therefore, it is necessary to specify G43, G44 or H to establish the tool length compensation vector.
- In case of the parameter No.5002,#5 = 1 (hold the tool length compensation vector by reset), the tool length compensation vector is held by pressing the RESET button.

(7) Associated parameters

No.5002, #0 Change in offset amount is made effective starting with:

= 0 the block in which D/H codes are next specified.

= 1 the block in which next buffering takes place.

No.5002, #4 = 0 Tool length compensation is always for Z axis.

= 1 Tool length compensation is always for axis assigned by program.

No.5002, #5 = 0 Tool length compensation vector is cleared by resetting.

= 1 Tool length compensation vector is not cleared.

No.3402

#3	#2	Mode Setting
0	0	G49
0	1	G43
1	0	G44

(8) Associated alarms

No.160 Tool length compensation error

(#001) G code which cannot be specified exists in tool length compensation command block.

(#002) When tool length compensation are applied to 2 axes, H has been individually commanded in G43/G44 modes.

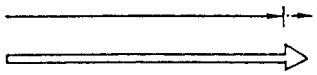
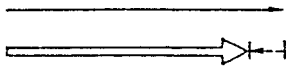
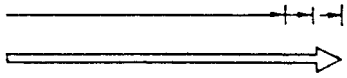
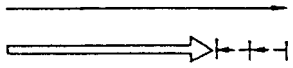
(#003) Tool length compensation are to be applied to 3 axes or more.

(#004) Tool length compensation axis cannot be determined in G43/G44 command block.

11-2 Tool Offset (G45 - G48)

This command extends or contracts the program-given stroke by the specified offset amount. In case of arc, however, tool offset can be specified only for 1/4 and 3/4 circles orthogonal to the axes.

(1) G code

G45	Extends by the offset amount.	
G46	Contracts by the offset amount.	
G47	Extends doubly by the offset amount.	
G48	Contracts doubly by the offset amount.	

However, Program-specified stroke

 Offset amount

Actual stroke

(2) Command format

{G45 / G46 / G47 / G48} ..... D _ ;

This command applies tool offset to the axis specified in the block.

G45 through G48 are one-shot commands.

The offset vector for tool offset is held against the move commands in the blocks following this block.

(3) Sample program

```
G17  G54  G90  G00  X0  Y0 ;
G01  G91  F200 ;
N1  G46  X20. Y20. D01 ;          Contracts the X and Y axes by the offset amount.
N2  G45  X40. ;                   Extends the X axis by the offset amount.
N3  G45  G03  X20. Y20. J20. ;    Extends the X and Y axes by the offset amount.
N4  G45  G01  Y20. ;              Extends the Y axis by the offset amount.
N5  G47  X60. ;                   Extends the X axis doubly by the offset amount.
N6  G47  Y-40. ;                  Extends the Y axis doubly by the offset amount.
N7  G46  X-20.Y-20. ;             Contracts the X and Y axes by the offset amount.
```

(4) Caution

(a) By parameter setting, all H code can be used in place of a B code.

(b) If D00 is specified, tool offset cannot be applied.

(c) If the sign of the offset amount specified with a D code is set to "-", extension/ contraction is reversed.

(d) Tool offset is processed after tool diameter compensation.

(e) If G45 - G48 is specified as a move axis for the simultaneous 2 axes, there may take place a nipping or an uncut part.

(f) A proper profile cannot be obtained if tool offset is applied during the coordinate rotation mode.

(g) G45 - G48 cannot be specified during the tool diameter compensation mode.

(5) Associated parameters

No. 5002, #1 =	0	The offset number for tool offset is a D code.
	1	The offset number for tool offset is an H code.
No. 5002, #2 =	0	Disables an arc command for tool offset.
	1	Enables an arc command for tool offset.

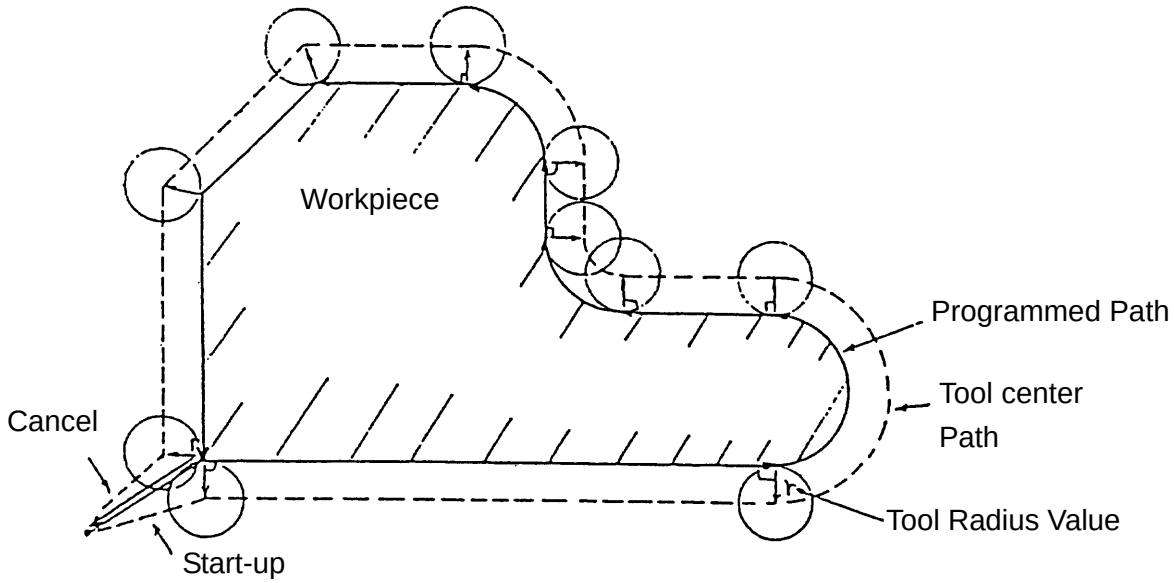
(6) Associated alarms

No. 161	Tool offset was specified in the G02/03 mode.

11-3 Tool Diameter Compensation (G38 - G42)

This command can offset the tool center path outside or inside the programmed path by the tool radius value specified with a D code.

If the tool radius value is specified with the D code when machining the outer figure or inner figure with an end mill, using this function can cut the target profile which had the tool center path offset outside or inside the programmed path by that value.



(1) G code

G40 : Tool diameter compensation cancel

G41 : Tool diameter compensation to left (offset to left)

G42 : Tool diameter compensation to right (offset to right)

G38 : Tool diameter compensation vector hold

G39 : Tool diameter compensation corner arc

(2) Command format

$$\left\{\begin{matrix}G00\\G01\end{matrix}\right\} \quad \left\{\begin{matrix}G41\\G42\end{matrix}\right\} \quad \alpha_\ \beta_\ D_\ \ldots\ ;$$

This command places tool diameter compensation in the start-up state.

$\left\{\begin{matrix}G00\\G01\end{matrix}\right\}$ $\left\{\begin{matrix}G40\\D00\end{matrix}\right\}$ $\alpha_$ $\beta_$;

This command cancels tool diameter compensation.

$\left\{\begin{matrix}G00\\G01\end{matrix}\right\}$ G38 $\alpha_$ $\beta_$;

With this command, tool diameter compensation offset vector can be retained.

$\left\{\begin{matrix}G00\\G01\end{matrix}\right\}$ G38 I_ J_ K_ ;

With this command, tool diameter compensation offset vector can be changed.

G39 I_ J_ K_ ; or G39 ;

This command allows the tool to move along an arc at the corner.

(3) Offset plane

Specify the plane where tool diameter compensation is to be applied, using plane selection(G17, G18, G19).

G17 : X_P Y_P plane
G18 : Z_P X_P plane
G19 : Y_P Z_P plane

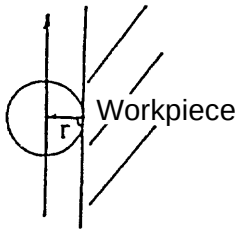
where; X_P : X axis or its parallel axis
Y_P : Y axis or its parallel axis
Z_P : Z axis or its parallel axis

(4) Offset direction

The offset direction for tool diameter compensation is determined by the G41/42 and the sign of the tool radius value specified with a D code.

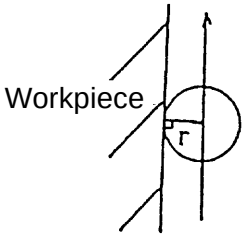
Offset Direction	G Code	Sign of Tool Radius Value
Offset to left	G41	+
	G42	-
Offset tool right	G41	-
	G42	+

Advance Direction



Offset to Left

Advance Direction



Offset to Right

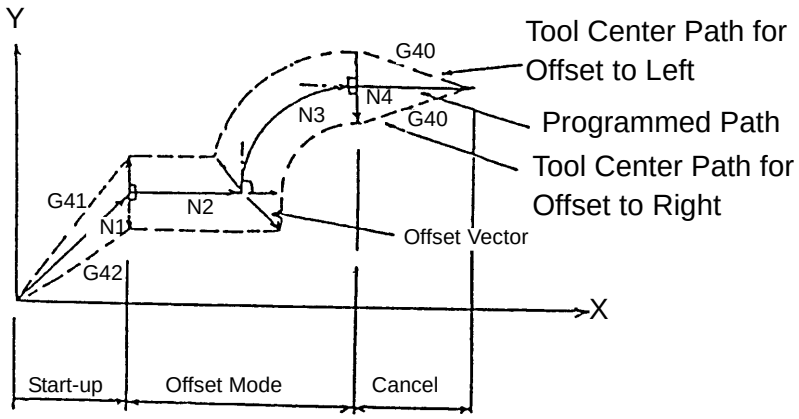
(5) Sample program for tool diameter compensation

[Offset to left] where; D10 = 20.

```
G90  G00  X0  Y0 ;
N1  G17  G01  G90  G41  X50. Y50. D10 F200;        Start-up
N2  X100.;
N3  G02  X150. Y100. 150.;                         Offset mode
N4  G01  G40  X200.;                               Cancel
```

[Offset to right] where; D10 = 20.

```
G90  G00  X0  Y0 ;
N1  G17  G01  G90  G42  X50. Y50. D10 F200;        Start-up
N2  X100.;
N3  G02  X150. Y100. 150.;                         Offset mode
N4  G01  G40  X200.;                               Cancel
```



Start-up : If G41 or G42 is specified in the cancel mode, the tool moves to the position offset by the radius value.

Offset mode : If you enter the tool diameter compensation mode with G41 or G42, the tool moves along the offset path until next G40 or D00 is specified.

Cancel : If G40 or D00 is specified in the offset mode, tool diameter compensation is cancelled and the tool moves to the programmed end point. The offset direction of the previous block will be on the normal of the end point.

(6) Tool diameter compensation vector hold (G38)

During the offset mode, the offset vector of the previous block can be held or its direction can be changed with a G38 command.

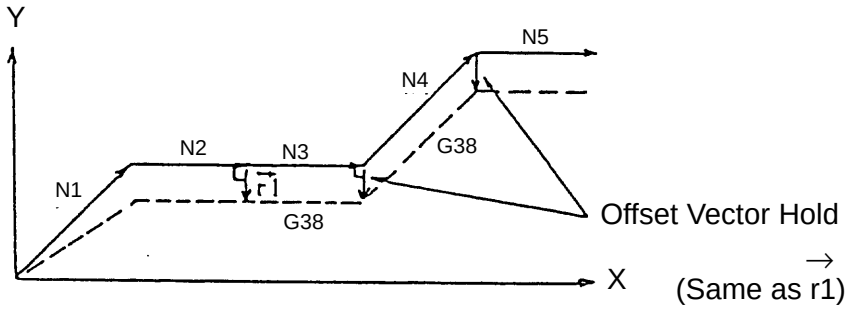
(a) Offset vector hold

$\left\{\begin{matrix} G00 \\ G01 \end{matrix}\right\}$ G38 α_ β_ ;

This command holds the offset vector at the end point position of the previous block without creating the offset vector.

[Sample Program] where; D10 = 20.

```
G54  G90  G00  X0  Y0 ;
N1  G17  G01  G42  X50. Y50. D10 F200 ;
N2  X100.;
N3  G38  X150. ;                          Offset vector hold
N4  G38  X200. Y100. ;                    Offset vector hold
N5  X250. ;
```



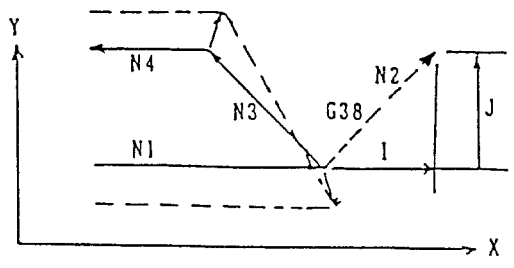
(b) Change of the offset vector

$\left\{\begin{matrix} G00 \\ G01 \end{matrix}\right\}$ G38 I _ J _ K _ ;

With this command, you can change the offset vector located at the end point of the previous block by specifying the end point vector with I, J and K of the offset plane.

[Sample Program] where; D10 = 20.

```
G42  G01  D10  F200 ;
       :                                  Offset vector change
N1   G01 G91 X100. ;
N2   G38 I50. J50. ;
N3   X-50.Y50. ;
N4   X-50. ;
```



(7) Tool diameter compensation corner arc (G39)

During the offset mode, a G39 command allows the tool to move along an arc at the corner.

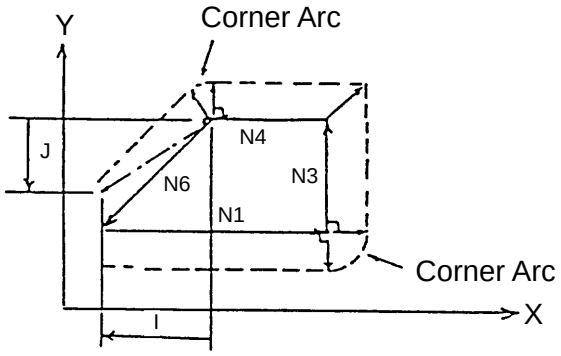
(a) G39 ; If I, J and K are omitted in the block containing G39, the tool moves along a corner arc which allows its end point vector to be perpendicular to the start point of the next block.

(b) G39 I _ J _ K _ ;

If I, J and K are specified in the block containing G39, the tool moves along a corner arc which allows its end point vector to be perpendicular to the vector specified with I, J and K.

[Sample Program] where; D10 = 20.

```
G42  G01  D10  F200 ;
   :
   :
N1  G01 G91 X100. ;
N2  G39  ;                  Corner arc
N3  Y50. ;
N4  X-50. ;
N5  G39 I-50. J-35. ;       Corner arc
N6  X-50. Y-50. ;
```



(8) Cautions

(a) An alarm results if the offset plane is switched during the tool diameter compensation mode.

(b) If there is no axis move command in the 3 blocks counting from the block next to the G41/42 specified one, the program starts from the subsequent axis move command specified block, cancelling the start-up.

(c) An alarm results if a move command is specified in the G39 speified block.

(d) If you specify 3 or more blocks, which do not contain an axis move command, during the offset mode, the workpiece may be partly left uncut or cut too much.

(e) If the following commands are given during the offset mode, an alarm results.

G31 ,
G37
G53 ,
G73, G74, G76, G81 ~ G89
G45 ~ G48
G302 ~ G305
G322 ~ G333

(f) During the offset mode, an interference check (excessive cut) alarm results in the following cases.

(i) When machining the inside of the arc smaller than the tool radius.
(ii) When machining the groove smaller than the tool radius.
(iii) When machining the level difference smaller than the tool radius.

(g) In case of the start-up or cancel mode, if the G code of the same group other than G00 or G01 is given, an alarm results.

(h) If tool diameter compensation is applied to an arc, a full circle may be turned into a short arc and vice versa. In such a case, divide the arc and make programming.

11-3-1 Detailed Description of Tool Diameter Compensation

(1) Start-up

If G41 or G42 is specified in the cancel mode (G40), the tool center path moves to the position offset by the tool radius value.

As described above, an action changing from the cancel mode to the offset mode is called start-up.

[Conditions]

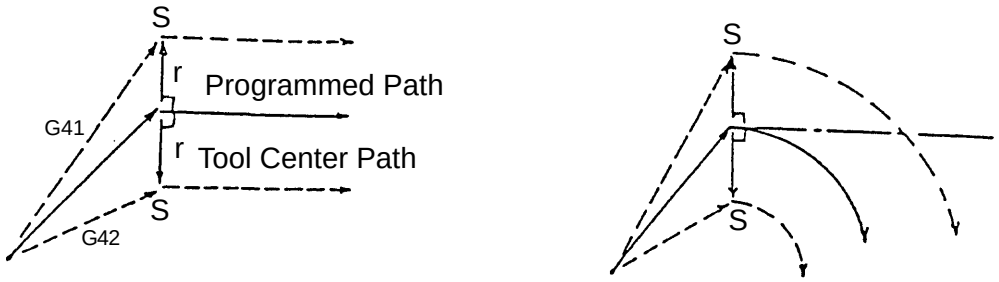
(a) When G41 or G42 is or has been specified.

(b) When a D code other than D00 is or has been specified.

(c) When the move axis, whose stroke is not 0, has been specified in the offset plane.

(d) When the move axis, whose stroke is not 0, has been specified in the offset plane of the next block. (The next block is the block skipping the block with no move axis within the con-tinuous blocks following the next block.)

(e) When the offset amount specified with a D code is not 0.



Where; r : Tool redius value
S : Single block stop position

(Note 1) An alarm results if you specify as the next block the block with an axis stroke of 0 within the offset plane.

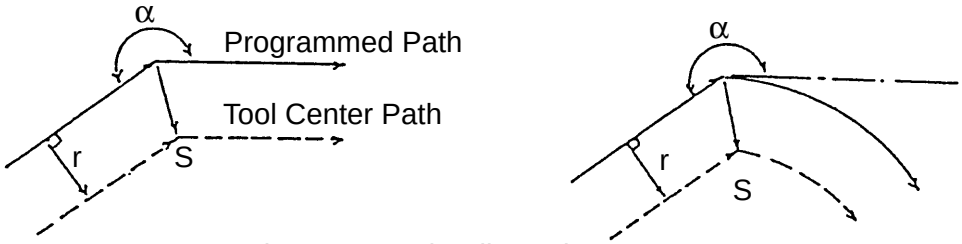
(2) Offset mode

In the offset mode, the tool center path is offset by the tool radius value against the programmed path.

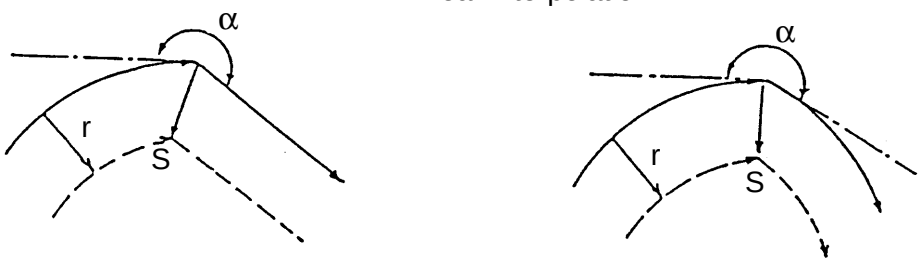
In the offset mode, 2 blocks are normally preread, and when the block with no axis move command is included, up to 4 blocks are preread.

The tool center path in the offset mode will be as follows depending on the angle of each block.

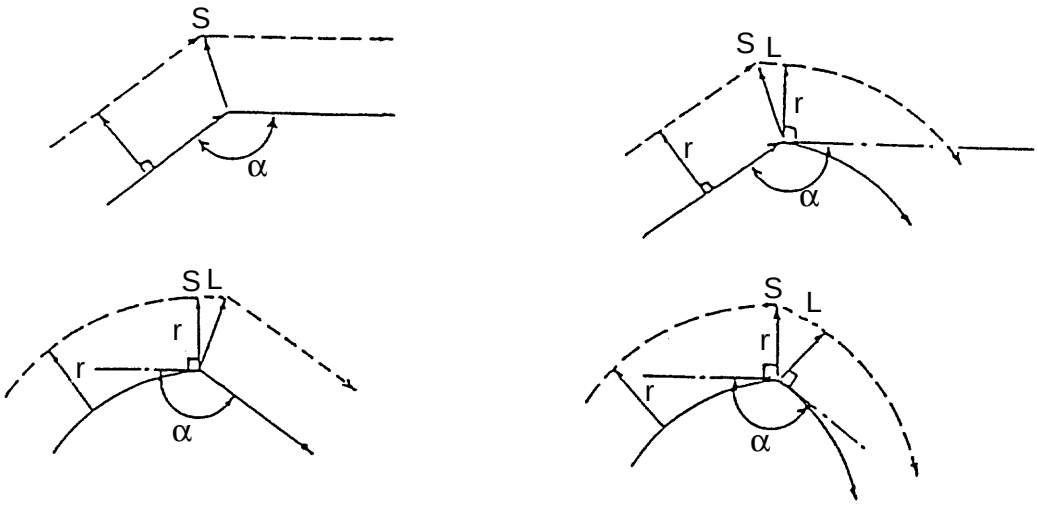
(a) When the tool moves inside ($180° \leqq \alpha < 359°$)



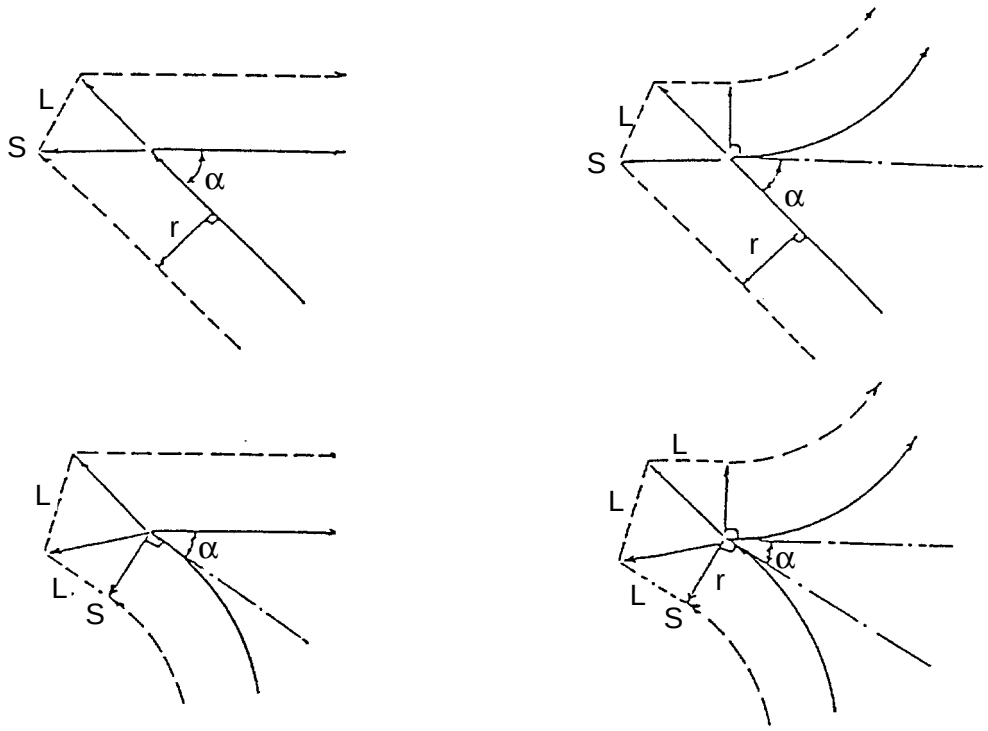
Where; r : Tool radius value
s : Single block stop position
L : Linear interpolation



(b) When the tool moves outside ($90° \leqq \alpha < 180°$)

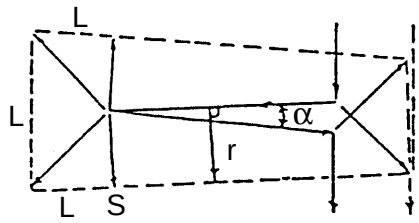


(c) When the tool moves outside at an acute angle ($\alpha < 90°$)



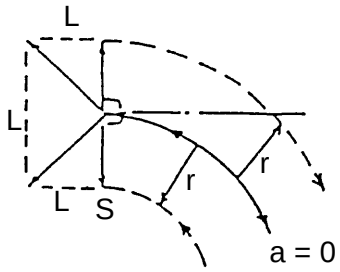
(d) When the tool moves inside ($359° \leqq \alpha$ or $\alpha < 1°$)

(i) When the tool moves inside in case of line-to-line, and the offset vector is large.($a < 1°$)



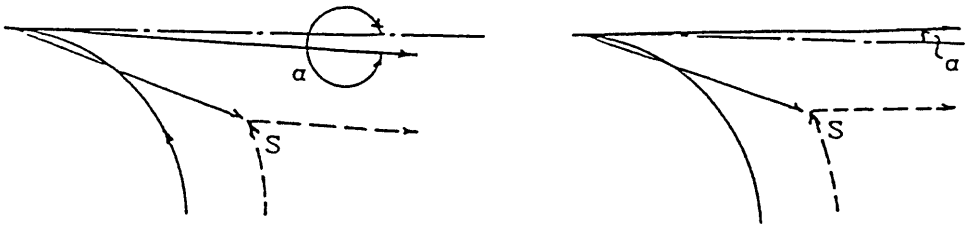
(ii) When the tool moves inside in case of line-to-arc, arc-to-line or arc-to-arc, and the offset vector is large or cannot be obtained.

(359° ≦ α or α < 1°)



(iii) When the tool moves inside in case of line-to-arc, arc-to-line or arc-to-arc, and the normal offset vector can be obtained.

(359° ≦ α or α < 1°)



(3) Cancel

If the block which satisfies even one of the following conditions is executed during offset mode, tool diameter compensation is cancelled and the tool moves to the programmed end point.

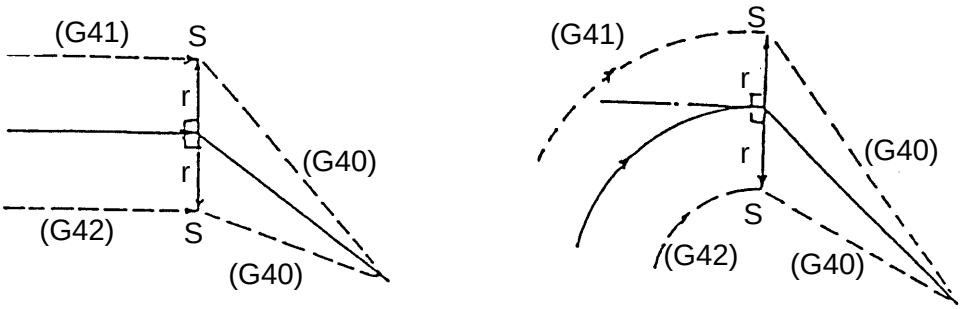
[Conditions]

(a) When G40 is specified.

(b) When D00 is specified.

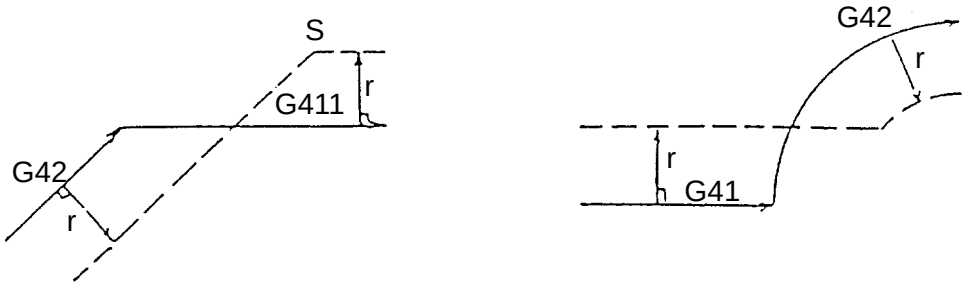
(c) When the offset amount specified to be 40 with a D code is 0.

When the block with no axis move command is specified together with a cancellation, the offset vector is created perpendicularly to the end point of the previous block and the offset amount is cancelled in the next block which contains an axis move command.

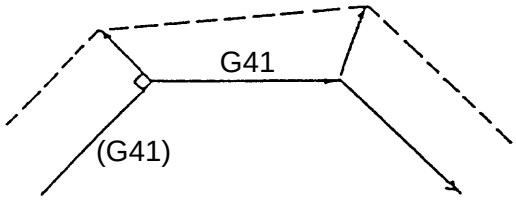


(4) Special uses

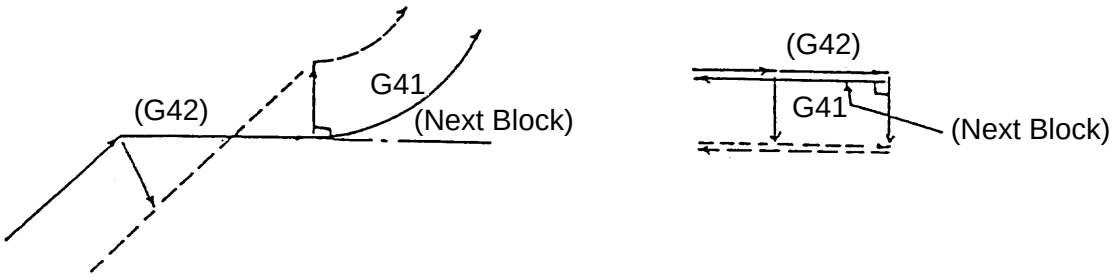
(a) When the offset direction is changed over by specifying G41/G42 during the offset mode, an intersecting point is obtained.



(b) When the offset direction is not changed over by specifying G41/G42 during the offset mode, the vector is created perpendicularly to the end point of the block before the G41/G42 command.



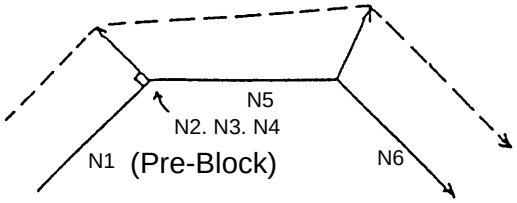
(c) When there is no intersecting point in the tool center path, the vector is created perpendicularly to the start point of the next block.



(d) When 3 or more blocks which have no axis move command are specified continuously, or when one or more block where an axial stroke is 0 is specified, the vector is created perpendicularly to the end point of the previous block.

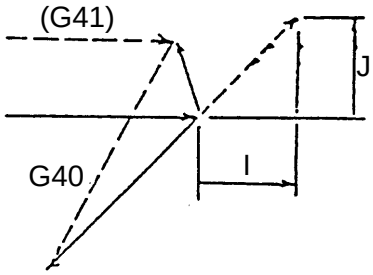
```
G17 G41 G91 G00 X10. Y10. D10  F200 ;
N1  G01 X50. Y50. ;
N2  M09 ;
N3  G04  X1. ;
N4  Z-50. ;
N5  X50. ;
N6  X50. Y-50. ;
```

Blocks with no axis move command



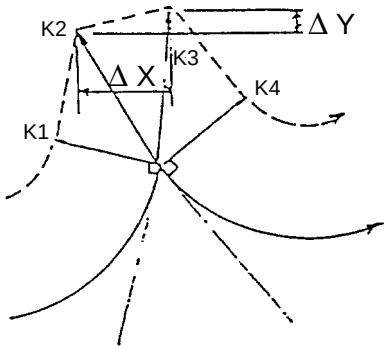
(e) The tool relief direction can be specified with I, J and K by giving G40 α_ β_ I _ J _ K _ ;

```
G17    G41    D10  ;
          :
          :
G91    G01    X50. ;
G40    X-50. Y-50. I30. J30. ;
```



(5) Move at the corner

(a) When 2 or more offset vectors are created at the end point of the block and they are almost matching, the latter vector is invalidated.
When the next block is an arc, however, the offset vector perpendicular to the start point of the next block becomes invalid.



[Conditions]

Δ X ≦ (Parameter value)

Δ Y ≦ (Parameter value)

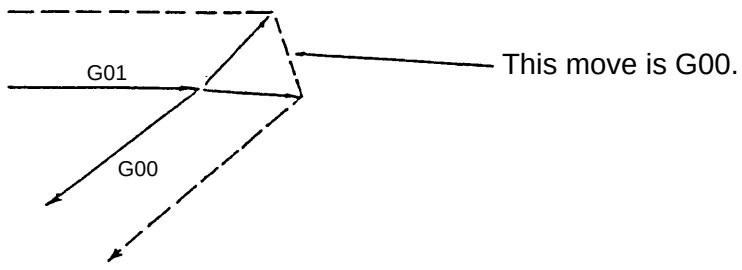
(i) When K2 and K3 are compared and their difference is smaller than the parameter set value, K3 is invalidated.

