SEIKI - SEICOS
$\sum$10L/$\sum$16T/$\sum$18T/$\sum$21L

INSTRUCTION MANUAL

PROGRAMMING

42 Edition 1.01   NO-0000-1-0211-E-1-01

HSD   *Hitachi Seiki Deutschland*
Werkzeugmaschinen GmbH

INTRODUCTION

This manual explains about the programming system of SEICOS-Σ10L, Σ16T,Σ18T and Σ21L.
The manual contains explanation on all functions, however, there are certain functions that are not applicable depending on the type of the machine used. On this matter, please refer to the instruction manual of each machine model when you utilize these functions.

Items requiring attention when reading this manual.
(1) In this manual, those functions not specifically remarked "able" should be understood as "unable".
(2) The contents of this manual may be changed without notice to meet a future machine improvement.
(3) In this manual, all the descriptions take G18 (Z-X) plane as the subject plane, unless otherwise specifically mentioned. For cutting processes made on a plane other than G18, refer to "7.8 Lathe turning on other than the G18 plane".

Σ10L, Σ16T, Σ18T and Σ21L
The processing programs of Σ10L, Σ16T, Σ18T and Σ21L are nearly 100% compatible.
As exceptions, Σ21L does not have the following functions.

    [1] 2.4 Helical interpolation
    [2] 2.5 Virtual axis interpolation
    [3] 4.6 Automatic corner override
    [4] 5.2 Inside arc cutting speed change
    [5] 6.5 Floating reference point return

CONTENTS

1. G CODE

1.1 G Code System

Three kinds of G code systems including A, B, and C are available for selection. Any G code systems are almost the same in their functions and programming methods except only part of the G codes are different.

When Specifying the position of each axis, however, there is a difference in the absolute and incremental programming methods between the A system and non-A systems.

1.1.1 A system

Absolute programming and incremental programming use an axial address to specify an axial position. When X, Y, Z, B, or C is used for the axial address, they assume absolute programming, and when U, V, W, D, or H is used, they assume incremental programming. The following table shows the relations between axial names and axial addresses.

Axial Name	Axial Address	
	Absolute	Incremental
X axis	X	U
Y axis	Y	V
Z axis	Z	W
A axis	A	$- - - - -$
B axis	B	D
C axis	C	H

Axis A does not have the address of the incremental command. It is always specified by absolute. However, only for commands G28, G30 and G301, absolute commands and incremental commands of axis A can be switched over as follows.

(1) When address Q is present on the same block
  - Q=0 → Absolute command
  - Q≠0 → Incremental command

(2) When address Q is not present on the same block
  - Parameter AINC=0 → Absolute command
  - Parameter AINC=1 → Incremental command
  (AINC means 3 bits of parameter number 3404.)

1.1.2　B and C Systems

The following G codes are used to specify either absolute programming or incremental programming.

    G90 : Absolute programming

    G91 : Incremental programming

These G codes are modal ones of Group 03.

(Note 1)　*The G code system A, B, and C are selected by the parameters GSB and GSC (parameter No.3400.)*

(Note 2)　*This command manual is described with the G codes of the A system unless otherwise specified.*

1.2 List of G Code Groups

Group	Function	Remarks
00	Non-modal	
01	Positioning/linear interpolation/circular interpolation	
02	Plane designation	
03	(Absolute programming/incremental programming)	
04	Stored stroke check	
05	Feed per minute/feed per revolution	
06	Inch/metric conversion	
07	Cutter compensation	
08		*2
09	Canned cycle for drilling	
10	Initial point return/R-point return	
11		*2
12	Work length modification	
13	Cutting mode/exact stop mode/automatic corner override mode	
14	Macro modal call	
15	Programmable mirror image	
16	Groove width compensation	
17	Constant surface speed control	
18	Tool life management	
19		*2
20		*2
21		*2
22	Polar coordinate interpolation	
23	Spindle speed fluctuation detection	
24		*2
25	Mirror image for double turrets	*
26		*2
27	Automatic tool nose radius compensation/cutter compensation enable/disable	
28		*2
29		*2
30		*2
31		*2

(Notes) *2 Spare G code group for improvement of the functions.*

1.3 List of G Codes

Group	G code system A	G code system B	G code system C	Function	Remarks
01	G00	G00	G00	Positioning	
	G01	G01	G01	Linear interpolation	
	G02	G02	G02	Circular interpolation/helical interpolation CW	
	G03	G03	G03	Circular interpolation/helical interpolation CCW	
00	G04	G04	G04	Dwell	
	G07	G07	G07	Virtual axis interpolation	
	G09	G09	G09	Exact stop	
	G10	G10	G10	Data setting	
	G11	G11	G11	Data setting mode cancel	
02	G17	G17	G17	Xp-Yp plane designation Xp: X axis or its parallel axis	
	G18	G18	G18	Zp-Xp plane designation Yp: Y axis or its parallel axis	
	G19	G19	G19	Yp-Zp plane designation Zp: Z axis or its parallel axis	
06	G20	G20	G70	Inch input	
	G21	G21	G71	Metric input	
04	G22	G22	G22	Stored stroke check ON	
	G23	G23	G23	Stored stroke check OFF	
23	G25	G25	G25	Spindle speed fluctuation detection OFF	
	G26	G26	G26	Spindle speed fluctuation detection ON	
00	G27	G27	G27	Reference point return check	
	G28	G28	G28	Reference point return	
	G29	G29	G29	Return from reference point	
	G30	G30	G30	2nd, 3rd, 4th reference point return	
	G301	G301	G301	Floating reference point return	
	G31	G31	G31	Skip function	
01	G32	G32	G32	Thread cutting	
	G34	G34	G34	Variable lead thread cutting	
00	G38	G38	G38	Tool nose radius compensation/cutter compensation vector hold	
	G39	G39	G39	Tool nose radius compensation/cutter compensation corner arc	
07	G40	G40	G40	Cutter compensation cancel	
	G41	041	G41	Cutter compensation to the left	
	G42	G42	G42	Cutter compensation to the right	

Group	G code system			Function	Remarks
	A	B	C		
01	G50	G92	G92	Coordinate system setting/spindle maximum speed setting	
	G52	G52	G52	Back face machining mode	Back
	G53	G53	G53	Machine coordinate system selection	
12	G54	G54	G54	Work length modification 1	
	G55	G55	G55	Work length modification 2	*1
00	G59	G59	G59	Local coordinate system setting	
13	G61	G61	G61	Exact stop mode	
	G62	G62	G62	Automatic corner override mode	
	G63	G63	G63	Tapping mode	
	G64	G64	G64	Cutting mode	
00	G65	G65	G65	Macro calling	
14	G66	G66	G66	Macro modal call	
	G67	G67	G67	Macro modal call cancel	
00	G70	G70	G72	Finishing cycle	
	G71	G71	G73	O.D./I.D. roughing cycle	
	G72	G72	G74	End face roughing cycle	
	G73	G73	G75	Closed loop turning cycle	
	G74	G74	G76	End face cutting-off cycle	
	G75	G75	G77	O.D./I.D. cutting-off cycle	
	G76	G76	G78	Compound type thread cutting cycle	
09	G80	G80	G80	Drilling cycle cancel	
	G81	G81	G81	Drilling cycle, spot drilling cycle	
	G82	G82	G82	Drilling cycle, counter boring cycle	
	G83	G83	G83	Peck drilling cycle	
	G831	G831	G831	Peck drilling cycle	
	G84	G84	G84	Tapping cycle	
	G841	G841	G841	Counter tapping cycle	
	G842	G842	G842	Direct tapping cycle	*1
	G843	G843	G843	Counter direct tapping cycle	*1
	G85	G85	G85	Boring cycle	
	G86	G86	G86	Boring cycle	
	G861	G861	G861	Fine boring cycle	
	G87	G87	G87	Back boring cycle	
	G88	G88	G88	73. Boring cycle	
	G89	G89	G89	74. Boring cycle	

Group	G code system			Function	Remarks
	A	B	C		
01	G90	G77	G20	O.D./I.D. turning cycle	
	G92	G78	G21	Single type thread cutting cycle	
	G94	G79	G24	End face turning cycle	
17	G96	G96	G96	Constant surface speed control	
	G196	G196	G196	Constant surface speed control (Back)	Back
	G97	G97	G97	Constant surface speed control cancel	
05	G98	G94	G94	Feed per minute	
	G99	G95	G95	Feed per rotation	
03	– – –	G90	G90	Absolute programming	
	– – –	G91	G91	Incremental programming	
22	G120	G120	G120	Polar coordinate interpolation mode cancel	
	G121	G121	G121	Polar coordinate interpolation mode	
00	G128	G128	G128	Scroll cutting speed control	
18	G130	G130	G130	Tool life management OFF	
	G131	G131	G131	Tool life management ON	
27	G140	G140	G140	Automatic tool nose compensation/cutter compensation cancel mode	
	G143	G143	G143	Automatic tool nose compensation enable mode	
	G144	G144	G144	Automatic tool nose radius compensation enable mode (G144 = G143)	
	G145	G145	G145	Cutter compensation enable mode	
00	G141	G141	G141	Automatic tool nose radius compensation to the left	
	G142	G142	G142	Automatic tool nose radius compensation to the right	
16	G150	G150	G150	Groove width compensation cancel	
	G151	G151	G151	Groove width compensation for end face	
	G152	G152	G152	Groove width compensation for O.D./I.D.	
00	G159	G159	G159	Automatic Conversion of Rotation ATC-C Tool Offset	
25	G170	G170	G170	Face machining mode	Back
	G171	G171	G171	Back face machining mode	Back
00	G194	G194	G194	External measurement compensation	

Group	G code system			Function	Remarks
	A	B	C		
10	G198	G198	G198	Canned cycle for drilling initial point return	
	G199	G199	G199	Canned cycle for drilling R-point return	
01	G212	G212	G212	Circular thread cutting CW	*1
	G213	G213	G213	Circular thread cutting CCW	*1
	G216	G216	G216	Spline interpolation	*1
	G222	G222	G222	Involute interpolation CW	*1
	G223	G223	G223	Involute interpolation CCW	*1
	G232	G232	G232	Exponential function interpolation CW	*1
	G233	G233	G233	Exponential function interpolation CCW	*1
00	G251	G251	G251	Multibuffer	
	G261	G261	G261	S-designation for the spindle	
	G262	G262	G262	S-designation for the rotary tool	
	G263	G263	G263	S-designation for the subspindle	Back
	G271	G271	G271	Cylindrical interpolation	
15	G501	G501	G501	Resetting programmable mirror image	
	G511	G511	G511	Setting programmable mirror image	
00	G921	G921	G921	Work coordinate system preset	

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

(Note 2) The G code systems, A,B, are selected by the parameters GSB and GSC (parameter No. 3400).

(Note 3) "Back" in the Remarks column indicates availability for the back machining system.

2. INTERPOLATION FUNCTION

2.1　Positioning (G00)

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

2.1.1　Command Format

　　　G01　X＿＿＿　Y＿＿＿　Z＿＿＿　......　F＿＿＿　;

2.1.2　Sample Program

(1) Absolute programming
　　　G00　X50, Z100, ;
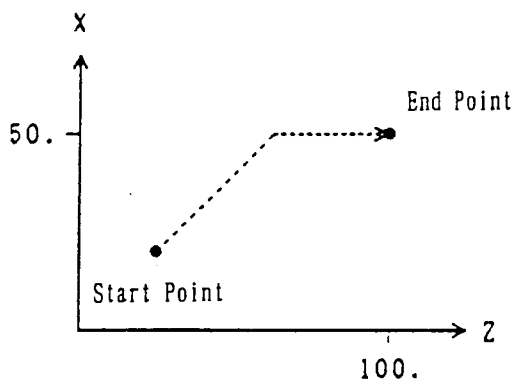
(2) Incremental programming
　　　G00　X50, Z100, ;
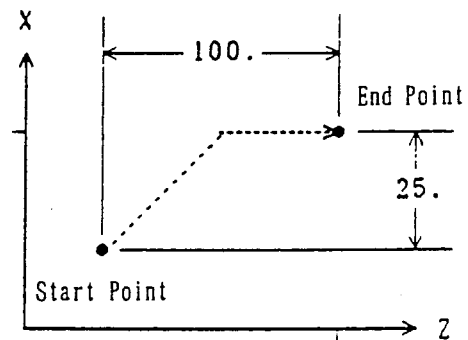
2.1.3　Cautions

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

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

(3) 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.

(4) The tool path can be made linear by altering the parameter.
　　　G00 X50, Z100, ;
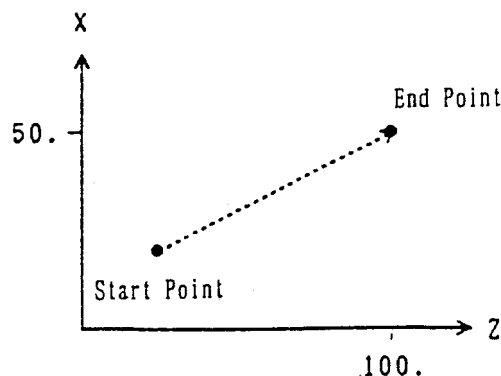
When linear interpolation positioning has been selected, shifting takes place in the speed which assures the shortest positioning time within the scope not exceeding rapid traverse rate for each axis.

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

2.1.4 Associated Parameters

No.1401, #1 = 0   Positioning system is non-linear type.

              = 1   linear type (linear interpolation).

No.1401, #6 = 0   Dry run disabled for rapid traverse command.

                      Dry run enabled for rapid traverse command.

No.3402, #0 = 0   Reset state is G00 mode.

              = 1   Reset state is G01 mode.

No.1420   Rapid traverse rate for each axis.

No.1620   Time constant of rapid traverse linear acceleration/deceleration for each axis.

2.2 Linear Interpolation

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

2.2.1 Command Format

2.2.2 Sample Program

(1) Absolute programming
    G01 X50. Z100. F200 ;
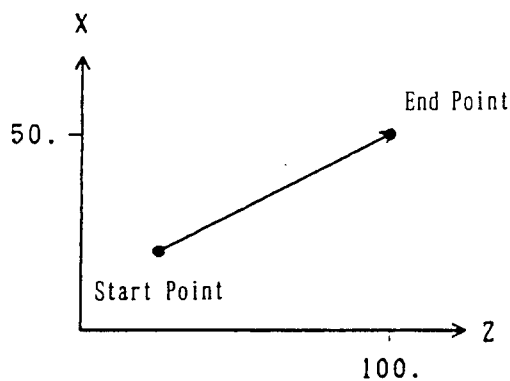
(2) Incremental programming
    G01 X50. W100. F200 ;
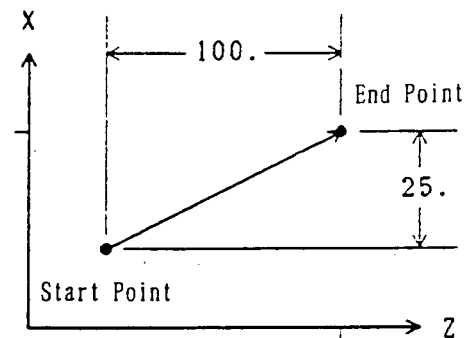
2.2.3 Cutting Feed Rate

The cutting feed rate specified with an F code is the speed at which the toolmoves 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 Ua Vb Wc Ff ;

X-axis cutting feed rate $Fx = af/2L$

Y-axis cutting feed rate $Fy = bf/L$

Z-axis cutting feed rate $Fz = cf/L$

where; $L = \sqrt{(a/2)^2+b^2+c^2}$

(Note) *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.*

2.2.4 Cautions

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

(2) Exponential type acceleration/deceleration is applied.

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

2.2.5 Associated Parameters

No.3402, #0 = 0   The reset state is the G00 mode

          = 1   The reset state is the G01 mode

No.1422   Maximum cutting feed speed (common to all axes)

No.1622   Time constant of cutting feed exponential type acceleration/deceleration

No.1623   FL speed of cutting feed exponential type acceleration/deceleration

2.2.6 Associated Alarms

No.102   F has not been specified in cutting feed. Or, F0 has been specified.

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.

2.3.1 Command Format

(1) Xp-Yp plane

$$\text{G17} \begin{bmatrix} \text{G02} \\ \text{G03} \end{bmatrix} \text{Xp\_} \quad \text{Yp\_} \quad \begin{bmatrix} \text{I\_} & \text{J\_} \\ \text{R\_} & \end{bmatrix} \text{F\_} \quad ;$$

(2) Zp-Xp plane

$$\text{G18} \begin{bmatrix} \text{G02} \\ \text{G03} \end{bmatrix} \text{Zp\_} \quad \text{Xp\_} \quad \begin{bmatrix} \text{K\_} & \text{I\_} \\ \text{R\_} & \end{bmatrix} \text{F\_} \quad ;$$

(3) Yp-Zp plane

$$\text{G19} \begin{bmatrix} \text{G02} \\ \text{G03} \end{bmatrix} \text{Yp\_} \quad \text{Zp\_} \quad \begin{bmatrix} \text{J\_} & \text{K\_} \\ \text{R\_} & \end{bmatrix} \text{F\_} \quad ;$$

where, Xp : X axis or its parallel axis
   Yp : Y axis or its parallel axis
   Zp : Z axis or its parallel axis

2.3.2 Arc Rotating Direction

   G02 : Clockwise (CW)
   G03 : Counterclockwise (CCW)
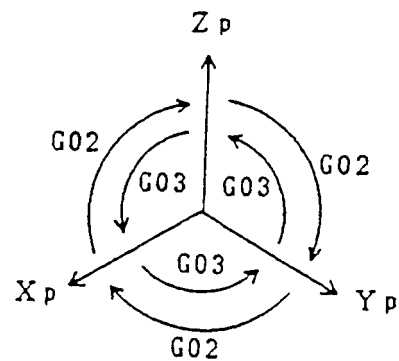
2.3.3 Arc Plane

The arc plane is specified with G17, G18, or G19.

    G17 : Xp-Yp plane
    G18 : Zp-Xp plane
    G19 : Yp-Zp plane

2.3.4 Arc Center

The arc center is specified with I, J, or K corresponding to Xp, Yp, and Zp, respectively. In this case, I, j, and K are the vector components when viewing the arc center from its start point.
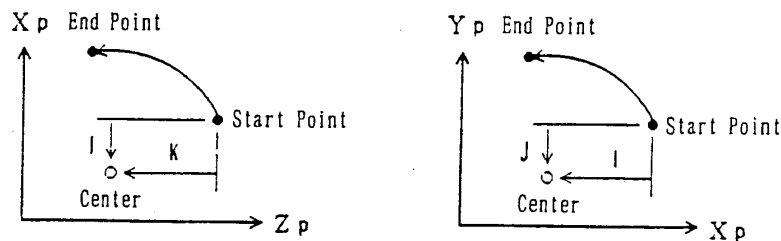
(Note 1) *Instead of using I, J, and K, you can use R to assign an arc radius. (R assignment of arc radius) When applying R assignment to an arc of 180° and above, specify an R in minus.*

(Note 2) *A full circle (360°) is not applicable in R assignment. Use I, J, and K for a full circle.*
When an arc close to 180° or to 360° is specified in R assignment, calculation error may be produced, resulting in center deviation.
In this case, assign the center by using I, J, and K.

2.3.5 Cutting Feed Rate

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

2.3.6 Program Example

(1) Absolute Command Using I, J, and K:

    G18 G00 X140. Z13.397 ;
    G02 X240. Z100. I-50. K86.603 F200 ;
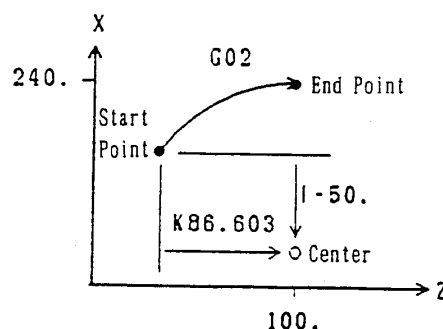
(2) Incremental Command Using I, J, and K:
G18 G02 U100. W86.603 I-50. K86.603 F200 ;
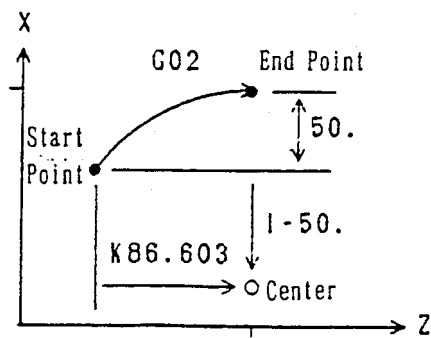
(3) An Arc of 180° or Less Using Radius R Assignment:
G18 G02 U200. W100. R100. ;
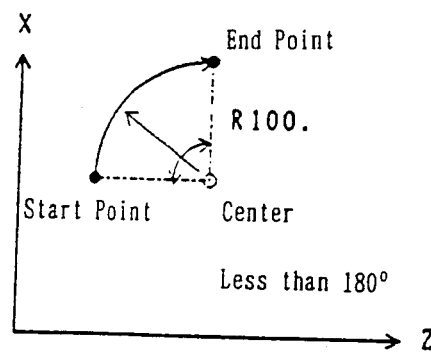
(4) An Arc of 180° or more Using Radius R Assignment:
G18 G02 U200. W100. R100. ;
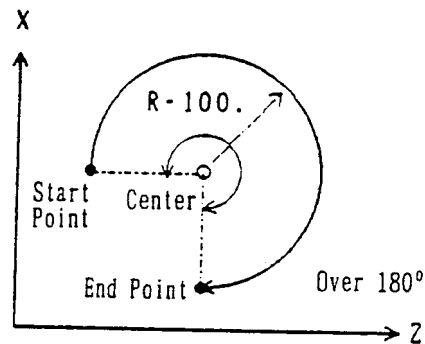
2.3.7   Cautions

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

(2) Exponential type acceleration/deceleration is applied.

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

(4) I0, J0, and K0 are omissible.

(5) 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.
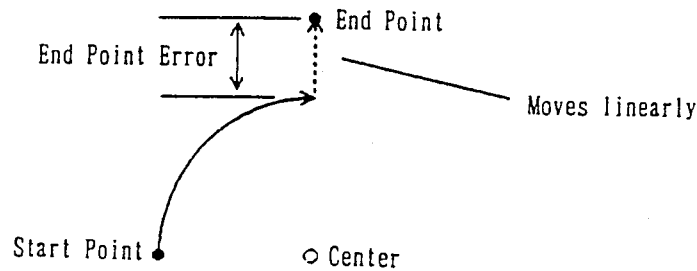
(6) An alarm results if the axis not for the arc plane is specified.
(7) When R is specified in the same block as I, J, and K, R is given priority.
(8) When the canter of a circular arc is not calculated, alarm takes place.

2.3.8   Associated Parameters

No.1422   Maximum cutting feed speed (common to all axes)
No.1622   Time constant of cutting feed exponential type acceleration/deceleration
No.1623   FL speed of cutting feed exponential type acceleration/deceleration
No.3459   Arc radius error limit value

2.3.9   Associated Alarms

No.102   No F has been commanded in cutting feed. Or, F0 has been commanded.
No.131   The R value in arc radius R assignment is erroneous.
No.132   An arc for which difference in radius values between the start point and the end point is larger than the parameter set value has been commanded.
(#001)   No center command (I, J, K)
(#002)   Center command (I, J, K) is 0.
(#003)   Arc end point allowable error is too large.

(Note) When an option for helical interpolation is not added, specifying the axis for other than the arc plane results in the alarm #191 (OPTION COMMAND).

2.4 Helical Interpolation (G02, G03).... Not available with Σ21L.

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.

2.4.1 Command Format

(1) Xp-Yp plane

$$G17 \begin{bmatrix} G02 \\ G03 \end{bmatrix} Xp\_ \quad Yp\_ \quad \begin{bmatrix} I\_ \quad J\_ \\ R\_ \end{bmatrix} a\_ \quad F\_ \quad ;$$

(2) Zp-Xp plane

$$G18 \begin{bmatrix} G02 \\ G03 \end{bmatrix} Zp\_ \quad Xp\_ \quad \begin{bmatrix} K\_ \quad I\_ \\ R\_ \end{bmatrix} a\_ \quad F\_ \quad ;$$

(3) Yp-Zp plane

$$G19 \begin{bmatrix} G02 \\ G03 \end{bmatrix} Yp\_ \quad Zp\_ \quad \begin{bmatrix} J\_ \quad K\_ \\ R\_ \end{bmatrix} a\_ \quad F\_ \quad ;$$

where; Xp : X axis or its parallel axis
Yp : Y axis or its parallel axis
Zp : Z axis or its parallel axis
α : Any optional linear axis for other than circular interpolation (up to 2 axes)
F : Arc speed

2.4.2 Feed Speed

Feed speed command F in helical interpolation is equal to the speed in which a tool moves on an arc.
Speed of the straight line axis = F $\times$ (length of the straight line axis/the arc length)

2.4.3 Sample Program

G17 G03 U-200. V100. R100. W50. F200 ;

Tool Path
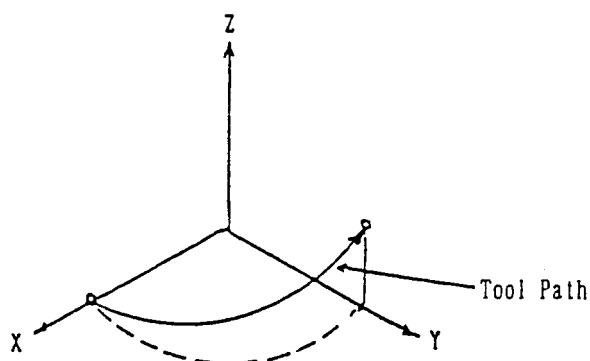
2.4.4 Cautions

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

(2) Cutter compensation is applied to circular interpolation.

(3) The axes for other than circular interpolation can be specified up to 2 axes. Specifying 3 axes or more results in an alarm.

(4) The tool speed can be the composite speed of arc and linear speeds by parameter setting.

2.4.5 Associated Parameters

2.4.6 Associated Alarms

No.149   In helical cutting, 3 or more linear axes have been commanded.

2.5   Virtual Axis Interpolation (G07) .... Not available with Σ21L.

When a virtual axis is assigned, axis shift does not take place.
In helical interpolation, by making one of circular command axes as a virtual axis, you can perform SIN interpolation.

2.5.1   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 α axis as the virtual axis

where ;   α : Any one axis

2.5.2   Sample Program

G07 Y0 ;
G17 G03 U0 V0 I-50. Z100. F200 ;
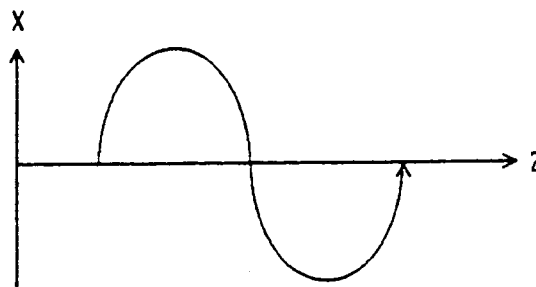
2.5.3   Cautions

(1) Virtual axis assignment is applicable only to 1 axis. If 2 or more axes should be assigned, alarm takes place.
(2) When axial shift has been commanded for the axis assigned as a virtual axis, dwell state equal to the shift time is produced.
(3) Command the virtual axis in incremental.

2.5.4   Associated Alarms

No.139   Two or more virtual axes are specified.

2.6   Cylindrical Interpolation (G271)

The stroke of the rotary axis internally specified in terms of angle is converted into the circumferential distance by specifying the stroke of the linear axis and the angle of the rotary axis with a program command. Since the circumferential distance can be regarded as the stroke of the linear axis on the circumference, linear interpolation and circular interpolation can be performed with other linear axes. After interpolation, it is reconverted into the angle of the rotary axis.

2.6.1   Command Format

G271   C Cylinder radius ;   Cylindrical interpolation ON

Cylindrical interpolation mode

G271   C0 ;   Cylindrical interpolation cancel

C denotes the rotary axis.

The move angle of the rotary axis is calculated back from the circumferential stroke.
For example, when you want to move 100.0 on the circumference of a cylinder whose radixis is 50.0, the move angle of the rotary axis is obtained from the following expression.

r   : Cylinder radius
θ   : Move angle
s   : Circumferential stroke

$$\text{Move angle} = \frac{\text{(Circumferenttial stroke)}}{\text{(Cylinder radius)}}$$

$$= \frac{360 \times 100.0}{2 \times \pi \times 50.0} = 114.591$$
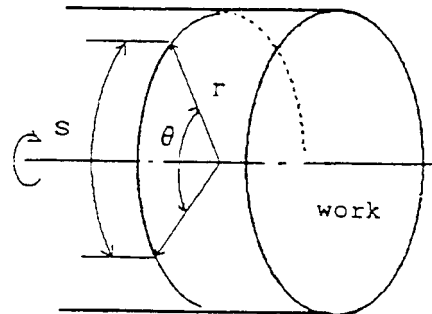
2.6.2 Feed Rate

Feed rate command F in the cylindrical interpolation mode is the speed at which the tool moves around the perimeter of a cylinder.

2.6.3 Circular Interpolation Axis

A linear as well as a rotating axis for which circular interpolation is performed are set in parameters beforehand (Straight line axis: No.3426, Rotary axis: No.3427).
The set range for both parameters, however, should be within 1~ no. of control axes and the parameters should not be the same for both axis.

2.6.4 Circular Interpolation Plane

During the circular interpolation mode, a plane for which the rotating axis is set to 1st plane axis and the straight line axis to 2nd plane axis is automatically selected.

2.6.5 Sample Program (X Axis is Diameter Specification)

(C-Z plane selected with Parameter No. 3426/3427.)

```
    T0100 ;
    G98 G145 G40 G80 ;
    G00 X120.0 Z-120.0 C0 ;
    G145 ;
    G271 C50.0 ;                          Cylindrical interpolation mode ON
N1  G42 G01 Z-40.0 F500 ;                 (Cylinder radius = 50.0)
    G01 X100.0 F100 ;
N2  C90.0 F500 ;
N3  Z-100.0 C180.0 ;
N4  C260.0 ;                              In the cylindrical
N5  G03 Z-80.0 C282.918 R20.0 ;           interpolation mode
N6  G01 Z-60.0 ;
N7  G02 W20.0 H22.918 R20.0 ;
N8  G01 C360.0 ;
N9  G40 G01 Z-120.0 ;
    G271 C0 ;                             Cylindrical interpolation mode OFF
    G00 Z50.0 C0 ;
```

Developed View of Cylinder Surface with Radius of 50.0
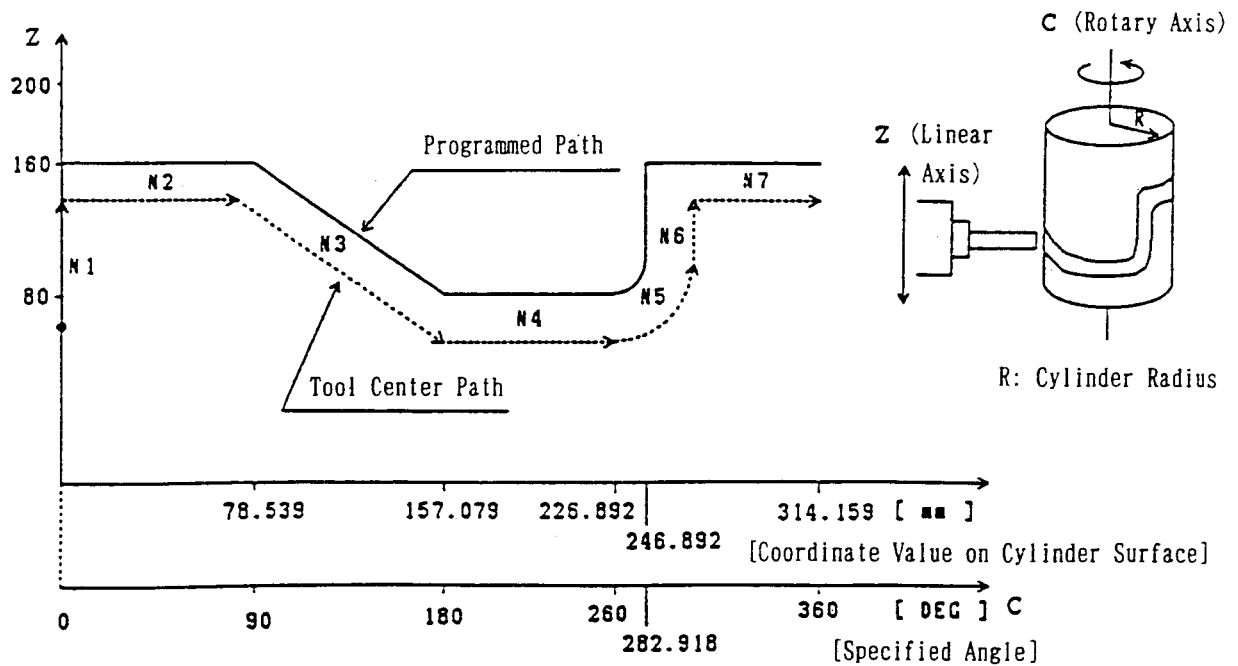
2.6.6 Cautions

(1) Arc radius programming with I, J, or K is not allowed during the cylindrical interpolation mode. Use radius designation on arc and specify it with the coordinate value on the cylinder surface.

(2) During cylinder interpolation mode, the plane (plane selected by G17 - G19) before entering the cylinder interpolation node will be canceled once, and then restored after completion of the cylinder interpolation mode.

(3) The circular arc radius specification method (which of the addresses I, J, K to use) when performing circular arc interpolation during the cylinder interpolation mode, will be decided depending on which axis of the basic coordinate system the second axis (straight line axis) of the cylinder interpolation plane corresponds to, as shown below.
- When the straight line axis is the X axis or parallel to it, Zp-Xp plane will be taken, specified by K, I.
- When the straight line axis is the Y axis or parallel to it, Xp-Yp plane will be taken, specified by I, J.
- When the straight line axis is the Z axis or parallel to it, Yp-Zp plane will be taken, specified by J, K.

Also, the circular arc radius can be specified by the R command.

(4) (G271 Cxx)

(5) An alarm results if 2 or more rotary axes are specified with the G271 command.

(6) Cylindrical interpolation is not available for the following functions:
- Machining break point return
- Manual intervention by manual absolute ON

(7) An alarm results if the following commands are given during the cylindrical interpolation mode.

G28, G30, G53	Machine coordinate system
Thhttxx; (T command), G54, G50	Work coordinate system
G70-G76, G81-G89, G831-G861, G861	
G90-94	Various canned cycles
G31, G121, G232	Others

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

(9) Due to the above conversion error, a circular interpolation alarm may occur if circular interpolation with a small circular arc radius is performed during the cylindrical interpolation mode. When using, care should be taken. Also, a cutter compensation alarm may occur at the time of cutter compensation due to the above reason.

(10) An alarm results if G271 Cxx; (C ≠ 0) (cylindrical interpolation mode ON) is specified during the cylindrical interpolation mode.

(11) The remaining stroke indicates the value at the time of moving on the perimeter of the cylinder.

2.6.7   Related Parameters

No.3426   Axis number (1 - No of control axes) of the straight line axis for performing
            circular interpolation

No.3427   Axis number (1 - No of control axes) of the rotating axis for performing circular
            interpolation

2.6.8   Related Alarm

No.126   An error exists in cylindrical interpolation command.
 (#001)   No rotary axis command exists.
 (#002)   No independent command exists.
 (#003)   Modal G code is not in correct state.
No.127   An erroneous command has been give while in cylindrical interpolation mode.
 (#001)   An erroneous G code has been commanded.
 (#003)   Rotary axis shifting command has been given in G00 mode.
 (#004)   Cylindrical interpolation has been commanded again while in cylindrical
            interpolation mode.
No.189 (G271)   Parameter setting is erroneous.

- Parameters No. 3426/No.3427 do not fall within 1 ~ no. of control axes. Or,
  No.3426 and No.3427 are equal.
- The straight line axis set in Parameter No.3427 does not correspond to or
  parallel to any of the basic three axes (X, Y, Z).

2.7   Polar Coordinate Interpolation (G120, G121)

Polar coordinate interpolation is a function to provide contour control by converting the command programmed in the orthogonal coordinate system into the movement of the linear axis (tool movement) and that of the rotary axis (workpiece rotation).

<Orthogonal Coordinate System>   <Polar Coordinate System>
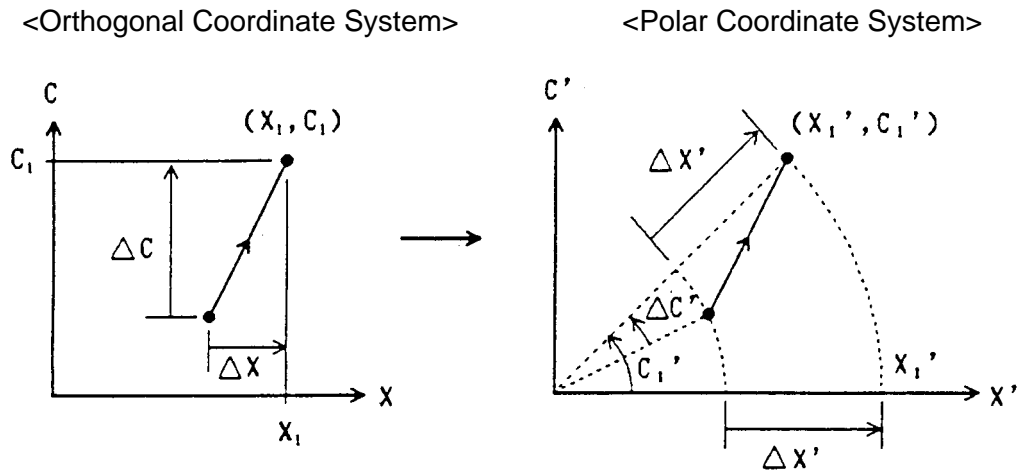
The relationship between (X1, C1) and (X1', C1') is as follows.

$$X_1' = \sqrt{X_1^2 + C_1^2} \qquad\qquad C_1' = \tan^{-1}(C_1/X_1)$$

2.7.1   G Codes

G121 : Polar coordinate interpolation mode
G120 : Polar coordinate interpolation mode cancel

2.7.2   Command Format

G121 ;   Turns polar coordinate interpolation mode ON

$\left.\begin{array}{l} \vdots \\ \vdots \\ \vdots \\ \vdots \\ \vdots \end{array}\right]$ Performs polar coordinate interpolation.

G120 ;   Cancels polar coordinate interpolation mode.

2.7.3   Polar Coordinate Interpolation Axes

A straight line axis as well as a rotating axis for the polar coordinate interpolation are set in the parameters beforehand (Straight line axis: No.3418, Rotating axis: No.3419).
The set range for both parameters should be within 1 ~ no. of control axes and the parameters should not be the same for both axis.

2.7.4   Polar Coordinate Interpolation Plane

When G121 is specified, the polar coordinate interpolation mode is put in effect, and the plane (polar coordinate interpolation plane) is selected, which assumes the origin of the work coordinate system as that of the coordinate system, the linear axis as the first plane axis, and virtual axis (rotary axis) orthogonal to the linear axis as the second plane

axis. Polar coordinate interpolation is performed on this plane. For the position specified by G121, the angle will be taken as 0 and polar coordinate interpolation started, regardless of the actual position. Therefore, create the program by taking the work coordinate value of the rotating axis as 0 when G121 has been specified.

2.7.5   Program Command

(1) The program command in the polar coordinate interpolation mode is given in terms of the orthogonal coordinate value of the polar coordinate interpolation plane.
The rotary axis address is used for the command address of the 2nd plane axis. However, the programmable unit is not degrees; the same unit as that for the 1st plane axis, linear axis (mm or inch). The 2nd plane axis assumes the same designation as the rotary axis; it is neither diameter designation nor radius designation.

(2) During the polar coordinate interpolation mode, linear interpolation (G01) and circular interpolation (G02, G03) can be specified. Both absolute programming and incremental programming are allowed.

(3) When performing circular interpolation on the polar coordinate interpolation plane, the circular arc specifying-method (which one of I, J, and K is to be used as the address) is determined as follows depending on to which axis of the basic coordinate system the 1st plane axis (linear axis) corresponds (parameter setting: No.1011).

- When the linear axis is either X axis or its parallel axis, the Xp-Yp plane is assumed, and I/J is used to specify the circular arc radius.
- When the linear axis is either Y axis or its parallel axis, the Yp-Zp plane is assumed, and J/K is used to specify the circular arc radius.
- When the linear axis is either Z axis or its parallel axis, the Zp-Xp plane is assumed, and K/I is used to specify the circular arc radius.

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 for the profile after cutter compensation.
However, specify G121 and G120 in the cutter compensation cancel mode (G40).

(5) For feed rate, specify the shift speed of the tool in the polar coordinate interpolation plane (orthogonal coordinate system) by F.
Since this is normally specified by feed per minute (G98), the units of F will be mm/min, inch/min.

2.7.6 Sample Program (X Axis [Diameter Assigned]: Linear Axis/C Axis[Radius Assigned]: Rotary Axis)

```
        G00 X130.0 C0 ;              Positioning to start position
        G145 ;                       Tool Diameter Compensation Effective mode
        G121 ;                       Polar coordinate interpolation started.
N1  G42 G01 X80.0 F100 ;      ─┐
N2  C40.0 ;                    │
N3  G03 X60.0 C50.0 R10.0 ;    │
N4  G01 X-60.0 ;               │   Profile program,
N5  C-15.0 ;                   │   Program with orthogonal coordinate
N6  G03 X-50.0 C-20.0 I5.0 J0 ;│   values on the X-C' plane
N7  G01 X80.0 ;                │
N8  C0 ;                       │
N9  G40 X130.0 ;              ─┘
        G120                         Polar coordinate interpolation cancelled.
```

2.7.7 Speed Clamping of the Rotating Axis

The maximum cutting feed speed for the polar coordinate interpolation can be set in the parameter. (Parameter No.3464)

When any speed above this level is specified while in polar coordinate interpolation, it is clamped to this speed. With the set value being 0, clamping is performed to the ordinary maximum cutting feed speed (Parameter 1422) even while in polar coordinate interpolation.

Further, in polar coordinate interpolation, as a shape having been programmed in the orthogonal coordinate system are converted into movement of the straight line and the rotating axes, rotating axis speed is increased in movement around the work center (origin of the coordinate system), even exceeding the limit speed of Machine.

In the above case, the allowable speed of the rotating axis can be set in the parameter as follows:

Allowable speed of rotating axis = Max. cutting feed speed (Parameter No. 3404) x Allowable ratio (Parameter No.3420)

When the speed component of the rotating axis exceeds the allowable speed while in polar coordinate interpolation, override shown below is automatically applied (Automatic override):

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

When the speed component of the rotating axis still exceeds the allowable speed even with override being applied, feed speed is clamped so that the speed component of the rotating axis does not exceed the maximum cutting feed speed. (Automatic speed clamp)

2.7.8 Rapid Traverse (G00) Operation

You can issue the G00 command during the polar coordinate interpolation mode. However, only the start point and the end point will be converted to the positions of the polar coordinate system for the G00 block.

There will be no conversion during the path from the start point to the end point.

Example: In the G121 mode, if the following command is issued:

    G00 X100. C50.0 ;

The path will be the same as specified in the following command in the G120 mode:

    G01 X141. 421 C45.0 ;

In the shift of the rotating axis by G00, close rotation (shift within +/-180 degrees) will be performed and the processing time will be reduced.

2.7.9 Special Specification A for Polar Coordinate Interpolation (Optional)

Through parameter setting, machining by a program of the X-Y coordinate system is made available in the same manner with the machining center.

In G121 mode, the following conversion processes will be performed immediately after the program is read in.

- X axis commands are processed as radius commands.
  (X axis command is doubled)
- Y axis commands are converted to C axis commands. (Y → C, V → H)

(Note 1) *Since the above conversion processes are performed on the basis of axis name, the parameters must be set with the straight line axis as the X axis and the rotating axis C axis that are subject to the polar coordinate interpolation.*

(Note 2) *Although X axis commands are possible with radius values, the coordinate indication will follow the convention of the mechanical specification. (usually diameter values)*

(Note 3) *Evan if the special specification A for polar coordinate interpolation is employed, either the standard of special specification can be selected with the parameters.*

2.7.10 Cautions

(1) G120 and G120 should be specified in an independent block.

(2) Before specifying G121, the work coordinate system where the center of the rotary axis serves as the origin of the coordinate system must be set. During the G121 mode, the coordinate system must not be altered.

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

(4) The following G codes can be specified in the G121 mode:
G00, G01, G02, G03, G04, G09, G40, G41, G42, G65, G66, G67, G98, G99, G140, G143, G144, G145

(5) For the axis move command outside the plane of the G121 mode, the axis moves regardless of polar coordinate interpolation.

(6) The display of the "remaining stroke" in the G121 mode represents that in the polar coordinate plane (orthogonal coordinates).
For G00 block, however, the unit for the ordinary case is applied.

(7) The following functions are not available for the blocks which are in the G121 mode:
Program restart, Machining break point return, Manual intervention by manual absolute ON

(8) On completion of polar coordinate interpolation (G120), the work coordinate of the rotating axis can be rounded at 360° through parameter setting. ($0° \leqq$ coordinate value $< 360°$)

(9) When the work coordinate value of straight line axis during G121 command is negative for the first shift block, only the end point will be converted to the polar coordinate system position. There will be no conversion during the shift. (Same as the straight line type G00 in the G121 mode.)
From the next shift block, polar coordinate conversion will be performed for the shift path. For the first, shift block, only G00 and G01 can be specified; G02 and G03 cannot be specified.

(10) Through parameter setting, you can specify G codes (G120/G121) for polar coordinate interpolation by using G126/G127.
In specification with G126/G127, they are converted into standard G codes (G120/G121) when an NC program is read into the buffer.
Therefore, command values and alarms are displayed in the standard G codes.

2.7.11 Associated Parameters

No.3404, #4 = 0   Work coordinates of rotating axis is not rounded on completion of polar coordinate interpolation.

       = 1   Work coordinates of rotating axis is rounded at 360° completion of polar coordinate interpolation.

No.3404, #6 = 0   Y-axis command in the polar coordinate interpolation is moved as Y-axis command.

       = 1   Y-axis command in the is moved by being replaced by C-axis command.

No.3404, #7 = 0   X-axis command in polar coordinate interpolation mode follows machine specifications.

       = 1   X-axis command in polar is taken as radius assignment.

No.3409, #0 = 0   G codes for polar coordinate interpolation are G120/G121 (standard)

       = 1   G126/G127

No.3410, #1 = 0   Neither automatic override nor automatic speed clamp is performed while in polar coordinate interpolation mode.

       = 1   Automatic override and automatic speed clamp are performed while in polar coordinate interpolation mode.

No.3418   Axis number of the linear axis for which polar coordinate interpolation is performed

No.3419   Axis number of the rotary axis for which polar coordinate interpolation is performed

No.3420   Axis number of the linear axis for which polar coordinate interpolation is performed.

No.3464   Maximum cutting feed speed while in polar coordinate interpolation.

2.7.12 Associated Alarms

No.113   An error exists in the polar coordinate interpolation command.
(#001)   G120/G121 commands is not independent type.
(#002)   When G120/G121 is specified, cutter compensation has not been cancelled.
(#003)   When the work coordinate value of the linear axis is in minus on specification of G121, a command other than G00/G01 has been specified in the first shift block.
No.114   An erroneous command has been specified while in the polar coordinate interpolation mode.
(#001)   A G code not useable has been specified.
(#002)   A T code has been specified.
No.189   (G121) An error exists in parameter setting.

- Parameters No.3418/No.3419 are not within 1 ~ no. of control axes, or No.3418 is equal to No.3419.
- The linear axis set in Parameter No.3418 is none of the basic three axes (X/Y/Z), or not parallel to any of them.

2.8 Angle Designated Linear Interpolation

By commanding an angle (A) formed between X or Z axis shifted and +Z axis, you can perform angle designated linear interpolation.

2.8.1 Command Format

G01 X___ A___ ;
G01 Z___ A___ ;

2.8.2 Angle A

(1) Scope of angle A

-360.000 ≦ A ≦ 360.000 (deg)

(2) Angle A code

Plus .... Counterclockwise angle from +Z axis
Minus .... Clockwise angle from +Z axis
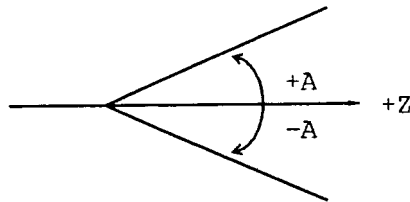
2.8.3 Program Example

G18 G01 U20. A60. ;
W20. ;
W10. A-30. ;
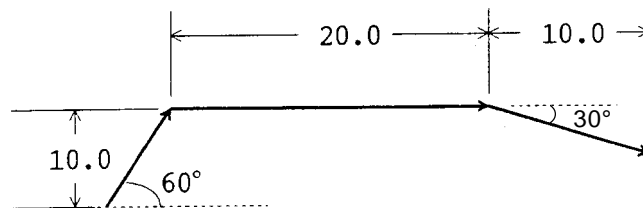
2.8.4 Cautions

(1) When A is used as an axis name, this function is made invalid.
   Address A indicates A axis, not an angle.

(2) Angle designated linear interpolation on other planes (G17/G19) is available.
   As this angle A, an angle formed by the horizontal axis of the coordinate system in plus direction (G17:+X, G19:+Y) is commanded.

(3) When the shift direction of the commanded axis and the shift direction after calculation are opposite, alarm takes place.

   G01 W100. A100. ;

<<Available A Range for Commanding in +Z Direction Command>>

$$0° ≦ A < 90° (-360° ≦ A < -270°)$$

$$270° < A ≦ 360° (-90° < A ≦ 0°)$$
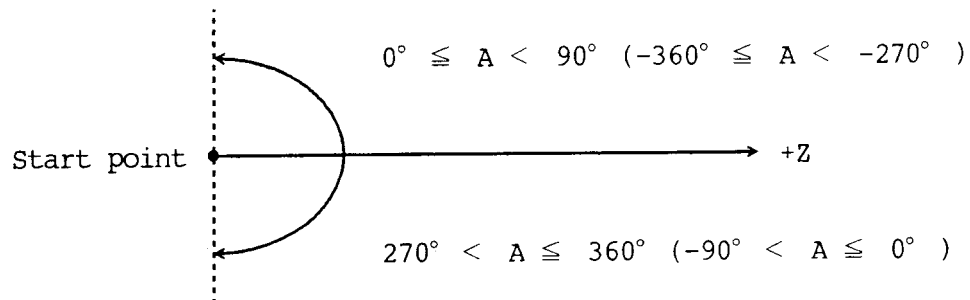
2.8.5 Associated Parameters

2.8.6 Associated Alarms

No.177	An error exists in the command of angle designate linear interpolation.
(#001)	Angle A value is out of the range.
(#002)	An angle in opposite shift direction has been commanded.

2.9 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.

2.9.1 Command Format

G31 X___ Y___ Z___ .... F___ ;

2.9.2 Sample Program

N1 G98 G31 W50. F100 ;
N2 G01 U50. W25. ;
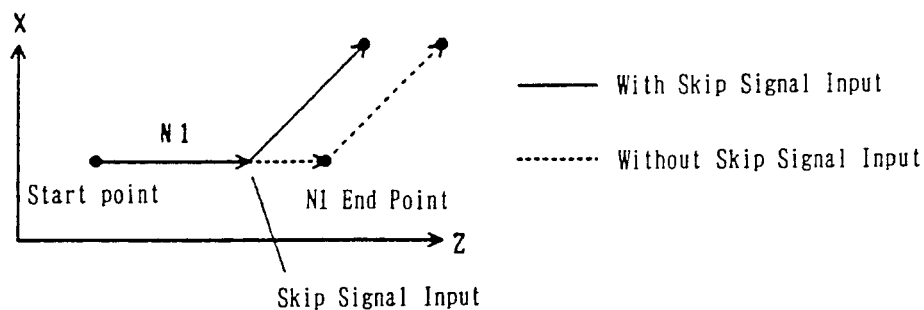
2.9.3 Cautions

(1) G31 cannot be specified while in the corner R compensation/tool system compensation mode.
(2) When the block next to the G31 command has been given by incremental programming, the next block functions in the incremental manner from the positioninterrupted by the skip signal.
(3) The G31 command is effective only in the block where it is specified.
(4) Position where skip has been input (work coordinates) can be read by the macro system variable.

2.9.4 Associated Parameters

No.6200, #7 = 0   Automatic acceleration/deceleration is made invalid to G31 skip command.
            = 1   Automatic acceleration/deceleration is made valid to G31 skip command.

2.9.5 Associated Alarms

3. THREAD CUTTING

3.1 Thread Cutting (G32)

With a G32 command, straight threads and tapered threads can be cut ata equal leads synchronously with the pulses from the spindle encoder.

3.1.1 Command Format

(1) Straight thread cutting at equal leads

G32 α___ F___ (E___) ;

where; α : Any one axis

F (E) : Thread lead

(2) Tapered thread cutting at equal leads

G32 α___ β___ F___ (E___) ;

where; α, β : Any one axis

F (E) : Thread lead

(3) Thread cutting at equal leads by specifying the number of threads per inch

G32 α___ β___ E___ ;

where; α, β : Any one axis

E : No. of threads per inch

(Note) Use a parameter to set whether to specify the thread lead with an E code or the number of threads per inch.

3.1.2 Sample Program

(1) Straight thread cutting at equal leads

G32 W-10. F2.0 ;

(2) Tapered thread cutting at equal leads

G32 U-10. W-20. F2.0 ;
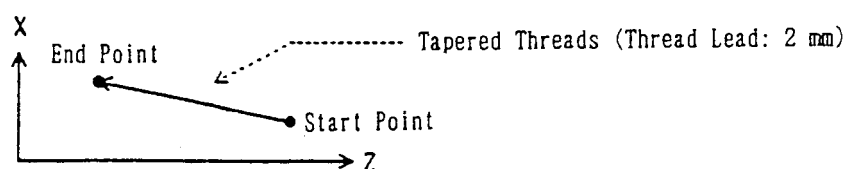
(3) Straight thread cutting at equal leads by specifying the number of threads per inch

G32 W-50. E5.0 ;

(5 threads per inch; Thread lead = 5.08 mm)

3.1.3 The Range of Thread Lead will be as Follows

(a) Metric programming

F35 : 0.00001 ~ 999.99999 (mm/rev)

F26 : 0.000001 ~ 99.999999(mm/rev)

(b) Inch programming

F26 : 0.000001 ~ 99.999999 (inch/rev)

F17 : 0.0000001 ~ 9.9999999 (inch/rev)

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

3.1.4 Cautions

(a) The feed hold function is disabled during thread cutting. If the FEED HOLD button is pressed during thread cutting, the program stops at the end point of the block next to the non-thread cutting block.

(b) Cutting feed override and spindle override is disabled and fixed at 100 % during thread cutting.

(c) An alarm results if 3 axes or more are specified in the same block.

(d) Give the thread lead (F or E) in the first G32 mode.

(e) If E is given during the G32 mode, it remains effective until F is given. Also, if F and E are given in the same block, F is given priority.

3.1.5 Associated Parameters

No.1401, #5 = 0   Dry run disabled during thread cutting and tapping.

         = 1   Dry run enabled during thread cutting and tapping.

No.3401, #4 = 0   F/E code for thread cutting in inch input is F26.

         = 1   F/E code for thread cutting in inch input is F17.

No.3401, #5 = 0   F/E code for thread cutting in mm input is F35.

         = 1   F/E code for thread cutting in mm input is F26.

No.3403, #7 = 0   E code for thread cutting is equal to the number of threads per inch of the thread lead.

3.1.6　Associated Alarms

No.103　　An error exists in the thread cutting command.

(#001)　　Commands have been given for 3 or more axes.

3.2　Continuous Thread Cutting (G32)

Continuous thread cutting is enabled by continuously specifying the thread cutting command blocks.

3.2.1　Sample Program

N1　G32　U-10.0　W-20.0　F3.0 ;

N2　W-10.0 ;

N3　U10.0　W20.0 ;
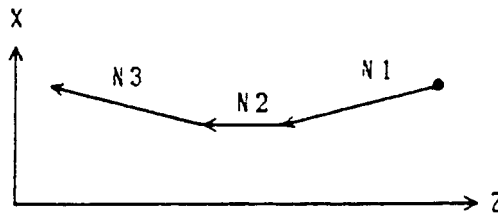
3.2.2　Cautions

(a)　Stop at single block is not possible during thread cutting.

3.3　Multi-thread Cutting

If you specify Q together with the thread cutting command (G32, G34, G76, G92), you can shift the thread cutting start angle by the specified shift Q.

If you execute thread cutting of the same shape after changing the Q value, you can execute multi-thread cutting.

3.3.1　Command Format

G32 α___ β___ Q___ F___ ;

Where, alpha, beta: Any one axis

F : Thread lead in the direction of the longitudinal axis

Q : Thread cutting start shift angle (0 to 360 degrees)

Other thread cutting commands are similar to the one described above.

3.3.2　Example of Program

G32 W-30.0 F2.0 Q180 ;

Shift by 180 degrees with respect to the reference point.

3.3.3　Precautions

(1) Either 1 degree or 0.001 can be selected as the unit for shift angle Q, by parameter setting.

(2) If you have selected 1 degree as the unit for Q, specify Q without the decimal point.
- Q1 → Indicates 1 degree
- Q1.0 → Indicates 1000 degrees

(3) If you have selected 0.001 as the unit for Q, specify Q with the decimal point.
- Q1 → Indicates 0.001 degrees
- Q1.0 → Indicates 1.0 degree

(4) When you use a macro variable in Q, select 0.001 degree as the unit for Q. Example: Q#510

(5) Even if you select 0.001 in the unit for Q, since the feedback pulse from the spindle encoder is normally 4096 pulse / rev. , the actual resolution will become 360 degrees/ 4096 = 0.088 degree.

(6) The Q specification for continuous thread cutting will be effective only in the thread cutting start block.

(7) If the Q specification exceeds the 0 to 360 degree range, alarm will occur.

3.3.4　Related Parameters

No.3404, #2 = 0　　Unit for the shift angle Q is 1 degree.

　　　　　= 1　　Unit for the shift angle Q is 0.001 degree.

3.3.5　Related Alarm

No.103　　An error exists in the thread cutting command.

(#001)　　Commands have been given for 3 or more axes.

(#002)　　Q command value has exceeded the set range.

3.4 Variable Lead Thread Cutting (G34)

Variable lead threads can be cut by specifying an incremental or decremental amount per revolution of thread in the G34 command block.

3.4.1 Command Format

G34 α___ β___ F___ K___ ;

where; α, β : Any one axis

F : Thread lead in the longitudinal direction at thread cutting start time

K : Incremental amount (decremental amount for a nagative value) per revolution

The programmable range of K is as follows.

Metric system : ±0.001 to ±99999.999 (mm/rev)

Inch system : ±0.0001 to ±9999.9999 (inch/rev)

3.4.2 Sample Program

Straight thread cutting at variable leads

(Thread lead at start time: 2.0 mm, Incremental amount per rev. : 0.5mm)

G34 W-10.0 F2.0 K0.5 ;
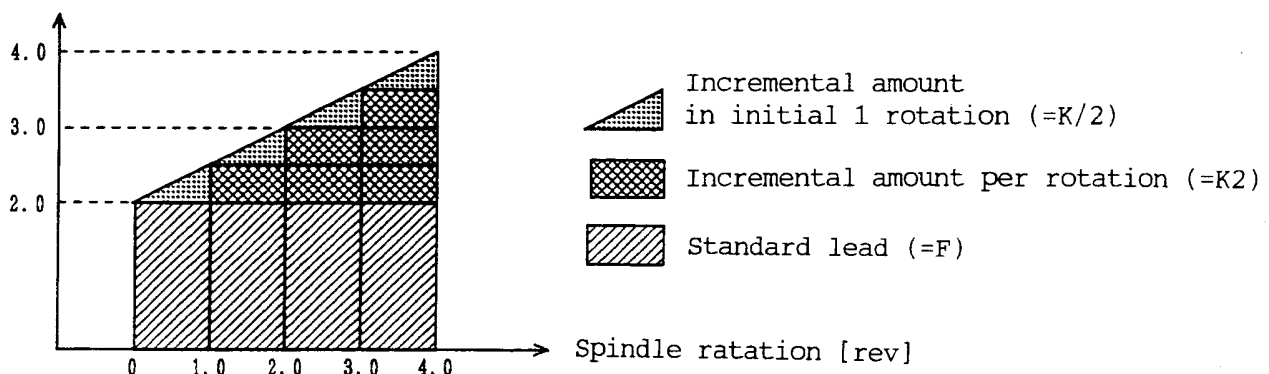
In variable lead thread cutting, feed speed is changed constantly as above so that lead is changed for every rotation.

The lead for 1st rotation in thread cutting becomes "F± (K/2)" and the lead for 2nd rotation on is increased/decreased by K for every rotation.

3.4.3 Cautions

(1) A G34 command with K omitted results in thread cutting at equal leads (G32 command).

4.1 Feed Per Minute (G98)

Until G99 is specified after G98, specify the stroke per minute (mm/min., inch/min.) with a numerical value following F as to the cutting feed rate.
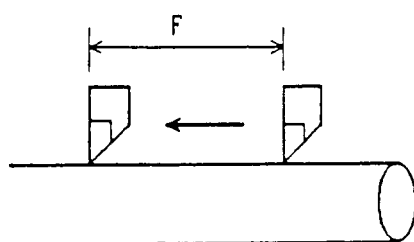
Stroke per Minute

(mm/min, inch/min)

4.1.1 Command Format

G98 ;

4.1.2 Sample Program

G98 F100 ; The cutting feed rate is 100 mm per minute.

4.1.3 The F Code Programmable Range is as Follows

(1) Metric system
F60 : 1~ 999999 (mm/min)
F61 : 0.1~ 999999.9 (mm/min)
(2) Inch system
F51 : 0.1~ 99999.9 (inch/min)
F52 : 0.01~ 99999.99 (inch/min)

4.1.4 Cautions

(1) One of F60, F61, F51, and F52 is selected by. parameter setting.
(2) When F61 is specified without a decimal point in the metric system, the least input increment of 1 mm/min. or 0.1 mm/min. can be selected by parameter setting.
(3) In case of thread cutting (G33), feed per revolution is assumed temporarily even during the G98 mode.

4.1.5 Associated Parameters

No.3400, #1 = 0   The unit without a decimal point for metric input F61 is 1mm/min.
         = 1   The unit without a decimal point for metric input F61 is 0.1mm/min.
No.3401, #2 = 0   Feed per minute for inch input is F51.
         = 1   Feed per minute for inch input is F52.
No.3401, #3 = 0   Feed per minute for metric input is F60.
         = 1   Feed per minute for metric input is F61.

4.1.6 Associated Alarms

No.102   F has not been specified in cutting feed. Or, F0 has been specified.

4.2 Feed Per Revolution (G99)

Until G98 is specified after G99, specify the stroke per minute (mm/rev., inch/rev.) with a numerical value following F as to the cutting feed rate.
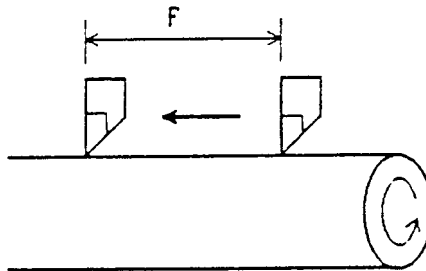
4.2.1 Command Format

G99 ;

4.2.2 Sample Program

G99 F12 ; The cutting feed rate is 1.2 mm per revolution.

4.2.3 The F Code Programmable Range is as Follows

(1) Metric system

F32 : 0.01 ~ 999.99 (mm/rev)

F33 : 0.001 ~ 999.999 (mm/rev)

(2) Inch system

F23 : 0.001 ~ 99.999 (inch/rev)

F24 : 0.0001 ~ 99.9999 (inch/rev)

4.2.4 Cautions

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

4.2.5 Associated Parameters

No.3401, #0 = 0   Feed per rotation for inch input is F23.

            = 1   Feed per rotation for inch input is F24.

No.3401, #1 = 0   Feed per rotation for metric input is F32.

            = 1   Feed per rotation for metric input is F33.

4.2.6 Associated Alarms

No.102   F has not been specified in cutting feed. Or, F0 has been specified.

4.3 Dwell (G04)

A G04 command can delay migration of operation to the next block.

When specifying by time: The command causes the machine to wait for the specified time.

When specifying by revolutions: The command causes the machine to wait while the spindle rotates the number of revolutions specified.