(ii) When K1 and K3 are compared and their difference is smaller than the parameter set value, K2 and K3 are invalidated.

(iii) When K2 and K3 are compared and their difference is smaller than the parameter set value, K2 and K3 are invalidated.

(iv) When K1 and K4 are matching, K4 is invalidated.

(b) A move at the corner is G00 when the next block is G00, and G01 when it is G01, G02 or G03.



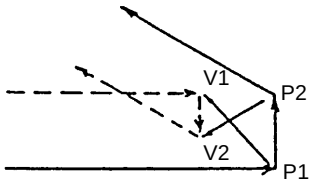
(6) Interference check

If tool diameter compensation is applied, the tool may cut in the workpiece when it has a special shape. With this function used, you can check whether the tool may cut into the workpiece before execution and prevent cut-in.

However, all cut-ins cannot be prevented. Also, even when a cut-in does not actually take place, an interference check alarm may result.

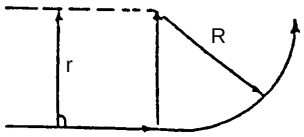
(a) Interference criteria in the interference check If one of the following conditions is satisfied, it is regarded as an interference.

(i) When the move direction in the programmed block and that in the corresponding block of the tool center path is 90° or more and 270° or less.



An interference results because the move directions of $\overrightarrow{V1V2}$ and $\overrightarrow{P1P2}$ forms an angle of 180°.

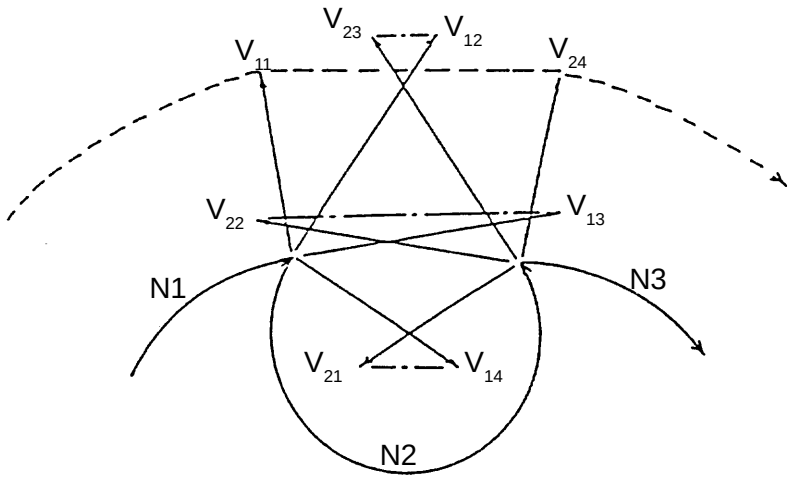
(ii) When an offset is applied to the inside of the arc which is smaller than the tool radius.



An interference results in case of $r \geqq R$.

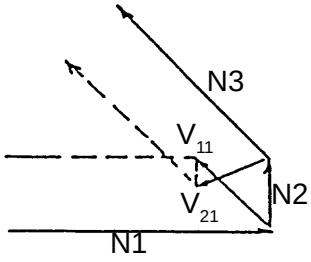
(b) Offset vector processing in case of interference An interference check is made sequentially starting at the offset vectors whose distance is closer, and when they interfere, they are erased when they are not the last one.

When the number of offset vectors is one at the time of first interference, an alarm results.



Interference check at V_{14} and V_{21} : Erases V_{14} and V_{21} due to interference
Interference check at V_{13} and V_{22} : Erases V_{13} and V_{22} due to interference
Interference check at V_{12} and V_{23} : Erases V_{12} and V_{23} due to interference
Interference check at V_{11} and V_{24} : No interference

(Note) The tool moves linearly from V_{11} to V_{24}.



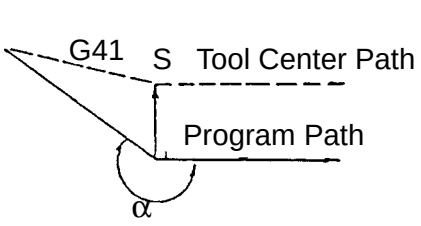
Interference check at V_{11} and V_{21} : Interference.
If V_{11} and V_{21} are erased, an alarm results because the offset vectors do not exist.

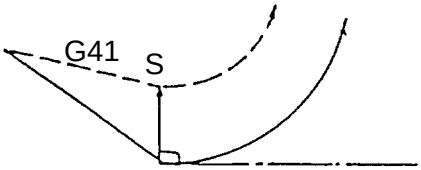
(7) Type B start-up and cancellation

By parameter setting, you can change over the start-up and cancellation methods to Type B.

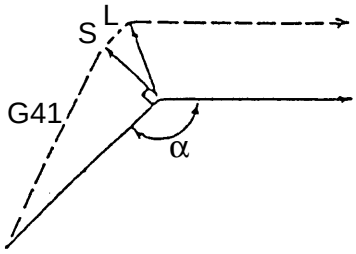
(a) Type B start-up

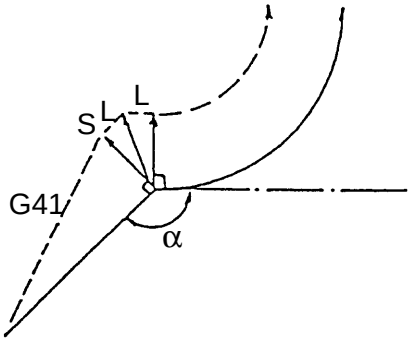
(i) When the tool moves inside($180° \leqq \alpha$)



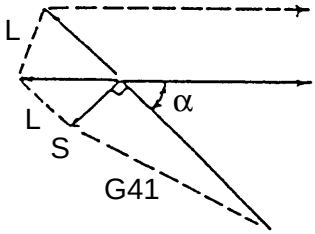


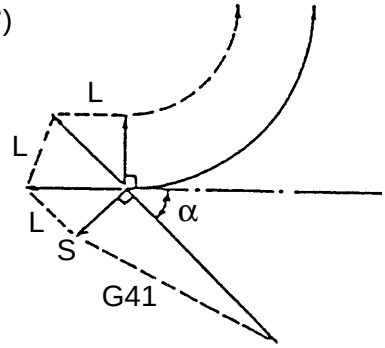
ii) When the tool moves outside (90° ≦ α < 180°)



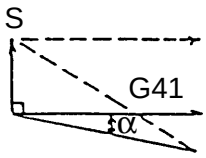


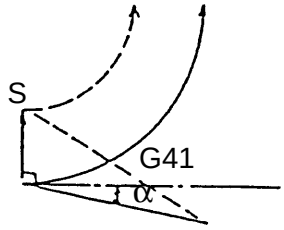
(iii) When the tool moves outside (1°≦ α <90°)





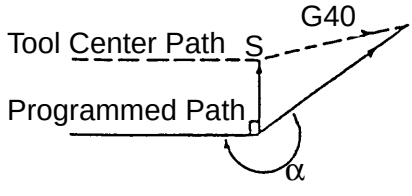
(iv) When the tool moves outside (α < 1°)

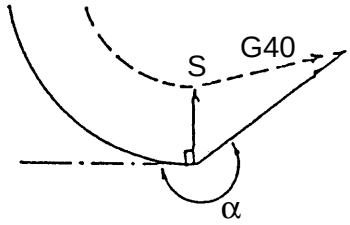




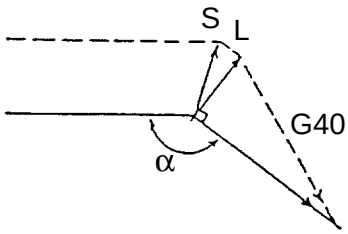
(b) Type B cancellation

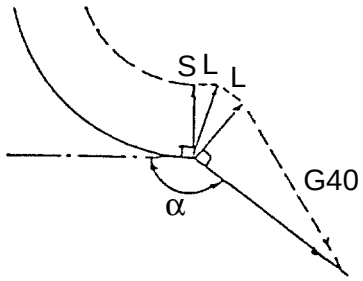
(i) When the tool moves inside (180° ≦ α)



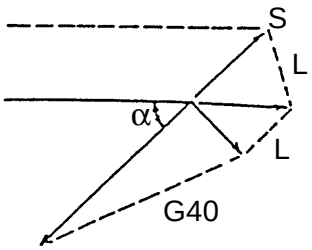


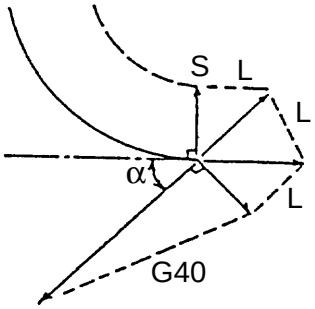
(ii) When the tool moves outside (90° ≦ α < 180°)



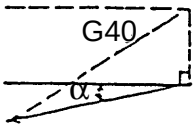


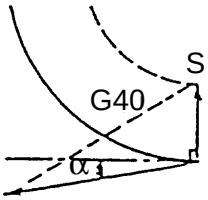
(iii) When the tool moves outside (1° ≦ α ≦ 90°)





(iv) When the tool moves outside (α ≦ 1°)





(8) Associated parameters

NO. 5003, #0 =	0	The start-up and cancellation methods are Type A.
	1	The start-up and cancellation methods are Type B.
NO. 5003, #1 =	0	Interference check : Offset vector difference 90° ~ 270° = Effective
	1	Interference check: Offset vector difference 90° ~ 270° = Ineffective
NO. 5003, #2 =	0	All the interference checks are effective.
	1	All the interference checks are ineffective.
NO. 5003, #3 =	0	Does not move by the offset amount in case of G04; block.
	1	Moves by the offset amount in case of G04; block.
NO. 5025		Limit value ignoring a little move amount around the corner.

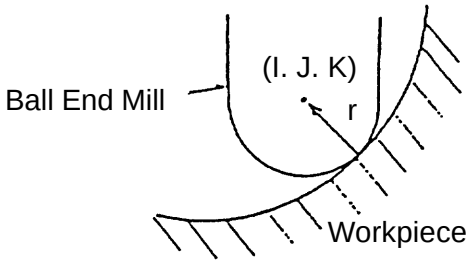
(9) Associated alarms

NO.115	Tool diameter compensation start-up or cancel has been specified in code other than G00/G01.
NO.117	Excessive cutting has occurred in tool diameter compensation.
(#001)	Arc radius < Tool diameter compensation amount
(#002)	Other interference
NO.118	No intersection exists in Tool Diameter Compensation mode.
NO.119	An erroneous command has been given while in Tool Diameter Compensation mode.
(#001)	Change in plane in current block.
(#002)	Change in plane in next block.
(#003)	Erroneous G code in current block.
(#004)	Erroneous G code in next block.
(#005)	G38 command in code other than G00/G01 in current block. (Offset vector retained)
(#006)	G38 command in code other than G00/G01 in next block. (Offset vector changed.)
(#008)	Axis command inside plane with G39 command in next block (Corner arc)

11-4 3-D Tool Offset (G40 - G41)

This command can offset the tool center path outside or inside the program path by the tool radius value in accordance with the 3-D vector.

If this function is used, the tool can be offset by the spherical radius value when machining the 3-D curved surface by using a ball end mill.



r: Tool spherical radius value
I, J and K : Tool spherical center vector

(Note) It is necessary to specify the vector components from the contact point of the work and the tool and the tool spherical center by I, J, K for each block.

(1) G code

G40 : 3-D tool offset cancel
G41 : 3-D tool offset (offset in the I, J and K directions)
G42 : 3-D tool offset (offset in the opposite directions of I, J and K)

(2) Command format

$$\left\{\begin{matrix}G00\\G01\end{matrix}\right\}\ \left\{\begin{matrix}G41\\G42\end{matrix}\right\}\ X_P_\ Y_P_\ Z_P_\ I_\ J_\ K_\ D_\ ;$$

where; X_P : X axis or its parallel axis
Y_P : Y axis or its parallel axis
Z_P : Z axis or its parallel axis
I : Offset direction of the X_P axis
J : Offset direction of the Y_P axis
K : Offset direction of the Z_P axis
D : Offset number

If this command is specified with tool diameter compensation cancelled, the 3-D tool offset mode results.

$$\left\{\begin{matrix}G00\\G01\end{matrix}\right\}\ G40\ X_P_\ Y_P_\ Z_P_\ ;$$

If this command is specified in the 3-D offset mode, 3-D tool offset is cancelled.

(3) Designation of the 3-D tool offset axis

The axis to which 3-D tool offset is to be applied is determined by the address of the move axis specified in the G41/G42 specified block.

G41 X _ I _ J _ K _ ; X, Y and Z axes

G41 U _ I _ J _ K _ ; U, Y and Z axes

(4) Offset vector of 3-D tool offset

In the 3-D tool offset mode, the offset vector is created in the direction specified with I, J and K at the end point of each block.

The offset vector is calculated as follows.

$Vxp = r \times \frac{i}{P}$: Offset vector component in the X_P axis direction

$Vyp = r \times \frac{j}{P}$: Offset vector component in the Y_P axis direction

$Vzp = r \times \frac{k}{P}$: Offset vector component in the Z_P axis direction

where ; r : Offset amount (tool spherical radius value)

i : Value of I

j : Value of J

k : Value of K

$$P = \sqrt{i^2 + j^2 + k^2}$$

(Note) P can use the value set in the parameter.

(5) Cautions

(a) The X, Y and Z addresses can be omitted in the G41/G42 specified block. However, the parallel axis cannot be omitted.

(b) Be sure to specify I, J and K in the G41/G42 specified block. If even one of them is omitted, tool diameter compensation results.

(c) When X_P , Y_P and Z_P are all omitted in the G40 specified block, only the offset amount is cancelled.

(d) When circular interpolation or helical interpolation, the vector created with the previous vector is saved as it is.

(e) If you specify the block, where I, J and K are all omitted, during the 3-D tool offset mode, the vector created with the previous vector is saved as it is.

(f) If 3-D tool offset is specified during the tool diameter compensation mode, it is invalidated.

(g) If the offset axis is changed over by specifying G41/G42 during the 3-D offset mode, an alarm results.

(h) The following G codes result in an alarm if they are specified during the 3-D offset mode.

G31, G37
G28, G30, G53
G45 ~ G48
G73, G74, G76, G81 ~ G89
G302 ~ G305
G322 ~ G333

(i) If surface cutting using the ball-end mill, it is possible to combine 3-D coordinate conversion and tool diameter compensation to substitute for 3-D tool compensation.

(6) Associated parameters

No.5026 Denominator constant (P) by 3-D tool offset

$P = \sqrt{i^2 + j^2 + k^2}$ when setting is 0.

(7) Associated alarms

No.158 The format for 3-D tool offset has an error.

(#001) G code which cannot be specified exists in Compensation mode.

(#002) The axis does not exist among the three base axes (X/Y/Z axes).

No.159 No intersecting point is found in 3-D tool offset.

11-5 H and D Functions

The tool offset number is specified with a 4-digit number following the address H or D.

(1) Command format

H _ ; or D _ ;

This command validates the offset amount specified with an H code and that specified with a D code. The H and G codes are modal.

Once they are specified, they remain effective until the next command is given.

H00 ; or D00 ;

H00 or D00 means the offset amount 0.

They have special meanings depending on the functions such as tool length compensation, tool diameter compensation and tool offset.

(2) Sample program

H01 ;

Validates length compensation (profile + wear) specified with the tool 001.

Offset amount specified with H01 = Profile(200.) + Wear(-0.01)
= 199.990

D05 ;

Validates diameter compensation (profile + wear) specified with the tool 005.

Offset amount specified with D05 = Profile(30.) + Wear(-0.01)
= 29.990

Tool Offset Memory

Tool	Tool name	Length Compensation		Diameter Compensation	
		Profile	Wear	Profile	Wear
001		200.000	-0.010	0.000	0.000
002		0.000	0.000	0.000	0.000
003		0.000	0.000	0.000	0.000
004		0.000	0.000	0.000	0.000
005		0.000	0.000	30.000	-0.010
006		0.000	0.000	0.000	0.000

(3) Function using the H and D codes

Tool length compensation : H code
Tool offset : H code or D code
Tool diameter compensation : D code

(Note) Whether the H or D code is used for tool offset depends on parameter setting.

(4) Cautions

(a) The offset amount specified with the H or D code are validated when the H or D code specified block is buffered.
Also, even if the H or D code is not specified, the offset amount can be validated by parameter setting when the next block is buffered.

(5) Associated parameters

NO.5002, #0 Offset volume, when changed, becomes effective starting with:
= 0 the block in which D/H codes are next specified.
= 1 the block in which next buffering takes place.

(6) Associated alarms

No.112 Specified tool No. has exceded the max. tool compensation number.

11-6 Tool Offset by Tool Number

This function automatically selects tool length compensation and tool diameter compensation for the spindle tool.

(1) Sample program (basic use)

T02 M06; ....①

G43 G90 G00 Z100.; ....②

① Call the No.2 tool to the spindle. Upon completion of M06 operation, tool offset No.2 is effected.

② The tool nose moves to Z100 of the work coordinates.

[Calculation of Tool Offset]

Tool	Tool name	Length Compensation		Diameter Compensation	
		Geometry	Wear	Geometry	Wear
001		102.015	0.000	12.232	0.000
002		200.000	-0.010	30.000	-0.010
003		164.770	-0.001	0.000	0.000
004		158.256	-0.008	16.188	-0.012

No.2 tool offset is; Tool length compensation amount

= Geometry(200.) + Wear(-0.01) = 199.990

Tool diameter compensation amount

= Geometry(30.) + Wear(-0.01) = 29.990

(2) Offset number

(a) Upon completion of M06 operation, the spindle tool number becomes the offset number.

(b) Either H or D code is used for the tool offset number.

(c) When H or D code is used, they are given priority over the spindle tool number.

(3) Compensation by Spindle Tool Number

(a) Tool length compensation

The work coordinate system is shifted by the difference of the tool length compensation amount corresponding to the spindle tool number from the previous offset amount.

The work coordinate system is shifted in the following cases.

(i) At completion of manual origin reset at power ON

(ii) Upon completion of the ATC (M06)

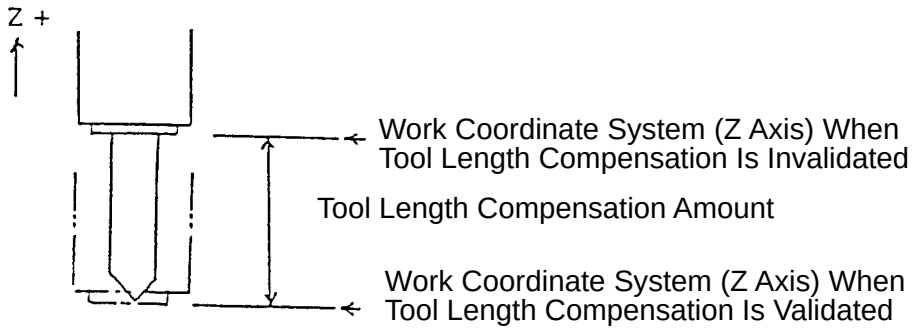
(iii) G43/G44/G49 specified block

(iv) Machining plane selection (G240 - G245) command block

(Note) Reset state will be G43

[Example]

```
T02 M06        ; Shifts the Z-axis work coordinate system by the offset amount of T02.
G00 G90 Z0     ; Positions to Z0 of the work coordinate system
    :
    :
T03 M06        ; Shifts the Z-axis work coordinate system by the offset amount of T03.
G00 G90 Z0     ; Positions to Z0 of the work coordinate system
    :
    :
```



(Note) In case of the G44 mode, the work coordinate system is shifted in the opposite direction.

(b) Tool diameter compensation

Tool diameter compensation is validated from the G41/G42 specified block.

[Example]

```
T06  M06  ;
      :
      :
G41 .... ;
      :
      :
G40  ;
```

Tool diameter compensation is applied with the offset amount of T02.

(4) Multiple offset (Compensation by H code, D code)

(a) Tool length compensation

With H ? ;, tool length compensation is applied with the offset amount specified with an H code, not the spindle tool number. The work coordinate system is shifted just by the tool length offset amount.

H _ ... ; Turns on multiple offset with an H code.
H00 ... ; Cancels multiple offset with an H code.

(Note) If T_M06; is specified, multiple compensation by the H code is cancelled.

[Example] When performing tool offset with the tool number by shifting the work coordinate system.

```
T02  M06  ;          Shifts the Z-axis work coordinate system by the offset amount of
                     T02.
G00  G90  Z0 ;
  :
  :
H106  ;              Turns on multiple offset with an H code Shifts the Z-axis work
                     coordinate system by the offset amount of H106.
G00  G90 Z0 ;
  :
  :
H00                  Cancels multiple offset with an H code Shifts the Z-axis work
  :                  coordinate system by the offset amount of T02.
  :
  :
  :
H107 G00 G90 Z0 ; Turns on multiple offset with an H code Positions to Z0 of the work
  :                  coordinate system after shifting the Z-axis coordinate system by
  :                  the offset amount of H107.
  :
  :
T03  M06  ;          Shifts the Z-axis work coordinate system by the offset amount of
  :                  T03.
  :                   (Cancels multiple offset with an H code)
```

(Note) The H00 command does not cancel tool length compensation, but returns you to the offset amount of the spindle tool number.

(b) Tool diameter compensation

With the command D _ ; tool diameter compensation is applied with the offset amount specified with an D code, not the spindle tool number. The G49 command becomes the same as the H00 command.

D _ ..... ; Turns on multiple offset with an D code.
D00 ..... ; Cancels multiple offset with an D code.

(Note) If T_M06; is specified, multiple compensation by the D code is cancelled.

[Example]

```
T02  M06 ;
G41 ..... ;
   :                          Tool diameter compensation is applied
   :                          with the offset amount of T02.
   :
G40 ;
   :
   :
   :
   :
G41 .....D102 ;               --------Turns on multiple offset with a D code.
   :
   :                          Tool diameter offset is applied with
   :                          the offset amount of D102.
   :
   :
G40 ;
D00 ;                          Cancels multiple offset with a D code.
```

(Note) The D00 command does not cancel tool diameter compensation, but returns you to the offset amount of the spindle tool number.

(5) Cautions

(a) Note that the D code (or H code) used in tool offset is taken as that for multiple offset.
(b) Multiple offset is cancelled upon completion of ATC (M06) operation.
(c) A tool change M code (M06) cannot be specified together with other M code in the same block.

(6) Associated parameters

No. 3407, #5 = 0 Performs tool offset with the tool number.
1 Does not perform tool offset with the tool number.

No. 5018 Tool change M code. Taken as M06 in case of 0.

No.3402

#3	#2	Mode setting
0	0	G49
0	1	G43
1	0	G44

(7) Associated alarms

No.112 Specified tool No. has exceeded the max. tool compensation number.

12. CONVERTING FUNCTION

12-1 Programmable Mirror Image (G501, G511)

With this command given, mirror image is applied for each axis to the shape specified work program.

(1) G code

G511 : Programmable mirror image ON

G501 : programmable mirror image cancel

(2) Command format

G511 X a Y b X c ... ;

This command applies the mirror image, with the position of (Xa, Yb, Xc, ...) as a mirror point.

X, Y, Z, ... in the G511 block are the axes to specify the mirror point and irrelevant to an axial move.

G501 X a Y b X c ... ;

This command cancels the programmable mirror image of the specified axes. X, Y, Z, ... in the G501 block are the axes to cancel the mirror image and irrelevant to an axial move.

G501 ;

This command cancels the programmable mirror image of all the axes.

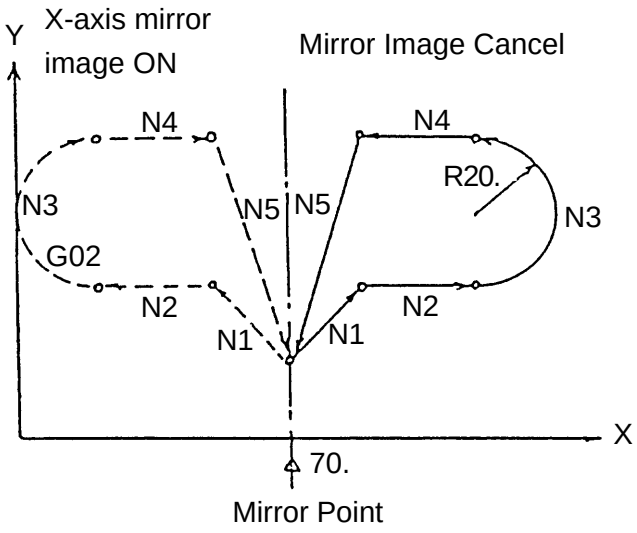
(3) When the mirror image is applied to only one axis of the specified plane

(a) Arc command (G02, G03) : Clockwise and counterclockwise are reversed.

(b) Tool diameter compensation (G41, G42) : Offset to right and offset to left are reversed.

(4) Sample program

```
G54  G90  G00  X70. Y20. ;
G511  X70. ;                          X-axis mirror image ON
N1  G01  X90. Y40. F200 ;
N2  X120. ;
N3  G03  Y80. R20. ;                  The mirror image is applied to the X axis
N4  G01  X90. ;
N5  X70. Y20. ;
G501  X0 ;                            X-axis mirror image cancel
```



(5) Cautions

(a) Specify the G511 and G501 commands in the independent block.
Otherwise, an alarm will result.

(b) Position display indicates the coordinate value after the programmable mirror image is applid.

(c) When the programmable mirror image and setting mirror image are applied, the former works first, and then, the latter.

(d) The following commands disable the programmable mirror image.
G28, G30, G53, G27 and G29
Shift amount of fine boring (G76) and back boring (G87)

(e) Specify the G511 and G501 commands in the G69 or G50 modes.
If they are specified in the G68 or G51 mode, and alarm results.

(f) To specify coordinate rotation and a programmable mirror image, follow the steps below:

```
G511 X_ Y_ Z_ ; Programmable mirror image ON
G68           ; Coordinate rotation ON
  :
G69           ; Coordinate rotation cancelled
G501          ; Programmable mirror image cancelled
```

(g) The first move command after the block which contains G511 or G501 should be specified by absolute programming.

(h) If arc is specified as the first move command after the block which contains G511 or G501, a proper arc may not be obtained.

(6) Associated parameters

No.3406, #1 =	0	Mirror image processing is performed before scaling and coordinate rotation.
	1	Mirror image processing is performed after scaling and coordinate rotation.
No.3406, #2		For mirror image coordinate rotation, axis switching, etc., with the middle point (G28, G30) being specified with an incremental command, middle point position inside NC is:
	= 0	converted into absolute value.
	= 1	remains as incremental value.

(7) Associated alarms

No.144	G511/G501 commands are not for independent blocks.

12-2 Setting Mirror Image

The mirror image can be applied to each axis by on/off operation in the Setting screen or by turning on/off an external input signal(PC → NC).

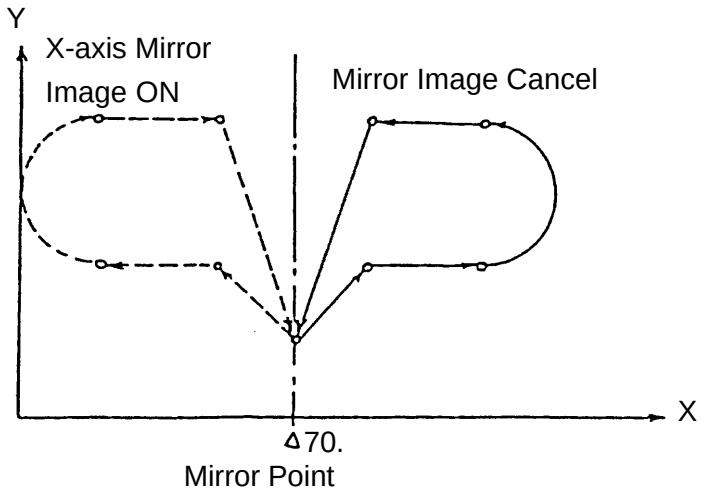
(Note) Whether absolute/incremental programming is used, the mirror image is applied taking as the mirror point the coordinate value when it is turn on.
If G54 or other work coordinate system setting is specified, the mirror image is applied with the mirror point as 0.

(1) On/off operation in the Setting screen
  (a) Press [SET], and then,[C].
  (b) Using the cursor keys, [↑] , [↓] , [←], and [→] ,adjust to enable/disable the specified mirror image axes.
  (c) Press [INPUT] .

(2) When mirror image has been applied only to one axis of assigned plane:
  (a) Arc command (G02, G03) : Clockwise and counterckickwise rotations are reverted.
  (b) Tool diameter compensation (G41, G42) : Right-side offset and leftside offset are inverted.

(3) ample program

```
G54  G90  G00  X70. Y20. ;
;                                          X-axis mirror image ON
N1  G01  G90  X90. Y40. F200 ;
N2  X120. ;
N3  G03  Y80. R20. ;
N4  G01  X90. ;
N5  X70. Y20. ;
;                                          Mirror image OFF
```



(4) Cautions

(a) On/off switching of mirror image is made effective in the next buffering block on.

(b) Position display indicates the coordinate value after the setting mirror image is applied.

(c) When the programmable mirror image and setting mirror image are applied, the former works first, and then, the latter.

(d) The following commands disable the setting mirror image.

G28, G30, G53, G27 and G29.

Shift amount of fine boring (G76) and back boring (G87)

(e) The setting mirror image on position and off position should be identical.

(f) The setting mirror image is turned off in the reset state.should be specified by absolute programming.

(g) When the setting mirror image is switched on/off, the first axis command should be specified by absolute programming.

(h) If the setting mirror image is switched from OFF to ON and an arc is specified for first axis command, a proper arc may not be obtained.

(5) Associated parameters

No. 3406, #1 = 0 Mirror image processing is performed before scaling and coordinate rotation.

1 Mirror image processing is performed after scaling and coordinate rotation.

No. 3406, #2 For mirror image, coordinate rotation, axis switching, etc., with the middle point (G28, G30) being specified with an incremental command, the middle point position inside NC is:

= 0 converted into an absolute value.

1 remains as an incremental value.

No. 3416, #0 = 0 Setting mirror image for each axis is made invalid.

1 Setting mirror image for each axis is made valid.

(6) Associated alarms

12-3 Scaling (G50, G51)

This command allows you to enlarge/reduce the profile given by the machining program at the specified scale factor.

(1) G code

G50 : Scaling cancel

G51 : Scaling ON

(2) Command format

G51 Xa Yb Zc ... [P _]; [ ] is omissible

This command causes the move commands after the next block to enlarge/reduce the profile at the scale factor specified with P, taking the position on (Xa, Yb, Zc) as the scaling center.

G50 ; This command cancels the scaling mode.

(3) Magnification factor (P)

The following two magnifications are available for parameter setting (No. 3405, #1):

0.00001 ~ 999.99999 or 0.001 ~ 99999.999

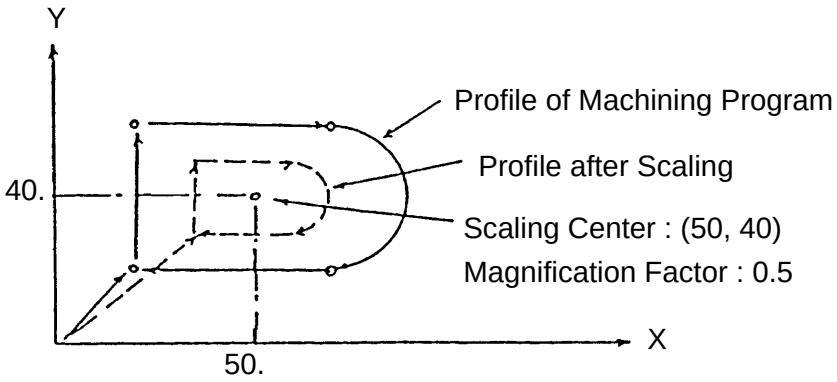
With P omitted in G51 command block, magnification set with the parameter (No.3460) is made effective.

(4) Scaling enabled axis

Scaling function, to made it valid or invalid, is set for each axis with Parameter (No.3416, #3).