4.3.1 Command Format

(1) When specifying by time

  G98 G04 P___ ;

  P: Dwell time (0.001 to 99999.999 sec.)

(2) When specifying by revolutions

  G99 G04 P___ ;

  P: Dwell revolutions (0.001 to 99999.999 rev.)

(Note) X or U can be used instead of P.

4.3.2 Sample Program

  G98 G04 P2000 ;   Dwell time 2 seconds

  G04 X2.0 ;   Dwell time 2 seconds

4.3.3 Cautions

(1) It is normally set to dwell specified by time (per-minute dwell).
  Through parameter setting, however, dwell specified by rotation (per-rotation dwell) can be set in the per-rotation feed mode.

4.3.4 Associated Parameters

No.3400, #5 = 0   G04 always specfies by time

           1   G04 follows G94 or G95

4.4 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, executes the next block.

4.4.1 Command Format

G09 .... ;

4.4.2 Sample Program

N1 G09 G01 W50.0 F2.0 ;
N2 G01 U-50.0 ;

The corner has an edge when G09 is specified

The corner is rounded when G09 is not specified
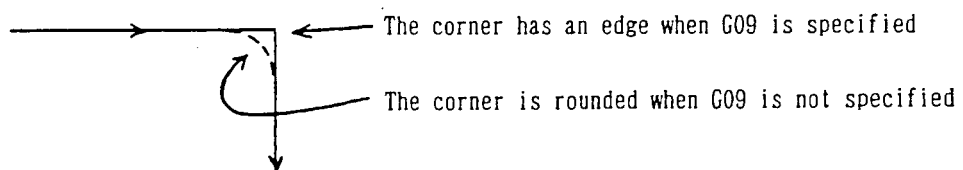
4.4.3 Associated Parameters

No.1801, #4 = 0   Imposition width for cutting feed is Parameter No.1826 (Common to rapid traverse)

= 1   Imposition width for cutting feed is Parameter No.1827 (Common to rapid traverse)

No.1826   Imposition width of each axis

No.1827   Imposition width of each axis for cutting feed

4.5 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.

4.5.1 Command Format

G61 ;

4.5.2 Sample Program

N1 G61 G01 W-100. F1.0 ;
N2 U50. ;
N3 W-100. ;

The corner has an edge when the G61 mode is effective.
The corner is rounded when the G61 is ineffective.
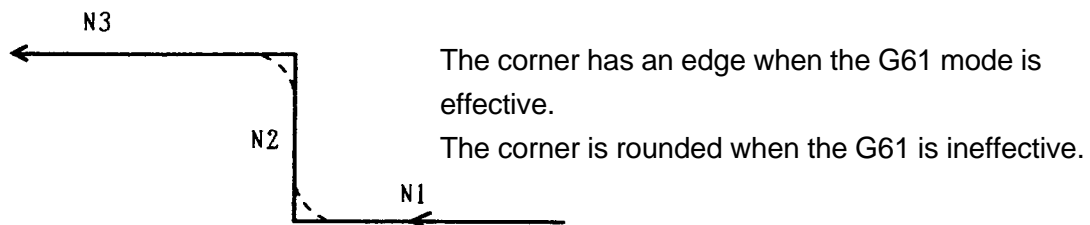
4.5.3 Switching Into M Code

Through setting a parameter, you can specify Exact Stop mode by using M21 command instead of using G61 command.

M21, as it is replaced by G61 inside NC for execution, is not output in Machine side. Also, with M21 command, G62 to G64 modes are cancelled.

4.5.4 Associated Parameters

No.1801, #4 = 0   Imposition width for cutting feed is Parameter No.1826 (Common to rapid traverse)

        = 1   Imposition width for cutting feed is Parameter No.1827 (Common to rapid traverse)

No.3409, #1 ON/OFF command system for Exact Stop mode is:

        = 0   G61 (ON) / G64 (OFF)

        = 1   G61 or M21 (ON) / G64 or M22 (OFF)

No.1826   Imposition width of each axis

No.1827   Imposition width of each axis for cutting feed

4.6   Automatic Corner Override Mode (062).... Not available with Σ21L

When cutter 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 ate to reduce the cutting load.
Once G62 is specified, it remains effective until G61, G63, or G64 is specified.

4.6.1   Command Format

G62 ;

4.6.2   Override Conditions

When the following conditions are met with respect to the blocks which are located before and after the corner in the cutter compensation mode, an override is applied to cutting feed automatically.

(a) When the G code of Group 01 is G1, G02, or G03.
(b) When an offset amount is not 0 in the cutter compensation offset mode.
(c) When an offset is inside at 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
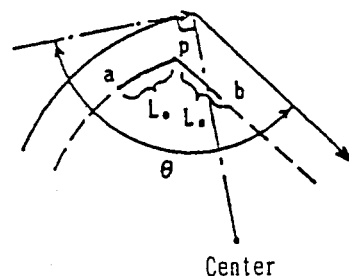
where; θ : Inner criterion angle
Le : Start distance
Ls : End distance
An override is applied to cutting feed from the point a through the point b.

4.6.3   Override Amount

$1 \leqq$ Automatic corner override $\leqq 100$ (%)
An actual cutting feed rate is as follows.

F $\times$ Automatic corner override $\times$ Cutting feed rate override

4.6.4 Sample Program

N1 G62 G42 G00 U40. V50. ;
N2 G01 U100. F200 ;
N3 G03 U60. V-30. R30. ;
N4 G64 G40 G00 U40. ;

An override is applied
from the point a through
the point b.
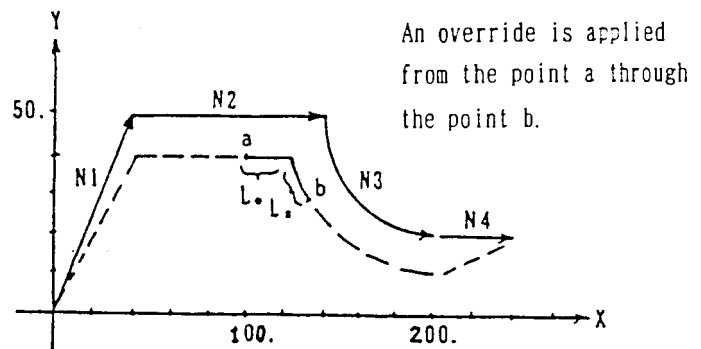
4.6.5 Cautions

(1) When a block not shifting has been specified between two blocks subject to automatic corner override, tool diameter compensation is supplied while automatic corner override is not.

4.6.6 Associated Parameters

No.1711   Inner criterion angle of automatic corner override (θ : deg.)
No.1712   Override of automatic corner override (%)
No.1713   Start distance (Le) of automatic corner override
No.1714   End distance (Ls) of automatic corner override

4.7 Tapping Mode (G63)

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

(1) Cutting feed rate override fixed at 100 %
(2) Spindle override fixed at 100 %
(3) Single block disabled
(4) Dry run disabled
(5) Feed hold disabled
(6) Decelerated stop disabled at the joint of the blocks (imposition check disabled)
(7) Tapping mode signal output

The G63 mode is effective until G61, G62, or G64 is specified.

4.7.1 Command Format

        G63 ;

4.7.2 Sample Program

        N1 G63 G01 X10. F200 ;
        N2 Z0 ;
        N3 X20. ;
        N4 Z-5. ;   Tapping mode effective blocks.
        N5 X30. ;
        N6 Z-10. ;
        N7 G64 X40. ;

4.7.3 Cautions

(1) By parameter setting, you can invalidate dry run disabled.

4.7.4 Associated Parameters

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

4.8 Cutting Mode (G64)

Until G61, G62, or G63 is specified after G64 was specified, the next block is executed continuously without decelerating to a stop between the blocks.
The G64 mode is effective until G61, G62, or G63 is specified.
When cutting is performed in the G64 mode, the corner may be rounded at the time of cutting feed.
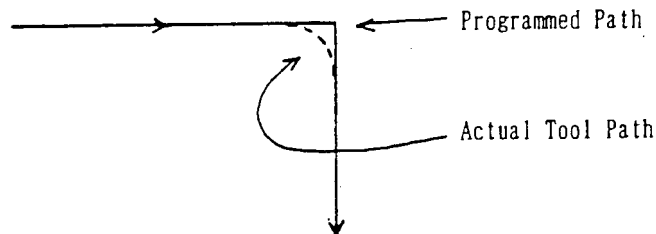
4.8.1 Command Format

4.8.2 Sample Program

```
N1 G64 G01 X10. F200 ;
N2 Z0 ;
N3 X20. ;
N4 Z-5. ;          Cutting mode effective blocks
N5 X30. ;
N6 Z-10. ;
N7 G61 X40. ;
```

4.8.3 Switching Into M Code

Through setting a parameter, you can specify Cutting mode by using M22 command instead of using G64 command.
M22, as it is replaced by G64 inside NC for execution, is not output in Machine side. Also, with M22 command, G61 to G63 modes are cancelled.

4.8.4 Associated Parameters

No.3409, #1 ON/OFF command system for Exact Stop mode is:
  = 0   G61 (ON) / G64 (OFF)
  = 1   G61 or M21 (ON) / G64 or M22 (OFF)

4.8.5 Cautions

(1) G64 is set in the reset state.
(2) A G00 command decelerates the machine to a stop regardless of the G64 mode.

4.9 Multibuffer (G251)

During automatic operation, the number of blocks to be preread is as follows.

(1) During the automatic tool nose radius compensation enable mode (G143) → 4 blocks

(2) During cutter compensation (G41/G42 in the G145 mode) → Max. 4 blocks

(3) Others → 1 block

A multibuffer command prereads up to 12 blocks.

This function makes it possible to reduce a stop time between the blocks when executing the program having very small continuous blocks.

4.9.1   Command Format

G251 P0 ; or G251 ;   Multibuffer mode ON

G251 P1 ;   Multibuffer mode ON

4.9.2   Sample Program

G251 ;   Multibuffer mode ON

                                Prereads up to 12 blocks in this portion.

G251 P1 ;   Multibuffer mode OFF

4.9.3   Cautions

(1) The macro statement is not included in the number of preread blocks.

(2) Specify the G251 command in an independent block.

(3) When the very small blocks are executed continuously, there will be no preread blocks, making an execution time shorter than the processing time of one block.

(4) The blocks may not be preread even in the multibuffer mode ON state, depending on the commands.

    (Example) G10, G20, G21, G22, G23

              G28, G30, G31, G53

              M00, M01, M02, M30

              Coordinate system setting by a T command

(5) The reset state can be turned to the multibuffer mode ON state by parameter setting.

4.9.4   Associated Parameters

No.3402, #6 = 0   Multibuffer mode OFF at reset time

          = 1   Multibuffer mode ON at reset time

4.9.5   Associated Alarms

No.130 There is an erroneous command in the block.

    (#001) G251 is not an independent command.

4.10 Acceleration/Deceleration Control

4.10.1 Automatic Acceleration/Deceleration

Acceleration/deceleration are automatically applied at the start and the end of shifting so that the machine system is protected from shock.
Acceleration/deceleration include the following types:

[Linear type acceleration/deceleration]

[Exponential type acceleration/deceleration]

[Bell type acceleration/deceleration]

- - - - - Operation in interpolation

───── Operation following acceleration/ deceleration
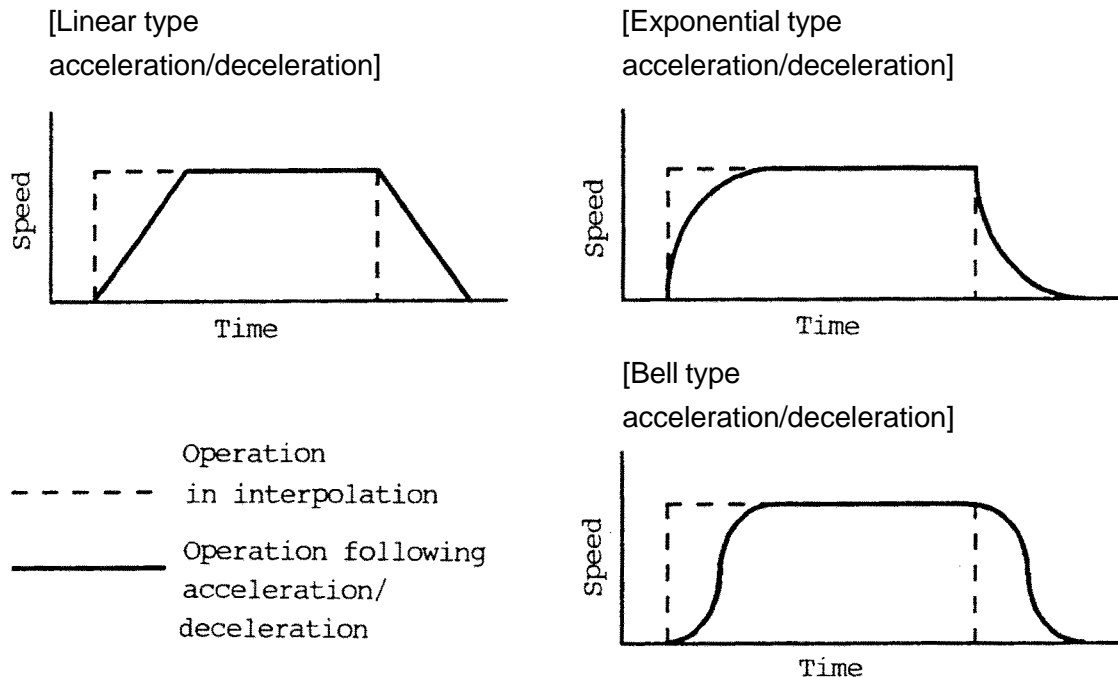
(1) Acceleration/deceleration types for standard setting are as follows:
   1) Rapid traverse .... Linear acceleration/deceleration
   2) Cutting feed (G01, G02, G03, G32) .... Exponential type acceleration/ deceleration
   3) Manual continuous feed (JOG feed) .... Exponential type acceleration/ deceleration

4.10.2 Other Acceleration/Deceleration (Optional)

(1) Rapid Traverse Bell Type Acceleration/Deceleration
   Rapid traverse bell type acceleration/deceleration, through application of smooth acceleration/deceleration to rapid traverse speed, can reduce mechanical shock due to change in acceleration.
   Compared with linear acceleration/deceleration, this enables setting of smaller time constant, thus shortening time required for acceleration/deceleration.

(2) Linear Acceleration/Deceleration Following Cutting Feed Interpolation
   Compared with the exponential type acceleration/deceleration, the linear acceleration/deceleration following cutting feed interpolation reduces delay to a command to 1/2, when given the same time constant, thus reducing time required for acceleration/deceleration.

Further, in the arc interpolation especially in high speed cutting, the actual tool path subject to acceleration/deceleration has an error to the arc in its radius direction. This error,. also, is reduced compared with that in the exponential type acceleration/decelation.

(3) Bell Type Acceleration/Deceleration Following Cutting Feed Interpolation
Bell type acceleration/deceleration following cutting feed interpolation is a function to reduce shock given to the mechanical system by smoothing acceleration and deceleration.

4.10.3 Associated Parameters

No.1610, #0/#1   Acceleration/Deceleration Available for Cutting Feed
                      (Including feed by dry run) :

Parameter No.1610		Acceleration/Deceleration
#1	#0	
0	0	Exponential type acceleration/deceleration
0	1	Linear acceleration/deceleration following interpolation
1	0	Bell type acceleration/deceleration following interpolation

No.1610, #4 = 0   Acceleration/deceleration for manual continuous feed (JOG feed) is by exponential type acceleration/deceleration.

　　　　　 = 1   Acceleration/deceleration for manual continuous feed (JOG feed) is by the same way as that of cutting feed.

No.6200, #7 = 0   Automatic acceleration/deceleration of skip function (G31) is made invalid.

　　　　　 = 1   Automatic acceleration/deceleration of skip function (G31) is made valid.

No.1620   Time constant of linear acceleration/deceleration of rapid traverse for each axis or time constant of rapid traverse bell type acceleration/deceleration T1.

No.1621   Time constant of linear acceleration/deceleration of rapid traverse for each axis T2.

No.1622   Time constant of cutting feed exponential type acceleration/deceleration for each axis, time constant of bell type acceleration/deceleration following cutting feed interpolation, or time constant of linear acceleration/deceleration following cutting feed interpolation.

No.1623   FL speed of exponential type acceleration/deceleration of cutting feed for each axis.

No.1624   Time constant of exponential type acceleration/deceleration of manual continuous feed for each axis, time constant of bell type acceleration/ deceleration following interpolation, or time constant of linear acceleration/ deceleration following interpolation.

No.1625   FL speed of exponential type acceleration/deceleration of manual continuous feed for each axis.

No.1626   Time constant of exponential type acceleration/deceleration of thread cutting cycle for each axis.

No.1627   FL speed of exponential type acceleration/deceleration of thread cutting cycle for each axis.

5. SPEED CONTROL

5.1 Feed Speed Command (F Data)

5.1.1 Modal In Per-Minute Feed and Per-rotation Feed

Feed rate "F", once commanded, remains valid (modal) until the next command is given. However, as "F" role is totally different between per-minute feed and per-rotation feed, it is hardly of any use even if "F" is held modal in per-minute feed and per-rotation feed. Oppositely, it can even cause danger.

For example, it can be dangerous if, after "F" having been commanded in per-minute feed, axis shift is performed in per-rotation feed without commanding "F".

   N1 G98 G01 Z__ F2000 ; Speed is 2000mm/min

   N2 G99 Z__ ;   Speed is 20.00mm/rev

Modal "F", therefore should be treated as follows:

> Per-minute feed and per-rotation feed (incl. thread cutting) are held modal independently.

In other words, NC holds two "F" data including ones for per-minute feed and the other for per-rotation feed. Depending on the feed mode for execution, the "F" to be used is determined.

They are explained in details as follows:

The number of effective digits below a decimal point for "F" are 0 for per-minute feed/ 2 for per-rotation feed/ 5 for thread cutting. And, "F" being made valid in that block is expressed in [ ].

	(Per-Minute Feed)	(Per-Rotation Feed)
On Supply of Power	0	0.00
N1 G99 F1.23 ;	0	[1.23]
N2 ——— ;	0	[1.23]
N3 G98 F1000 ;	[1000]	1.23
N4 ——— ;	[1000]	1.23
N5 G32 F2.34567 ;	1000	[2.34567]
N6 ——— ;	1000	[2.34567]
N7 G99 ;	1000	[2.34]
N8 ——— ;	1000	[2.34]
N9 G98 ;	[1000]	2.34
N10 —— ;	[1000]	2.34
N11 G32 ;	1000	[2.34567]

Also, when the feed per minute F is specified, the feed per revolution F can be made 0 by parameter setting (No. 3401, #7=1) ; conversely, when the feed per revolution F is specified, the feed per minute F can be made 0.

In this case, for the previous example, the feed per revolution F = 0 in N3 and the feed per minute F = 0 in N5, therefore, the effective F in N9, N10, N11 will become 0.

When the inch input/mm input is changed by using the G code (G20/G21), inch/mm conversion will be performed for modal data "F" also.

$$1000mm/min \longleftrightarrow 39.3inch/min$$

5.1.2   F Command for Rotating Axis by Inch Input

The shift (deg) of rotating axis (C) is converted to inch and used.

[Example 1] G98 G01 U10.0 W20.0 H30.0 F20.0 ;

30 (deg) will be treated as 30 (inch).

Interpolation, distance (L) is:

$$L = \sqrt{(10/2)^2 + 20^2 + 30^2} = 36.401 \text{ (inch)}$$

Interpolation distance (T) is:

$$T = L/F = 36.401/20 = 1.82 \text{(min)}$$

Speed (Fc) of axis C becomes:

$$Fc = 30/T = 16.483 \text{ (deg/min)}$$

[Example 2] G98 G01 H360.0 F180.0 ;

$$L = 360 \text{ (inch)}$$
$$T = 360/180 = 2 \text{ (min)}$$
$$Fc = 360/T = 180 \text{ (deg/min)}$$

5.2 Changing of Inner Circular Cutting Speed

.... Not available with Σ21L

For circular cutting which is offset internally while in the tool diameter compensation mode, the override equal to the ratio of a radius of the tool diameter center path to a radius of the program path is applied to the specified cutting feed speed.

Cutting rate of the tool diameter center path is as follows:

$F \times Rc/Rp$ where   Rc : Radius of tool center path

Rp : Radius of program path
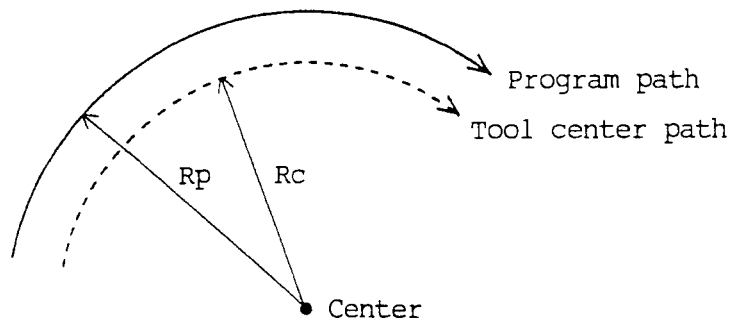
5.2.1 Cautions

(1) Inner circular cutting speed change is part of the automatic corner override function. Therefore, this function is invalid unless the automatic corner override function is added.

(2) Through parameter setting, changing of inner circular cutting speed is made valid in G62 mode.

(3) When Rc/Rp value is smaller than the parameter set value, use the minimum override of the parameter.

(4) When automatic corner override is added to inner circular cutting, the cutting feed speed gets as follows:

$F \times (Rc/Rp) \times$ Auto corner override $\times$ Cutting feed rate override

5.2.2 Associated Parameters

No.3406, #0 = 0   Change of inner circular cutting speed is held always valid.

= 1   Change of inner circular cutting is speed valid in G62 mode.

No.1710   Minimum override of inner circular cutting speed

5.3 Scroll Cutting Speed Control (G128)

Scroll (vobute) shape can be cut by performing straight line interpolation of rotating axis and the straight line axis that moves in the direction of the diameter.

However, in the straight line interpolation (G01) of the straight line axis and rotating axis, the cutting speed normally does not become constant. This is because the interpolation sped is constant, regardless of the position (radius of circle) of the straight line axis.This function controls the interpolation speed of the straight line axis and the rotating axis based on the position of the straight line axis so that the cutting speed becomes equal to the command speed at that position.

5.3.1 Command Format

G128 G98 G01 α___ β___ F___ (E___) (Q___) (p___) ;

G128	:	One-shot G code that indicates that this function is performed
G98	:	Feed per minute
G01	:	Straight line interpolation
α	:	End point position of straight line axis (mm, inch)
β	:	End point position of rotating axis (deg)
F	:	(First) cutting speed (mm/min, inch/min)
(E)	:	Second cutting speed (mm/min, inch/min)
(Q)	:	Speed changeover remaining angle (degree)
(P)	:	Inside cutting / outside cutting specification (1: Outside, 2: Inside)
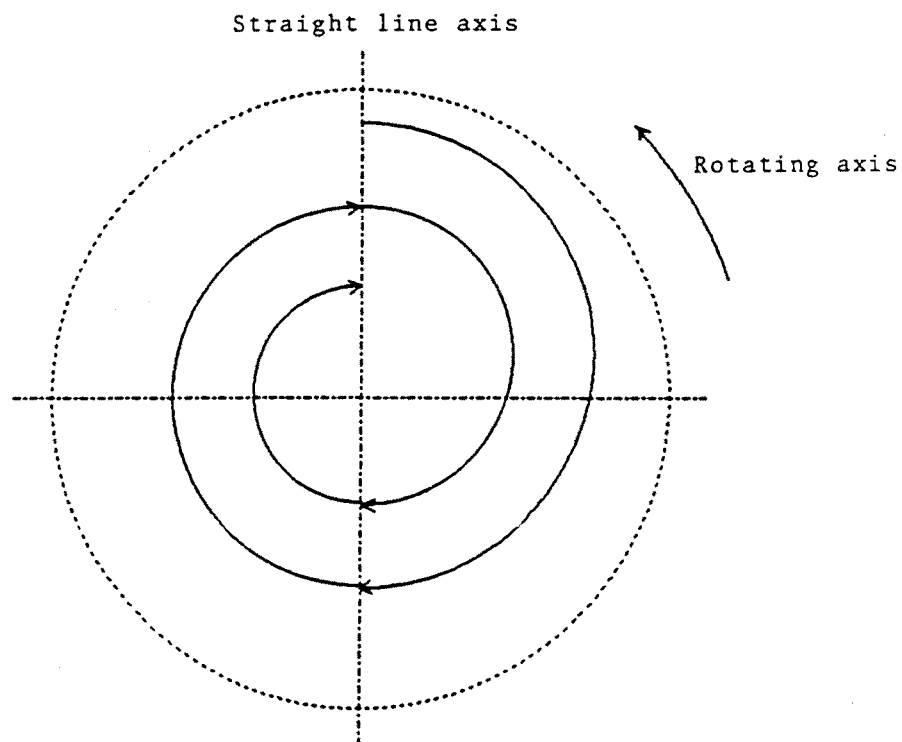
5.3.2 Control Axis

It is necessary to set in advance, in the parameters (linear axis: No.3418, rotating axis: No.3419), the straight line axis and the rotating axis which are subject to scroll cutting speed control.

Where, however, No.3418 ≠ No.3419. These parameters will be used for polar coordinate interpolation (G121) also.

5.3.3 Speed Control Method

The interpolation speed is changed according to the change in the cutting radius, so that the cutting speed at that position becomes the command speed.

Since a work coordinate value of a straight line axis is employed for the cutting radius (1/2 when the diameter is specified), the origin of the work coordinate system must match the center of the work rotation.

Since the work coordinate value is not showing the cutting position but the tool center, however, the speed will not be controlled for the cutting position but the tool center if the work coordinate is selected as the cutting radius. (The acutual cutting position is equal to the work coordinate position ± the tool radius.)

When the address P is specified, the cutting radius can be the corrected value of the absolute work coordinate with the tool radius (addition or subtraction).

P=1 : Outer cutting correction (subtracting the tool radius)

P=2 : Inner cutting correction (adding the tool radius)

Value of P other than above or P omitted: No compensation

(Note 1) *Specify address P without the decimal point.*

(Note 2) *Set the tool compensation amount "R (shape), Q (wear)" in the tool diameter.*
The tool diameter becomes R + Q.
Set 9 in the virtual tip point (T). Compensation will not be performed if the value is other than 9.


```
                                    Cutting surface (P2)
              R0                    Programmed path
                                    Cutting surface (P1)
     R1              R2
                                    R1 = R0 - tool diameter
                                    R2 = R0 + tool diameter


              Work center
```
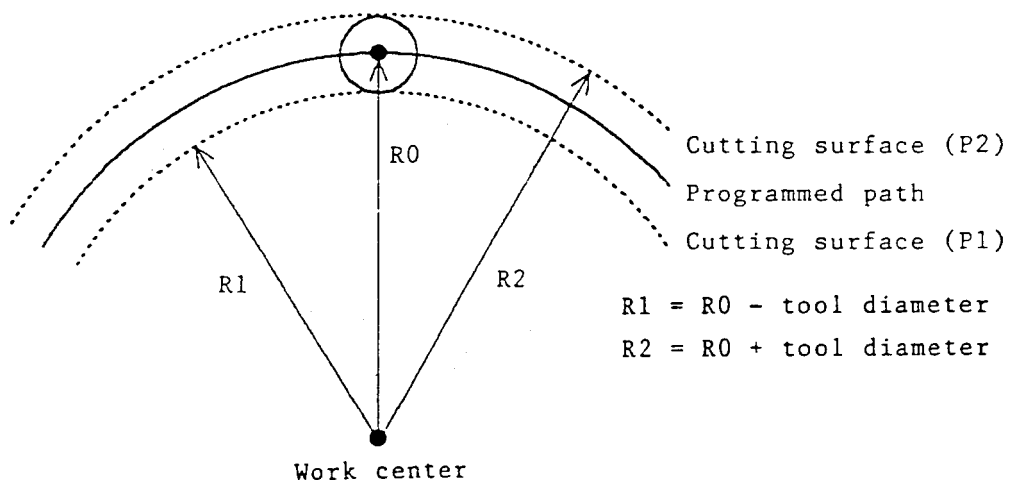
5.3.4 Speed Changeover Function

By specifying address E (second cutting speed) and address Q (speed changeover remaining angle) in the same block as G128, the cutting speed can be changed during execution of that block. When the remaining shift of the rotating axis becomes less than Q, the cutting speed will be changed from F to E. Both the remaining shift of the rotating axis and the Q value will be treated as absolute values (no symbols).

This function will be effective when both E and Q are specified and whez Q is not equal to 0. If this condition is not satisfied, the cutting speed will become the normal first cutting speed F.

Q at this stage indicates the angle, therefore, the position of the decimal point of the data is the same as the position of the decimal point of the address.
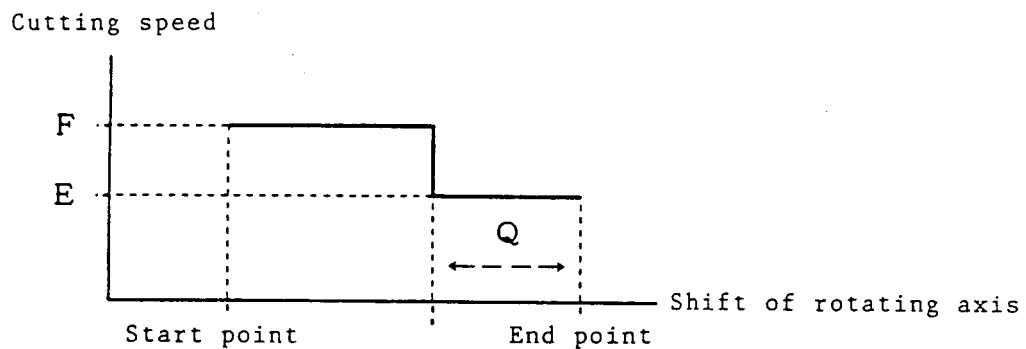
5.3.5 Program Example (Straight Line Axis: X Axis, Rotating Axis: C Axis)

```
N1  T     00 ;
N2  M13 ;                          Rotating tool forward rotation
N3  G00  X150.0  Z10.0  C0 ;
N4  X85.0 ;
N5  Z-10.0 ;
N6  G98  G01  X90.0  F2000 ;       Approach the start position
N7  G128  X30.0  C-720.0 ;
    F800  W600  Q90.0  P2 ;        Scroll cutting (inside)
```

Tool diameter: 5 mm   · · · Tool center path (800 mm/min)
———— Gutting point path   − − − Tool center path (600 mm/min)
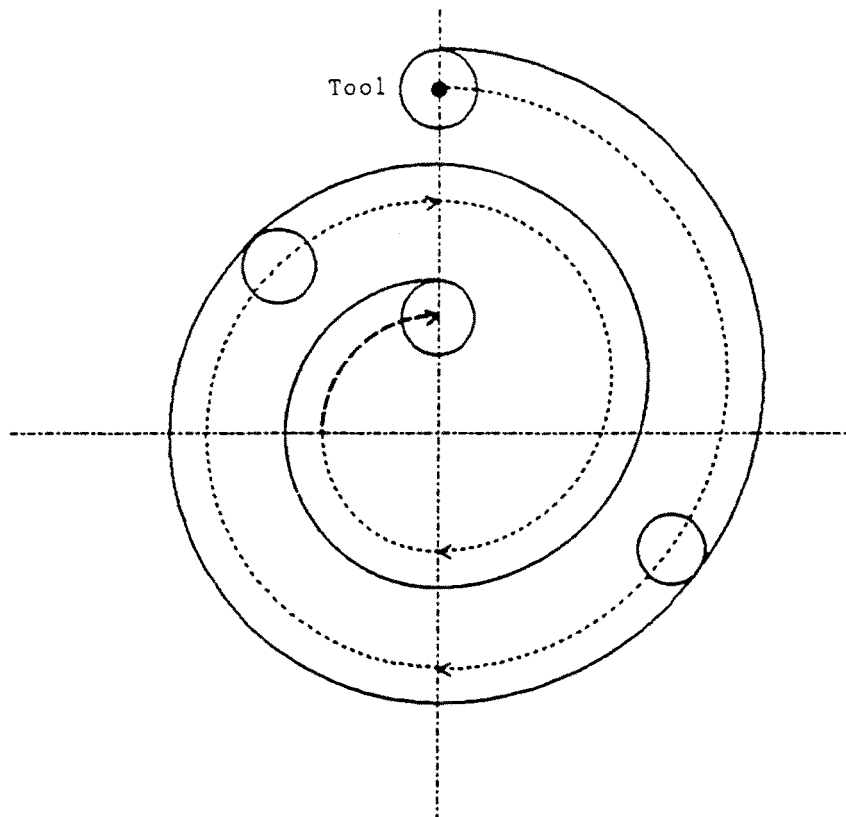
5.3.6 Precautions

(1) G128 is a one-shot G code and is effective only in the specified block.

(2) The G128 function can be executed only during straight line interpolation (G01) in feed per minute (G98).

(3) The axes that can be controlled are two in number - the straight line axis and the rotating axis set in the parameters.
Make sure you specify the two axes in the G128 block and do not specify any other axis. If this condition is not satisfied, alarm will be output.

(4) When the shift of the straight line axis or the rotating axis is 0, the G128 function becomes ineffective, similar to the condition when G128 is not specified. (No alarm will be output.)

(5) Alarm will be output if there is an error in the parameter setting.
• When Parameter No.3418 is equal to No.3419.

(6) Feed override is effective. The specified speed to which override is applied, will become the cutting speed.

(7) Dry run is effective but the dry run speed will become the compensated speed, and not the cutting speed. (Same as the condition when G128 is not specified.)

(8) The work coordinate of the straight line axis will be used in the radius value at the cutting position. Therefore, set the work coordinate system so that the origin of the work coordinate system is at the center of the work rotation.

(9) The following restrictions will be applied to the coordinate values of the start and end points of a straght line axis.
• Neither of the specified coordinate values of the start and end points should be zero.
• The signs of the spcified coordinate values of the start and end points should be equal. (When the start point is positive, the end point should be also positive. When the start point is negative. the end point should be also negative.)
• Both of the cutting radiuses of the start and end points should not be zero or negative.
Unless these conditions are met, the alarm will be given.

5.3.7 Related Parameters

No.3418   Axis number of straight line axis performing G128 (1 ~ no. of control axes)

No.3419   Axis number of rotating axis performing G128 (1 ~ no. of control axes)

5.3.8   Related Alarms

No.183   Error in the G128 command.

(#001)   It is not G98/G01.

(#002)   An error exists in an axial command.

(#003)   Start/end points of straight line axis are not correct.

No.189 (G128)   Error in the parameter setting.

5.4 Speed Control of Independent Axis

Offset speed F of the normal axis NC (X, Y, Z etc.) has following meanings.

[1]   Supplement of straight line (G01) : speed on diagonal line

[2]   Supplement of arc (G02 and G03) : speed on circumference

[3]   Wrenching (G32) : speed of an axis with greater amount of movement (lead of screw)

However, the normal speed control is insufficient for axes which move during the planing and are not involved in the planing, namely "the oscillation halt axis" which is used to halt oscillation generated during rotation in a long work.

If the oscillation halt axis and the planing axis are specified in the same block, the actual planing speed does not equal the specified speed F. (The more the movement amount of the oscillation halt axis increases, the more the planing speed decreases.)

Therefore, the oscillation halt axis is called "independent axis", and the other axes are called "the ancillary axes". And the following function is added.

> **"Speed control function which makes the planing speed of the ancillary axis F regardless of the movement amount of the independent axis."**

This control function is called "speed control function of the independent axis".

5.4.1   Specification of Indepandant Axis

It is possible to control, as an independent axis, only one among CNC control axes. The independent axis is assigned in Parameter No.1012, #6.

(Note) When two or more axes being assigned as independent axes, an axis with the smallest axis number is controlled as an independent axis.

5.4.2   Control Contents

(1)   Block in which speed control of the independent axis is performed

Speed control of the independent axis performed only in blocks which meet the following conditions.

- G01 (straight line supplement), G32 (wrenching), G34 (wrenching of variable leads) However, G01 in G128 (scroll planing speed control) is excepted.
- The ancillary axes and independent axes are moving in the same block.

(2) Speed control of independent axis

When speed control of the independent axis is performed, speed of each axis is controlled in order that;

- Speed of the ancillary axis equals the specified speed F (The same as when independent axis is not specified.), or
- Speed of the independent axis equals the speed of simultaneous termination of supplement with ancillary axis.

If the supplement distance of the ancillary axes is Ld, and the movement distance of the independent axes is Li, speed of the independent axis (Fi) is expressed as follows.

$$Fi = Li \times F / Ld \quad (F: \text{Command speed} = \text{Speed of ancillary axis})$$

(Notes)   Feeding override is effective. (Speed of each axis is overridden.)
In the case of dry run, speed of ancillary axis becomes the dry run speed.

5.4.3   Operation Examples to be Controlled

If axes X and Z are ancillary axes, axis A is an independent axis, and each axis makes the following commands from the work coordinates 0, speed of the independent axis is expressed as follows.

(1) G01 (straight line supplement)

G98 G01 X50.0 Z100.0 A200.0 F100 ;

• Supplement distance of ancillary axis

$$Ld = \sqrt{(50/2)^2 + 100^2} = 103.078 \text{ (mm)}$$

• Speed of independent axis

$$Fi = \frac{Li}{Ld} \times F = \frac{200 \times 100}{103.078} = 194.028 \text{ (mm/min)}$$

(*1) The X axis is processed by radius value conversion.

(*2) Feeding for each rotation is processed by radius value conversion.

(2) G32 (Wrenching)

G32 X50.0 Z100.0 A200.0 F10.0 ;

• Supplement distance of ancillary axis.

Ld = 100 (mm) → Movement amount of the X-axis (25mm) is larger than that of the Z-axis (100mm)

• Speed of independent axis

$$Fi = \frac{Li}{Ld} \times F = \frac{200 \times 100}{100} = 20.0 \text{ (mm/rev)}$$

(*1) In the case of taper wrenching, though this processing method has been traditionally used, more than two axes can not be specified.

(*2) In the case of G34 (variable lead wrenching), speed of the A-axis varies as the speed of the independent axis changes.

5.4.4   Operation Not to be Controlled

Special processing is not performed for the code G other than above. And an axis which is specified as an independent axis is treated as an ancillary axis.

Main codes G not to be controlled are described below.

(1) In the case of G00 (location setting)

Location setting of the non-straight form (standard) and the straight form is performed as before.

(2) In the case of G02 and G03 (arc supplement)

Conventional processing is applied.

• The independent axis is inside the arc surface.

→ This cannot happen, but if it is specified, usual arc is executed.

• The independent axis is outside the arc surface.

→ Helical supplement is executed. Straight line command upon the Helical supplement effective in only two axes. Therefore, there is no problem in the execution.

However, alarm is generated, if option of the Helical supplement is not attached.

(3) In the case of fixed cycle (single, complex, holing)

Usually independent axis is not specified in the fixed cycle.

If fixed cycle is specified, and is executable, it will be executed. However, operations conform to their specifications, because they are composed of G00, G01 and G32.

5.4.5   Notes

(1) If speed control of the independent axis is performed, speed of the independent axis will be proportional to the ratio of the supplement distance of the ancillary axis and the movement distance of the independent axis.

Therefore, if [supplement distance of ancillary axis] << [movement distance of independent axis] (Right side is much larger than left side.), speed of the independent axis will be excessively high. Therefore, this should be considered when creating programs.

(2) For the block in which speed control of the independent axis is performed, manual absolute becomes OFF. (Offset of the amount of manual intervention will not be performed for the movement after the manual intervention.)

6.1   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.

6.1.1   Command Format

G28 X___ Y___ Z___ ...... ;

6.1.2   Sample Program

G28 U100. W25. ;
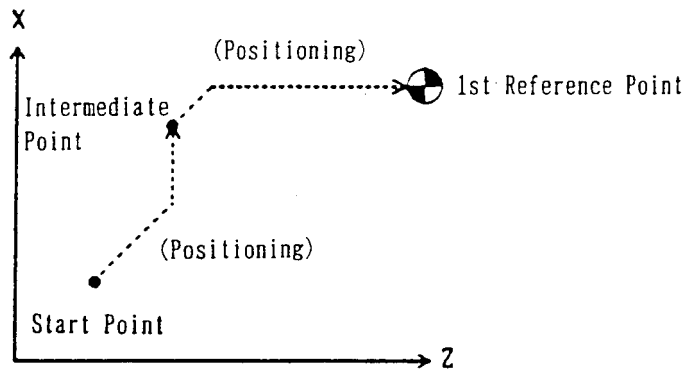
6.1.3   When G28 is Specified without Completing Manual Reference Point Return After Turning on the Power.

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 takes place before completion of the reference point returning, following returning to the deceleration dog, reference point returning is performed once again.

6.1.4   Cautions

(1) If G28 G53 X___ Y___ Z___ ... ; is specified, the axes are positioned to the specified position of the machine coordinate system, and then, they return to the reference point.

(2) The G28 command confirms imposition upon return to the reference point.

6.1.5   Related Parameters

No.1225   Machine coordinate values of 1st reference point for each axis

6.2　Reference Point Return Check (G27)

After positioning the axes to the program-specified position, a G27 command checks whether that position is the 1st reference point, and if not, an alarm results.

6.2.1　Command Format

G27 X＿＿＿ Y＿＿＿ Z＿＿＿ ...... ;

6.2.2　Sample Program

G27 X100. Z-50. ;

After the X axis moved to 100.0 and the Z axis to -50.0, it is checked whether they have returned to the 1st reference point.

6.2.3　Cautions

(1)　The position reached with a G27 command includes tool offset, tool nose radius compensation, and cutter compensation amounts.
(2)　Only the axis specified in the G27 block is checked whether it has returned.
(3)　No reference point return check is not made on the axis for which the machine lock function is turned on.
(4)　The G27 command confirms imposition after positioning.

6.2.4　Associated Parameters

6.2.5　Associated Alarms

No.F092　　　It has not been returned to the origin.

6.3   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.

6.3.1   Command Format

G29 X___ Y___ Z___ ...... ;

6.3.2   Sample Program

N1 G28 U100. W25. ;
N2 G29 U-70. W-10. ;
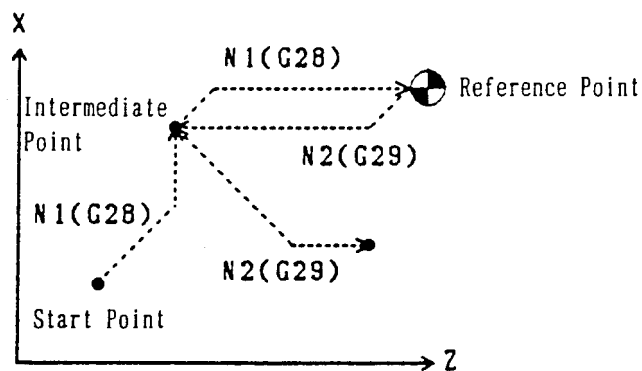
6.3.3   Cautions

(1) If 029 is specified with no G28 or G30 specified once, the intermediate point becomes zero.

6.4 2nd-4th Reference Point Return (G30)

A G30 command can automatically return program-specified axes to the 2nd-4th reference point after positioning them to the intermediate point.

The 2nd ~ 4th reference points are set in parameters at positions set especially for the machine.

6.4.1 Command Format

G30 P___ X___ Y___ Z___ ...... ;

P = 2, 3, and 4 correspond to the 2nd through 4th reference points, respectively.

If P is omitted, the 2nd reference point return is assumed.

6.4.2 Sample Program

G30 P3 U100 . W25. ;
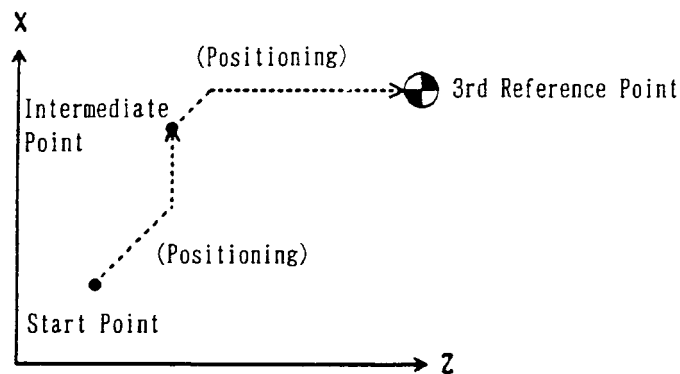
6.4.3 Cautions

(1) If G30 G53 X___ Y___ Z___ ...... ; is specified, the axes are positioned to the specified position of the machine coordinate system, and then, return to the reference point.

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

6.4.4 Associated Parameters

No.1226   Machine coordinate value of 2nd reference point of each axis

No.1227   Machine coordinate value of 3rd reference point of each axis

No.1228   Machine coordinate value of 4th reference point of each axis

6.5 Floating Reference Point Return (G301).... Not available with Σ21L

A G301 command can automatically return the axes specified by the program to the floating reference point after positioning them to the intermediate point.
The floating reference point is the set position on the machine.
The floating reference point can be set in accordance with the following operation.

6.5.1 Command Format

G301 X___ Y___ Z___ ...... ;

6.5.2 Sample Program

G301 U100. W25. ;
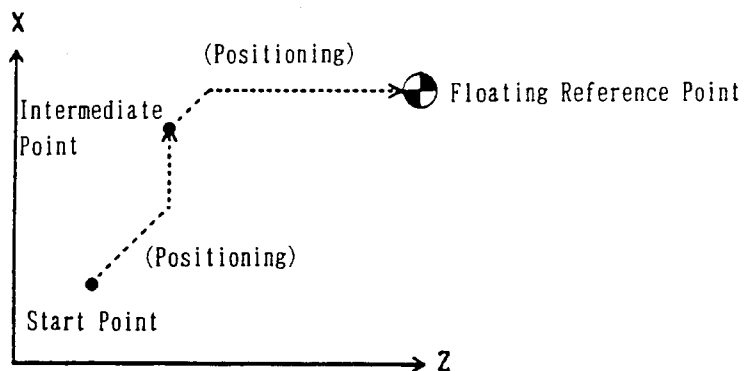
6.5.3 Cautions

(a) G301 G53 X___ Y___ Z___ ; returns the axes to the floating reference point after positioning them to the specified position of the machine coordinate system.

(b) The G301 command confirms imposition upon return to the floating reference point.

6.5.4 Associated Parameters

No.1229   Floating reference point of each axis

7.1   Tool Nose Coordinate System

The tool nose coordinate system always presents the distance between the tool nose position and machining origin correctly.

Therefore, at the time of zero point return, Z setter, Q setter, or turret index, set the coordinate system simultaneously to correct to proper coordinate values.

7.1.1   Configuration of the Tool Nose Coordinate System
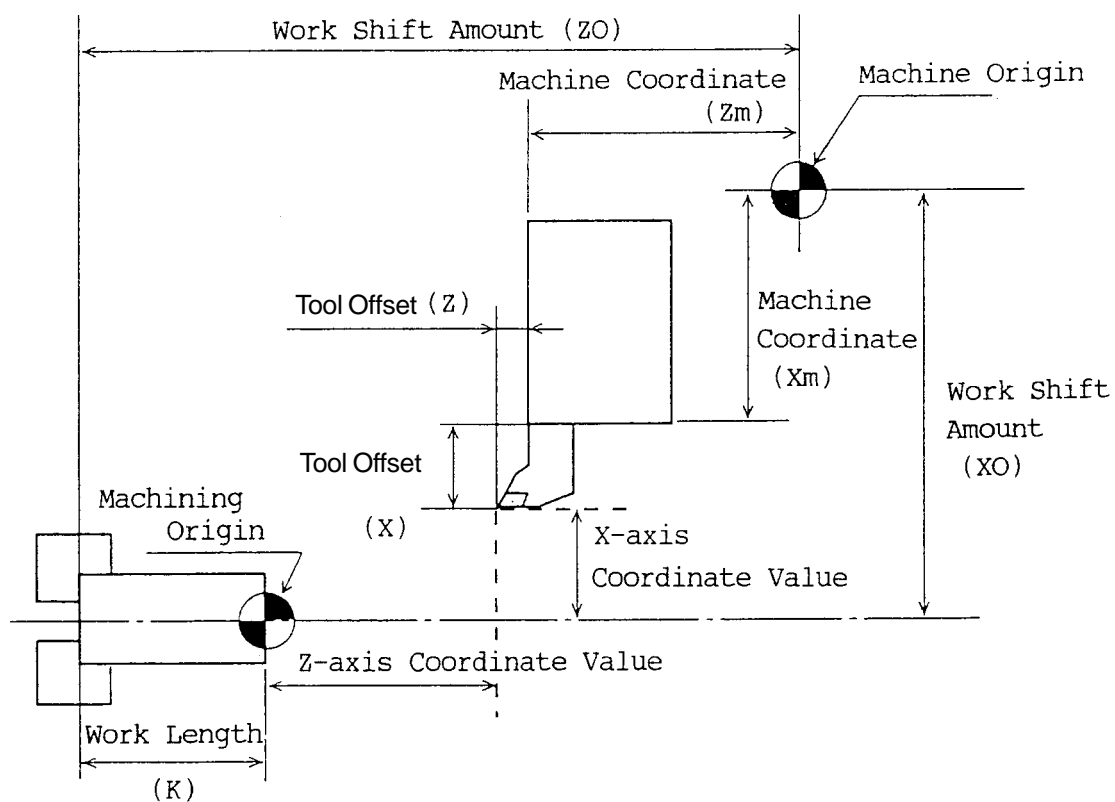
Coordinate System of z-x Plane

Since the tool nose coordinate system is defined as the distance between the tool nose position and machining origin, the coordinate values are obtained by the following expressions.

X axis: Work shift amount (XO) + Machine coordinate (Xm) - Tool offset (X) (-External work origin off set) (Xop)

Y axis: Work shift ainount (YO) + Machine coordinate (Ym) - Tool offset (Y) (- External work origin offset) (Yop)

Z axis: Work shift amount (ZO) + Machine coordinate (Zm) - Tool offset (Z)-Work Length [K] (- External work origin off set)(Zop)

C axis: Work shift amount (CO) + Machine coordinate (Cm) (- External work origin offset)
(Cop)

B axis: Work shift amount (BO) + Machine coordinate (Bm) (-External work origin off
set)(Bop)

(Note) *The OP work origin offset indicates the stun of the "external work origin offset" and the "thermal displacement work origin offset", each of which is used optionally.*

(1) Work shift amount (X0, Y0, Z0, ...)

Length between the machine origin and jaw end face.

It can be input from [Machining Reference Point Shift] of the [Work Coordinate] screen.

No input, however, can be made for x axis on this page. It needs to be input on the parameter page.

(2) Work length (K)

Length between the chuck jaw end face and machining origin.

It can be input from [Work Length] of the [Work Coordinate] screen.

It can be also altered by executing G54 Z (W)___ ;.

(3) Machining origin

Reference point of the machining program

(4) Tool offset (X, Y, Z)

Tool offset amount. It can be altered in the [Tool] screen.

(5) Outer work origin offset (Xe, Ye, Ze, ...)

Those are offset amount that shift the whole work coordinate system.

Input is possible from the [Outer Offset] in the [Work coordinate] screen.

The specified amount of the outer work origin offset will not, however, become valid unless the coordinate system (T command, manual turret index and the like) is set. (The offset amount will not be reflected in the coordinate system.)

7.1.2 Setting of Tool Nose Coordinate System

If the tool is indexed to the cutting position by turret index operation, the coordinate values of the "tool nose position" will change.

This is because setting of the coordinate system is done in order to correctly display the distance between the tool nose position and machining origin at any time.

Setting of the coordinate system is done in the following cases:

(1) When T is specified in the automatic (memory, MDI) mode.

(2) When the tool rest is rotated by pressing the turret index switch in the manual (handle, feed, return to the origin) mode.

(3) When the work length is altered with G54 Z(W) __ ; in the automatic mode.

(4) When the automatic mode is started from the reset state (OP = 0). But ouly when the parameter No.1202, #3 = 1.

(5) When manual return to the origin is performed.

But only the initial return to the origin as for the straight lire axis.

(6) When measurement is made with Q setter or Z setter.

(7) When an offset value for the tool at a cutting position is altered through the screen input in the manual mode.

(8) When both the work shift and work length are altered in the manual mode.

Axes subject to setting coordinate systems for the above (1) through (8) will be as follows:

(1) through (4): all the axes.

        But, a coordinate system will not be set for a rotating axis (Parameter No. 1010, #0=1)

(5) through (8): control axes only

7.1.3 Cautions

(1) If G50 X___ Y___ Z___ ...... ; is executed and the coordinate system is changed, the coordinate value displayed at [Tool Nose Position] will not coincide with the distance from the machining origin to the tool nose position.

(2) When the T command and coordinate system setting (G50) are specified in the same block like G50 X___ Y___ Z___ T___ ;, the T command becomes effective and G50 is ignored.

(3) As described above, the T-command does not set the coordinate system for the rotary axis. Therefore, even if a work shift amount for the rotary axis is altered using a G10 command or custom macro, the T-command cannot reflect it in the coordinate system.

If this is the case, use "work coordinate system preset (G921)" which allows you to set the work coordinate system independently for each axis with a program command. (For G921, see Section 7.7.)

7.1.4 Associated Parameters

No.1209   Work length

No.1233   Work shift amount (Work reference point shift amount)

7.2 Plane Designation (G17, G18, G19)

With a G17, G18, or G19 command, this function specifies the plane in which circular interpolation, cutter compensation, etc. are performed.

7.2.1 Command Format

G17 Xp____ Yp____ ;	Xp- Yp plane
G18 Zp____ Xp____ ;	Zp-Xp plane
G19 Yp____ Zp____ ;	Yp-Zp plane
G17 ;	X-Y plane
G18 ;	Y-Z plane
G19 ;	Z-Y plane

where; Xp: X axis or its parallel axis
          Yp: Y axis or its parallel axis
          Zp: Z axis or its parallel axis

(Note) The α - β plane refers to the orthogonal coordinate system which assumes the α-axis to be the axis of abscissas and the β-axis to be the axis of ordinates.

$+\beta$

$+\alpha$
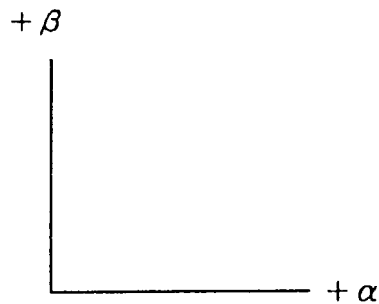
7.2.2 Sample Program

G17 U0 H0 ;	Selects the X-C plane.
	Enables the X-C plane.
G18 W0 D0 ;	Selects the Z-B plane.
	Enables the Z-B plane.

7.2.3 Cautions

(1) Set with the parameter to which one of the X, Y, and Z axes the additional axis is parallel.

(2) The reset state is the G18 mode.

(3) Set properly the relations between the parameter No.1023 (programmed axis name of each axis) and No.1024 (sets which axis of the basic coordinate system each axis is to be).

(4) An alarm results when the plane is not determined in the block where G17, G18, or G19 was specified.

(Example) G17 X___ Y___ D___ ;

When the B axis is parallel to the Y axis, it is uncertain whether the X-Y or X-B plane is selected.

(5) A move command is irrelevant to plane selection.

(6) As an ordinary lathe does not have Y axis, nothing except G18 (Z-X) plane can be selected.

On the other hand, as a drilling axis in the drilling fixed cycle is set to the basic axes (X, Y, Z) other than those of the plane, neither X nor Z axis can be assigned as a drilling axis for G18 plane.

In the above case, set a Y-axis set value (2) to an axis other than control axes of Parameter No.1024. A vertual Y-axis is set on the coordinate system, where selection of G17(X-Y)/G19(Y-Z) planes is enabled and drilling with the X or Z axis can be performed.

(As it is the virtual Y axis on the coordinate system, command of Y axis, even if it is made, is ignored.

7.2.4 Associated Parameters

No.1023   Programmed axis name of each axis

No.1024   Sets which axis of the basic coordinate system each axis will be.

7.2.5 Associated Alarms

No.106   A plane selection (G17-G19) command has an error.

7.3 Work Coordinate System Change (G50)

By specifying G50, you can create the work coordinate system in which a current position will be a specified one.

7.3.1 Command Format

G50 X___ Y___ Z___ ...... ;

7.3.2 Sample Program

N1 G00 X50. Z25. ;
N2 G50 X150. Z50. ;

If the N2 block is executed, the work coordinate system will be set so that the current position will be X = 150.0, Z = 50.0.
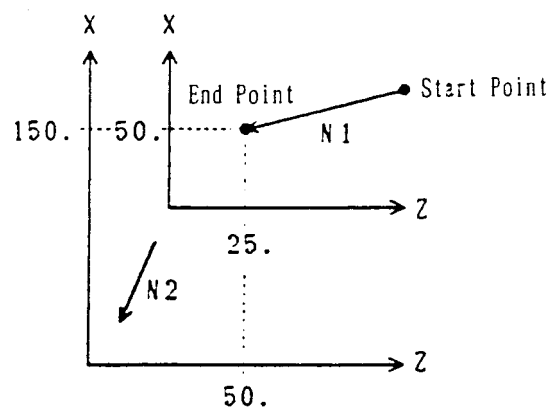
7.3.3 Cautions

(1) G50 is effective only in the block where it is specified.
(2) G50 can be specified by both absolute programming and incremental programming.
     The axes for absolute programming assume specified coordinate values.
     For the axes for incremental programming, the coordinate system is shifted by a specified amount.
(3) If G50 is specified with tool nose radius compensation or cutter compensation applied, the work coordinate system is set so that the position before compensation will be a specified one.
(4) The work coordinate system is validated with a G50 command as follows.
     • Set the coordinate system with a T command.
       (However, the coordinate system of the rotating axis makes no change.)

7.4 Work Length Modification (G54/G55)

Work length of the front (spindle)/rear (subspindle) sides are changed.

7.4.1 Command Format

G54 Z___ ; (Spindle side: Input of the absolute value of the work length)

     W___ ; (Spindle side: Input of the incremental value of the work length)

G55 B___ ; (Subspindle side: Input of the absolute value of the work length)

     D___ ; (Subspindle side: Input of the incremental value of the work length)

7.4.2 Notes

(1) Command data of G54 are set in Parameter No.1209.
   A command value is written in a parameter in G54, whereas a command value is added to a parameter in W command.

(2) Command data of G55 are set in Parameter No.1210.
   A command value is written in a parameter in B command, whereas a command value is added to a parameter in D command.

(3) Following G54/G55 command, the coordinate system is automatically set.

(4) Work length is used in the calculation of the corner R coordinate system.

(5) G55 command is useable only when the rear processing function has been provided. As for the rear face coordinate system, see "19.1 Rear processing."

7.4.3 Associated Parameters

No.1209   Work length 1 (Spindle side)

No.1210   Work length 2 (Subspinle side)

7.5　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.

7.5.1　Command Format

G53　X___　Y___　Z___　...... ;

7.5.2　Sample Program

G53　X50. Z50. ;
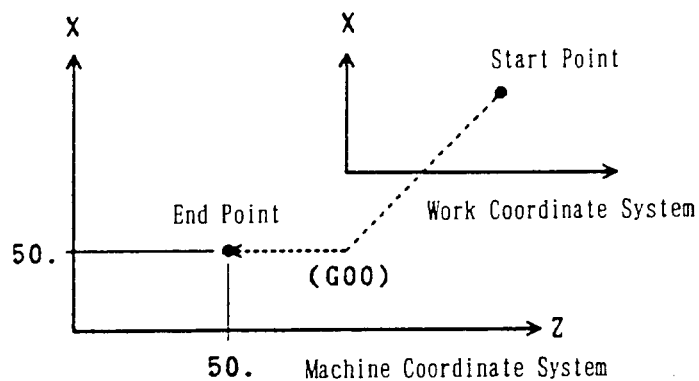
7.5.3　Cautions

(1) An alarm results if an incremental axis command is given.
(2) Specify G53 after cancelling tool nose radius compensation/ cutter compensation.
(3) The G53 command is effective only in the block where it is specified.

7.5.4　Associated Parameters

7.5.5　Associated Alarms

No.146　　An incremental axis command has been given for machine Coordinate System Selection (G53.)

7.6 Setting the Local Coordinate System (G59)

You can set another coordinate system (local coordinate system) in the work coordinate system by specifying the G59 command.

7.6.1 Command Format

G59 X___ Y___ Z___ ...... ;

X, Y, Z: Offsets of local coordinate system(position of origin of local coordinate system taking the origin of the work coordinate system as the reference)

7.6.2 Program Example

N1 G00 X150. Z50. ;
N2 G59 X100. Z25. ;

If you execute the N2 block, the local coordinate system will be set with the values X = 100 and Z = 25 of the work coordinate system as the origin.
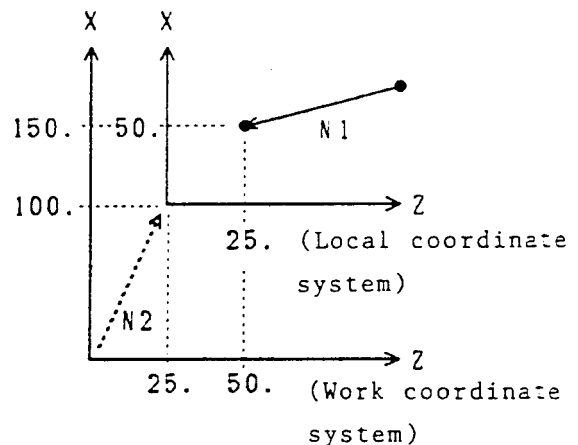
7.6.3 Precautions

(1) G59 is effective only for the specified block.

(2) There is no differentiation in absolute commands/incremental commands for G59.G59X10.; and G59U10.; are the same.

(3) The local coordinate system will be canceled only for the command axis in case of the following:
- When an offset of 0 is specified in G59
- When coordinate system is set in G50.

(4) The local coordinate system of all the axes will be canceled in case of the following:
- Reset
- Coordinate system is set in the T command.

7.7 Work Coordinate System Preset (G921)

A G921 command can be used to preset the work coordinate system independently for each axis (setting of the tool nose coordinate system).
It is valid for coordinate system setting of the rotary axis (parameter No.1010, #0 = 1) for which the coordinate system cannot be set with a T-command.

7.7.1 Command Format

G921 X___ Y___ Z___ C___ ...... ; Specification of the axis for which you want to preset the coordinate system.

7.7.2 Sample Program (C-axis = Rotary Axis)

N1 G28 H0 ; .... Returns the C axis to the zero point.
N2 G50 C0 ; .... Sets the C-axis zero point to C0.
    ≀
N3 G10 P0 C___ ; .... Alters a C-axis work shift amount.
N4 G921 C0 ; .... Presets the C-axis work coordinate system.
    ≀

7.7.3 Cautions

(1) G921 simply specifies the axis for which you want to preset the coordinate system, and its value has no meaning; X0 and X1.0 have the same meaning.
Also, absolute programming and incremental programming do not make any difference. (X1 = U1)

7.8 Lathe Turning Other than the G18 Plane (Z-X)

The command system and operations in the functions given below differ according to the selected plane (G17 to G19).

 [1]   Circular arc interpolation [including helical interpolation] (G02, G03)

*[2]   Chamfering, corner R   (I, J, K, R)

 [3]   Arbitrary angle chamfering, corner R   (,C/,R)

*[4]   Angle-specified straight line interpolation

*[5]   Automatic tip R compensation

*[6]   Groove width compensation   (G150~G152)

 [7]   Tool diameter compensation   (G40~G42)

*[8]   Single fixed cycle   (G90, G92, G94)

*[9]   Compound fixed cycle   (G70~G76)

[10]   Drilling fixed cycle   (G80~G89, G831, G841, G861)

[11]   Direct tap cycle   (G842, G843)

(Note) *Items marked with '*' are functions that are unique to lathes.*

Lathe turning is normally performed in the G18 plane (horizontal axis: Z axis, longitudinal axis: X axis).

In this instructional manual also, functions unique to the lathe are described in the G18 plane, but to rend general purpose features to the NC unit, each function can be executed in other planes (G17, G19) also. Within the NC unit, the axis commands in the selected plane are regarded as horizontal axis (longitudinal direction) commands and longitudinal axis (diameter direction) commands of the plane, and all the commands are executed.

Therefore, commands in other planes are generally executed as described below.

"Axis command"   Replaces the X axis and Z axis in the G18 plane by longitudinal axis, horizontal axis of the selected plane.

"I, J, K"   Specifies the X axis direction as I, Y axis direction as J, and Z axis direction as K.

             <G17>   <G18>   <G19>

            +Y   +X   +Z

                      +X   +Z   +Y
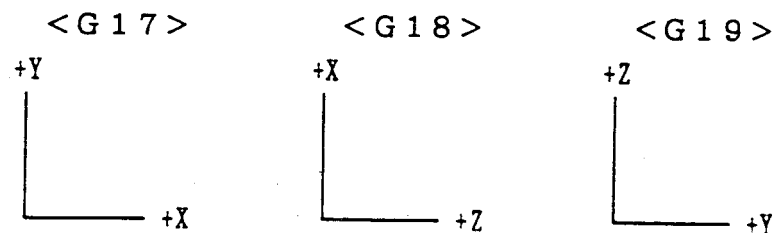
From the axis configuration of the machine, the Z axis becomes the longitudinal direction, the X axis becomes the diameter direction, and if the Y axis exists then the Y axis also becomes the diameter direction.

No lathe turning is assumed in the G17 plane but if it is convenient to perform turning in the Y axis instead of the X axis, then lathe turning in the G19 plane becomes necessary.
However, for the G19 plane, the longitudinal direction becomes the Y axis, and the diameter direction becomes the Z axis, therefore, the G19 plane cannot be used for functions that issue the command recognizing the longitudinal direction and the diameter direction ([6], [8], and [9] mentioned above).

Therefore, the longitudinal direction and the diameter direction for the G19 plane are regarded as Z axis and Y axis respectively, and the functions of [6], [8] and [9] are executed. (Take care because these functions will be executed as they are in the G17 plane.)

(Note) *Set the virtual tip point (control point) of the tool according to the axis configuration of the standard plane. (See figure below.)*
Automatic tip R compensation will be performed on this plane.
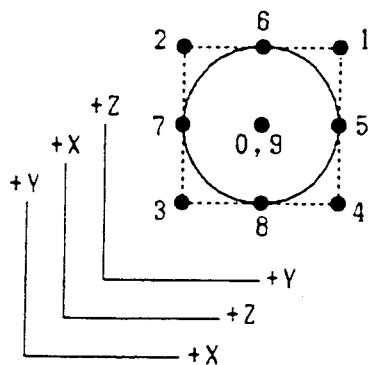
< G19: When the horizontal axis is taken as the Z axis >

7 - 12
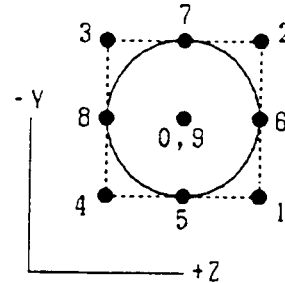
Explanations about the "groove width compensation," "single fixed cycle," "compound fixed cycle" are given below. For detailed explanations, refer to the respective items in the G18 plane.

(1) Groove width compensation (G150 - G152)

[Groove width compensation in the Y-Z plane]

G150 : Cancel groove width compensation
G151 : Groove width compensation for end face / rear end face (Y coordinate system shift)
G152 : Groove width compensation for outer diameter /inner diameter (Z coordinate system shift)
The virtual tip point will be changed as shown below.
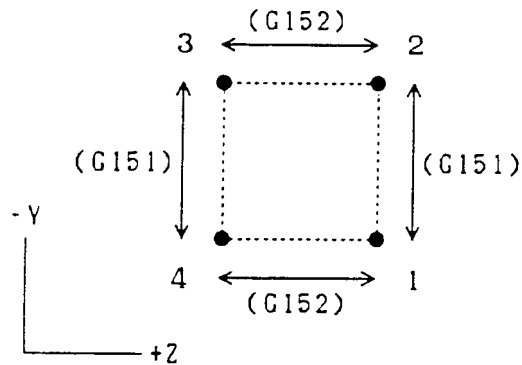
(2) Single fixed cycle (G90, G92, G94)
Explanations about the fixed cycle of G90 are given here but these are similar for other fixed cycles also (G92, G94).

[Single fixed cycle in the Y-Z plane (G90)]

G90 Y___ Z___ J___ F___ ;
(Note the J notation carefully)

R : Rapid traverse
F : Cutting feed
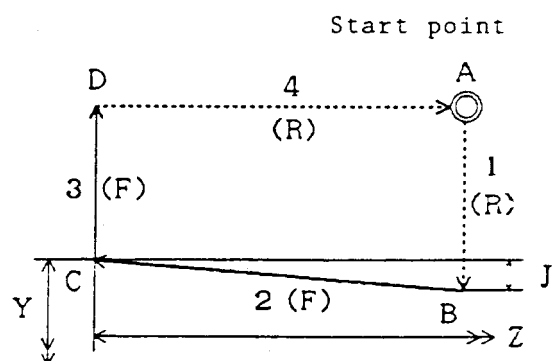
(Note 1) *Specify the position of C point by Y, Z.*
(Note 2) *J is the incremental value (radius value) seeing point B with reference to point C in the Y direction.*

(3) Compound fixed cycle (G70 - G76)

Specify the X axis direction command "X, U, I" of the G18 plane by the Y axis direction command "Y, V, J" in the G19 plane.

Explanations about the fixed cycle of G71 are given here, but these are similar to the explanations for other fixed cycles (G72 - G76).

[Compound fixed cycle (G71) in the Y-Z plane]
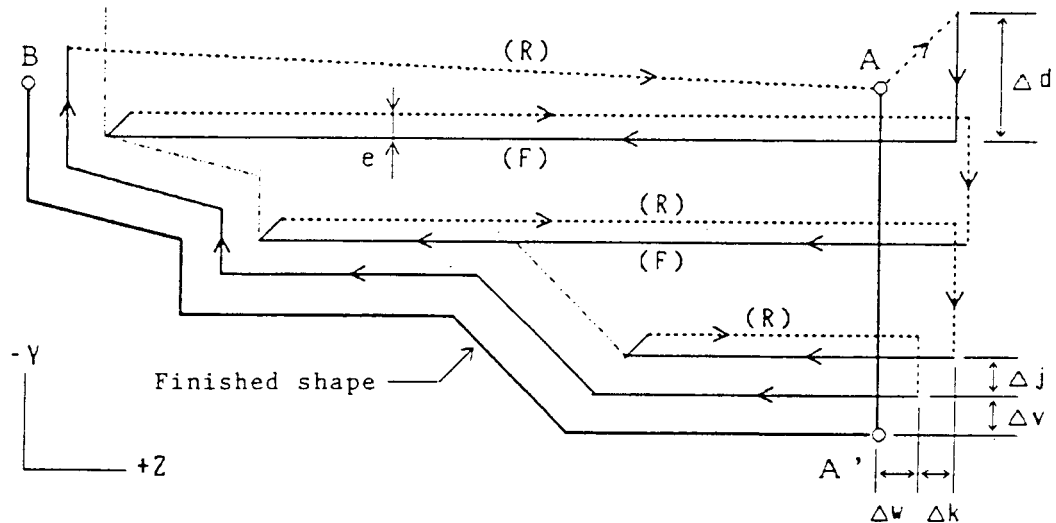
F : Cutting feed, R : Rapid traverse
e : Escape amount (PRM6213)

G71 P(ns) Q(nf) V±__ W±__ J±__ K±__ D__ F__ S__ ;

N(ns) ------ ;
Specify the finished shape of A - A' - B in the block from sequence number (ns) to (nf).
N(nf) ------ ;

P : First sequence number of the finished shape block (ns)
Q : Last sequence number of the finished shape block (nf)
V : Finished shape in the Y direction (Δv)
W : Finished shape in the Z direction (Δw)
J : Y direction allowance of the roughing cycle (ΔJ)
K : Z direction allowance of the roughing cycle (Δk)
D : Cutting depth (Δd) ...... Specify without notation.

7 - 14

8.1 Diameter Designation and Radius Designation

The dimensions of each axis can be specified in two methods; using a diameter value or a radius value. The former is called diameter designation and the latter radius designation. Although which one is to be specified can be selected by the parameter DIAX (parameter No.1010; available for each axis), it has been predetermined for each machine. So, make setting suitable for the mechanical system.

Normally, the NC lathe assumes diameter designation for the X axis, and radius designation for the other axes. As this may vary from one machine to another, refer to the instruction manual for each machine.

D1, D2 ...... Diameter designation
R1, R2 ...... Radius designation
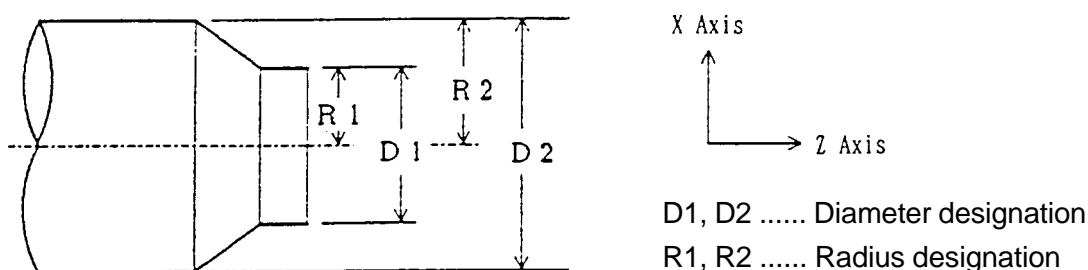
When diameter designation is assumed for the X axis, pay attention to the conditions listed in the following table.

Itemrks	Remarks
Non-X-axis command (including the incremental command by the address V. W, D, or H)	Regardless of the diameter/ radius
X-axis command (including the incremental command by the address U)	Specify with a diameter value.
Coordinate system setting (G50)	Specify the X-axis coordinate value with a diameter.
X value of the tool position offset amount	Specify with a diameter value.
Parameters (D, I, K) for X-axis depth of cut, etc. used for the canned cycle (G90, G92, G94, G70-G76, etc.)	Specify with a radius value.
Radius for circular interpolation (R, I, K)	Specify with a radius value.
Chamfering, corner R amount (R, I, K)	Specify with a radius value.
Tool nose radius	Specify with a radius value.
Feed rate in the X-axis direction	Radius change/rev., Radius change/min.
Display of the X-axis position	Display with a diameter value.

(Note 1) *This manual is described assuming diameter designation for the X axis and radius designation for the other axes, unless otherwise specified.*

8.2 Absolute/Incremental Programming (G90, G91)

These G codes are available only when the G code system B or C is selected.
You can select either absolute programming which gives the specified position of the coordinate system with a value following the axis address, or incremental programming which gives a stroke from a current position.

8.2.1 Command Format

G90 : Absolute programming
G91 : Incremental programming

8.2.2 Sample Program
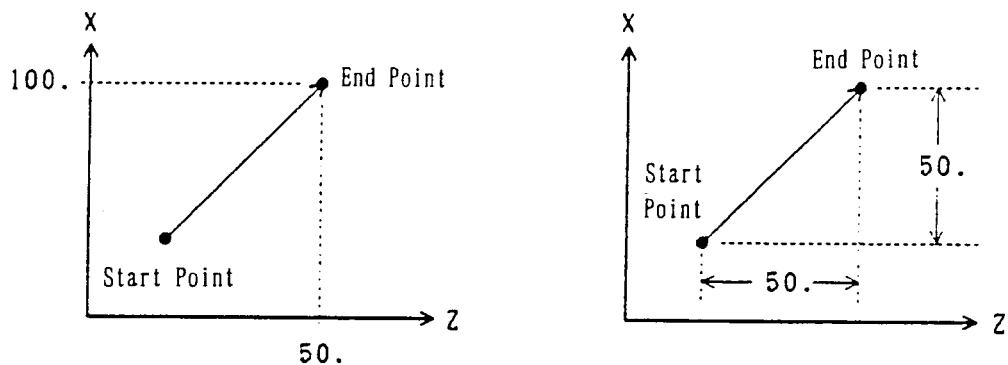
8.2.3 Cautions

(1) When the G code system A is selected, specify with an axis command address instead of using G90/G91. (Refer to "1.1 G Code System")

(2) The G90 and G91 commands are modal G codes. If one of them is specified, it remains effective until the other one is specified.

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

8.2.4 Associated Parameters

No.3402, #1 = 0   The reset state is the G91 mode.
          = 1   The reset state is the G90 mode.

8.3   Inch/mm Input (G20, G21)

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

8.3.1   Command Format

G20 : Inch system
G21 : mm system

8.3.2   The Following Systems of Units are Changed with the G20/G21 Command

(1) Feed rate command with an F code
(2) Commands related to the position
(3) Work origin shift amount
(4) Tool offset amount
(5) Part of parameters
(6) Unit of one graduation of the manual pulse generator

8.3.3   Cautions

(1) Specify G20/G21 command in an independent block after returning the control axis to the machine origin.
    Note that, with changeover of inch input/mm input (change in G20/G21 command or the parameter), an axis not on the machine origin enters the origin returning incomplete state.
(2) When the G20/G21 command is executed, preset the work coordinate system.

8.3.4   Associated Parameters

No.1000, #2 = 0   The input increment system is mm.
          = 1   The input increment system is inch.
No.1010, #0 = 0   Enables inch/mm switching for each axis (linear axis)
          = 1   Disables inch/mm switching for each axis (rotary axis)

9. SPINDLE FUNCTIONS

9.1 Spindle Functions (Function S)

When the rotation number of the spindle (rpm) is specified by a maximum of 8 digits (S00000000 to S99999999) following the address 5, binary code signal, strobe signal (SF), analog signal according to the rotation number of the spindle motor, gear selection singnal etc. are output to the machine side.

In this section, the command S in the status where fixed control of the cycle speed is canceled (mode G97) is described. For the details of the command S in the cycle speed fixed control status, refer to the section 9.2 Cycle Speed Fixes Control.

9.1.1 A Unit of the Command S is Made in One Block.

9.1.2 Example Program

    G97 S1000 ; Rotation number of spindle 1000 rpm

9.1.3 Command S for Rotation Tools and Subspindle

The rotation number command of the rotation tools and subspindle are made by the command S.

Command methods are as follows. However, all of these command methods are not compatible with all kinds of machine compositions. Therefore, refer to manuals attached to each machine.

(1) Direct tapping mode → S for rotation tools

(2) Other than direct tapping mode

    <Command S only: automatically identified depending on the machine status>

    [1] During the front processing mode (also without rear processing function)
- Planing mode (The axis C is not linked with the spindle.) → S for main spindle
- Rotation tool mode (The axis C is linked with the spindle.) → S for rotation tools

    [2] During the rear processing mode
- Planing mode (Subspindle is not identified.) → S for subspindle
- Rotation tool mode (Subspindle is identified.) → S for rotation tools

    (Note) For details of the rear processing mode, refer to the item 19.1 Rear Processing.

    <If Code G + command S: specified by code G>
- G261　S___ ; → S for main spindle
- G262　S___ ; → S for rotation tools
- G263　S___ ; → S for subspindle

    (Note) G261 to G263 are code G of one-shot.

9.1.4 Notes

(1) Command S for setting of the maximum rotation number (G50) is processed only within NC, and signals to the machine side is not output.
(2) If negative number is specified to the command S, it is treated as positive number.

9.1.5 Related Parameters

No.3403 #1 = 0   Code S of G50 represents the maximum rotation number (clamping value).

= 1   Code S of G50 represents the rotation number.

9.2 Constant Surface Speed Control (G96, G97)

Spindle (or subspindle) rotation is controlled according to tool position of X axis (1st axis) so that speed at the cutting point is always held at the specified circumferential speed.

9.2.1 Conurtand Format

G96 S___ ;   Constant surface speed control ON
        S: Surface speed (in/min, feed/min.)
G196 S___ ; Constant subspindle speed control ON
        S: Spindle speed (in/min, feet/min)
G97 S___ ;   Constant surface speed control OFF
        S: Spindle speed (rpm)

9.2.2 Sample Program

G96 P1 S10 ;   Performs constant surface speed control with one axis. (Constant surface speed: 10 m/min.)
G97 S500 ;   Constant surface speed control cancel (Spindle speed: 500 rpm)

9.2.3 Cautions

(1) When performing constant surface speed control, see to it that the center of the rotary axis will be the origin of the work coordinate system.
(2) Only when the rear side processing function is available, Constant control of speed per cycle for subspindie (G196) can be used. (See "19.1 Processing of Rear Side".)

9.2.4 Associated Parameters

9.3 Maximum Spindle Speed Setting (G50)

The maximum spindle speed (rpm) can be specified with a numerical value following S specified in the same block as G50.
Also, the maximum rotation number of rotation tools and subspindle as well as main axis can be specified.

9.3.1 Command Format

G50 S___ (P1) ; The maximum rotation number of the main axis. (P1 can be omitted.)

G50 S___ P2 ; Setting of the maximum rotation number of rotation tools.

G50 S___ P3 ; Setting of the maximum rotation number of subspindle.

9.3.2 Sample Program

G50 S1000   ; Rotation number of the main axis is clamped at 1000rpm.

G50 S2000 P2 ; Rotation number of the rotation tool is clamped at 2000rpm.

9.3.3 Cautions

(1) The maximum spindle speed set with a G50 command is effective regardless of the G96/G97 mode.

9.3.4 Associated Parameters

No.3403, #1 = 0   S code of G50 is the maximum rotation number. (Clamp value)

= 1   S code of G50 is the number of rotations

9.4   Spindle Speed Variation Detection (G25/G26)

When the spindle speed has increased or decreased by a command due to machine conditions, an overheat alarm is produced together with the spindle speed variation detection alarm signal being output to the machine side.
This is used for prevention of seizure of the guide bush.

9.4.1   Command Format

(1) Spindle speed variation detection enabled:

G26 (P___) (Q) (R___) ;

P___   : Time required from change in the specified rpm to start of spindle speed variation detection (msec)

Q___   : Allowable rate of rpm at which the spindle is taken to have reached the specified rpm (1%, 01%)

R___   : Spindle variation rate at which the spindle speed variation detection alarm is not produced. (1%, 0.1%)

(2) Spindle speed variation detection disabled:

G25 ;

9.4.2   Detecting System of Spindle Speed Variation

The following method is used to detect whether or not a spindle rpm has varied from the specified rpm by an amount above the allowable rpm.
In other words, Sd or Sr mentioned below whichever being larger in rpm is taken as the allowable variation rpm Sm. When an actual spindle rpm has exceeded Sm against the specified rpm, Sc; variation above the allowable level is detected and an alarm is produced.

Sd : Allowable variation range of fixed amount which does not depend on the specified rpm (Parameter No.4913)

Sr : Allowable variation range obtained when the specified rpm Sc being multiplied by the constant ratio r (Parameter No.4912)

Sm : Sd or Sr, whichever is larger in rpm

9.4.3 Conditions for Starting Spindle Speed Variation Detection

On occurrence of change in the specified rpm of the spindle Sc, spindle speed variation detection is started when either one of the following conditions has been satisfied:

1) Actual spindle rpm has reached the scope of (Sc - Sg) ~ (Sc + Sq)

where:

$$Sq = Sc \times \frac{q}{100}$$

q : Allowable ratio in rpm with which the spindle is taken to have reached the specified rpm. (Parameter No.4914)

2) Time p which has been set in Parameter No.4914 has passed since the specified rpm Sc was changed.

9.4.4 Notes

(1) Make G25/G26 commands into independent blocks.

(2) P/Q/R values commanded in 026 are set in parameters.

P : Parameter No.4914
Q : Parameter No.4911
R : Parameter No.4912

Specify P/Q/R without setting a decimal point.
(Specify Q/R by using the unit set in the parameters.)
No parameter setting is performed with those for which commanding has been omitted. Checking, in that case, is performed with the value having been already set.

(3) On occurrence of an alarm, Machine is brough to a single block stop while in auto operation.

(4) No checking is performed while in spindle stop (*SSTP=0).

(5) With the actual spindle rpm being 0 rpm, an alarm starts when on second or more has passed.

9.4.5 Associated Parameters

No.3708, #4 = 0   Spindle speed variation detection disabled when SIND signal is ON.

        = 1   Spindle speed variation detection enabled when SIND signal is ON.

No.4900, #0   The unit for the allowable ratio (q)/variation ratio (r) set in Parameters No.4911/4912 for spindle speed variation detection is:

        =0   1%

        =1   0.1%

No.4911   Allowable ratio of rpm at which the spindle is taken to have reached the specified rpm (q)

No.4912   Spindle variation ratio with which a spindle speed variation detection alarm is not produced (r)

No.4913   Spindle variation rpm with which a spindle speed variation detection alarm is not produced (d)

No.4914   Time required from change in the specified rpm to start of spindle speed variation detection (p)

9.4.6 Associated Alarm

No.F704   Overheat: Spindle

        (Spindle overheat detected by spindle speed change)

10. TOOL FUNCTION

10.1 Tool Function (T Function)

When numeral values of max. 8 digits which are preceded by address T have been specified, BCD code signal (T11~T88) and strobe signal (TF) are output in Machine side. Further, through combination of T codes, the following processings are carried out:

[1] Tool nose coordinate system setting (normal and multitool)

[2] Tool multiple offset ON/OFF

   For details of the above [1] ~ [2] , refer to "7.2 Tool Nose Coordinate System" and "12.4 Multiple Offset".

10.1.1 T Code Configuration

T code configuration, depending on use of ATC and available kinds of combination of tool offset, gets as follows:

<With ATC> ... Specified by 6- or 8-digit T code.

(1) 6-digit specification → Available kinds of tool offset 32/64/99

       T hh tt △△ ;

                            Multiple offset number, multitool offset number

                            Tool number

                            Thrret plane number

(2) 8-digit specification → Available kinds of tool offset 200

       T hhh ttt △△ ;

                            Multiple offset number, multitool offset number

                            Tool number

                            Thrret plane number
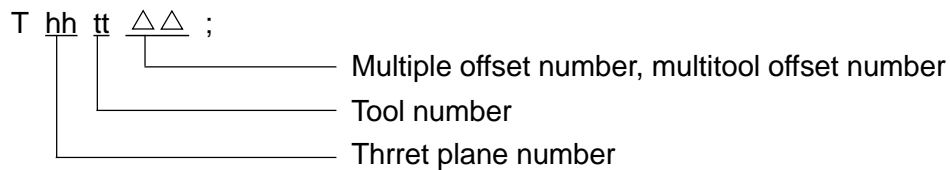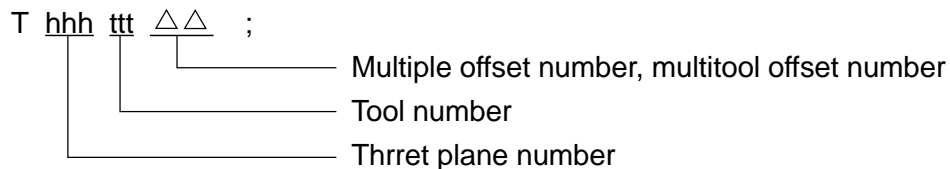
<Without ATC> .... Specified by 4-digit T code.

              → Available kinds of tool offset 32/64/99

       T tt △△△ ;

                            Multiple offset number, inultitool offset number

                            Tool number (= Turret plane number)
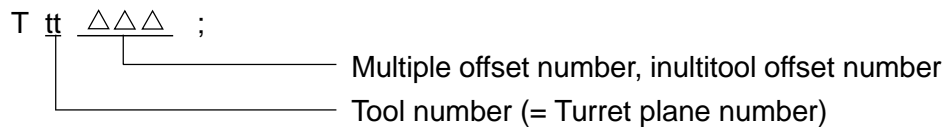
10.1.2　Sample Program

    (1)　T6 digits with ATC

        T021200 ; Tool No. 12 is called in Turret 02 plane.

    (2)　T4 digits without ATC

        T0200 ; Turret 02 plane (Tool No.02) is called.

10.1.3　Tool Number and Offset Number

The tool number and compensation number of a T code are made in the following specifications so that they can deal with the case for which two tools (compensations) are needed for one turret plane [multiple offset] or the case for which two or more compensations are synthesized with one tool [multitool offset].

The following description assumes "tt" to be a tool number and "$\triangle\triangle$" to be an offset number to represent the T 4-digit/T 6-digit specification.

Both tool number and offset number assume 3 digits in the T 8-digit specification, but take it in the same manner. A turret face number is omitted in this description because it has no meaning.

    (1)　At normal cutting

        T <u>XX</u> <u>00</u> ;

                          00 (zero)

                          Tool offset number

        The coordinate system with the tool offset amount of NO.XX is set at the same time of turret index. Also, the multiple offset state is cancelled. (The offset amount of NO.XX is also used for groove width offset.)

    (2)　Multiple Offset

        T <u>XX</u> <u>$\triangle\triangle$</u> ;

                          Multitool offset number

                          Tool offset number

        The coordinate system with the tool offset amount of NO.$\triangle\triangle$ is set at the same time of turret index. (The offset amount of NO.$\triangle\triangle$ is also used for groove width offset.)

        Also, if this T command is executed in the status of multiple offset, the multiple offset is canceled.

(3) At multiple offset

T XX △△ ;

                                 Offset number (multiple offset)

                                 Tool number (Same as the one currently in use.)

If a multiple offset T-command is executed independently, the axis will be moved by the offset amount of no.△△ . If a T-command is given simultaneously with an axis command, the axis will be moved by an original move amount plus the offset amount of no.△△.

(Note) *For multiple offset, see 12.4 Multiple Offset*

10.1.4   Classification of the Offset Amount at Multitool Cutting and Multiple Offset

There are two types of tool offset; one is to indicate physical tool dimensions, and the other is to indicate an offset amount as multiple offsets. This concept serves as a judgment criterion whether it is multitool cutting or multiple offsets. Concretely, it is judged based on the virtual tool nose point and tool nose radius value of the number indecated by offset number (△△) of the T code as follows.

T tt △△ ;

Meaning of △△

- When the virtual tip point is not equal to 0, △△ will be treated as the multi-tip tool number.
- When the virtual tool nose point is 0 and the tool nose radius is 0, △△ is considered a multiple offset number.
  When the turret number or tool number has changed from the previous T-command, or when the G52 command (back machining mode selection) is in the same block, however, an alarm (no. 182) will result
- When the virtual tip point = 0, and tip radius is not equal to 0, alarm (No. 182) will be output.

10.1.5   Cautions

(1) The effective number of digits for a T command, as mentioned in "10.1.1 T Code Configuration", is either 8 or 6 digits with ATC, or 4 digits without TC.

(2) No offset number (multiple/combined) is not output in a code signal to the machine side.
     The code signal in case of T123456 ; is "00001234".

(3) If the T command is given, the NC unit normally stops prereading.
   In the following cases, however, it does not stop prereading.
   Tool multiple offset ON/OFF
   T9****** ; Command

(4) In the following cases, the T-command is treated only inside the NC unit, outputting no code signal or strobe signal to the machine.
   • Tool multiple offset ON/OFF
   • Machine external measurement compensation (G194)

(5) When the coordinate system is set by the T-command (normal command/multicut offset command), the display of "TOOL T" and OFFSET DATA (OFFSET X/Z, TOOL NOSE R, TOOL WIDTH, TOOL NOSE POINT T) in the Overall screen is updated. Since the multiple offset T-command does not set the coordinate system (the tool itself does not change), the previous tool data which is the base for multiple offset are displayed without updating the above-mentioned display data.

(6) When a negative value is specified in the T command, it is treated as a positive value.

10.1.6   Associated Parameters

No.5009   Max. number of planes of the tool post
No.5010   Max. number of pots (no. of tool post planes + no. of ATC magazine pots)
           ...... Only when the ATC is attached

10.1.7   Associated Alarms

No.182   An error exists in T command.
 (#001)   An error exists in the number of digits of T code.
 (#002)   The turret face number has an error. (Face number = 0 or > maximum number of faces)
 (#003)   The tool number has an error. (Tool number = 0 or > maximum number of tool offsets)
 (#004)   The offset number has an error. (Offset number > maximum number of tool offsets)
 (#005)   The multiple offset command has an error. (TOOL NOSE POINT = 0, but TOOL NOSE RADIUS ≠ 0)
 (#006)   When the ATC is attached, there is an error in a command for the turret face number in the 90s (90, 91, 99)
 (#007)   The multiple offset command has an error. (In the G02/G03 mode, axis command given in the same block)
 (#008)   The multiple offset command has an error. (Turret face number or tool number changed)
 (#009)   The multiple offset command has an error. (Multiple offset command in the same block as G52 command)

10.2 ATC Canned Cycle

For Machines provided with ATC, a command is given by T codes of either 6 or 8 digits. (As for details of the T codes, see "10.1.1 T Code Configuration".)

When a T code is specified, "turret index" or "ATC operation" is performed according to the specified content.

This chapter covers "ATC operation".

The ATC includes three types of A, B, and C, whose differences are briefly explained as follows. The commanding methods are the same to all of them.

• Types A/B .... Tool mounting on each turret face is performed first in the Z-axis direction. The ATC position is limited to one.

        Type A: The magazine moves together with Z axis.

        Type B: The magazine does not move.

• Type C .... Tool mounting is performed first in the Z-axis for some turret faces and in the X-axis for the other. The ATC, positions include two. (Z-axis direction/X-axis direction)

"As for details of ATC, see the instruction manual of each Machine."

10.2.1 Command Format

T commands, normally, consist of the following:

(1) For T6 digits

    T hh tt △△ ;

                         — Offset number

                       — Tool number

                    — Turret plane number

(2) For T8 digits

    T hh ttt △△△ ;

                         — Offset number

                       — Tool number

                    — Turret plane number

These commands indicate that "a specified tool t be mounted on a specified turret face h and the turret face be indexed to the cutting face."

When the tool t has already be mounted on the turret face h, turret indexing alone is performed by the above T command.

(Note) Since the offset number has nothing to do with ATC processing and, also, T-6/T-8 digits only cause difference in the maximum values of the tool (offset) numbers, the description hereafter uses the following:

• Offset number = 0

• T6 digits

10.2.2 Special Commands

When a number of 90's has been specified as a turret face number, it serves as the following special command:

(1) Tool calling

   T <u>99</u> tt 00 ; (tt : tool number)

   The specified tool held in a magazine is called out on the magazine-side arm.

(2) Pull-out of magazine-side arm tools

   T <u>90</u> 00 00 ;

   Tools on the magazine-side arm are pulled into the magazine. If there is no empty space in the magazine, however, they are pulled into the turret.

(3) Pull-out of turret-side arm tools

   T <u>91</u> 00 00 ;

   Tools on the turret-side arm are pulled into the magazine. If there is no empty space in the magazine, however, they are pulled into the turret.

   (Note 1) In any case, the offset number needs to be 0.

   Further, in "pull-out of magazine-side arm tools" and "pull-out of turret-side arm tools", a tool number cannot be assigned. Give a command by using the tool number =0.

   (Note 2) The special T-code command is a preread stop command and has an effect on automatic tool nose radius compensation and cutter compensation. Give the T-code command in the non-compensation start-up or non-compensation mode block as much as possible. For the effect of the preread stop command on automatic tool nose radius compensation and cutter compensation, see 12. COMPENSATION FUNCTION.

10.2.3 Basic Operation ...... T hh tt 00;

Regardless of ATC type, the basic operation becomes as follows:

(1) Output T hh tt 00; in Machine. Further if there being any M command or S command in the same block, it is output together with the T code.

(2) The axes assigned by parameters move to the ATC position in the assigned order. However, the methods to assign ATC position differ between Types A/B and Type C.

   [Types A/B]

   ATC positions for all axes are set by parameters.

   [Type C]

   The ATC position (3rd origin/4th origin) for X, Y and Z axes follow the instruction from Machine (PMC). Assignment of the ATC position for other axes are set by the parameters in the same way as Types A/B.

(3) On completion of move into the ATC position, perform tool changing.

(4) On completion of a T function, perform coordinate system setting by using a new tool offset volume.

(5) The axes assigned by parameters move to the completion position according to the assigned order and position (1st origin/3rd origin/4th origin).

(Note) *The above (2) and (3) are performed while T function of (1) is in execution ("TF"= 1).*

10.2.4　Interlock by Tool Type

Tools are largely divided into the following three types:

"Fixed tool"/"Turning tool"/"Rotary tool"

Prior to starting ATC, it is necessary to set the tool type on Tool List screen. The following interlocks are prepared concerning a type of the assigned tool and the assigned turret face.

(1) In any type, "rotary tools" cannot be subject to ATC.

(2) In Types A/B, "rotary tools" can be mounted only on an even-numbered turret face.

(3) In Type C, "X-axis rotary tools" can be mounted only on the 5th face of a turret, whereas "Z-axis rotary tools" on the 7th.

Through use of G159 command (Rotary tool offset auto conversion), a tool is made attachable either to the 5th or the 7th face. (As for details, see Item "10.3".)

For incapable operation, the NC unit outputs the ATC alarm signal into Machine, displaying an alarm message.

10.2.5　Cautions

(1) An shift axis to the ATC position and an shift axis from the ATC position to the ATC completion position can be set for the ATC canned cycle.

(2) The lst/3rd/4th reference points can be selected for each axis, which serve as the ATC position and the completion position. As for X/Y/Z axes of Type C, however, ATC position setting is not available.

(3) The order for shifting can be set for each axis.

(4) The axis command, if given within the same block, can produce an alarm. (Excepting a T command of combined compensation.)

(5) Whether or not to stop for each operation can be set for Single Block.

10.2.6 Associated Parameters

No.5009   Max. number of planes of the tool post

No.5010   Max. number of pots (no. of tool post planes + no. of ATC magazine pots)

No.5103, #4 = 0   Single Block is made invalid in ATC canned cycle.

        = 1   Single Block is made valid in ATC canned cycle.

No.5108, #0 = 0   Each axis does not move to ATC position in ATC canned cycle.

        = 1   Each axis moves to ATC position in ATC canned cycle.

No.5108, #1 to #2 Selecting ATC Position for Each Axis (Excepting X/Y/Z axes of Type C)

#2	#1	ATC Position
0	0	1st origin
0	1	3rd origin
1	0	4th origin

No.5108, #5 to #7 Selecting Shift Order of Each Axis to ATC Position (Two or more axes movable simultaneously.)

#7	#6	#5	Shift Order to ATC Position
0	0	0	1st
0	0	1	2nd
0	1	0	3rd
0	1	1	4th
1	0	0	5th
1	0	1	6th
1	1	0	7th
1	1	1	8th

No.5109, #0 = 0   Each axis does not move to ATC completion position in ATC canned cycle.

        = 1   Each axis does not move to ATC completion position in ATC canned cycle.

No.5109, #1 to #2 Selecting ATC Completion Position of Each Axis

#2	#1	ATC Completion Position
0	0	1st origin
0	1	3rd origin
1	0	4th origin

No.5109, #5 to #7 Selecting Shift Order of Each Axis to ATC Completion Position (Two or more axes movable simultaneously.)

#7	#6	#5	Shift Order to ATC Completion Position
0	0	0	1st
0	0	1	2nd
0	1	0	3rd
0	1	1	4th
1	0	0	5th
1	0	1	6th
1	1	0	7th
1	1	1	8th

10.2.7 Associated Alarms

No.182   T Command error

(#001)   An error exists in T command.

(#002)   An error exists in the turret face number. (Face number= 0 or > max.number of faces)

(#003)   An error exists in the tool number. (Tool number= 0 or > max.number of tool offsets)

(#004)   An error exists in the offset number. (Offset number > max.number of tool offsets)

(#005)   An error exists in the multiple offset command. (Tool nose point ≠ 0, where tool nose point= 0)

(#006)   An error exists in a command for a turret face number in the 90's (90, 91, 99) in ATC specification.

(#007)   An error exists in the multiple offset command. (The axis command is given within the same block in G02/G03 mode.)

(#008)   An error exists in the multiple offset command. (Turret face number or tool number has changed.)

(#009)   An error exists in the multiple offset command. (Multiple offset command in the same block as G52 command)

(#100)   The assigned tool does not exist. (ALM2)

(#101)   An error exists in the type of the assigned tool. (ALM3)

(#102)   An error exists in the tool post side arm. (ALM1)

(#103)   No empty pot exists in the magazine. (ALM2)

(#104)   No empty face exists in the tool post. (ALM2)

(#105)   ATC command has been given to the fixed tool. (ALM3)

(#106)   No dummy tool exists in MG-TP-TP operation. (ALM2)

(#107)   An error exists in G159 command.

(#108)   An axis command exists within the same block with T command except of multiple offset.

(#109)   A turret internal tool change command has been given.

(#150)   The machining mode is inconsistent with the T-code command.

(#151)   An error exists in the turret angle command value.

(#152)   A turret angle command has been given in T02xxxx.

(#154)   A type unset tool has been specified.

10.3 Rotary Tool Offset Auto Conversion (G159)

In Type C ATC, a rotary tool can be mounted both on the X-axis rotary tool station and on the Z-axis rotary tool station of the turret.

Normally, however, it can be mounted on only one of the rotary tool stations because of interlock by tool type. (As for interlock by tool type, see "10.2 ATC Canned Cycle".)

By G159 and a T code being commanded in the same block, tool type is converted (rotary X $\leftrightarrow$ rotary Z) for ATC to be performed.

Further, by X-axis/Z-axis tool offset being converted according to tool direction, coordinate system setting is carried out with the tool offset obtained from the conversion. Further, by parameter setting, the virtual tool nose point is, also, subject to conversion.

<Z-axis rotary tool station>   <X-axis rotary tool station>
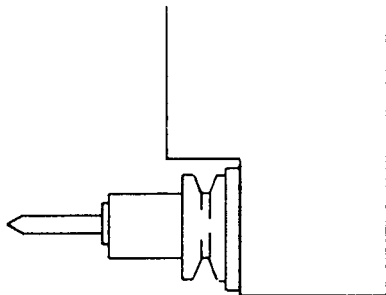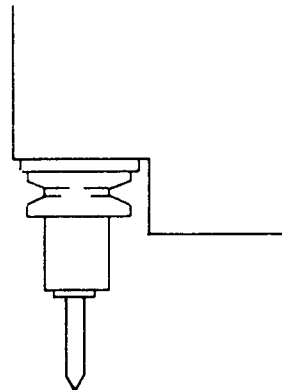
(Note) Even where single tool can serve both as the X-axis rotary tool and as the Z-axis rotary tool, it is recommended to avoid complication of a program that two separate tools be used without application of G159 whenever affordable.

10.3.1 Command Format

G159 T△△ ◎◎ xx ;

Through using this command, convert tool type (rotary X $\leftrightarrow$ rotary Z) to perform ATC and, following ATC operation, perform tool offset conversion (X axis $\leftrightarrow$ Z axis) and coordinate system setting.

Conversion is performed only when all the following conditions are satisfied:

[1] When G159 and a T code has been assigned within the same block.

[2] The specified tool (No.◎◎) is either a rotary X or rotary Z of type.

[3] The specified tool is not the converted one.

[4] ATC operation needs to be practiced by a T command.

(Note) G159 is a one-shot G code.

10.3.2 Tool Type Conversion

With G159 command being given, tool type interlock checking is performed where a rotary X is taken as a rotary Z and a rotary Z as a rotary X.

ATC operation to fit a rotary Z tool to the rotary X station and a rotary X tool to the rotary Z station is made enabled here.

Further, through tool type conversion, the rotary X and rotary Z are converted into *X and *Z respectively.

(Note) *Checking the tool type interlock is performed for the tool types before conversion.*
*When specifying a "T command" to the type conversion tool (*X, *Z) be sure to specify "G159."*

10.3.3 Tool Offset Conversion

With tool offset conversion being practiced, the tool offset of X and Z axes are converted as follows. Coordinate system setting is performed with the tool offset obtained from conversion.

Conversion Type	Axis	Before Conversion		After Conversion	
		Profile	Wear	Profile	Wear
Rotary X ↔ *X	X	x	u	A	0
(Rotary Z)	Z	z	w	(x/2)-C	u/2
Rotary Z ↔ *Z	X	x	u	2(z+C)	2w
(Rotary X)	Z	z	w	B	0

Where, A/B/C indicate values having been set in parameters prior to Machine shipment.

   A: Difference between the inner diameter center and the Z-axis rotary tool center (Xaxis direction) ... Standard value = 0

   B: Difference between the X-axis rotary tool center and the reference outer diameter tool nose (X axis direction)

   C: Difference between the X-axis tool offset and the Z-axis tool offset
For details, see "10.3.7 Associated Parameters".

10.3.4 Virtual Tool Nose Point Conversion

For use in auto tool offset measurement (Auto Presetter), conversion of the virtual tool nose point, also, is performed by parameter setting.
Details of conversion are as follows:

Conversion Type	Change in Virtual Tool Nose Point										
Rotary X ↓ *Rotary X (Rotary Z)	Before conversion	0	1	2	3	4	5	6	7	8	9
	After conversion	0	4	1	2	3	8	5	6	7	9
Rotary Z ↓ *Z (Rotary X)	Before conversion	0	1	2	3	4	5	6	7	8	9
	After conversion	0	2	3	4	1	6	7	8	5	8

10.3.5 Conversion Cancel

When a converted tool is detached from the specified turret face, the conversion is cancelled; returning the tool type, the tool offset value and the virtual tool nose point to the states before conversion.
There are two ways available in actual practice:

[1] When the converted tool has been subject to ATC conversion with any other tool:
  G159 T051300; Tool No.13 offset conversion
  T051400; After cancelling of tool No.13 conversion, coordinate system setting is performed with tool No.14.

[2] When the converted tool fitted on the rotary X or the rotary Z station has been moved to any other position (other station or magazine) by manual operation:
  Practice the conversion cancelling in timing of coordinate system setting such as on manual turret indexing and on the first zero point returning from power supply.

Further, when a tool type or tool offset value has been changed in conversion state (manual input, Q setter, etc), Conversion Cancel is made as follows:
<When tool type has been changed.>
The conversion state, simply, is internally cancelled, where none of the tool type, the tool offset value, and the virtual tool nose point are changed.

<When tool offset value has been changed.>
Offset values are reversely converted as in the following table. (The tool types and virtual tool nose points are returned to pre-conversion states.)

• Converted state of "*X"

Axis	Pre-conversion value		Post-conversion value		Post-cancel value	
	Profile	Wear	Profile	Wear	Profile	Wear
X	x	u	α	p	$2(\beta+c)$	2q
Z	z	w	β	q	z	w

• Converted state of "*Z"

Axis	Pre-conversion value		Post-conversion value		Post-cancel value	
	Profile	Wear	Profile	Wear	Profile	Wear
X	x	u	α	p	x	u
Z	z	w	β	q	$(\alpha/2)-c$	p/2

(Note) *A virtual tool nose point, if being changed, does not affect conversion cancelling.*

10.3.6 Cautions

(1) Converted tool types, tool offset values and virtual tool nose points are retained even after cutting-off of power supply unless conversion cancelling is performed.

(2) G159 is not useable for any other ATC except the one of Type C.

10.3.7 Associated Parameters

No.5005, #7 = 0   Virtual tool nose point is not converted by Rotary Tool Offset Auto Conversion.

          = 1   Virtual tool nose point is converted by Rotary Tool Offset Auto Conversion.

No.5028 (Parameter A)   Difference between the inner diameter center and the Z-axis rotary tool center (X axis direction) ... Standard value= 0

No.5029 (Parameter B)   Difference between the X-axis rotary tool center and the reference outer diameter tool tip (Z direction)

No.5030 (Parameter C)   Difference between the X-axis tool offset and the Z-axis tool offset

(※) As for Parameters A/B/C, see the following drawing.

Reference outer tool tip

$C = (X/2) - Z$
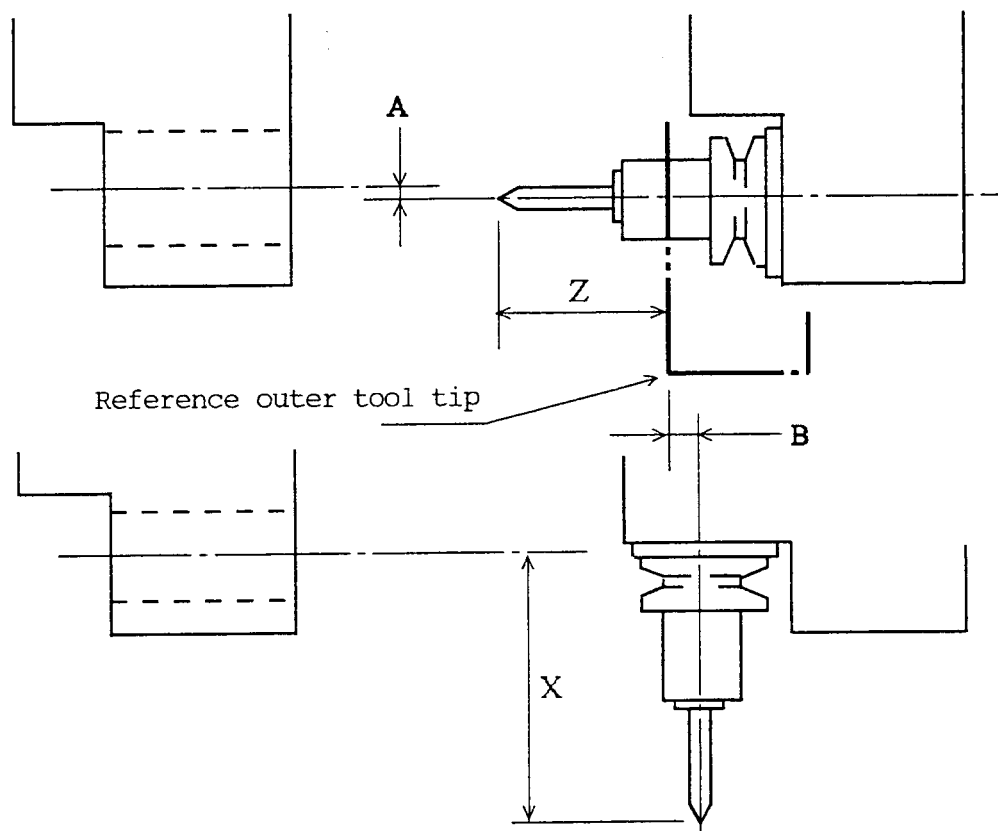
X/Z are the tool offsets when the same rotary tool is measured in Q-setter.

    X :   X-axis tool offset when measured as an X-axis rotary tool

    Z :   Z-axis tool offset when measured as a Z-axis rotary tool

10.4 ATC Type-E Offset Automatic Change

With the ATC Type-E, the tool offset reference changes depending on the turret face specified by a T-code command. Referencing the position where the rotary tool face has been indexed to the spindle side, the NC unit internally calculates a tool offset reference shift amount by a turret command and turret rotation angle to set the coordinate system. For the tool offset, set a projection amount from the turning tool/rotary tool face.
(A fixed tool also assumes the projection amount from the rotary tool face.)

10.4.1 Turret Face and Indexing Angle

A turret command specifies the tool mounting turret face and turret indexing angle. The following table shows the relations between the turret commands and turret faces, and indexing angles.

	Turret Face	Indexing Angle
T01tt00	Turning tool face	Indexes the turning tool face in the -Z direction (spindle direction)
T02tt00	Rotary tool face	Indexes the rotary tool face in the -Z direction(spindle direction).
T03tt00	Fixed tool face	Indexes the rotary tool face in the -Z direction(spindle direction).

10.4.2 Machining Mode Interlock

The turret commands are interlocked with the machining mode as follows, corresponding to the indexing angle.
(1) T01tt00 can be given only in the front mode(G170).
(2) T02tt00 and T03tt00 can be given in both front and back modes.

10.4.3 Fixed Tool Command

When using a fixed tool located below the rotary tool face, index it to the angle determined by a turret command regardless of the tool mounted onto the turret face, by giving T03tt00, and set the coordinate system, using the offset number tt.
When this is done, it is necessary to set a projection amount from the datum level of the rotary tool face in the offset number tt as an offset amount.
This is equivalent to T02tt00 except that the T-code command only performs indexing regardless of the tool mounted onto the rotary tool face.

10.4.4 Tool Reference Shift Amount

As the tool offset reference position(gauge line) varies depending on the T-code command, change a tool reference shift amount according to the T-code command. The tool reference shift amount 0 refers to the gauge line of the rotary tool face at the time of giving T02tt00.

	Axes	
	X	Z
T01tt00	2 (C-2D)	B-A
T02tt00	0	0

A, B, and C are the values preset in the parameters upon shipment of the machine. See the figure below.

A: Distance from the rotation center of the turret to the gauge line of the rotary tool face (Parameter No. 5028)

B: Distance from the rotation center of the turret to the gauge line of the turning tool face (Parameter No. 5029)

C: Distance from the center of the rotary tool pot to that of the turning tool pot (Parameter No. 5030)

D: Distance from the rotation center of the turret to the center of the rotary tool pot (Parameter No. 5033)

(Note 1) Set a radius value for all of A, B, C, and D.

(Note 2) Set each parameter, assuming that the arrow direction indicates a positive value.
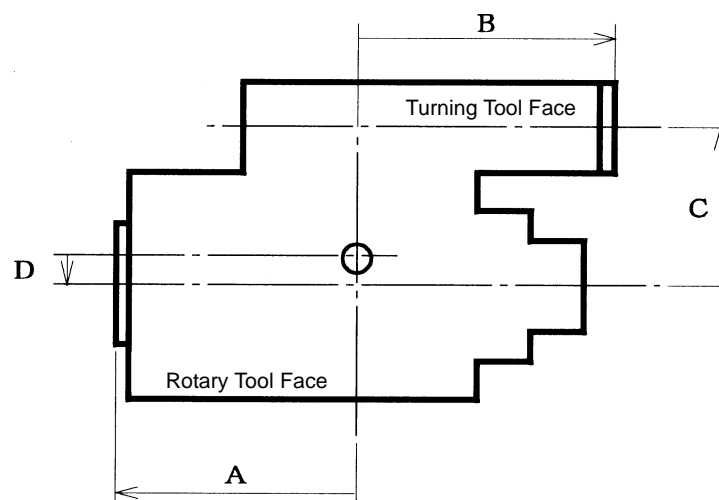
10.4.5 Turret Angle Command

With the ATC Type-E, you can perform oblique machining by giving an arbitrary angle with a turret angle command. This command, however, can be given only to the rotary tool face.

With the turret angle being zero degree when the rotary tool face is directed to the -Z side, give the command, assuming the right-handed screw direction to be a positive value, facing the +Y axis side. An angle range is 0=<B=<180.

A command format is as follows:

T02tt00 <u>Bxx</u>

Angle Command

The turret angle command can be given as a program command in the MDI or memory mode. Manual indexing is allowed only to 0, 90, and 180 degrees.
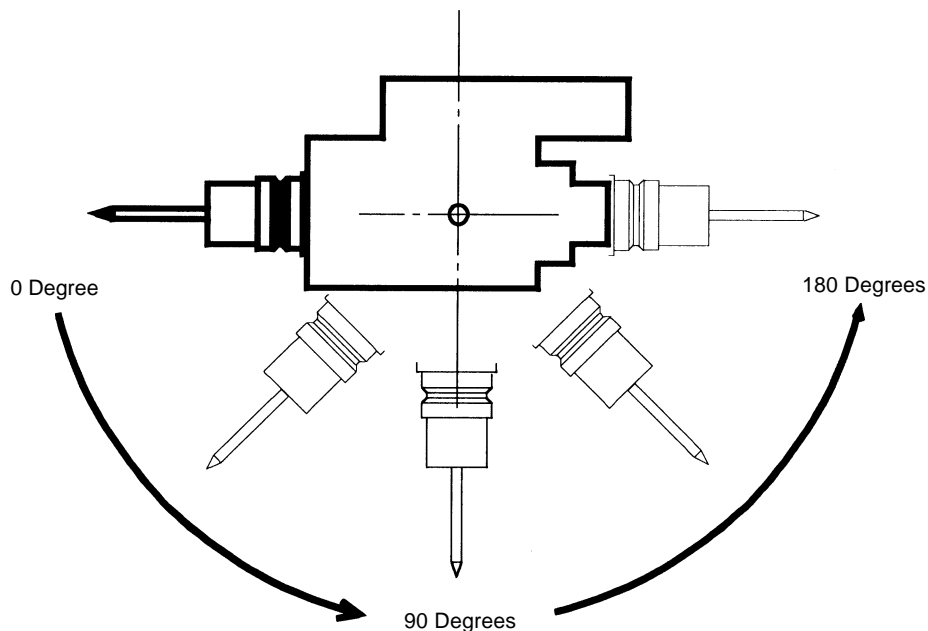
10.4.5.1 Tool Offset Conversion

Tool offset(X1, Z1) is converted as follows by the turret angle command.

$X1 = X\cos\theta + 2Z\sin\theta$

$Z1 = -(1/2) X\sin\theta + Z\cos\theta$

    X: X-axis geometry offset + X-axis wear offset

    Z: Z-axis geometry offset + Z-axis wear offset

    θ: Turret angle command

    (Note) When the turret angle command is 90 or 180 degrees, tool offset setting(Tool Offset screen) is altered; it is not altered at other angles.

10.4.5.2 Tool Reference Shift Amount Conversion

The tool reference shift amount(X2, Z2) is converted as follows by thee turret angle command.

$X2 = 2 (A\sin\theta + D\cos\theta - D)$

$Z2 = A\cos\theta + D\sin\theta - A$

    A: Distance from the rotation center of the turret to the gauge line of the rotary tool face

    D: Distance from the rotation center of the turret to the center of the rotary tool pot

    θ: Turret angle command

10.4.5.3 Tool Nose Point Conversion

The tool nose point is converted as follows when the turret angle command is 90 or 180 degrees.

Turret Angle	Change of Virtual Tool Nose Point									
90 degrees	Before conversion	0	1	2	3	4	5	6	7	8	9
	After conversion	0	2	3	4	1	6	7	8	5	9
180 degrees	Before conversion	0	1	2	3	4	5	6	7	8	9
	After conversion	0	3	4	1	2	7	8	5	6	9

10.4.6 ATC Type-E Tool Nose Coordinate System

The tool nose coordinate values for the ATC Type-E are obtained by the following formulas:

    X-axis: Work shift amount [X0] + Machine coordinate [Xm] - (Tool offset [X] + Tool reference shift amount [Xs]) + (External work origin offset [Xe])

    Y-axis: Work shift amount [Y0] + Machine coordinate [Ym] - Tool offset [Y] + (External work origin offset [Ye])

    Z-axis: Work shift amount [Z0] + Machine coordinate [Zm] - (Tool offset [Z] + Tool reference shift amount [Zs]) + Work length [K] + (External work origin offset [Ze])

    C-axis: Work shift amount [C0] + Machine coordinate [Cm] + (External work origin offset [Ce])

    ZB-axis: Work shift amount [ZB0] + Machine coordinate [ZBm] + (External work origin offset [ZBe])

(Note 1) For the tool offset and tool reference shift amount, see 10.4.4, 10.4.5.1, and 10.4.5.2.

(Note 2) For the ATC Type-E, the tool offset + tool reference shift amount corresponds to the tool offset in tool nose coordinate calculation for the conventional ATC.

(Note 3) The tool reference position refers to the gauge line of the rotary tool face(Z-axis direction) and center of the rotary tool pot(X-axis direction) when the rotary tool face is indexed to the spindle side; the position where the tool reference shift amount will be zero(0).

Work Shift Amount [Z0]
Machine Coordinate [Zm]
Machine Origin
Tool Reference Shift Amount [Zs]
Tool Reference Position
Machine Coordinate [Xm]
Tool Reference Shift Amount [Xs]
Work Shift Amount [X0]
Tool Offset [X]
Machining Origin
X-axis X Coordinate Value
Work Length [K]
Z-axis Coordinate Value
Tool Offset [Z]
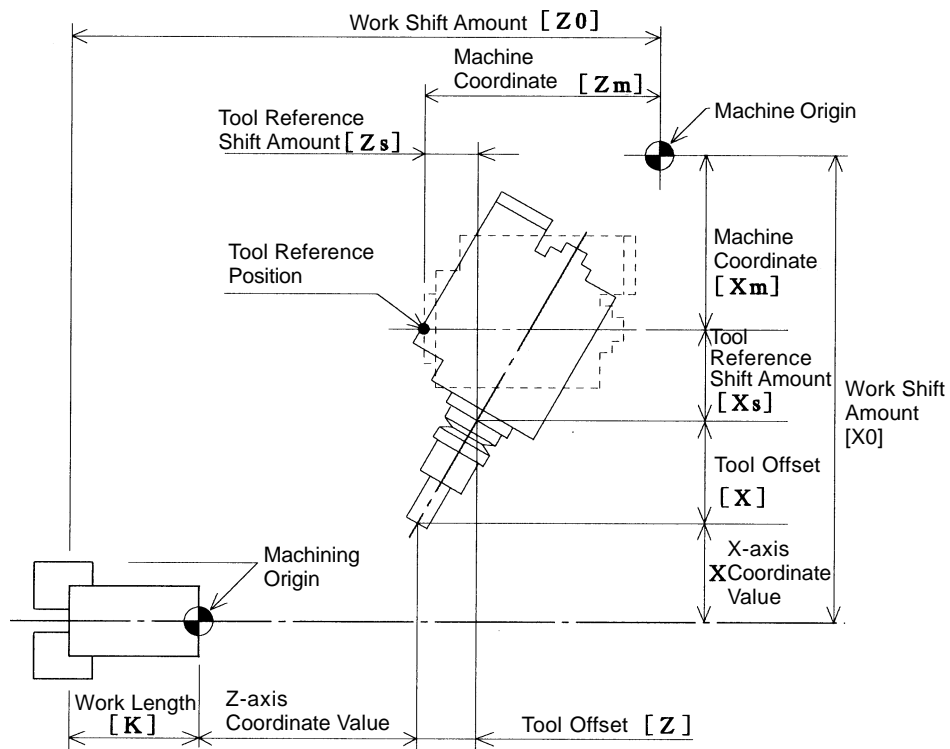
Z-X Plane Coordinate System

10.4.7   Stoke Limit 2/3 Switching

A tool prohibited area is set as a stroke limit value, using a machine coordinate value. Normally, the tool prohibited area is set, considering the tool length. For the ATC Type-E, however, the tool or turret interference position changes depending on the turret indexing angle.

Stroke Limit Value
Machine Origin
Stroke Limit Value
Machine Origin
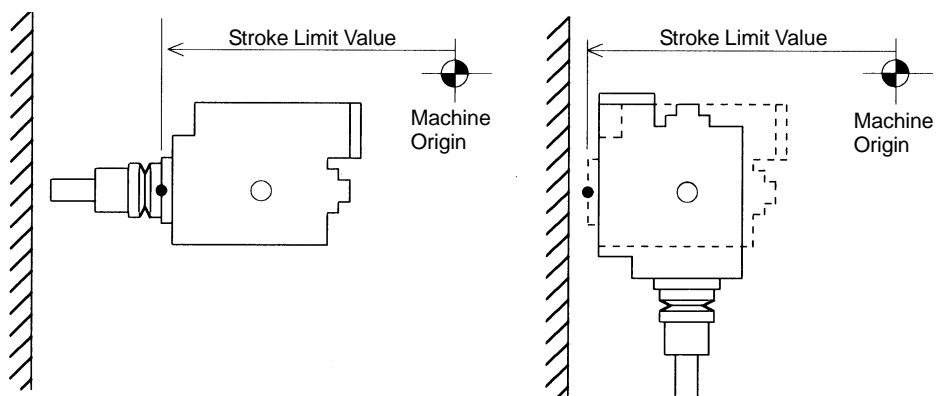
For the ATC Type-E, therefore, alter the values of the stroke limits 2 and 3. The stroke limit parameters(No. 1322 to No. 1325) are altered to the values for each area at the time of a T-code command or manual indexing.

Divide the turret indexing angle θ into the following areas;

Area A ($0 \leq \theta <45$) Area

B ($45 \leq \theta \leq 135$)

Area C ($135 \leq \theta \leq 180$)

and set the stroke limit values according to the respective angles.

(Note) The turret indexing angle is 0 degree when the rotary tool face is directed to the spindle, 90 degrees when directed downward, and 180 degrees when the turning tool face is directed to the spindle(rotary tool face directed to the back spindle).

Area A   Area C

Area B
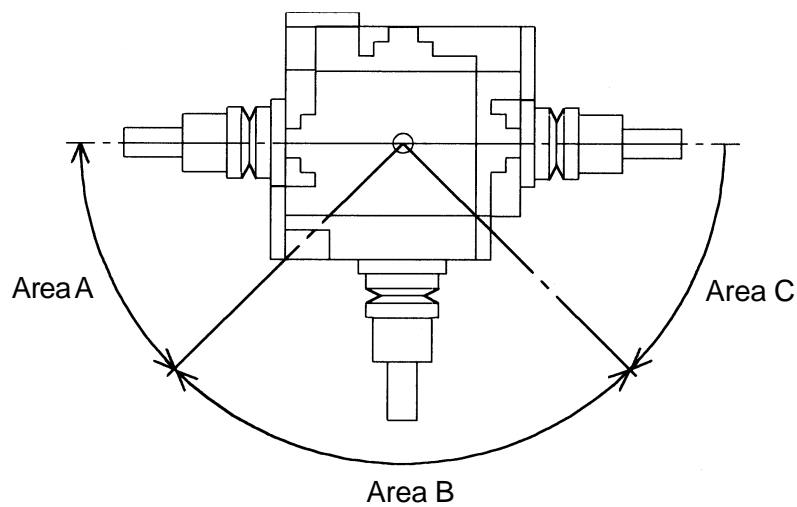
10.4.8   Associated Alarms

No. 182   The T-code command has an error.

(#109)   Turret internal tool change has been specified.

(#150)   The machining mode is inconsistent with the T-code command.

(#151)   The turret angle command value has an error.

(#152)   The turret angle command has been given in other than T02xxxx.

(#154)   The type unset tool has been specified.

10.4.9   Associated Parameters

No. 1006, #6 = 0   For the ATC Type-E, the stroke limits 2 and 3 are not altered depending on the turret indexing angle.

             = 1   For the ATC Type-E, the stroke limits 2 and 3 are altered depending on the turret indexing angle.

No. 5006, #0 = 0   Turret angle command in increments of 45 degrees

             = 1   Turret angle command in increments of 5 degrees

No. 1100   Stroke limit 2 plus directional machine coordinate value for area A

No. 1101   Stroke limit 2 minus directional machine coordinate value for area A

No. 1102   Stroke limit 2 plus directional machine coordinate value for area B

No. 1103   Stroke limit 2 minus directional machine coordinate value for area B

No. 1104   Stroke limit 2 plus directional machine coordinate value for area C

No. 1105   Stroke limit 2 minus directional machine coordinate value for area C

No. 1106   Stroke limit 3 plus directional machine coordinate value for area A

No. 1107   Stroke limit 3 minus directional machine coordinate value for area A

No. 1108   Stroke limit 3 plus directional machine coordinate value for area B

No. 1109   Stroke limit 3 minus directional machine coordinate value for area B

No. 1110   Stroke limit 3 plus directional machine coordinate value for area C

No. 1111   Stroke limit 3 minus directional machine coordinate value for area C

No. 1322   Stroke limit 2 plus directional machine coordinate value

No. 1323   Stroke limit 2 minus directional machine coordinate value

No. 1324   Stroke limit 3 plus directional machine coordinate value

No. 1325   Stroke limit 3 minus directional machine coordinate value

No. 5028   Distance from the rotation center of the turret to the gauge line of the rotary tool face

No. 5029   Distance from the rotation center of the turret to the gauge line of the turning tool face

No. 5030   Distance from the center of the rotary tool pot to that of the turning tool pot

No. 5033   Distance from the rotation center of the turret to the center of the rotary tool pot

11. MISCELLANEOUS FUNCTION

11.1 Miscellaneous Function (M Function)

If the address M followed by an up to 8-digit numerical value (M00000000 to M99999999) is specified, the BCD 8-digit code signal (M11 ~ M88) and strobe signal (MF) are output to the machine side.

11.1.1 Specify a Set of M Command in One Block.

11.1.2 The Following M Commands are Used as Special Functions.

(1) 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.

(2) M01 (Optional stop)

If the OPTIONAL 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.

(3) M02 (Program end)

Specify M02 at the end of the program. If M02 is specified during automatic operation, automatic operation stops after executing that block, and the NC unit is reset.

(4) M30 (Progam end)

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

(5) M98 (Subprogram call)

Call a subprogram with the command; M98P_ ;.

(6) M99 (Subprogram end)

Specify M99 at the end of the subprogram.

(Note) For uses of M98 and M99, refer to "21. CUSTOM MACROS"

11.1.3 Sample Program

G97 S1000 M03 ;	Spindle forward rotation
G00 X50. Z10. ;	
G99 G01 Z0 F1.0 ;	
:	
M05 ;	Spindle stop
M30 ;	Program end

11.1.4 Cautions

(1) If M00, M01, M02, or M30 is specified, the NC unit stops prereading.
Also, the arbitrary code M set in the parameter can halt the advanced reading.
(Note) Parameter settings can make the following M codes the fixes M codes that halt advanced reading.