(5) Sample program

```
G90  G00  X0  Y0 ;
G51  X50. Y40. P500 ;       Scaling ON
N1  G00  G90  X20. Y20. ;
N2  G01  G91  Y40. F200. ;
N3  X50. ;
N4  G02  Y-40. R20. ;
N5  G01  X-50. ;
N6  G50 ;                   Scaling cancel
```



(6) Cautions

(a) Scaling is not applied to the offset amounts for tool diameter compensation, tool length compensation and tool offset.

(b) Specify the G51 command in the independent block. Otherwise, an alarm results.

(c) When scaling is applied to one axis in the plane, do not specify an arc command.

(d) The first move command after the block which contains G51/G50 should be given by absolute programming.

(e) When scaling is applied to the two axes in the plane, it is also applied to optional angle chamfering corner R (, C _, R _).

(f) When a scaling factor is set for each axis by parameter setting, the scale factor(P) specified by the program is invalidated.

(g) Specify the G51 command in the coordinate rotation cancel (G69) mode.

(h) The following commands disable scaling.

G28, G30, G53, G27 and G29

Shift amount of fine boring (G76) and back boring (G87)

Scaling factor = 0

(i) Command format is different from SEICOS-MⅡ/A.

(7) Associated parameters

No.3405, #1 =	0	Scaling factor increment 0.001-fold
	1	Scaling factor increment 0.00001-fold
No.3416, #4 =	0	Disables the scaling factor of each axis
	1	Enables the scaling factor of each axis
No.3416, #3 =	0	Disables scaling of each axis.
	1	Enables scaling of each axis.
No.3460		Scaling factor when P is omitted in the block containing G51
No.3475		Scaling factor by axis (the axis for which No. 3416, #4 = 1 is set)

(8) Associated alarms

No. 142	The format of the block containing G51 has an error.
(#001)	G51 command is not for independent block.

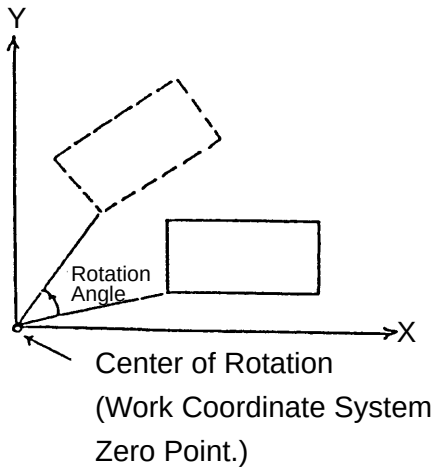
12-4 Coordinate Ratation (G68, G69)

This command can rotate the profile specified by the machining program by the specified angle. There are the following two kinds of coordinate rotation.

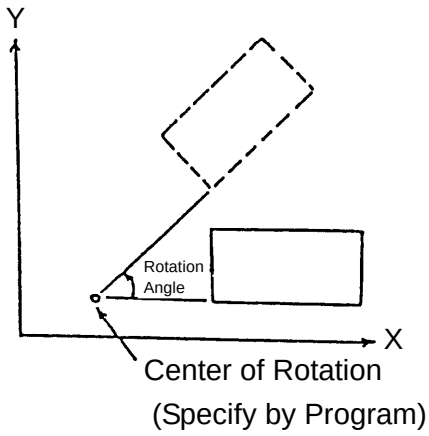
(a) When assuming the center of rotation as the work coordinate system zero point .....Type A

(b) When specifying the center of rotation by the program .....Type b

Type A



Type B



(1) G code

G68 α_ β_ R_ ;	Coordinate rotation ON
G69 ;	Coordinate rotation cancel

(2) Command format

G68 α_ β_ R_ ... ; (Coordinate rotation Type B)

With this command, shift command for the next block on is made into a format as having been turned by the angle assigned with R centering around (α_ β_) position.

α/β are specified in absolute values for the two axes on the plane assinged by G17/G18/ G19.

R is available as follows depending on parameter setting (NO.3405, #2):

- 360.000~ +360.000 or
- 360.00000~ +360.00000

The counterclockwise direction is in plus.

G68 ; (Coordinate rotation Type A)

With this command, shift command for the next block on is made into a format as having been turned by the angle assigned with R of the work coordinate page centering around the zero point of the work coordinate system.

G69 ;

With this command, Coordinate Rotation mode is cancelled.

(a) To use Type A and B combined, apply Type A after applying Type B.

```
G68 α_ β_ R_ ; Coordinate rotation Type B ON
G68 ;          Coordinate rotation Type A ON
 :
 :
G69 ;          Coordinate rotation cancel
```

(b) When using coordinate rotation Type A

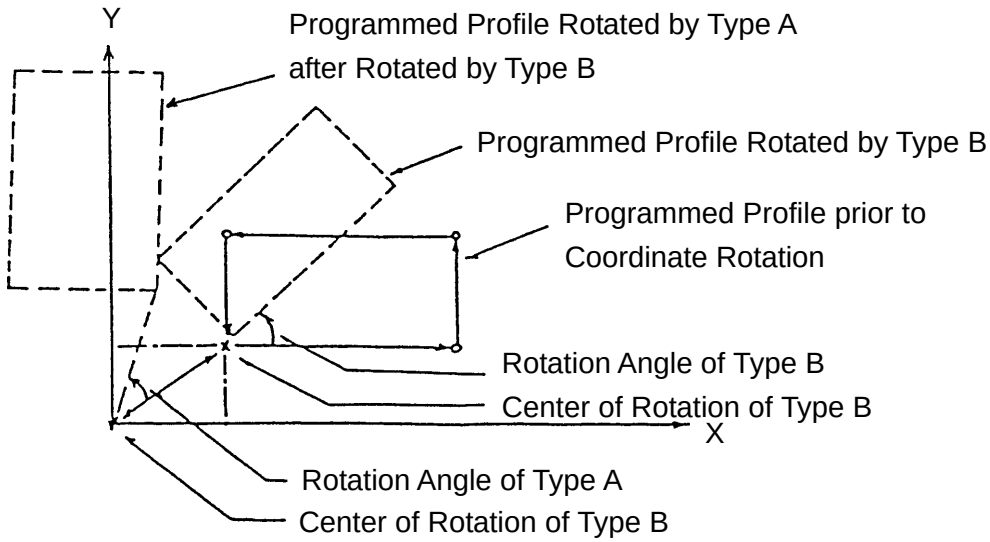
```
G68 ;          Coordinate rotation Type A ON
 :
 :
G69 ;          Coordinate rotation cancel
```

(c) When using coordinate rotation Type B

```
G68 α_ β_ R_ ; Coordinate rotation Type B ON
 :
 :
G69 ;          Coordinate rotation cancel
```

(3) Sample program

```
G17  G54  G90  G00  X0  Y0 ;
G68  X30. Y20. R45. ;          Coordinate rotation Type B ON
G68  ;                         Coordinate rotation Type A ON
N1  G01  G90  X30. Y20. F200 ;
N2  G91 X60.;
N3  Y30. ;
N4  X-60. ;
N5  Y-30. ;
G69  X-30.  Y-20. ;            Coordinate rotation cancel
```



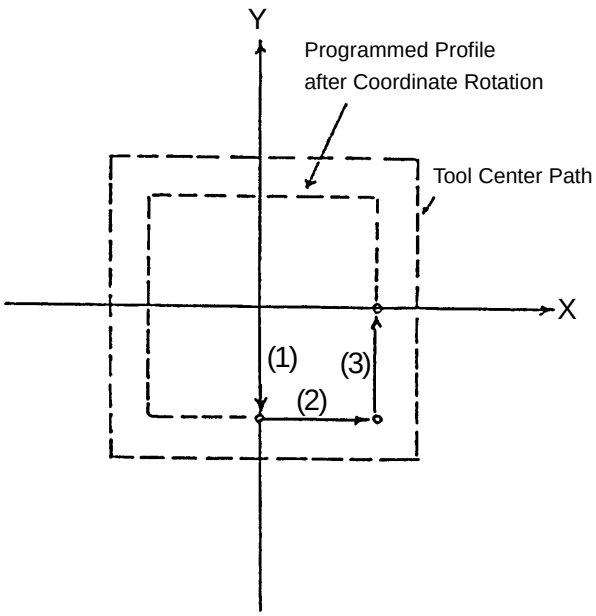
(4) When you use coordinate rotation together with tool diameter compensation, scaling and programmable mirror image, specify them in the order as follows.

```
G511 ..... ;          Programmable mirror image ON
G51 ..... ;           Scaling ON
G68 α_ β_ R_ ;        Coordinate rotation Type B ON
G68 ;                 Coordinate rotation Type A ON
G41 ..... ;           Tool diameter compensation ON
  :
  :
G40 ;                 Tool diameter compensation cancel
G69 ;                 Coordinate rotation cancel
G50 ;                 Scaling cancel
G501 ;                Programmable mirror image cancel
```

(5) When specifying repeatedly

By setting the parameters, you can register one program as a subprogram and call that program, changing the angle.

```
G17 G54 G90 G00 X0 Y0;
G68 X0 Y0 R0;
M98 P100;
M98 P200 L3;
G00 G90 X0 Y0;
G69;

0100;
G90 G01 G42 X0 Y-10.D10; ... (1)
X10.;                    ... (2)
Y0;                      ... (3)
G40;
M99;

0200;
G91 G68 X0 Y0 R90.;
G90 M98 P100;
M99;
```



(6) Cautions

(a) Specify a G68 command in the independent block. Otherwise, an alarm will result.

(b) An alarm results if the plane is changed by specifying plane selection (G17, G18, G19) during the G68 mode.

(c) The first move command after the block which contains G68/G69 should be given by absolute programming.

(d) The following commands disable coordinate rotation.

Shift amount of fine boring (G76) and back boring (G87)

G28, G29, G30, G53, G31 and G27

(e) When G92 or G52 is specified, the center of rotation for coordinate rotation Type A is (0, 0).

(f) A correct profile is not obtained if tool position compensation (G45 - G48) is effected in the coordinate rotation mode.

(7) Associated parameters

No. 3405, #0 =	0	The coordinate rotation angle is always of absolute programming.
	1	The coordinate rotation angle depends on G90/G91.
No.3405, #2 =	0	The input increment of the coordinate rotation angle is 0.001 degree.
	1	The input increment of the coordinate rotation angle is 0.00001 degree.
No.3405, #3		With "G68;", the rotation center and the rotation angle are:
=	0	Rotation center: zero point of G54~G59 Rotation angle : set value R of G54~G59
	1	Rotation center: current position Rotation angle : set value of Parameter(NO.3461)
No.3406, #2		For mirror image, coordinate rotation, axis switching, etc., with a middle point (G28,G30) being specified in an incremental command, the middle point position inside NC is:
=	0	converted into an absolute value.
	1	remains as an incremental value.
No.3461		Rotation angle of coordinate rotation (G68) (Valid with No.3405,#3=1)

(8) Associated alarms

No.128	G68 command format has an error, or a plane has been changed while in G68 mode.
(#001)	G68 command is not for an independent block.
(#002)	Plane has been changed while in coordinate rotation Type B.
(#003)	Plane has been changed while in coordinate rotation Type A.

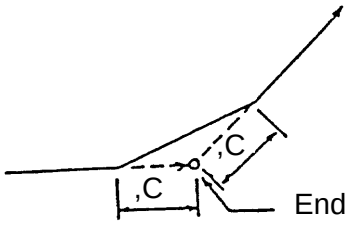
12-5 Optional Angle Chamfering/Corner R (, C, R)

Chamfering or corner R can be inserted by specifying:, C" or ",R" in linear interpolation or circular interpolation.

(1) Command format

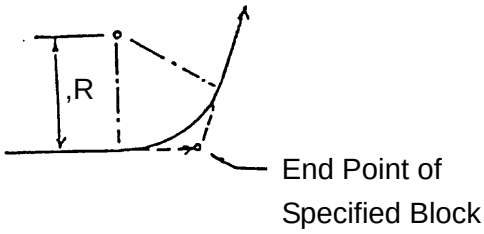
(a) Optional angle chamfering

$\begin{Bmatrix} G01 \\ G02 \\ G03 \end{Bmatrix}$, C_ ;



(b) Optional angle corner R

$\begin{Bmatrix} G01 \\ G02 \\ G03 \end{Bmatrix}$ ....., R_ ;



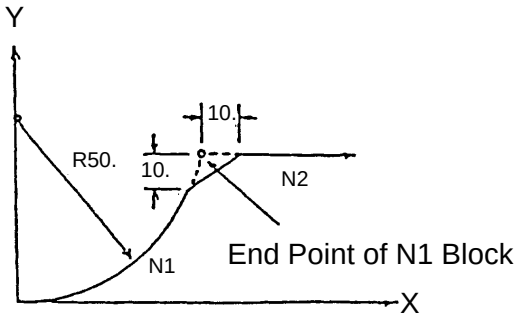
(2) Specify the plane for optional angle chamfering/corner R with plane selection (G17, G18, G19).

(3) In the next block where optional angle chamfering/corner R (, C, R).
Specify linear or circular interpolation.

(4) Sample program

(a) Optional angle chamfering

```
G17  G54  G90  G00  X0  Y0  ;
N1  G03  X50. Y50. R50., C10. F200 ;
N2  G01  X90.,
```

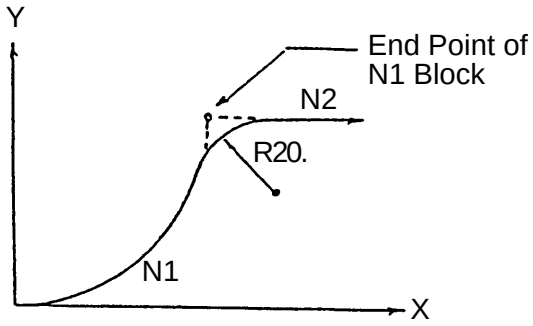


(b) Optional angle corner R

G17 G54 G90 G00 X0 Y0 ;

N1 G03 X50. Y50. R50., R20. F200 ;

N2 G01 X90.,



(5) Cautions

(a) If the plane is switched by specifying plane selection (G17, G18, G19).

(b) A single block stop results in the end point of the newly inserted block for chamfering corner R.

(c) An alarm results if the following commands are specified in the ,C or R specified block and the next block.

G92, G52, G54 ~ G59,

G28, G29, G30, G53,

G31, G45 ~ G48

(d) When a difference of angle is within +/-1 % between the, ,C or R specified block and the next block, chamfering/corner R is invalid.

In case of arc, the difference of angle is obtained from the contact point at the intersecting point.

(e) When there is no axis move command in the plane in the ,C or R specified block and next block, chamfering/corner R results in an alarm.

(f) When the original movable range is exceeded as aresult of inserting chamfering or corner R, an alarm results.

(6) Associated parameters

(7) Associated alarms

NO.124	The original specified range is exceeded in optional angle chamfering/ corner R.
(#001)	,C/,R command value is in minus.
(#002)	Both ,C and ,R exist in one block.
(#003)	Current block is not G01~G03.
(#004)	No axis shift takes place inside plane of current block.
(#005)	Erroneous G code exists in current block.
(#006)	Next block is not G01 ~ G03.
(#007)	No axis shift takes place inside plane of next block.
(#008)	Erroneous G code exists in the next block.
(#009)	Plane change exists in next block.
(#010)	Original shift amount has been exceeded in current block.
(#011)	Original shift amount has been exceeded in next block.
(#012)	Calculation error.

13. MEASURMENT

13-1 Skip Function (G31)

Linear interpolation is performed by a G31 command. If an external skip signal is input during linear interpolation, the program proceeds to the next block, stopping the axes and discarding the remaining stroke.

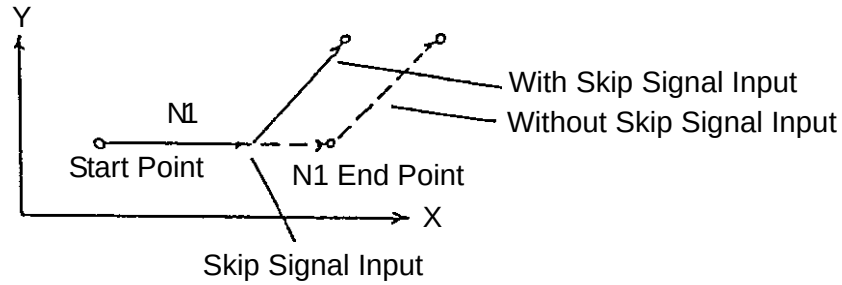
(1) Command format

G31 X _ Y _ Z _ ... F _ ;

(2) Sample program

N1 G91 G31 X100. F200 ;

N2 X50. Y50. ;



(3) Cautions

(a) By parameter setting, the feed rate set with the parameter is enabled regardless of the F command of the program.

(b) G31 cannot be specified in the tool diameter compensation mode.

(c) When the block next to the G31 command has been given in incremental programming, the next block functions in the incremental manner from the position interrupted by the skip signal.

(d) The G31 command is effective only in the block where it is specified.

(4) Associated parameters

(5) Associated alarms

13-2 Automatic Measurement of Tool Length (G37)

Coordinates at a measuring point specified by G37 is compared with those obtained in actual measurement to use its difference as the wear correction of a tool currently used.

(1) Command Format

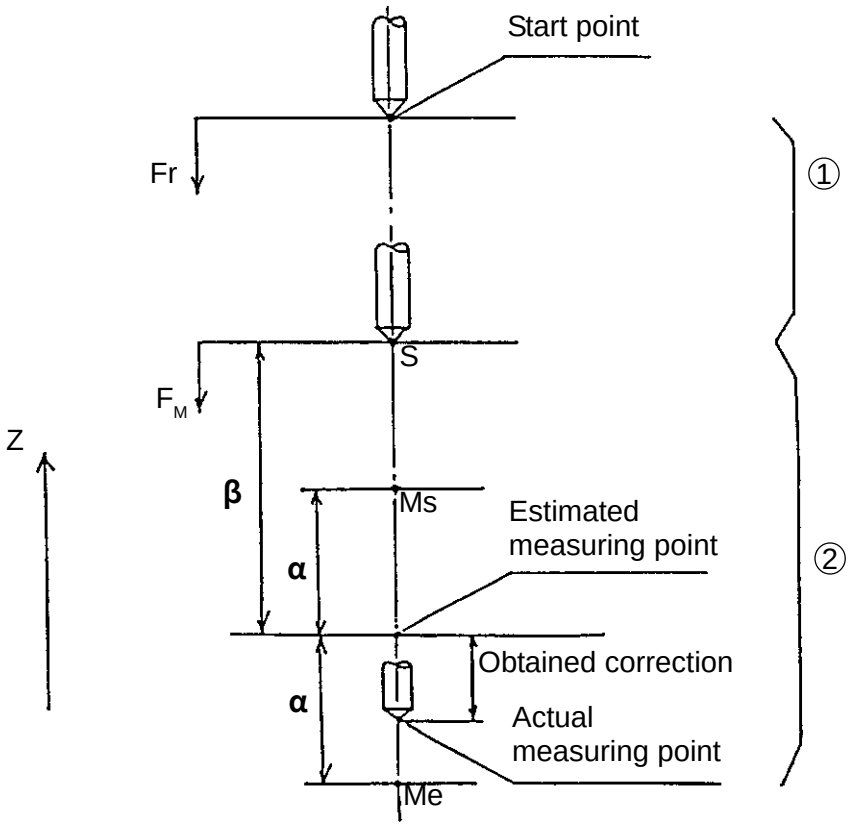
If Z axis is the measuring axis:

G37 Z_ [F_] ; Parameter shown in bracket may be omitted.

In F_ omission, see Parameter No.6276.

(2) Operation

For Z axis



Operation ① : Rapid shift to Point S apart by B from start and estimated measuring points.

Operation ② : Shift at measuring rate Fm Point S. If the measuring point reach signal is not caught within an area remote by (±) α from the estimated measuring point, a value of reached coordinates is stored to skip to a next step. But, if the reach signal is received outside the area or the signal is not received, error occurs.

Corrective calculation :

Coordinate values of the estimated measuring point and the actual measuring point are compared, whose difference is then substituted as the new wear corrective value.

(3) Cautions

(a) Command axis is one of three basic axes.

(b) Command is available only in an absolute value.

(c) With H command given in G37 block, alarm is produced. Place H command before G37 block.

(d) Before G37 command, cancel tool diameter compensation by using G40.

(4) Associated Parameters

No.6276	Fm: Measurement feed rate
No.6277	α: Measuring length
No.6279	β: Measuring speed length

(5) Associated Alarms

No.120	An error exists in tool length auto measurement command.
(#001)	Command has not been given in absolute.
(#002)	An axis other than three basic axes has been specified.
(#003)	Measuring length = 0
(#004)	Measuring length > measuring speed length
(#005)	Measuring distance < measuring speed length
No.121	An error exists in H code command in tool length auto measurement.
(#001)	H command exists in G37 block.
(#002)	H00
No.123	Measuring position reach signal has not been turned ON within the measuring scope.

13-3 Safety Guard (Tool Length)

This function measures the tool length of the tool used for the machining program in the AUTO mode.

(1) Operation method

① Perform zero point return.
② Attach a touch probe to the spindle and reference block to the table.
③ Measure the reference block position.
(Omissible if having been already measured.)
For measuring method, see “5-7 Setting Operation of Tool Setter” of “5. Operation Concerning W Setter” in the operation version or (2) Reference Gauge.
④ Call the machining program in the [AUTO] mode.
⑤ Turn ON the [TOOL LENGTH] button.
⑥ Press [ACTIVATE].
⑦ Execute the machining program’s T code, M06.
⑧ Tool length measurement is carried out with H command.
(Note 1) Tool length measurement is carried out when offset No. has been changed.
(Note 2) Measurement is carried out in M06 when tool offset by tool No. is effective.
(Note 3) As for measuring operation, see (4) Description of Measurement Operation.
(Note 4) Commands except for M00, M01, M02, M06, M30, M98, M99 and T are not available.
(Note 5) Neither M00 nor M01 operate when Parameter No.6243, #3 = 1.
⑨ Date of the safety guard page are cleared with M02 or M03. For the safety guard page, push [F3] (tool)→[F6] (safety guard).
⑩ Set [Tool Length] button to OFF.

(2) Reference gauge

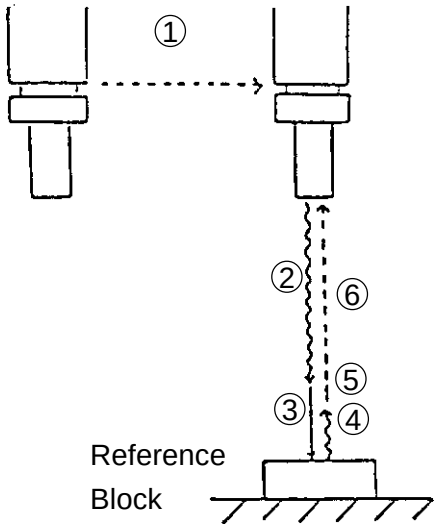
① After setting [Handle] or [Feed] mode, push ON [Tool Setter] button.
② Press [F3] (TOOL), [F9] (SELECT/FUNC), [F2] (REFERENCE GAUGE) in that order.
→ The Reference Gauge Setting screen appears.
③ Put the touch probe into contact with the Z directional contact surface of the reference block. → The buzzer sounds and the parameters No.6251, 6258 and 6259 are rewritten.
④ Relieve the Z axis until the buzzer around goes off.
⑤ Press [F2] (REFERENCE GAUGE) and [F9] (SOFT KEY CLEAR) in that order.
⑥ Set [Tool Setter] button to OFF.

(3) Sample program

```
00001 ;
N1  G54  G90  G00  X0  Y0 ;
N2  G30  G91  X0  Y0  Z0 ;
N3  T01  M06 ;                    ..... Executes T01 and M06.
N4  G00  G90  X100. Y100. ;
N5  G43  Z-100. H1 ;              ..... Measures the H1 tool length.
N6  M98  P2 ;
N7  T02 ;                         ..... Executes T02.
N8  G30  G91  X0  Y0  Z0 ;
N9  M06 ;                         ..... Measures the H2 tool length.
N10  G00  G90  X200. Y200. ;
N11  G43  Z-100.  H2 ;
N12  M98  P2 ;
N13  M30 ;                        ..... Clears the data on the screen.
%

00002 ;
N100  G01  G91  X10.  F200 ;
N101  G02  J-10. ;
N102  G01  Z50. ;
N103  M99 ;
%
```

* The tool whose length has been measured is not measured until M30 or M02 is executed. For example, if the program is restarted from the beginning after it was executed as far as N5, tool length measurement is performed from the next block, invalidating H1 in the N5 block.

(4) Description of Measuring Operation



Tool length measurement

① The X and Y axes move to the reference block position at rapid traverse rate.

② Bring the Z axis close to the reference block by the manual pulse generator or jog feed.
(Operate just in the automatic mode.)

③ Apply the Z axis to the reference block by the manual pulse generator or jig feed. → The buzzer sounds and the offset amount of the H code is rewritten.

④ Retract from the block by the manual pulse generator or jog feed until the buzzer goes off.

⑤ Press the CYCLE START switch.

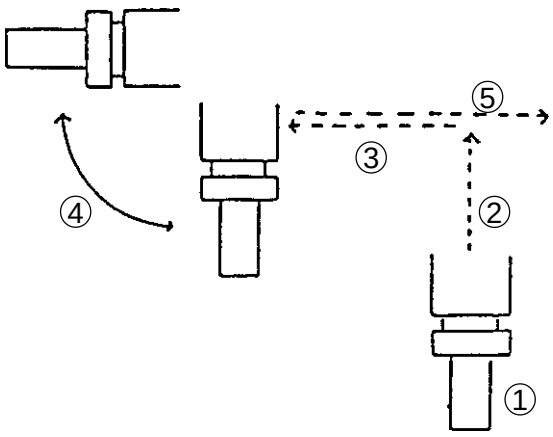
⑥ The Z axis moves to the ATC position at rapid traverse rate and the program proceeds to the next block.

* If the CYCLE START switch is pressed after completing the movement ①, the program proceeds to the next block, canceling the movement ② on.

(5) ATC Operation

Ordinary ATC operations, as mentioned below, vary by machine type, which operate in the same way as ATC canned cycle.

ATC (M06)



① The coolant stops and the spindle stops.

② The Z axis moves to the ATC position at rapid traverse rate.

③ Spindle orientation is performed, and the X and Y axes move to the ATC position at rapid traverse rate.

④ ATC

⑤ The X and Y axes move to the reference block position at rapid traverse rate.

(Note) As with the ATC canned cycle, set the parameters in advance.

(6) Parameters

Refer to 14-4 Safety Guard (Comparison).

(7) Alarms

No.213 Safety guard tool length operation error

[#001] “Tool length” button has been pushed except in reset state.

[#002] After tool length measurement is started , prior to resetting (M02, M30, Reset key, %) being applied, “tool length” button is set OFF.

[#003] Measurement has taken place for 2 or more axes simultaneously.

[#004] On start of tool length measurement for 5-sided machine, a horizontal shaft head is fitted. Or, a horizontal shaft head is fitted while in measurement.

[#005] Either the manual or the handle mode switch has been pressed.

No.715 Measurement error Used in machine-lock state.

(8) Cautions

① Operate tool length measurement with the SINGLE BLOCK switch turned off.

② When using tool offset or multiple corrections using a tool number, the tool length is measured per correction number.

③ Perform operation with machine-lock OFF state.

④ Perform tool length measurement as holding Auto mode.

⑤ As ATC canned cycle is operated in M06, set the parameter for ATC canned cycle beforehand.

⑥ In case of the program using a spare tool, the tool length of the currently selected tool will be measured.

· Sample program (Program using the spare tool)

```
T1100M06 ;
M45 ;
G43Z_ H12/2 H11/1 H10 ;
M46 ;
```

Call No. T1100			Block Skip
Order	Tool	Status	Signal
1	10	Life	0
2	11	Used	1
3	12	Unused	2

If the tool no. 11 has been selected in the sample program above, H11 measurement will be made.

13-4 Safety Guard (Comparison)

This function executes the machining program in the AUTO mode with the X and Y axes moving and the Z axis machine-locked, measures the workpiece profile (Z-axis direction) in an optional Z-axis positioning block, and checks for an interference with workpiece.

(1) Operation method

① Perform zero point returning.

② Mount the spindle touch sensor.

③ Set touch sensor scope. Setting, if having been already done, can be omitted.
As for setting procedures, see “5-7 Setting Operation of Tool Setter” of “5. Operation Concerning W Setter.”

④ Call a machining program in [AUTO] mode.

⑤ Push ON [VERIFY] button.

⑥ Push [START].

* G00 Zxx ;
  Measures the workpiece profile when executing the block containing a Z-axis positioning command, and at the same time, checks for an interference with the workpiece.

* Block other than G00 Zxx ;
  Executed with Z axis machine locked, M06 invalidated and dry run.

(Note) No verification is carried out with G00 Zxx; command which is given while in Special Canned Cycle or in Canned Cycle mode or together with canned cycle cancel command block.

(EX) G80 G00 Zxx ; ... No verification carried out.

⑦ With M02 or M30, data of the safety guard page are cleared.
For the safety guard page, push [F3] (tool) → [F6] (safety guard).

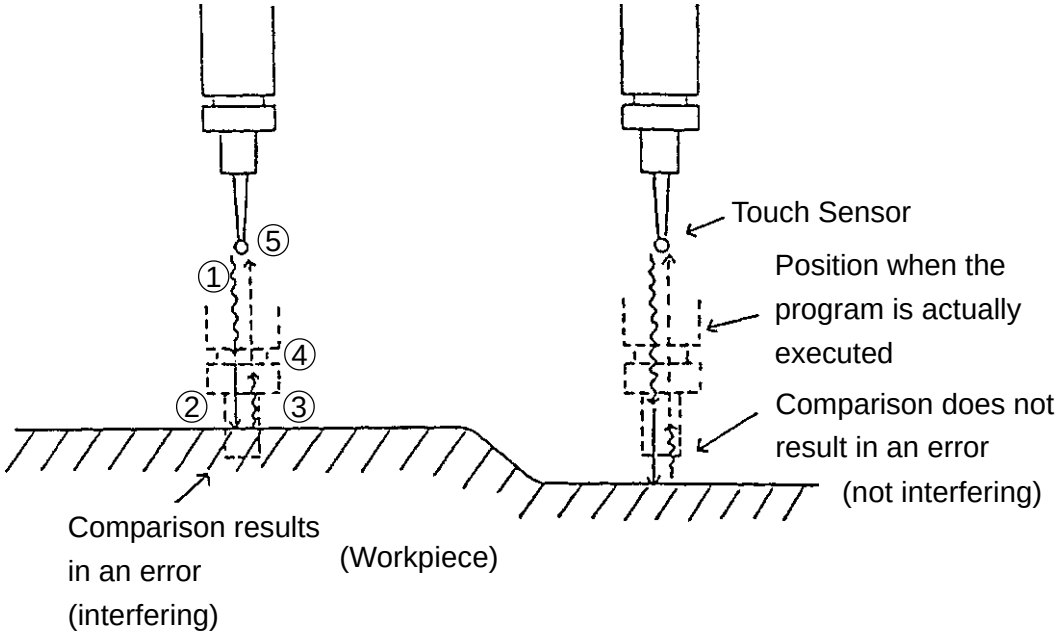
⑧ Set [VERIFY] button to OFF.

⑨ Perform zero point returning of Z axis.

(2) Sample program

```
O0001 ;
N1  G54  G90  G00  X0  Y0 ;
N2  G30  G91  X0  Y0  Z0 ;
N3  T01  M06 ;
N4  G00  G90  X100.  Y100. ;
N5  G43  Z-100.  H1 ;          ..... Comparison
N6  M98  P2 ;
N7  T02 ;
N8  G30  G91  X0  Y0  Z0 ;
N9  M06 ;
N10  G00  G90  X200.  Y200. ;
N11  G43  Z-100.  H2 ;         ..... Comparison
N12  M98  P2 ;
N13  M30 ;                     ..... Clears the data
%                                    on the screen.

O0002 ;
N100  G01  G91  X10.  F200 ;
N101  G02  J-10. ;
N102  G01  Z50. ;
N103  M99 ;
%
```

(3) Comparison movements



① Bring the Z axis close to the workpiece by the manual pulse generator or jog feed.

② Apply the Z axis to the workpiece by the manual pulse generator or jog feed. → The buzzer sounds and the workpiece profile is measured.

③ Retract from the block by the manual pulse generator or jog feed until the buzzer goes off.

④ Press the [CYCLE START] switch. (execution of comparison) → A warning message appears if the position when the program is actually executed interferes with the workpiece.