- M12 : Work counter
- M31 : No material check
- M32 : Top cut check
- M33 : Top cut reset

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

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

11.1.5 Associated Parameters

No.3404, #0 = 0 Fixed M code for stopping pre-reading is disabled.
          = 1 Fixed M code for stopping pre-reading is enabled.

No.3434~3441 M code for stopping pre-reading (individual setting)

No.3442~3449 Minimum value/maximum value for the M code group for stopping pre-reading (Group setting)

11.2   2nd Miscellaneous Function

If an up to 8-digit numerical value (0 to 99999999) is specified subsequent to the address (A, B, or C) set with a parameret, the BCD 8-digit code signal (B11 ~ B88) and strobe (BF) are output to the machine side.

11.2.1   Specify a Set of 2nd Miscellaneous Function in One Block.

11.2.2   Cautions

(1) Set the command address of the 2nd miscellaneous function for the parameter.

(2) When a negative value is given to the second miscellaneous function, it is treated as a positive value.

11.2.3   Associated Parameters

No.1020   Command adress of the 2nd miscellaneous function.

12. COMPENSATION FUNCTION

12.1 Automatic Tool Nose Radius Compensation and Cutter Compensation

Provides both tool nose radius compensation and cutter compensation functions in order to meet needs for multifunctional NC lathes.

12.1.1 G Codes to Change Over the Type of Compensation

The following G codes are used to enable tool nose radius compensation or cutter compensation.

G140 : Tool nose radius compensation/cutter compensation cancel mode
G143 : Tool nose radius compensation enable mode
(G144 : Tool nose radius compensation enable mode)
G145 : Cutter compensation enable mode

These G codes are modal ones of Group 27.

(Note 1) *G144 is provided for compatibility with the SEICOS-LII ; it is treated as G143 inside the NC unit.*

(Note 2) *The G143 mode is selected at power-on or reset time.*

12.1.2 Setting of the Tool Nose Radius and Tool Diameter

Set the tool nose radius data and tool diameter data at "R (Profile), Q (Wear)" of the Tool Offset screen.

The value of R + Q is treated as a tool nose radius value when the virtual tool point(T) is 1-8, and as a tool diameter value when it is 9.

That is, R + Q (tool nose radius) is treated as 0 in the G143 mode when the virtual tool nose point is 0 or 9, and R + Q (tool diameter) as 0 in the G145 mode when the virtual tool nose point is other than 9.

(Note) *When both tip radius compensation and tool compensation are used in one tool, as in the ball end mill, the tool diameter compensation can be rendered effective by parameters even if the virtual tip point is not 9.*

Parameter No. 5003

#7 = 0 : Tool diameter compensation is effective only when the virtual tip point is 9

#7 = 1 : Tool diameter compensation is effective when virtual tip point is between 1 to 9

12.2 Automatic Tool Nose Radius Compensation

When normally programming, the tool nose is assumed to be one point, but the actual cutters have a tool nose radius.

The tool nose radius is irrelevant when cutting parallel to the axis such as end face, outer diameter, inner diameter, etc. When chamfering or cutting an arc or a grade, however, this tool nose radius causes insufficient or excessive cut.

Automatic tool nose radius compensation makes the NC unit perform computation inside it and controls the tool nose automatically to prevent insufficient or excessive cut.
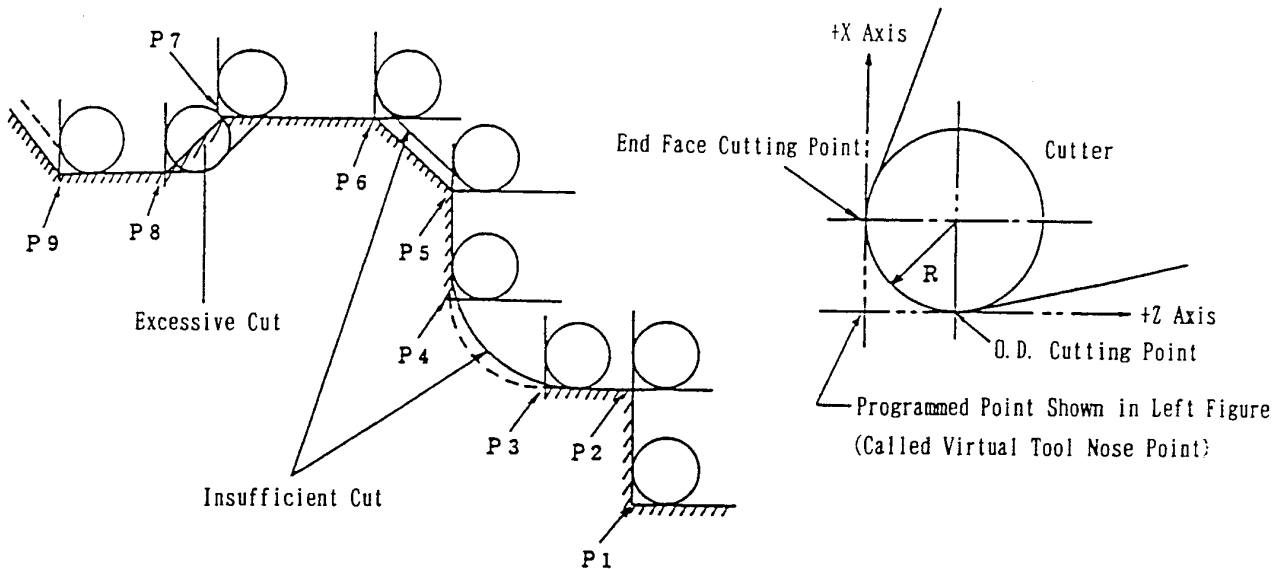
P1-P9 are original programmed points

12.2.1 Virtual Tool Nose Point Input

Set the virtual tool nose point viewed from the center of the tool nose radius for "T" of the Tool Offset screen. (T = 0-9)
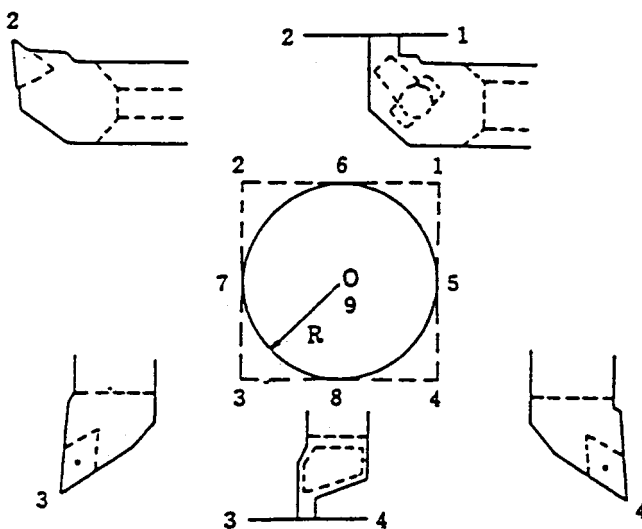
<Virtual Tool Nose Point>
This refers to the programmed
point of the tool nose radius.
Depending on this selection,
the programming method or
tool offset method will differ
completely.

12.2.2 Automatic Tool Nose Radius Compensation State

Automatic tool nose radius compensation has the following three states.

State	Tool Path	Material
Offset cancel state	Moves on the programmed path.	
Offset to left	Moves on the left side of the programmed path advance direction	Right side of the advance direction
Offset to right	Moves on the right side of the programmed path advance direction	Left side of the advance direction

When performing automatic tool nose radius, the program repeats the cycle of starting at the tool nose radius compensation cancel state and returning to the compensation cancel state again through the tool nose radius compensation state.

Compensation Cancel ⟶ Compensation State
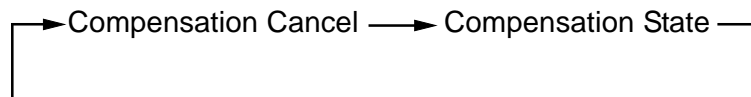
They are divided into three states and each block is called as follows.
(1) Start-up block:
Block to change over from rapid traverse to cutting feed (G00 → G01, G02, G03)
(2) Offset block:
Cutting feed continuous block (G01, G02, G03 → G01, G02, G03)
(3) Cancel block:
Block to change over from cutting feed to rapid traverse (G01, G02, G03 → G00)

12.2.3 Automatic Tool Nose Radius Compensation Direction

<Compensation direction: Right>
The tool nose is located to the right of the material in the advance direction.
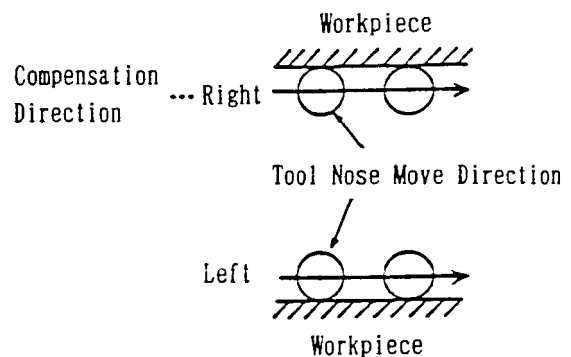
<Compensation direction: Left>
The tool nose is located to the left of the material in the advance direction.

12.2.4 Determination of the Compensation Direction of the Block Under Cutting

The automatic tool nose radius compensation direction under cutting is determined as shown in the table on the next page.

In case of single axis return move, however, the axis returns at the position where the tool nose does not exceed the vector which is set perpendicualr to the move axis.

(Example 1) For the tool nose point 3
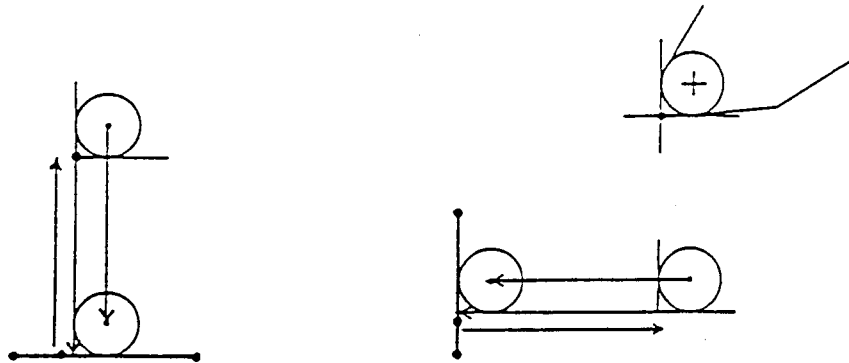
(Example 2) For the tool nose point 8

<<List of Compensation Directions (During Cutting)>>

Tool Nose Point \ Move Direction	→	↗	↑	↖	←	↙	↓	↘
1	Right	-	Left	Left	Left	-	Right	Right
2	Right	Right	Right	-	Left	Left	Left	-
3	Left	-	Right	Right	Right	-	Left	Left
4	Left	Left	Left	-	Right	Right	Right	-
5	-	Left	Left	Left	-	Right	Right	Right
6	Right	Right	-	Left	Left	Left	-	Right
7	-	Right	Right	Right	-	Left	Left	Left
8	Left	Left	-	Right	Right	Right	-	Left
0.9	×	×	×	×	×	×	×	×

(Note 1) *"-" denotes the same compensation direction as the previous block because the compensation direction cannot be determined. It is also "-" for an arc command.*

(Note 2) *When the virtual tool nose point is 0 or 9, tool nose radius compensation is not applied.*

(Note 3) *"Under cutting" refers to being in the G01/G02/G03 mode.*

<Examples of Tool Move Including the Block Where the Compensation Direction Cannot Be Determined>   For virtual tool nose point = 3

(Example 1)

	(Move Direction)	(Compensation direction)
[1]	↓	Left
[2]	↙	Left
[3]	↓	Left

* The compensation direction at [2] is the same as that at [1].

(Example 2)
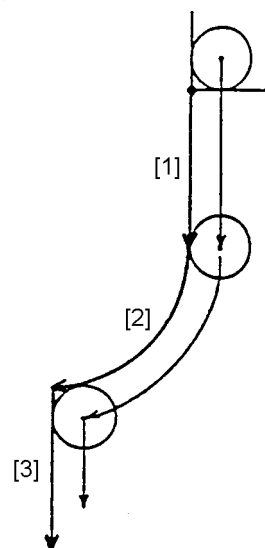
	(Move Direction)	(Compensation direction)
[1]	↓	Left
[2]	↺	Left
[3]	↓	Left

* The compensation direction at [2] is the same as that at [1].

(Example 3)
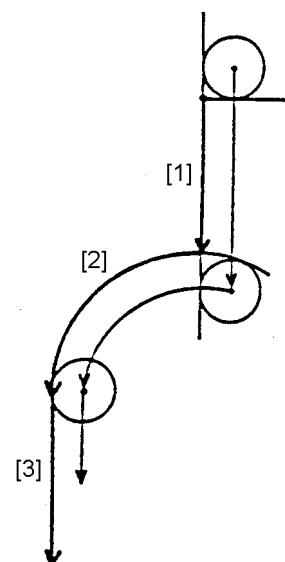
	(Move Direction)	(Compensation direction)
[1]	↓	Left
[2]	↻	Left
[3]	↓	Left

* The compensation direction at [2] is the same as that at [1].

12.2.5 Forced Determination of the Compensation Direction

Normally, the tool nose radius compensation direction is automatically determined as shown in the table on the previous page.

However, you may want to apply tool nose radius compensation in the direction different from normal cutting, such as approach for indentation machining, etc.

In such a case, you can specify the tool nose radius compensation direction by the program to forces determination of the compensation direction.

<<How to Specify the Compensation Direction>>
Use the following G codes:
  G141 : Automatic tool nose radius compensation to left
  G142 : Automatic tool nose radius compensation to right

(Note 1)   *Effective to the G01, G02, and G03 move blocks only.*

(Note 2)   *Only one specified block is effective.*

(Note 3)   *When the compensation direction in indefinite in the block next the specified one, the same compensation direction as the specified one is assumed.*

<Examples of Forced Determination of Compensation Direction>

For virtual tool nose point = 3

(Example 1)

	(Move Direction)		(Compensation direction)
		G141 (G142) Unspecified	G142 Specified at [1]
[1]	↓	Left	G142 (Right)
[2]	↙	Left	Right
[3]	↓	Right	Right

* Full line (—) : When G141 (G142) is not specified

Broken line (- - -) : When G142 is specified at [1]

(Example 2)

	(Move Direction)		(Compensation direction)
		G141 (G142) Unspecified	G142 Specified at [1]
[1]	↓	Left	G142 (Right)
[2]	⌒	Left	Right
[3]	←	Right	Right

* Full line (—) : When G141 (G142) is not specified

Broken line (- - -) : When G142 is specified at [1]

(Example 3)

	(Move Direction)		(Compensation direction)
		G141 (G142) Unspecified	G142 Specified at [1]
[1]	↓	Left	G142 (Right)
[2]	⌒	Left	Right
[3]	←	Left → Right	Right

* Full line (—) : When G141 (G142) is not specified

Broken line (- - -) : When G142 is specified at [1]

12.2.6 Determination of the Compensation Direction of Start-up and Cancellation Blocks (Approach, Release)

At approach or release time, determine the automatic tool nose radius compensation as follows.

(1) i) When the specified stroke is X/2 > Z(*1), the vector is created parallel to the Z axis.
   ii) When the specified stroke is X/2 ≦ Z(*2), the vector is created parallel to the X axis.

(2) Determine "±" viewing the compensation direction of the move axis. (determination of the virtual component direction)
   (※)When G141/G142 has been specified, determine "±" viewing the G141/G142 compensation direction.

(3) Determine the virtual component compensation direction (right, left).

(4) Make a calculation.

   (Note 1) *The following results when the move command at (1) is of an arc.*

   i) *When the vector in the tangent direction of the arc has an angle larger than 45° with respect to the Z axis, the vector is created parallel to the Z axis.*

   ii) *When the vector in the tangent direction of the arc has an angle of 45° or less with respect to the Z axis, the vector is created parallel to the X axis.*

   iii) *The vector in the tangent direction of the arc is created from the arc start point in case of approach, and from the arc end point in case of release.*

   (Note 2) *When the compensation direction of the move axis cannot be determined at (1), determine "±" viewing the compensation direction of the next move axis in case of approach, and that of the previous axis in case of release respectively. (same for the arc command)*

   (Note 3) *When the compensation direction of the virtual component cannot be determined at (3) (for the virtual tool nose point 5-8), the same compensation direction as that at (2) is assumed.*

   (Note 4) *The start-up block and cancellation block refer to those switching from G00 to G01/G02/G03 (approach), and vice versa (G00), respectively.*

   (※1) X/2 > Z ----When the move direction is larger than 45° with respect to the Z axis.

   (※2) X/2 ≦ Z ----When the move direction is 45° or less with respect to the Z axis.

(Example 1) For virtual tool nose point = 3

[1] The vector is created parallel to the X axis because of X/2 < Z.

Move Axis
30°
Vector Parallel to X Axis
Virtual Vector
Tool Nose Point 3
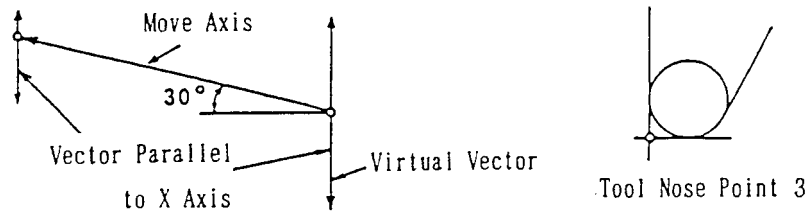
[2] Determine the vector in the same direction as the compensation side of the move axis (+X axis because of the right direction).

Compensation Direction Right
+X Axis
+X Axis
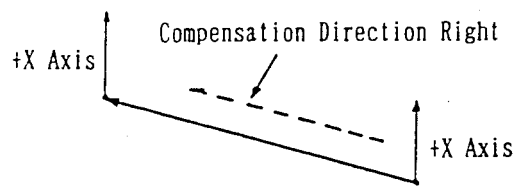
[3] Determine the compensation side of the virtual vector, and then, its intersecting point.
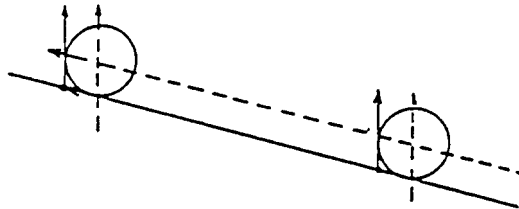
(Example 2) For virtual tool nose point = 6

[1] The vector is created parallel to the X axis because of X/2 < Z.

Move Axis
Virtual Vector
Tool Nose Point 6
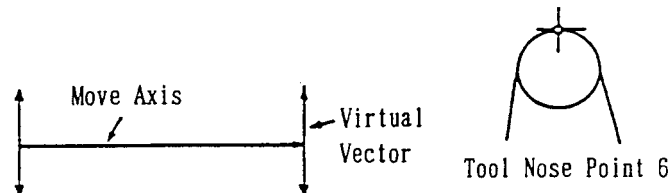
[2] Determine the vector on the -X side because the compensation side of the move axis is right.

Right Side
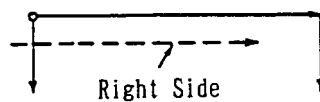
※[3] Since the compensation side of the virtual vector cannot be determined, specify it in the same direction as the compensation side of the move axis.

Same Compensation
Direction (Right Side)

12 - 10
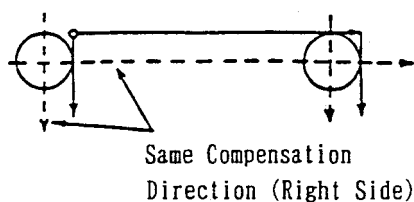
(Example 3) A. For virtual tool nose point = 3

① (Line Parallel to Z Azis)

Virtual Vector

Larger than 45°

G01

G00

②(Perpendicular to Z Axis)

Tool Nose Point 3

G00

G01

45° or Less

Virtual Vector

G00

B. For virtual tool nose point = 8

G00

Virtual Vector

②

45° or Less

G01

G01

Virtual Vector

Larger than 45°

G00

G01

①

Virtual Vector
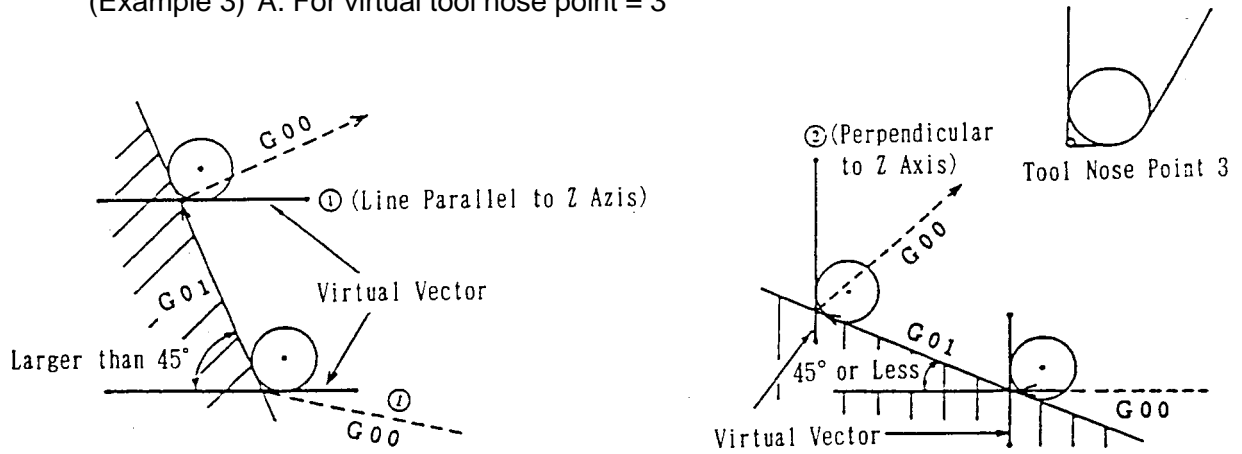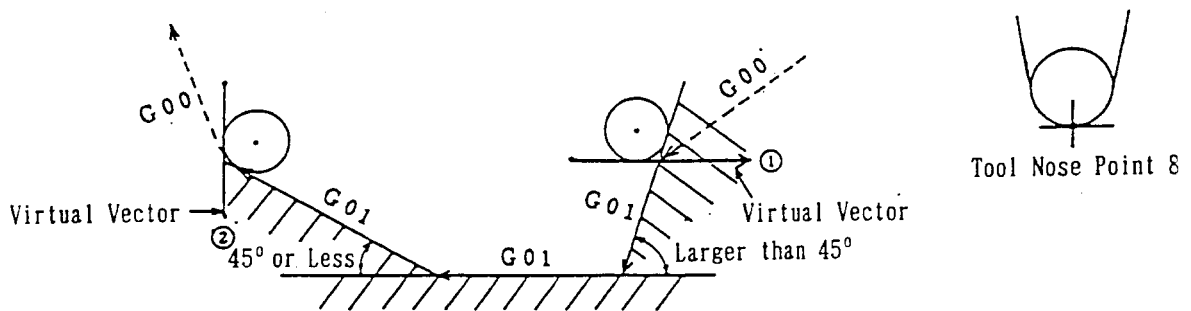
Tool Nose Point 8

(Example 4) For virtual tool nose point = 3 (G00 G02 G03)

G00

G03

G00

/G00

G02

G00

G00

45° or Less

G03

Larger than 45°

①

G03

12 - 11
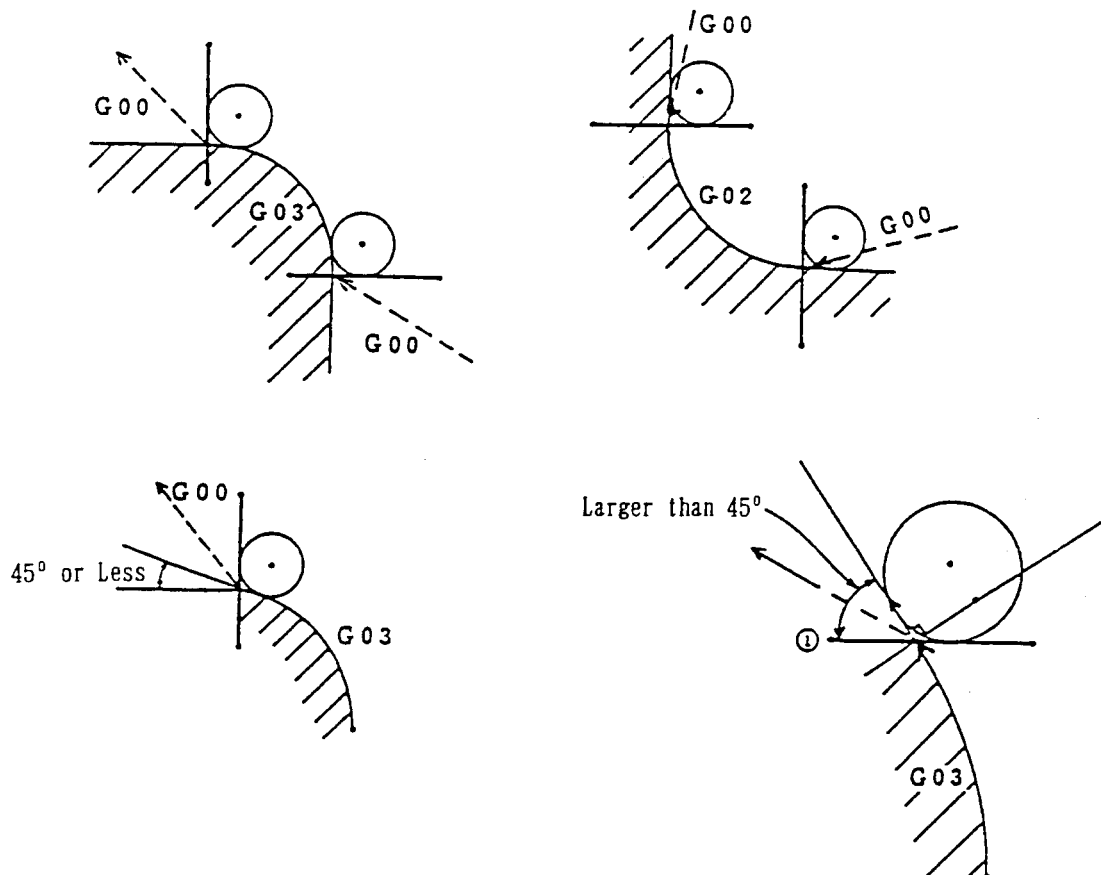
12.2.7 Start-up and Cancellation Conditions

The NC unit has a 4 blocks worth of buffer for automatic tool nose radius compensation(cutter compensation).

This buffer is used for calculation of automatic tool nose radius compensation.

N1 .... ;
N2 .... ;
N3 .... ;
N4 .... ;

(1) Start-up conditions

When all of the following conditions are met in the cancellation state, automatic tool nose radius compensation starts up in the current block.

a) G143 mode

b) Virtual tool nose point (T) = 1-8

c) Tool nose radius value $(R + Q) \neq 0$

d) The current block is G00 and there is an axis move within the plane.

e) The next block is G01/G02/G03, there is an axis move within the plane, and the compensation direction can be determined. (including when G141/G142 is specified)

(Note 1) *G00 for the condition (d) can be replaced by G01 by parameter setting. When started up in the G01 block, a start-up interlocking point calculation will be the same as an interlocking point calculation in the mode, being different from the case of G00.*

(Note 2) *When the conditions of the next block are not met, those of the 2nd next and 3rd next blocks are checked sequentially. When there is a block other than G01/G02/G03, a condition check is cancelled for the subsequent blocks. When the conditions are not met for the 3 blocks ahead, automatic tool nose radius does not start up.*

(2) Cancellation conditions

When any one of the following conditions is met in the compensation state, compensation is cancelled.

a) G00 has an axis move within the plane.

b) Coordinate system setting with a T command

c) Not G143 mode

12.2.8 Cautions

(1) The intersecting point of the tool nose radius center between each block is calculated in the same manner as that of the tool center at the time of cutter compensation.
In the following cases, however, a special calculation method is used. (stated earlier)

   a) Start-up and cancellation blocks

   b) G01 single axis return block

(2) An interference check also conforms to cutter compensation.

(3) With a G39 command, the tool can be moved along an arc when moving around outside the corner larger than 180° , similar to cutter compensation.

(4) Automatic tool nose radius compensation is not performed by MDI operation.

(5) Although ordinary start-up is performed only in G00 block, it is made possible in G01 block through parameter setting.
When start-up is made in G01 block, however, the method to calculate the intersecting point of the nose R center becomes different from that for G00, being made the same with the intersecting point in mode.

(6) When 3 or more blocks free of shifting have appeared continuously in a mode (including those by the pre-reading stop command), a vertical offset vector is usually created at the end of the current block in the same way as in tool diameter compensation. Through parameter setting, however, the offset vector can be temporarily eliminated (offset vector temporary clear).

   (※) For (1) through (3) and the general cautions during compensation, refer to "12.5 Cutter Compensation".

12.2.9 Associated Parameters

No.5003, #1 = 0   Interference check: Enabled if offset vector difference is 90° to 270°

           = 1   Interference check: Disabled if offset vector difference is 90° to 270°

No.5003, #2 = 0   Interference check: All enabled

           = 1   Interference check: All disabled

No.5005, #0   When 3 or more blocks free of shifting have appeared continuously (including those by the pre-reading stop command):

           = 0   Vertical offset vector is created at the end of the current block.

           = 1   Offset vector is temporarily cleared at the end of the current block.

No.5005, #1 = 0   Start-up takes place in G00 block.

           = 1   Start-up takes place in G00 or G01 block.

No.5025   Limit value at which small shift amount is ignored in corner outside shifting.

12.2.10 Associated Alarms

No.117	Excessive cut occurred during tool nose radius compensation.
(#001)	Arc radius is smaller than the compensation amount.
(#002)	Others
No.118	An intersecting point does not exist in tool nose radius compensation.
No.119	An erroneous command has been given while in tool nose R compensation.
(#001)	A plane has been changed in the current block.
(#002)	A plane has been changed in the next block.
(#003)	An erroneous G code has been specified in the current block.
(#004)	An erroneous G code has been specified in the next block.
(#005)	G39 has been specified except in G00 or G01 in the current block.
(#006)	G39 has been specified except in G00 or G01 in the next block.
(#008)	Axial shift within a plane has been specified in G39 command.

12.3 Groove Width Compensation (G150, G151, G152)

When a groove cutting tool is used, programming is done with one of the virtual tool noses(for example, 4) and an offset is input. Also, it is also necessary to offset the other virtual tool nose (for example, 4). At this time, this function performs offsetting the groove width by specifying a G code.

12.3.1 G Codes

G150 : Groove width compensation cancel

G151 : Groove width compensation ON for end face/back end face (X coordinate system shift)

G152 : Groove width compensation ON for outer/inner diameter (Z coordinate system shift)

12.3.2 Description of Compensation

(1) Compensation ON

A G151/G152 command reads the tool width (H + J) and virtual tool nose point (T) in the Tool Offset screen, shifts the coordinate system by the tool width, and changes over the virtual tool nose point.

If groove width compensation (G151, G152) is specified, the virtual tool nose point is changed as shown in the figure.

For example, the G151 command changes the virtual tool nose point from 1 to 4, and changes to 1, if it is 4.

(2) Compensation cancel

Compensation is cancelled by G150 or a T command. (Both coordinate system and virtual tool nose point are restored to the original ones.)

```
= = = = Groove Width Compensation for End Face (G151) = = = =            = = = = Groove Width Compensation for
                                                                          Inner/Outer Diameter (G152) = = = =
              ①              ②

┌──────────────────┐                     ┌──────────────────┐                        ┌────────────────┐
│                  │         ↕      ↕     │                  │                        │   Compensation │
│ Compensation for │                     │ Compensation for │                        │   for O.D. ON  │
│ Back End Face ON │                     │ End Face ON      │                        │                │
│                  │                     │                  │                        └────────────────┘
└──────────────────┘                     └──────────────────┘                         ③   ←──────→   ④
              ④              ③
```
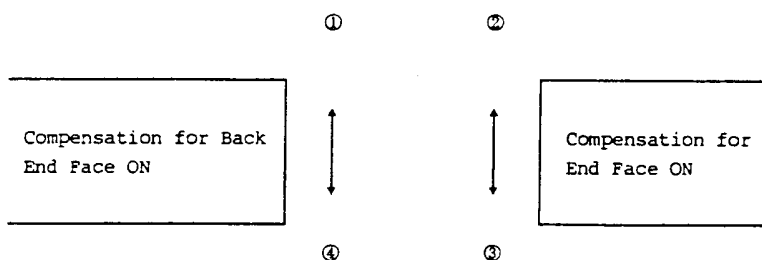
These ①,②,③,④ are indicative of virtual tool nose.

```
                                                                          ②   ←──────→   ①
                                                                         ┌────────────────┐
                                                                         │   Compensation │
                                                                         │   for I.D. ON  │
                                                                         │                │
                                                                         └────────────────┘
```
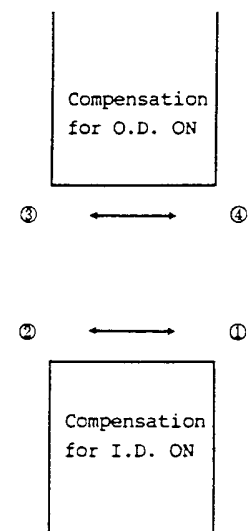
Command	Change in tip point	Change in coordinate system
G151	1 → 4	X → X - (H+J)
	2 → 3	X → X - (H+J)
	3 → 2	X → X + (H+J)
	4 → 1	X → X + (H+J)
G152	1 → 2	Z → Z - (H+J)
	2 → 1	Z → Z + (H+J)
	3 → 4	Z → Z + (H+J)
	4 → 3	Z → Z - (H+J)

12.3.3 Sample Program (Virtual Tool Nose Point = 3)

```
       G18 G00 X100.0 Z-50.0 ;
N1     G99 G01 X50.0 F0.5 ;
N2     Z-40.0 ;
N3     G00 X100.0 ;
N4     G152 ;
N5     G00 Z-30.0 ;  ┐
N6     G01 X50.0 ;    │ Groove width being compensated
N7     Z-40.0 ;       │
N8     G00 X100.0 ;  ┘
N9     G150 ;
```
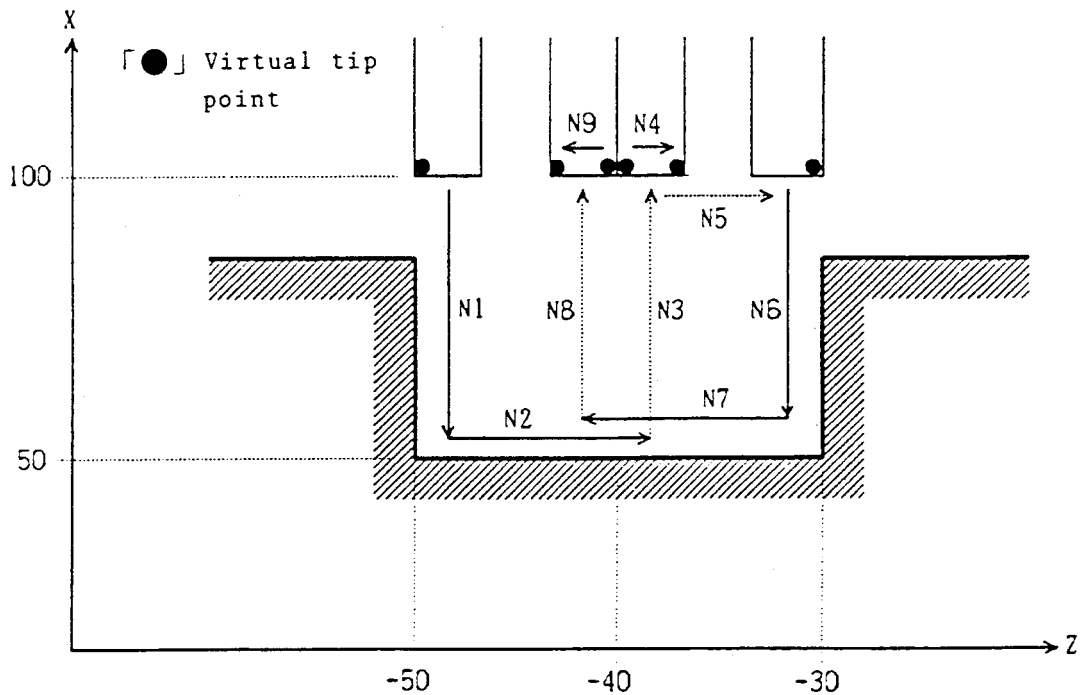
12.3.4 Cautions

(1) An alarm results if the virtual tool nose point is other than 1-4.

(2) When G151/G152 is specified continuously in the program, cancel current compensation and apply new compensation.

(3) When G150-G152 and an axis move command are specified simultaneously, or when the plane is changed over during compensation, an alarm results.

(4) A reset during compensation cancels compensation, but the coordinate system does not change.

(5) Even if a groove width offset command is executed at TOOL WIDTH = 0, groove width offset will not be normally applied. However, it can be applied by parameter setting. If groove width offset is applied at TOOL WIDTH = 0, only the virtual tool nose point will change, not the coordinate system, affecting automatic tool nose radius offset.

12.3.5 Associated Parameters

No.3111, #0 = 0   Does not update the tool nose point display by groove width offset.

        = 1   Update the tool nose point display by groove eidth offset.

No.3406, #4 = 0   Does not apply offset when a groove width offset command is given at TOOL WIDTH = 0.

        = 1   Applies offset when a groove width offset command is given at TOOL WIDTH = 0.

12.3.6 Associated Alarms

No.179   Error in the groove width compensation command.

 (#001)   Axial shift has been specified.

 (#002)   A plane has been changed.

 (#003)   An error exists in the tool nose point.

12.4 Multiple Offsets

Depending on the machine or machining state, one tool may require two or more offsets.
In this case, through assignment of an offset number (lower 2 or 3 digits) of the T code,
offset can be applied independently from the ordinary tool nose coordinate system setting.
Offset, in this case, is performed not by shifting of the coordinate system but by shifting by
the amount of offset.

T01<u>05</u> ;	(No.25 multiple offset ON)
G00 Z1000. T01<u>03</u> ;	(No.23 multiple offset ON)
	(Multiple offset enabled)
T01<u>00</u> ;	(Multiple offset cancel)

12.4.1 Multiple Offset Command

When the offset number (lower 2 or 3 digits) of a T code is not equal to 0 as well as the
virtual tool nose point indicated by the offset number =0 and the tool nose R =0, the T
code is taken as the multiple offset command.

When the offset number (lower 2 or 3 digits) in the T-code is 0 in the multiple offset
mode, this T-code is considered a multiple offset cancel command. The turret face
number and tool number specified with these T-codes must be the same as the numbers
selected before the respective T-codes had been given.

12.4.2 Multiple Offsets Operation

(1) Multiple Offset Start-Up Operation

    a) When T command has been independently given:

        The tool moves by the offset amount of the offset number specified with the T
        code. When an axial shift command is given henceforth, it moves with the tool
        path being shifted by this offset amount.

12 - 18
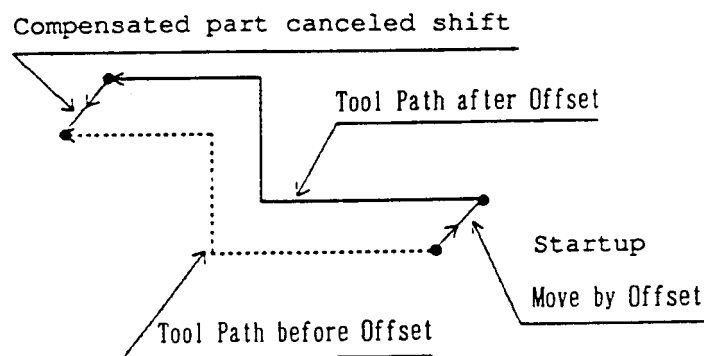
b) When it is specified in the same block as an axis move command, the tool moves by "specified axis move amount + offset amount", and the end point assumes the coordinate value shifted by the offset amount.
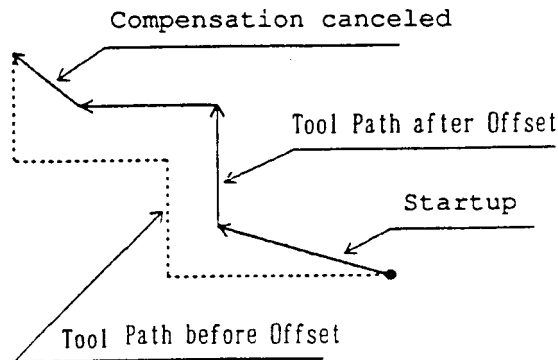
(2) Multiple Offset Cancelling Operation
   a) When the turret number and tool number of the T-command match the currently selected numbers and the offset number is 00 (T0100), the tool moves in the opposite direction by the currently applied offset amount.

       T0125 ;
       T0100 ;   (The compensated amount of compensation No. 25 returns and cancels the compensation.)

   b) When the tool number by T command is different from that currently being selected and any other tool has been called, coordinate system setting is simply performed in that tool number, where no shifting by the cancelled amount takes place.

       T0224 ;
       T0300 ;   (Set the coordinate system in tool No. 03 and cancel the compensation.)

(3) Even if groove width compensation (G151, 152) is executed, compound compensation will be retained as it is and will not be affected.

12.4.3 Example of Compound Compensation Program

```
        T0100 ;
N1      G00 X50.   Z0 ;
N2      T0123 ;                 (Compound compensation in compensation No.23)
N3      G01 Z-30.  F100 ;
N4          X70. ;
N5          Z-60.    T0125 ;    (Compound compensation in compensation No.25)
N6          X-100.   T0100 ;    (Cancel compound compensation)
N7      G00 X200. Z200. ;
```
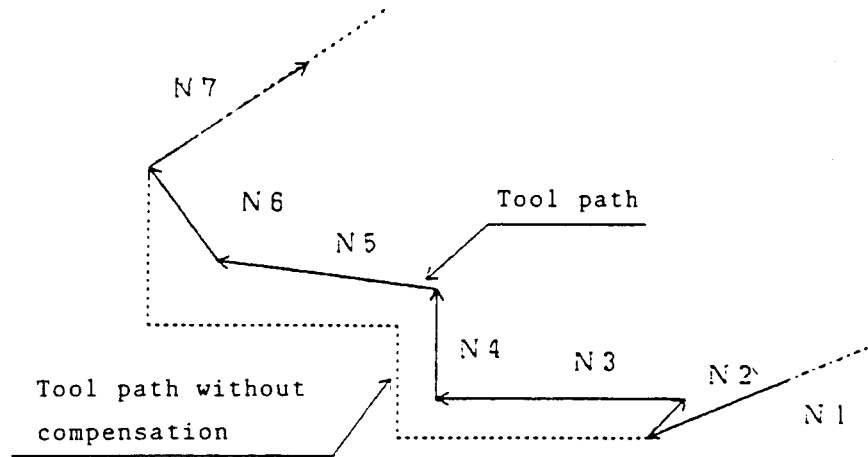

12.4.4 Cautions

(1) When the offset number (lower 2 digits or 3 digits) in the T-code is non-zero and the tool offset amount indicated by that offset number has TOOL NOSE POINT = 0 and TOOL NOSE RADIUS = 0, this T-code is treated as a multiple offset command.

To actually apply a multiple offset, the following condition must be met.

"The turret number or tool number should not be changed by the multiple offset command."

If this condition is not met, an alarm will result.

    T0200 ; .... Tool number 2, turret indexing + coordinate system setting

    T0222 ; .... Offset number 22, multiple offset

    T0424 ; .... Alarm because the tool number (turret number) changed
                          from 2 to 4

(2) When the offset number in the T-code is non-zero and the tool offset amount indicated by that offset number has TOOL NOSE POINT = 0 and TOOL NOSE RADIUS = non-zero, an alarm results because it is impossible to distinguish between multicut offset an multiple offset.

(3) When a multiple offset T-code is given in the same block as a G52 command (back machining mode selection) in the back spindle specifications, an alarm results because the multiple offset command runs counter to coordinate system setting in the back machining mode.

(4) Only a linear axis move command (G00/G01/G32/G34) can be given in the same block as multiple offset. If an axis move command and multiple offset command are executed in the circular arc (G02/G03) mode, an alarm will result.

(5) When applying multiple offset to the multicut tool, give multiple offset after executing a multicut offset command with a T-command.

      T0111 ; .... Coordinate setting with the multicut tool of the offset number 11

      T0121 ; .... Multiple offset to the multicut tool number 11 with the offset number 21

      T0100 ; .... Multiple offset cancellation after moving the tool by the offset amount of the offset number 21.

(6) Tool nose radius compensation is applied to the path to which multiple offset was applied.

12.4.5 Associated Alarms

No.182   The T-command has an error

(#001)   There is an-error in the number of digits of the T-code.

(#002)   The turret face number has an error. (Face number = 0 or > maximum number of faces)

(#003)   The tool number has an error. (Tool number = 0 or > maximum number of tool offsets)

(#004)   The offset number has an error. (Offset number > maximum number of tool offsets)

(#005)   The multiple offset command has an error. (TOOL NOSE POINT = 0, but TOOL NOSE RADIUS $\neq$ 0)

(#006)   When the ATC is attached, there is an error in a command for the turret face number in the 90s (90, 91, 99)

(#007)   The multiple offset command has an error. (In the G02/G03 mode, axis command given in the same block)

(#008)   The multiple offset command has an error. (Turret face number or tool number changed)

(#009)   The multiple offset command has an error. (Multiple offset command in the same block as G52 command)

12.5 Cutter Compensation (G38-G42)

This command can offset the tool center path to the right or left of the programmed path by the tool radius value. This function is effective when machining the outer figure or inner figure with an end mill.
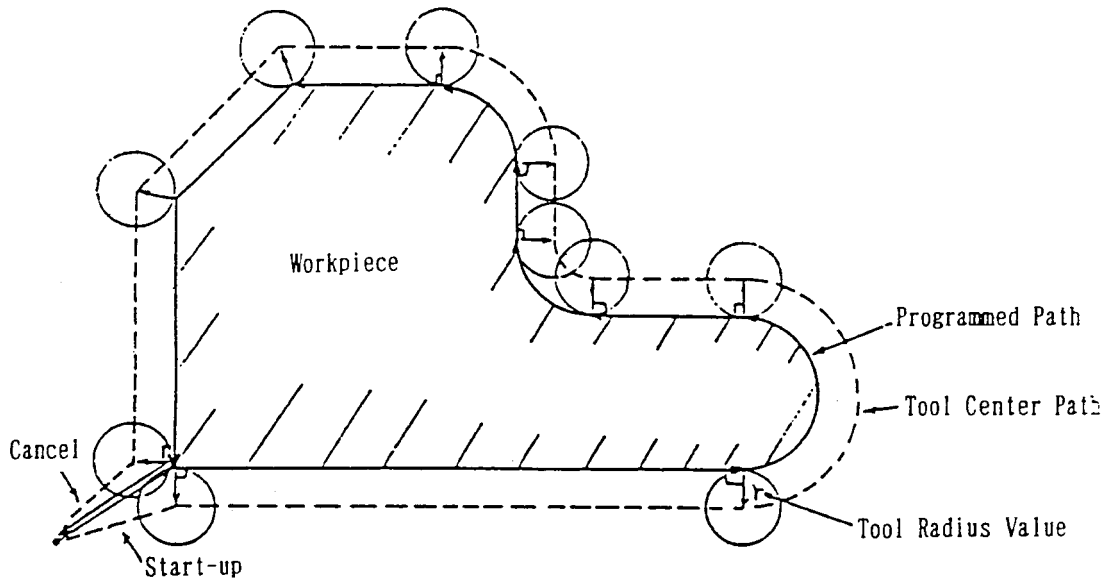
12.5.1 G Codes

G40 : Cutter compensation cancel
G41 : Cutter compensation to left
G42 : Cutter compensation to right
G38 : Cutter compensation hold/change
G39 : Cutter compensation corner arc

12.5.2 Command Format

(1) Cutter compensation start-up

$$\begin{bmatrix} G00 \\ G01 \end{bmatrix} \begin{bmatrix} G41 \\ G42 \end{bmatrix} \alpha\_ \quad \beta\_ \quad ;$$

This command starts up cutter compensation.

(2) Cutter compensation cancel

$$\begin{bmatrix} G00 \\ G01 \end{bmatrix} G40 \quad \alpha\_ \quad \beta\_ \quad ;$$

This command cancels cutter compensation.

(3) Cutter compensation vector hold/change

$\begin{bmatrix} G00 \\ G01 \end{bmatrix}$ G38 α_ β_ ;

   This command allows the offset vector for cutter compensation to be held.

$\begin{bmatrix} G00 \\ G01 \end{bmatrix}$ G38 I_ J_ K_ ;

   This command allows the offset vector for cutter compensation to be changed.

(4) Cutter compensation corner arc

   G39 I__ J__ K__ ; or G39 ;

   This command allows the tool to move along an arc at the corner.

12.5.3 Offset Plane

Specify the plane where cutter compensation is to be applied, using G17, G18 or G19.
   G17 : Xp-Yp plane
   G18 : Zp-Xp plane
   G19 : Yp-Zp plane
   where;   Xp: X axis or its parallel axis
             Yp: Y axis or its parallel axis
             Zp: Z axis or its parallel axis

12.5.4 Offset Direction

The offset direction for cutter compensation is specified with G41/G42.
   G41 :   Offset to left
   G42 :   Offset to right

Advance Direction   Advance Direction

Workpiece   Workpiece

Offset to Left   Offset to Right

12 - 23
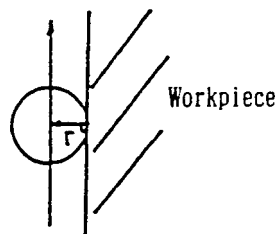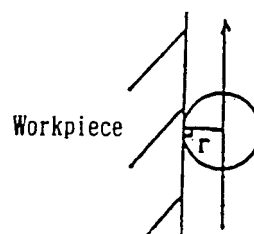
12.5.5 Sample Program for Cutter Compensation (Tool radius = 20.)

[Offset to left]

```
    G98  G00  X0  Y0 ;
N1  G17  G01  G41  X100.  Y50.  F200 ;          Start-up
N2  X200. ;                                     Offset mode
N3  G02  X300.  Y100.  150. ;                   Cancel
N4  G01  G40  X400. ;
```

[Offset to right]

```
    G98  G00  X0  Y0 ;
N1  G17  G01  G41  X100.  Y50.  F200 ;          Start-up
N2  X200. ;                                     Offset mode
N3  G02  X300.  Y100.  150. ;                   Cancel
N4  G01  G40  X400. ;
```
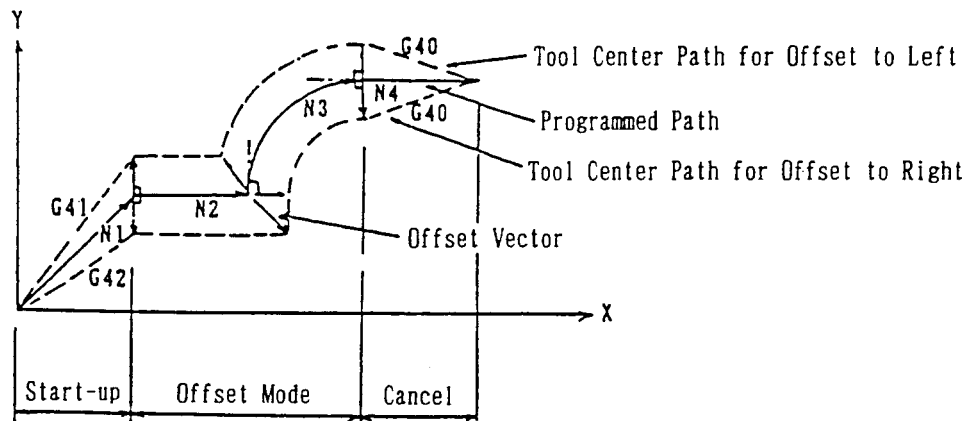

(1) Start-up:

If G41 or G42 is specified in the cancel mode, the tool moves to the position offset by the radius value.

The offset direction will be on the line which is vertical to the shift direction at the start point of the next block.

(2) Offset mode:

If you enter the cutter compensation mode with G41 or G42, the tool moves along the offset path until next G40 is specified.

(3) Cancel:

If G40 is specified in the offset mode, cutter compensation is cancelled and the tool moves to the programmed end point.

The offset direction for the preceding block will be on the line vertical to the shift direction at the end point.

12.5.6 Cutter Compensation Vector Hold/Change (G38)

During the offset mode, the offset vector of the previous block can be held or changed.

(1) Offset vector hold (α and β are the axes within the plane)

$$\begin{bmatrix} G00 \\ G01 \end{bmatrix} \quad G38 \quad \alpha\_ \quad \beta\_ \quad ;$$

This command holds the offset vector at the end point position of the previous block without creating the offset vector.

[Sample program]

```
G98 G00 X0 Y0 ;
N1 G17 G01 G42 X100. Y50. F200 ;
N2 X200. ;
N3 G38 X300. ;                 Offset vector hold
N4 G38 X400. Y100. ;           Offset vector hold
N5 X500. ;
```
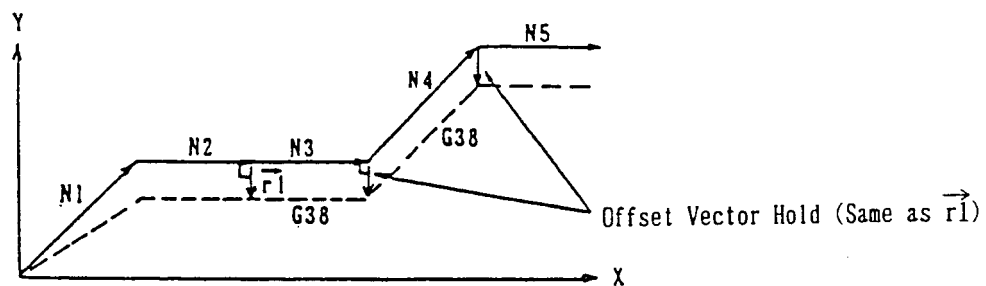

(2) Offset vector change

$$\begin{bmatrix} G00 \\ G01 \end{bmatrix} \quad G38 \quad I\_ \quad J\_ \quad K\_ \quad ;$$

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]

```
G17 G42 G01 ...... ;


N1 G01 U200. ;
N2 G38 I50. J50. ;        Offset vector change
N3 U-100. V50. ;
N4 U-100. ;
```
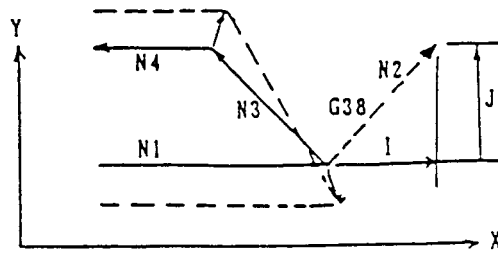
12.5.7   Cutter Compensation Corner Arc (G39)

During the offset mode, a G39 command allows the tool to move along an arc at the corner.

(1) G39 ;

If I, J, and K are omitted in the block containing G39, the tool moves along an arc which allows its end point vector to be perpendicular to the start point of the next block.

(2) 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]

```
    G17 G42 G01 ...... ;
        |
        |
        |
    N1 G01 U200. ;
    N2 G39 ;                        Corner arc
    N3 V50. ;
    N4 U-100. ;
    N5 G39 I-50. J-35. ;            Corner arc
    N6 U-100. V-50. ;
```
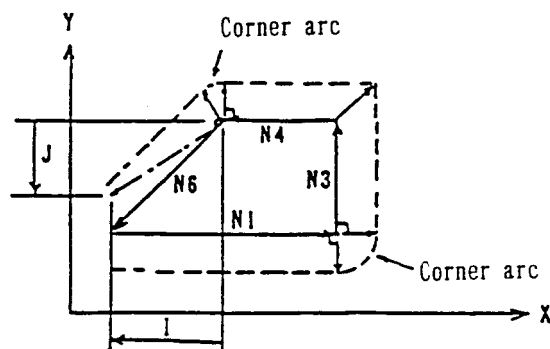
12.5.8   Cautions

(1) An alarm results if the offset plane is switched during the cutter compensation mode.

(2) If there is no axis move command in the 3 blocks counting from the block next to the G41/G42 specified one, the program starts from the subsequent axis move command specified block, cancelling the start-up.

(3) An alarm results if a move command is specified in the G39 specified block.

(4) When 3 or more blocks free of axial shift have been specified continuously in the offset mode or the pre-reading stop command has been performed, vertical vector is created at the end point of the block shifted for the last time, where workpieces may be left partly uncut or cut too much.

(5) If the following commands are given during the offset mode, an alarm results.
   G31, G37, G53,
   Canned cycle by the G code of Group 09

(6) 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.

(7) An alarm results if the start-up or cancellation block is specified with a G code other than G00 or G01.

(8) If cutter compensation is applied to an arc, a full circle may be turned into a short arc and vice versa.
   In such a case, specify with the arc divided.

12.6 Detailed Description of Cutter Compensation

12.6.1 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) G145 mode

(b) The tool radius (R + Q) of the selected tool is not 0.

(c) The virtual tool nose point (T) of the selected tool is 9.

(d) G41/G42 is or has been already specified.

(e) The move axis, whose stroke is not 0, has been specified.

(f) When the move axis, whose stroke is not 0, has been specified in the offset plane of the next block.
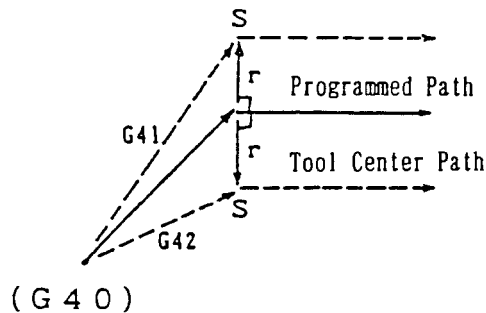
where;   r : Tool radius value

           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.*

12.6.2   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 between blocks.

(1) When the tool moves inside (180° ≦ α < 359°)
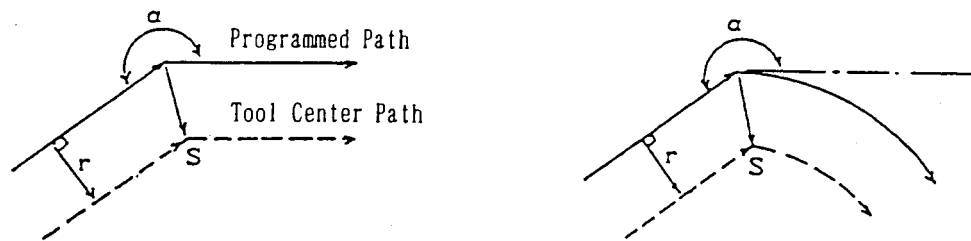

```
where;   r: Tool radius value
         s: Single block stop position
         L: Linear interpolation
```
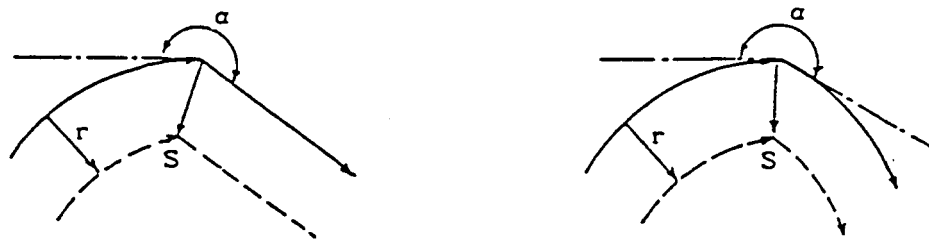

(2) When the tool moves outside (90° ≦ a < 180°)
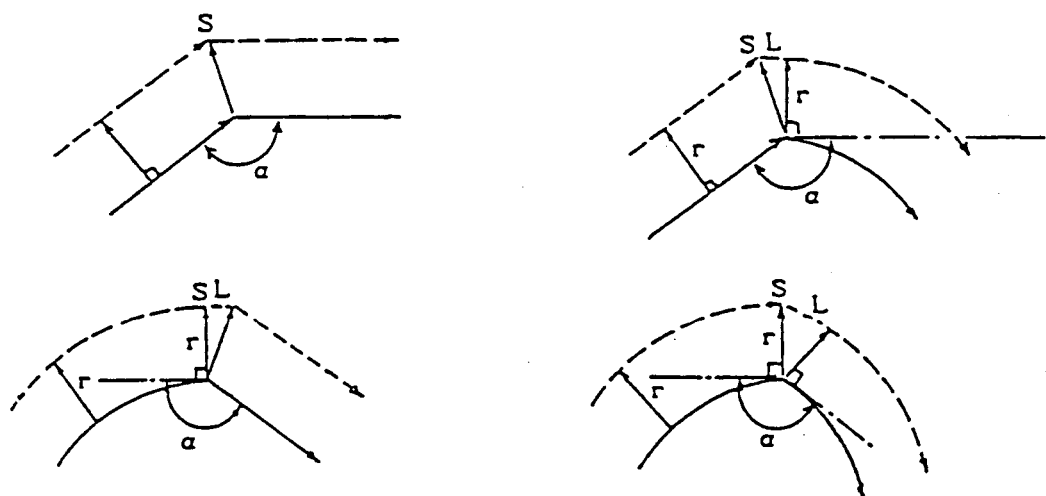
(3) When the tool moves outside at an acute angle ($\alpha < 90°$)
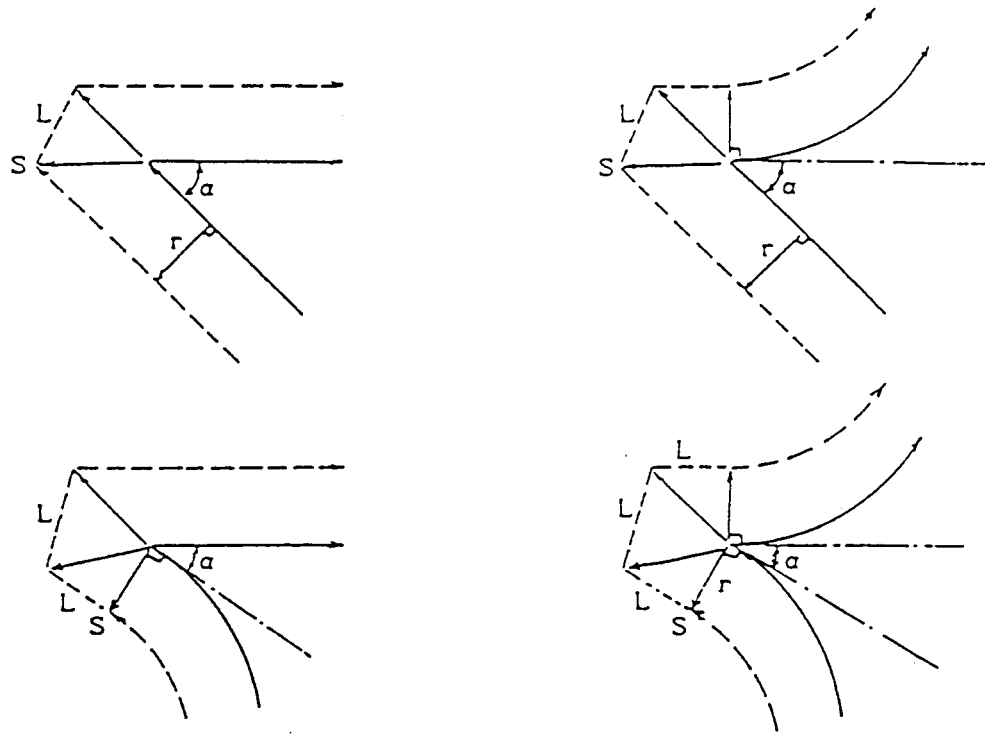
(4) When the tool moves inside ($359° \leqq \alpha$ or $\alpha < 1°$)
   i) When the tool moves inside in case of line-to-line, and the offset vector is large.
   ($\alpha < 1°$)
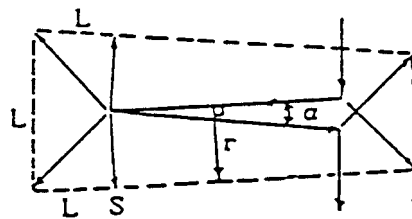
   ii) When the tool moves inside in case of line-to-arc, arc-to-line, or arc-to-arc, and
   the offset vector is large or cannot be obtained. ($359° \leqq \alpha$ or $\alpha < 1°$)
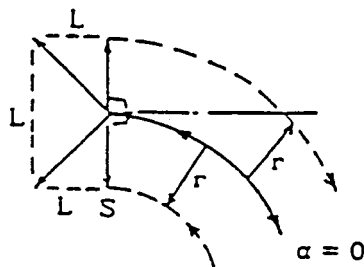
iii) When the tool moves inside in case of line-to-arc, arc-to-line, or arc-to-arc, and the normal offset vector can be obtained.
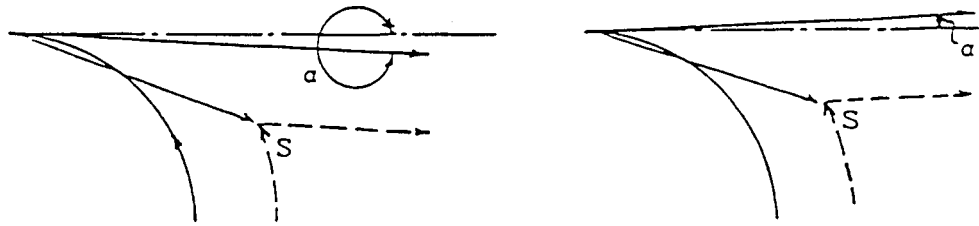
12.6.3 Cancel

If the block which satisfies even one of the following conditions is executed during offset mode, cutter compensation is cancelled and the tool moves to the programmed end point.

(a) G40 has been specified.

(b) Not in the G145 mode.

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.
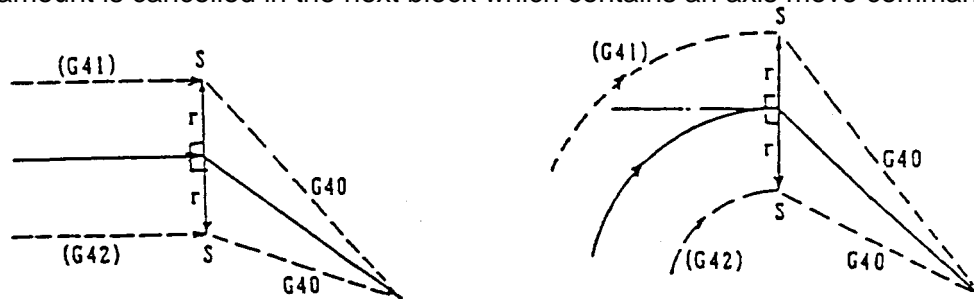
12.6.4 Special Uses

(1) When the offset direction is changed over by specifying G41/G42 during the offset mode, an intersecting point is obtained.
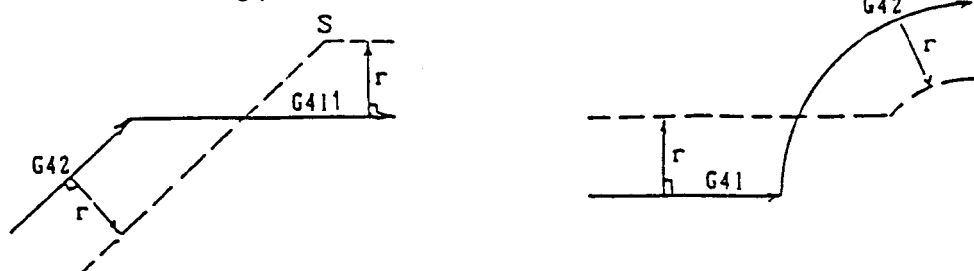
(2) When the offset direction is not changed over by specifying G41/G42 during the offset mode, the vector is created perpendicularly to the end point of the block before the G41/G42 command.
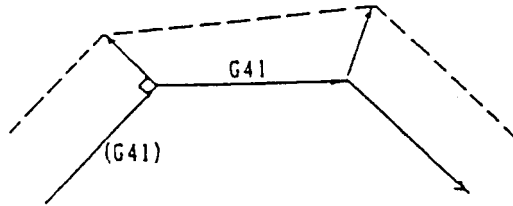
(3) When there is no intersecting point in the tool center path, the vector is created perpendicularly to the start point of the next block.
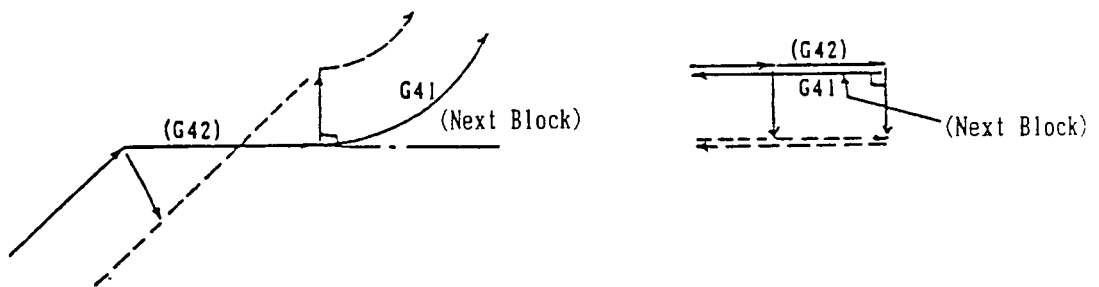
(4) When 3 or more blocks free of axial shift have been specified, the pre-reading stop block has been specified, or a block with 0 axial shifting amount has been specified, a vertical vector is created at the end point of the preceding block.

    G98 G17 G41 G00 U20. V10. ;

N1 G01 U100. V50. F200 ;

N2 M09 ;

N3 G04 X1. ;   Blocks with no axis move command

N4 W-50. ;

N5 U100. ;
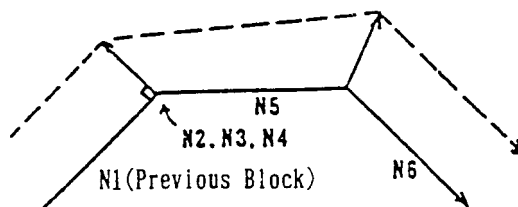
N6 U100. V-50. ;

(5) The tool release direction can be specified with I, J, and K by giving G40 α___ β__
I___ J___ K___ ; .

G17 G41 ...... ;
      ⋮
      ⋮
      ⋮
G01 U100. ;
G40 U-100. V-50. 130. J30. ;
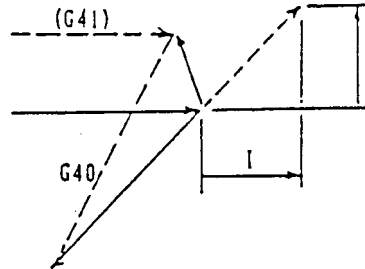
12.6.5 Move at the Corner

(1) When 2 or more offset vectors are created at the end point of the block and they are almost matching, the latter vector us invalidated.
When the next block is an arc, however, the offset vector perpendicular to the start point of the next block becomes invalid.
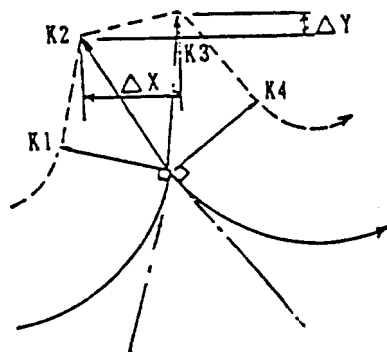
[Conditions]
$\Delta X \leq$ (Parameter value)
$\Delta Y \leq$ (Parameter value)

  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, K1 and K3 are invalidated.
  iii) When K2 and K4 are compared and their difference is smaller than the parameter set value, K1 and K3 are invalidated.
  iv) When K1 and K4 are compared, K4 is invalidated.

(2) A move at the corner is G00 when the next block is G00, and G01 when it is G01, G02, or G03.
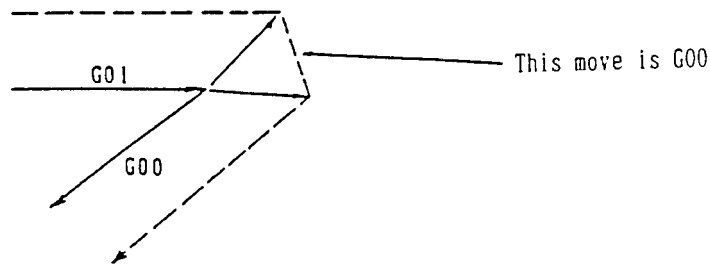
12.6.6 Interference Check

If cutter compensation is applied, the tool may cut into the workpiece excessively 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 it.

However, all excessive cuts cannot be prevented. Also, even when an excessive cut does not actually take place, an interference check alarm may result.

(1) 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 is the corresponding block of the tool center path is 90° or more and 270° or less.
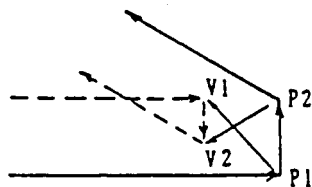
An interference results because the move directions of V1V2 and P1P2 forms an angle of 180°.

   ii) When an offset is applied to the inside of the arc which is smaller than the tool radius.
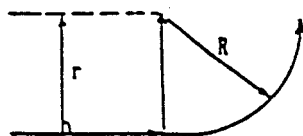
An interference results in case of r > R.

(2) 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 they still interfere, or when there is only one offset vector at one time from the beginning and it interferes, an alarm results.
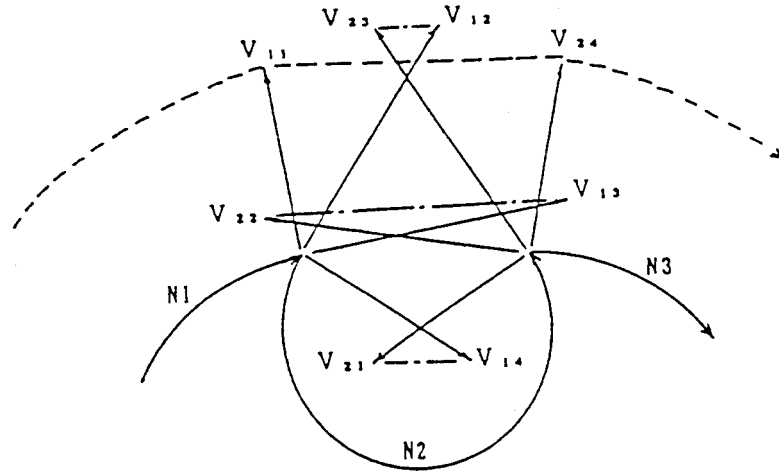
Interference check at V_{14} and V_{21} :Erases V_{14} and V_{21} at due to interference

Interference check at V_{13} and V_{22} :Erases V_{13} and V_{22} at due to interference

Interference check at V_{12} and V_{23} :Erases V_{12} and V_{23} at due to interference

Interference check at V_{11} and V_{24} : No interference

(Note) *The tool moves linearly from V_{11} from V_{24}.*
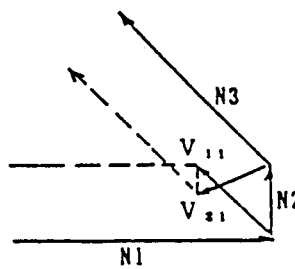
Interference check at V_{11} and V_{21} : Interference. If V_{11} and V_{21} are erased, an alarm results because the offset vectors do not exist.

12.6.7 Type B start-up and Cancellation

By parameter setting, you can change over the start-up and cancellation methods to Type B.

(1) Type B start-up

   i) When the tool moves inside (180° ≦ α)
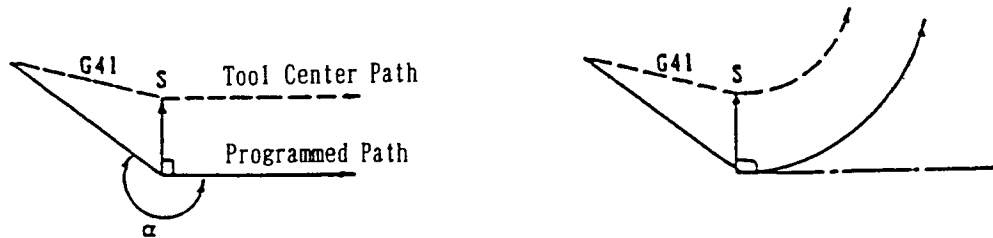
   ii) When the tool moves outside (90° ≦ α <180°)
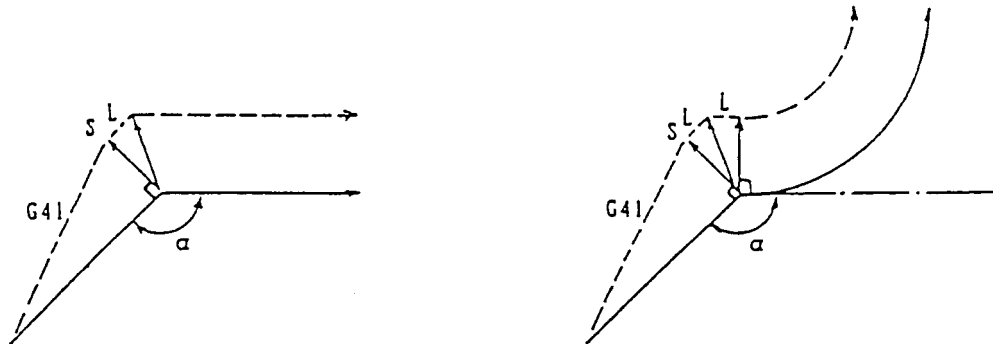
   iii) When the tool moves outside (1° ≦ α < 90°)
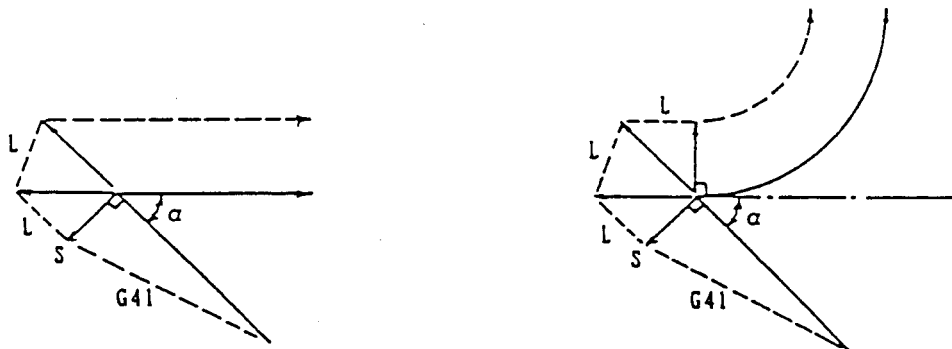
   iv) When the tool moves outside (α < 1°)

(2) Type B cancellation

    i) When the tool moves inside (180° ≦ α)
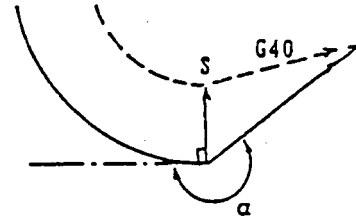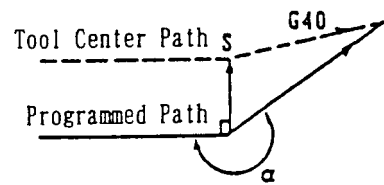
    ii) When the tool moves outside (90° ≦ α <180°)
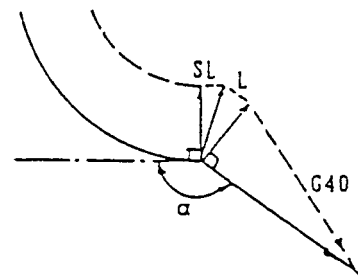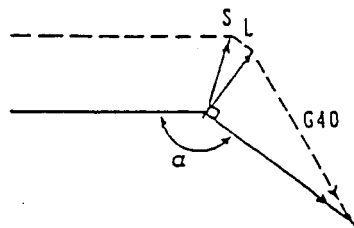
    iii) When the tool moves outside (1° ≦ α < 90°)
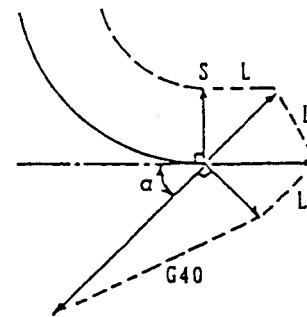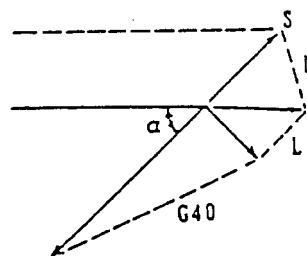
    iv) When the tool moves outside (α < 1°)
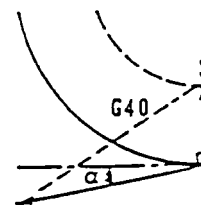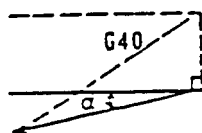
12.6.8 Associated Parameters

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 : Enabled if offset vector difference is 90° to 270°.

        = 1   Interference check : Disabled if offset vector difference is 90° to 270°.

No.5003, #2 = 0   Interference check : All enabled

        = 1   Interference check : All disabled

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 at which minor shifting amount is ignored on corner outside shifting.

12.6.9 Associated Alarms

No.115   Start-up/cancel of tool diameter compensation has been specified with a G code other than G00/G01.

No.117   An excessive cut was caused in the cutter compensation mode.

(#001)   The arc radius is smaller than the offset amount.

(#002)   Others

No.118   No intersecting point is found in the cutter compensation mode.

No.119   A wrong G code was specified in the cutter compensation mode.

(#001)   A plane has been changed in the current block.

(#002)   A plane has been changed in the next block.

(#003)   An erroneous G code has been specified in the current block.

(#004)   An erroneous G code has been specified in the next block.

(#005)   G39 has been specified except in G00 or G01 in the current block.

(#006)   G39 has been specified except in G00 or G01 in the next block.

(#008)   Axial shift has been specified within a plane with G39 command.

13. CONVERTING FUNCTION

13.1 Programmable Mirror Image (G501, G511)

The mirror image function can be applied to each axis with the G command of the program.

13.1.1 G Codes

G511 :   Programmable mirror image ON
G501 :   Programmable mirror image cancel

13.1.2 Command Format

(1) G511 X____ Y____ Z____ ...... ;
This command applies the mirror image to the specified axes, with the position of (X____, Y____, Z____, ...) as a mirror point.
The axis command of this block has the meaning given below. It does not have any relation to the axis shift.
• Specifies the axis that applies the mirror image in the axis address.
• Specifies the mirror point in the axis data.

(2) G501 X____ Y____ Z____ ...... ;
This command cancels the programmable mirror image of the specified axes.
In the axis command for this block, only the axis that cancels the programmable mirror image is specified; its value has no significance and it has no relation to the shift of the axis.

(3) G501 ;
This command cancels the programmable mirror image of all the axes.

13.1.3 When the Mirror Image is Applied to Only One Axis of the Specified Plane

(1) Arc command (G02, G03) : Clockwise and counterclockwise are reversed.
(2) Cutter compensation (G41, G42) : Offset to right and offset to left are reversed.
(3) Automatic tool nose radius compensation left (G141) and right (G142) are reversed.

13.1.4   Sample Program

```
G98 G17 G00 X140. Y20. ;
G511 X140. ;                    X-axis mirror image ON      (mirror point: X140.)
N1 G01 X180. Y40. F200 ; ┐
N2 X240. ;               │      The mirror image is
N3 G03 Y80. R20. ;       │      applied to the X axis.
N4 G01 X180. ;           │
N5 X140. Y20. ;          ┘
G501 X0 ;                       X-axis mirror image cancel
```


Mirror Point (140.)
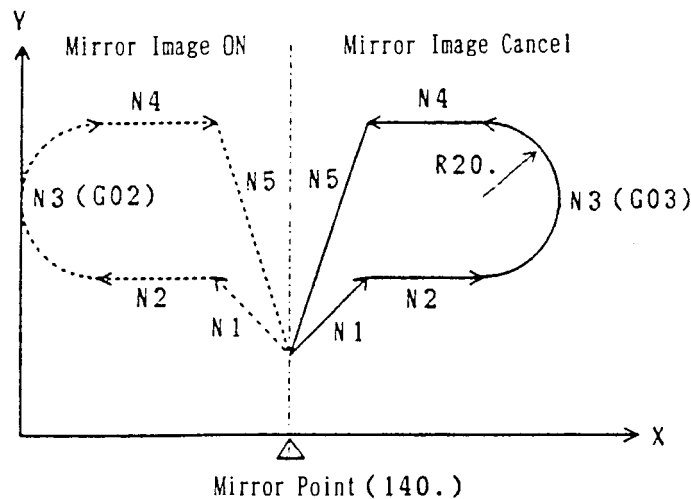
13.1.5   Cautions

(1) Specify the G511 and G501 commands in the independent block.
Otherwise, an alarm will result.

(2) Position display indicates the coordinate value after the programmable mirror image is applied.

(3) When the programmable mirror image and setting mirror image are applied, the former works first, and then, the latter.

(4) The following commands disable the programmable mirror image.
i)   G28, G30, G301, and G53
However, the programmable mirror image is applied to the intermediate point of G28, G30, and G301.
ii) Shift amount of fine boring (G76) and back boring (G87)

(5) The first move command after the block which contains G511 or G501 should be specified by absolute programming.

(6) If an arc is specified as the first move command after the block which contains G511 or G501, a proper arc may not be obtained.

13.1.6   Associated Parameters

13.1.7   Associated Alarms

No.144   G501/G511 block is not in an independent command.

13.2   Setting Mirror Image

The mirror image can be applied to each axis by turning on/off the Setting screen or turning on/off an external input signal (PC → NC).

13.2.1   On/off Operation with the Setting Screen

ON/OFF of the mirror image of each axis can be specified on the setting (data) screen.

13.2.2   Mirror Image on/off Switching becomes Effective from the Next Buffering Block

(Note 1)   *The mirror image is applied taking a mirror image ON-OFF switching position as a mirror point, whether the program is of absolute programming or incremental programming.*

(Note 2)   *Through parameter setting, irrespective of position where the mirror image has been changed ON → OFF, the mirror point can be made 0.*

13.2.3   When the Mirror Image is Applied to Only one Axis of the Specified Plane.

(1)   Arc command (G02, G03) : Clockwise and counterclockwise are reversed.

(2)   Cutter compensation (G41, G42) : Offset to right and offset to left are reversed.

(3)   Automatic tool nose radius compensation left (G141) and right (G142) are reversed.

13.2.4 Sample Program

```
G98 G17 G00 X140.0 Y20.0 ;
M△△△ ;              X-axis mirror image ON (mirror point: X140.)
N1 G01 X180.0 Y40.0 F200 ;
N2 X240.0 ;                      The mirror image is
N3 G03 Y80.0 R20.0 ;             applied to the X axis.
N4 G01 X180.0 ;
N5 X140.0 Y20.0 ;
M □□□ ;             X-axis mirror image cancel
```


Mirror Point（140.）
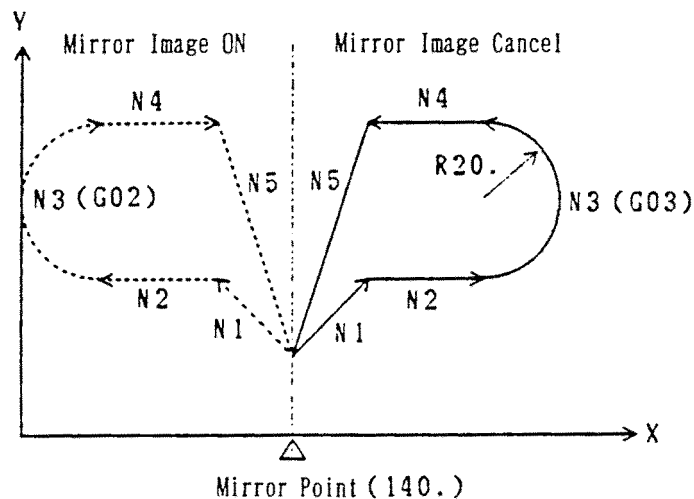
13.2.5 Cautions

(1) When either the mirror image of the setting screen or external input signal is ON, the setting mirror image will become ON.
(Becomes ON even if both are ON.)

(2) Position display indicates the coordinate value after the setting mirror image is applied.

(3) When the programmable mirror image and setting mirror image are applied, the former works first, and then, the latter.

(4) The following commands disable the programmable mirror image.
   i) G28, G30, G301, and G53
      However, the setting mirror image is applied to the intermediate point of G28, G30, and G301.
   ii) Shift amount of fine boring (G76) and back boring (G87)

(5) In the reset state, the setting mirror image is turned off.

(6) After switching on/off the setting mirror image, specify the first move command by absolute programming.

(7) If an arc is specified as the first move command after switching on/off the setting mirror image, a proper arc may not be obtained.

13.2.6   Associated Parameters

No.3406, #3 = 0   Mirror point of the setting mirror image is equal to the buffering
                          position.

            = 1   Mirror point of the setting mirror image is equal to the buffering 0
                          position.

No.3416, #0 = 0   Mirror image for each axis OFF

            = 1   Mirror image for each axis ON

13.3   Chamfering, Corner R

Chamfering or corner R can be inserted in between the blocks of linear interpolation (G01)
which turns at a right angle.

Chamfering is specified with the addresses I and K, and corner R with the address R.

13.3.1   Command Format

(1) Chamfering

      G01 X___ K___ ;

      G01 Z___ I ___ ;

(2) Corner R

      G01 X___ R___ ;

      G01 Z___ R___ ;

(Note) *The chamfering/corner R direction is specified with the plus/minus sign of I, K,*
and R.
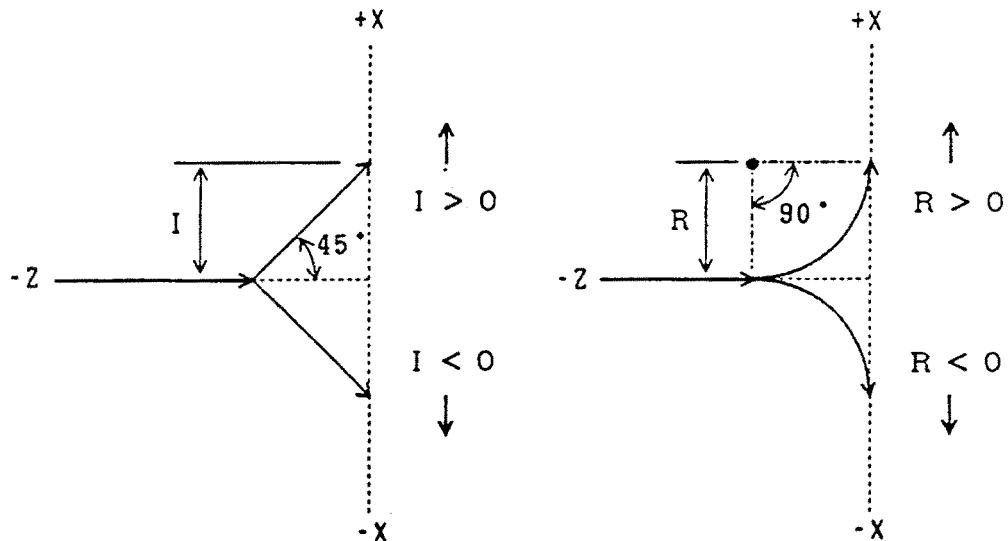
13.3.2 Sample Program

```
G00  X0  Z0 ;
G18  G01  X30.  K-5.  F1.0 ;
Z-25.  R5. ;
X60. ;
Z-35. ;
```
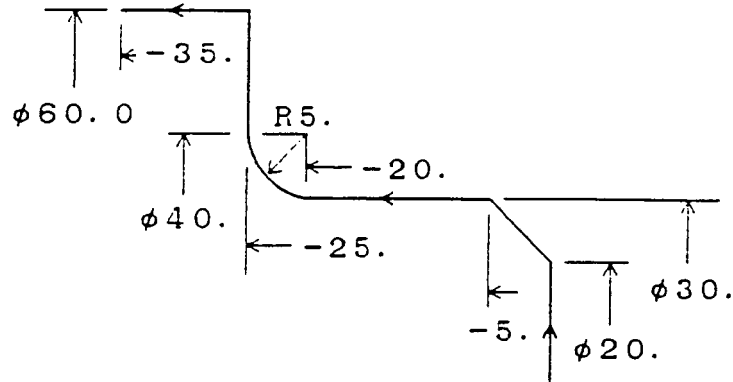

13.3.3 Cautions

(1) Movement to be specified with G01 for chamfering/corner R should be concerned with only one axis within the plane, and the next block must be one axis perpendicular to that axis within the plane.

(2) A stop point in the SINGLE BLOCK mode is immediately after chamfering/corner R.

(3) It is also possible to perform chamfering/corner R in the other planes (G17, G19). Use the address J for chamfering in the Y-axis direction.

(4) An alarm results from the following conditions.
   (a) When the block next to chamfering/corner R is not a G01 move perpendicular to the previous block, or when a G code (G17-G19) for plane selection is specified.
   (b) When a chamfering/corner R amount is larger than an original move amount.

13.3.4 Associated Parameters

13.3.5 Associated Alarms

No.178   An error exists in the chamfering/corner R command.
 (#001)   I/J/K/R value has exceeded the original shifting amount in 1st block.
 (#002)   The 2nd block is not equal to G01.
 (#003)   G17 ~ G19 has been specified in 2nd block.
 (#004)   The 2nd block has not shown right-angled shift to 1st block.
 (#005)   An error exists in a G code command in 2nd block.
 (#006)   I/J/K/R value has exceeded the original shifting amount in 2nd block.

13.4 Optional Angle Chamfering/Corner R (, C, R)

Optional angle chamfering or corner R can be inserted by specifying ",C" or ", R" in linear interpolation or circular interpolation.

13.4.1 Command Format

(1) Optional angle chamfering

```
⌈G01
|G02    ...... ,  C_ ;
⌊G03
```
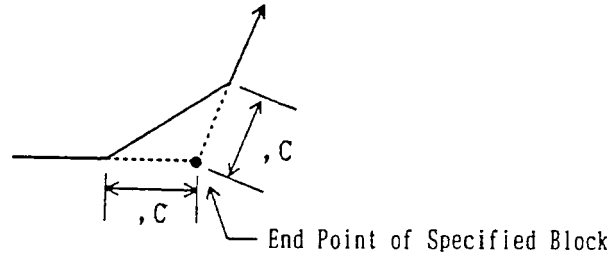

(2) Optional angle corner R

```
⌈G01
|G02    ...... ,  R_ ;
⌊G03
```
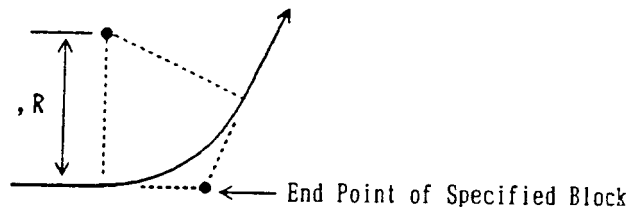

(Note 1) *Specify the plane for optional angle chamfering/corner R with G17, G18, or G19.*

(Note 2) *In the block next to one where optional angle chamfering/corner R was specified, specify linear interpolation or circular interpolation.*

13.4.2 Sample Program

(1) Optional angle chamfering
```
G18  G00  X0  Z-50. ;
G01  X20.  F1.0 ;
N1  G03  X120.  Z0  R50. , C10. ;
N2  G01  Z50. ;
```
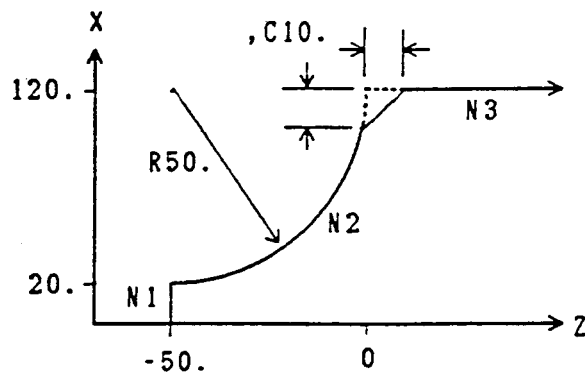
(2) Optional angle corner R

G18 G00 X0 Z-50. ;
G01 X20. F1.0 ;
N1 G03 X120. Z0 R50. , R20. ;
N2 G01 Z50. ;
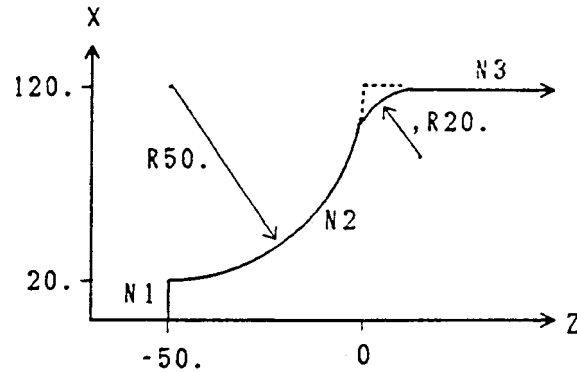
13.4.3 Cautions

(1) An alarm results if you change over the plane in the block next to one where chamfering/corner R was specified.

(2) A stop point at single block time is the end point of the chamfering/corner R block.

(3) An alarm results if the following commands are specified in the block where chamfering/corner R was specified, or in the next block.

   G50, G28, G29, G30, G53, G31

(4) When a difference of angle is within ±1 % between the chamfering/corner R specified block and next block, chamfering/corner R is invalid.

(5) An alarm results if there is no axis move within the plane in the chamfering/corner R specified block or next block.

(6) When the original movable range is exceeded as a result of inserting chamfering/corner R, an alarm results.

13.4.4 Associated Parameters

13.4.5 Associated Alarms

No.124   An error exists in optional angle chamfering/corner R command.

(#001)   The ,C or ,R command value is in minus.

(#002)   The ,C and ,R have been specified in the same block.

(#003)   The current block is not equal to G01-G03.

(#004)   No axial shift exists within a plane in the current block.

(#005)   An error exists in a G code command in the current block.

(#006)   Next block is not equal to G01-G03.

(#007)   No axial shift exists within a plane in the next block.

(#008)   An error exists in a G code in the next block.

(#009)   A plane has been changed in the next block.

(#010)   The original shifting amount has been exceeded in the current block.

(#011)   The original shifting amount has been exceeded in the next block.

(#012)   A calculation error has occurred.

13.5 Three Dimensional Coordinate Conversion (G268, G269)

Three dimensional coordinate conversion is a feature to convert the coordinates to around a desired axis by specifying the rotation center of the orthogonal coordinate system, and the direction and angle of the rotation center axis.

13.5.1 G-codes

G268: Three dimensional coordinate conversion mode ON
G269: Three dimensional coordinate conversion mode OFF

13.5.2 Command Format

G268 [X_] [Y_] [Z_] [B_] [I_] [J_] [K_];

　　　　　　　Three Dimensional Coordinate Conversion Mode ON

　　　　　　　Three Dimensional Coordinate Conversion Executed

G269 ;　Three Dimensional Coordinate Conversion Mode OFF

X, Y, Z:　Center of rotation (Default:0)
I, J, K:　Vector for the rotation center axis (Default: 0; J = 1.0 when I, J, and K are all omitted)
B:　Rotation angle (Default: Turret indexing angle)

N1 G268 Xx_1 Yy_1 Zz_1 Ii_1 Jj_1 Kk_1 Bb_1 ;
N2 G268 Xx_2 Yy_2 Zz_2 Ii_2 Jj_2 Kk_2 Bb_2 ;

Three dimensional coordinate conversion can be applied twice by parameter setting. In the N1 block, specify the center of initial rotation, and the direction and rotation angle of the rotation center axis. By executing this command, it can be assumed that the coordinate system X'Y'Z', which was decentered by X_1, y_1, z_1 from the original coordinate system and rotated by the angle b_1, has been newly created around the vector specified with i_1, j_1, k_1.

Next, in the N2 block, specify the center of second rotation, and the direction and rotation angle of the rotation center axis. The X-, Y-, Z-, I-, J-, K-, and B-code commands assume the values in the coordinate system X'Y'Z' created by conversion in the N1 block. By executing this command, it can be assumed that the coordinate system X"Y"Z", which was decentered by x_2, y_2, z_2 from the coordinate system X'Y'Z'and rotated by the angle b2, has been newly created around the vector specified with i_2, j_2, k_2.

Subsequent program command values, x_p, y_p, z_p are those in the coordinate system X"Y"Z".

If x_2, y_2, and z_2 have not been specified in the N2 block, the center of rotation will be the same position as that in the N1 block. The N2 block is not required if one rotation will do.

Example)   G268 Xx$_0$ Yy$_0$ Zz$_0$ I0 J0 K1 βα
           G268   I1 J0 K0 βα
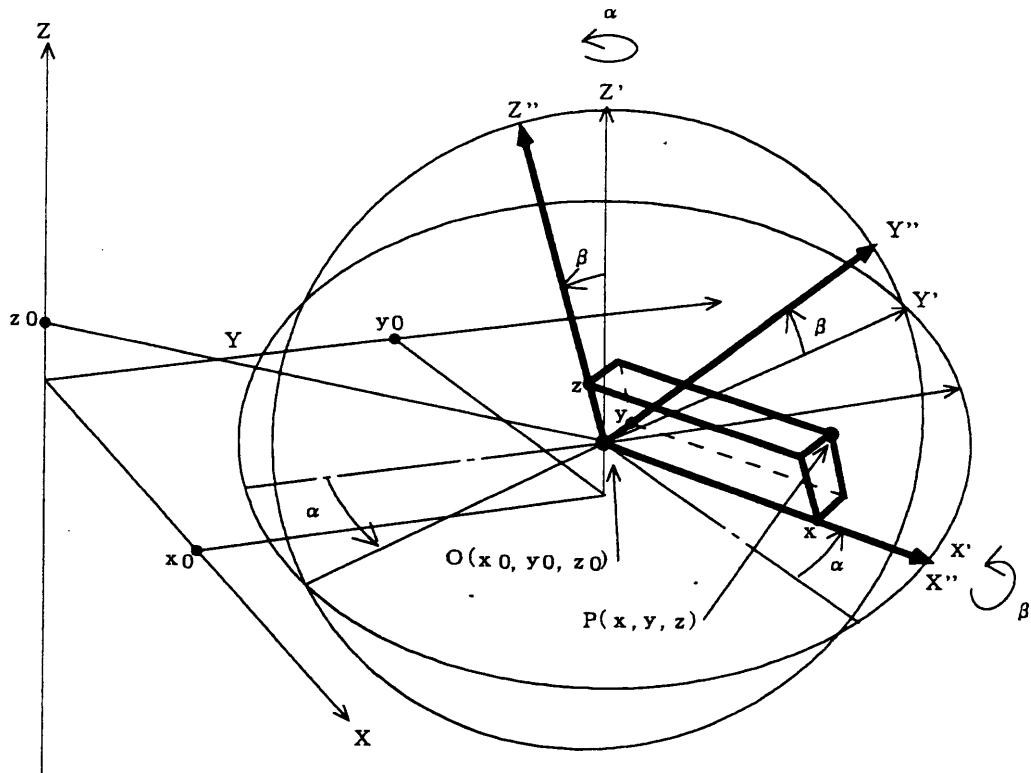
XYZ:	Coordinate system before conversion
X'Y'Z':	Coordinate system after first conversion
X"Y"Z":	Coordinate system after second conversion
α:	First rotation angle
β:	Second rotation angle
O(x$_o$, y$_o$, z$_o$):	Center of rotation
P(x, y, z):	Coordinate values in the coordinate system X"Y"Z"

13.5.3 Three Dimensional Coordinate Conversion Formula

The following conversion formula is generally used to represent the relations between the coordinate values(x',y',z') in the coordinate system after conversion and those(x, y, z) in the coordinate system before conversion.

$$
\begin{pmatrix} x \\ y \\ z \end{pmatrix} = \begin{pmatrix} M_1 \end{pmatrix} \begin{pmatrix} x' \\ y' \\ z' \end{pmatrix} + \begin{pmatrix} x_1 \\ y_1 \\ z_1 \end{pmatrix}
$$

The following formula applies to second conversion.

$$
\begin{pmatrix} x \\ y \\ z \end{pmatrix} = \begin{pmatrix} M_1 \end{pmatrix} \begin{pmatrix} M_2 \end{pmatrix} \begin{pmatrix} x' \\ y' \\ z' \end{pmatrix} + \begin{pmatrix} M_1 \end{pmatrix} \begin{pmatrix} x_2 \\ y_2 \\ z_2 \end{pmatrix} + \begin{pmatrix} x_1 \\ y_1 \\ z_1 \end{pmatrix}
$$

where;   x, y, z: Coordinate values in the original coordinate system

         x',y',z':   Coordinate values in the coordinate system after conversion

         x_1, y_1, z_1:   Center of rotation for first conversion

         x_2, y_2, z_2:   Center of rotation for second conversion (Values in the coordinate system created by first conversion)

         M_1:   First conversion matrix

         M_2:   Second conversion matrix

M_1 and M_2 are conversion matrixes determined by the rotation angle and rotation center axis and generally expressed as follows:

$$
\begin{pmatrix} n_1^2+(1-n_1^2)\cos\theta & n_1n_2(1-\cos\theta)-n_3\sin\theta & n_1n_3(1-\cos\theta)+n_2\sin\theta \\ n_1n_2(1-\cos\theta)-n_3\sin\theta & n_2^2+(1-n_2^2)\cos\theta & n_2n_3(1-\cos\theta)+n_1\sin\theta \\ n_1n_3(1-\cos\theta)-n_2\sin\theta & n_2n_3(1-\cos\theta)+n_1\sin\theta & n_3^2+(1-n_3^2)\cos\theta \end{pmatrix}
$$

where; n_1:   X-axis directional cosine of the rotation center axis   i/p

         n_2:   Y-axis directional cosine of the rotation center axis   j/p

         n_3:   Z-axis directional cosine of the rotation center axis   k/p

         θ: Rotation angle

p is determined as follows:

$$
p = \sqrt{i^2+j^2+k^2}
$$

The conversion matrix looks like the following for coordinate conversion in the ZX plane.

$$
\begin{pmatrix} M \end{pmatrix} = \begin{pmatrix} \cos\theta & 0 & \sin\theta \\ 0 & 1 & 0 \\ -\sin\theta & 0 & \cos\theta \end{pmatrix}
$$

13.5.4 Sample Program (X-axis [Diameter Designation])

G28 U0 V0 W0 ;
T020200 B60.0 ;
G97 S1000 M13 ;
G145 ;
G17 :
G00 X54.642
G268 X20.0 Z-10.0 ;
Z50.0 ;
G81 Z-15.0 R10.0 D-8.0 F500 E3000 L0
X40.0 ;
Y20.0 ;
G198 X20.0 ;
G80 ;
G269 ;
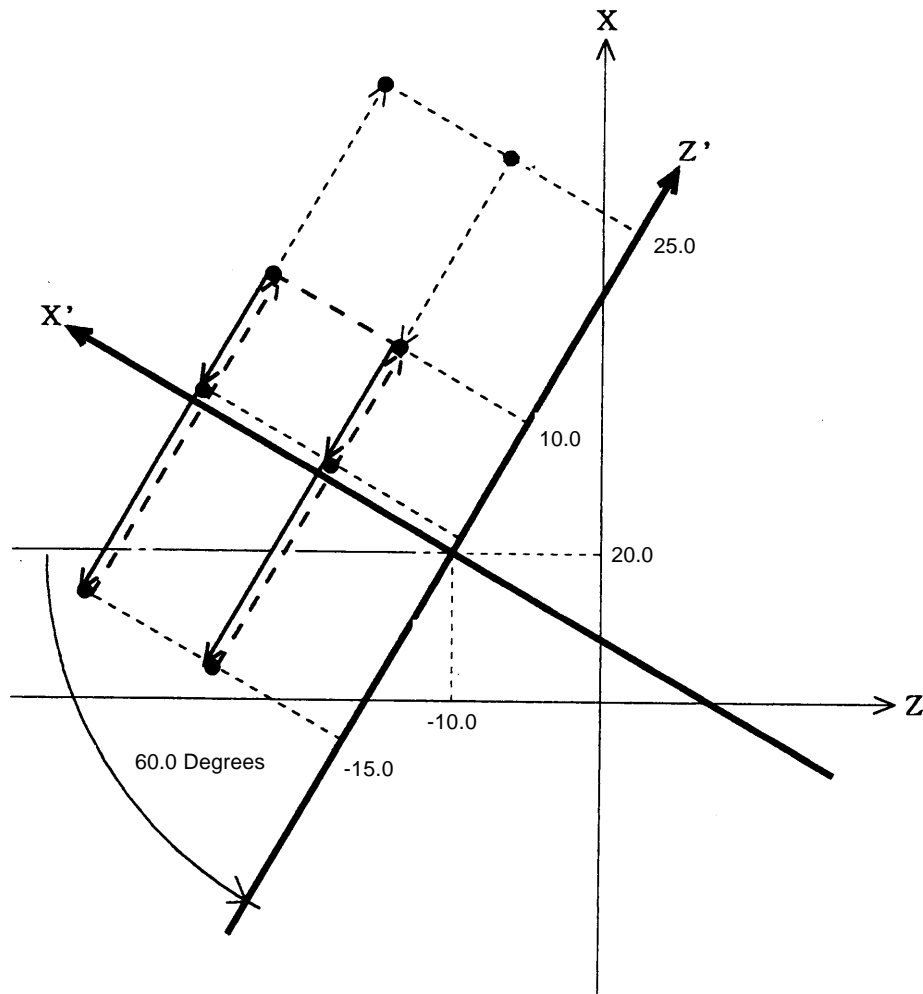
13.5.5 Precautions

(1) The coordinate system cannot be set in the three dimensional coordinate conversion mode. It is impossible to give any command which causes coordinate system setting.(The T-code command or manual indexing are not allowed.)

(2) Model G-codes at three dimensional coordinate conversion command(G268/G269)
The G-codes in the following groups must be selected as follows:
Group 07: G40 (Cutter compensation cancel)
Group 09: G80 (Drilling cycle)
Group 15: G501 (Programmable mirror image cancel)
Group 16: G150 (Groove width compensation cancel)
Group 22: G120 (Polar coordinate interpolation

(3) Commands available in the three dimensional coordinate conversion mode
G00, G01, G02, G03, G04, G10, G11, G17, G18, G19, G28, G29, G30, G38, G39, G40, G41, G42, G53, G65, G66, G67, G80, G81, G82, G83, G831, G84, G841, G842, G843, G85, G86, G861, G87, G88, G89, G98, G99, G140, G143, G145, G141, G142, G198, G199, G501, G511

(4) The three dimensional coordinate conversion mode is not canceled until a cancel command is executed, unless the power is turned off.

(5) Three dimensional coordinate conversion is applied to the basic three axes.

(6) Three dimensional coordinate conversion can be doubly applied by parameter setting. When this is done, the coordinates of the rotation center assume the values in the coordinate system where first three dimensional coordinate conversion has been applied.

(7) For rotation angle B, rotation in the right-handed screw direction is assumed to be positive, facing the rotation axis direction.

(8) When applying automatic nose radius compensation or cutter compensation, three dimensional coordinate conversion is turned on after compensation.

(9) Three dimensional coordinate conversion is not applied to the commands in the machine coordinate system.

(10) Programmable mirror image can be specified, but not external mirror image. Three dimensional coordinate conversion can be applied after programmable mirror image.

(11) If a drilling axis is commanded to move at rapid traverse rate in the canned cycle for drilling in the three dimensional coordinate conversion mode, it will move by cutting feed which assumes the feed rate to be the maximum cutting feed rate.

(12) In the three dimensional coordinate conversion mode, an X-axis command can be given with a radius value through parameter setting. When this is done, the tool nose position is also displayed in terms of the radius value.

(13) With the turret turned by 90 or 180 degrees, three dimensional coordinate conversion is disabled.

(14) To give a three dimensional coordinate conversion command, automatic nose radius compensation and cutter compensation should be canceled.

(15) Do not specify other G-codes in the G268 or G269 block.

(16) If three dimensional coordinate conversion is specified in the back machining mode, coordinate conversion will be performed at the angle 180 degrees minus the command. This moves the tool closer to the workpiece with a +Z-axis command in the back machining mode. However, coordinate conversion can be executed at the specified angle even in the back machining mode by switching the parameter.

13.5.6 Associated Parameters

No. 3413, #1 = 0   Disables three dimensional coordinate conversion double rotation

= 1   Enables three dimensional coordinate conversion double rotation

No. 3413, #2 = 0   In the three dimensional coordinate conversion mode, the X-axis command abides by radius designation.

= 1   In the three dimensional coordinate conversion mode, the X-axis command abides by No. 1010, #3.

No. 3413, #3 = 0   In the back machining mode, the three dimensional coordinate conversion rotation angle assumes the specified angle minus 180 degrees.

= 1   In the back machining mode, the three dimensional coordinate conversion rotation angle assumes the specified angle.

No. 3466   First rotation angle

No. 3467   Second rotation angle

13.5.7 Associated Alarms

No. 170   Three dimensional coordinate conversion command error

(#001)   A modal G-code is not correct.

(#002)   A rotation center direction command is not correct.

(#003)   Three dimensional coordinate conversion has been doubly applied(triply when double conversion is enabled by the parameter)

(#005)   A rotation angle command is not correct.

(#006)   A turret indexing angle is not correct.

(#007)   G268/G269 cannot be given in the direct tapping mode.

(#008)   Tool nose radius/cutter compensation has not been canceled.

No. 171   Three dimensional coordinate conversion mode error.

(#001)   An disallowed G-code has been given.

(#004)   A T-code command has been given.

No. 191   No option

(G268)   No three dimensional coordinate conversion option.

14. SINGLE TYPE FIXED CYCLE

Use of single type fixed allows you to specify in one block a series of actions, "Cut-in to Cutting(or Thread cutting) to Release to Retract" given in the normal program.
Furthermore, when repeating it, you only specify newly changed values; it is very effective in simplification of the program.

14.1 O.D./I.D. Cutting Cycle (G90)

Capable of performing O.D./I.D. straight or taper cutting.

14.1.1 Command Format

G90 X__ Z__ (I__) F__ [E__] ;

(1) Straight Cutting
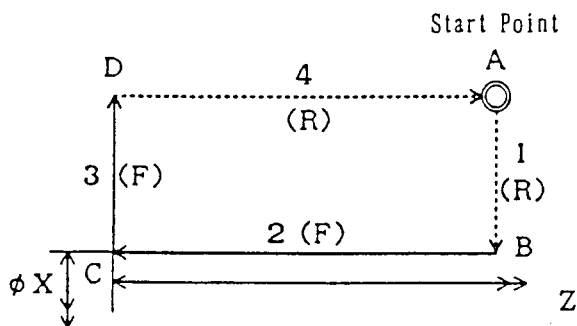
(2) Taper Cutting

G90 X__ Z__ F__ ; (I=0)
R: Rapid traverse
F: Cutting feed
(F code designation)
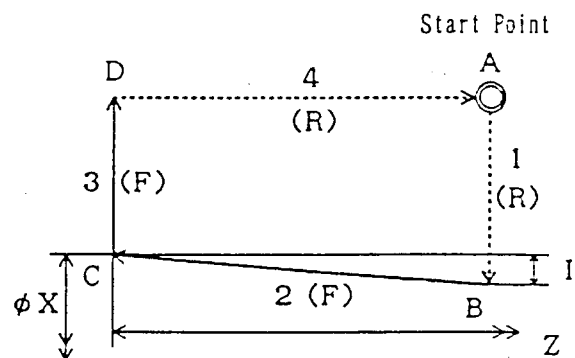
G90 X__ Z__ I__ F__ ;
(Pay attention to the sign of I)

(Note 1) *Specify the position of the point C with X and Z.*

(Note 2) *I refers to an incremental value(radius value) when viewing the point B from the point C in the X-axis direction.*

(Note 3) *With E being commanded instead of F, shift speed of Operation 3 can be changed.*

- *E > 0 : Operation 3 indicates cutting feed at speed E.*
- *E = 0 : Operation 3 indicates rapid traverse.*

14.1.2 Sample Program

G00 X100. Z10. ;
G90 X50. Z-40. F1.0 ;
X40. ;
X30. ;
G00 Z100. ;

φ100.

φ50.
φ40.
φ30.

−40.

−10.
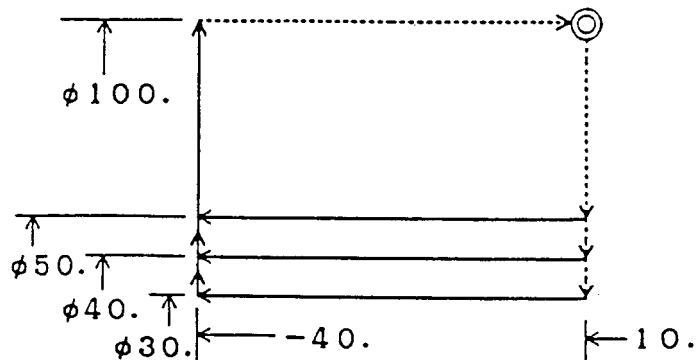
14.2 Canned Cycle for Thread Cutting (G92)

Capable of performing O.D./I.D. straight or taper thread cutting.

14.2.1 Command Format

G92 X___ Z___ (I___) (Q___) F___ ;

(1) Straight Cutting

(2) Taper Cutting

Start Point

D   4
(R)

A

3 (R)

1
(R)

2 (F)   B

φX   C

Z

Start Point

D   4
(R)

A

3 (R)

1
(R)

2 (F)   B

φX   C   I

Z
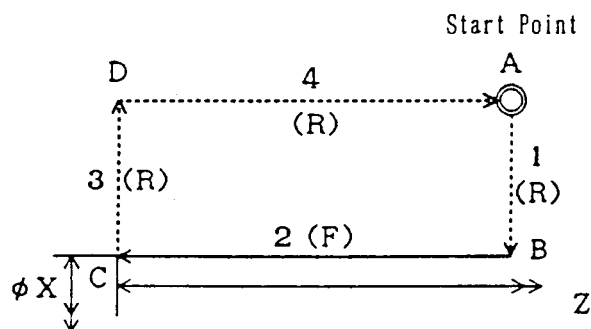
G92 X___ Z___ F___ ; (I=0)
R: Rapid traverse
F: Thread cutting
(F code designation)
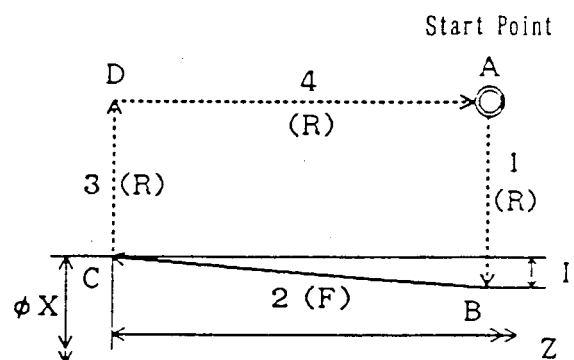
G92 X___ Z___ F___ ; (I=0)
(Pay attention to the sign of I)

(Note 1) *Specify the position of the point C with X and Z.*

(Note 2) *I refers to an incremental value (radius value) when viewing the point B from the point C in the X-axis direction.*

(Note 3) *Thread lead assignment is available in two ways as in the ordinary thread cutting, i.e. direct assignment by F/assignment of no. of threads per inch. (As for details, see "3.1 Thread Cutting.")*

(Note 4) *The unit of Q, whether to be 1 degree or 0.001 degree, can be selected by a parameter. (As for details, see "3.3 Multiple Thread Cutting".)*

(Note 5) *In all the steps (A → B → C → D → A), the main axis override will be invalid.*

14.2.2 Chamfering

Chamfering can be done at the point C by a signal from the machine side.
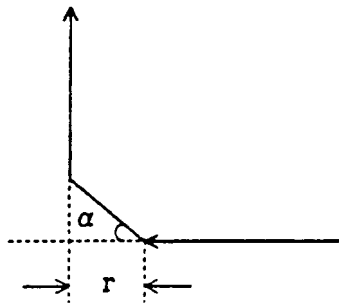
Suppose a thread lead is L, the value of chamfering r is set for the parameter in an increment of 0.1L in a range of 0.1L to 12.7L.

The value of chamfering angle a is also set for the parameter in an increment of 1° in a range of 1° to 89°.

(Note) *When the parameter values above are beyond the setting range, it is assumed to be r = 0.1L and a = 45°.*

14.2.3 Thread Cutting Cycle Retract

When Feed Hold has been applied while in thread cutting (Operation 2), even with the halt lamp being lit up, operation goes on until single block stop takes place at the start point.

When the "thread cutting cycle retract" function is attached, and feed hold is applied during thread cutting, the tool is released immediately chamfering the thread, and the X and Z axes retract to the initial point in that order.

——— Normal chamfering
········· Movement at feed hold

Feed hold applied here
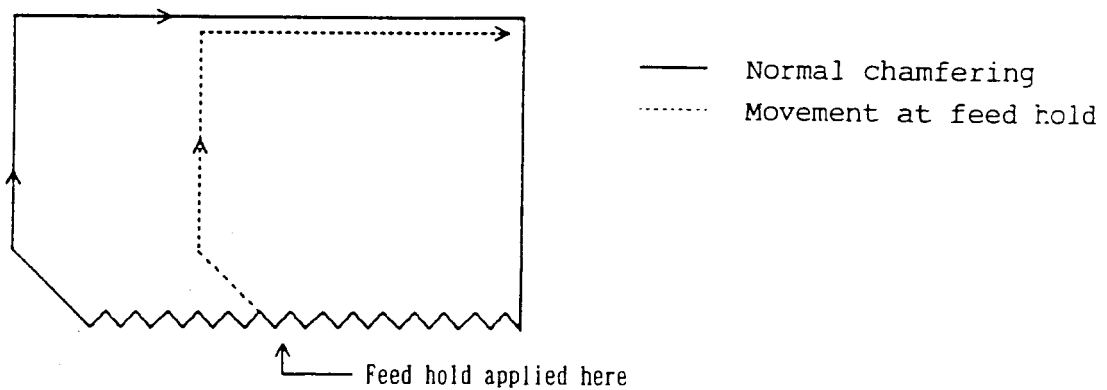
(Note 1) *The chamfering amount upon release is the same as that at the end point.*
          While the tool is being released, additional feed hold is not effective.
(Note 2) *This machine tool, when restarted to run at the start point, follows the same course of operation (A → B → C1 → C2 → D).*
          Also, the tool can be retracted while in re-run.

14.2.4 Sample Program

G00 X100. Z10. ;
G92 X50. Z-40. F1.0 ;
X40. ;
X30. ;
G00 Z100. ;
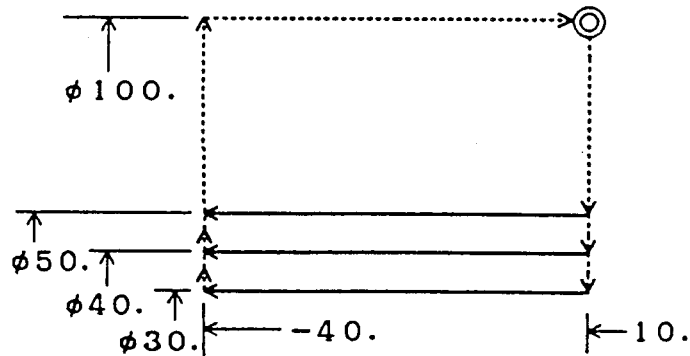
14.2.5 Rapid Traverse Override Clamp of X Axis Release

X-axis release (C → D) in threading cycle moves in rapid traverse by the same exponential function type acceleration/deceleration as that of threading (B → D). When the acceleration/deceleration time constant of threading is reduced so that the incomplete screw thread part is shortened in threading, shock may be created to the Machine system.

In X-axis release, the rapid traverse override having been selected on Machine operation panel can be clamped with the parameter set value so that shock mentioned above can be lessened. The parameter set range includes 1 to 100% and any setting beyond this is regarded as 100%, where no clamping is performed.

In rapid traverse other than that of X-axis release (A → B, D → A), no clamping takes place for rapid traverse override.

14.2.6 Assciated Parameters

No.1626   Time constant of exponential function type acceleration/deceleration in threading cycle of each axis (msec)
No.1627   FL speed of exponential function type acceleration/deceleration in threading cycle of each axis (mm/min)
No.5111   Screw chamfering amount ($\times$ 0.1L)
No.5112   Screw chamfering angle (deg.)
No.5114   Rapid traverse override clamp value in X-axis release (%)

14.3 End/side Cutting Cycle (G94)

Capable of performing end/side straight or taper cutting.

14.3.1 Command Format

G94 X__ Z__ (K__) F__ [E__] ;

(1) Straight Cutting

(2) Taper Cutting

R: Rapid traverse
F: Cutting feed
(F code designation)

G94 X__ Z__ F__ ;
(K=0)

G94 X__ Z__ K__ F__ ;
(Pay attention to the sign of K)
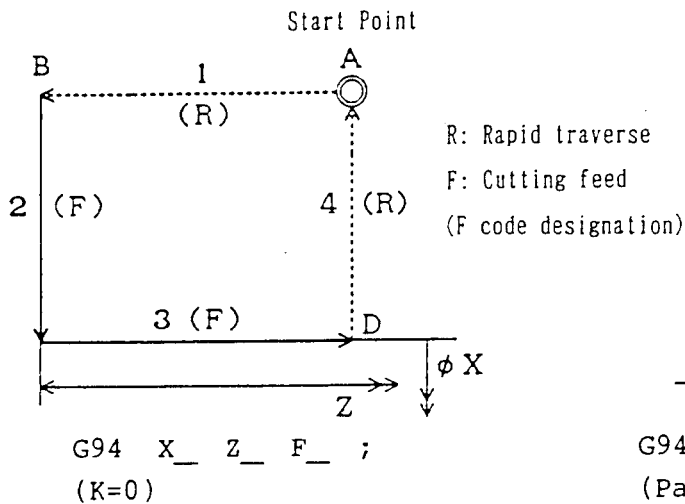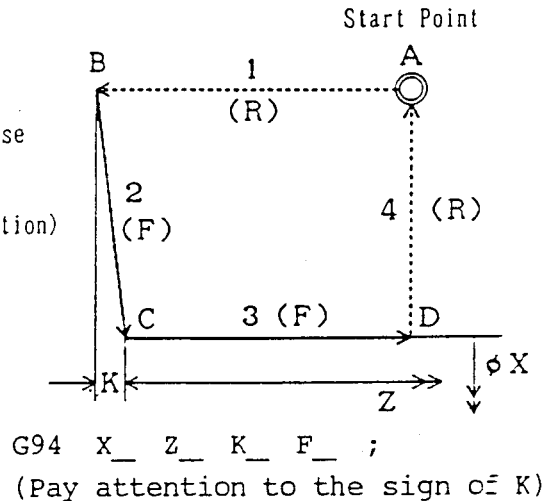
(Note 1) *Specify the position of the point C with X and Z.*
(Note 2) *K refers to an incremental value(radius value) when viewing the point B from the point C in the X-axis direction.*
(Note 3) *With E being commanded instead of F, shift speed of Operation 3 can be changed.*
  - *E > 0 : Operation 3 indicates cutting feed at speed E.*
  - *E = 0 : Operation 3 indicates rapid traverse.*

14.3.2 Sample Program

G00 X100. Z5. ;
G94 X50. Z-10. F1.0 ;
Z-15. ;
Z-20. ;
G00 Z100. ;

ø100.

ø50.

← -10.
← -15.
← -20.
← 5.

14 - 5
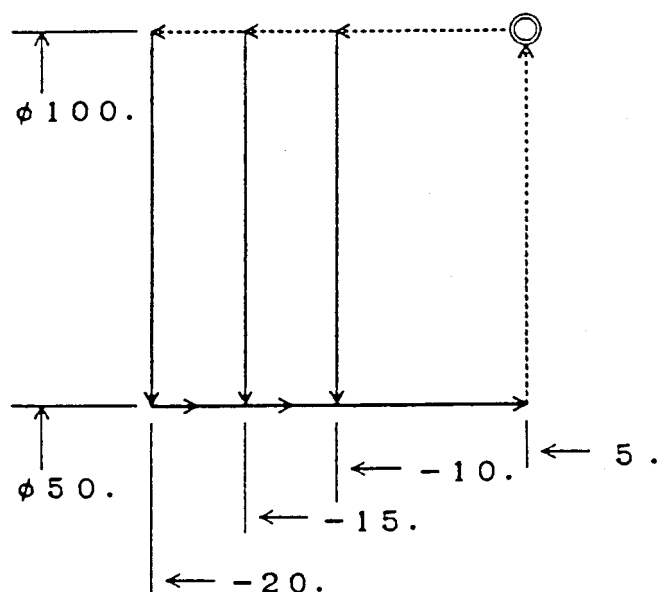
14.4 Cautions Concerning Single Type Fixed Cycle

14.4.1 Cautins

(1) The data X, Z, I, and K in the canned cycle are modal values.
Unless X, Z, I, and K are newly specified, the previous ones remain effective. In the G90/G92/G94 block, however, specify all required data.