⑤ The Z axis moves to the ATC position at rapid traverse rate and the program proceeds to the next block.

* If the [CYCLE START] switch is pressed after completing the movement ①, the program proceeds to the next block, canceling the movement ② on.
* If the [CYCLE START] switch is pressed when a warning No. 215 (the tool entered the disabled area in safety guard comparison) occurred, operation can be continued, canceling the warning.

(4) Parameters

No.6243,	#0 = 0	Compares only first G00 Zxx coming after a T command.
	1	Compares all G00 Zxx.
	#1 = 0	The measurement position Z of comparison is the difference between a command value and a measured value.
	1	The measurement position Z of comparison is the work coordinates of the touch position.
	#2 = 0	Does not position to the 2nd reference point at comparison start time.
	1	Positions to the 2nd reference point at comparison start time.
	#3 = 0	M00, M01 is executed on tool length execution.
	1	M00, M01 is ignored on tool length execution.
No.6251		Distance from the reference point to the reference gauge Z end face.
No.6255		Distance from the spindle nose to the -Z directional end face of the probe sphere.
No.6256		Retouch return amount
No.6258		Reference gauge position (X axis)
No.6259		Reference gauge position (Z axis)

(5) Alarms

No.214	Safety guard verification operation error	
[#001]	“VERIFY” button has been pushed ON except in reset state.	
[#002]	After verification is started, prior to resetting (M02, M30, Reset key, %) being applied, “VERIFY” button has been set OFF.	
[#003]	Measurement has been performed for 2 or more axes simultaneously.	
[#004]	On start of verification measurement for 5-sided machine, a horizontal shaft head is fitted. Or, in measurement, a horizontal shaft head is fitted.	
[#005]	Manual or Handle mode switch has been pushed.	
No.215	Safety guard verification error	Tool has entered the disabled area in safety guard verification.
No.715	Measurement error	Used in machine-locked state.

(6) Cautions

① Operate safety guard comparison with the SINGLE BLOCK switch turned off.
② Be sure to perform zero point return after operating safety guard comparison.
③ Perform operation with machine-lock OFF.
④ Perform verification as holding Auto mode.
⑤ Collation does not accommodate the program using the spare tool.

14. DATA SETTING

14-1 Data Setting (G10)

14-1-1 Tool offset amount setting

The tool offset amount can be set by a program command.

(1) Command format

G10 L10 P_ R_ ; Sets the tool length profile offset amount.
G10 L11 P_ R_ ; Tool length wear offset amount setting
G10 L12 P_ R_ ; Sets the tool diameter profile offset amount.
G10 L13 P_ R_ ; Tool length wear offset amount setting

where ; P : Offset number
R : Offset amount

(2) Cautions

(a) Whether the specified offset amount is of absolute/incremental programming depends on G90/G91.
(b) If G10 L1 P_ R_ ; is specified, the tool length profile offset amount is set.
(c) Do not add a decimal point to P.

(3) Associated Alarms

No.100 G10 command has an error.
(#001) Command is insufficient.
(#002) Command value is erroneous.
(#003) Unnecessary command exists.

14-1-2 Work coordinate system offset amount setting

(1) Command format

G10 L2 P_ X_ Y_ Z_ ... R_ ; Sets G54 - G59.
where ; P0 : Specify the external work zero point offset.
P1 - P6 : Specify corresponding to the work coordinate system G54 - G59.
X, Y, Z ... : Work zero point offset amount of each axis
R : Rotation angle
G10 L20 P_ X_ Y_ Z_ ... R_ ; Sets G540 - G599.
where ; P0 : Specify the external work zero point offset.
P1 - P60 : Specify corresponding to the work coordinate system G54 - G59.
X, Y, Z ... : Work zero point offset amount of each axis
R : Rotation angle

G10 L21 P_ X_ Y_ Z_ ... R _ ; Sets the common zero point shift amount.

where ; P0 - P5 : Common zero point shift amount number

X, Y, Z ... : Common zero point shift amount of each axis

R : Length of attachment (effective only for P5)

(2) Cautions

(a) The following commands are also possible.

G10 L2 P_ X_ Y_ Z_ ... R_ ; Sets G54 - G59.

where ; P54-p59: Specify corresponding to the work coordinate system G54 - G59.

G10 L20 P_ X_ Y_ Z_ ... R_ ; Sets G540 - G599.

where ; P540-P599: Specify corresponding to the work coordinate system G540 - G599.

(b) Do not add a decimal point to P.

(c) Absolute/incremental for specified offset amount, rotation angle, etc. follow G90/G91.

(3) Associated Alarms

No.100 G10 has an error.

(#001) Command is insufficient.

(#002) Command value is erroneous.

(#003) Unnecessary command exists.

14-2 Programmable Parameter Input (G10)

The parameters of the NC unit can be input by a program command.

(1) Command format

G10 L50 ;	Programmable parameter input ON
N _ P _ R _ ;	Programmable parameter input mode
G11 ;	Programmable parameter input OFF
where ; N _ ;	Parameter number
P _ ;	Axis number (1 ~ 8 : For axis type parameter)
R _ ;	Parameter set value

(2) Cautions

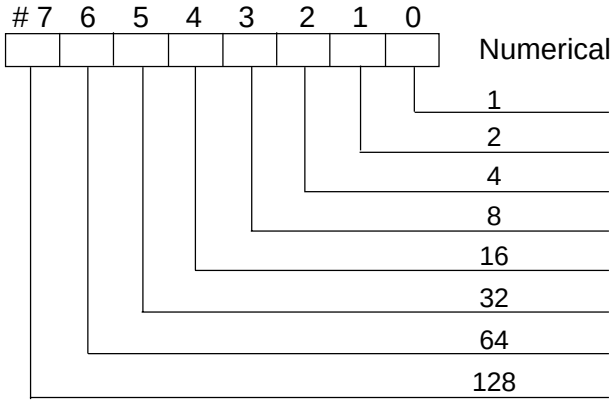
(a) Specify G10 L50 ; and G11 ; commands in a single block.

(b) Other NC statements cannot be specified during the programmable parameter input mode.

(c) Specify the numerical value of the address R without a decimal point.

(d) Some of parameters cannot be made effective unless the power is turned down once.

(e) The setting method of the bit type parameters is as follows.



(Example) When setting #7 to 1 ;
N _ R128 ;

(f) Specify programmable parameter input in the canned cycle cancel state (G80).

(g) When Bit No. Q is specified, set 0 or 1 as R.

(h) Writing of parameters larger than PRA9000 is disabled.

(i) Specify the end of a changed parameter with G11.

(3) Associated alarms

No.100 G10 command has an error.

(#011) Parameter No. error (N)

(#012) Parameter axis No. error (P)

(#013) Parameter bit No. error (Q)

(#014) Parameter set value error (R)

(#015) Unnecessary command exists.

(#016) Unwritable parameter has been specified.

14-3 Plotting Parameter Setting

It is possible to set plotting parameters by the G10 command.

(1) Command Format

G10 L80 P0 ; Plotting screen clear

G10 L80 P1 R_ ; Plotting plane select

R0:XYZ R3:ZX R6 XZ
R1:XY R4:YX R7:XZY
R2:YZ R5:ZY R8:XYXZ

G10 L80 P2 R_ Q_ ; Plotting rotation angle

R_ : Horizontal rotation angle (deg)
Q_ ; Vertical rotation angle (deg)

G10 L80 P3 X_ Y_ Z_ I_ J_ K_ ; Plotting range

Maximum coordinate
Maximum coordinate

(2) Cautions

(a) If plotting parameters are set by the G10 command, the plotting screen is also cleared.

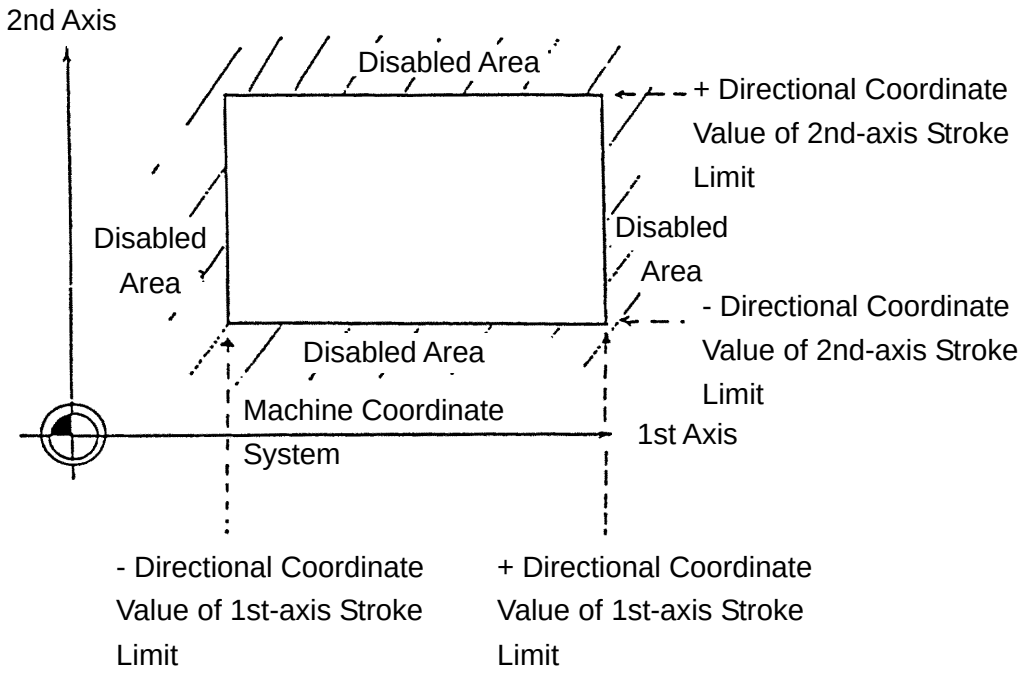
(3) Associated Alarms

No.100 G10 command has an error.
(#061) P command error
(#062) R command error
(#063) Q command error

15. SOFT OT

15-1 Soft OT (Stored Stroke Limit 1)

Each axis has the outside stroke disabled area set by soft. If the axis enters the set disabled area, distribution stops in case of automatic operation, and a move in the disabled direction stops in case of the JOG or HANDLE mode.



(1) If even one axis enters the set disabled area during automatic operation, an alarm results and all the axes decelerate to a stop. In the manual operation, only the axis in the set disabled area displays an alarm and decelerates to a stop.

(2) When the axis enters the set disabled area, move it in the release direction out of the stroke disabled area by manual operation toward the outside of the disabled area, and then, press the RESET switch to reset an alarm.

(3) Cautions

(a) Soft OT has the stroke disabled area set peculiar to the machine.

(b) The stroke limit coordinate value of each axis refers to the position in the machine coordinate system.

(c) The distance required for the axis to stop after entering the disabled area depends on the feed rate, etc.

(4) Associated parameters

No.1320		+ directional coordinate value of the stroke limit 1 of each axis
No.1321		- directional coordinate value of the stroke limit 1 of each axis
No.1300, #6		During a period from supply of power to manual reference point recovery ;
	= 0	Soft OT checking is performed.
	= 1	Soft OT checking is not performed.
No.1300, #7		When there is a command with which Soft OT is exceeded ;
	= 0	Alarm takes place after Soft OT is exceeded.
	= 1	Alarm takes place before Soft OT is exceeded.

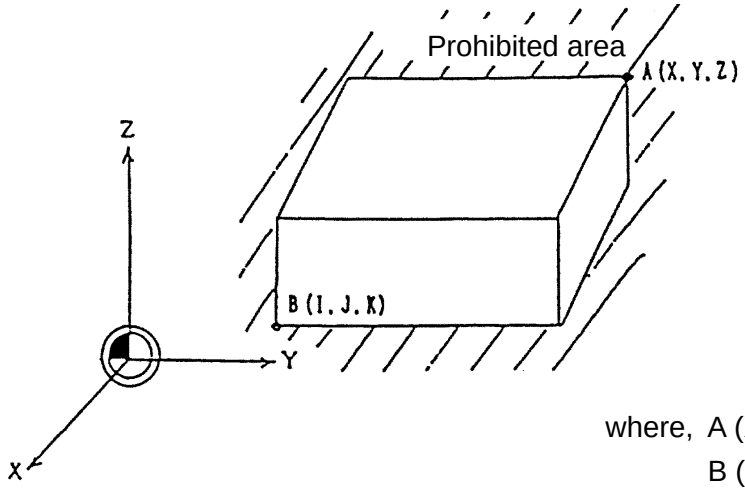
(5) Associated alarms

No.F500	Soft -OT	+ directional disabled area
No.F501	Soft -OT	- directional disabled area

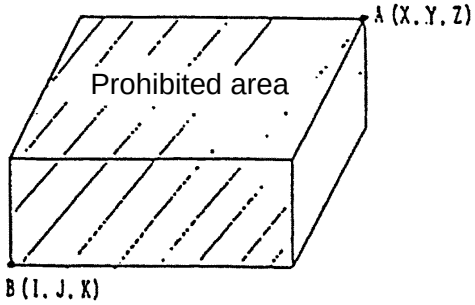
15-2 Stored Stroke Limits 2 and 3 (G22 and G23)

The prohibited area of stored stroke limit 2 can be specified by the G22 command.
With input from the set page, stored stroke limit 2/3 disabled areas can be set.
Upon entering the specified prohibited area, distribution is stopped in the automatic operation mode and movement in a prohibited direction is stopped in the manual feed or handle mode.

(a) Outside prohibited area



(b) Inside prohibited area



where, A (X, Y, Z): Upper coordinate value
B (I, J, K): Lower coordinate value
X > I
Y > J
Z > K

(1) G code
G22 Stored stroke limits 2 and 3 ON
G23 Stored stroke limits 2 and 3 OFF

(2) Command format
(a) “Valid”/”Invalid” of Stored Stroke Limit
G22 ;
With this command, stored stroke limit checking is made valid.
G23 ;
With this command, stored stroke limit checking is made invalid.

(b) Setting of Stored Stroke Limit 2

G22 X_ Y_ Z_ I_ J_ K ;

X : Plus-side boundary of X axis

Y : Plus-side boundary of Y axis

Z : Plus-side boundary of Z axis

I : Minus-side boundary of X axis

J : Minus-side boundary of Y axis

K : Minus-side boundary of Z axis

Inside or outside of the set boundary is the disabled area. Whether it is to be inside or outside is determined by parameters.

(a) There is no need to specify an axis which is not changed.
To change only the minus side of X axis, use G22 I_ ;.

(b) To change both the plus and minus side boundaries, the following conditions must be satisfied:

X > I, Y > J, Z > K

(c) Axes other than X , Y, and Z cannot be changed with a program command.

(d) Boundary can be set also on the set page.

(3) Setting of Stored Stroke Limit 3

A boundary is set on the set page. Inside of the set boundary is the disabled area.

(4) Upon entering the specified prohibited area, in the manual operation mode, the axis is moved from the stroke prohibited area to a cancel direction and then up to the stroke prohibited area. An alarm can be canceled by pressing the reset button.

(5) Cautions

(a) The coordinate values of stored stroke limits at each axis is in the position of the machines coordinate system.

(b) Stored stroke limits 2 and 3 are effective only for axes that have completely been returned to the reference point.

(c) The distance required for the axis to stop after entering the prohibited area varies with the feed speed.

(d) G22 and G23 commands must be specified in a single block.

(6) Relevant parameter

No.1322 Coordinate value in positive direction of stored stroke limit 2 at each axis

No.1323 Coordinate value in negative direction of stored stroke limit 2 at each axis

No.1324 Coordinate value in positive direction of stored stroke limit 3 at each axis

No.1325 Coordinate value in negative direction of stored stroke limit 3 at each axis

No.1300, #0 = 0 Stored stroke limit 2 is at the inside prohibited area

= 1 Stored stroke limit 2 is at the outside stored stroke area

No.1810, #0 = 0 Stored stroke limit 2 at each axis is ineffective

= 1 Stored stroke limit 2 at each axis is effective

No.1310, #1 = 0 Stored stroke limit 3 at each axis is ineffective

= 1 Stored stroke limit 3 at each axis is effective

(7) Relevant alarm

No.F502 Entered the prohibited area of stored stroke limit 2 during movement in a positive direction.

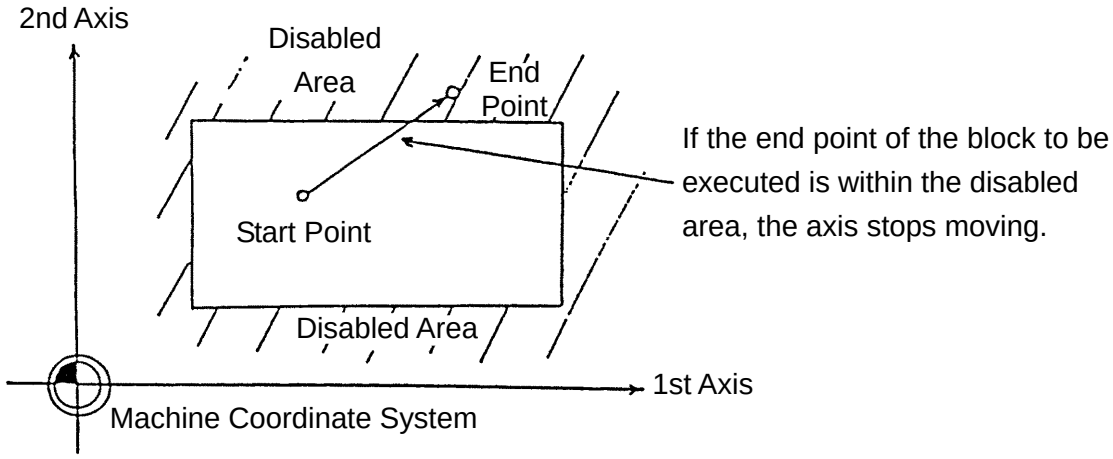
No.F503 Entered the prohibited area of stored stroke limit 2 during movement in a negative direction.

No.F504 Entered the prohibited area of stored stroke limit 3 during movement in a positive direction.

No.F505 Entered the prohibited area of stored stroke limit 3 during movement in a negative direction.

15-3 Soft-OT before Move

In auto operation, when the end coordinate of a block to be executed has entered the set disabled area, distribution is stopped with an alarm indication.
In manual operation, it will be invalidated.



(1) When the “soft-OT before move” alarm results, press the RESET switch to reset it.
(2) The end point of the block to be executed can be calculated out of the machine coordinated + remaining stroke.
(3) Cautions
  (a) The disabled area applicable here indicate those of Stroke Limit 1/2/3.
  (b) A block move path is not checked.
  (c) The axis for which the machine lock function is turned on is not checked.
  (d) Only the axis which has completed reference point return is checked.
  (e) No checking is performed while in the following modes:
    · Cylindrical Interpolation mode
    · Polar Coordinate Interpolation mode

(4) Associated parameters

No.1301, #2	= 0	G31 block is subject to checking.
	= 1	G31 block is not to checking.
No.1301, #7	= 0	The soft stroke limit before move is invalidated.
	= 1	The soft stroke limit before move is validated.

(5) Associated alarms

No.F510	By the stroke check before move, the axis was found in the disabled area in the + direction.
No.F511	By the stroke check before move, the axis was found in the disabled area In the - direction.

16. AXIS CONTROL

16-1 Rotary Axis Controlling Function

It is possible to specify to rotate the rotary table by setting parameters.

(1) Command Format

(a) If Incremental Command

The command value becomes the moving amount.

A720. ; Rotates 720. deg in the positive direction (CCW).

A-720. ; Rotates 720. deg in the negative direction (CW).

(b) If Absolute Command

Selection can be made out of Types A, B, and C through parameter setting.

[Type A] After the end coordinate is rounded within 1 rotation, shifting takes place by an amount equal to difference to the start point.

[Type B] Positioning is performed in the shortest distance within one rotation.

[Type C] Positioning is performed in the command direction within one rotation.

(2) Example of Program of Type A

G90 A0 ; Moves to the position of 0 degree.

A390. ; Moves to the position of 30 degrees by rotating 30 degrees in the positive direction.

A300. ; Moves to the position of 300 degrees by rotating 90 degrees in the negative direction.

A-45. ; Moves to the position of 315 degrees by rotating 15 degrees in the positive direction.

A-0. ; Moves to the position of 0 degree by rotating -315 degrees in the negative direction.

G91 A360. ; Moves to the position of 315 degrees by rotating 360 degrees in the positive direction.

A-360. ; Moves to the position of 0 degree by rotating 360 degrees in the negative direction.

(3) Example of Program of Type B

```
G90 A0      ;   Moves to the position of 0 degree.
    A390.   ;   Moves to the position of 30 degrees by rotating 30 degrees in the positive
                direction.
    A300.   ;   Moves to the position of 300 degrees by rotating 90 degrees in the negative
                direction.
    A-45.   ;   Moves to the position of 315 degrees by rotating 15 degrees in the positive
                direction.
    A-0.    ;   Moves to the position of 0 degree by rotating 45 degrees in the positive
                direction.
G91 A360.   ;   Moves to the position of 0 degree by rotating 360 degrees in the positive
                direction.
    A-360.  ;   Moves to the position of 0 degree by rotating 360 degrees in the negative
                direction.
```

(4) Example of Program of Type C

```
G90 A0      ;   Moves to the position of 0 degree.
    A390.   ;   Moves to the position of 30 degrees by rotating 30 degrees in the positive
                direction.
    A300.   ;   Moves to the position of 300 degrees by rotating 270 degrees in the
                positive direction.
    A-45.   ;   Moves to the position of 315 degrees by rotating 345 degrees in the
                negative direction.
    A-0 .   ;   Moves to the position of 0 degree by rotating -315 degrees in the negative
                direction.
G91 A360.   ;   Moves to the position of 0 degree by rotating 360 degrees in the positive
                direction.
    A-360 .;    Moves to the position of 0 degree by rotating 360 degrees in the negative
                direction.
```

(5) Associated Parameters

No.1012, #0 =	0	Type A for each axis' rotary axis control
	1	Type B for each axis' rotary axis control
No.1012, #1 =	0	Rotary axis control of each axis follows No.1012, #0.
	1	Rotary axis control of each axis follows Type C.
No.1010, #0 =	0	Axis (linear axis) requiring inch-metric switching
	1	Axis (rotary axis) not requiring inch-metric switching
No.1011, #2 =	0	Relative coordinate not rounded by moving amount per rotation.
	1	Relative coordinate not rounded by moving amount per rotation.
No.1011, #2 =	0	Work coordinate value not rounded by moving amount per rotation.
	1	Work coordinate value rounded by moving amount par rotation.
No.1024		Setting of which axis of the basic coordinate system each axis becomes
No.1232		Each axis' moving amount per rotary axis rotation

(Note) Set parameters as follows.

No. 1010, #0 = 1

No. 1011, #0 = 1

No. 1011, #2 = 0

No. 1024 = 0

16-2 Oscillation Function (G113, G114)

With this command, one of the reference axes X, Y and Z in other than the plane specified with G17, G18 or G19 (plane selection) can be reciprocated over the width specified asynchronously.

(1) The reciprocating axes are as follows :

	Oscillation Axis
G17 ;	Z axis in case of Xp -Yp plane
G18 ;	Y axis in case of Zp -Xp plane
G19 ;	X axis in case of Yp -Zp plane

(2) Command format

{G17 / G18 / G19} ;

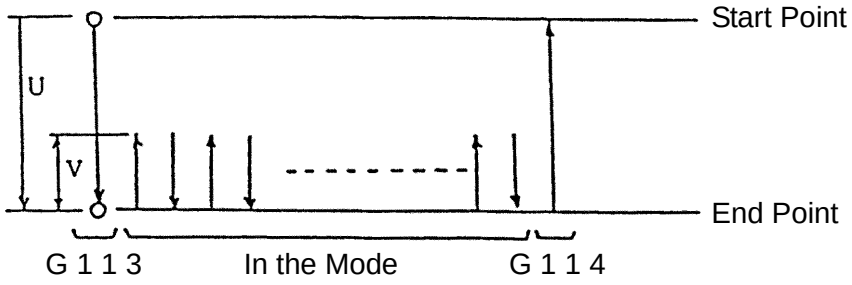
G113 U _ V _ E _ ;

- E _ : Oscillation axis feed rate
- V _ : Oscillation axis return amount (positive value)
- U _ : Oscillation axis end point (incremental value)

This command turns on the oscillation function. Hereafter, the specified axis is reciprocated until G114 is specified.

G114 ;

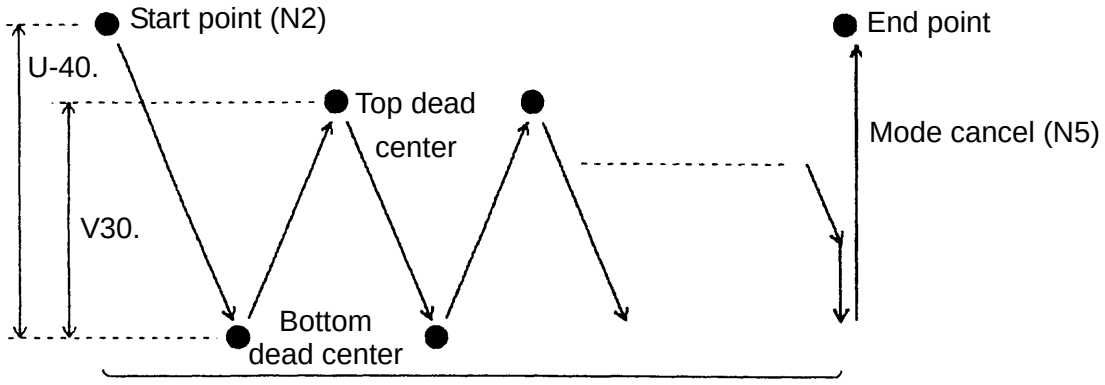
This command causes the oscillation axis to return to the start point at the feed rate specified with F and cancels the oscillation function.



Parameter No.7803, #0 = 0: Positive value. Moving to opposite of U.
Parameter No.7803, #0 = 1: Signed. Moving to sign direction.

(3) Sample program

N1 G17 ; ..... Specifies the oscillation axis.
N2 G90 G00 X0 Y0 Z100. ; ..... Turns on the oscillation function.
N3 G113 U-4. V30. E1000 ; .....
N4 G01 X100. F200 ; ..... In the oscillation mode
(Z axis reciprocating)
N5 G114 ; ..... Cancels the oscillation function

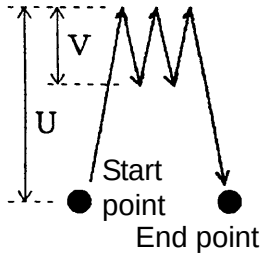
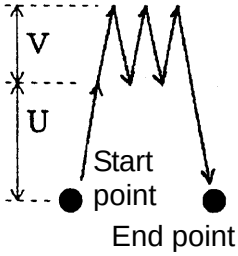
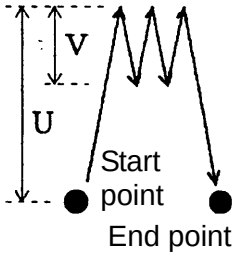
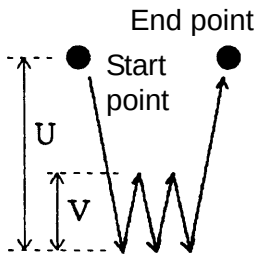
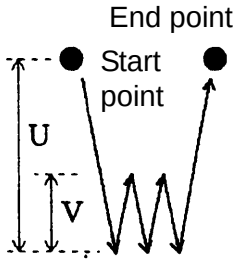
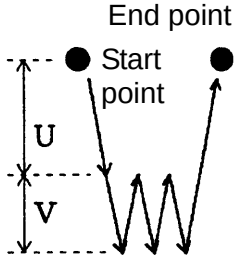


Oscillation mode (N3~N4) * Feed rate ... F200 (x axis), E1000(Z axis)

(4) Cautions

(a) Specify the G113/G114 command in a single block.
(b) If the oscillation axis is specified during the oscillation mode, an alarm results.
(c) When plane selection is changed over during the oscillation mode, an alarm results.
(d) An alarm results if U_ is omitted in the block containing G113.
(e) Use U_ when V_ is omitted in the block containing G113.
(f) Use F_ when E_ is omitted in the block containing G113.
(g) Feed rate override for the oscillation axis and dry run are invalidated in the section of G113 through G114.
(h) If the canned cycle or special cycle is specified during the oscillation mode, an alarm results.
(i) The additional axis cannot be specified as the oscillation axis.
(j) No manual intervention is available while in Oscillation mode.

(k) G113 command operates as follows according to U,V command marks and parameter setting :

	Parameter N0.8656, #0=0 V mark is ignored (Absolute value recovered.)		Parameter N0.8656,#0=1 V command mark used. (MII interchangeable)	
	V(+) command	V(-) command	V(+) command	V(-) command
U(+) command		Same as left.		
U(-) command		Same as left.		

(1) Stop position of an oscillation axis is as follows:

Stop Condition	Stop Position	Resumption
Emergency stop Servo alarm	Stops instantaneously.	Not possible
NC alarm NC reset	Stops after returning to start point.	
Halt Single block stop	Stops after returning to	possible
Cancel command	Stops after returning to start point.	--

(m) To change an operation mode when an oscillation axis is in shifting, halt once and, after the axis is fully stopped, change the mode.
(For example, operated as Memory mode → HALT button → axis stop → FEED mode.)

(n) To carry out oscillation in machine-locked state, do not rewrite the relative coordinate system. If machine-lock is turned ON/OFF in Oscillation mode, alarm starts.

(5) Associated Parameters

No.8656, #0 The second operation of oscillation axis:

= 0 makes return by amount equal to V absolute value.

= 1 shifts in V mark direction.

(6) Associated alarms

No.210 Oscillation alarms

No.210 (#000) G113 has been specified together with other G code.

(#001) An erroneous address has been specified in G113 block.

(#002) No U command exists in G113 block.

(#003) E0 has been specified in G113 block.

(#004) G113 has been specified while in Canned Cycle mode.

(#010) Plane has been changed while in Oscillation mode.

(#011) Canned cycle has been specified while in Oscillation mode.

(#012) Oscillation axis has been specified while in Oscillation mode.

(#013) G113 has been specified while in Oscillation mode.

(#014) G322~G333 have been specified while in Oscillation mode.

(#020) G114 has been specified together with other G code.

(#021) An erroneous address has been specified in G114 block.

(#022) F0 has been specified in G114 block

(#030) Machine ON/OFF has occurred in Oscillation mode.

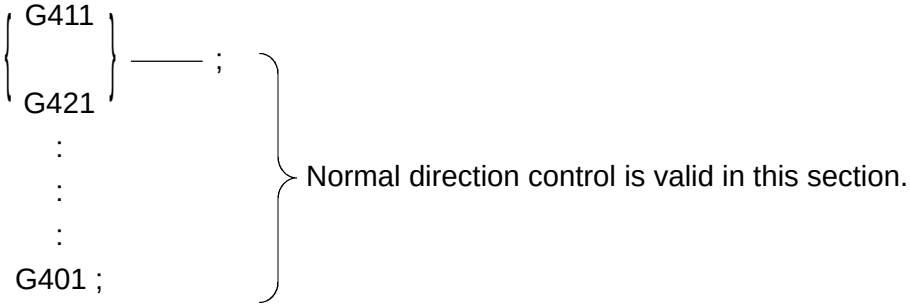
16-3 Normal Direction Control (G411, G421, G401)

With the G411 or G421 command, the rotary axis (C-axis) is always controlled in the normal direction during cutting with respect to X- or Y-axis contouring.

(1) G-code

G411	Normal direction control left ON
G421	Normal direction control right ON
G401	Normal direction control cancel

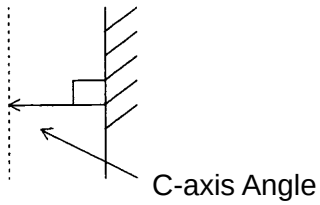
(2) Command format



Normal direction control takes effect after the G411 or G421 command block. It is cancelled by the G401 command.

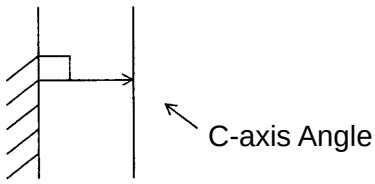
(3) Rotary axis (C-axis) control direction

Advance Direction



G411: Left Side ON

Advance Direction



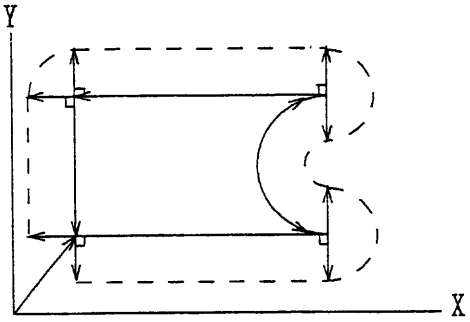
G421: Right Side ON

(4) Rotary axis (C-axis) feed rate

In the normal direction control mode, the feed rate of the rotary axis (C-axis) will assume the speed set with the parameter. In case of the arc command block, the rotary axis (C-axis) will move synchronously with an arc.

(5) Sample program

```
G54 G90 G00 X20. Y20. ;
F421 ;
N1 G01 G90 X100. F500 ;
N2 G02 Y70. R25. ;
N3 G01 X20. ;
N4 Y20. ;
G401 ;
```



(6) Cautions

(a) In the normal direction control mode, the rotary axis (C-axis) always takes a shortcut movement which forms an angle of 180° or less.

(b) In the normal direction control mode, specifying the rotary axis (C-axis) results in an alarm.

(c) To use this function, a helical interpolation option is required.
Helical interpolation cannot be specified during the normal direction control mode.

(7) Associated parameters

No.3421		Axis number of the rotary axis controlled in the normal direction
No.3450		Feed rate of the rotary axis controlled in the normal direction
No.3462		Limit insertion angle of the rotary axis controlled in the normal direction
No.3463		Limit move amount executed at the angle in the previous block in normal direction control
No.3414, #0	= 0	Shortcut enabled in normal direction control
	= 1	Shortcut disabled in normal direction control (Abides by No.3414, #1)
No.3414, #1	= 0	Rpm in normal direction control is plus directional rotation
	= 1	Rpm in normal direction control is minus directional rotation

(8) Associated alarms

No.130 (#002)	The command in the block has an error. The rotary axis (C-axis) was specified during normal direction control.
No.189	Parameter setting has an error.

19. AUTOMATIC OPERATION

19-1 Program Restart

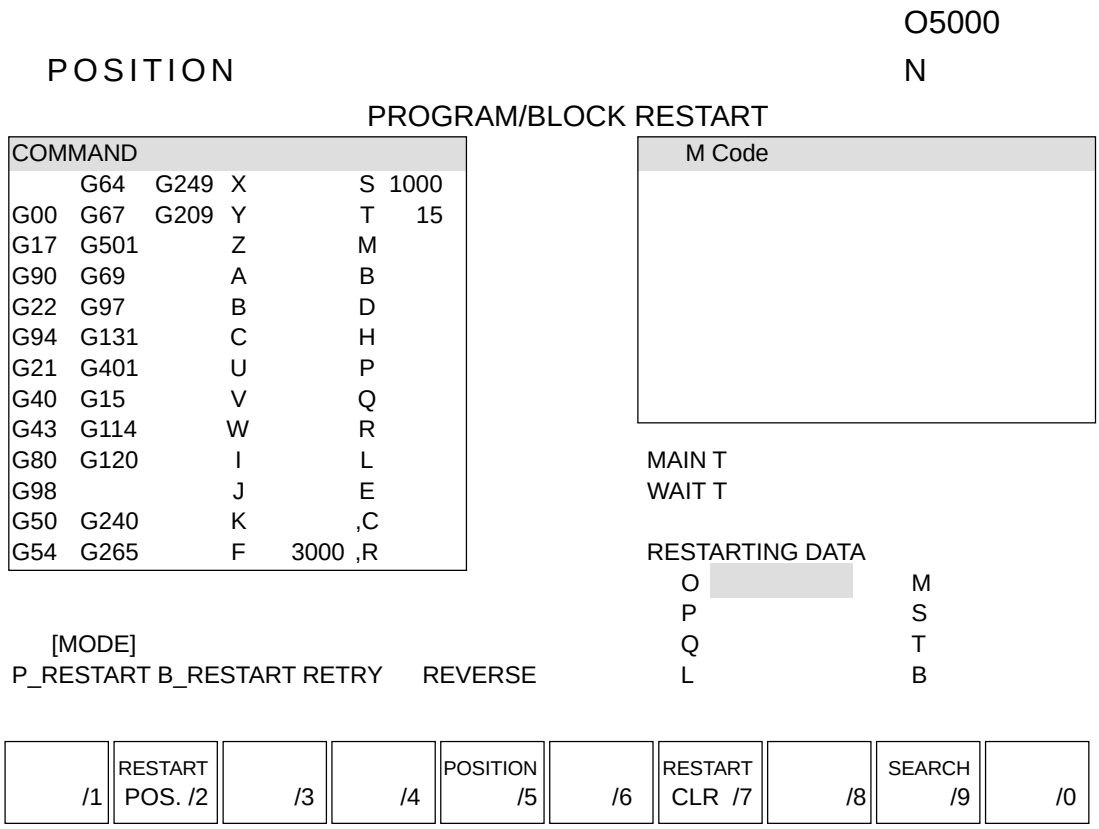
The program can be restarted from the given block by specifying the sequence number and the number of repeats.

There are two types of block restart ; P and Q types.

P type : When the tool is broken

Q type : When the power is turned down

With this function used, machining can be restarted from the block of the specified sequence number when the tool is broken or the power is turned down. It is also available as a high-speed tape check function.



(1) P type (When the tool is broken)

P-type program restart is used when the coordinate system does not change when machining is restarted.

(2) Q type (When restarting machining later in the following cases)

(a) When the power is turned down.

(b) When the EMERGENCY STOP switch is pressed

(c) When the coordinates are altered after interrupting automatic operation.

· When G92 is given from MDI

· When the coordinate system is shifted

· When the automatic coordinate system is set from reference point return

(3) Procedure 1

(a) P Type

① Push HALT .

② Manually retracting a tool, change it with new one. If necessary, change offset amount. (To Procedure 2)

(b) Q type

① When the power is turned on or an emergency stop is cancelled, perform everything required at that time, including reference point return, etc.

② Move the tool to the machining start point by manual operation, set the model information and coordinate system in the same condition as at machining start time.

③ If necessary, set or alter the offset amount. (To Procedure 2)

(4) Procedure 2 (Common to P Type/Q Type)

① Select the EDIT switch of the machine operation panel and press the RESET button of the NC operation panel. The program returns to its beginning.

② Select the automatic mode and press the Program Restart switch of the machine operation panel.

③ Select the Program/Block Restart screen.

④ Specify the block you want to restart.

O No. of the program to be resumed.

P Sequence No. of the block to be resumed in P type.

Q Sequence No. of the block to be resumed in Q type.

L No. of times the sequence number is repeated.

⑤ Pressing the SEARCH key starts a search.

⑥ When the search is completed, the values at [RESTART DATA] disappear.

⑦ Turn off the Program Restart switch of the machine operation panel.

⑧ When you look at the screen and there are the M, S, T and B codes you want to output, select the MDI mode and output the M, S, T and B codes through the Program/ Block Restart screen by operating as follows.

(a) Input, with keys, M, S, T, and B codes.

(b) Push “start” on the machine operation panel.

⑨ When moving form the current position to the restart position, confirm that the tool does not interfere with the workpiece, etc.
If it interferes, select the manual mode and moves it to an interference-free position.

⑩ Select the AUTO mode and press the CYCLE START switch. At this time, the tool moves to the machining restart position one axis by dry run in accordance with the order specified with the parameters, and then, restarts automatic operation from the block of the specified sequence number.

(5) Program/Block Restart screen

Rest position ..... Indicates the machining restart position.

Rest distance ..... Indicates the distance from the current tool position to the machining restart position.

M ..... Displays the M codes specified in the recent 32 times.

Spindle tool T ..... Indicates the tool number of the tool set in the spindle.

Wait tool T ..... Indicates the tool number of the tool set in the standby position.

S ..... Displays the S code specified last.

B ..... Displays the B code specified last.

(Note) To select the screen, press POSITION (F1) , RESTART (F5) in that order.

(6) Cautions

(a) When the FEED HOLD switch or RESET switch is pressed during a search, redo program restart operation from the beginning.

(b) When the Program Restart switch of the machine operation panelis turned on, the CYCLE START switch is ignored.

(c) When moving to the machining restart position one axis by one axis, a single block stop can be applied after completing the movement of one axis. However, MDI cannot be allowed in.

(d) In the following cases, the program cannot be restarted properly.

(i) When intervened manually with manual absolute OFF.

(ii) When operated with the machine lock function applied and the Z-axis ignored.

(iii) When the coordinate system is not set initially in the incremental program.

(iv) When intervened manually halfway moving the axes for return.

(v) When the program is restarted in the block prior to absolute programming after the block where a skip (G31) command was specified.

(vi) When the coordinate system is set, altered or shifted after completing a search.

Note that none of the above cases results in an alarm.

(e) An alarm results when the sequence number cannot be collated.

(7) Associated parameters

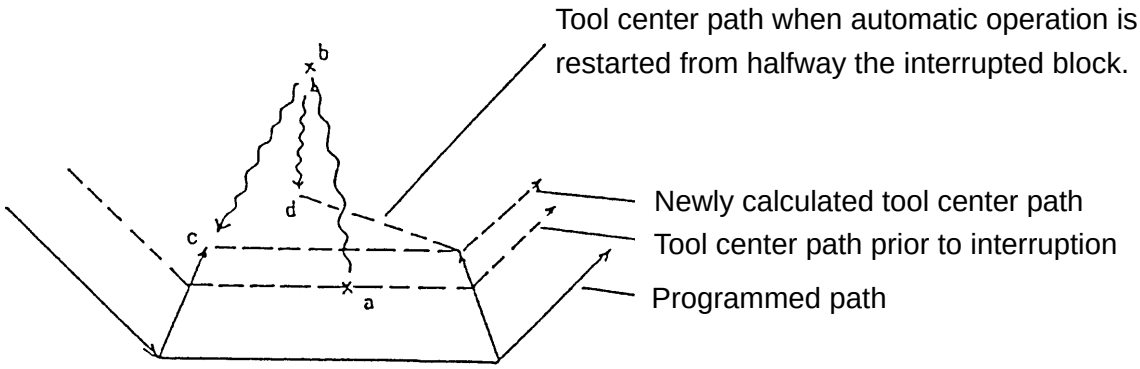
No.8703 Order of the axes which are moved by dry run in restarting the program.

(8) Associated alarms

No.140	Program restart operationis erroneous.
(#001)	Reset start has not been created on start of resumed block searching. (OP=1)
(#002)	HALT signal (SP)=0 on start of resumed block searching.
(#003)	Assigned program does not exist on start of resumed block searching after 0-number has been assigned.
(#004)	When 0-number is not assigned, an executed program has not been called out.
(#010)	Resumption signal (SRN)=0 while in resumed block searching.
(#011)	Resumed block has not been found. (M02/M30 has been detected.)
(#012)	Resumed block has not been found. (% has been detected.)
(#020)	Manual intervention has taken place to the shift completed axis while in approach to the resumption place.
No.189	There is error in the parameter setting. (If parameter No.8703 is not set)

19-2 Block Restart

When a trouble such as tool breakage takes place during machining and automatic operation is interrupted, you can manually relieve the tool from a machining break point, changes the tools, alter the tool offset amount, move the tool to the start point or halfway point of the interrupted block by manual continuous feed (jog), thus restarting automatic operation.



where ;

: Manual operation

: Automatic move

a : Machining break point

b : Retract position (tool changed, offset amount altered, block return ON)

c : Start point of the interrupted block

d : Halfway position of the interrupted block

[Operation Procedure]

(a) A trouble occurred. (tool breakage etc.)

(b) Suspend automatic operation (Press the Feed HOLD switch of the machine operation panel.) However, when press the RESET, the Block Restart is disabled.

(c) Relieve the tool manually, and if necessary, change the tools.

(d) If necessary, after the tool offset amount.

(e) Select the JOG mode and turn on the BLOCK RESTART (BRT) switch.

(f) In the JOG mode, move the tool to the start point of the interrupted block or close to it. When this is done, the tool stop automatically when it reaches the start point of the interrupted block.
(Note) If handle feed, not stopped at the start point of the suspended block.

(g) Select the automatic mode.

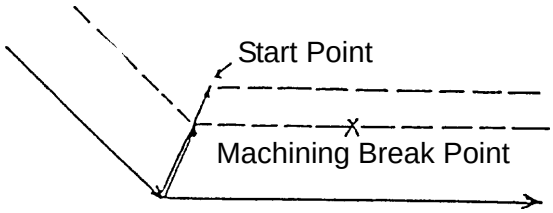
(h) Press ACTIVATE with BLOCK RESTART on.
(i) Turn off BLOCK RESTART after automatic run is restarted.

There are the following two types of block restart :

(a) When the BLOCK RESTART switch is pressed in the manual mode, the start point of the interrupt block is calculated.
(b) When the CYCLE START switch is pressed in the automatic mode with the BLOCK RESTART switch turned on, the tool center path is newly calculated and automatic operation restarts.

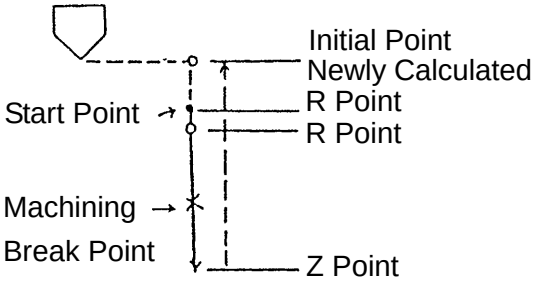
(1) When the BLOCK RETURN switch is pressed in the manual mode
[Operation method]
(a) A trouble such as tool breakage occurred.
(b) Suspend automatic operation by pressing the FEED HOLD switch of the machine operation panel or other method. When the machine is placed in the reset state by pressing the RESET or EMERGENCY STOP switch, the block restart function is disabled.
(c) Relieve the tool manually, and if necessary, change the tools.
(d) If necessary, after the tool offset amount.
(e) Select the JOG mode and turn on the BLOCK RESTART (BRN) switch.
(f) In the JOG mode, move the tool to the start point of the interrupted block or close to it. When this is done, the tool stops automatically when it reaches the start point of the interrupted block.

[Start point of the interrupted block]
(a) During the tool diameter compensation mode



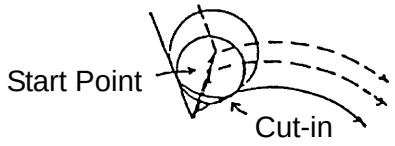
The start point is obtained by calculating the offset vector against a new offset amount from the offset vector at the start point of the interrupted block.

(b) During the canned cycle (G73, G74, G76, G81 ~ G89)



The start point is always a newly calculated R point.

(Note) With an arc command in the tool diameter offset mode, when the start point of the interrupted block is located inside the corner, the workpiece may be cut in or left uncut.



[Program/Block Restart screen]

Rest position ..... Indicates the start point of the interrupted block in terms of the work coordinate system.

Rest distance ..... Indicates the distance from the current tool position to the machining restart position.

B-RESTART ..... In the block restart mode, "*" is added to B-RESTART.

Note) To select the screen, press POSITION(F1). RESTART(F5) in that order.

[Start point of the interrupted block]

The start point of the interrupted block is calculated as follows.

[Programmed absolute coordinates of the specified point]

+

[Tool offset amount previously compensated]

+

[Tool length compensation amount newly calculated]

+

[Tool diameter compensation amount newly calculated]

(2) When the CYCLE START switch is pressed in the automatic mode with the BLOCK RESTART switch turned on.

[Operation method]

(a) A trouble such as tool breakage occurred.

(b) Operate in the same manner as when the BLOCK RETURN switch is pressed in the manual mode mentioned in (1) above, or operate as follows.

(i) Relieve the tool manually, and if necessary, change the tools.

(ii) If necessary, alter the tool offset amount.

(iii) By manual operation, move the tool close to the break point of the interrupted block.

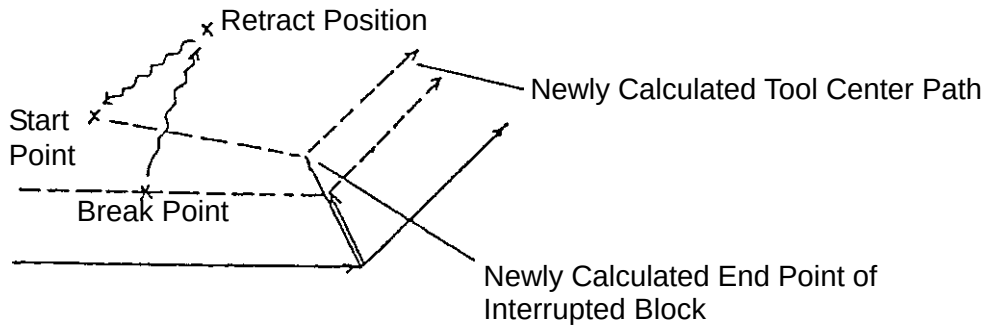
(c) Select the automatic mode, and if the BLOCK RESTART switch is turned off, turn it on.

(d) Pressing the CYCLE START switch restarts automatic operation.

(e) After restarting automatic operation, turn off the BLOCK RESTART switch.

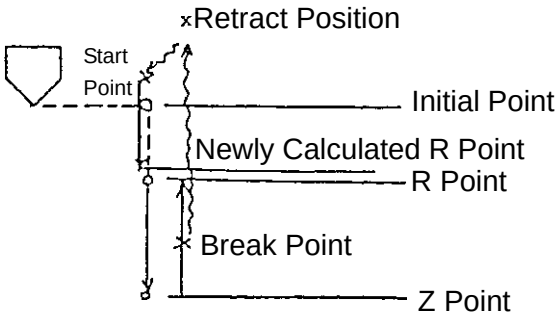
[Moving the interrupted block]

(a) During the tool diameter compensation mode



The end point of the interrupted block is obtained by calculating the offset vector against a new offset amount from the offset vector at the end point of the interrupted block.
Automatic operation restarts from the current tool position toward the newly calculated end point of the interrupted block.

(b) During the canned cycle (G73, G74, G76, G81 ~ G89)



Automatic operation restarts again from the current tool position toward the newly calculated R point.

[End point of the interrupted block]

The end point of the interrupted block is calculated as follows.

[Programmed absolute coordinates of the specified point]

+

[Tool offset amount previously compensated]

+

[Tool length compensation amount newly calculated]

+

[Tool diameter compensation amount newly calculated]

(3) Cautions

(a) In case of the following commands, since the tool does not return to a correct position even if the block is restarted, a warning alarm is displayed. Restarting automatic operation causes the warning alarm to disappear.

- G28, G30, G53, G31
- During the high-speed distributed machining mode (G05)
- Cylindrical interpolation mode (G271)
- Block without a move command

(b) The offset amount should be set before turning on the BLOCK RESTART switch.

(c) During the tool diameter compensation mode or 3-D tool offset mode, the tool does not return to a correct position, even if the block is restarted with the offset amount = 0.

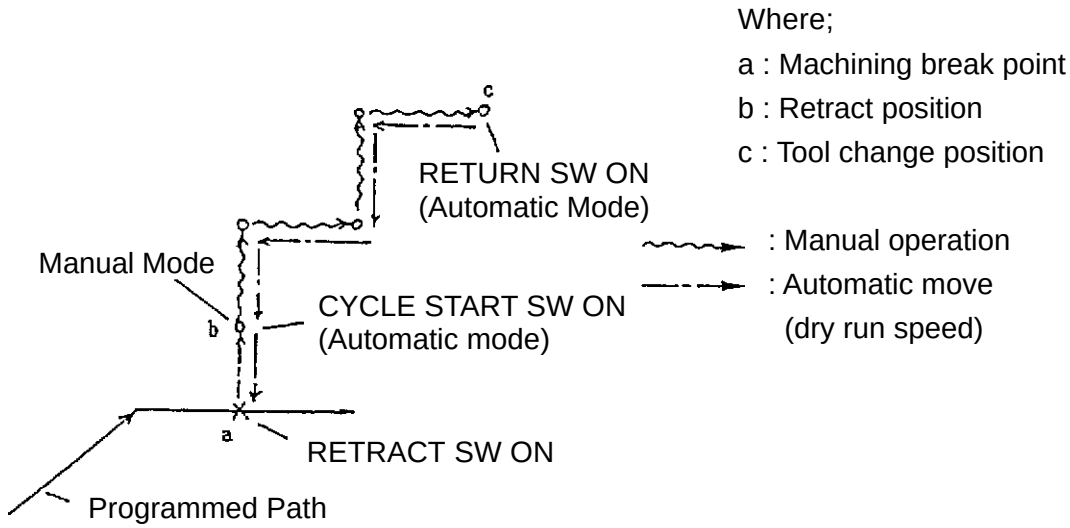
(d) When the tool offset number differs between the machining interrupted block and the preread block, the tool does not return to a correct position even if the block is restarted.

(e) Block restart is disabled in the feed hold disabled area in the tapping cycle (G74, G84, G741, G841), thread cutting mode (G33) or tapping mode (G63).

(f) When the portion from the R point to the machining break point (area ranging from the movement ③ through ⑤) is covered by dividing it into 50 times or more during the canned cycle (G73, G74, G76, G81 ~ G89), an alarm results.

19-3 Machining Break Point Return (G206)

When the tool is broken during machining, you can relieve it from the machining break point, replace it a new tool, return the new tool to the machining break point, and restart machining.



(1) Enabling/disabling machining break point return

$$\left\{\begin{matrix} G90 \\ \\ G91 \end{matrix}\right\} \quad G206\ X_\ Y_\ Z_\ \cdots;$$

This command allows you to specify the machining break point return enabled axes and the retract position for the subsequent blocks.
In case of absolute programming (G90), specify the retract position in the work coordinate system. In case of incremental programming (G91), specify the stroke from the machining break point to the retract position.

G206 ;

This command can disable machining break point return for the subsequent blocks. To disable it, omit all the axis commands in the G206 block.

(2) Operational procedure

① A trouble such as tool breakage occurred. (This function is disabled by pressing the RESET switch.)

② Press RETRACT on the machine operation panel while automatic operation is being started, stopping, or suspended. The tool moves to the retract position specified with the G206 command, interrupting execution of the block. The RETURN POS. lamp is illuminated upon completion of move.

③ Switch to the manual mode.

④ Select the JOG or HANDLE mode and move the axes.
When this is done, an axis select signal is valid for only one axis, and up to 10 manual retract paths can be stored.

⑤ If necessary, change the tools.

⑥ Switch to the automatic mode. If you turn on the RETURN switch of the machine operation panel and turn it off, the tool moves to the retract position along the retract path.

⑦ Change over to the automatic mode and press TOOL RETURN on the machine operation panel. The tool moves to a retract position along a retract route. The RETURN POS. lamp is illuminated upon completion of move.

⑧ If START is pressed at the retract position, the tool returns to a machining break point. Pressing START again resumes automatic operation.

(Note 1) If there are axis selection signals for two or more axes during tool retract by manual operation, Alarm No.193 is issued. Operation can be continued by having the axis selector signal for one axis again.

(Note 2) Even if a tool offset amount is altered after changing tools and the tool is returned to the machining break point, the altered tool offset amount is not taken into account.

(3) Cautions

(a) During the canned cycle (G73, G74, G76, G81 ~ G89) mode, all the retract positions are the R point in the section between the movements ③ and ⑤ (single block disabled). A return continues again from the R point.
However, machining break point return is disabled in the movements ③ through ⑤ of G74/G84/G87.

(b) Machining break point return is disabled in the feed hold prohibited block.

(c) When the BLOCK RESTART switch is pressed during the retract mode, block restart is performed. In this case, subsequent machining break point return is disable.

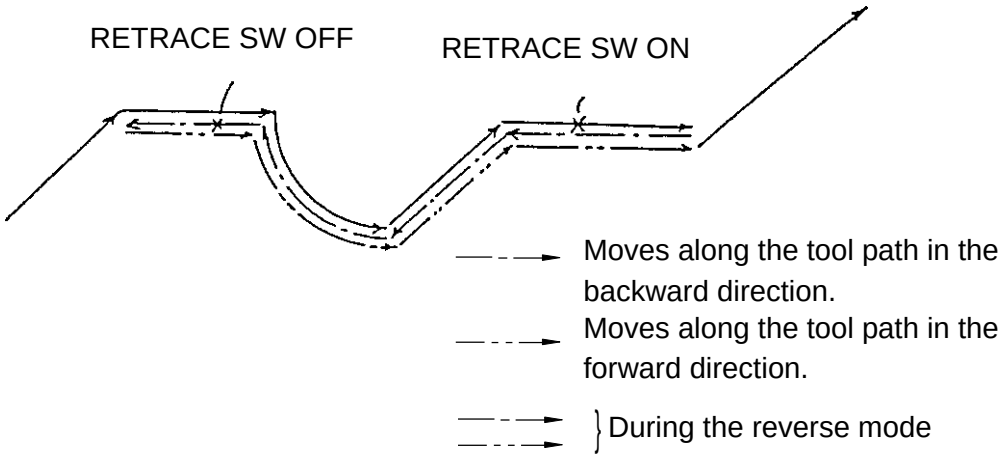
(d) During the canned cycle (G73, G74, G76, G81 ~ G89) mode, if the portion from the R point to the machining break point (movements ③ through ⑤) is divided into 50 or more sections and executed, an alarm results.

(e) If the retract switch is turned on during the G05/G271 mode, an alarm results.

(f) During the return operation, the key Stop must not be pressed. If the key Stop is pressed, the feed hold status is entered and the return operation is discontinued.

19-4 Reverse Movement

If the RETRACE switch is turned on during automatic operation, the tool can be moved backward from the end point of the executed block along the tool path where it has passed. If the RETRACE switch is turned off, the tool can be moved along the original tool path from the end point of this block.



(1) Operational procedure

(a) When automatic operation is starting/stopping

① Turn on the RETRACE switch.
When the block is being executed, the tool moves to the end point, and then, moves backward along the tool path.
The RETRACE lamp of the machine operation panel is illuminated and "*" is added to REVERSE on the Program/Block Restart screen.

② Turn off the RETRACE switch.
The tool moves to the end point of the block being executed when the retrace switch is turned off, and then, moves along the tool path in the forward direction.

③ If you exit the reverse mode, the RETRACE lamp of the machine operation panel goes off and "*" disappears from REVERSE on the Program/Block Restart screen.

(Note) In the reverse mode, single block stop and feed hold (FEED HOLD switch ON) are effective.
When it comes to the block which does not allow reverse movement, the warning alarm No.196 is displayed and operation is switched to forward movement automatically.

(b) When automatic operation is being suspended

① Press the FEED HOLD switch of the machine operation panel.
Automatic operation is suspended.

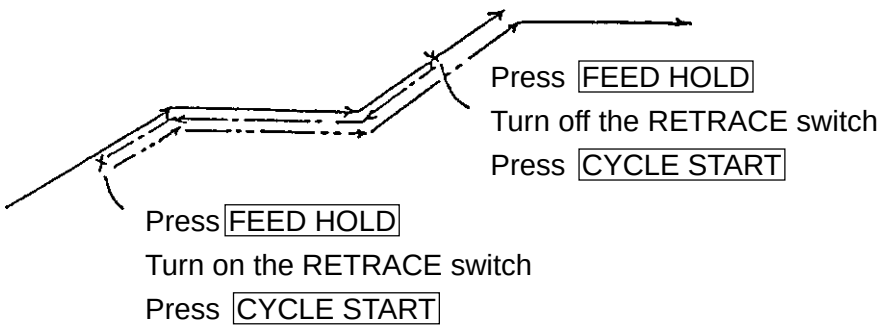
② Turn on the RETRACE switch.
The RETRACE lamp of the machine operation panel is illuminated and "*" is added to REVERSE on the Program/Block Restart screen.

③ Press the CYCLE START switch of the machine operation panel.
The tool moves along the tool path in the backward direction from the break point of the executed block.

④ Press the FEED HOLD switch of the machine operation panel.

⑤ After turning off the RETRACE switch, press the CYCLE START switch. The tool moves along the tool path in the forward direction from the position where automatic operation is suspended.

⑥ If you exit the reverse mode, the RETRACE lamp of the machine operation panel goes off and "*" disappears from REVERSE on the Program/Block Restart screen.



(3) Cautions

(a) Single block stop and feed hold are effective during the reverse mode. When automatic operation is stopped or suspended during the reverse mode, operation can be continued by pressing the CYCLE START switch of the machine operation panel.

(b) During the reverse mode, the M, S, T and B codes are output from the NC unit to the machine.
(c) The tool can move along the tool path in the backward direction at the speed set with the parameter.
(d) Any blocks containing the following commands cannot move backward during the reverse mode.
(e) The number of blocks which allow reverse movement is about 50 blocks for completion of execution.

(4) Associated parameters
No.8686 Feed rate at backward movement (ineffective if 0)

19-5 Sequence Number Comparison and Stop

When you specified the sequence number of the block you want to stop after completion of execution, if the specified sequence number is encountered during program execution, the single block stop state results after executing that block.

(1) Setting of the sequence number
Through setting, enter the sequence number of the block you want to stop after completion of execution. The set sequence number becomes zero when it matches or after reset.

(2) Cautions
(a) It is impossible to compare and stop with the sequence number in the block containing the command which is processed only inside the NC unit.
(b) When one program contains multiple identical sequence numbers, the machine stops after executing the program which matches first.
(c) When the sequence number of the block where the number of repeats has been specified matches, the machine stops after executing the last repeat.
(d) Sequence number comparison and stop is disabled during the high-speed distributed machining (G05) mode.

(3) Associated parameters

No.3474 Sequence number for sequence number comparison and stop

19-6 Reset (Reset Associated with Automatic Operation)

Pressing the RESET switch resets the NC unit. The NC does the following.

(a) Deletes the preread buffer and execution buffer.
(b) Initializes the G command.
(c) Cancels tool length compensation and tool diameter compensation (does not perform offset operation).
(d) Erases the numerical value of the F code.
(e) Resets an alarm.
(f) Stops an axis move.

19-6-1 Details of Reset (Reset Associated with Automatic Operation)

Pressing the RESET switch resets the NC unit. The NC unit provides the following processings.

(1) Cancelling the offset amount of the local coordinate system (G52).
Eliminating the local coordinate system offset amount of the work coordinate system.

(2) Deleting the preread buffer.
Deleting the [NEXT] block.

(3) Initializing the execution buffer [COMMAND].
(a) G command
Group 00 : Deletion
Group 01 : Follows the parameter No.2402, #7 and #0.
Group 02 : Follows the parameter No.2402, #7 and #5.
Group 03 : Follows the parameter No.2402, #7 and #1.
Group 04 : (G23 when no option is added)
Group 05 : Follows the parameter No.2402, #7 and #4.
Group 06 : Follows the parameter No.0001, #0.
Group 08 : Follows the parameter No.2404, #7, #3 and #2.
Group 12 : Follows the parameter No.2402, #7.
Group 17 : Follows the parameter No.2402, #7.
Group 24 : Follows the parameter No.2402, #7.
Group 26 : Follows the parameter No.2402, #7.

(Note) All the G codes other than those above are initialized.

(b) The numerical values of the addresses 0, N, M and B2 are held.

(c) The numerical values of the addresses H, S, T and F follow the parameter No.2402, #7.

(4) Tool diameter compensation and 3-D tool offset are cancelled (no offset is performed).

(5) Tool length compensation follows the parameter No.6000, #5.

(6) Associated parameters

No.3402 , #0 = 0 — Reset state : G00

1 — G01

, #1 = 0 — G91

1 — G90

, #2 = — 1 0 0 1

, #3 = — 0 1 0 1

G43 G44 G49

, #4 = 0 — G94

1 — G95

, #5 = 0 — G17

1 — G18

, #7 = 0 — G00 ~ G03, G17 ~ G19, G54 ~ G59 — Clears

1 — G93 ~ G95, G96 ~ G97, G90 ~ G91 — Does not clear
G43 ~ G49, F, H, S, T.
G248 ~ G249, G240 ~ G245

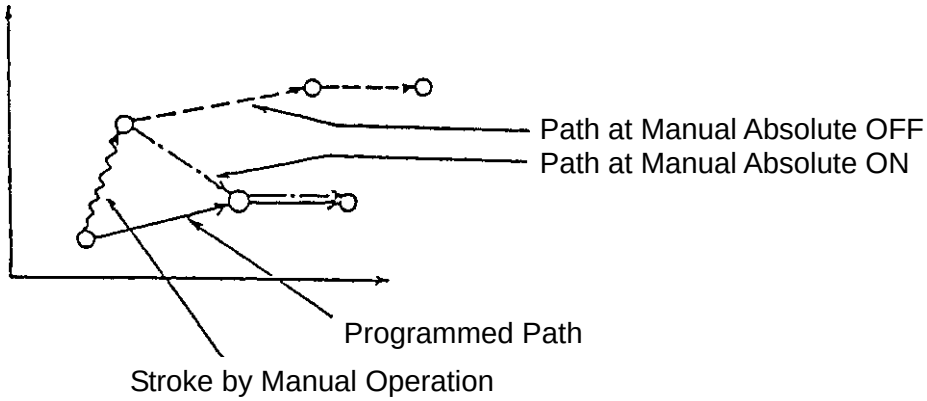
No.5002 , #5 = 0 — Reset state : Clears the tool length compensation vector.