(2) The canned cycle mode is cancelled by specifying the G code of Group 00 other than G04, or that of Group 01 other than G90, G92, and G94.

(3) In case of the SINGLE BLOCK mode, the actions 1-4 are performed by one cycle start.

(4) The canned cycle is not executed in the block without axis command, which is in the canned cycle mode.

(5) In the canned cycle mode, the M and S codes are output at the start of the action 1. The action 2 is performed upon completion of the M and S functions.

(6) An alarm results from the following conditions.

[1] When the two axes in the plane are not specified in the G90, G92, or G94 block.

[2] When the axis outside the plane is specified in the canned cycle mode.

[3] When the move direction of the action 1 becomes opposite to the specified move direction, depending on the I and K data.
(Example) G90 U-50. W-100. I60. F1.5 ;

[4] When the value of chamfering r is larger than the original stroke of the action 2 in the G92 canned cycle.

[5] When the plane (G17 to G19) is changed in a command block other than G90, G92, G94 during fixed cycle mode.

14.4.2 Associated Alarms

No.176   An error exists in the command of single type fixed cycle.

(#001)   An error exists in axis command.

- The two axes in the plane have not been specified in the fixed cycle command block.
- An axis outside the plane has been specified while in fixed cycle.

(#002)   An error exists in I/J/K value. (The shifting direction of approaching operation gets opposite to the original shifting direction.)

(#003)   Plane has been changed.

15. MULTIPLE REPETITIVE CYCLE

There are 7 kinds of multiple repetitive cycles provided to simplify the program.

G Code	Cycle Name	Remarks	
G70	Finishing cycle		
G71	I.D./O.D. roughing cycle (Type 1/2)	Capable of	Capable of tool nose
G72	End face roughing cycle (Type 1/2)	finishing with	radius compensation
G73	Closed loop cutting cycle	G70	
G74	End face cutting-off cycle	No tool nose R compensation applies.	
G75	Outer figure cutting-off cycle		
G76	Thread cutting cycle		

1. G70-G76 are the non-modal G codes of Group 00.
2. G70-G73 are available only for memory operation.

<Types of Complex Fixed Cycles>

Complex fixed cycles are classified into three types, i.e. A, B and C.

The table below shows functions available to each types.

Functions /Types	A	B	C
G70	○	○	○
G72 G72 Type 1	○	○	○
G71 G72 Type 2	✕	✕	○
G73	✕	○	○
G74 G75	✕	○	○
G76	○	○	○

○... Available

✕ ... Not available

15.1 Rough Planing of Inside & Outside Diameter (G71)

If the final shape is programmed as A→A'→B, rough planing is performed through the path, as shown below, which has been automatizally determined.
Whereas if the command (Z or W) of the Z axis is not present on the first block of the final shape, type 1 will be applied, if the command is present, type 2 will be applied.

15.1.1 Type 1

(F) G01
(R) G00
(Note) ... Follows A→A' in the final shape
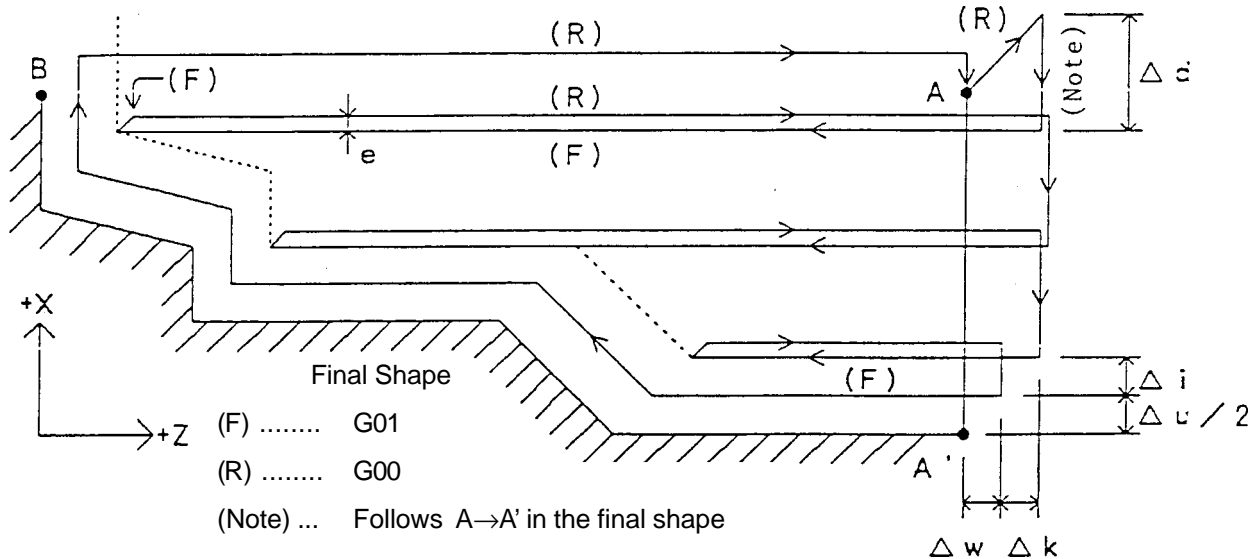
G71 P (ns) Q (nf) U±_ W±_ I±_ K±_ D_ F (1) S (1) ;
N (ns) .... ;

   .... ;

                         Movement along the final shape of
                         A-A'-B is specified through the
         F (2)   block from ns to nf sequentially.
         S (2)

N (nf) .... ;

   P : Sequential number (ns) of the first block of the final shaze
   Q : Sequential number (nf) of the last block of the final shape
   U : Finish toward X .... (Δu)
   W : Finish toward Z .... (Δw)
   I : Determination of the direction X of the final rough planing .... (Δi)
   K : Determination the direction Z of the final rough planing .... (Δk)
   D : Amount of cutting........ (Δd) ... Specified without sign.
   e : Amount of clearance (Parameter No.5145)

(1) Notes on type 1 of G71
   [1] Commands F and S ([2]) are disregarded which have been specified between (ns) and (nf), and commands which have been specified on the block of G71 or on the block before G71 ([1]) become effective.
   [2] Between A and A', commands must be made on the block of the sequential number (ns). Also, command of the Z axis cannot be contained.

[3] Between A' and B, monotonously increasing or decreasing pattern must be applied to both of X axis and Z.

[4] If G00 between A → A', cutting along A → A' is performed at G00.
If G01 between A → A', cutting along A → A' is performed at G01.
- If parameter No.5102, #5=1, override can be set to parameter No.5113 as a unit of per cent (%) for the amount of cutting (D command).

$$\Delta d = D \times \frac{\text{Parameter No. 6202}}{100}$$

(2) Clearance movement after planing

[1] After planing toward Z axis, it moves at the derection of 45 degrees.
- Set the amount of clearance (e) for parameter No.5145 by the value of radius.
- Movement to the X axis is not made in excess of (starting point + U + I).
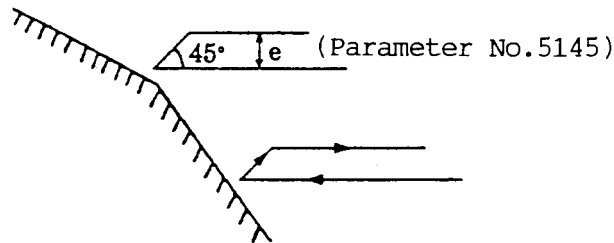
(3) Signs of commands U, W, I and K
Four kinds of shapes can be planed by G71. Each of these kinds of shapes are planed by the tool in parallel the Z axis.
Signs of commands U, W, I and K vary depending on shapes.

15 - 3
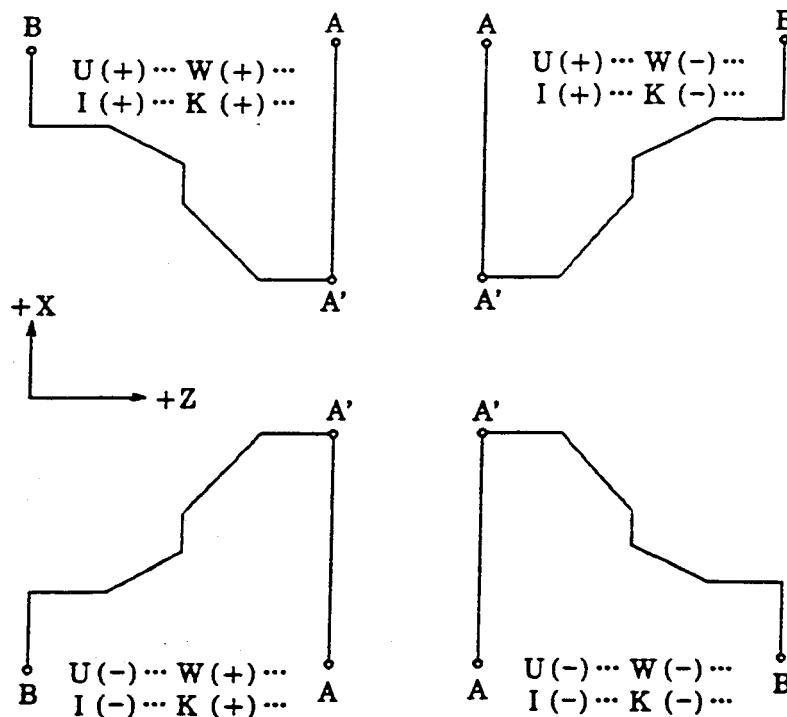
(4) In case where I and K are omitted

Units of the cutting amount (d) are planed excepting the finish carts (U and W) and the planing (rough finish) is performed in accorcance with the final shape program. However, the rough finish can be omitted by setting parameter No.5102, #4 to 1. (In this case, it returns to the starting point through the dotted line after the final rough planing.)
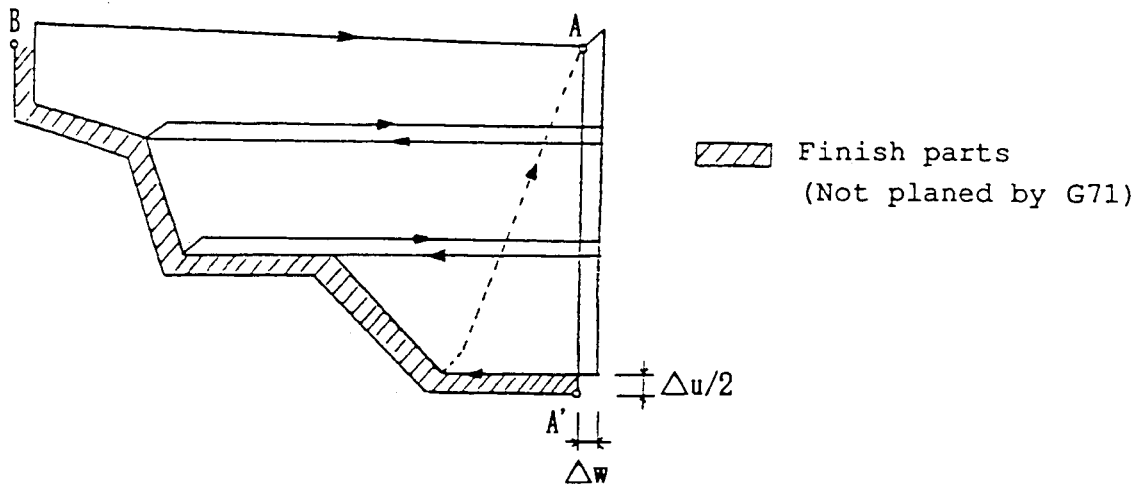
(5) In case where I and K are specified

Units of the cutting amount (d) are planed excepting the finish parts + rough finish parts (U+I, W+K), and the final rough finish parts (I and K) are planed.

In this case, the rough finish planing is performed regardless of parameter No.5102, #4.
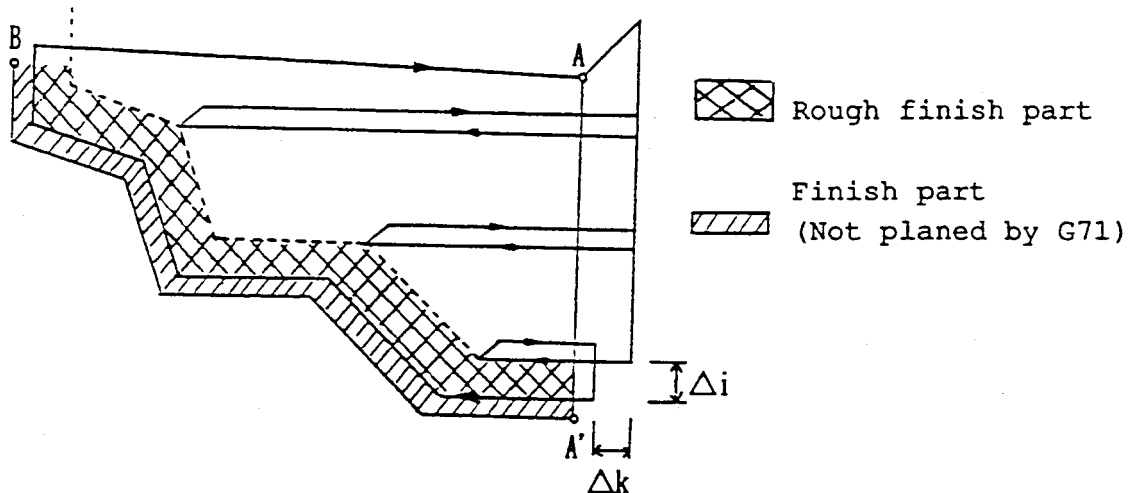
If the final rough planing part is (C→C') + I ≤ D, (C→C') + I is planed at a stretch.

15.1.2   Type 2
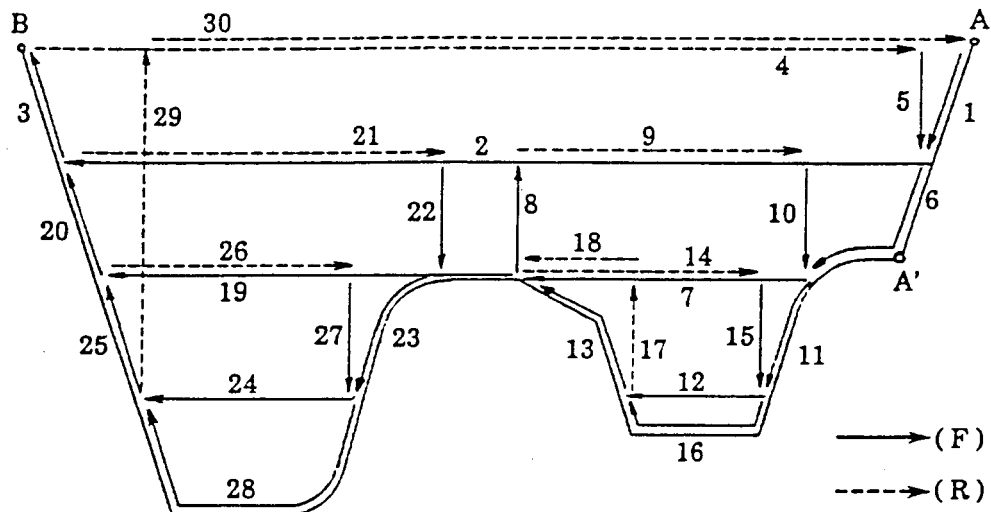
(1)   Notes on Type 2 of G71

[1]   In the case of type 2, a shape whose direction toward X axis is not monotonous incremental (or decremental) is permitted.
However, the shape of the direction toward Z axis must be incremental (or decremental).

[2]   Between A and A', it must be specified by the blocks of sequential number (ns). Also, commands of the Z axis must be contained.

[3]   If the amount of movement of the Z axis between A and A' is 0, the same sendings (G00/G01) as for the finish shape are applied to the cutting (Note) along the A → A'.

[4]   If the Z axis moves between A and A', the sending (G01) is applies to the cutting (Note) along the A → A', even when the sending (G00) is applied to the finish shape.

(2)   Movement after planing

[1]   Vartical movement is made after cutting along the shape.
Amount of movement (e) is set for parameter No.5145 by the value of radius.

[2]   Movement toward the X axis is not made in excess of (starting point + U + I).
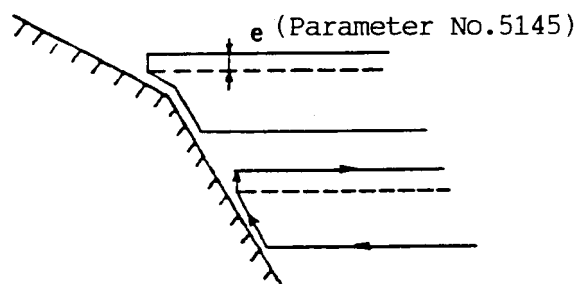
[3] The planing after the planing of one pocket is performed as shown below.

σ (Parameter No.514␣

(F) (R)
(F)
(R)
(F) (F)
(F)
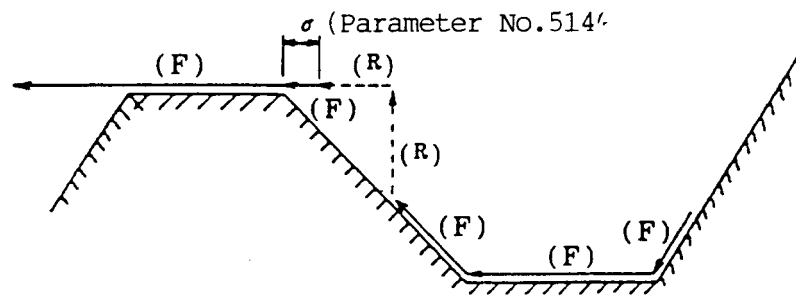
(3) Offset of edge R

Type 2 can be planed during the automatic offset mode of the edge R (G143 and G144) . The finish shape is a shape for which the offset is canceled at the starting point of the cycle. (For the details of the edge R during the complex fixed cycle, refer to "15.8 Notes".)

Finish Profile

Tool Nose Center Path after Tool Nose Radius Compensation

(I, K, U, W = 0)
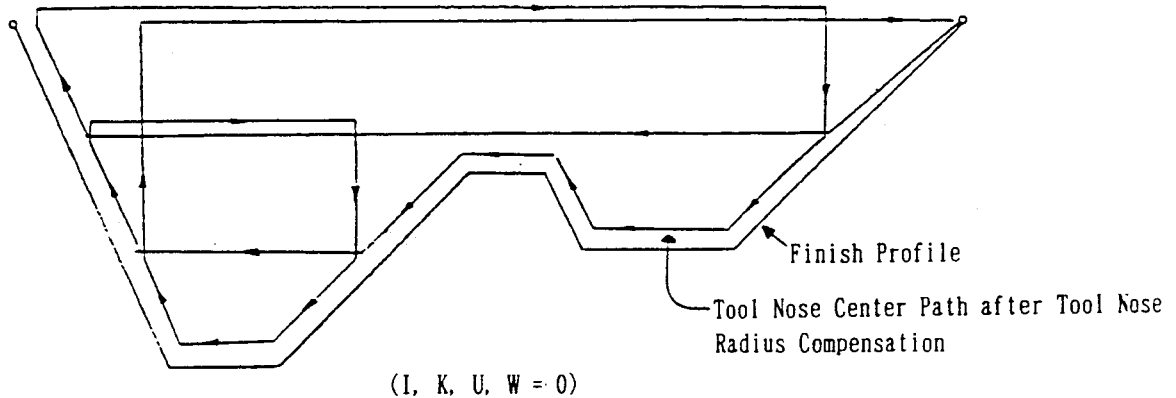
Otherwise, the same planing as for type 1 is performed.

15.1.3 Program Example

(1) Type 1
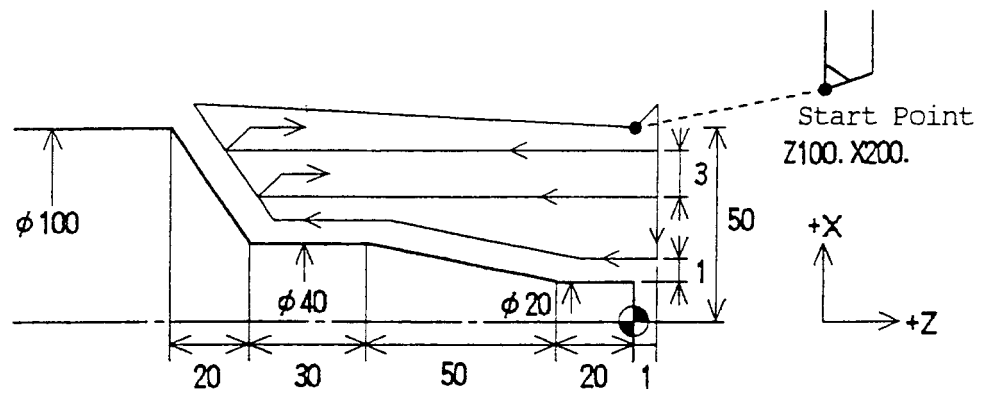

```
          G00  X200.  Z100. ;
          X110.  Z5. ;
          G99  G01   X100.  Z0  F2.0 ;
          G71  P101  Q102  U2.  W1.  D3.  F0.1 ;
N101      G00  X20. ;
          G01  Z-20.  F0.05 ;
          X40.  Z-70. ;
          Z-100. ;
N102      X100.  Z-120. ;
           ;
          G70 P101 Q102 ;
```

(2) Type 2
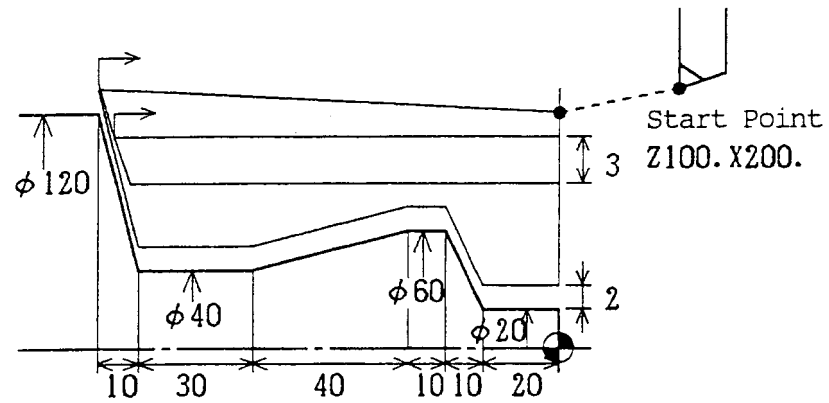

```
        G00  X200. Z100. ;
        X130. Z5. ;
        G99  G01  X120. Z0 F2.0 ;
        G71  P121  Q122  U4. D3. F0.1 ;
N121    G00  X20. W0 ;
        G01  Z-20. F0.05 ;
        X60.  Z-30. ;
        Z-40. ;
        X40.  Z-80. ;
        Z-110. ;
N122    X120. Z-120. ;
          ;
        G70  P121  Q122
```

15.2　Rough Planing Cycle of End Side (G72)

Whereas the rough planing for G71 is performed toward Z, the rough planing for G72 is performed toward X. Otherwise, the same planin as for G71 is performed.

15.2.1　Type 1


```
+X  (F) ··· G01
    (R) ··· G00
    (Note) Follow A → A'
```
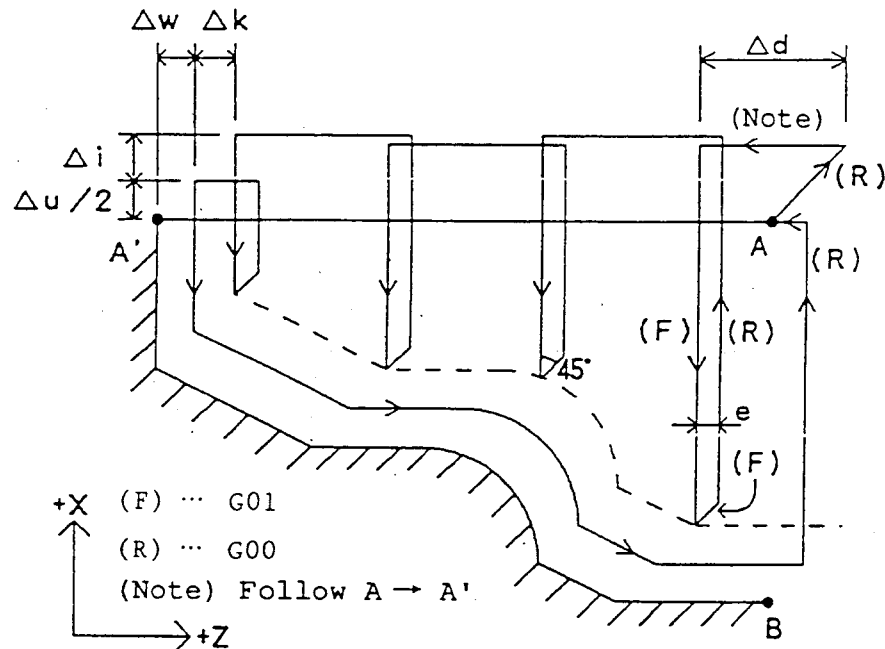
G72 P (ns) Q (nf) U± W± I± K± D_ F (1) S (1) ;

N (ns) .... ;

.... ;

F (2)

S (2)

Finish shape movement of A-A'-B is specified in the block from the sequential numbers (ns) to (nf).

N (nf) .... ;

P : Sequential number (ns) of the top of the finish shape block
Q : Sequential number (nf) of the last block of the final shape
U : Finish toward X .... (Δu)
W : Finish toward Z .... (Δw)
I : Last rough cutting part toward X .... (Δi)
K : Last rough cutting part toward Z .... (Δk)
D : Amount of cutting........ (Δd) ... Specified without sign.
e : Amount of clearance (Parameter No.5145)

(1) Notes on type 1 of G72
   [1] Commands F and 5 (2) between (ns) and (nf) are disregarded, and commands on the block of G72 or less (1) become effective.
   [2] Commands between A and A' must be made by the block of the sequential number (ns). Also, commands of the X axis cannot be contained.

(2) Signs of commands U, W, I and K

There are kinds of shapes which can be planed by G72. Each of then are planed in parallel with the X axis of the tool.

Signs of the commands U, W, I and K vary depending on shapes.
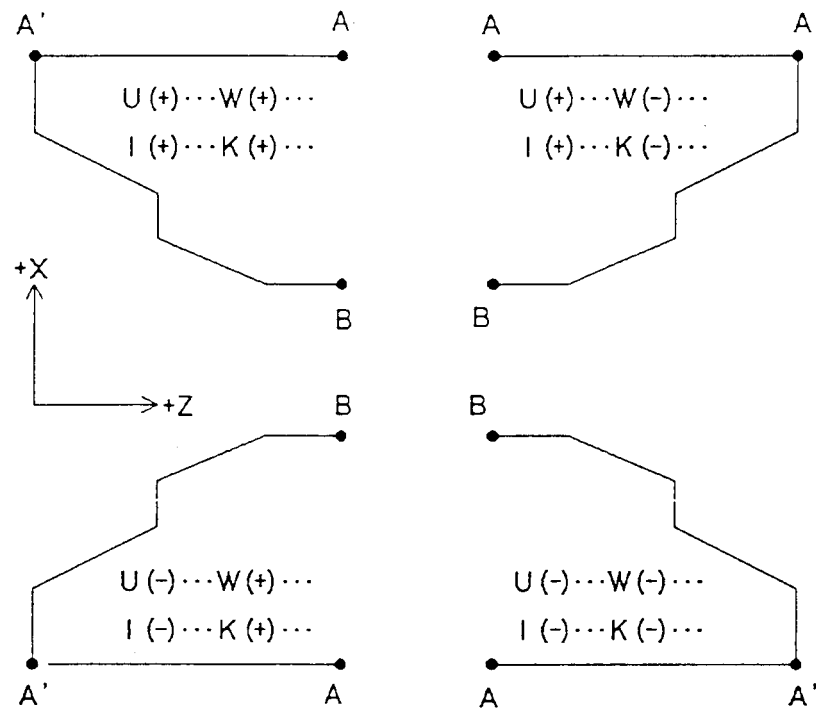
15.2.2 Type 2
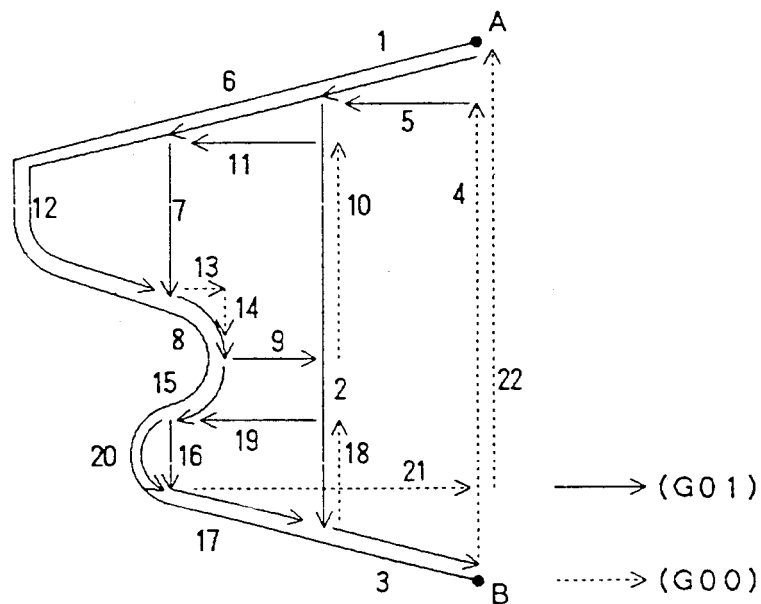
15.2.3   Program Example
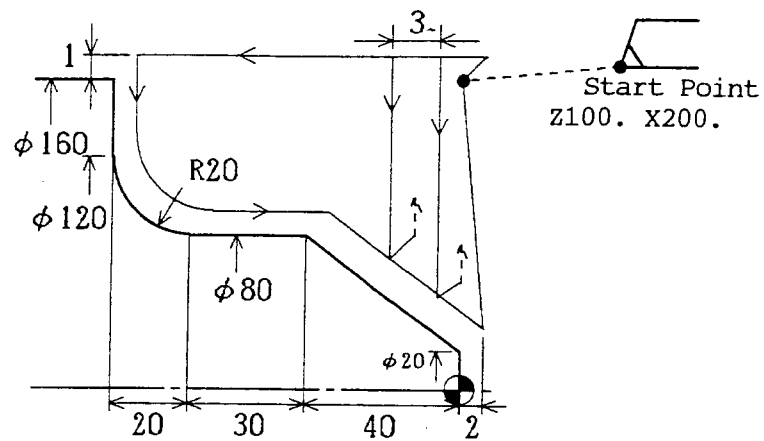

```
            G00  X200. Z100. ;
            X170. Z5. ;
            G99  G01  X160. Z0  F2.0 ;
            G72  P201  Q202  U2.  W2.  D3.  F0.1 ;
N201        G00  Z-90. ;
            G01   X120.  F0.05 ;
            G03   X80.  Z-70.  R20. ;
            G01  Z-40. ;
N202        X20.  Z0 ;
               ;
            G70  P201  Q202 ;
```

15.3 Planing Cycle of Close Loop (G73)

This is a cycle in which planing is repeated by shifting the planing patter little by little.


```
(R).....G00
(Note) .Follow A->A'->B.
```
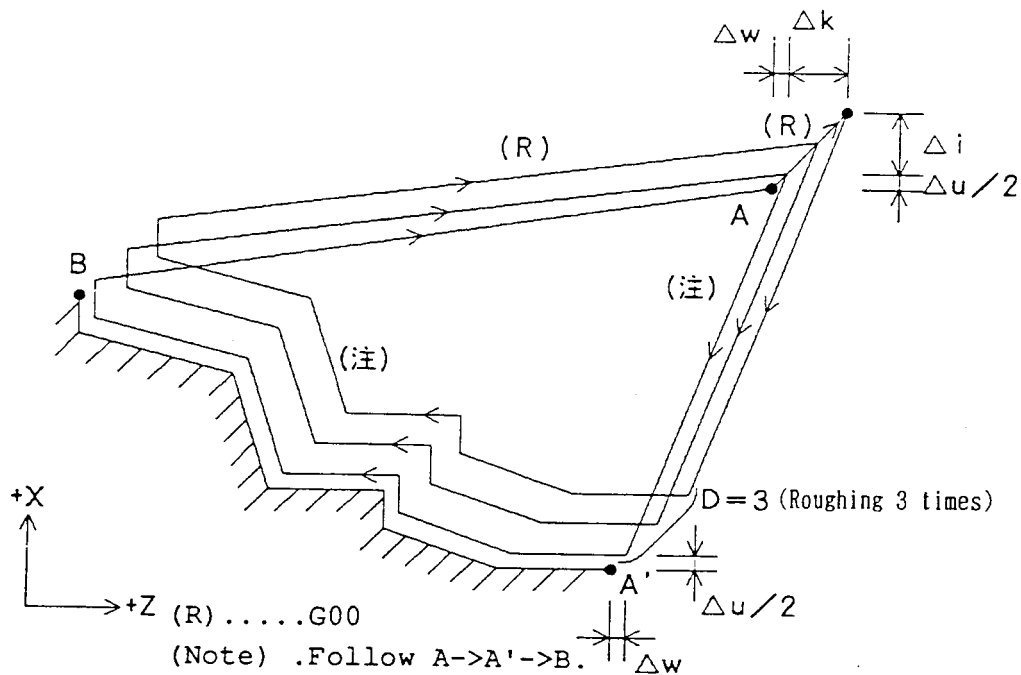
G73 P̲(̲n̲s̲)̲ Q̲(̲n̲f̲)̲ U±̲ W±̲ I±̲ K±̲ D̲ F̲(̲1̲)̲ S̲(̲1̲)̲ ;
N (ns) .... ;

   .... ;

                        Finish shape movement commands of A-A'-B are
        F̲(̲2̲)̲   made in the block of the sequential numbers from
        S̲(̲2̲)̲   (ns) to (nf).

N (nf) .... ;

   P : Sequential number (ns) of the top of the finish shape block
   Q : Sequential number (nf) of the last block of the final shape
   U : Finish toward X ....   (Δu)
   W : Finish part toward Z .... (Δw)
   I : Part toward X .... (Δi)
   K : Part toward Z .... (Δk)
   D : Number of rough planing

(1) Notes on type 1 of G73
     Commands F and S between (ns) and (nf) ([2]) are disregarded, and commands on
     the block G73 or less ([1]) become effective.

(2) Signs of commands U, W, I and K

There are four kinds of shapes which can be planed by G73.

Signs of the commands U, W, I and K vary shapes as follows.

(3) Program Example

Start Point
Z100. X200.
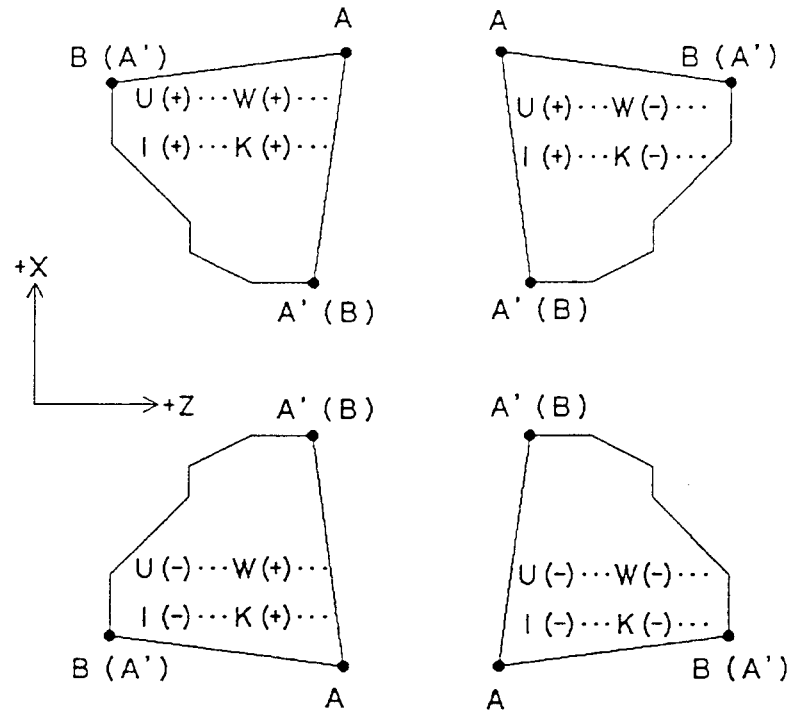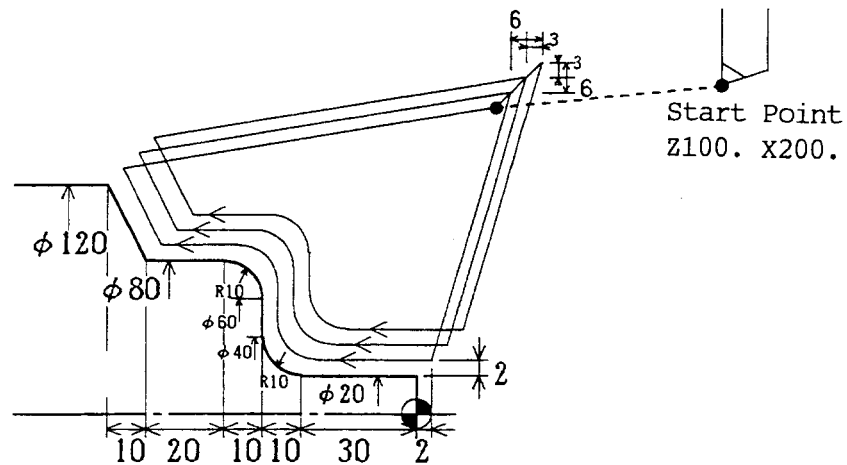
```
        G99 G00 X200. Z100. ;
        X160. X20. ;
        G73 P301 Q302 U4. W2. I6. K6. D3 F0.1 ;
N301    G00 X20. Z-0 ;
        G01 Z-40. R10. F0.05 ;
        X80. R-10. ;
        Z-70. ;
N302    X120. Z-80. ;
           ;
        G70 P301 Q302 ;
```

15.4 Finish Cycle (G70)

After planing roughly through the use of G71 to G73, planing by G70 can be performed.

G70 P (ns)   Q (nf) ;

   P : Sequential number (ns) of the top of the finish shape block

   Q : Sequential number (nf) of the last of the finish shape block

- Commands in G71 to G73 are disregarded, and commands F and S between (ns) and (nf) become effective.
- A maximum of three finish shapes performed in G71 to G73 can be saved simultaneously. The fourth shape and onward will require longer cycle time, because the memory is searched again when executing G70.
- Save of the finish shapes can be deleted by executing G70 or reset.
  (Execution of G70 is still available even after memory has been erased.
  When G70 has been executed for a shape without memory, however, a slightly longer time is required for memory to be searched over again.)

```
          :
          :
[1] ┌   G71  P101  Q102 ...... ;
    │   N101 ...... ;
    │      :
    │      :
    │      :
    │   N102 ...... ;
    └      :

[2] ┌      :
    │   G72  P201  Q202 ...... ;
    │   N201 ...... ;
    │      :
    │      :
    │      :
    │   N202 ...... ;
    └      :

[3] ┌      :
    │   G73  P301  Q302 ...... ;
    │   N301 ...... ;
    │      :
    │      :
    │      :
    │   N302 ...... ;
    └      :
```

```
          :
[4] ┌   G71  P401  Q402 ...... ;
    │   N401 ...... ;
    │      :                    (Memorizing [4] clears [1] )
    │      :
    │      :
    │   N402 ...... ;
    └      :
          :
          :
          :
          :
       G70  P401  Q402 ;  — [4]
          :
       G70  P101  Q102 ;  — [1]  → Cuts after
          :                          searching the
       G70  P201  Q202 ;  — [2]     memory
          :
       G70  P301  Q302 ;  — [3]
          :
          :
          :
```
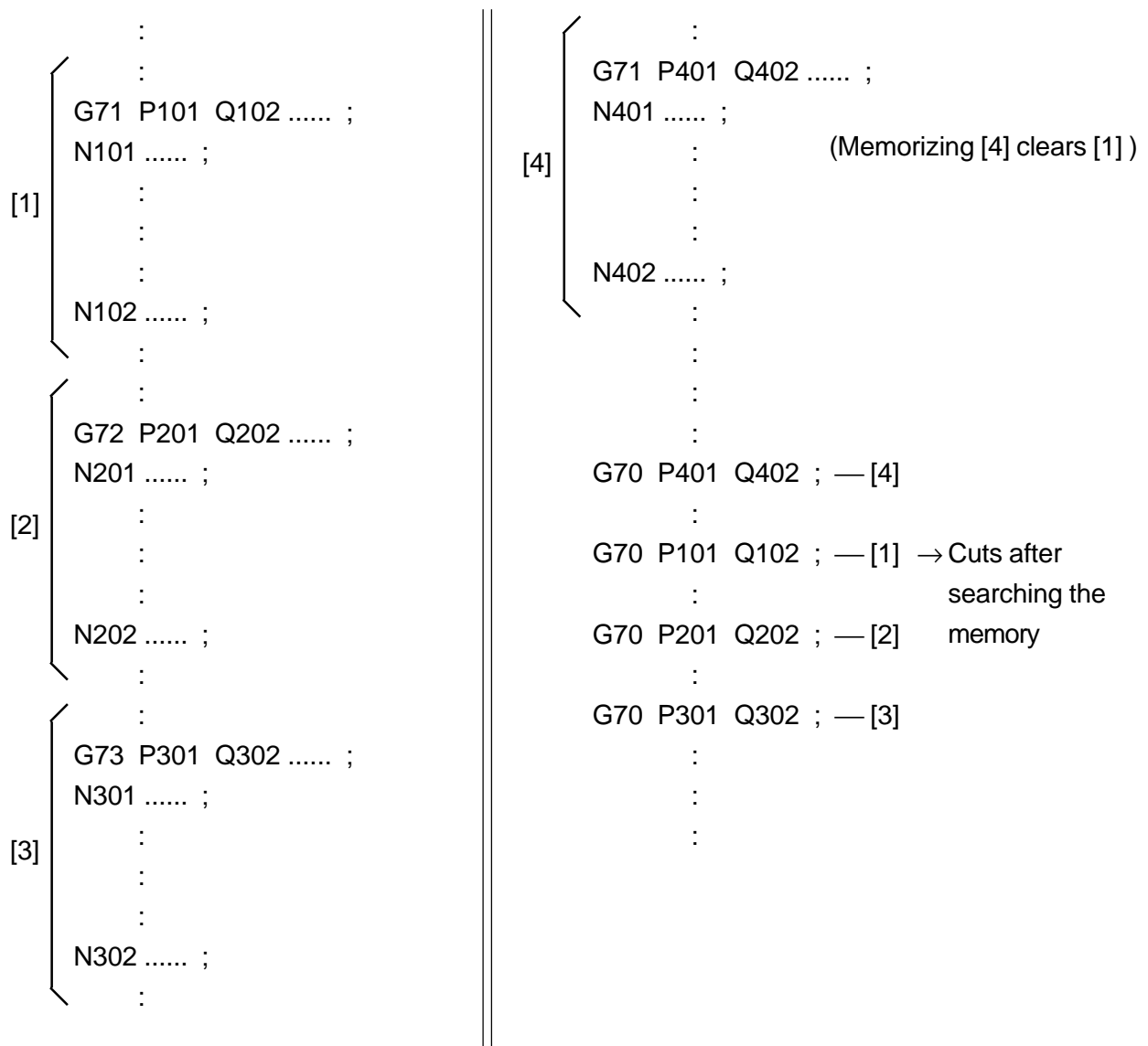
15.5 Edge Cutting Cycle (G74)

This command enables the disposal of the wastes generated during the cutting of outside diameter. Also, if X (U) and I are omitted, the deep hole drill cycle toward the Z axis will be entered.
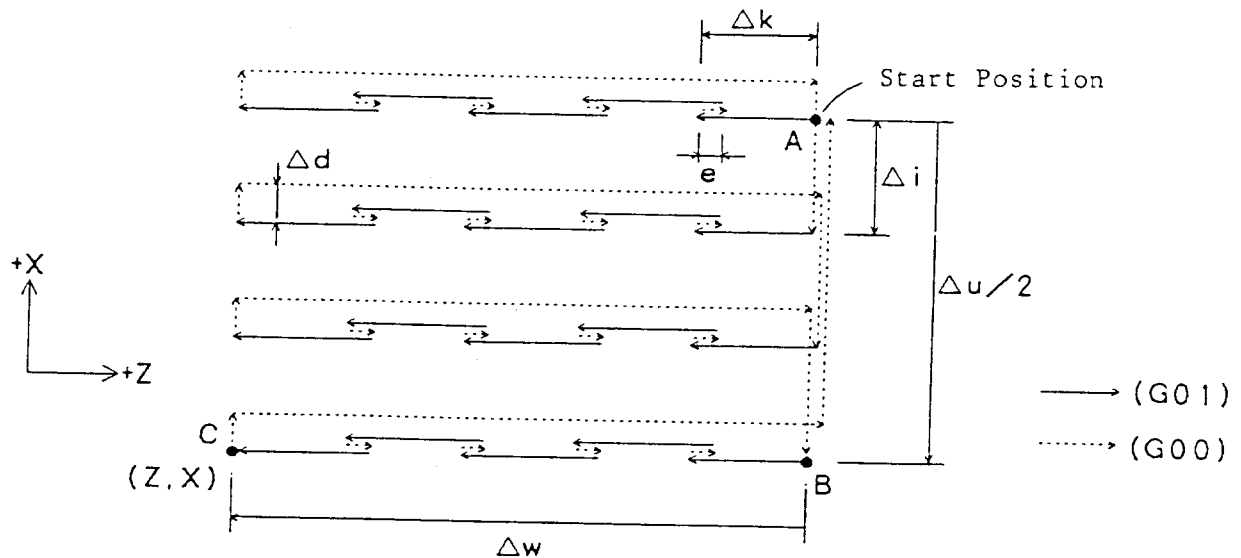
G74 X (U)___ Z (W)___ I___ K___ D___ F___ R___ ;

X : Point C

U : Incremental amount toward X of A → C (Δu)

Z : Point C

W : Incremental amount toward Z of A → (Δw)

 I : One-time cutting amount toward X (absolute value) (Δi)

K : One-time cutting amount toward Z (absolute value) (Δk)

D : Movement amount of tool at the bottom of cutting (Δd)

F : Feeding speed

e : Return amount (parameter No.5147)

R : If 0 is specified or omitted, it returns by e during Z return movement (Refer to the figure). If 1 is specified, it returns to the start point (A - B) every time.

(Note) At the bottom of cutting, it moves from B to A. If X does not move, it moves toward the sign of D.

(1) Program example (Deep hole cycle)

Start Point Z100. X200.
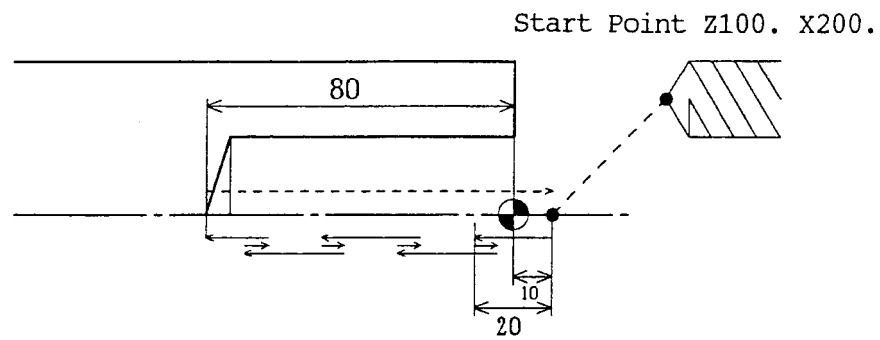
G98 G00 X200. Z100. ;
X0 Z10. ;
G74 Z-80. K20. F600 ;

15.6 Outside Diameter Edge Cutting Cycle (G75)

This command enables the disposal of the wastes generated during the planing of the edge. Also, if Z (W) and K are omitted, grooving processing and edge cutting processing can be performed.
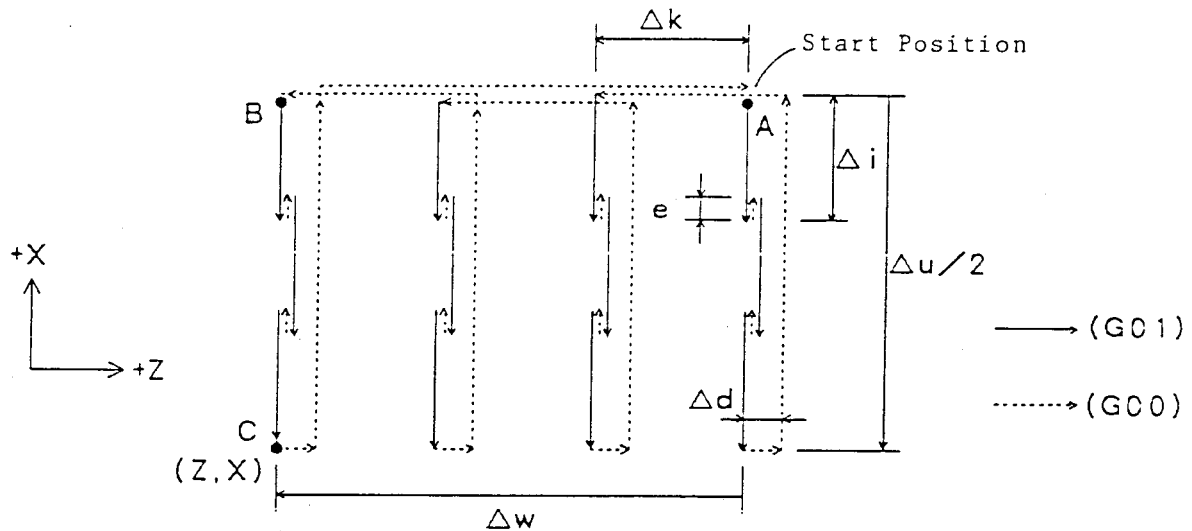
G75 X (U)___ Z (W)___ I___ K___ D___ F___ R___ ;

  X : Point C

  U : Incremental amount toward X of A→C (Δu)

  Z : Point C

  W : Incremental amount toward Z of A→C (Δw)

  I : One-time cutting amount toward X (absolute value) (Δi)

  K : One-time cutting amount toward Z (absolute value) (Δk)

  D : Movement amount of tool at the bottom of cutting (Δd)

  F : Feeding speed

  e : Return amount (parameter No.5147)

  R : If 0 is specified or omitted, it returns by e during X return movement (Refer to the figure). If 1 is specified, it returns to the start point (A - B) every time.

(Note) At the bottom of cutting, it moves from B to A. If Z does not move, it moves toward the sign of D.

(1) Program example (X-Direction cutting-off working)

G97 S600 ;
G00 X200. Z100. ;
G96 S60 ;
X100. Z-80. ;
G75 X-4. I20. F600 ;

15.7   Combined Type Thread Cutting Cycle (G76)

This command enables the wrenching cycle with the fixed amount of planing.

15.7.1   Command Format
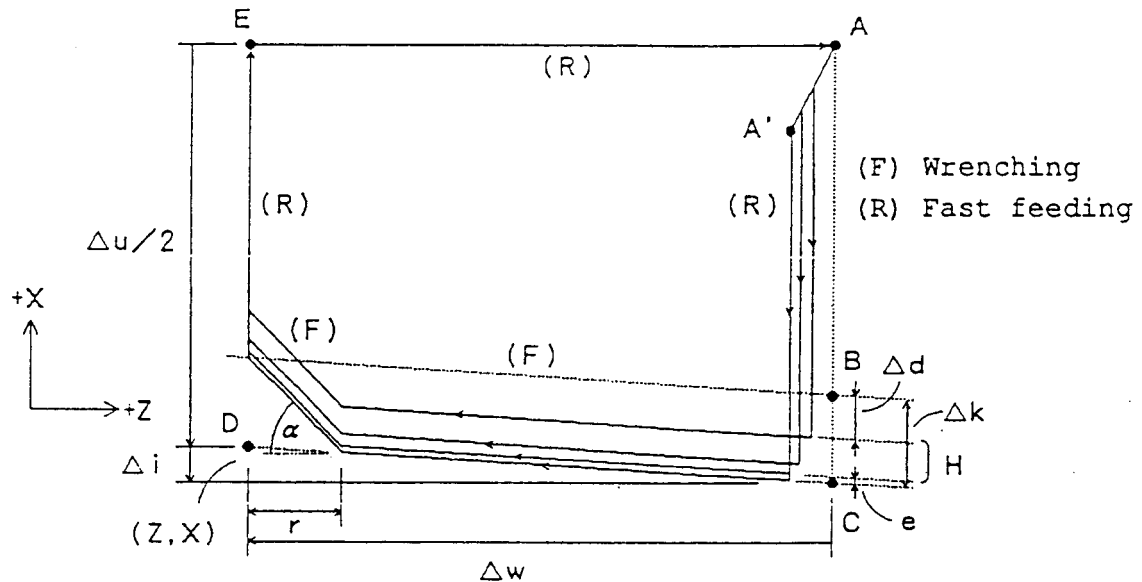
G76 X (U)____ Z (W)____ I____ K____ D (H)____ A____ Q____ F____ ;

- X :   Point D
- U :   Incremental amount toward X of A → D (Δu)
- Z :   Point D
- W :   Incremental amount toward Z of A → D (Δw)
- I :   Incremental amount of D → C (Δi)
- K :   Size of screw (Δk)
- D :   First cutting amount (Δd)
- H :   Number of cuttings
- A :   Edge angle (Set within the range from 0 to 120 degrees.)
- Q :   Shift amount of starting angle of wrenching
- F :   Lead of screw
- e :   Finishing part ...... (parameter No. 5149)
- r :   Chanferring amount (parameter No. 5111)
- α :   Chanferring angle (parameter No.5112)

(1) Notes on G76

   [1] Wrenching of the lead F is performed between C and D. Otherwise, fast feeding
        is performed.

   [2] Value of r (parameter No.5111) is set within the range from 1 to 127 (if the lead is
        L, it is set withing the range from 0.1L to 12.7L)

   [3] Value of α (parameter No.5112) is set within the range from 1 to 89 degrees.

   [4] Using P command, select one among 4 kinds of depth setting methods.
        As for details, see the next page on.

[5] Using Q command, you can shift a threading start angle. Use this command to facilitate multiple thread cutting.

[6] In single block, stopping takes place at A and A' points. (One time of cycle start applies for cutting from A' back to A.)

[7] Finishing stock e is set to parameter No. 5149.

[8] When depth of cut for one time gets smaller than clamp value g. (parameter No.5148), the depth of cut is clamped to g.

[9] The number of times of final finishing can be set in parameter No.5129.

[10] As for details including screw lead (F)/threading retract (Feed Hold while in thread cutting)/chamfering/taper screw/thread cutting starting position (Q), etc., see "7-1 Single Tye Thread Cutting Cycle (G92)".

[11] With P2, P4 (staggered cutting) commanded, specify G in an even number (2, 4, 6, ...).

[12] For the last two times of P2/P4 (staggered) rough cutting, cut their margins each by half.

[13] Throughout the entire G76 process, spindle override is held invalid.

[14] Rapid traverse override of X-axis release (C → D) can be clamped with No.5114 value. (As for details, see Item 14.2.5.)

15.7.2 Depth Setting Method

(1) Fixed Cut Amount/Half Side Cutting (P1 assignment)

Parameter No.5148
(When one-time cut amount < g)

For H command
Cut at H rotation.

Cutting repeated (for finishing) by no. of times set in Parameter No.5129.
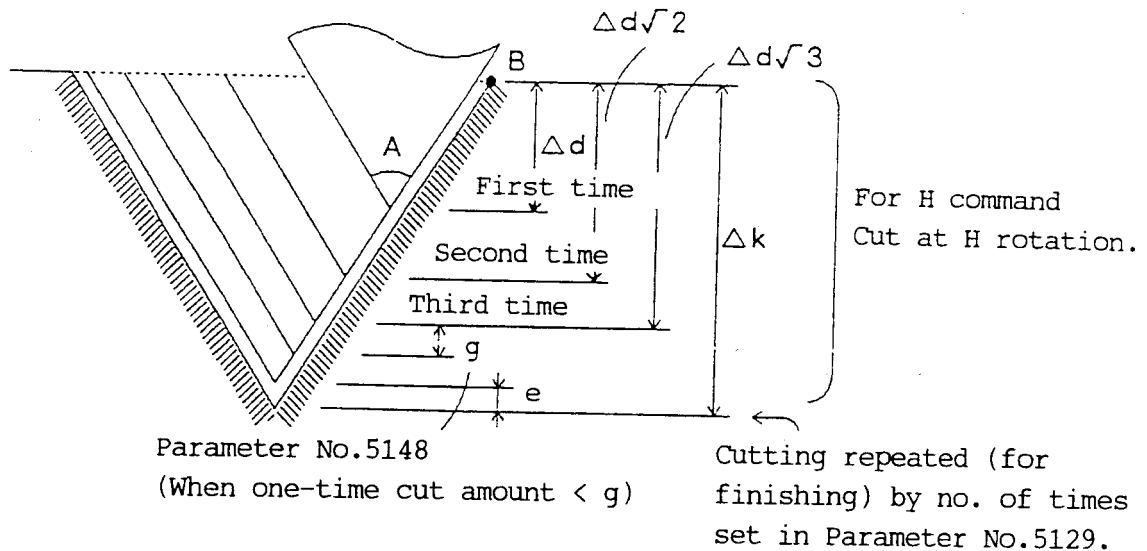
Depth of cut on 1st time .... Δd
2nd time ...... $\Delta d\sqrt{2}$
3rd time .... $\Delta d\sqrt{3}$
4th time .... $\Delta d\sqrt{4}$
5th time .... $\Delta d\sqrt{5}$

(2) Fixed Cut Amount/Staggered Cutting (P2 assignment)
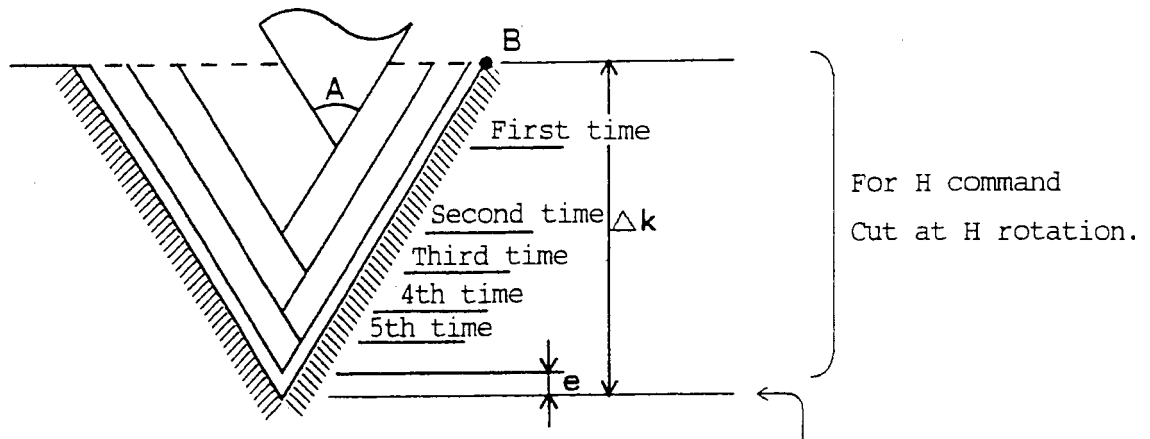
For H command
Cut at H rotation.

Cutting repeated (for finishing) by no. of times set in parameter No.5129.

Depth of cut on 1st time ...... Δd, $\dfrac{\sqrt{2}}{2}$

2nd time...... Δd, $\sqrt{2}$

3rd time ...... Δd, $\dfrac{\sqrt{2} + \sqrt{4}}{2}$

4th time ...... Δd, $\sqrt{4}$

5th time ...... Δd, $\dfrac{\sqrt{4} + \sqrt{6}}{2}$

6th time ...... Δd, $\sqrt{6}$

(3) Fixed Cut Amount/Half Side Cutting (P3 assignment)

For H command
Cut at H rotation.

Cutting repeated (for finishing) by no. of times set in parameter No.5129.
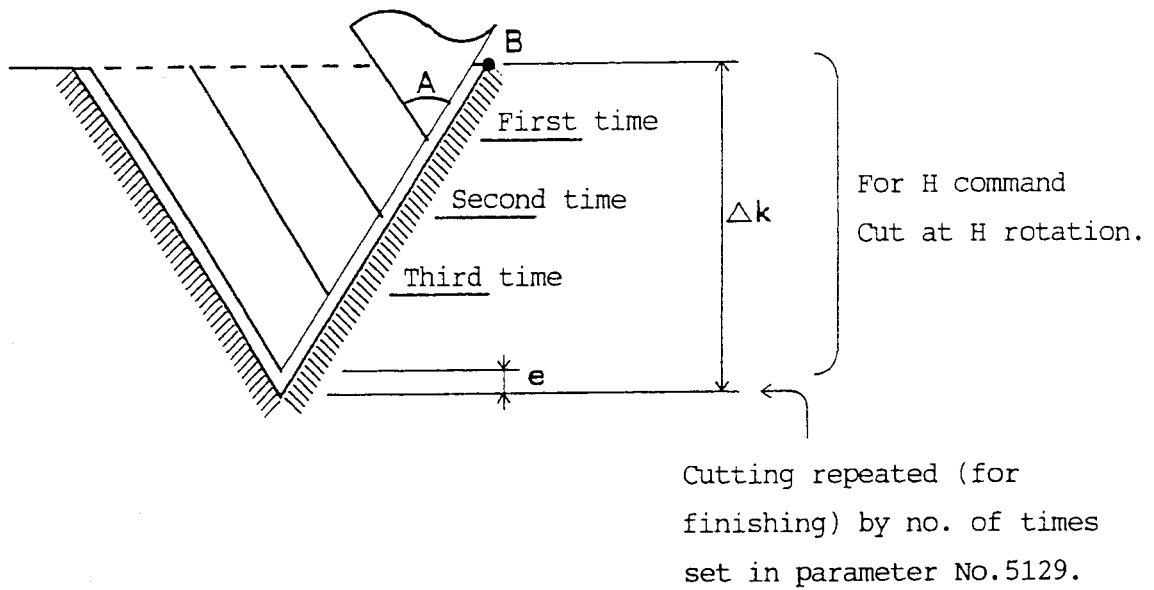
Depth of cut on 1st time .... Δd
2nd time ...... $\Delta d \cdot 2$
3rd time .... $\Delta d \cdot 3$
4th time .... $\Delta d \cdot 4$

(4) Fixed Cut Amount/Staggered Cutting (P4 assignment)

For H command
Cut at H rotation.