1 — Does not clear the tool length compensation vector.

20. MANUAL OPERATION

20-1 Manual Absolute ON/OFF

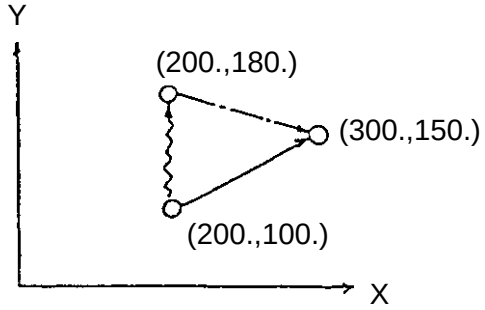
If manual absolute is turned on, the stroke by manual operation is added to the program coordinate value (work coordinate, machine coordinate, relative coordinate), and the then manual intervention amount is generally processed at next block execution time.
If manual absolute is turned off, the stroke by manual operation is not added to the program coordinate value. Therefore, the tool is shifted by the manual intervention amount with respect to the original program.



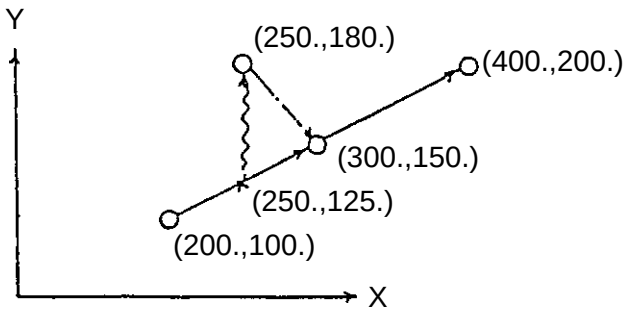
(1) Sample program for manual absolute ON

```
N1  G01  G90  X200. Y100. F500 ;
N2  X300. Y150. ;
N3  X400. Y200. ;
N4  Y250. ;
```

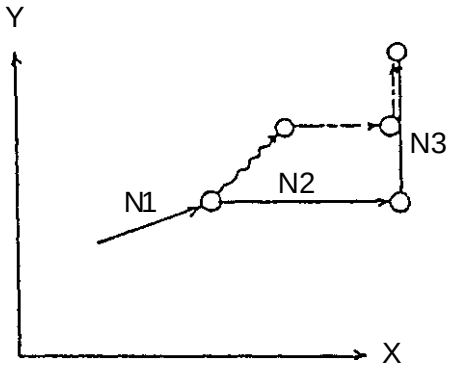
(a) When manual intervention is applied upon completion of move in the N1 block, and the N2 block is executed after moving the Y axis by +80.



(b) When the FEED HOLD switch is pressed during execution of the N2 block, and the CYCLE START switch is pressed again after making manual operation intervene to move the Y axis by +80.



(2) When the X and Y axes are moved by intervention of manual operation at manual absolute ON, only the axis specified in the next one-axis command returns.



(3) Cautions

(a) When the next block is of incremental programming, you can set with the parameter whether to process the manual intervention amount by manual operation.

(b) When processing the manual intervention amount by manual operation in the next block, offset of the manual intervention amount is invalidated if the next block is circular interpolation or helical cutting.

(c) To return the manual intervention amount by the axis command of axis move 0, it is necessary to set to parameter No.2403, #6 = 1.

(4) Associated parameters

No.3403, #4 =	0	The G91 specified block next to manual intervention at manual absolute ON assumes the same path as G90.
	1	Same path as at manual absolute OFF
No.3403, #6 =	0	To return the manual intervention amount, a command block to move the axis.
	1	To return the manual intervention amount, the axis' command block.

(5) Associated alarms

21. TEST RUN

21-1 Miscellaneous Function Lock

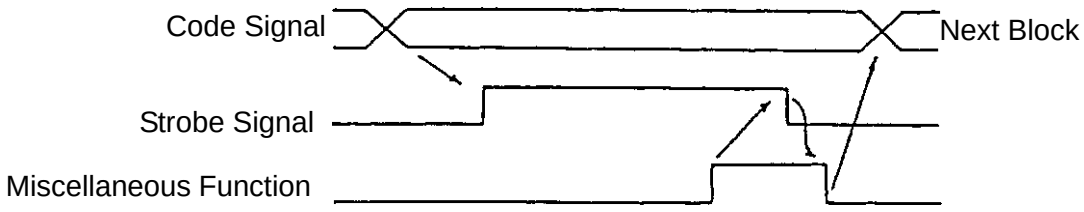
If the miscellaneous function lock of the machine operation panel is turned on, the M- , S- , T- and B-code (2nd miscellaneous function) commands are invalidated.
This is normally combined with the machine lock function to check the NC program.

(1) If the miscellaneous function lock is turned on, the code signal and stroke signal are not output.
(2) The following M codes are effective even if the miscellaneous function lock is turned on.
M00 , M01 ,
M02 , M30 ,
M98 , M99 ,
(3) When the miscellaneous function lock is turned on during automatic operation, it is effective from the block next to the one being executed.

21-2 Miscellaneous Function Finish

If the miscellaneous function(M), spindle function(S), tool function(T) or 2nd miscellaneous function, the code signal and strobe signal are output to the machine side. When all the specified actions are completed, the machine notifies the NC unit of miscellaneous function finish(FIN).

(1) Operational sequence



After the code signal is output, the strobe signal is output, which is delayed by the time set with the parameter.
If miscellaneous function finish is received over the time set with the parameter, the strobe signal is turned off.
The NC unit proceeds to the next block after confirming that miscellaneous function finish is turned off.

(2) Cautions

(3) Associated parameters

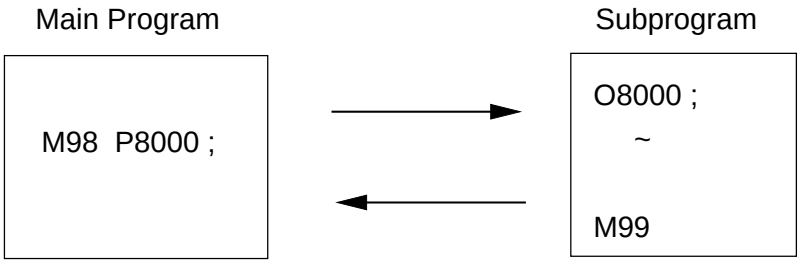
No.3017	Strobe signal delay time
No.3018	Miscellaneous function finish (FIN) acceptance width

(4) Associated alarms

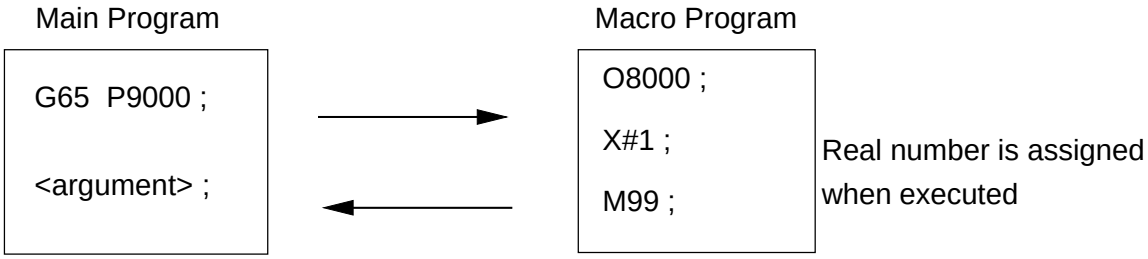
22. CUSTOM MACROS

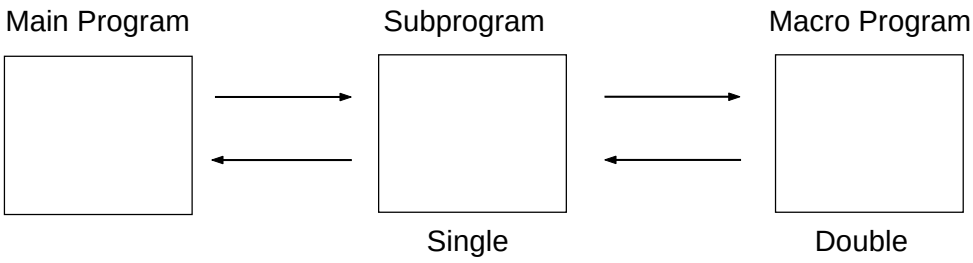
22-1 Outline

A pattern, which is repeatedly used in the program, is registered in the memory as a subprogram in advance. That registered subprogram can be called with a representative instruction and executed. This representative instruction is referred to as a subprogram command.

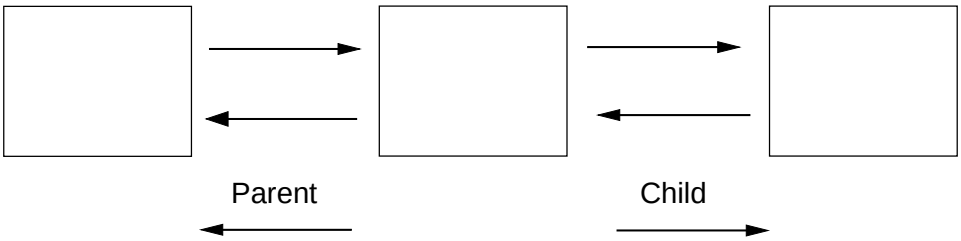


A pattern, which has part of the program replaced with variables, is registered in the memory as a macro program in advance. That registered macro program is called with a representative instruction and can be executed with real numbers assigned to variables. This representative instruction is referred to as a macro call instruction.





It is possible to call another program from a called program. This is called multi-call.
When a certain program is calling another program, that called program is referred to a child program.
When a certain program is being called from another program, that calling program is referred to a parent program.



22-2 Call Commands and Return Command

The following table shows the program call and return commands.

No.	Call/Return	Code Used
1	Subprogram call	M98
2	Macro simple call	G65
3	Macro modal call	G66, G67
4	Arbitrary G code call	Gxx (parameter)
5	M code macro call	Mxx (parameter)
6	M code subprogram call	Mxx (parameter)
7	T code subprogram call	Txxxx
8	S code subprogram call	Sxxxx
9	2nd miscellaneous function subprogram call	Bxxxx (2nd miscellaneous function code)
10	Return to parent program	M99

22-2-1 Types of Command

(1) Subprogram call

M98 P.... Q.... L.... ;

This command calls the program beginning with the sequence number Q of the program number specified with P and runs it L-times. If P is omitted, the commend calls the program beginning with the sequence number Q of the current program number.

If Q is omitted, the command calls the program beginning with the top of the program number specified with P. If L is omitted, the program will be run once. No argument can be specified.

If there is any other address than O, N, P, or L in the identical M98 block, the machine will make a single block stop at that block.

Otherwise, it will not stop. Multiplicity of local variables does not change. Multiplicity of subprograms is allowed up to octuple, together with the macros.

(2) Macro simple call

M65 P ..... L ..... < argument > ;

This command calls program whose program number was specified with P and executes it L times. If L is omitted, the program is executed once. The argument can be specified Specify G65 before all the address other than O and N.

Multiplicity of local variables increases by one. Multiplicity of macros is allowed up to quadruple.

(3) Macro modal call

G66 P ..... L ..... < argument > ;

This command can specify the macro call mode.

G67 ;

This command can cancel the macro call mode. In the block containing a move command during the macro call mode, the specified macro is called after executing that move command.

G66 P9800 < argument > ;

X ..... ;
M ..... ;
Z ..... ;

The macro is called in the block containing a move in this portion.

G67 ;

Specify G66 before all the addresses other than O and N.

Specify G67 in an independent block.

Specify G66 and G67 as a pair within the same program.

(4) Arbitrary G code call

The Gxx < argument > ; command can be the same as G65 P.....< argument > ; or G66 P..... < argument > ; If a P code is specified, it is stored in #16 as an argument.

In this case, any 10 sets of G code can be set in the parameters out of G01 through G999.

Specify the G code for macro call before all the addresses other than O and N.

Parameter			G Code Calling
PRA	6030		O9010
	6031		9011
	6032		9012
	6033		9013
	6034		9014
	6035		9015
	6036		9016
	6037		9017
	6038		9018
	6039		9019

When parameter setting is 0, arbitrary G code call is not done. That is, the macro cannot be called with G0. When parameter setting is a positive number, simple call is done, and when it is a negative number, modal call is done.
For example, when PRA7020 = -100, 09010 is called by G100, that is, modal call.

(5) Macro call by M code

Mxx < argument >;
This command can call the macro.
In addition to Type I and Type II, G, P and L are available as an argument; G: #10, L : #12, P : #14.
In this case, any 10 sets of M codes can be set in the parameters out of M01 through M99999999.
Specify the M code for macro call before all the addresses other than O and N. The MF and M codes are not sent out.

Parameter			G Code Calling
PRA	6050		O9010
	6051		9021
	6052		9022
	6053		9023
	6054		9024
	6055		9025
	6056		9026
	6057		9027
	6058		9028
	6059		9029

When parameter setting is 0 or a negative number, macro call by M code is not done.

(6) Subprogram call by M code

Mxx ;

This command can call the subprogram. No argument can be specified. In this case, any 9 sets of M codes can be set in the parameters out of M01 through M999999.

The MF and M codes are not sent out.

Parameter			G Code Calling
PRA	6050		O9001
	6042		9002
	6043		9003
	6044		9004
	6045		9005
	6046		9006
	6047		9007
	6048		9008
	6049		9009

When parameter setting is 0 or a negative number, subprogram call by M code is not done.

(7) Subprogram call by T code

Txx ;

This command calls the program O9000 as the subprogram.

The T code becomes the argument of the common variable #149.

Other arguments than the above cannot be specified.

TF and T codes are not sent out.

Parameter

	7							0
PRA6000								TCS

TCS = 0 : Does not call the subprogram by the T code.

1 : Calls the subprogram by the T code.

(8) Subprogram call by the S code

Sxx ;

This command calls the program O9029 as the subprogram.

The S code becomes the argument of the common variable #147.

Other arguments than the above cannot be specified.

SF and S codes are not sent out.

Parameter

	7							0
PRA6000							SCS	

SCS = 0 : Does not call the subprogram by the S code.

1 : Calls the subprogram by the S code.

(9) Subprogram call by the 2nd miscellaneous function code

With a 2nd miscellaneous function code, the program O9028 is called as the subprogram. Set the address of the 2nd miscellaneous function in the parameter PRA1020. For example, if the address of the 2nd miscellaneous function is B, the command Bxx ; calls the subprogram. The 2nd miscellaneous function code becomes the argument of the common variable #146. Other arguments that the above cannot be specified. BF and 2nd miscellaneous function codes are not sent out.

Parameter

PRA1020	

Sets the address which specifies the 2nd miscellaneous function.

Address	Set Value
A	65
B	66
C	67
U	85
V	86
W	87

Parameter

	7							0
PRA6000						BCS		

BCS = 0 : Does not call the subprogram by the 2nd miscellaneous function code.

= 1 : Calls the subprogram by the 2nd miscellaneous function code.

(10) Return from the program

M99 ;

This command causes you to return from the currently executed subprogram or macro program to the parent program.

When the same block as M99 contains the address other than O, N, P and L, the machine stops at that block(single block stop); otherwise, it does not stop.

When M99 ; is specified in the main program, control branches to the beginning of the main program, and therefore, operation can be repeated.

M99 Pxxxx ; causes you to return to the block of the sequence No.Nxxxx in the parent program.

When M99 Pxxxx ; is specified in the main program, control branches to the sequence No.Nxxxx in the main program.

M99 Pxxxx takes longer execution time.

The M99 Lxxxx ; command forces to L times the number of remaining repeat times of the subprogram or custom macro. L is ignored when this command is specified in the main program.

22-2-2 Multi-call

(1) Multiplicity

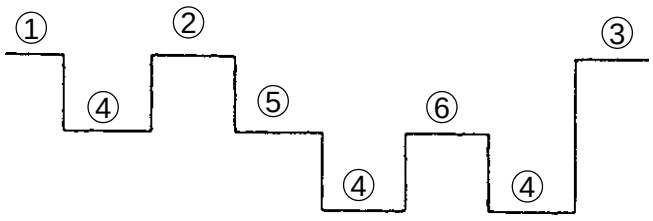
The custom macro can be called up to the quadruple level. The Subprogram can be called up to the octuple level in combination with the multiplicity of the custom macro.

(2) Modal multi-call

When modal macros are multiply specified, the next macro is called every time a macro move command is executed. Macros are sequentially called starting at the one specified last.

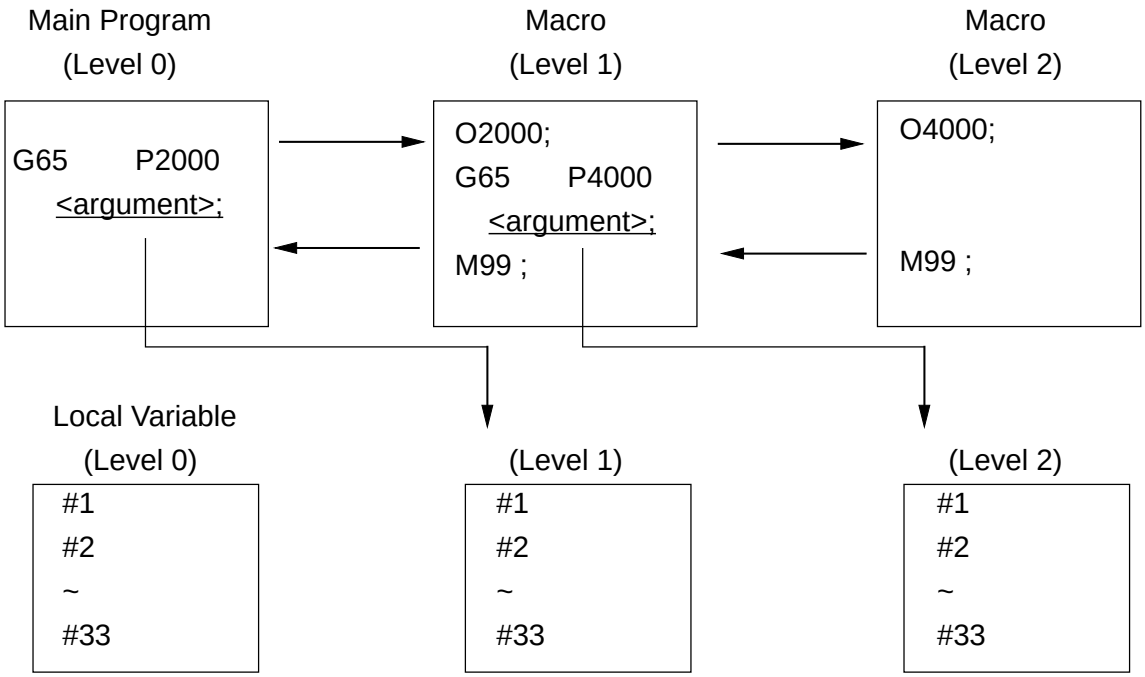
```
G66P1000;
Z1000;    ①
G66P2000;
Z1500;    ②
G67             P2000 cancel
G67             P1000 cancel
Z2500;    ③
 ~

1000;
X5000;    ④
M99;
%

O2000;
Z600;     ⑤
Z500;     ⑥
M99;
%
```

The program above is executed in the following order.



(3) Macro multiplicity and local variable

If the macro is called, macro multiplicity (level) increases by one. The local variable level also increases by one, accompanying it.



① If the macro is called, the local variable of the parent program is stored, and that for the child program is newly provided.
The argument can be transferred to this by specifying the argument.

② If you are returned from the child program to the parent program by M99, the local variable stored in ① is provided in the same state as stored one.

Note) Subprogram call does not change macro multiplicity. That is, the local variables do not change before and after a call.

(4) Modal call and local variable succession

The local variable of the macro called by modal call is succeeded to during that modal call mode.

Parent Program	Local Variable	Child Program	Local Variable
#1=0	#1=0	Argument transfer	
G66P1000A1.0;			
	#1=0		
		O1000;	#1=0
Z1000;			
		#1=#1+1;	#1=2
		M99;	
	#1=0		Succeeded
		O1000;	
Z2000;			#1=2
		#1=#1+1;	
	#1=0	M99	#1=3
G67;			
	#1=0	Arfument tranfer	×Not
G66P1000A1.0;			Succeeded
	#1=0		
Z4000;		O1000;	#1=1
	#1=0		
		#1=#1+1;	#1=2
G67;		M99	

With the parameter, it is possible to disable local variable succession. In this case, <argument> data in the G66 block is transferred at every modal call.

Parameter 7000

7	6	5	4	3	2	1	0
		LVCON					

0 :Succeeds to the local variable by modal call.

1 :Does not succeeds to the local variable by modal call.

(5) When making the special call multiply

Arbitrary G code call, M code macro call, M code subprogram call, T code subprogram call, S code subprogram call, and 2nd miscellaneous function code subprogram call are referred to special calls.

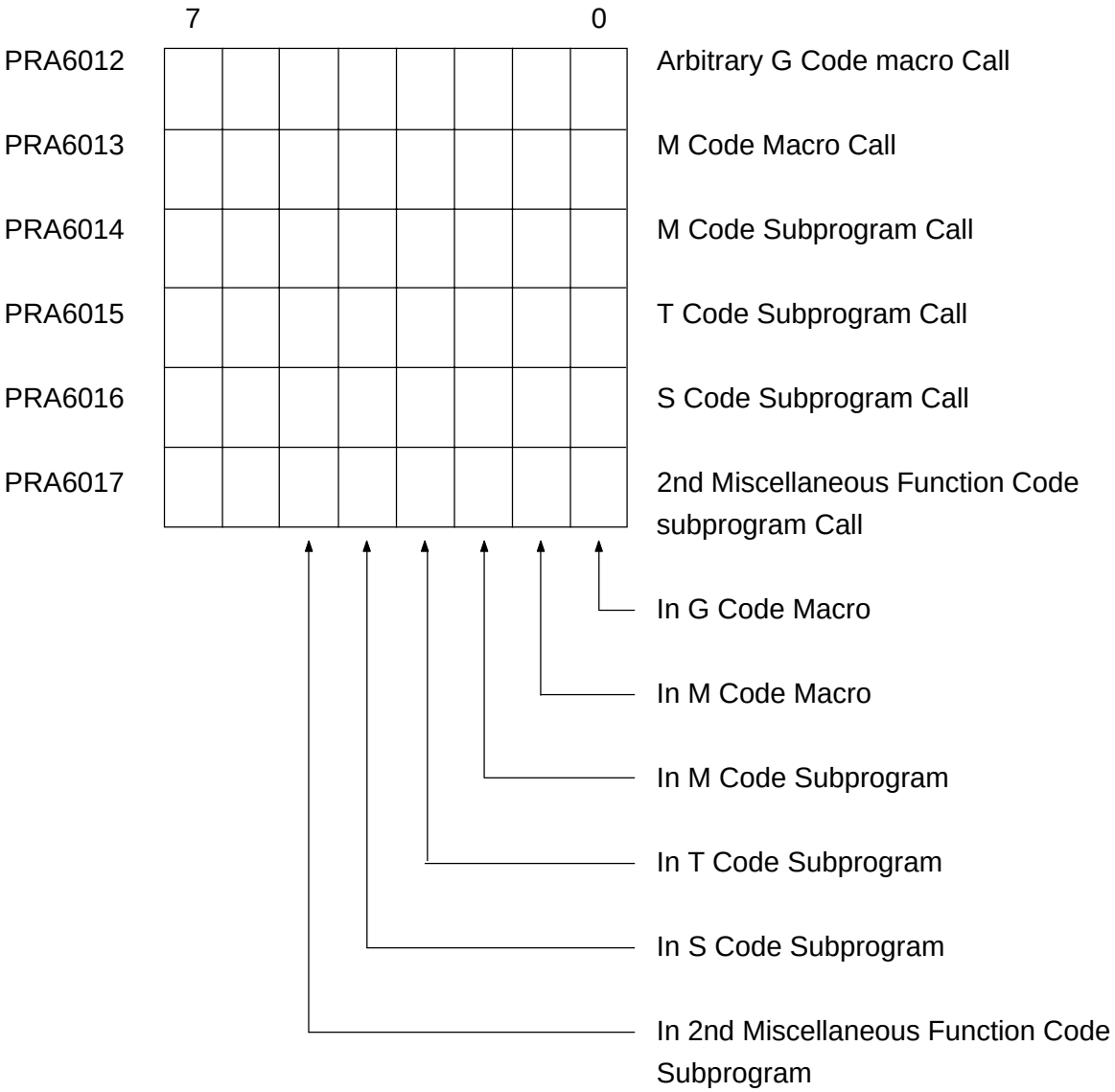
Identical special call cannot be made multiply. For example, if arbitrary G code call is specified in the program called by arbitrary G code call, it is treated as a normal G code without making a call.

The following table shows special call disabled combinations.

	2nd Mis-cellaneous Function Code Sub-program	S Code Sub-program	T Code Sub-program	M Code Sub-program	M Code Macro	Arbitrary G Code Macro
Arbitrary G Code Macro	○	○	○	○	○	×
Arbitrary M Code Macro	○	○	○	×	×	○
Arbitrary M Code program	○	○	○	×	×	○
T Code Subprogram	○	○	×	○	○	○
S Code Subprogram	○	×	○	○	○	○
In 2nd Mis-cellaneoous Function subprogram	×	○	○	○	○	○

Since this prohibitive condition is set with the parameter, it can be cancelled with the parameter.

Parameter



In the parameter above, when each bit is ;

0 : Enables a call

1 : Disables a call

22-2-3 Argument Designation

Argument designation means to assign a real number to the local variable used in the custom macro.

There are two types of argument designation ; Type Ⅰ and Type Ⅱ. Both can be used freely.

(1) Argument designation Ⅰ

Address	Corresponding Variable
A	#1
B	#2
C	#3
I	#4
J	#5
K	#6
D	#7
E	#8
F	#9
H	#11
M	#13
Q	#17
R	#18
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

(2) Argument designationⅡ

Address	Corresponding Variable
A	#1
B	#2
C	#3
I_1	#4
J_1	#5
K_1	#6
I_2	#7
J_2	#8
K_2	#9
I_3	#10
J_3	#11
K_3	#12
I_4	#13
J_4	#14
K_4	#15
I_5	#16
J_5	#17
K_5	#18
I_6	#19
J_6	#20
K_6	#21
I_7	#22
J_7	#23
K_7	#24
I_8	#25
J_8	#26
K_8	#27
I_9	#28
J_9	#29
K_9	#30
I_{10}	#31
J_{10}	#32
K_{10}	#33

(3) Argument's decimal point position

In argument designation, signs and a decimal point can be used for the addresses where they are not allowed originally.

< Example >

G65 P1 H-2.0 M-9.6 ;

The following table shows the decimal point positions when the decimal point is omitted.

Address	Description
A,B,C,U,V,W,X,Y,Z (in case of axial address)	Conforms to the least input increment for each axis. Refer to Subtable a.
A,B,C,U,V,W (in case of 2nd miscellaneous function address)	○
A,B,C,U,V,W,X,Y,S (other than the cases above)	Conforms to the least input increment for the reference axis. Refer to Subtable a.
Q,R,	Conforms to the least input increment for each axis. Refer to Subtable a.
I,J,K	Replace I with X, J with Y, and K with Z, and the same as the decimal point position of that address.
E,F	Conforms to the least input increment for each axis. Refer to Subtable b.
D,G,H,L,M,P,S,T	○

Subtable a.

	Metric (G21)	Inch (G20)	Rotary (Axis)
IS-A	2	3	2
IS-B	3(2)	4(3)	3(2)
IS-C	4(3)	5(4)	4(3)

· The values in parentheses are applied when IPR (bit 2 of the parameter 1007) is ‘‘1’’.

· All are ‘‘0’’ in case of pocket calculator type decimal point input.

Subtable b.

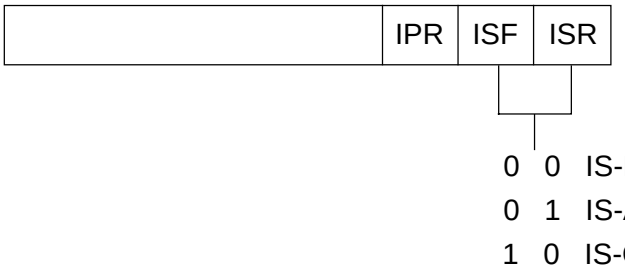
	Metric (G21)	Inch (G20)
inverse time (G93)	3	3
Feed per minute (G94)	MM1 = 0 0 MR1 = 1 1*	IM2 = 0 1 IM2 = 1 2
Feed per revolution (G95)	MR3 = 0 2 MR3 = 1 3	IR4 = 0 3 IR4 = 1 4
Thread cutting (G33)	MS6 = 0 5 MS6 = 1 6	IS7 = 0 6 IS7 = 1 7

· 0 when the parameter F61 is ‘‘1’’

· All are ‘‘0’’ in case of pocket calculator type decimal point input.

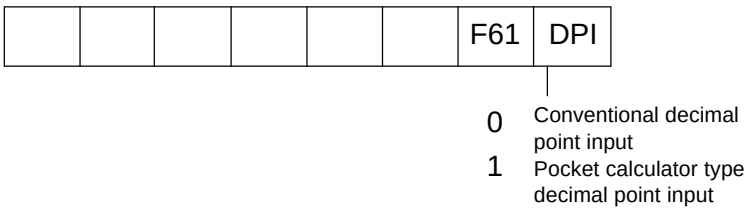
Parameter

PRA1003



PRA1010

PRA3400



PRA3401

		MS6	IS7	MM1	IM2	MR3	IR4

(4) Cautions

a. Argument Assignment I and II can be used as combined. When a variable has been argument assigned by more than twice, the one assigned last is made valid.
b. For both Argument Assignment I and II, assign Addresses I, J, and K only in alphabetical order.
c. For the custom macro call command, assign a call code prior to assignment of all arguments.

22-3 Variables

With a variable specified to a certain address within the macro program instead of directly giving a numerical value to it, when this variable is called during execution, a variable value can be taken out to be as an address value. There are local variables, common variables and system variables, and their uses and natures vary respectively.

(1) Local variables (#1 through #33)
Variables used locally for each macro. With a variable area independently secured for each macro call, these variables can store values through argument designation or save results of computation executed within the macro. The local variables are reset at every macro call and registered by argument designation. Unspecified variables become ‘‘null’’ They are reset to ‘‘null’’ at power-on time or reset time.

(2) Common variables (#100 through #199, #500 through #999)
Common variables can be commonly used throughout all macros and those of any multiplicity.

① The common variables #100 through #199 are reset to ‘‘null’’ when the power is turned on.

② The common variables #500 through #599 are not reset to ‘‘null’’ when the power is turned on.

(3) System variables
These are variables whose uses are fixed in the system.

① Interface input signal (#1000 through #1031, #1032, #1033, through #1035)
By reading #1000 ~ #1031, the status of 32-point input signals exclusive for the macro program can be known.

System Variable	Points	Interface Input Signal		System Variablw	Points	Interface Input Signal	
#1000	1	2^0	UI000	#1016	1	2^{16}	UI016
#1001	1	2^1	001	#1017	1	2^{17}	017
#1002	1	2^2	002	#1018	1	2^{18}	018
#1003	1	2^3	003	#1019	1	2^{19}	019
#1004	1	2^4	004	#1020	1	2^{20}	020
#1005	1	2^5	005	#1021	1	2^{21}	021
#1006	1	2^6	006	#1022	1	2^{22}	022
#1007	1	2^7	007	#1023	1	2^{23}	023
#1008	1	2^8	008	#1024	1	2^{24}	024
#1009	1	2^9	009	#1025	1	2^{25}	025
#1010	1	2^{10}	010	#1026	1	2^{26}	026
#1011	1	2^{11}	011	#1027	1	2^{27}	027
#1012	1	2^{12}	012	#1028	1	2^{28}	028
#1013	1	2^{13}	013	#1029	1	2^{29}	029
#1014	1	2^{14}	014	#1030	1	2^{30}	030
#1015	1	2^{15}	015	#1031	1	2^{31}	031

Variable Value	Input Sugnal
1.0	Contact closed
0.0	Contact open

The 32-point input signals can be read at one time by reading #1032 ~ #1035.

System Variable	Points	Interface Input Signal
#1032	32	UI000~UI031
#1033	32	UI100~UI131
#1034	32	UI200~UI231
#1035	32	UI300~UI331

$$\#1032 \sum_{i=0}^{20}\{1000 + i\} * 2i - \#1031 * 2^{31}$$

$$\#[1032 + n] = \sum_{i=0}^{30}\{2i * V\ i\} - 2^{31} * V_{31}$$

where ; Vi = 0 when UIni is open
V1 = 1 when UIni is closed
n = 0 ~ 3

② Interface output signals (#1100 through #1131, #1132, #1133 through #1135)
The ON or OFF signal is sent to the 32-point output signals, which are exclusively used for the macro program, by substituting the values for #1100 through #1131.

System Variable	Points	Interface Input Signal		System Variablw	Points	Interface Input Signal	
#1100	1	2^0	UO000	#1116	1	2^{16}	UO016
#1101	1	2^1	001	#1117	1	2^{17}	017
#1102	1	2^2	002	#1118	1	2^{18}	018
#1203	1	2^3	003	#1119	1	2^{19}	019
#1104	1	2^4	004	#1120	1	2^{20}	020
#1105	1	2^5	005	#1121	1	2^{21}	021
#1106	1	2^6	006	#1122	1	2^{22}	022
#1107	1	2^7	007	#1123	1	2^{23}	023
#1108	1	2^8	008	#1124	1	2^{24}	024
#1109	1	2^9	009	#1125	1	2^{25}	025
#1110	1	2^{10}	010	#1126	1	2^{26}	026
#1111	1	2^{11}	011	#1127	1	2^{27}	027
#1112	1	2^{12}	012	#1128	1	2^{28}	028
#1113	1	2^{13}	013	#1129	1	2^{29}	029
#1114	1	2^{14}	014	#1130	1	2^{30}	030
#1115	1	2^{15}	015	#1131	1	2^{31}	031

Variable Value	Input Sugnal
1.0	Contact closed
0.0	Contact open

The 32-point input signals can be sent all at one time by substituting the values for #1132 through #1135.

System Variable	Points	Interface Input Signal
#1132	32	UI000~UI031
#1133	32	UI100~UI131
#1134	32	UI200~UI231
#1135	32	UI300~UI33

$$\#1132 \sum_{i=0}^{20}\{1100 + i\} * 2i - \#1131 * 2^{31}$$

$$\#[1132 + n] = \sum_{i=0}^{30}\{2i * V\,i\} - 2^{31} * V_{31}$$

where ; Vi = 0 when UIni is open
Vi = 1 when UIni is closed
n = 0 ~ 3

Note : ‘‘Null’’ is regarded as 0.
Those other than ‘‘Null’’ and 0 are regarded as 1.

③ Tool offset amount (#2001 through #2800, #10001 through #13999)
The offset amount can be known by reading the values of #2000 ~ #2800 and #10000 ~ #13999, and it can be altered by substituting the values for them.

System Variable Numbers for Tool Offset Amount
(For Memory C, 200 Offsets)

Offset NO.	HCode		D code	
	Profile	Wear	Profile	Wear
1	#2001	#2201	#2401	#2601
2	#2002	#2202	#2402	#2602
3	#2003	#2203	#2403	#2603
:	:	:	:	:
:	:	:	:	:
199	#2199	#2399	#2599	#2799
200	#2200	#2400	#2600	#2800

Note :#2001 ~ #2200 in case of the memory A, without distinguishing D/H and profile/wear.

When the number of tool offsets exceeds 200, the following is assumed.

Offset NO.	HCode		D code	
	Profile	Wear	Profile	Wear
1	#10001	#11001	#12001	#13001
:	:	:	:	:
:	:	:	:	:
999	#10999	#11999	#12999	#13999

Note: #10000 ~ #10999 for the memory A.
The offset amount for the offset number of up to 200 can be referred to with #2001 ~ #2800.

④ Alarm (#3000)

When a condition occurs in the program, which you want to be an alarm, the system can be placed in the alarm state.

#3000 = n (<alarm message>) ; (n ≦ 4095)

This command specifies the alarm message (up to 32 characters) enclosed by the alarm number n and ‘‘(‘‘,‘‘)’’.

Note : For the alarm number, use one which the system has not specified. If the alarm number specified by the system is used, the alarm message of #3000 appears and that alarm remains effective until next reset.

⑤ Clock (#3001, #3002)

The time of the clock can be known by reading the values of #3001 and #3002, and the time can be preset by substituting the values for those variables.

Type	System Variable	Unit	At Power-on	Count Condition
Clock 1	#3001	msec.	0	Any time
Clock 2	#3002	Hour	Saved	When STL signal is turned on

Note : Clock accuracy is 10 msec.Overflows at 4294967295 msec.and returns to 0.

⑥ Single block stop, miscellaneous function complete signal wait control (#3003)

The controls shown in the table below can be provided by substituting the values in the table below for #3003.

#3003	Single Block Stop	Miscellaneous Function Complete Signal
0	Not suppressed	Waited
1	Suppressed	Waited
2	Not suppressed	Not waited
3	Not suppressed	Not waited

Note :No distribution complete signal is output when not waiting for the miscellaneous function complete signal. #3003 is cleared to 0 by reset.

⑦ Feed hold, feed rate override, exact stop check enabled/disabled (#3004)
The following controls can be provided by substituting the values shown in the table below for #3004.

#3004	Feed Hold	Feed Rate Override	Exact Stop Check
0	Enabled	Enabled	Enabled
1	Disabled	Enabled	Enabled
2	Enabled	Disabled	Enabled
3	Disabled	Disabled	Enabled
4	Enabled	Enabled	Disabled
5	Enabled	Enabled	Disabled
6	Enabled	Disabled	Disabled
7	Disabled	Disabled	Disabled

Note :When the FEED HOLD button is pressed while the feed hold disabled block is being executed ;

(a) Keep pressing the FEED HOLD button stops the machine by single block. However, it does not stop if single block stop is also suppressed.

(b) When the FEED HOLD button is pressed and released, the FEED HOLD lamp blinks, but the machine does not stop. It stops at the first block where the feed hold function is enabled.
#3004 is cleared to 0 by reset.

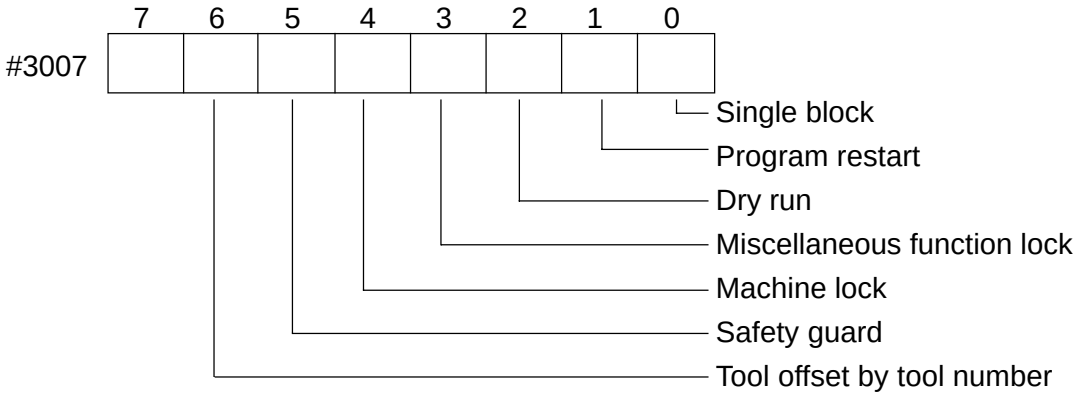
⑧ Mirror image condition (#3007)
By reading #3007, the then mirror image status can be known per axis.
Each axis corresponds per bit at the time of binary display.

	7	6	5	4	3	2	1	0
#3007	8	7	6	5	4	3	2	1 st axis

Mirror image disabled (0) / enabled (1) is indicated per bit.

⑨ Operational condition information (#3010)

By reading #3010, the then operational condition can be known. Each condition corresponds per bit at the time of binary display.



Disabled (0) / enabled (1) is indicated per bit.

⑩ Time (#3011, #3012)

By reading #3011 and #3012, the year, month, day, hours, minutes and seconds can be known.

Type	System Variable
Year, month, day	#3011
hours, minutes, seconds	#3012

(Example) 11 : 07 : 05, Jan. 23, 1989

#3011 = 19890123

#3012 = 110705

⑪ Run hour timer (#3021, #3022)

Three types of integrating time can be read, which indicate the running condition of the NC unit.

Machine type	System Variable	Unit	At Power-on	Count Condition
Automatic operation time	#3021	Hour	Saved	Automatic operation ON
Cutting time	#3032	Hour	Saved	Cuttin geed ON

Note : Clock accuracy is in milliseconds.

⑫ Work counter

Reading/writing of the work counter of the running state screen are enabled.

System Variable	Function
#3901	No. of works machined
#3902	Set no. works
#3903	Total no. of works for machining

Note : Enter numerical values of 0 to 9999.

⑬ Modal information (#4001 ~ #4120, #4201 ~ #4330)

By reading the values of #4001 ~ #4120, the modal commands specified so far (up to the preceding block) can be known.

By reading the values of #4201 ~ #4320, the modal commands in the block being executed can be known.

The unit at the time of giving the command is assumed.

System Variable	Modal Information in Preceding Block	System Variable	Modal Information in Block Being Executed
#4001	G Code (Group 01)	#4201	G Code (Group 01)
.	.	.	.
.	.	.	.
.	.	.	.
#4023	G Code (Group 23)	#4223	G Code (Group 23)
#4102	B Code	#4302	B Code
#4107	D Code	#4307	D Code
#4108	E Code	#4308	E Code
#4109	F Code	#4309	F Code
#4111	H Code	#4311	H Code
#4113	M Code	#4313	M Code
#4114	Sequence number	#4314	Sequence number
#4115	Program number	#4315	Program number
#4119	S Code	#4319	S Code
#4120	T Code	#4320	T Code
		#4400	Spindle tool number

⑭ Position information in the canned cycle (#4500 ~ #4504)

Next position information and machining data can be known by reading #4500 ~ #4504 in the canned cycle.

System Variable	Position Information/ Machining Data
#4500	R point
#4501	Initial point
#4502	Z point
#4503	Q Data
#4504	P Data

⑮ Position information (#5001 ~ #5108)

Various position information can be known by reading the values of #5001 through #5108. The unit at the time of giving the command is assumed.

System Variable	Position Information	Read in Move
#5001	1st-axis block final position (ABSIO)	Enabled
#5002	2nd-axis block final position (ABSIO)	
#5003	3rd-axis block final position (ABSIO)	
.	.	
.	.	
#5008	8th-axis block final position (ABSIO)	
#5021	1st-axis current position (ABSMT)	Disabled
#5022	2nd-axis current position (ABSMT)	
#5023	3rd-axis current position (ABSMT)	
.	.	
.	.	
#5028	8th-axis current position (ABSMT)	
#5041	1st-axis current position (ABSOT)	Disabled
#5042	2nd-axis current position (ABSOT)	
#5043	3rd-axis current position (ABSOT)	
.	.	
.	.	
#5048	8th-axis skip signal position (ABSKP)	
#5061	1st-axis skip signal position (ABSKP)	Enabled
#5062	2nd-axis skip signal position (ABSKP)	
#5063	3rd-axis skip signal position (ABSKP)	
.	.	
.	.	
#5068	8th-axis skip signal position (ABSKP)	
#5081	1st-axis tool length offset amount	Disabled
#5082	2nd-axis tool length offset amount	
#5083	3rd-axis tool length offset amount	
.	.	
.	.	
#5088	8th-axis tool length offset amount	
#5101	1st-axis servo position deviation amount	Disabled
#5102	2nd-axis servo position deviation amount	
#5103	3rd-axis servo position deviation amount	
.	.	
.	.	
#5108	8th-axis servo position deviation amount	

Symbol	ABSIO	ABSMT	ABSOT	ABSKP
Meaning	End point position of the block just before	Specified current position	Specified current position	Skip signal ON position in the G31 block.
Coordinate system	Work coordinate system	Machine system	Work coordinate system	Work coordinate system
Tool offset, tool length/diameter compensation	Not included, Tool nose position	Included, Tool reference point position	Included, Tool reference point position	Included, Tool reference point position

⑯ Work offset amount (#5200 ~ #5328)

The offset amount can be known by reading the values of #5200 ~ #5328, and it can be altered by substituting the values for them.

Variable No.	Controlled Axis	Coordinate System
#5200 #5201 #5202 . . #5208	Coordinate rotation angle 1st-axis work offset 2nd-axis work offset . . 8th-axis work offset	External offset
#5220 #5221 #5222 . . #5228	Coordinate rotation angle 1st-axis work offset 2nd-axis work offset . . 8th-axis work offset	G54
#5240 #5241 #5242 . . #5248	Coordinate rotation angle 1st-axis work offset 2nd-axis work offset . . 8th-axis work offset	G55
#5260 #5261 #5261 . . #5268	Coordinate rotation angle 1st-axis work offset 2nd-axis work offset . . 8th-axis work offset	G56
#5280 #5281 #5282 . . #5288	Coordinate rotation angle 1st-axis work offset 2nd-axis work offset . . 8th-axis work offset	G57
#5300 #5301 #5302 . . #5308	Coordinate rotation angle 1st-axis work offset 2nd-axis work offset . . 8th-axis work offset	G58
#5320 #5321 #5322 . . #5328	Coordinate rotation angle 2nd-axis work offset 2nd-axis work offset deviation amount . . 8th-axis work offset	G59

⑰ Common work zero point offset amount (#7220 ~ #7328)

The common work zero point offset amount can be known by reading the values of #7220 ~ #7328, and it can be altered by substituting the values for them.

Variable No.	Controlled Axis	Coordinate System
#7220 #7221 #7222 . . #7228	Coordinate rotation angle 1st-axis offset 2nd-axis offset . . 8th-axis offset	P0
#7240 #7241 #7242 . . #7248	Coordinate rotation angle 1st-axis offset 2nd-axis offset . . 8th-axis offset	P1
#7260 #7261 #7262 . . #7268	Coordinate rotation angle 1st-axis offset 2nd-axis offset . . 8th-axis offset	P2
#7280 #7281 #7282 . . #7288	Coordinate rotation angle 1st-axis offset 2nd-axis offset . . 8th-axis offset	P3
#7300 #7301 #7302 . . #7308	Coordinate rotation angle 1st-axis offset 2nd-axis offset . . 8th-axis offset	P4
#7320 #7321 #7322 . . #7328	Coordinate rotation angle 1st-axis offset 2nd-axis offset . . 8th-axis offset	P5

⑰ Additional work offset amount

G540 - G599 work offset amount can be known by reading the values of #7400 - 7998, and it can be altered by substituting the values from them.

Variable No.	Controlled Axis	Coordinate System
#7400 #7401 #7402 . . #7418	Coordinate rotation angle 1st-axis offset 2nd-axis offset . . 8th-axis offset	G540
#7410 #7411 #7412 . . #7418	Coordinate rotation angle 1st-axis offset 2nd-axis offset . . 8th-axis offset	G541
#7420 #7421 #7422 . . #7428	Coordinate rotation angle 1st-axis offset 2nd-axis offset . . 8th-axis offset	G542
. .	. .	. .
#7500 #7501 #7502 . . #7508	Coordinate rotation angle 1st-axis offset 2nd-axis offset . . 8th-axis offset	G550
. .	. .	. .
#7990 #7991 #7992 . . #7998	Coordinate rotation angle 1st-axis offset 2nd-axis offset . . 8th-axis offset	G559

System variable number = 7000 + (Coordinate system number - 500) $\times$ 10 + Axis number.

⑲ Life Management Information (#21001 - #24999)

It is possible to know the tool life management information by reading #21001 ~ #24999. It is also possible to rewrite the information by substituting values.

	Tool No.	Variable	Remarks		
Setting unit	1 2 : 998 999	#21001 #21002 : #21988 #21999	0:Minute 1:Time 2:Length 3:Hole		
Setting data	1 2 : 998 999	#22001 #22002 : #22988 #22999			
Setting unit	1 2 : 998 999	#23001 #23002 : #23988 #23999			
Setting unit	1 2 : 998 999	#24001 #24002 : #24988 #24999	0:Unused 1:Used 2:Monitor used 3:N/A 4:Life	5:Wear life 6:Skip 7:N/A 8:Tip measurement NG 9:Word measurement NG	10:Overload error 11:No-load error 12:Extarnal error

⑳ Axis names (#3041 to #3048)

Each axis name can be learned by reading #3041 to #3048.

System Variable	Description	return Value and Meaning	
#3041	1st axis name	88:X	67:C
#3042	2st axis name	89:Y	85:U
#3043	3st axis name	90:Z	86:V
:	:	65:A	87:W
#3048	8st axis name	66:B	

㉑ Axis numbers (#3061 to #3069)

Each axis number can be learned by reading #3061 to #3069.

System Variable	Description
#3061	X-axis number
#3062	Y-axis number
#3062	Z-axis number
#3064	A-axis number
#3065	B-axis number
#3066	C-axis number
#3067	U-axis number
#3068	V-axis number
#3069	W-axis number

(4) Character string variables

An alphanumeric string can be used as a variable by prefixing it with $. Substituting a value for the character string variable is called creating the variable. It can be later used in the representation of the <expression>.

```
$STR1 = 123.0 ;
$STR2 = $STR1*2 ;
```

When replacing a numeral following an address with a variable, it is necessary to enclose the variable by brackets.

```
X [$STR1]  Y [$STR2] ;
```

Caution : The maximum allowable character string length is 22 characters, including $. These variables are commonly available through all the macros. They are all cleared by reset.

22-4 Representation of Variables

The variable is represented by the variable number following ‘‘#’’ as follows ;

#i (i = 1, 2, 3, ...)

#1, #2, #3

Or, it is represented by using the < expression > as follows ;

#[< expression >]

#[#100], #[#500 + 1], #[#20/2]

In the following description, #1 can be replaced by # [< expression >].

22-5 Citation of Variables

A numerical value following an address can be replaced by a variable. By giving the command ‘‘<address >#i’’or ‘‘<address > - #i’’ a variable value or its complement is taken as a command value for that address. When #11 is 20.0, X#11 is the same as a command of X20.0.

< Notes >

a. Addresses /, O and N cannot cite variables.

b. The variable number cannot be replaced by the variable.

##20 ..... Incorrect

#[#20] ..... Correct

c. When the variable is used as address data, the portion below the significant digit of that address is rounded.

d. You cannot give a command exceeding the maximum command value of each address.

e. A numerical value following the address can be replaced by an expression.
By specifying ‘‘<address >[< expression >]’’or ‘‘ ‘‘<address > - [< expression >], the value of the < expression > is assumed as it is, or its complement as a command value for its address.

#[#1 + #2 * #3]

Y - [#1 * #2 * COS[#3]]

f. The constant without a decimal point used in square brackets is regarded as having a decimal point at its end.

X1000 is 1.000 mm, but X[1000] is 1,000 mm.

22-6 Undefined Variables

The value of an undefined variable is null. #0 is always used as a null variable.
the undefined variable occurs in the following cases ;

1) Local variable for which no argument has been designated in the macro call command.
2) Common variables #100 through #1XX when the power is turned on.
3) Variables for which “null’’ has been written by manual operation.

(1) Citation of variables

If the undefined variable is cited, the address itself is also ignored.
When #1 is “null’’ X100 Y#1 ; is the same as X100 ; .
The complement for “null’’is “null’’ itself.
When #1 is “null’’ X100 Y-#1 ; is the same as X100 ;

(2) Computation

In a combination by an operator or in a function, the undefined variable is treated the same as the constant 0.
When #1 is “null’’ ;

#2 = #1 ; → #2 = “null’’
#2 = #1 + 1 ; → #2 = 1
#2 = #1*5 ; → #2 = 0
#2 = # [#1] → #2 = “null’’
#2 = # [#1 + #1] → #2 = “null’’

(3) Relative computation

“null’’ and 0 differ from each other only in case of EQ and NE.
When #1 is “null’’ ;

#1EQ#0 = True
#1EQ0 = False
#1NE#0 = False
#1NE0 = True
#1GE#0 = True
#1GE0 = True

22-7 Expression and Computation

The expression refers to a general numerical expression where constants and variables are combined with operators, or simply numerical values or variables.

In the following description, the constants may be used instead of #i and #j.

(1) Addition type computation

#i + #j ..... Addition

#i - #j ..... Subtraction

#iOR#j ..... OR (for each bit of 32 bits)

#iXOR#j ..... Exclusive OR (for each bit of 32 bits)

(2) Multiplication type computation

#i * #j ..... Multiplication

#i/#j ..... Division

#iAND#j ..... AND (for each bit of 32 bits)

#iMOD#j ..... Remainder

(3) Relative computation

When a judgment result is true, an expression value is 1, and when false, the value is 0.

#iEQ#j ..... Judgment of #i = #j

#i = #j

#iNE#j ..... Judgment of #i ≠ j

#i<>#j>

#iGT#j ..... Judgment of #i > #j

#i>#j

#iGE#j ..... Judgment of #i ≧ j

#i>=#j

#iLT#j ..... Judgment of #i < #j

#i<#j

#iLE#j ..... Judgment of #i ≦ j

#i<=#j

(4) functions

SIN[#i]	Sine	(unit: degree)
COS[#i]	Cosine	(unit: degree)
TAN[#i]	Tangent	(unit: degree)
ASIN[#i]	Inverse sine	(unit: degree)
ACOS[#i]	Inverse cosine	(unit: degree)
ATAN[#i]/[#j]	Inverse tangent	(unit: degree)
ABS[#i]	Absolute value	
SQRT[#i]	Square root	
EXP[#i]	Exponent with “e’’ as a base.	
LN[#i]	Natural logarithm	
ROUND[#i]	Rounding to an integer	
FIX[#i]	Omission of figures below a decimal point	
FUP[#i]	Raising of figures below a decimal point to a unit	
BIN[#i]	Conversion from BCD	
BCD[#i]	Conversion to BCD	
ADP[#i]	Addition of a decimal point	
SPA[#i]	Tool number to be called next	
SPB[#i]	Tool number actually used	

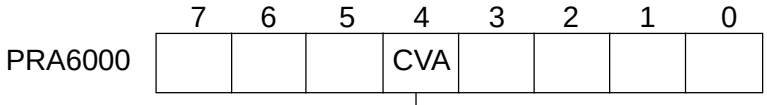
(a) Concerning the ADP function

By specifying ADP[#n](n = 1 to 33), a decimal point can be added to the argument passed without the decimal point on the macro program side. On the custom macro side called by G65PnnnnX10; , the value of ADP[#24] becomes equivalent to the argument with the decimal point added to the end, that is 10. This is used when you do not want to be conscious of the setting unit on the macro program side.

If a numerical value is substituted for the local variable on the custom macro side, the ADP function does not act on the ADP function thereafter.

```
G65   P100   X10   Y20 ;  (for the metric system)
└→ O100 ;
     #1 = #24;              #1 = 0.01
     #1 = ADP[#24];         #1 = 10
     #1 = #25;              #1 = 0.02
     #1 = ADP[#25];         #1 = 20
     #24 = ADP[#24];        #24 = 10
     #1 = ADP[#24];         #1 = 10
     #25 = 0.05;            #25 = 0.05
     #1 = ADP[#25];         #1 = 0.05
```

You can disable the ADP function with the parameter.



0 : Enables the ADP function

1 : Disables the ADP function

With G65 P_ X10 ; and in the macro ;

	CVA = 0	CVA = 1
#24	0.01	0.01
ADP[#24]	10.0	0.01

Note) If the ADP function is used for “null’’ , “null’’ is returned.

(b) Functions dealing with tool life management

If the tool runs out of life under tool life management, the programmed tool number will not match the actually used tool number. In that case, the actually called or used tool number can be learned by using these functions.

① SPX [x]

x: Call number

When the tool is specified with the call number x, the actually called tool number

is returned.

(Note) If x is the tool number, that tool number will be returned.

② SPB [x]

x: Call number

The tool actually called by the T-command is returned.

(Note) If x is the tool number, that tool number will be returned.

<Example of Use>

<Program>

```
 01000 ;
 #1 = SPA[1000] ;
 T1000M06 ;
#2 = SPB[1000] ;
M99 ;
```

<Setting of Life management>

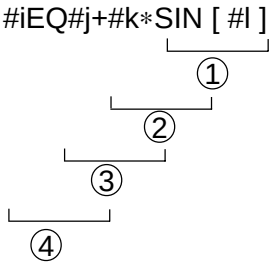
Call No.			Stting Count: 10
Oder	Tool	Use	Status
1	1	9	Used
2	2	0	
3			

<Results>

	1st Time	2nd Time
#1	1,000	2,000
#2	1,000	2,000

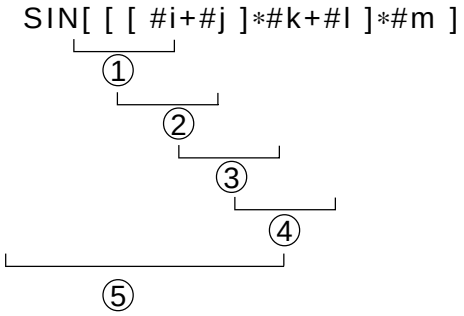
(5) Combination of computations

Computations and functions can be combined. Computations are given priority in the order of function multiplication type, addition type, and relative computations.



(6) Alteration of computation order by square brackets ([ ])

Using square brackets, you can enclose the portion of computation order to which you want to give priority. The square brackets can be quintuply used, including those for functions.



22-8 Substitution Command

By specifying #i = < expression >;, the value of the < expression > is substituted for the variable " i ". The < expression > can be used instead of " i ".

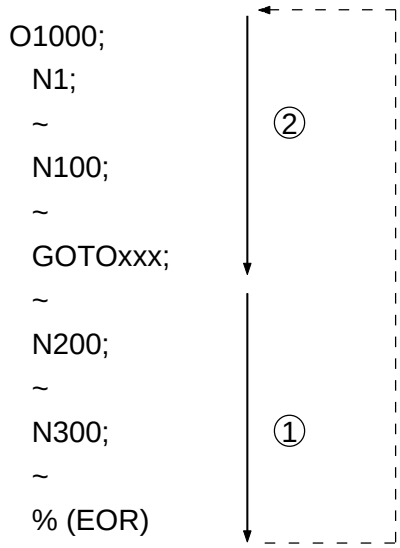
```
#1 =6 ;
#2 =#1+ABS[#2]/3 ;
#[#1+500] =1 ;
```

22-9 Branch Command

Control jumps to the block having the sequence number “n’’ within the same program by specifying “GOTO n ;’’

The < expression > can be used instead of “n’’ When this is done, the value of the < expression > is obtained and control jumps to the block having that value as the sequence number.

The sequence number is searched for in the next turn. When there are two or more same sequence numbers, control jumps to the block having the first found sequence number.



< Notes >

a. On the program, backward branching takes a longer execution time than forward branching.
b. Even if the sequence number, to which control jumps, is not found at the beginning of the block, execution starts from the beginning of the block.

22-10 Repeat Command

DO m ; (M = 1, 2 or 3)

~

END m ;

By specifying as above, the blocks between DOm and ENDm are repeatedly executed. The following special uses are also available.

(1) Conditional repeat

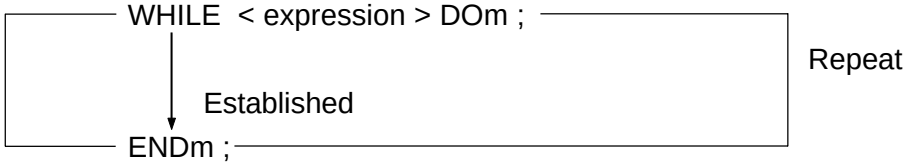
WHILE < expression > DOm ; (m = 1, 2 or 3)

~

ENDm ;

By specifying as above, the blocks between DOm and ENDm are repeatedly executed while the value of the < expression > is true (not 0).

When the value of the < expression > is false (0), control jumps to the block next to ENDm.



Unestablished

< Notes for the repeat command >

(a) Specify DOm before ENDm.

(b) For m of DOm and ENDm, specify the same number.
The value of m is called an identity number ; only 1, 2 or 3 can be used.

(c) The same identity number can be used as many times as you want, except when a repeat range overlaps.

(d) You can jump to outside the DO loop with GOTOn, but not vice versa.

(e) Multiplicity of the DO loop is allowed up to 3 levels.

< Error-incurring programs >

(a) DO1 ;
DO1 ; ← There is no corresponding DO
END1 ;

(b) DO1 ;
END1 ; ← There is no corresponding DO.

(c) DO1 ;
DO2 ;
END1 ; The loop cannot intersect.
END2 ;

(d) DO1 ;
E100 ;
END1 ;
GOTO100 ;

22-11 Naming Command

Part of the common variables can be named within 12 characters, respectively.
With the command “SETVNn (name 1, name 2, name 3, ...) ;, the variable ‘‘n’’ can be given a name “name1’’ variable “n + 1’’ a name “name 2’’ variable “n + 2’’ a name “name 3’’ and so on.
The characters available for names are numbers, alphabets, decimal point, +, -, ∗, /, $, ?, #, [,], !, {, }, & and @.
The common variables to which names can be given are those in the range selected from #500 with an option. Those names are not deleted even if the power is turned off.

22-12 IF Command

(1) 1-line format

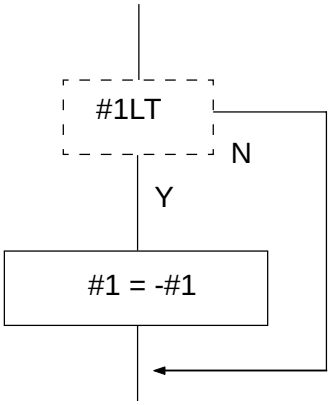
IF < expression > THEN macro command

This command allows a conditional branch. When the value of the < expression > is true (not 0), the macro command is executed, and when the value is false (0), nothing is done.

Here, the macro command refers to the substitution command, branch command, naming command and external output command.

< Example >

IF[#ILTO] THEN #1=-#1;



THEN can be omitted only when the macro command after THEN is GOTO.

< Example >

IF [#1EQ4] GOTO100 ; and IF [#1EQ4] THEN GOTO100; have the same meaning.

< Note >

Note that although the IF command and repeat command can be combined, they have a different action from the conditional repeat command.

IF < expression > THEN DOm ;

~

ENDm ;

If this is specified, the blocks between DOm and ENDm are repeated when the value of the < expression > is true (not 0).

When the value is false (0), the block next to DOm is executed, but DOm is not executed. Then, if ENDm is executed, an alarm results because there is no corresponding DOm (since not executed at IF).

(2) Block format

IF <expression 1> THEN ;

①

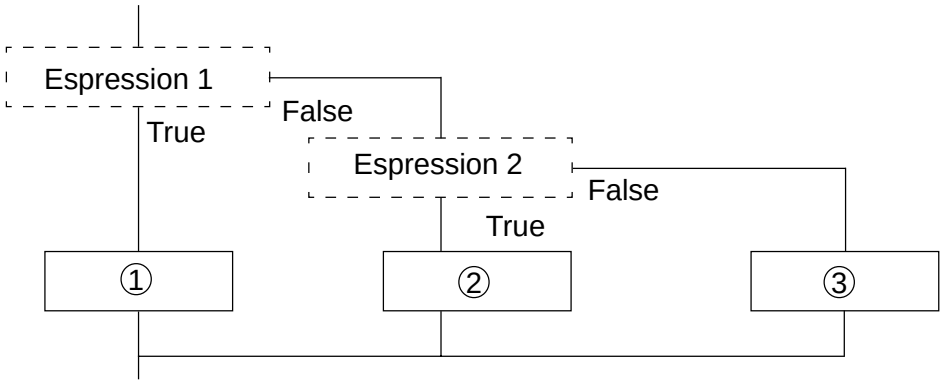
ELSE IF <expression 2> THEN ;

②

ELSE ;

③

ENDIF ;



Continues to ENDIF onward

If the value of <expression 1> is true, the program will branch to the block next to ENDIF after executing ①. If false (0), the program will branch to the next ELSE statement. If the IF command comes after that, the program will execute the next block according to that conditional expression or branch to ELSE. The block after ELSE up to ENDIF can be omitted.