Cutting repeated (for finishing) by no. of times set in parameter No.5129.
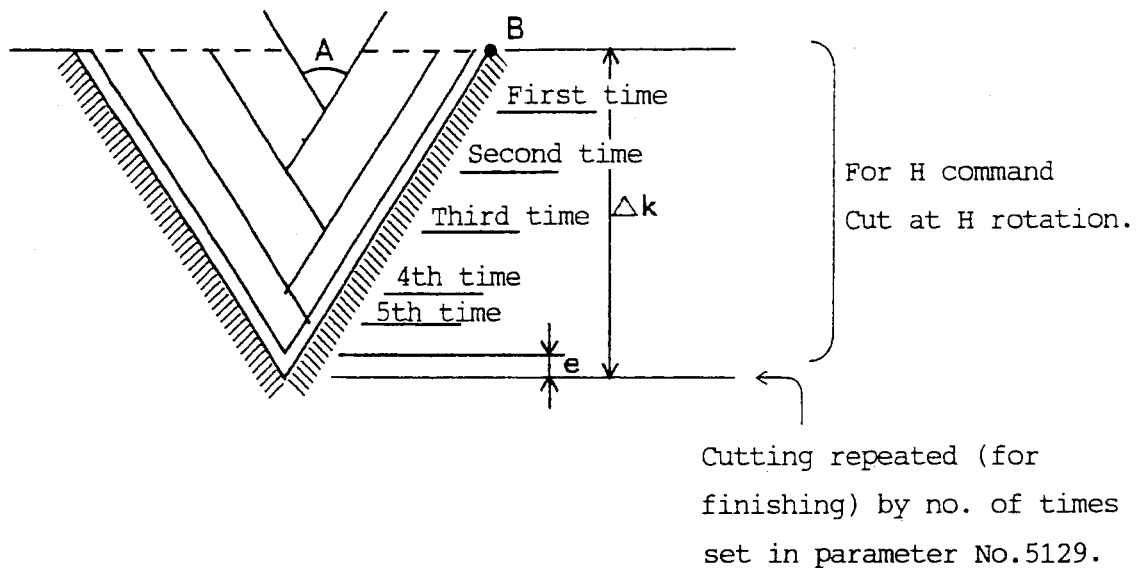
Depth of cut on 1st time .... Δd
2nd time ...... $\Delta d \cdot 2$
3rd time .... $\Delta d \cdot 3$
4th time .... $\Delta d \cdot 4$

15.7.3   Program Example
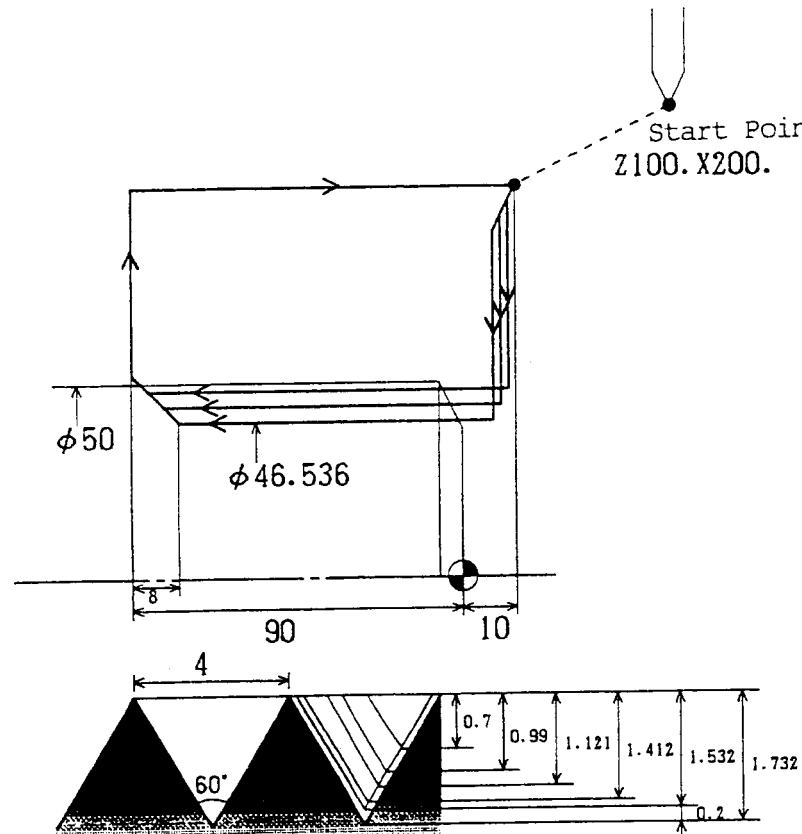

```
G00 X200. Z100. ;
X100. Z10. ;
G76 X46.536 Z-90. K1.732 D0.7 F4.0 A60 ;
```

(Note) *Where parameters are: No. 5111 = 20*
No. 5112 = 45
No. 5148 = 200
No. 5149 = 200

15.8 Cautions Relating to Combined Type Fixed Cycle

15.8.1 Notes

(1) No. of Blocks of Finish Shape
  [1] Finish shapes used in G70 to G73 have a maximum of 45 blocks.
     However, if chamfered/corner R is specified during the finish shaping, less number of blocks are memorized because they are divided into two parts.
  [2] Alarm is generated if the finish shape exceeds 45 blocks.
  [3] A maximum of 44 mountain shapes can be specified during the finish shaping of type 2 of G71 and G72.
     A maximum of 45 valley shapes can be specified during the finish shaping of type 2 of G71 and G72.
  [4] If the automatic offset of edge R is made to G71 and G72, the number of blocks which can be memorized may be slightly reduced.
     Refer to (3).

(2) Start and End Positions of Finish Shape
  [1] Values of Xs of starting point of G71 and terminal point of the finish shaping must be set equal.
  [2] Values of Zs of starting point of G72 and terminal point of the finish shaping must be set equal.
  [3] Starting points of G71 to G73 and G70 must be overlapped.

(3) Tool Nose R Compensation
  [1] G70 to G73 can be specified during the automatic offset mode of the edge P. (G143 and G144) . In each shapes, cancellation is made at the starting point of the cycle.
     The offset direction is determined by the starting point of the cycle, terminal point of the first block of the finish shape and terminal point of the finish shape. (The center of the edge passes through the triangle.)
  [2] If the first block of the finish shape is other than G00 and G01, offset is not made.
  [3] Offset is made to G70 and G73.
  [4] Offset is not made to the rough planing of type 1 of G71 and G72.
  [5] Offset is made by the parameter to the rough planing of the type 2 of G71 and G72. (No.5102, #6 = 1 $\rightarrow$ Offset is made.)
  [6] Offset is made to the rough finish of G71 and G72.
     (If offset is made to the shape outside the corner, link must be made as an arc.)
  [7] Total of the number of the finish shapes before offset (including the blocks divided by chamfered/corner R) and the number of parts linked by an arc exceed 60 blocks, alarm is generated.

[8] Upon rough finish of type 1 of G71 and G72, only the first block is corrected inversely.
- First block is G00.
- Z (when G71) and X (when G72) are not moved on the first block.
- The corner of the first block to the second block does not exceed 90 degrees.

(4) Subprogram cannot be called from the finish shape.
   Also, Macro statement cannot be used in the finish shape.

(5) The first and last sequential numbers of the finish shape cannot be the same.
   Also, same numbers cannot be used in a program.

(6) After the execution of the cycle of G71 to G73, it is resumez from the blocks following the last block of the finish shape.
   After the execution of the G70 cycle, it is resumed from the block following G70.

(7) When executing G71 and G72 on a single block, the following notes must be considered.
- It is stopped on the newly created block (an arc linking outside the corner) due to the offset of the edge R.
- It is stopped on the block dividing chamfered/corner R.
- Upon the rough planing, it is stopped at every node of during each operation.
- Every block whose movement amount is 0 is stopped.

(8) Command G which can be made within the finish shape is as follows.

Code	Group	Function	Remarks
G00	01	Location setting	
G01	01	Direct clearance	
G02	01	Arc supplement (clockwise)	Herical/SIN supplement not available
G03	01	Arc supplement (counterclockwise)	Herical/SIN supplement not available
G04	00	Dwell	
G09	00	Exact stop	
G61	13	Exact stop mode	
G64	13	Planing mode	
G96	17	Constant control of cycle speed	
G97	17	Cancellation of constant control of cycle speed	
G98	05	Feeding per minute	
G99	05	Feeding per rotation	
G196	17	Constant control of cycle speed (rear side)	Optional function

*Do not make G commands other than described above in the finish shape.

(9) How to Command Type 1/Type 2 in G71/G72

When one of the following applies, it is judged as Type 2. In other cases, it is judged as Type 1.

When "R1" has been commanded in the same block as G71/G72.

When two axes of X and Z have been commanded in the start block of the finish shape.

15.8.2 Related Parameters

No.5102, #2   Be sure to set 0.

No.5102, #4 = 1   Do not finish the rough planing in G71 and G72.

        = 0   Finish the rough planing in G71 and G72.

No.5102, #5 = 1   Override of cutting amount of G71 and G72 is effective.

        = 0   Override of cutting amount of G71 and G72 is inffective.

No.5102, #6 = 1   Offset of the tool nose R of G71/G72 type 2 is valid.

        = 0   Offset of the tool nose R of G71/G72 type 2 is invalid.

No.5102, #7 = 1   Command A of G76 does not have a decimal point.

        = 0   Command A of G76 has a decimal point.

No.5111   Chanferring amount of the wrenching.

         $1 \sim 127$ $(0.1L \sim 12.7L)$ (Outside the range $\rightarrow 10$)

No.5112   Chanferring angle of the wrenching.

         $1 \sim 89°$ (Outside the range $\rightarrow 45$)

No.5113   Override of the cutting amount of G71 and 072.

         $1 \sim 255\%$ (Outside the range $\rightarrow 100$)

No.5114   Rapid traverse override clamp value for X-axis release in threading cycle.

         $1 \sim 100\%$ (Out of range $\rightarrow 100$)

No.5145   Movement amount of G71 and G72 (e)

         $0 \sim 99999999$ (Input unit)/(Outside the range $\rightarrow 0$)

No.5146   Clearance amount ( ) of G71 and G72.

         $0 \sim 99999999$ (Input unit)/(Outside the range $\rightarrow 0$)

No.5147   Return amount of G74 and G75 (e)

         $0 \sim 99999999$ (Input unit)/(Outside the range $\rightarrow 0$)

No.5148   Minimum cutting amount of G76 (g)

         $0 \sim 99999999$ (Input unit)/(Outside the range $\rightarrow 0$)

No.5149   Finish part of G76 (e)

         $0 \sim 99999999$ (Input unit)/(Outside the range $\rightarrow 0$)

No.5129   Number of repetition of the finish cycle of G76

         $1 \sim 255$ times/(Outside the range $\rightarrow 1$)

15.9 Alarms Relevant to Combined Type Fixes Cycle

No.175   Complex fixed cycle error

On the alarm screen, more detailed messages are displayed.

   [175]   (#090)   Complex fixed cycle error
    [1]   [2] [3]   [4]

[1] Alarm number

[2] G-code(G70 at 0, G71 at 1, G72 at 2 ......, G76 at 6)

[3] Alarm type

[4] Message

Contents of alarm are described below

No.	Contents	Cause and Action
#?01	The number of blocks of finish shapes overflowed.	Reduce the number of blocks. Check the PQ command value.
#?02	There is no finish shape.	and the N number. Reduce the number of blocks
#?0	The number of valley shapes over flowed	Draw the shape.
#?05	The number of mountain shapes overflowed	Check the decimal point. Check G01 after G02 and G03.
#?10	G71/G72 type 1 shape error	Check if there is an error in the movement direction of XZ. Check the PQ command value and the N number.
#?11	G71/G72 type 1 D command error	D0 command
#?20	G71/G72 cutting direction error	X (G71) and Z (G72) do not move on the first block. Simultaneous XU: Simultaneous ZW: ZW absent (G74)
#?40	G71/G75 command XZUW error	Simultaneous XU: XU absent Simultaneous ZW (G75)
#?43	074/G75 cutting amount error	Z (G74) and X (G75) do not move.
#?44	Command 1K error	KO: IO (G74) with X movement IO: KO (G75) with Z movement
#?60	Command DH of G76 error	$D \leq 0$: $H \leq 0$: DH absent
#?62	Command A error	Outside the range A (Check the decimal point)
#?63	Command EF error	E0: EF absent
#?64	Command XZUW of G76 error	Movement amount of X or Z is 0: Movement amount of $X < Z$
#?65	Command K of G76 error	$K \leq 0$: K< Finish part
#?70	Error in those other than the next block direct line	Other than the next block G01 of the chamfered and corner R
#?71	Chamfered and corner R command error	Check the direction of the command IJR.
#?72	The chamfered and corner R is too large.	Check the shape. : Reduce the size of IKR.
#?73	Error in the block following the chamfered and corner R	Check the direction of the command IJR.
#?74	Error in the supplement of the straight line which specifies the angle.	XY abset

No.	Contents	Cause and Action
*#?75	A command next block error	Check the related parameters.
*#?76	A command next block calculation error	Check the related parameters.
#?77	Chamfered of the arbitrary angle is too large	Check the shape. : Reduce the size of ,C.
#?78	Error in the command R of the chamfered and corner of the arbitrary angle	Other than ,C,R, block G01.:,C,R simultaneous.
#?79	The corner R of the arbitrary angle is too large	Check the shape. : Reduce the size of ,R.
#?7A	Error in the specification of the arc radius R	Enlarge R. (Decimal point is not present.) Check the shape. (Terminal location is mis-specified.)
#?90	N number (P) at the top of the finish shape is not found	Check PQ command value and the N number.
#?91	N number (q) at the last of the finish shape is not found	
#?92	Q is found prior to P of the N number.	Specify P.
#?93	The command P is not present.	Specify Q.
#?94	The command Q is not present.	Change the value of PQ command
#?95	command P = command Q	Check the command G.
#?96	The command G which is not permitted in the finish shape is present	Remove the command A.
#?97	The axis A is used in the finish shape	Remove the macro statement.
#?98	Macro statement is used in the finish shape	Draw the shape. Check the decimal point.
#?A0	G71/G72 type 2 shape error	Check G01 after G02 and G03
#?A1	G71/G72 type 2 cutting down error	Error in the movement direction of Z (G71) and X (G72)
#?A2	G71/G72 type 2 cutting error	
#?A3	G71/G72 type 2 cutting up error	Check the PQ command value and the N number.
#?A4	G71/G72 type 2 projection shape error	Draw the shape. Error may be prevented by slightly modifying the amount
#?A5	G71/G72 type 2 D command error	of the cutting (D). There is a hiqher than the starting point.
% #?C0	Shape table creation error	D0 command
% #?C1	Edge R calculation error	Slightly modify the shape.
% #?C2	G71/G72 type 1 cutting error	Slightly modify the value of the command D.
% #?C4	G73 start error	Slightly modify the value of the command D.
% #?C5	G73 cutting error	Slightly modify the value of the command DK.
% #?C6	Division by 0 of G76	Slightly modify the length of screw and the value of the command I.
% #?C7	Infinite looping	Slightly modify the shape.
#?0T	T-command error	Excludes a T-command (cannot be specified)

- Alarm having No. with "*" is not generated under the normal setting.
  If this alarm is generated, check the setting values of the related parameters.
- Alarm having No. with "%" is generated as a result of the internal calculation of NC, and will not be generated on the normal ccnditions. If this alarm is generated, inform us of the alarm number, message, and the programs or related parameters in which the alarm is generated.
- One of the numbers, 0 to 6, replaces "?" in the No. column.

16.1 Canned Cycle for Drilling (G80-G89, G831, G841, G861)

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

When drilling the same hole repeatedly, you only have to specify a hole position; it is very effective for simplification of the program.

16.1.1 G Codes

G80	:	Canned cycle for drilling cancel
G81	:	Drilling cycle, spot drilling cycle
G82	:	Drilling cycle, counter boring cycle
G83	:	Peck drilling cycle
G84	:	Tapping cycle
G85	:	Boring cycle
G86	:	Boring cycle
G87	:	Back boring cycle
G88	:	Boring cycle
G89	:	Boring cycle
G831	:	High-speed peck drilling cycle
G841	:	Counter tapping
G861	:	Fine boring cycle

16.1.2 Command Format

$\begin{bmatrix} G198 \\ G199 \end{bmatrix}$ G___ X___ Y___ Z___ R___, R___ Q___ P___ L___ F___ E___ ;

G198/G199	:	Return point
C___	:	C code for the canned cycle (G81-G89, G831, G841, G861)
X___/Y___	:	Drilling position <Note 3>
Z___	:	Position of the point Z R ___ : Position of the point R
D___	:	Location of comma D (This can be specified in the address ",R" depending on the setting of parameter.)
Q___	:	G831/G83 depth of cut or G861/G87 shift amount
P___	:	Dwell time
L___	:	No. of repeat times
F___	:	Cutting feed rate
E___	:	Cutting feed rate

(Note 1) *When the number of repeat times(L) is omitted, it is assumed to be L = 1. If L = 0 is specified, the tool moves to a drilling position and does not perform drilling.*

(Note 2) *I, J, and K may be used for G83, G87, G831, and G861.*

(Note 3) *The Z point position is specified with the address of the drilling axis.*

A drilling position is specified with the axial address of other than the drilling axis.

(Note 4) The R point, Z point, P, Q, I, J, and K are modal in the canned cycle mode.

16.1.3 Machining Cycle

The canned cycle generally consists of the following movements [1] through [7].

```
         ①
    o------>o  o  ——— Initial Point
        ¦  ¦^
   ②    ¦  ¦ ⑦
        v  ¦
        o  o  ——— R Point
   ③    ¦  ^
        v  ¦
        o  ¦     ——— D Point
           ¦  ⑥
   ④       ¦
        v  ¦
        o  o  ——— Z Point
         ⑤
where;  --> Rapid traverse
         →  Cutting feed
```
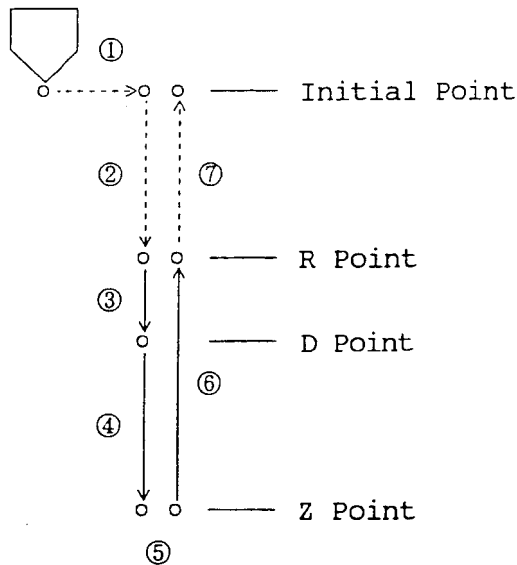
[1] Positioning to the drilling position
[2] Rapid traverse to the R point
[3] Cutting feed to the, D point (feed rate E)
[4] Drilling to the Z point (feed rate F)
[5] Movement at the Z point
[6] Return to the R point
[7] Rapid traverse to the initial point

(Note 1) In the movements [4] through [6] , single block operation does not stop the machine.

(Note 2) The D-point can be specified only in the following cycles.
* *G81, G82 (Drilling cycle)*
* *G83, G831 (Peck drilling cycle)*
When a D-command is omitted, the movement [4] (speed F) is executed from the R-point, omitting the movement [3] .
When an E-command is omitted, the movements at the feed rate E is replaced by the feed rate F.

(Note 3) An E-command in the tapping cycle (G84, G841) assumes the feed rate of the movement [6] . When the E-command is omitted, the movement [6] also moves at the feed rate F.

16.1.4 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.

16 - 2

16.1.5   R point, Z point, and D point

Altough the R and Z points can be specified by either absolute or incremental programming, the D point is always specified by incremental programming.
When the G code system A is used, the R point is always specified by absolute programming.
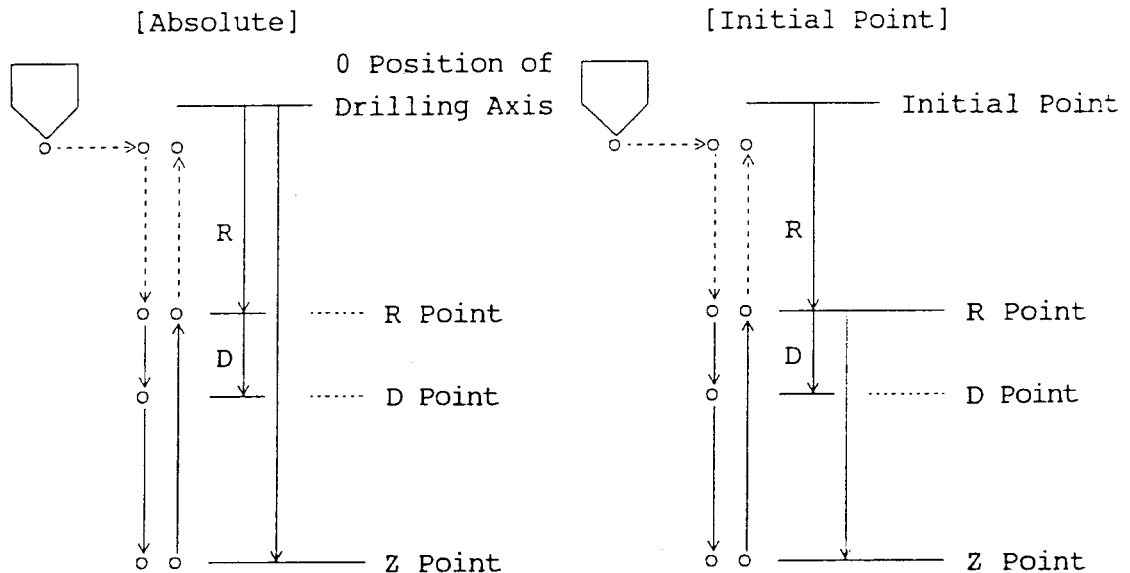
(Note 1)   The D point is an incremental position from the R point.
(Note 2)   For the address specifying the D point, either "R" or "D" can be selected by parameters. If "D" is selected, the incremental command for B axis by "D" during the fixed cycle Cannot be issued.

16.1.6   Drilling Axis, and Positioning Axis to the Drilling Position

The drilling axis is determined by G17-G19.
   G17 : Z axis
   G18 : Y axis
   G19 : X axis
The specified axis other than the drilling axis will be the positioning axis to the drilling position.

(Note 1)   When switching G17-G19, cancel the canned cycle.
(Note 2)   In machines that have no Y axis, the method for selecting G17 (X-Y plane) and C19 (Y-Z) plane is explained below.
          Plane selection is performed by setting parameter No.1024 (which axis of the basic coordinate system corresponds to each axis).
          Therefore, the axis (axis not included in the control axis number) not used in this parameter is set in the virtual Y axis.
          For instance, if the machine has 2 axes, X and Y, set the third item of parameter No. 1024 as "2." (Y axis of basic coordinate system)
          If settings are made this way, drilling fixed cycle can be executed in the G17, G19 planes.

16.1.7 Description of the Movements in the Canned Cycle

The folloing description of the movements in the canned cycle assumes the positioning axes to the drilling position to be the X and Y axes, and the drilling axis to be the Z axis.

(1) G81 (Drilling)

$$\begin{bmatrix} G198 \\ G199 \end{bmatrix}$$ G81 X___ Y___ Z___ R___ D___ L___ F___ E___ ;

[G198]   [G199]

```
(X,Y)                                   (X,Y)
····>o  o  ──── Initial Point           ····>o  o  ──── Initial Point
     ↓  ^                                    ↓
     o  |      ──── R Point                  o  o   ──── R Point
     |  |                                    |  ^
     ↓  |      ──── D Point                  ↓  |   ──── D Point
     |  |                                    |  |
     o  o      ──── Z Point                  o  o   ──── Z Point
```

(2) G82 (Drilling)

$$\begin{bmatrix} G198 \\ G199 \end{bmatrix}$$ G82 X___ Y___ Z___ R___ D___ P___ L___ F___ E___ ;

[G198]   [G199]

```
(X,Y)                                   (X,Y)
····>o  o  ──── Initial Point           ····>o  o  ──── Initial Point
     ↓  ^                                    ↓
     o  |      ──── R Point                  o  o   ──── R Point
     |  |                                    |  ^
     ↓  |      ──── D Point                  ↓  |   ──── D Point
     |  |                                    |  |
     o  o      ──── Z Point                  o  o   ──── Z Point
     (P)                                     (P)
```

(P): Dwell
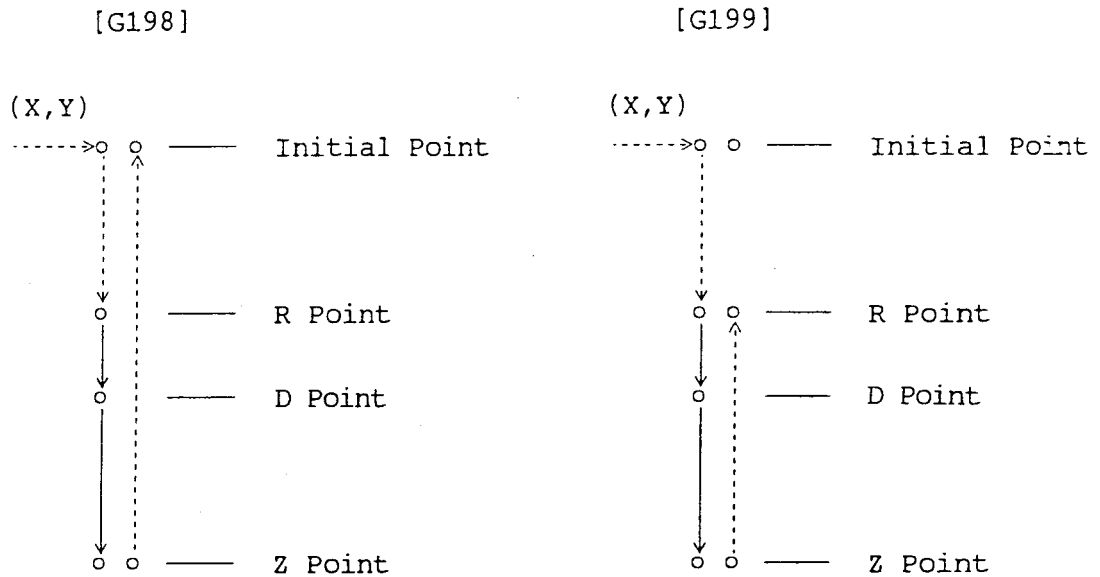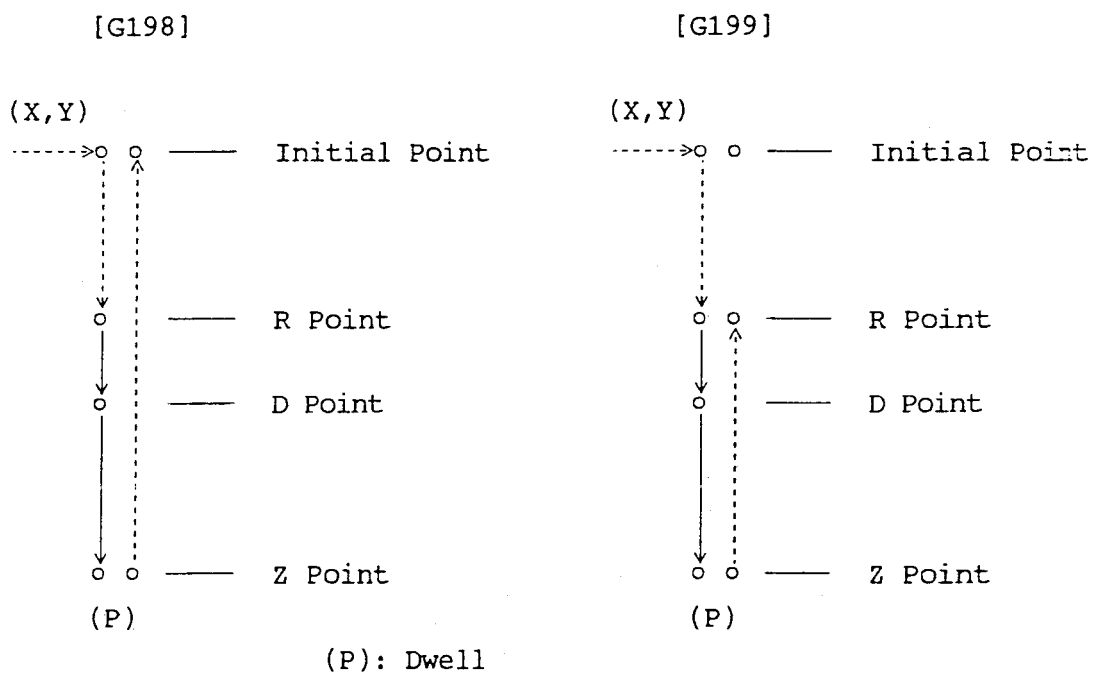
(3) G83 (Peck drilling)

$\begin{bmatrix} \text{G198} \\ \text{G199} \end{bmatrix}$ G83 X___ Y___ Z___ R___ D___ p___ Q___ L___ F___ E___ ;

[G198]

(X,Y)

·----->○ ————————⌀— Initial Point

○ ⌀—————— R Point

○ —————— D Point

Q | pr — D Point

Q | pr

Q | pr

⌀ —— Z Point

[G199]

(X,Y)

·----->○ ———————— Initial Point

○ ———————— R Point

○ —————— D Point

Q | pr

Q | pr

Q | pr

⌀ —— Z Point
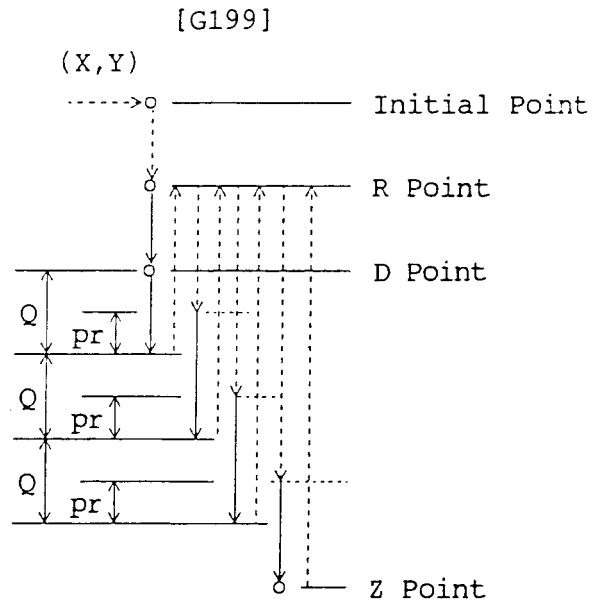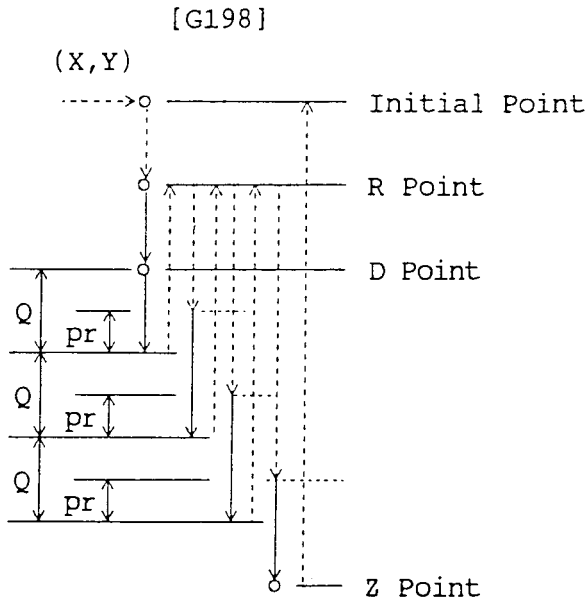
$\begin{bmatrix} \text{G198} \\ \text{G199} \end{bmatrix}$ G83 X__ Y__ Z__ R__ D__ P__ I__ J__ K__ L__ F__ E__ ;
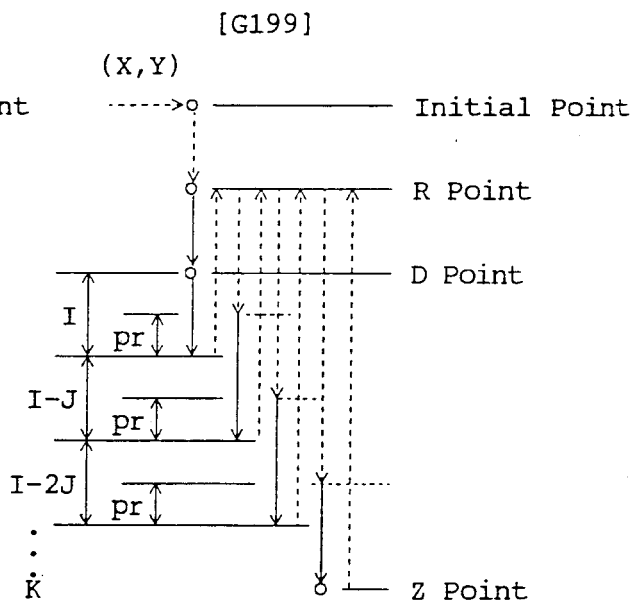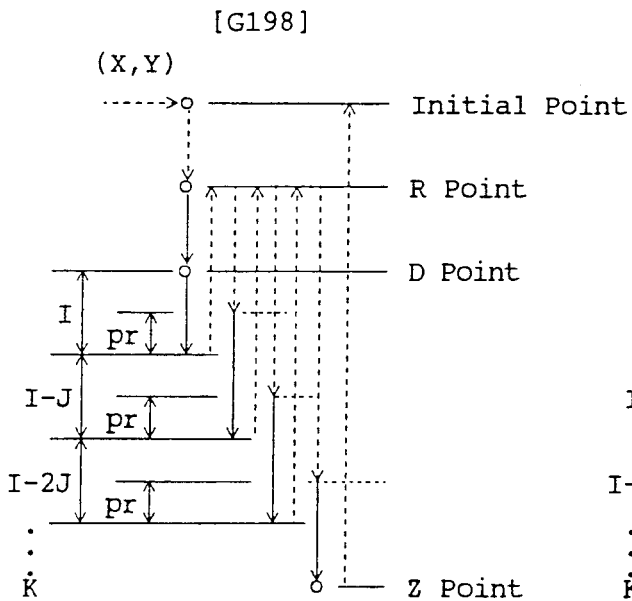
[G198]

(X,Y)

·----->○ ———————— Initial Point

○ ———————— R Point

○ —————— D Point

I | pr

I-J | pr

I-2J | pr

.
.
K ⌀ —— Z Point

[G199]

(X,Y)

·----->○ ———————— Initial Point

○ ———————— R Point

○ —————— D Point

I | pr

I-J | pr

I-2J | pr

.
.
K ⌀ —— Z Point

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) *Specifying parameters makes dwell possible at the bottom of a hole by a P command.*

(4) G84 (Tapping)

$\begin{bmatrix} \text{G198} \\ \text{G199} \end{bmatrix}$ G84 X___ Y___ Z___ R___ P___ L___ F___ E___ ;

```
        [G198]                                    [G199]
  (X,Y)                                    (X,Y)
------->o  o ——————— Initial Point    ------->o  o ——————— Initial Point
       ^                                      ^
       :  :                                   :  :
       :  :  (Ppr)                            :  :  (Ppr)
       :  :  (FWD)                            :  :  (FWD)
       v  :                                   v  :
       o  o ——————— R Point                   o  o ——————— R Point
       :  ^                                    :  ^
       :  :                                    :  :
       :  :                                    :  :
       v  :                                    v  :
       o  o ——————— Z Point                    o  o ——————— Z Point
  (Ppr)                                    (Ppr)
  (REV)                                    (REV)
```

(FWD): Tool forward
(REV): Tool reverse
(Ppr): Dwell(parameter setting)
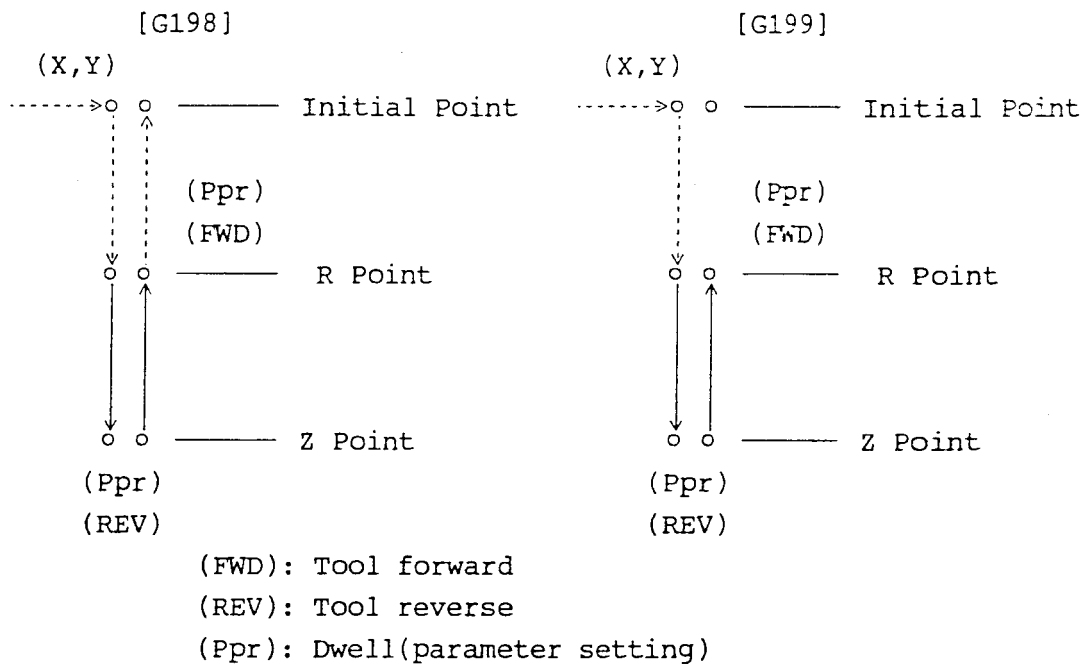
(Note 1) Specifying parameters makes dwell possible by a P command.
(Note 2) Feed hold and override are disabled during cutting.

(5) G85 (Boring)
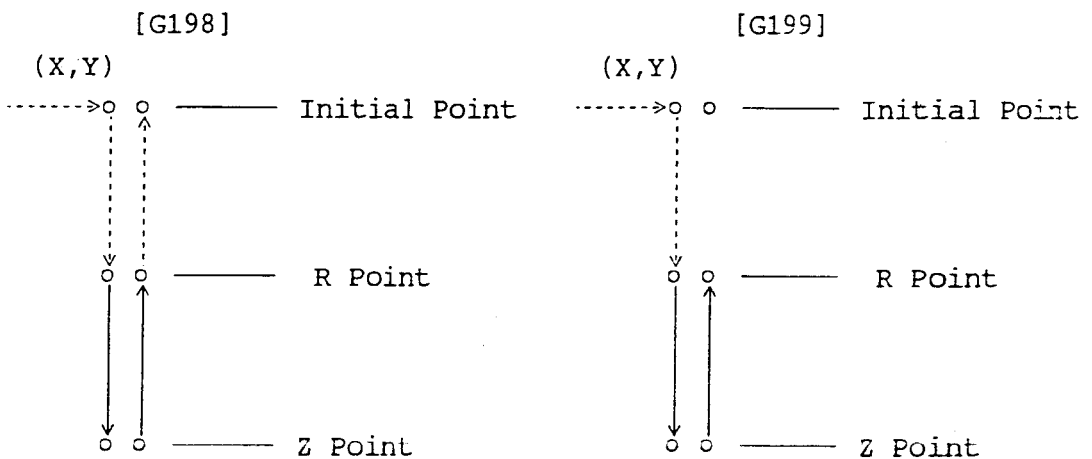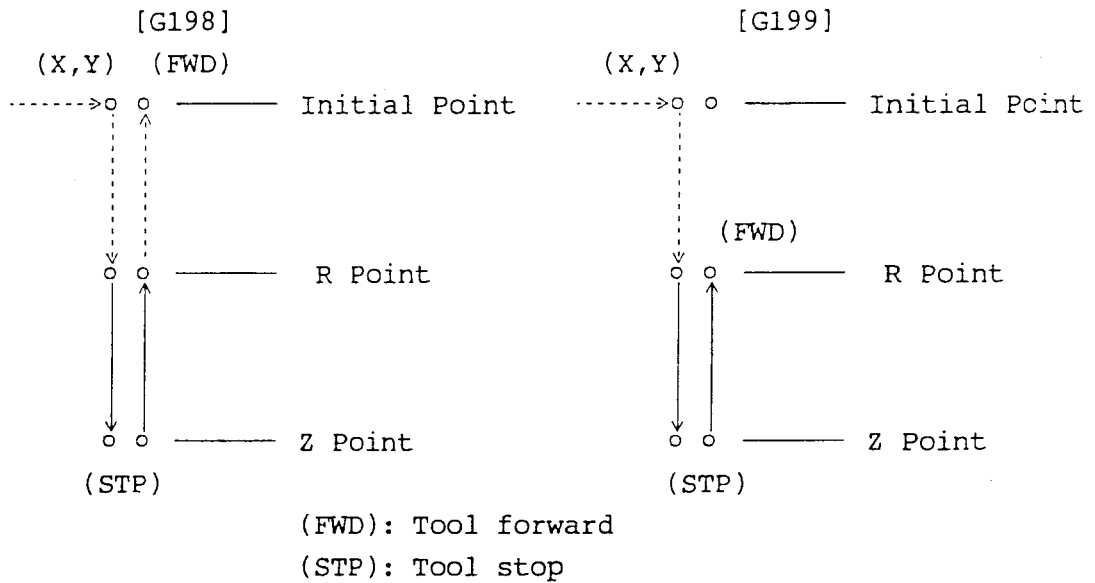
$\begin{bmatrix} \text{G198} \\ \text{G199} \end{bmatrix}$ G85 X___ Y___ Z___ R___ L___ F___ ;

```
        [G198]                                    [G199]
  (X,Y)                                    (X,Y)
------->o  o ——————— Initial Point    ------->o  o ——————— Initial Point
       ^                                      ^
       :  :                                   :
       :  :                                   :
       v  :                                   v  :
       o  o ——————— R Point                   o  o ——————— R Point
       :  ^                                    :  ^
       :  :                                    :  :
       v  :                                    v  :
       o  o ——————— Z Point                    o  o ——————— Z Point
```

(6) G86 (Boring)

$\left[\begin{array}{c} \text{G198} \\ \text{G199} \end{array}\right]$ G86 X___ Y___ Z___ R___ L___ F___ ;

[G198]   [G199]

(X,Y) (FWD)   (X,Y)
------>o o ——— Initial Point   ------>o o ——— Initial Point

                                              (FWD)

      o o ——— R Point   o o ——— R Point

      o o ——— Z Point   o o ——— Z Point
    (STP)   (STP)

                (FWD): Tool forward
                (STP): Tool stop

(7) G87 (Back boring)

$\left[\begin{array}{c} \text{G198} \\ \text{G199} \end{array}\right]$ G87 X___ Y___ Z___ R___ P___ Q___ L___ F___ ;

[G198]   [G199]

(X,Y)
------>o ⇨ o ——— Initial Point
         ⇦

                                    ┌ ─ ─ ─ ─ ─ ─ ─ ┐
                                    :   :
                                    :   Not Used   :
                                    :   :
                                    └ ─ ─ ─ ─ ─ ─ ─ ┘

      o ⇨ o ——— Z Point

                                    (P): Dwell
      o ⇦ o ——— R Point   (FWD): Tool forward
    |————————| (ORT): Tool index stop
    | Q | ⇨ : Shift (rapid traverse)

(Note) *By parameter setting, the shift amount can be specified with I, J, and K*
instead of Q.
G17 command : I, J (Xp-Yp plane)
G18 command : K, I (Zp-Xp plane)
G19 command : J, K (Yp-zp plane)

16 - 7

(8) G88 (Boring)

$\begin{bmatrix} G198 \\ G199 \end{bmatrix}$ G88 X___ Y___ Z___ R___ P___ L___ F___ ;

[G198]   [G199]
(X,Y) (FWD)   (X,Y) (FWD)
------>o o ———— Initial Point   ------>o o ———— Initial Point

        o ———— R Point   o o ———— R Point

        o o ———— Z Point   o o ———— Z Point
      (P)   (P)
    (STP)   (P): Dwell   (STP)
                      (FWD): Tool forward
                      (STP): Tool stop
                      ww-> : Jog feed
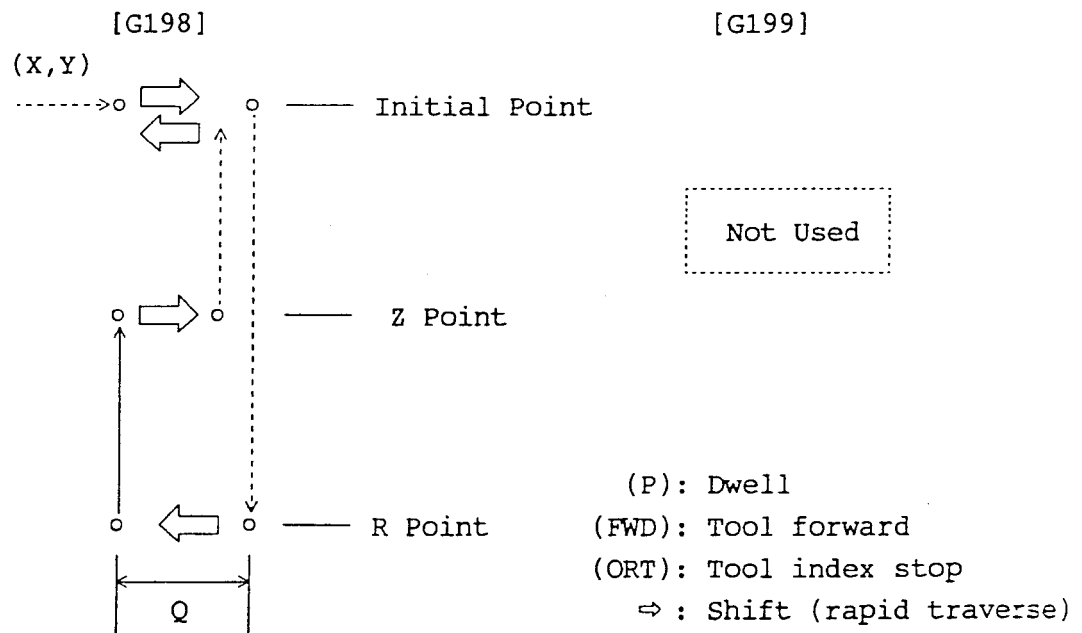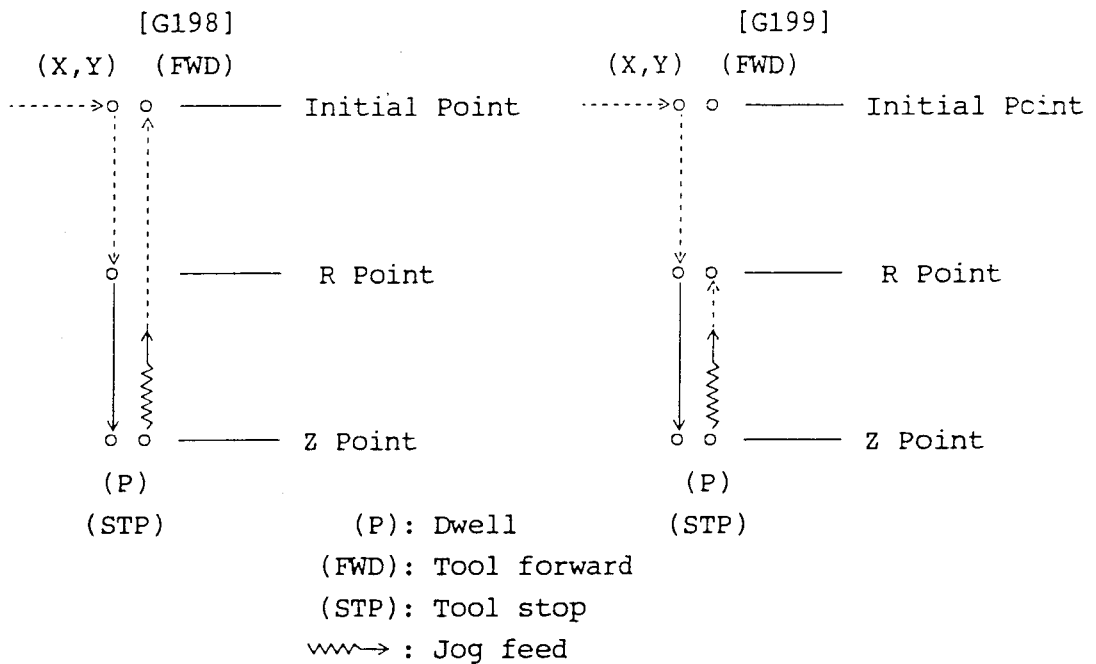
(Note) *If the Z point is reached and the tool stops rotating after swell, the machine is automatically placed in the single block stop state. You can select the manual mode and perform jog feed.*
Automatic operation is restarted by selecting the automatic mode and pressing the CYCLE START button.

(9) G89 (Boring)
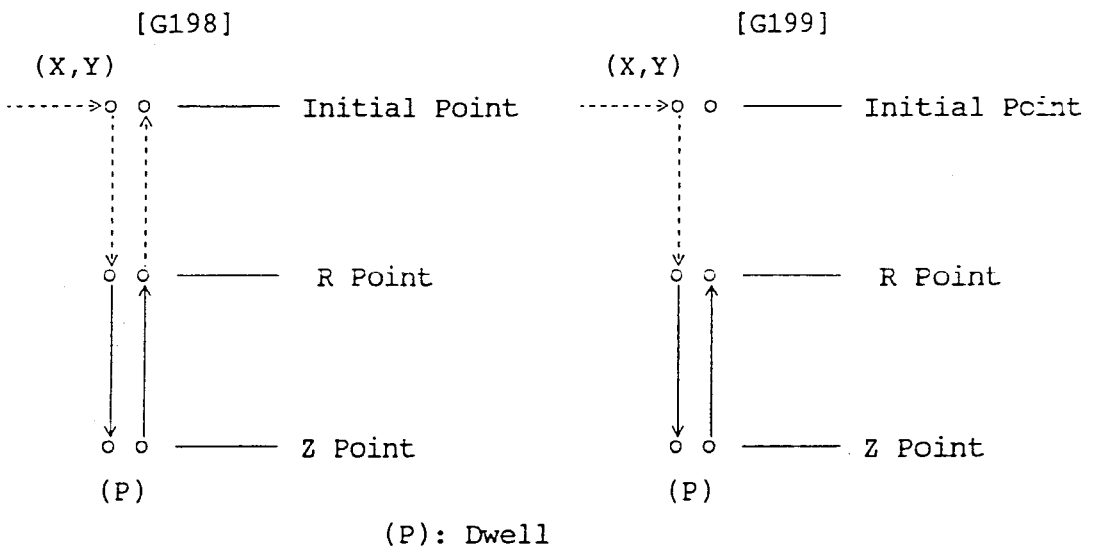
$\begin{bmatrix} G198 \\ G199 \end{bmatrix}$ G89 X___ Y___ Z___ R___ P___ L___ F___ ;

[G198]   [G199]
(X,Y)   (X,Y)
------>o o ———— Initial Point   ------>o o ———— Initial Point

        o o ———— R Point   o o ———— R Point

        o o ———— Z Point   o o ———— Z Point
      (P)   (P)
                    (P): Dwell

16 - 8

(10) G831 (High-speed peck drilling)

$\left[\begin{array}{c} \text{G198} \\ \text{G199} \end{array}\right]$ G831 X___ Y___ Z___ R___ D___ Q___ L___ F___ E___ P___ ;

[G198]   [G199]

(X,Y)   (X,Y)

------>o ————————— Initial Point   ------>o ————————— Initial Point

        o ————————— R Point   o ————————— R Point

        o ————————— D Point   o ————————— D Point

Q | pr   Q | pr

Q | pr   Q | pr

Q | pr   Q | pr

                o ——— Z Point   o ——— Z Point

$\left[\begin{array}{c} \text{G198} \\ \text{G199} \end{array}\right]$ G831 X___ Y___ Z___ R___ D___ Q___ L___ F___ E___ P___ ;

[G198]   [G199]

(X,Y)   (X,Y)

------>o ————————— Initial Point   ------>o ————————— Initial Point

        o ————————— R Point   o ————————— R Point

        o ————————— D Point   o ————————— D Point

I | pr   I | pr

I-J | pr   I-J | pr

I-2J | pr   I-2J | pr

K   o ——— Z Point   K   o ——— Z Point
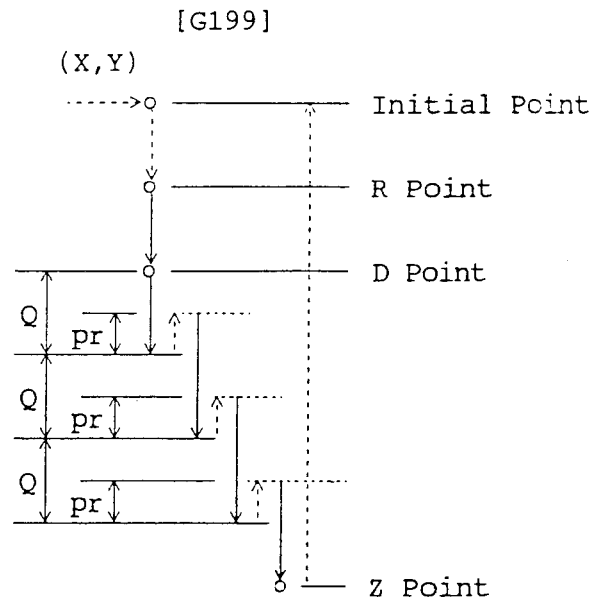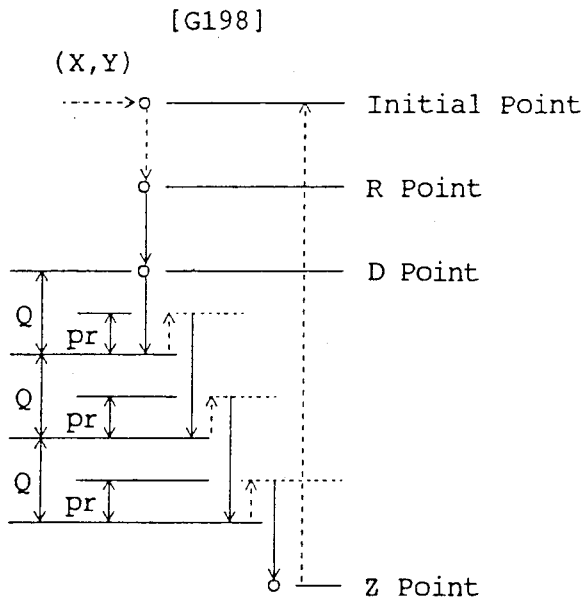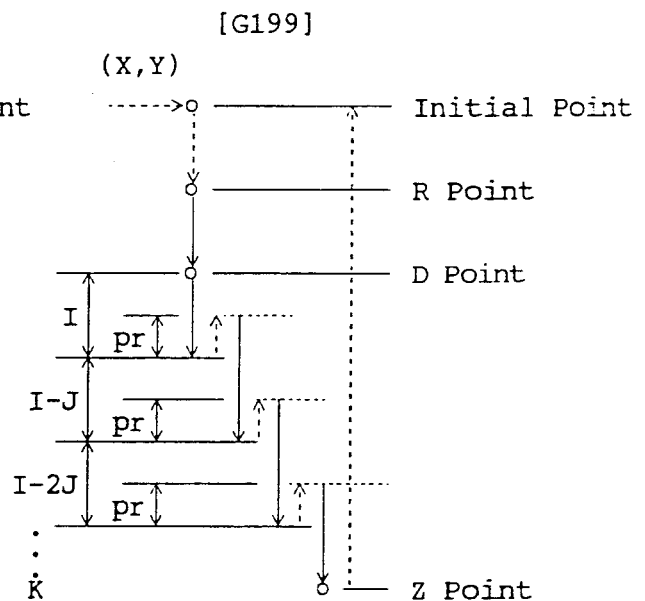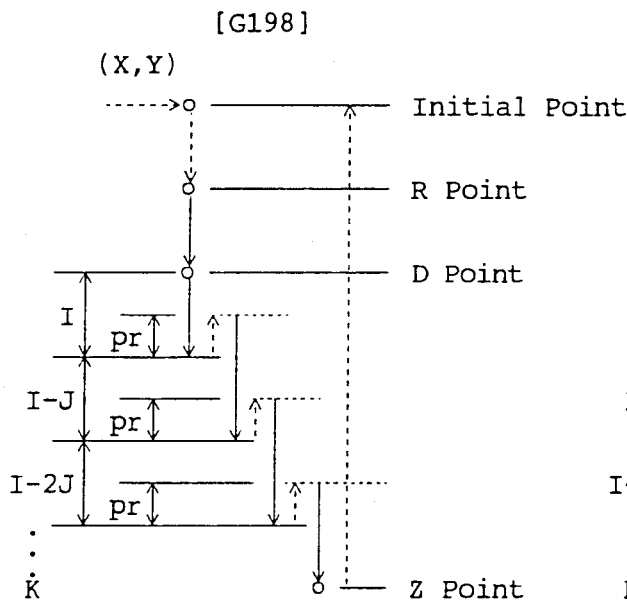
where; I : Initial value of the depth of cut (positive value)
        J : Decremental value in 2nd cut onward (positive value)
        K : Final value of the depth of cut (positive value)

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

(Note 2) *Specifying parameters makes dwell possible at the batten of a hole by a P command.*

16 - 9

(Note 3) *If " , C" is given, the tool will be withdrawn halfway drilling.*

G831 X___ Y___ Z___ R___ D___ P___ Q___ L___ F___

E___ ,C___ ;

The following figure shows an example of movement for initial point return.
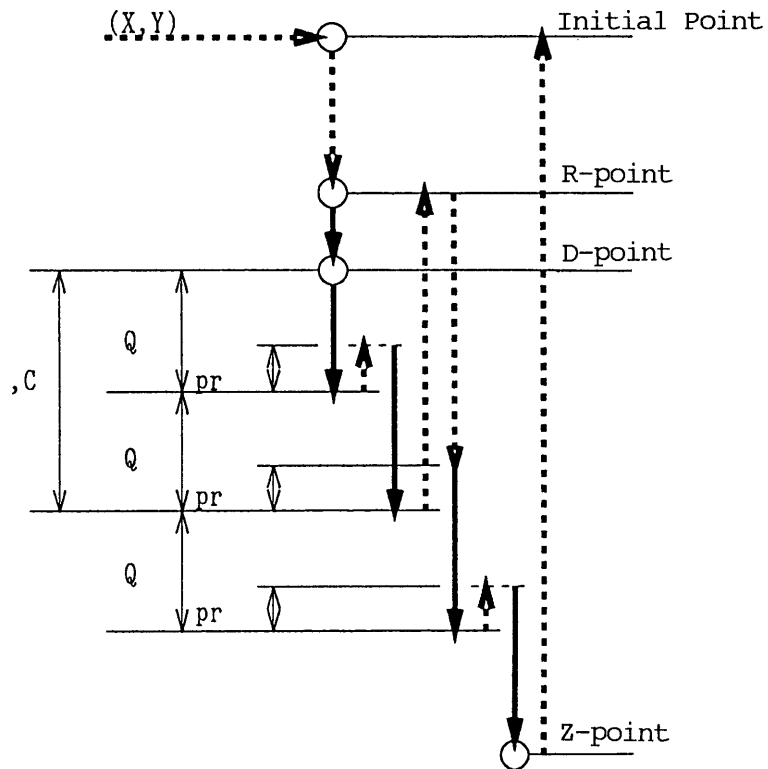
" , C" denotes an incremental value from the R-point to the Z-point.

A sign is invalid.

During the single block mode, the tool stops at the R-point.

The tool is withdrawn at every " , C" value. If " , C50." is given, for example, a withdrawing position will be 50., 100., 150., and so on.

When passing through the withdrawing position during drilling operation (move by the Q or I/J/K value), the withdrawing position will be the end point of that movement.

The address " , C" is valid only when it is specified in the same block as G73.

(11) G841 (Counter tapping)

$\begin{bmatrix} \text{G198} \\ \text{G199} \end{bmatrix}$ G841 X___ Y___ Z___ R___ P___ L___ F___ E___ ;

(Note 1) *Specifying parameters makes dwell possible by a P comniand.*

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

```
        [G198]                              [G199]

  (X,Y)                              (X,Y)
------>o  o ——————— Initial Point   ------>o ——————— Initial Point
        ^
        |  (Ppr)                            |  (Ppr)
        |  (REV)                            |  (REV)
  o  o ——————— R Point               o  o ——————— R Point
        ^                                   ^
        |                                   |
  o  o ——————— Z Point               o  o ——————— Z Point
  (Ppr)                              (Ppr)
  (FWD)   (FWD): Tool forward        (FWD)
          (REV): Tool reverse
          (Ppr): Dwell(parameter setting)
```

(12) G861 (Fine boring)
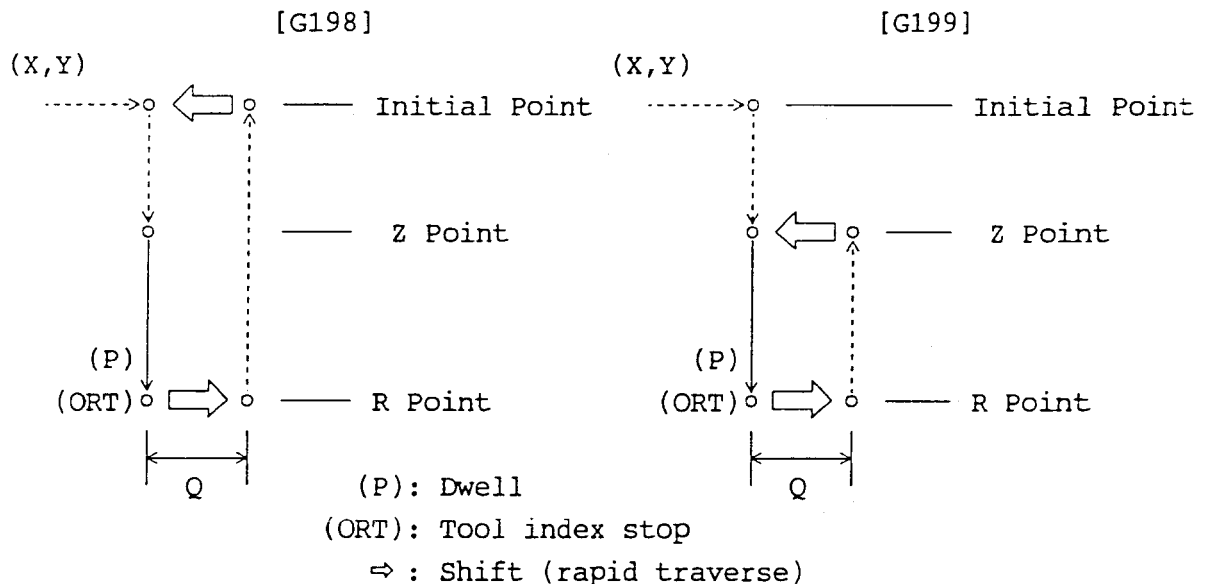
$\begin{bmatrix} \text{G198} \\ \text{G199} \end{bmatrix}$ G861 X___ Y___ Z___ R___ P___ Q___ L___ F___ ;

```
           [G198]                                [G199]

(X,Y)                                 (X,Y)
----->o <== o ——— Initial Point       ------>o ——————————— Initial Point
       ^
       o ——— Z Point                         o <== o ——— Z Point
       |                                      |
 (P)   |                               (P)    |
(ORT) o ==> o ——— R Point             (ORT) o ==> o ——— R Point
      |<——Q——>|                             |<——Q——>|

              (P): Dwell
              (ORT): Tool index stop
              ⇨ : Shift (rapid traverse)
```

(Note) *By parameter setting, the shift amount can be specified with I, J, and K instead of Q.*

G17 command : I, J (Xp-Yp plane)

G18 command : K, I (Zp-Xp plane)

G19 command : J, K (Yp-zp plane)

16 - 11

16.1.8 Sample Program

```
G17  G00  X0  Y0 ;
Z50. ;
G97  S1000  M113 ;          Rotary tool forward
G81  Z-30.  R20. ,R-18.     Machining data setting for the canned cycle
      F500  E3000  L0 ;
G199  X20.  Y10. ;          Drilling cycle in the position [1]
X40. ;                      Drilling cycle in the position [2]
Y20. ;                      Drilling cycle in the position [3]
G198  X20. ;                Drilling cycle in the position [4]
G80 ;                       Canned cycle cancellation
```
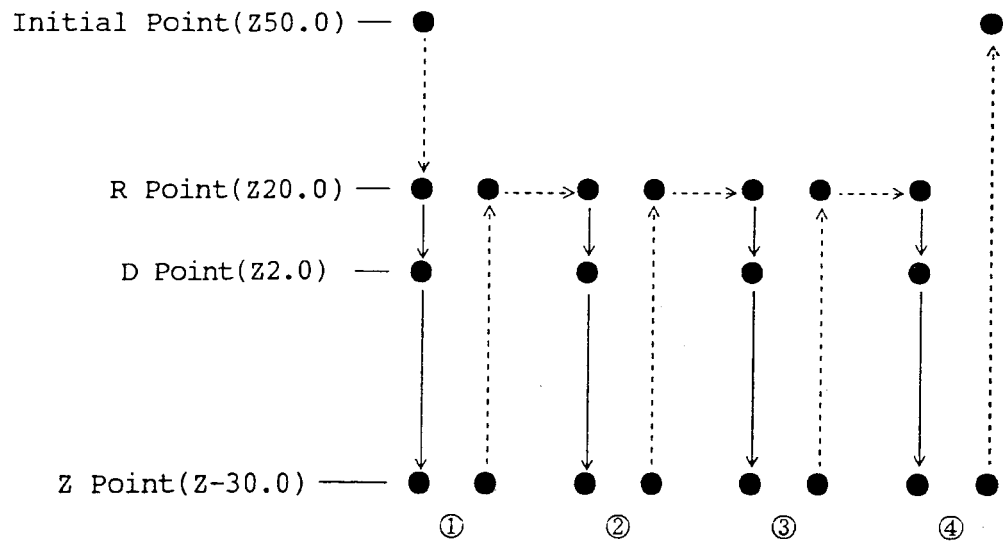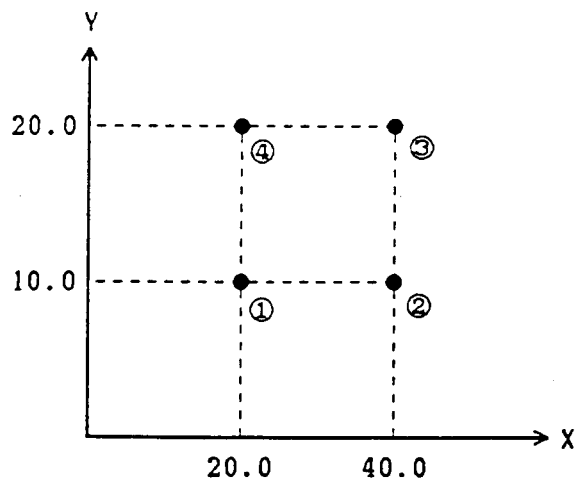
16.1.9 Cautions

(1) When the SINGLE BLOCK button is turned on, the tool stops at the end point of the movements [1] , [2] , [3] , and [7] . In this case, the FEED HOLD lamp is turned on at the end point of the movements [1] , [2] , and [3] , and the movement [7] when the number of repeat times is left incomplete.
A single block stop is enabled in the movements [4] through [7] of the cycle other than tapping (G84, G841).