22-13 External Output Commands

It is possible to output messages and the NC's internal data to the external unit via the RS232C data input interface. They are printed out if the external unit is a printer.

(a) PRINT
(b) BPRNT
(c) DPRNT
(d) POPEN
(e) PCLOS

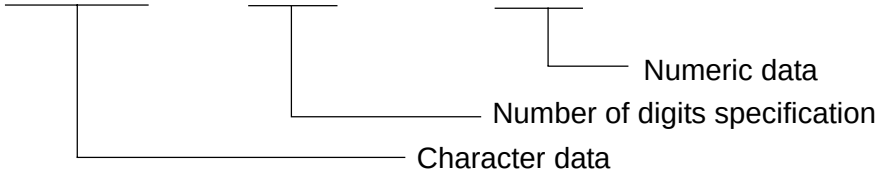
Specify in the following order.

①	Open command	POPEN Connection processing with I/O unit
②	Data output command	PRINT, BPRNT, DPRNT Outputs data
③	Close command	PCLOS Connection reset with I/O unit

(1) POPEN
Outputs the DC2 control code from the NC side.

(2) Data output command 1

PRINT (ABC ...) P <Expression> D <Expression>



PRINT outputs the character and the numeric per digit of variable value in the ISO, EIA, ASCII code.

a) As the character, the character in the range bracketed in '(' and ')' is just output. Alphanumeric characters, '+', '-', '∗', '/', '=', '&', '#', '[', ']', '$', '?', ',', '@', ' ','{','}' are available.
It is possible to omit character data.

b) Specify the number of digits above the decimal point and that below the decimal point of the numeric to be output subsequently to address P. The specification method: the unit digit of data P is the number of digits below the decimal point and its 10th digit is that above the decimal point.
Data of the looth digit and able is ignored.
If P13: The number of digits below the decimal point is 3 lines.
The number of digits above the decimal point is 1 line.
The number of digits specification is enabled until a P command comes anew in the same block.

PRINT P12 D#1 D#2 P23 D#3 ;
P12 enabled P23 enabled

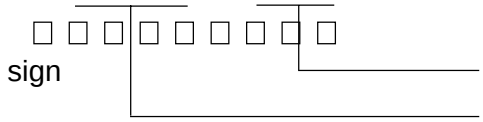
It is also possible to omit the number of digits specification. In that case, as floows.

	Number of digits above decimal point	Number of digits below decimal point
metric	4	3
inch	3	4
metric (SBMIC=1)	3	4
inch (SBMIC=1)	2	5

c) Specify the <expression> to be output subsequently to D. Output the value of the <expression> per digit.
Suppose the numeric data consists of a maximum of 8 lines. If the high-order digit is 0, nothing is output if parameter 6000's PRT = 1 and the space code is output if PRT = 0. The numeric below the decimal point is output whenever the number of digits below the decimal point is not 0. The decimal point is not output if the number of digits below the decimal point is 0. If the sign of the <expression> is positive, the space code is output if parameter PRT = 0 but nothing is output if PRT = 1. If the value of the <expression> is <null>, the space code is output for the number of significant digits if parameter PRT = 0 but nothing is output if PRT = 1.

PRINT P43 D#0 ;

If PRT = 0, output as follows.

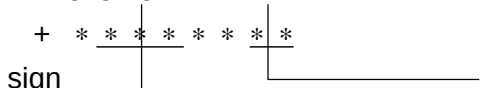


digits below the decimal point
digits above the decimal point

When the <expression> overflows, output '∗'for the number of significant digits.

PRINT P53 D#1 ;

When #1 + overflow



digits below the decimal point
digits above the decimal point

When #1 - overflow

\- ∗ ∗ ∗ ∗ ∗ ∗ ∗ ∗

If underflow, the <expression> regards as 0.

d) The EOB code is output after the command data output. The punching format at that time is determined according to parameter 0's ICR1, ICR2.

e) The EOB code is not output if there is ',' just after EOB.

PRINT (A B C D E F G), ;

Does not output EOB. This does not apply to that which is in '( )'however.

PRINT (A B C D E F G), ;

Outputs EOB.

this function is not available in BPRNT, DPRNT.

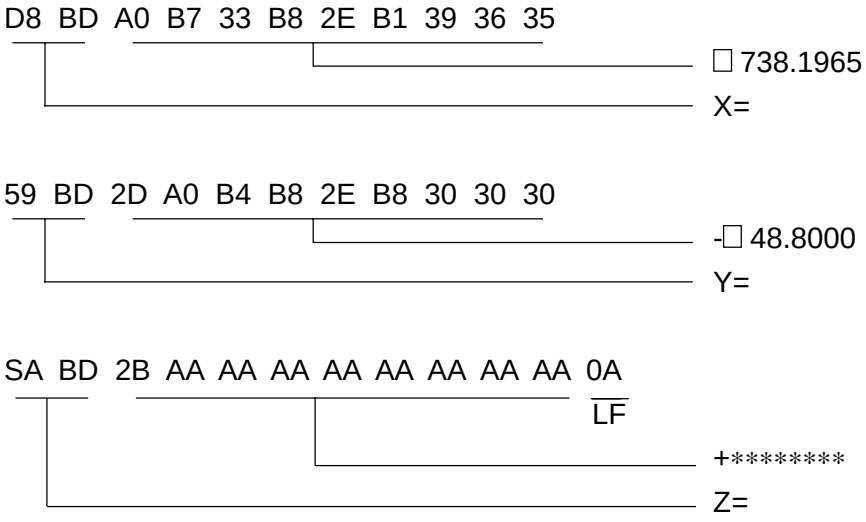
Example) PRINT P34(X=)D#1(Y=)D#2 P25(Z=) D#3 ;

#1= 738.196451

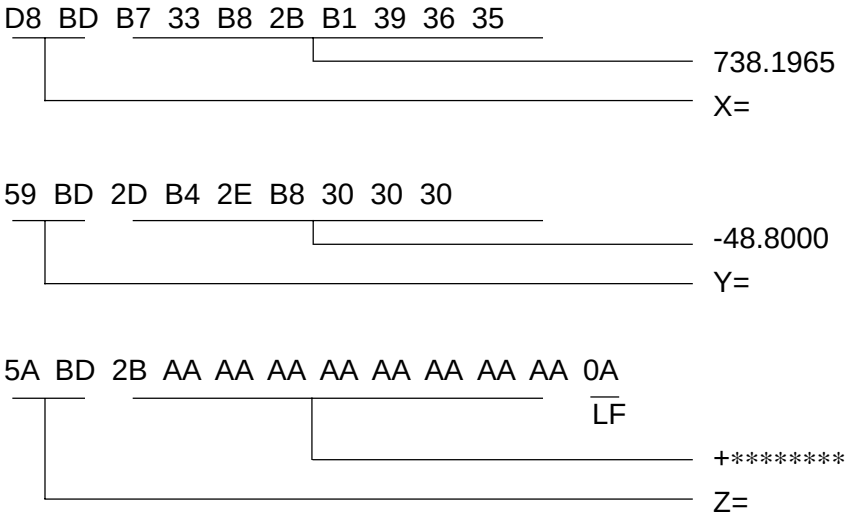
#2=-48.8

#3= 338.417

① Parameter PRT = 0



② Parameter PRT = 1



(3) Data output command 2.

BPRNT [A #1 [3] · · ·] ;

- [3] · · · The number of significant below the decimal point.
- #1 Variable number
- A Characters

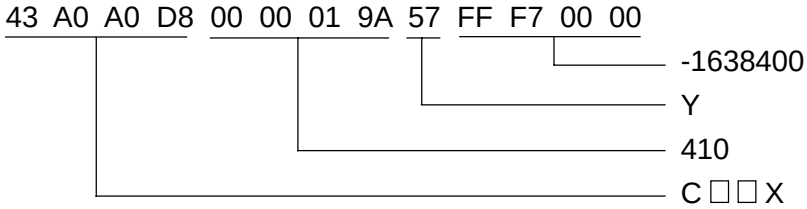
In the BPRNT, output binary of variable and characters.

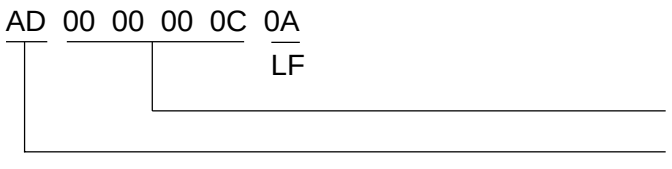
a) As the character, the specified character is just output. The following characters the specificable.
   · Alphabet (A - Z)
   · Numeric characters
   · Special characters (*, /, +, -)
   ‘‘*’’ is output in the space code, however.

b) Specify the number of significant lines below the decimal point subsequently to the variable command and by bracketing it. The value of a variable is handled as 2-word (32-bit) data in consideration of lines below the decimal point and is output from the high-order bio just as binary data.

c) The EOB code is output after the command data output. The punching format at that time is determined according to parameter 0’s ICR1, ICR2.

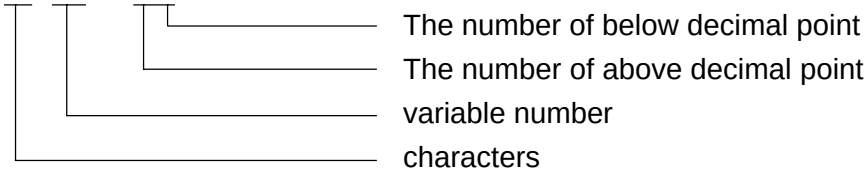
d) Same as the explanation in Item e) of the PRINT command.
   Example when BPRNT [C**X#100[3]Y#10[3]M#10[0]] ;
   #100 = 0. 40956
   #101 = -1638. 4
   #10 = 12. 34
   output as follows





(4) Data command 3

DPRNT [B #2 [43] · · ·]



In the DPRNT, output the variable number of each figure number, using ISO, EIA, ASCII code.

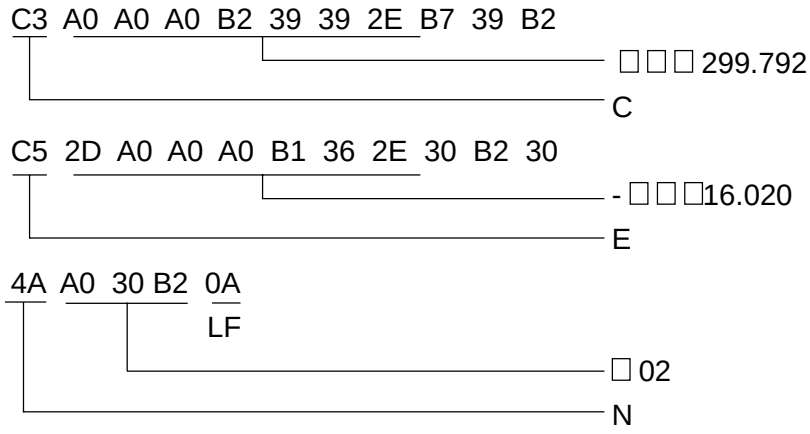
a) Same as the explanation in Item a), c), d) of the BPRNT.

b) When outputting the value of a variable, specify the variable number subsequently to # and, furthermore, specify the number of digits above the decimal point and that below the decimal point by bracketing them.
The value of a variable is output in the ISO or ASCII code per digit from the high-order digit for the specified number of digits. This is also the same for the decimal point.
Suppose the value of a variable consists of a numeric of a maximum of 8 digits. If the high-order digit is 0, nothing is output if parameter PRT = 1 and the space code is output if PRT = 0.
The numeric below the decimal point is output whenever the number of digits below the decimal point is not 0. The decimal point is not output if the number of digits below the decimal point is 0.
If the sign of the value of a variable is positive, the space code is output if parameter PRT = 0 but nothing is output if PRT = 1.
If the value of a variable is <null>, it is regarded as 0.

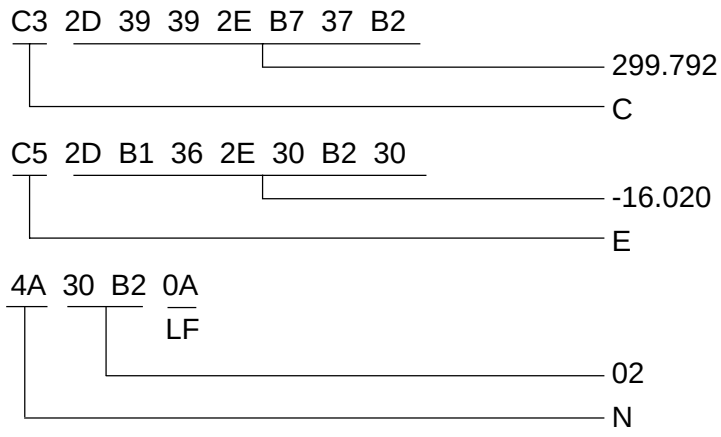
Example

When
DPRINT [C#1[53]E#2[53]N#3[20]]
#1=299.79246
#2=-16.02
#3=602.205

① If parameter PRT=0, output as follows.



② If parameter PRT=1, output as follows.



(5) Close Command PCLOS

Outputs the DC4 control code from the NC side.

(6) Restrictions

Data is output at the timing when the instruction above is executed in the program.
The number of characters outputtable by the instruction of 1 program is 256 bytes.

23. Interrupt Type Custom Macro

Other program can be called by inputting an interrupt signal (UNIT) from the machine side while running a program.

<Applications>

(1) Starting processing at tool error detection, using an external signal.
(2) Allowing other machining to interrupt a series of machining without cancelling on-going one.
(3) Reading the on-going machining information at constant time intervals.

23-1 Custom Macro Interrupt Operation

There are the following three types of custom macro interrupt operations.
(Parameters)

(1) When interrupting halfway the program block (Type-I)
If the interrupt signal(UNIT) is input, an on-going move (dwell included) will be interrupted to run an interrupt program.
   a. Reversion by M99
      i) When there is an NC statement in the interrupt program
      After finishing the interrupt program, control returns to the expected end point value of the interrupted block.
      ii) When there is no NC statement in the interrupt program
      After finishing the interrupt program, operation continues from halfway the interrupted block.
   b. Reversion by M99 Pxxxx
      After finishing the interrupt program, the original program is run from the block of the N-number specified by P.
(2) When interrupting at the end of the block (Type-II)
The interrupt program is run after one block created inside the NC unit is finished. After the interrupt program is finished, the original program is run from the next block or the block of the specified N-number.
(3) When interrupting after finishing the entire cycle (Type-III)
In case of a special canned cycle, etc., the interrupt program is run after entirely finishing one cycle. After the interrupt program is finished, the original program is run from the next block or the block of the specified N-number.

23-2 How to Specify

(1) Enabling conditions

a. Automatic operation, MDI, DNC mode

b. STL ON

c. Custom macro interrupt operation is already over

(2) Command format

M96 Enables the interrupt signal(UNIT) (Can be set with the parameter)

M97 Disables the interrupt signal(UNIT) (Can be set with the parameter)

```
M96  Pxxxx;
  :
  :
M97 ;
```

Specifies the interrupt program number.

(Modal information)

23-3 Interrupt Type Custom Macro Proper

There are two types of custom macro interrupts; subprogram type interrupt and macro type interrupt. (Parameters)

(1) Subprogram type interrupt

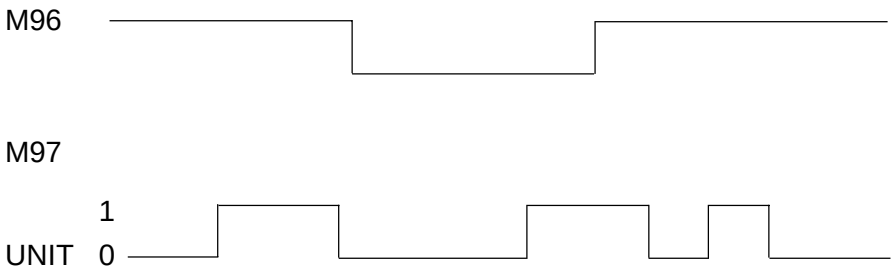
This is called as a subprogram, that is, the level of local variables does not change before and after an interrupt.

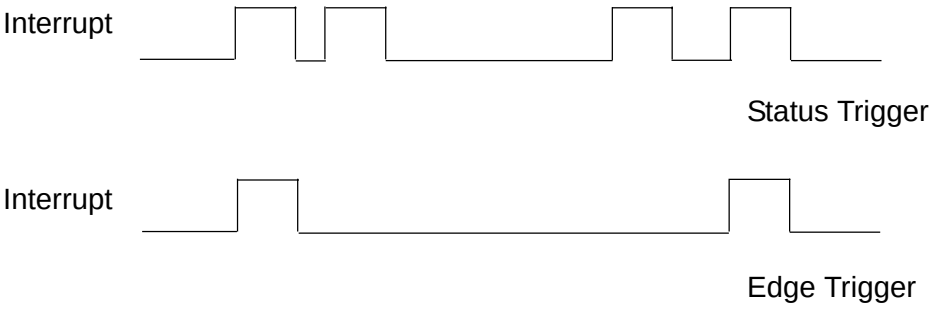
(2) Macro type interrupt

This is called as a macro, that is, the level of local variables changes; no argument can be specified. Either case is added to multiplicity of call.

23-4 Status Trigger Method and Edge Trigger Method (Parameters)

The interrupt signal(UNIT) becomes valid after M96 is input. The interrupt signal will be ignored even if input in the M97 mode, but if it held until M96 is specified, it will interrupt immediately if the status trigger method is selected; it does not interrupt in case of the edge trigger method.





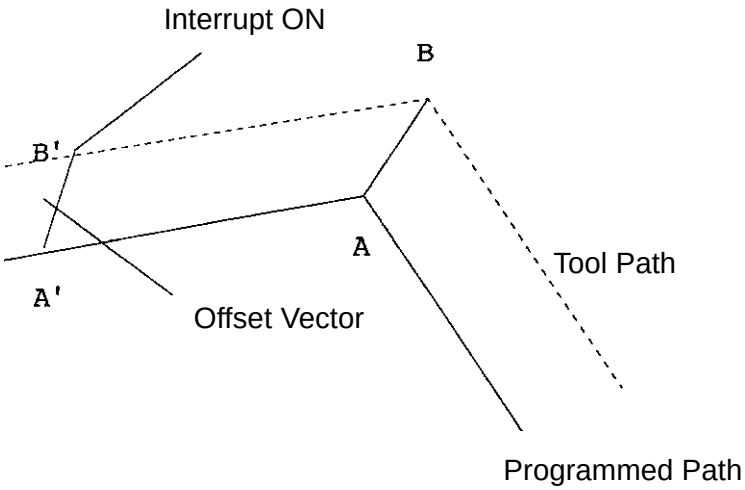
No additional interrupt is not generated while running the custom macro interrupt program, but that state is cancelled by reading M99. However, it is not cancelled until an NC command is started immediately after that.

23-5 Reversion and Modal Information

To return from the custom macro to the original program, specify M99. It is also possible to return to the sequence number in the original program. The modal information is taken over as follows depending on the command used.

(1) When returning by M99
Returns to the modal information existing before the interrupt. That is, the modal information altered in the interrupt program becomes invalid.
(2) When returning by M99 pxxxx
The modal information altered in the interrupt program becomes valid.

23-6 System Variable in Interrupt Program



(1) The coordinates of A can be read until an NC statement appears in #5001 onward.
(2) The coordinates of A' can be read after a moveless NC statement appears.
(3) The machine coordinates and work coordinates of the B' point can be read in #5021 onward and #5041 onward, respectively.

(4) The modal information in the block interrupted in #4201 to #4320 can be read.
(5) #4001 to #4120 continues to hold the original program's information until an NC statement appears.

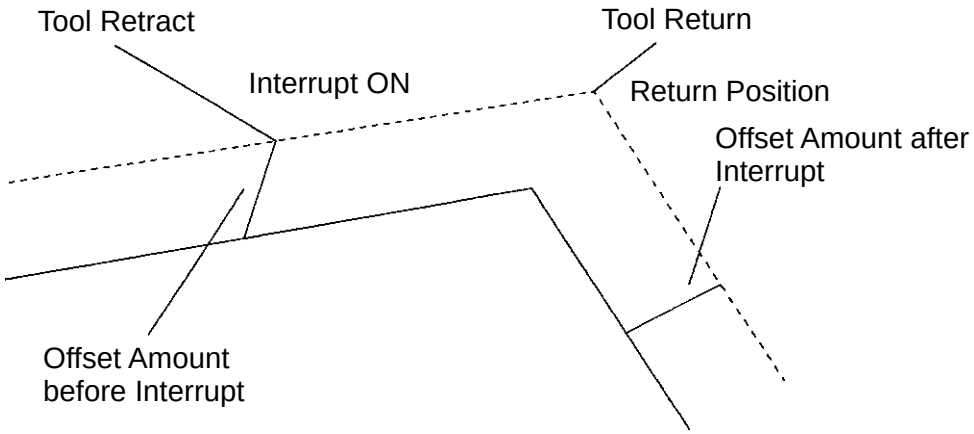
23-7 Custom Macro Interrupt and Custom Macro Modal Call

When a read program is called, a custom macro modal call is temporarily cancelled. If it is respecified in the interrupt program, however, it will be called. The previous status is restored when returning to the original program.

23-8 Interrupt Timing and Return Position in Each Mode

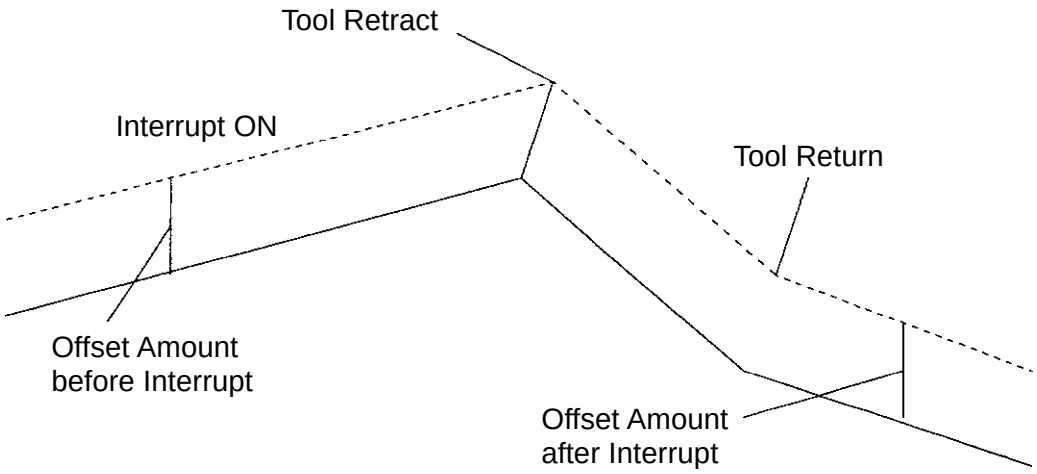
An interrupt program running timing and return position(return position by M99) depends on the G-code in the interrupt generated block.

(1) Normal move commands (G00, G01, G02, G03, etc.)
a. Type-I
When an interrupt takes place, the interrupt program runs, cancelling a move. After the interrupt program is finished, the machine returns to the scheduled end point position of the interrupted block. If an offset amount is altered in the cutter compensation mode or tool length compensation mode, the tool will be offset by the increased/decreased offset amount.



b. Type-II

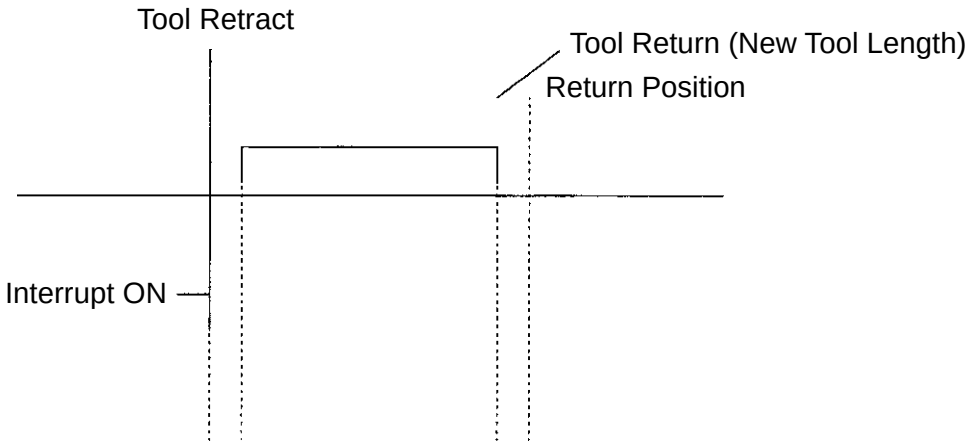
The interrupt program runs after finishing one block (one block created inside the NC unit). After the interrupt program is finished, the tool is positioned to the already calculated next block. If an offset amount is altered, the tool will be offset.



(2) Canned cycle for drilling (Tapping cycle excluded)

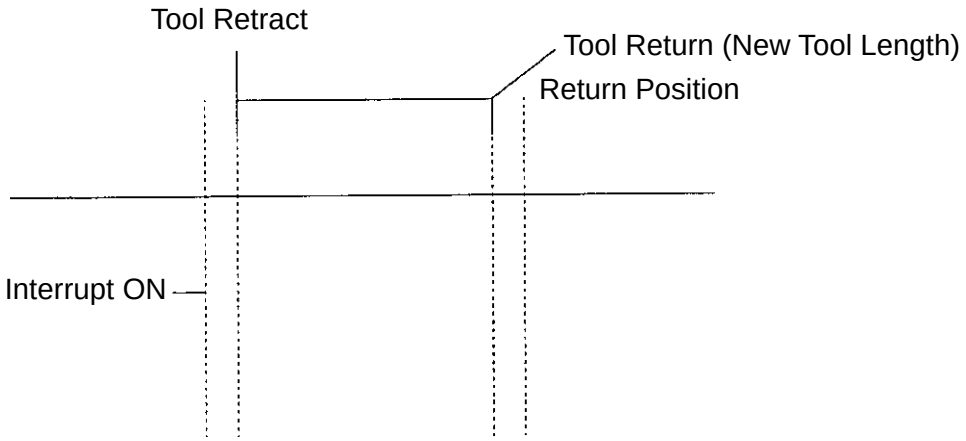
a. Type-I

When an interrupt takes place, the interrupt program runs, cancelling a move. After the interrupt program is finished, the tool is positioned to the next drilling position. If an offset length is altered, the tool will be offset.



b. Type-II

After drilling is finished(at R-point return), the interrupt program runs. After the interrupt program is finished, the tool is positioned to the next drilling position. If an offset length is altered, the tool will be offset.

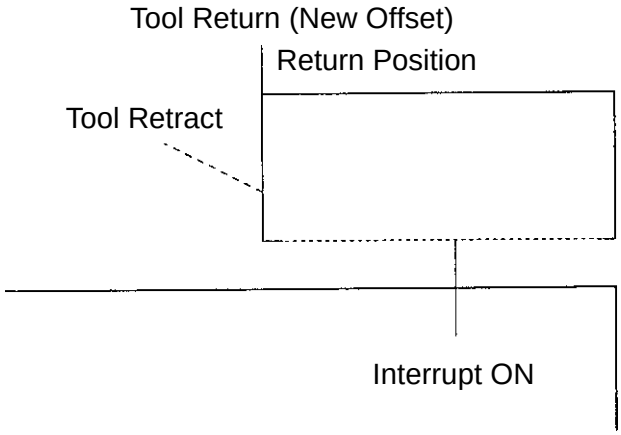


(3) Tapping cycle (G84, etc.)

There is no distinction between Type-I and Type-II. This cycle is fixed to Type-II of the canned cycle for drilling.

(4) Threading

The interrupt program does not run during threading operation. After the interrupt program is finished, the tool is position to a retract position.



(5) Special canned cycles (plane machining, pocket machining, etc.)

a. Type-I

When an interrupt takes place, the interrupt program runs, cancelling a move. After the interrupt program is finished, the tool is positioned to the scheduled end point position of the interrupted block. The tool is not offset even if an offset amount is altered.

b. Type-II

The interrupt program runs after finishing one block created inside the NC unit. After the interrupt program is finished, the tool is positioned to the already calculated next block. The tool is not offset even if an offset amount is altered.

c. Type-III

The interrupt program runs after entirely finishing one cycle. After the interrupt program is finished, the original program runs from the next block.

(6) Skip function (G31)

While the skip function is running, the program cannot be interrupted until its end.

(7) Move commands in the machine coordinate system (G28, G30, G53, etc.)

The program cannot be interrupted until the end of the block.

23-9 Associated Parameters

No.6070 M-code enabling a custom macro interrupt (M96 if 0)

No.6071 M-code disabling a custom macro interrupt (M97 if 0)

No.6001				MCI2	MCI1		TSE	MCSB

MSCB The interrupt macro is;
0: Subprogram
1: Macro type.

TSE The interrupt macro's interrupt signal is;
0: Status trigger
1: Edge trigger

MCI1, MCI2 Type of the interrupt macro

MCI2	MCI1	Type
0	0	1
0	1	2
1	0	3
1	1	3

24. MEMORY OPERATION IN OTHER COMPANIES' FORMATS

24-1 Memory Operation in FS15 Format

The programs in the FANUC's Series 15 data format can be run by setting "1" in the parameter no.3409' #7. The following commands can be operated. For the other data formats, it is necessary to comply with the Σ10M.

(1) Operatable commands

(a) Addition of work coordinate system pairs

G54.1 Pn

(b) Polar coordinate interpolation

G12.1 ; Polar coordinate interpolation mode

G13.1 ; Polar coordinate interpolation cancel mode

(c) Involute interpolation

G02.2 X_ Y_ I_ J_ R_ F_ ; Clockwise

G03.2 X_ Y_ I_ J_ R_ F_ ; Counterclockwise

(d) Floating reference point return

G30.1 X_ Y_ Z_ · · · ;

(e) Work coordinate system preset

G92.1 X0 Y0 Z0 · · · ;

(f) Rigid tap

G84.2 X_ Y_ Z_ R_ P_ F_ S_ L_ ; Rigid tap

G84.3 X_ Y_ Z_ R_ P_ F_ S_ L_ ; Counter rigid tap

(g) Programmable mirror image

G51.1 X_ Y_ Z_ ; Programmable mirror image ON

G50.1 X_ Y_ Z_ ; Programmable mirror image cancel

(h) True circular cutting

G12.2 I_ D_ F_ ; Clockwise

G13.2 I_ D_ F_ ; Counterclockwise

(i) Multibuffer

G05.1 P1 ; Multibuffer mode OFF

G05.1 ; Multibuffer mode ON

(j) High-speed machining

G05.1 Q1 ; Preceding control mode ON

G05.1 Q0 ; Preceding control mode OFF

24-2 Memory Operation in i80M Format

The programs in the YASUKAWA ELECTRIC's Series i80 data format can be run by setting "1" in the parameter no.3409' #6. The following commands can be operated. For the other data formats, it is necessary to comply with the Σ10M.

(1) Operatable commands

(a) Work coordinate system setting
G54(to G59) J1(to J9)

(b) Return to the basic coordinate system
G52

Presets the work coordinate system and cancels coordinate rotation.

(c) Local coordinate system
G52 Q2 X_ Y_

Sets the local coordinate system if there is any Q-command(not necessarily Q2).

(d) Scaling
G51 I_ J_ K_ P_

A magnification factor, P_, should be specified with a decimal point added.

(e) Solid tap mode command
G93

(f) True circular cutting
G12 I_ D_ K_ Q_ F_ ; Clockwise
G12 I_ D_ K_ Q_ F_ ; Counterclockwise

(g) Canned cycle for drilling
G77 Back boring cycle

(h) Tool offset by tool number
Limited to automatic programming

(2) Associated parameters

No.3405, #2 = 0 Does not apply coordinate rotation with a coordinate system selection command
= 1 Applies coordinate rotation with a coordinate system selection command

INSTRUCTION MANUAL
PROGRAMING
SEIKI-SEICOS Σ10M/16M/18M
Version 1.01
6-2000

First Edition 2-1998