(2) When the FEED HOLD button is pressed in the movements [4] through [6] of G84/G841, the FEED HOLD lamp is turned on immediately, and machine operation stops after continuing to [6] .

(3) The feed rate override for the movements [4] through [6] of G84/G841 is fixed at 100 %. Dry run can be enabled/disabled by parameter setting.

(4) specify Q, P, I, J, and K in the block where R or the axis has been specified.
In the other blocks, they are not treated as the data of the canned cycle.
Also, P, Q, I, J, and K are not treated as the data of the canned cycle in the block where the G code of Group 00 (excepz G09) as well.

(5) The canned cycle is cancelled by specifying G80 or the G code of Group 01 (G00, G01, ...).

(6) If M or S is specified in the block where the canned cycle has been specified, itis sent out at the time of the first movezent [1] (positioning to the drilling position).
When the number of repeat times (L) has been specified, M and S are sent out in the first time only.

(7) The numerical values of P, Q, L, F, E, etc. should be given with positive values.

(8) Q, I, J, and K should be always specified in terms of radius.
R/D should be specified in conformity with the designation of the drilling axis.
Drilling axis designated by radius .... Radius desighation for R/D

Drilling axis designated by diameter .... Diameter designation for R/D

(9) "R" can be selected instead of "D" for the address that specifies the "D" point by parameter setting. However, if "D" is selected, incremental command of B axis cannot be used during the drilling fixed cycle.

(10) With a spindle indexing M code being assigned in a parameter, drilling can be performed through M code command instead of shaft command.
Use this to perform drilling on every spindle indexing with a machine provided with spindle indexing function instead of C axis.

    [EX] N1 G81 R Z F L0 ;
           N2 Mxxx ;
           N3 M△△△ ;
           N4 G80 ;

When Mxxx is a spindle indexing M code and M△△△ is not; drilling is performed after Mxxx is output in N2 block, and no drilling is performed after M△△△ is output in N3 block.

16.1.10 Associated Parameters

No.1401, #5 = 0   Dry run is invalid in thread cutting/tapping command.
          = 1   Dry run is valid in thread cutting/tapping command.

No.3407, #6 = 0   Single block operation does not stop during drilling (movements [4] through [6]).
          = 1   Single block operation stops during drilling (movements [4] through [6]). (except for the tapping cycle)

No.5100, #2 = 0   The G87/G861 shift direction is specified with a parameter, and shift amount with Q.
          = 1   Both shift direction and shift amount are specified with I, J, and K.

No.5100, #3 = 0   Dose not output a rotation stop when changing the tool rotating direction.
          = 1   Outputs a rotation stop when changing the tool rotating direction.

No.5100, #4 = 0   Spindle indexing M code is not used.
          = 1   Spindle indexing N code is used. (No.5135/5136 are set.)

No.5100, #5 = 0   In the 084/G831 mode, addition of dwell by a P command is invalid.
          = 1   In the G84/G831 mode, addition of dwell by a P command is valid.

No.5100, #6 = 0   Disables addition of dwell by a P command in the G84/G841 mode.
          = 1   Enables addition of dwell by a P command in the G84/G841 mode.

No.5100, #7 = 0   Address that specifies the D point of the fixed cycle is ",R"
          = 1   Address that specifies the D point of the fixed cycle is "D"

No.5127   Direction to run off a tool following G87/G861 orientation of each axis.

No.5131   M code for tool forward (M13 when set to 0)

No.5132   M code for tool reverse (M14 when set to 0)

No.5133   M code for tool stop (M05 when set to 0)

No.5134   M code for tool orientation (M15 when set to 0)

No.5135   Minimum value to identify the spindle indexing M code

No.5136   Maximum value to identify the spindle indexing M code

No.5150   G831 return volume

No.5151   G83 clearance amount

16.1.11 Associated Alarms

No.133   Error in the drilling fixed cycle command.

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

(#001)   G87 (back polling) has been commanded in R point level recovery mode.

(#100)   In parameter No.5127, 0 is set.

(#101)   In parameter No.5127, an axis same as the drilling axis has been set.

16.2 Direct Tapping Cycle (G842, G843)

High-speed and high-precision tapping is performed using the same method as for the synchronization of the rotation tool and the feeding axis.
The conventional tapper is not required.

16.2.1 Command Format

$$\begin{bmatrix} G842 \\ G483 \end{bmatrix} \begin{bmatrix} G198 \\ G199 \end{bmatrix} \begin{bmatrix} G98 \\ G99 \end{bmatrix} \quad X\_\_\_ \ Y\_\_\_ \ Z\_\_\_ \ R\_\_\_ \ P\_\_\_ \ L\_\_\_ \ S\_\_\_ \ F\,(E)\_\_\_ \ ;$$

G842	:	Normal direct tap
G843	:	Inverse direct tap
G198/G199	:	Return point
X___ /Y___	:	Drilling position < Note 2 >
Z___	:	Z point location (hole bottom location) < Note 2 >
R___	:	R point location (;laning starting point)
P___	:	Dwell time
L___	:	Number of repetition
F___	:	G98 mode ...... Feed rate of drilling axis
		G99 mode ...... Tap pitch
E___	:	Number of threads per inch (effective only in the G99 mode)
S___	:	Number of rotations of rotation tool

(Note 1) *When the number of repeat operations (L) is omitted, L will be taken as l. If you specify L = 0, shift will be to the drilling position but drilling will not be executed.*

(Note 2) *Specify the position of Z point by the axis address of the drilling axis. Specify the drilling position by an axis address other than the drilling axis.*

(Note 3) *R point, Z point and P are modal during the fixed cycle.*

(Note 4) *"Return point," "R point," "Z point," "drilling axis" and positioning axis at the drilling position are the same as for "drilling fixed cycle." Therefore, please refer to section "16.1"*

16.2.2 Processing Cycle

Processing Cycle of the direct tap is composed off operations 1 to 8.

[G 8 4 2]
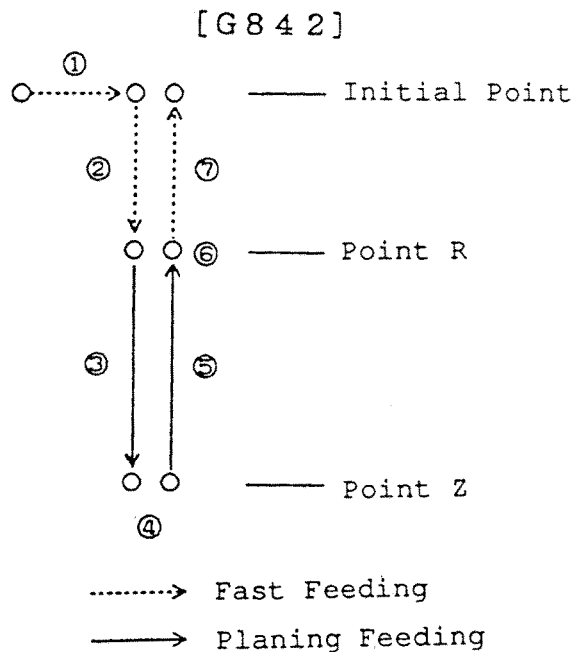
[1] Decision of the location to be holed.

[2] Feeding to the point R.

[3] Holing processing to the point Z with the tool rotated.

[4] Dwell depending on the parameter settings.

[5] Return to the point R with the tool inversely rotated, and stop of the rotation of the tool.

[6] Dwell depending on the parameter settings.

[7] Fast feeding to the initial point.

(Note) *In the case of G843, the tool is rotated inversely.*

16.2.3 Feed Rate and Pitch Specification (F Command)

The meaning of the F command varies during direct tapping for the feed per minute mode (G98) and the feed per revolution mode (G99).

Also, the E command can be used instead of the F command in the G99 mode.

• G98 mode: F indicates the feed rate of the drilling axis (mm/min, inch/min)

• G99 mode: F indicates the tap pitch (mm, inch)

E is the number of threads per inch (thread/inch)

The number of significant digits of the F command is the same as G01. The number of significant digits by parameter setting is given below.

• G98 mode: Metric input	F60:	1 ~ 999999 (mm/min)
	F61:	0.1 ~ 999999.9 (mm/min)
Inch input	F51:	0.1 ~ 99999.9 (inch/min)
	F52:	0.01 ~ 99999.99 (inch/min)
• G99 mode: Metric input	F32:	0.01 ~ 999.99 (mm/rev)
	F33:	0.001 ~ 999.999 (mm/rev)
Inch input	F23:	0.001 ~ 99.999 (inch/rev)
	F24:	0.0001 ~ 99.9999 (inch/rev)

(Note 1) *Even for the G99 mode, the motion will be in feed per minute.*

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

(Note 3) *If F and E are specified in the same block, F will become effective.*

(Note 4) *Pitch will be determined by F and S (tool rpm) in the G98 mode.*

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

16.2.4　Override on Pull-out Operation

Pull-out operation speed (point Z → point R) of direct tap can be varied to cutting speed (point R → point Z).

Override of pull-out speed to the cutting speed is set in a parameter.

With Parameter No.5200, #4 (D0V) 1, this override is made valid and shifting takes place as follows:

$$\text{Pull-out speed} = \frac{\text{cutting speed} \times \text{Parameter No. 5211}}{100}$$

16.2.5　Notes

(1) Specify direct tapping in the peripheral speed constant control canceled status (G97).

(2) In this item, holing location is the side XY, and the holing axis is the Z axis.

(3) Dwelling can be effective or ineffective depending on the parameter settings.

(4) During the tapping, the feeding override and the main axis override is 100% (ineffective).
     However, the dry run can be effective or ineffective depending on the parameter settings.

(5) When performed in a single block, it is stopped at the initial point and the point R.

(6) If "Stop" is pushed during the tapping, the feeding hold lamp becomes lit and is stopped after continued as far as the point R.

(7) Cancellation of the direct tapping specifies G80 or G01 group G code (G00, G01, G02...). However, the cancellation command and the other fixed cycle (G70, G71 etc.) must not bw specified in the same block.

(8) When performing the direct tapping, the cammand S for the rotation tool should be made in the specified block of G842 or G843 or the preceding block.

16.2.6 Ralated Parameters

No.1401, #5 = 0   Dry run is valid in thread cutting/tapping command.

         = 1   Dry run is invalid in thread cutting/tapping command.

No.5200, #0   Always set this to 1.

No.5200, #4 = 0   Override for pull-out operation of direct tap is invalid.

         = 1   Override for pull-out operation of direct tap is valid.

No.5200, #6 = 0   Feed hold/single block are invalid in direct tap.

         = 1   Feed hold/single block are valid in direct tap.

No.5210   Direct mode command M code

         (Although not commanded in a program, this M code is created internally and output in Machine side.)

No.5211   Override for direct tap pull-out operation

No.5300   Imposition width of the tapping axis of dilect tap

No.5301   Imposition width of the rotary tool of direct tool

16.2.7 Associated Alarms

No.138   An error exists in the direct tap command.

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

(#001)   S value is 0.

(#002)   F value is 0.

(#005)   Any other fixed cycle has been commanded in the same block as cancel command (G80, G00 ...).

(#006)   E value is 0.

(#010)   Direct tap has been specified in the thread cutting mode.

17. DATA SETTING

17.1 Programmable Data Input (G10)

Various data such as work shift, offset, tool life management, out-of-machine measurement, serial interface I/O port switching cn be altered on the NC program.

17.1.1 Program Input if Offset Amount

(1) Work shift amount input

    G10   P0   X (U)___ Y (V)___ Z (W)___ B (D)___ C (H)___ ;

        P0   : Work shift amount input designation
        X (U) : X-axis work shift amount
        Y (V) : Y-axis work shift amount
        Z (W) : Z-axis work shift amount
        B (D) : B-axis work shift amount
        C (H) : C-axis work shift amount

(2) Profile offset amount input

    G10   L10   P___ X (U)___ Y (V)___ Z (W)___ R___ Q___ H___ ;

        L10   : Profile offset input designation
        P   : Offset number (1~ max. compensation number)
        X (U) : X-axis profile offset amount
        Y (U) : Y-axis profile offset amount
        Z (W) : Z-axis profile offset amount
        R   : Tool nose radius (absolute)
        Q   : Virtual tool nose point (0-9)
        H   : tool width (absolute)

(3) Wear offset amount input

    G10 L11   P___ X (U)___ Y (V)___ Z (W)___ R___ H___ ;

        L11   : Wear offset amount input designation
        P   : Offset number (1~ max. compensation number)
        X (U) : X-axis wear offset amount
        Y (V) : Y-axis wear offset amount
        Z (W) : Z-axis wear offset amount
        R   : Tool nose radius (absolute)
        H   : Knife edge width (absolute)

(Note 1) *Only when absolute programming is used to input the profile offset amount, the wear offset amount of the input address is cleared to 0.*

(Note 2) *Only absolute programming is allowed R(tool nose radius) and H (knife edge width).*

17.1.2 Tool Life Management Data Input

(1) Tool setting   (omissible within [ ])

  G10 L20 P___ Q___ [A___] ;

    P : Tool setting position

      ○○ xx (Ex. 0101)

          Row (01-03)

          Line (1-99)

    Q : Tool setting data

      ○○ xx (Ex. 0101 for T0101)

          Offset number

          Turret number
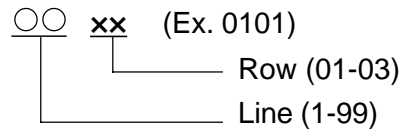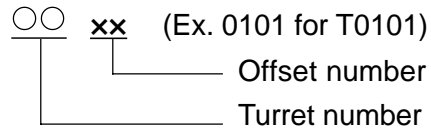
    A : Priority flag (No priority flag when omitted)

      0 : Without priority flag

      1 : With priority flag

(2) Tool life counter

  G10 L21 P___ D___ U___ [A___] ;

    P : Tool number (1-99)

    D : Set value data

    U : Use value data

    A : Time/frequency count designation (Time when omitted)

      0 : Time

      1 : Frequency

17.1.3 Input of Externally Measured Data (Data in [ ] are Omissible.)

(1) G10 L30 P___ X (Z)___ Q___ D___ ;

  P : Measured position (1-8)

  X (Z) : Axial data 0 : +X (+Z)

           1 : - X (- Z)

  Q : Measured point (1 ~ 4)

  D : Reference value

(2) G10 L31 P___ Q___ D___ ;

  P : Measured position (1- 8)

  Q : Measured data position (1-4) 1 : +NG

                  2 : +OK

                  3 : - OK

                  4 : - NG

  D : Compensation amount

(3) G10 L32 P___ Q___ D___ ;

      P : Measured position (1 ~ 8)

      Q : Repeat data position (1 ~ 6)   1 : ++NG

                                                                         2 : +NG

                                                                         3 : +OK

                                                                         4 : - OK

                                                                         5 : - NG

                                                                         6 : --NG

      D : Count set value

(4) G10 L33 P___ Q___ D___ ;

      P : Measured position (1 ~ 8)

      Q : Measured data position (1 ~ 6)   1 : ++NG

                                                                         2 : +NG

                                                                         3 : +OK

                                                                         4 : - OK

                                                                         5 : - NG

                                                                         6 : --NG

      D : Tolerance

(5) G10 L34 P___ D___ U___ ;

      P : Measured position (1 ~ 8)

      D : Measured pass after offset

      U : Measured pass after termination of tool life

(6) G10 L35 P___ Q___ D___ [A___] ; (Wear control)

      P : Tool No. (1 ~ 99)

      Q : Set value

      D : Wear amount

      A : Tool life flag (Omission: No change)   0 : Normal

                                                                                         1 : Abnormal

(7) G10 L36 P___ Q___ D___ ; (Setting the quantitative compensation amount)

      P : Measurement position (1 ~ 8)

      Q : Data position (1 ~ 3) 1 : Compensation amount 1

                                                            2 : Compensation amount 2

                                                            3 : Compensation amount 3

      D : Quantitative compensation amount

(8) G10 L37 P___ Q___ D___ ; (Group counter)

      P : Measurement position (1 ~ 8)

      Q : Dataposition (1 ~ 2) 1 : Group 1

                                                  2 : Group 2

      D : No. of work pieces

Caution

Without the optional unit, loading of data are not available in tool life control or external measurement.

17.1.4　Serial Interface I/O Port Switching

(1)　Changeover of custom macro external output device number

G10 L97 P＿＿＿ ；

P : device number (1 ~ 6)

(2)　Changeover of input device number

G10 L98 P＿＿＿ ；

P : device number (1 ~ 6)

(3)　Changeover of output device number

G10 L99 P＿＿＿ ；

P : device number (1 ~ 6)

17.1.5　Spindles

(1)　Phase shift of the subspindle in synchronous rotation (phase control ON)

G10 L100 R＿＿＿ ；

R : Phase shift angle (-359.999 to 359.999 in a step of 0.001 degree)

(2)　Setting Revolution Rate/Phase of Spindle/Rotary Tool in Polygon Turning Between Spindles

G10 L110 P＿＿＿ Q＿＿＿ R＿＿＿ ；

P : Revolution rate of spindle (1 to 999)

Q : Revolution rate of rotary tool (1 to 999/ -1 to -999)

R : Phase difference between spindle and rotary tool (1 deg. unit: 0 to 359)

(Note 1)　*For the parameter setting (No.7603, #1 (QDR)= 1) indicating "the rotating direction of a rotary tool follows that of the spindle", no minus value cannot be assigned in Q.*

(Note 2)　*For the parameter setting (No.7602, #5 (COF)= 1) indicating "no phase control", R cannot be assigned.*

As for Polygon turning, see "19.2 Polygon Turning."

17.1.6　NC Chuck

(1)　Selecting the chucking data file

G10 L200 P＿＿＿ ；

P: File number(1 to maximum number of files)

(2)　Altering the chucking data

G10 L201 P＿＿＿ Q＿＿＿ R＿＿＿ ；

P: File number(1 to maximum number of files)

Q: Data number(1 to 7)　　　1: Program number

2: Chucking force

3: Jaw opening

4: Chucking force

5: Chucking diameter irregularity

6. Mandrel diameter

7: Chucking force

R: Data

17.1.7 Associated Alarms

No.100   An error exists in G10 command.

(#001)   A command is lacking.

(#002)   An error exists in a command value.

(#003)   There is an unnecessary command.

(#071)   Neither P nor Q has been specified in the command before entering the polygon mode.

(#072)   In the polygon mode, only P or Q has been specified in the command.

(#073)   The P command value is out of the range.

(#074)   The Q command value is out of the range.

(#075)   The R command value is out of the range.

(#076)   R has been specified in disabling phase control(Parameter No. 7602, #5(COF) = 1)

(#077)   R has been specified in the polygon by the servo motor.

(#091)   P command error (No command or command value error)

(#092)   Q command error (No command or command value error)

(#093)   R command error (No command or command value error)17.2 Programmable Parameter Input

This function changes the parameters in the NC program.

17.2 Programmable Parameter Input

This function changes the parameters in the NC program.

17.2.1 Command Format

G10L50 ;   Parameter change start

N___   R___ ;

N___   R___   P___   Q___ ;

N___   R___   P___ ;

G11 ;   Parameter change end

    G10L50 : Parameter input mode ON (Independent command)

    G11   : Parameter input mode cancel (Independent command)

    N___   : Parameter number

    P___   : Axis number (1 ~ 8)

    Q___   : Bit number (0 ~ 7)

    R___   : Parameter set value (Input 0 or 1 when the bit number is specified)

17.2.2 Cautions

(1) Be sure to specify G11(parameter input cancel) at the end of parameter change.

(2) It is prohibited to write to the parameter No.9000 onward.

(3) When there is a Q command(bit number) in writing to a bit type parameter, the value of an R command should be 0 or 1.

(4) Specify G10L50 and G11 in an independent block.

17.2.3 Associated Alarms

No.100   An error exists in G10 command.

 (#011)   An error exists in a parameter No. (N).

 (#012)   An error exists in an axis No. (2).

 (#014)   An error eixsts in a parameter set value (P).

 (#015)   An unnecessary command exists.

 (#016)   Write inhibit parameter

 (#017)   An error exists in writing of a bit-type parameter.

 (#018)   Parameter table error

18. STROKE LIMIT

18.1   Stored Stroke Limit 1

A tool entry disabled area can be set for each axis.
Its boundaries are set with parameters. The outside of the set boundaries is the disabled area.
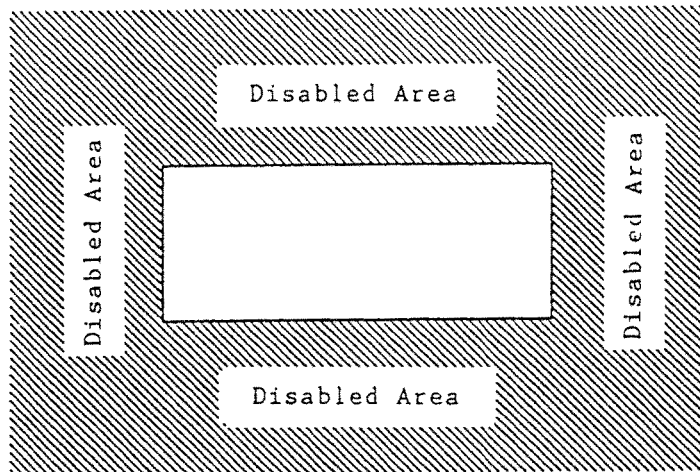
If even one axis enters the set disabled area during automatic operation, an alarm results and all the axes decelerate to a stop.
In the manual operation, only the axis in the set disabled area results in an alarm and decelerates to a stop.

18.1.1   Alarm Cancellation

When the tool enters the disabled area, get it out of the disabled area by manual operation and press the RESET button to reset an alarm.

18.1.2   Cautions

(1) Normally, stroke limit 1 has the stroke disabled area set, which is peculiar to the machine.
(2) The stroke limit boundary of each axis refers to the position in the machine coordinate system.
(3) Normally, Stroke Limit 1 is made effective immediately following supply of power. However, through parameter setting, only the axis for which reference point return has been completed can be made effective for Machine adjustment, etc.
(4) The distance required for the axis to stop after entering the disabled area depends on the conditions such as feed rate, etc.

18.1.3 Associated Parameters

No.1300, #6   Following supply of power, Stroke Limit 1 is subject to checking till reference point returning is performed.

       = 0   Performed.

       = 1   Not performed.

No.1300, #7   When a command with which the stroke limit is exceeded has been given;

       = 0   Alarm takes place after the stroke limit is exceeded.

       = 1   Alarm takes place before the stroke limit is exceed exceeded.

No.1320   Pulse directional coordinate value of the stroke limit 1 of each axis

No.1321   Minus directional coordinate value of the stroke limit 1 of each axis

18.1.4 Associated Alarms

No.F500   Overtravel : + n

       Stroke Limit 1 on + side of n axis has been exceeded.

No.F501   Overtravel : + -n

       Stroke Limit 1 on - side of n axis has been exceeded.

18.2　Stroke Limit 2 to 4 (G22, G23)

A tool entry disabled area can be set, dividing it into three ranges; stroke limit 2 through stroke limit 4. The boundaries of the stroke limits 2 through 4 are set with parameters. Either inside or outside of the set boundaries becomes a disabled area.

For the stroke limit 2, you can set with the parameter whether the inside or outside should be disabled.

For the stroke limits 3 and 4, the inside becomes the disabled area.

(a) Disabled area: Outside　　　　　　　(b) Disabled area: Inside
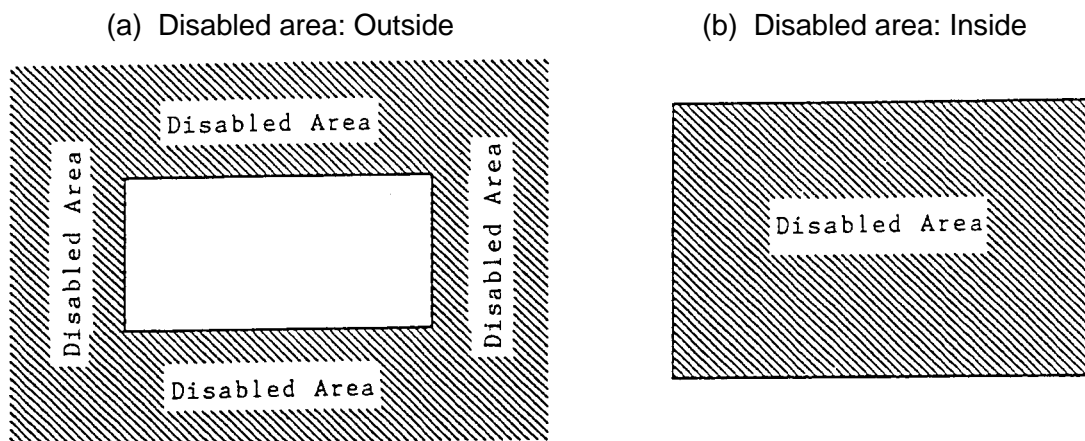
When the disabled area exists on the outer sides, alarm takes place on entering of even one axis into the area.

When the disabled area exists on the inner side, alarm takes place on entering of all axes into the area.

18.2.1　G Code

　G22 : Stroke limit 2 check ON
　G23 : Stroke limit 2 check OFF

Once either one of these G codes are specified, it remains effective until the other one is specified.

The stroke limits 3 and 4 are effective regardless of the G-code.

18.2.2　Command Format

(1)　Stroke limit check 2 enable/disable

　　G22 ;
　　　　　　　　　　　　　Stroke limit check 2 enable

　　G23 ;
　　　　　　　　　　　　　Stroke limit check 2 disable

(2) Stroke limit boundary setting

The following command can enable a stroke limit 2 check and set the boundary of the stroke limit 2 or 3. Set in the parameter as to which limit should be altered.

G22 X_ Y_ Z_ I_ J_ K_ ;

- X ： X-axis plus side boundary
- Y ： Y-axis plus side boundary
- Z ： Z-axis plus side boundary
- I ： X-axis minus side boundary
- J ： Y-axis minus side boundary
- Z ： Z-axis minus side boundary

(Note 1) *You do not have to specify the axis to which you make no change. When changing only the X-axis minus side, the command will be G22 I_ ;*

(Note 2) *When changing the boudary of the plus and minus sides, the following conditions must be met.*

$$X > I, Y > J, Z > K$$

(Note 3) *For the axes other than X, Y, and Z, the boundary cannot be changed with a program command.*

18.2.3 Alarm Cancellation

When the tool enters the disabled area, release it from the disabled area by manual operation and press the RESET button to reset an alarm.

18.2.4 Cautions

(1) The boundary of the stroke limit of each axis refers to the position in the machine coordinate system.

(2) The stroke limits 2 to 4 are validated only for the axis which has completed reference point return.

(3) The distance required for the axis to stop after entering the disabled area depends on the conditions such as feed rate, etc.

(4) Specify G22/G23 in an independent block.

(5) Through parameter setting, G codes (G22/G23) to turn ON/OFF the stroke limit 2 can be specified with G36, G38/G37, or G39.

G36, G38/G37, or G39, if used, is converted into the standard G codes (G22/G23) to be read when an NC program is read into the buffer.

Thus, the command values and alarms are displayed in the standard G codes.

18.2.5 Associated Parameters

No.1300, #0 = 0   The disabled area for Stroke Limit 2 is inside.

           = 1   The disabled area for Stroke Limit 2 is outside.

No.1300, #5 = 0   Stroke Limit 3 release signal is made invalid.

           = 1   Stroke Limit 3 release signal is made valid.

No.1300, #7   When a command with which the stroke limit is exceeded is given;

           = 0   Alarm takes place after the stroke limit is exceeded.

           = 1   Alarm takes place before the stroke limit is exceeded.

No.1310, #0 = 0   Stroke Limit 2 of each axis is made invalid.

           = 1   Stroke Limit 2 of each axis is made valid.

No.1310, #1 = 0   Stroke Limit 3 of each axis is made invalid.

           = 1   Stroke Limit 3 of each axis is made valid.

No.1310, #2 = 0   Stroke Limit 4 of each axis is made invalid.

           =1   Stroke Limit 4 of each axis is made valid.

No.1322   Machine coordinate value in plus direction of Storke Limit 2 of each axis.

No.1323   Machine coordinate value in minus direction of Stroke Limit 2 of each axis.

No.1324   Machine coordinate value in plus direction of Stroke Limit 3 of each axis.

No.1325   Machine coordinate value in minus direction of Stroke Limit 3 of each axis.

No.1328   Machine coordinate value in plus direction of Stroke Limit 4 of each axis.

No.1329   Machine coordinate value in minus direction of Stroke Limit 4 of each axis.

No.3409, #0   G code to specify ON/OFF of the stroke limit check 2

           = 0   G22 (ON)/G23 (OFF)

           = 1   G36 or G38 (ON)/ G37 or G39 (OFF)

No.8658, #7 = 0   A command to set the boundary with G22. Alters a boundary value of the stroke limit 3.

           = 1   A command to set the boundary with G22. Alters a boundary value of the stroke limit 2.

18.2.6 Associated Alarms

No.F502   Overtravel: +n

        Stroke Limit 2 on + side of n axis is exceeded.

No.F503   Overtravel: -n

        Stroke Limit 2 on - side of n axis is exceeded.

No.F504   Overtravel: +n

        Stroke Limit 3 on + side of n axis is exceeded.

No.F505   Overtravel: -n

   Stroke Limit 3 on - side of n axis is exceeded.

No.F512   Overtravel: +n

        Stroke Limit 4 on + side of n axis is exceeded.

No.F513   Overtravel: -n

        Stroke Limit 4 on - side of n axis is exceeded.

18.3 Stroke Limit Check before Move

If the end point position of the block to be executed by automatic operation enters the disabled area, it results in an alarm, stopping the axes.
Of the stroke limits 1, 2, 3, and 4 all effective ones are checked.
The end position of the execution block is calculated out of "Machine coordinates" + "Remaining shift volume".
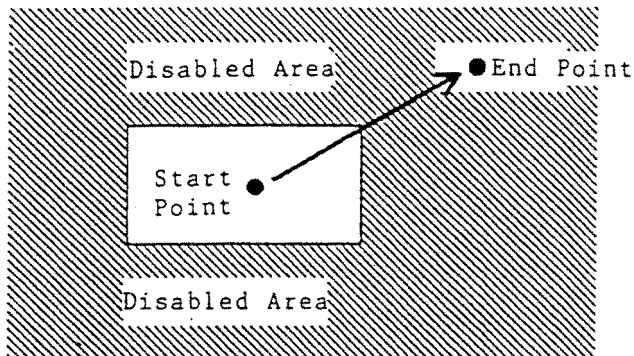
The axis stops moving if the end point of the block to be executed is in the disabled area.

18.3.1 Alarm Cancellation

When an "stroke limit before move" alarm occurs, press the RESET button to reset it.

18.3.2 Cautions

(1) A block move path is not checked.
(2) The axis for which the machine lock function is turned on is not checked.

18.3.3 Associated Parameters

No.1301, #2 = 0   G31 block is subject to checking in pre-shifting stroke limit checking.
　　　　　 = 1   G31 block is not subject to checking in pre-shifting stroke limit checking.
No.1301, #7 = 0   Disables the stroke limit before move check.
　　　　　 = 1   Enables the stroke limit before move check.
(Note) For the parameters for the disabled areas and effective axes of the stroke limits 1, 2, 3, and 4 refer to 18.1 and 18.2.

18.3.4 Associated Alarms

No.F510   Overtravel: +N
　　　　　 In pre-shifting stroke checking, the end position of the block is within the disabled area of the N axis + side stroke limit.
No.F511   Overtravel: -N
　　　　　 In pre-shifting stroke checking, the end position of the block is within the disabled area of the N axis - side stroke limit.

19. PROCESSING

19.1 Rear Processing

A machine which has a edge box for both front processing and rear Processing, and a main axis (sub-spindle) for rear processing, can perform double amount of work.

In this section, the relation between front processing and rear processing on a edge box (1 series) is described.

Front processing is performed in "the front processing mode", and rear processing is performed in "the rear processing mode".

T code described in this section means the T code which performs coordinates setting (setup), not the T code of multiple offset.

19.1.1 Switchover Between Front Processing Mode and Rear Processing Mode

Front processing mode and rear processing mode can be switched over by the modal code G (25 groups)

  G170 :   Front Processing Mode

  G171 :   Rear Processing Mode

(Note 1) *Upon powering on, G170 is selected.*

(Note 2) *This cannot be changed by resetting.*

Also, in the front processing mode (G170), the rear processing mode can be temporarily entered by specifying G52 (one shot) and the T code to the same block.

However, the modal code G (G170) cannot be changed by selecting the rear processing mode using G52.

Though G52 is one shot, the rear processing mode selected by G52 remains effective until G170 or the T code is specified, or manual turret index is performed.

19.1.2 Setting of Coordinate System (Setting Up)

Coordinate systems for front processing (front coordinate system) and rear processing (rear coordinate system) vary depending only on the Z-axis. That is, only the Z-axis has the front coordinate system and the rear coordinate system, and the other axes have only the front coordinate system.

Also, the 5-axis (axis with sub-spindle) is set up always with the hook offset amount set 0. (When specifying the setting up of the all axes, the B-axis will also be set up.)

(1) Setting Up upon Powering On

Setting up is performed in the front coordinate system.

(2) Setting Up by Manual Operation

[1] Return to Origin, Change of Offset Amount (Screen Input), Q Setter Depending on the selected processing modes (front or rear) , front coordinate system of rear coordinate system is selected and set

[2] Turret Index

Depending on the modal G code (G170 or G171), front coordinate system or rear coordinate system is selected and set up.

(3) Setting Up by Program command

[1] T code command

Depending on the modal code G (G170 or G171), front coordinate system or rear coordinate system is selected and set up.

However, even in the mode G170, setting up is performed in the rear coordinate system by the command "G52 T△△△△".

Also, only the B-axis can be set up by the command "T5100". However, this T code is processed within NC and is not output to the machine side.

[2] G54 and G55 Commands (Change of Work Length)

Depending on the selected processing mode (front or rear), the followings will be performed.

<Front Processing Mode>

G54 Z (W) ___ ; Change of work length 1 and setting up of Z-axis (Front coordinate system)

G55 B (D) ___ ; Change of work length 2

<Rear Processing Mode>

G54 Z (W) ___ ; Change of work length 1

G55 B (D) ___ ; Change of work length 2 and setting up of B-axis

(Note) *As there are relations between work length 1 and the coordinate system of the B-axis, and work length 2 and the rear coordinate system of the Z-axis, the B- and the Z-axis will be set up when performing setting up of the Z-axis and the B-axis, respectively.*

19.1.3 Program Example

Transition of the processing modes (front or rear) and the coordinate systems (front or rear) is described. The actual processing programs are not taken into consideration.

G170 ;	Front Processing Mode
T△△△△ ;	Setting Up of Front Coordinate System
G54 Z (W)___ ;	Change of Work Length 1 and Setting Up of Z-axis
G55 B (D) ___ ;	Change of Work Length 2
T△△△△ ;	Setting Up of Front Coordinate System
G52 T△△△△ ;	Rear Processing Mode and Setting Up of Rear Coordinate System
G54 Z (W)___ ;	Change of Work Length 1
G55 B (D)___ ;	Change of Work Length 2 and Setting Up of B-axis
T△△△△ ;	Front Processing Mode and Setting Up of Front Coordinate System
G171 ;	Rear Processing Mode
T△△△△ ;	Setting Up of Rear Coordinate System
T5100 ;	Setting Up of B-axis
T△△△△ ;	Setting Up of Rear Coordinate System
G170 T△△△△ ;	Front Processing Mode and Setting Up of Front Coordinate System
G5100 ;	Setting Up of B-axis

19.1.4 Synchronization Feeding of Spindle

(Feeding for every rotation; G99/Wrenchin; G32)

Before executing the synchronization feeding of spindle, spindle (main or sub) to be synchronized is automatically determined as follows.

Front Processing Mode ... In synchronization with the main spindle.

Rear Processing Mode ... In synchronization with the sub-spindle.

(Note) The conventional codes G (G110 to G112) cannot be used.
(Alarm will not be generated when specifying. However, it will be disregarded within NC.)

19.1.5   S Code Command

There are S code commands for the main spindle, rotation tool and sub-spindle.

Proper S code command is selected follows.

(1) Command of number of rotations (Constant control of the rotation speed is canceled; G97)

   [1] Command of S code unit

      Depending on the status of the processing mode and spindle, proper S code will be automatically identified.

      <Front Processing Mode>

         When the main spindle and the C-axis are not connected.; For main spindle

         When the main spindle and the C-axis are connected.; For rotation tool

      <Rear Processing Mode>

         When the sub-spindle is not identified.; For sub-spindle

         When the sub-spindle is identified.; For rotation tool

      (Note) This judgment is carried out in ladder processing on Machine side.

   [2] Commands of S and G codes

      Proper S code will be selected depending on the G code.

         G261 S___ ; For main spindle

         G262 S___ ; For rotation tool

         G263 S___ ; For sub-spindle

      (Note) G261 to G263 are one-shot G codes.

   [3] Switchover by the conventional M codes (M191 to M193) cannot be performed. M191 to M193 are output to the machine side as the normal M codes.

(2) Constant Control of Rotation Speed (G96 and G196)

   This is determined depending on the G codes.

      G96 S___ ; For main spindle

      G196 S___ ; For sub-spindle

   (Note 1) G96 and G196 are modal G codes.

   (Note 2) G96 can be specified only in the front processing mode. And G196 can be specified only in the rear processing mode.

   Also, switchover of the processing modes must be performed when the constant control of the rotation speed is canceled (G97).

   The G196 status in the front processing mode and the G96 status in the rear processing mode generate alarms. (Alarm No.184 Rear processing command is error.)

(3) Setting of Maximum Rotation Number (G50)

This is determined by the value of P which specifies the block of G50.

G50 S___ (P1) ; For main spindle

G50 S___ P2 ; For rotation tool

G50 S___ P3 ; For sub-spindle

(Note 1) *G50 is the one-shot G code.*

(Note 2) *P1 can be omitted.*

19.1.6   Mirror Image of Z-axis

By executing mirror image to the origin (location of 0) of the work coordinates system of the Z-axis, program for the front processing can be used for the rear processing.

This function can be implemented by using general-purpose programmable mirror images (G511 and G501)

G511 Z0 ; Mirror image is executed to 'Z0'.

Symbols of the movement commands of the Z-axis are handled inversely.

Z □□□ → Z - □□□ , W □□□ → W - □□□

G501 ; Cancellation of mirror images.

When mirror image is executed to the Z-axis on the G18 plane (Z-X),

- the clockwise rotation (G02) and the counterclockwise rotation (G03) of the arc command; and
- the left side (G141) and the right side (G142) of the automatic offset of the R edge; are handled inversely with program.

Automatic offset of the edge R are performed in accordance with shapes to which mirror image is executed. Therefore, regardless of mirror image, the virtual edge point of tool (control point) must be set on the same location as when performing the normal rear processing.

19.1.7   Work Delivery Function

The work delivery function, consisting of the lathe in rear spindle (sub-spindle) specification, serves to deliver work from the spindle to the sub-spindle as keep them rotating synchronously.

This function is composed of the following 5 independent functions:

[1]   Spindle synchronous control
   → Synchronously rotates (also, enabling phase control of) the spindle and the sub-spindle.

[2]   Torque clamp of rear spindle
   → Limits torque of the rear spindle.

[3]   Torque clamp of B axis
   → Limits torque of B axis

[4]   Torque limit skip
   → Limiting torque of B axis, pushes B axis (rear spindle) against the spindle work to stop B axis movement.

[5]   Work pull
   → Limiting torque of B axis and holding a work with both spindles, pulls the work with B axis. This is used in cutting-off of bar.

The above [1], [2] and [3] , in terms of program commands, are processed on Machine side (PMC ladder). The [4] and [5] functions are processed on the CNC, which are described as follows:

(1)   Work Delivery mode
   "In Work Delivery Mode" needs to be informed from Machine side so that torque limit skipping and work pulling functions can be automatically performed.
   Normally "In Work Delivery Mode" indicates the torque clamp state of B axis.
   (Note) An M code whose signals are changed in Work Delivery mode (B-axis torque clamp ON/OFF) should be made the M code to stop pre-reading.

(2)   Torque Limit Skip
   When cutting feed (G01) is commanded in the minus direction of B axis in Work Delivery mode, the CNC changes it into a torque skip command for execution.
   The B axis (rear spindle) is pushed against the spindle work in torque limit skipping and, when torque reaches the certain level, skips the rest of the shift commands to execute the following block.

(3) Work Pull

When a chuck close command is given for the sub-spindle in Work Delivery mode, the CNC, following completion of chuck closure, executes the work pulling command with B axis at the shift amount and speed having been set in the parameters.

The following is a sample program to perform work delivery:

G261 S___ M03 ;	Spindle forward rotation
G263 S___ M114 ;	Rear spindle reverse rotation
M160 ;	Synchronous Rotation mode ON
⟲	
M119 ;	Rear spindle chuck open
M128 ;	B axis torque clamp ON
G98 G01 B___ F___ ;	Work pushing (B-axis torque limit skip)
G04 P___ ;	
M126 ;	Rear spindle torque clamp ON
M118 ;	Rear spindle chuck closed (On completion, work is pulled.)
⟲	
(Cutting-off)	
⟲	
M69 ;	Spindle chuck open
G98 G01 B___ F___ ;	B axis shifted
M127 ;	Rear spindle torque clamp OFF
M129 ;	B axis torque clamp OFF
M161 ;	Synchronous Rotation mode OFF

[Associated Parameters]

[1] Spindle Synchronous Control

No.4032   Adjustable speed time constant on spindle synchronous control (rpm/sec)

No.4033   Spindle synchronous rpm reach level (rpm)

No.4034   Shift amount on spindle phase synchronous control (Pulse: 360°/4096)

No.4035   Spindle phase synchronous compensation data (Pulse/2m sec)

(Note) *No.4034 indicates a shift amount at start of phase synchronous control and, while in phase synchronous control, the shift amount can be added with the program command (G10 L100 R___ ;).*

[2] Rear Spindle Torque Clamp

No.4025   Torque limit value (%)

[3] B-Axis Torque Clamp

There is no parameter.

The clamp value is set with the PMC ladder. With a clamp value (%) being set, clamp ON. With the initial value (100%) returned, clamp OFF.

[4] Torque Limit Skip

There is no parameter.

[5] Work Pull

No.8690   M coder to perform work pulling (rear spindle chuck closed)

No.8691   B-axis shift amount for work pull

No.8692   B-axis shift speed for work pull (per-minute feed)

(Note) *When the set value for work pull M code is 0, it is taken as M118.*

When the set value for the B-axis shift amount or the B-axis shift speed is 0, no work pull operation is performed.

19.1.8 Notes

(1) "Front processing mode and rear processing mode" and "G170 and G171" described in this section mean the status and the G code, respectively.
Therefore, the processing mode and the C code may not match.

	"Processing Mode"	"G code"
G170 ;	Front	G170
G52 T△△△△ ;	Rear	*G170
T△△△△ ;	Front	G170
G171 ;	Rear	G171

(2) In the rear processing mode, "Z" of the work coordinates (edge location) is displayed with color which varies depending on the current processing are displayed.

(3) The following S-related data displayed at the lower left of the Overall screen differs depending on whether the front or back machining mode is selected.

- Rotation S (Surface speed V)
- Actual rotation
- Clamp S

The main spindle data are displayed during the front machining mode, and the subspindle data during the back machining mode, respectively.

(4) Rear processing mode is an optional function of NC.
When an option of the rear processing mode is not available, the following command generates alarm. (No.191 Optional Command)
    G171, G196, G55, G52

(5) Depending on the settings (parameter) of the turret plane numbers and the tool offset pair numbers, error checking of the T command is performed. The T code must meet the following conditions.
    <When ; T △ △ □ □ >
    $0 < △△ \leqq$ Number of turret planes
    $0 \leqq △△ \leqq$ Number of pairs of tool offsets
When these conditions are not met, alarm will be generated (No.182 T-command error). However, the "T5100" command is excluded.

(6) In the case of the setup command of all axes (T code and manual turret index), setup of the B-axis can be canceled depending on the parameter settings.
    Parameter No.1200, #7= 0 : B-axis is set up. (Standard)
                    = 1 : B-axis is not set up.

19.1.9　Details of Coordinate System

(1) With Rear Processing

<Front Coordinate System> The same conditions as when no rear processing is performed will be applied to the axes other than B-axis.

X axis : Work shift amount + Machine coordinated - Tool offset

   [Xf]　　　　[X0]　　　　　　　　[Xm]　　　　　　　　　[X]

(-External work origin offset)

           [xe]

Y axis : Work shift amount + Machine coordinated - Tool offset

   [Yf]　　　　[Y0]　　　　　　　　[Ym]　　　　　　　　　[Y]

(-External work origin offset)

           [Ye]

Z axis : Work shift amount + Machine coordinated - Tool offset

   [Zf]　　　　[Z0]　　　　　　　　[Zm]　　　　　　　　　[Z]

- Work length 1 (- External work origin offset)

     [W1]　　　　　　　　　　[Ze]

C axis : Work shift amount + Machine coordinated

   [Cf]　　　　[C0]　　　　　　　　[Cm]

(- External work origin offset)

           [Ce]

B axis : Work shift amount + Machine coordinated - Woke length 1

   [Bf]　　　　[B0]　　　　　　　　[Bm]　　　　　　　　[W1]

- Work length 2 (- External work origin offset)

   [W2]　　　　　　　　　　　　　　[Ze + Be]

(Note)　The value of work length 2 (W2) includes the amount of hook offset.

<Rear Coordinate System> The same conditions as when the rear processing is performed will be applied to the aexs other than Z-axis.

Z-axis : - { [Coordinate Value of B-axis] - [Front Coordinate Value of Z-axis] }

   [Zb]　　　　　　　　　　[Bf]　　　　　　　　　　　　　　　[Zf]

[Coordinate System of Z- and B-axes]
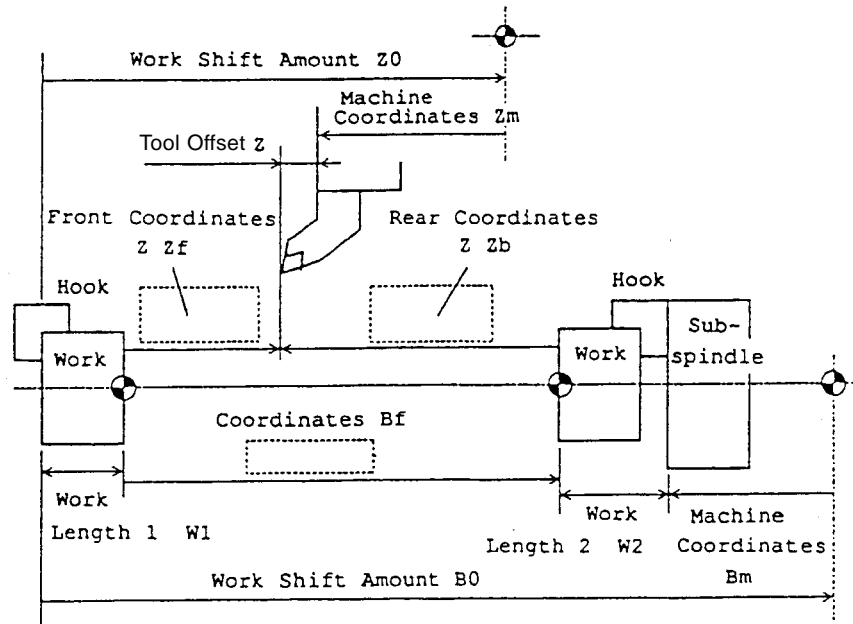
(2) Less Rear Processing

<Front coordinate system only>

X axis : Work shift amount + Machine coordinates - Tool offset

[Xf]   [X0]   [Xm]   [X]

(- External work origin offset)

[Xe]

Y axis : Work shift amount + Machine coordinates - Tool offset

[Yf]   [Y0]   [Ym]   [Y]

(- External work origin offset)

[Ye]

Z axis : Work shift amount + Machine coordinates - Tool offset

[Zf]   [Z0]   [Zm]   [Z]

- Work length 1 (- External work origin offset)

[W1]   [ze]

C axis : Work shift amount + Machine coordinates

[Cf]   [C0]   [Cm]

(- External work origin offset)

[Ce]

B axis : Work shift amount + Machine coordinates

[Bf]   [B0]   [Bm]

(- External work origin offset)

[Be]

(Note)   *For details, see "7.2 Tool Nose Coordinate System."*

19.2 Polygon Turning (Polygon Turning Between Spindles)

Polygon turning is used to work a polygon for which a work and the tool are turned at a fixed rate. Through changing of the work-tool revolution rate and the number of knives, a polygon becomes a tetragon or a hexagon.

It is more advantageous than the polygon machining with C and X axes through using polar coordinate interpolation as it requires less machining time.

On the other hand, it is defective since the polygon thus obtained is not quite perfect. Normally, it is applied in machining bolt heads of square/hexagon head bolts and hexagon nuts.

This function consists of two serial spindles including 1st spindle as the work turning axis (master axis) and 2nd spindle as tool turning axis (polygon synchronous axis) where spindle rotation is controlled at a fixed rate.

19.2.1 Command Format

(1) Mode Command

    G10 L110 P___ Q___ R___ ; .... Setting polygon rotation ratio and phase
    M△△△ ; .... Polygon Synchronous mode ON
                                                        (M△△△ indicates parameter setting) →
                                                        standard M156

Setting each command value (P, Q, R) in Polygon Synchronous mode with G10 command, start Polygon Synchronous mode by using an M code.

    P : Revolution rate of spindle (master axis)
        Command range: Integer of 1 to 999
                        (The rotation direction of the spindle is fixed with the
                        ordinary spindle rotation command [M03/M04, etc].)

    Q : Revolution rate of rotary tool (polygon synchronous axis)
        Command range: Integer of 1 to 999, -1 to -999
                        (The rotation direction of the rotary tool is fixed with a code
                        of the Q command value. However, for Parameter
                        No.7603, #1 (QDRC) =1, it is fixed by the spindle rotation
                        direction. In this case, a minus value cannot be
                        commanded in Q.)

    R : Commanding of phase for spindle and rotary tool (Commanding of absolute
        value)
        Command range :   Integer of 0 to 359
        Command unit : 1 degree

    (Note) *The R is omissible, where no commanding makes phase 0.*
            When phase control is made invalid in Parameter No.7602, #5 (PCOF) =1,
            an alarm takes place if R has been commanded.

P, Q, or R value having been set once in G10 command stays modal until Polygon Synchronous mode is cancelled.

While in Polygon Synchronous mode, with an S command given to the spindle, the rotary tool turns as the polygon synchronous axis at speed equal to SxQ/P, whose phase is controlled to become R.

(2) Cancel Command

M □□□ ; ...... Polygon Synchronous mode OFF

(M □□□ indicates parameter setting) → Standard: M157

With the above command, Polygon Synchronous mode is cancelled Whether or not to cancel Polygon Synchronous mode on resetting is determined by Parameter No.7603, #0 (RPLM).

19.2.2   Sample Program

When cutting a tetragon in polygon turning through use of a two-flute tool:

G00 X100.0 Z10.0 ;	Approach
G97 S1000 M03 ;	Spindle forward rotation
G10 L110 P1 Q2 ;	Polygon rotation rate setting (1 : 2, phase 0)
M156 ;	Start of polygon turning
≀	(Rotation start of rotary tool: S2000)
(Cutting)	
M157 ;	End of polygon turning (Rotation stop of rotary tool)

19.2.3   Cautions

(1) When using G10 command and M156/M157 commands, be sure to command them in independent blocks.

(2) In commanding of G10, while R is omissible, be sure to command P/Q.

(3) The M codes for starting and ending polygon turning are processed only inside the CNC, which are not output into Machine.

(4) The M codes for starting and ending polygon turning become M codes to stop pre-reading.

19.2.4   Associated Parameters

No.762, #0   While in Spindle-Spindle Polygon Turning mode, the rotation direction of the spindle (master axis) is:
= 0   Not reversely turned.
= 1   Reversely turned.

#1   While in Spindle-Spindle Polygon Turning mode, the rotation direction of the rotary tool (polygon synchronous axis) is:
= 0   Not reversely turned.
= 1   Reversely turned.

#3   When Spindle-Spindle Polygon Turning mode is provided with phase control, to select the spindle to perform phase shift for phase fitting:
= 0   Shift the rotary tool (polygon synchronous axis)
= 1   Shift the spindle (master axis)

#4   When Spindle-Spindle Polygon Turning mode is provided, the control system when the command for Spindle-Spindle Polygon Turning mode is given:
= 0   Enters Spindle-Spindle Polygon Turning mode as keeping the current spindle speed.
= 1   Enters Spindle-Spindle Polygon Turning mode after automatically stopping the spindle.

#5   While in Spindle-Spindle Polygon Turning mode, phase control is:
= 0   Provided.
= 1   Not provided.

No.7603, #0   Spindle-Spindle Polygon Turning mode on resetting is:
= 0   Cancelled.
= 1   Not cancelled.

#1   The rotation direction of the rotary tool (polygon synchronous axis) is:
= 0   Determined by the code of a Q command value (+/-).
= 1   Conforming to the rotation direction of the spindle (master axis).

No.7621   The maximum rpm of the polygon synchronous axis.

No.7631   The spindle rpm deviation allowable level while in spindle-spindle polygon turning.

No.7632   The steady-state confirming time while in spindle-spindle polygon turning.

No.8678   M code to start polygon turning (For 0: M156)

No.8679   M code to end polygon turning (For 0: M157)

19.2.5 Associated Alarms

No.100   An error exists in G10 command.

(#071)   In the command before entering Polygon mode, P or Q command does not exist.

(#072)   In the command while in Polygon mode, only one of P or Q has been commanded.

(#073)   The P command value is beyond the set range.

(#074)   The Q command value is beyond the set range.

(#075)   The R command value is beyond the set range.

(#076)   The R command exists when phase control is invalid.

No.218   An error exists in the polygon turning command.

(#001)   The M code command to start or to end polygon turning is not independent.

(#002)   An polygon turning rate has not bee commanded before starting of polygon turning.

20.1　Program Resumption

By assigning a sequence No. and the number of times of repetition, you can resume a program in the assigned block.

By using this function, on occurrence of tool failure or power suspension, you can resume processing in the block with the assigned sequence number.

Also, it is possible to use this as the high-rate tape checking function.

20.1.1　Operation Method

(1) On supply of power or cancellation of emergency stop, all steps required to be performed before auto operation including recovery of the reference point, etc. are carried out.

When a tool has been changed due to tool failure, etc., tool compensation amount is reset as required.

(2) Before starting work, call, to the cutting face on a tool rest, a tool to be used at the beginning of processing.

(3) Selecting [Edit Mode] on the machine operation panel, push [Reset] of the NC operation panel to reset the NC unit and call the program head.

(4) Selecting [Memory mode] on the machine operation panel, push [Program Resume].

(5) "PROGRAM RESTART" page is automatically displayed.

(6) Input, in "P" of "resumption data", the sequence No. of a block to be resumed. When the same sequence numbers should appear, input, in "L", the number of times the sequence No. being repeated.

To assign a program No. to be resumed, input a program No. in "O".

(7) With $\boxed{\text{F9/Search}}$ pushed, searching starts.

(8) On completion of searching, "Resumption Data" value is erased.
For "M Code", the M codes having been commanded in the past 32 times are displayed.
For "Tool T", a tool to be used when processing is resumed is displayed in a T code.
For "Spindle S", a S code commanded for the last time is indicated.
For "B", a B code commanded for the last time is indicated.

(9) [Program Resume] of the machine operation panel is turned off.

(10) When M/S codes to be output exist on the page, select MDI mode. Input a M code in "M" and a S code in "S" of "Resumption Data", and push [Start] on the machine operation panel.

(11) Pushing $\boxed{\text{F2/Resume Position}}$, confirm that no interference of a work occurs when the tool moves from the current position to the resumption position.
If it occurs, move it, in Manual mode, to a place where no interference takes place.
"Resume Position" indicates, in a work coordinate system (tip position), the place where processing is resumed.
"Resume remaining shift amount" indicates distance from the current tool position to the work resumed place.

(12) Selecting, on the machine operation panel, [Memory mode], push [Start).At this time, the tools move, in the order set in the parameters, to the work resumed place by one axis in dry run and auto operation is resumed in the block of the assigned sequence No.

20.1.2 Cautions

(1) With [RESET] pressed during sequence No. searching, start Program Restart operation all over again.

(2) When [PROGRAM RESTART] of the machine operation panel is ON, no [START] can take place.

(3) 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.

(4) In the following cases, the program cannot be restarted properly.
a) When operated with the machine lock function applied.
b) When the program is restarted in the block prior absolute programming after the block where a skip (G31) command was specified.
c) When, following completion of sequence No. searching, setting or changing of the coordinate system has been carried out.
Not that none of the above cases results in an alarm.

(5) You cannot restart the program from the following blocks:
a) Block while in the polar coordinate interpolation mode (G121)/the cylindrical interpolation mode (G271)
b) Halfway the finish profile block specified with P and Q of the multiple repetitive cycle (G71-G73)

(6) When a program No. has been assigned on start of sequence No. searching, you can choose in a parameter whether to start searching after calling the program No. or to start by using the current program No. and then, to search a sequence No. in the assigned program (sequence No. search within the subprogram.)

(7) An alarm results when sequence number cannot be collated.

(8) With ATC, program restart from the block with a T command other than compound compensation is not possible.
Alarm will be output.
In such cases, shift the T command to the next block.

    B100 T010100 ;   → N100 ;
                           T010100 ;

(9) When a C-axis command exists between the program head and the restart block in a machine using C axis control of the built-in spindle, perform zero point return before starting the sequence number search by switching to C axis.

20.1.3   Associated Parameters

No.8657, #0   When O number has been assigned on start of Program Restart,

      = 0   After the assigned O number is called, N number searching is performed.

      = 1   Searching is started with the currently called program followed by N number searching within the assigned program.

No.8657, #1   When the block while in drilling fixed cycle is searched with Program Restart,

      = 0   No. of repetition (L) assigned is equal to No. of holes

      = 1   No. of repetition (L) assigned is equal to No. of specified blocks.

No.8703   Order in which shifting takes place on Program Restart in dry run to the restart position

20.1.4   Associated Alarms

No.140:   Error in the program restart operation.

(#001)   N number search starting operation is carried out when it is not in the reset state (OP=1).

(#002)   N number search starting operation is carried out with the halt signal (*sp)=0

(#003)   Although an assigned 0 number has been tried to be called, there is no relevant program.

(#004)   The program for starting N number searching has not been called out.

(#005)   No tool No. has been selected on start of N number searching.

(#010)   "PROGRAM RESTART" signal has been turned off while in N number search.

(#011)   The assigned N number has not been found.

No.189:   Error in the parameters setting. (parameter No.8703)

20.2　Return to Machining Interruption Point

This is the function of returning the automatic operation to the machining interruption point after the axis has been shifted by manual operation during automatic mode (memory, MDI) for measuring the workpiece or for removing the chips.

```
Return to          <·········································○ Escape position
interruption point
                                            Shift by manual operation



                                 Interruption
                                 point
                                                    ←───────────  Program path
```
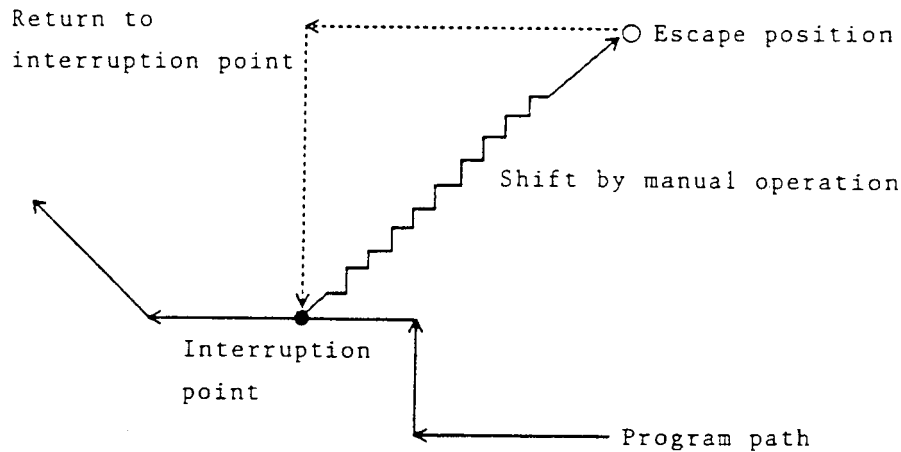
20.2.1　Operating Method

[1]　Interrupt the automatic operation by single block or feed hold (stop).
This position (work coordinate system) will be recorded as the machining interruption point.

[2]　In the manual mode (feed, handle), move the tool by manual operation.

[3]　Perform the workpiece measurement or remove the chips.

[4]　Turn on the [Machining Interruption Point Return] switch.

[5]　Move in the direction of the machining interruption point in the feet mode. [Machining interruption point return operation]

- The shift speed at this stage will be the same as the normal manual feed, and this will be the manual feed rate selected on the machine operation panel.
- When the machining interruption point is reached, even if you continue to press the manual feed button, the axis shift will star automatically.

[6]　Turn off the [Machining Interruption Point Return] switch.

[7]　Return to the automatic mode, press the start button and restart automatic operation.

20.2.2 Precautions

(1) Machining interruption point will be recorded for all the axes as the position of the work coordinate system when the automatic operation was last interrupted.
The recorded machining interruption point can be canceled by the reset operation.

(2) Manual feed when the machining interruption point has not been recorded will be the sane as the normal manual operation, even if the [Machining Interruption Point Return] switch is. ON.

(3) Only shift in the direction of the machining interruption point is possible in the operation for returning to the machining interruption point. Shift in the opposite direction is not possible.
Also, if you stop pressing the manual feed button during the return operation, the machine will stop at that position. If you press the manual feed button again, the return operation will continue.

(4) After recording the machining interruption point and until the return to he machining interruption point, if you shift the axis in the machine-locked status, you may not be able to return to the correct position.
(The work coordinates will be restored to the correct position but there will be an offset in the machine coordinates equal to the shift during the machine-locked status.)

20.3 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.

20.3.1 Setting of the Sequence Number

Input the sequence number you want to stop in the Setting screen.
The set sequence number becomes zero at the time of sequence number comparison and stop, or reset.

20.3.2 Cautions

(1) 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.

(2) When one program contains multiple identical sequence numbers, the machine stops after executing the program which matches first.

(3) When there is one block which contains multiple actions (for example, canned cycle), and its sequence number is consistent, the machine stops after performing the last action of that block.

(4) When the sequence number of the block where the number of canned cycle repeat times has been specified is consistent, the machine stops after executing the first repeat.

(5) When the sequence number subject to stopping exists both in the main program and the subprogram, not only the sequence number but also the program number can be assigned in the parameter.

With setting of the program number ≠ 0, the program number has been assigned. Assignment of the program number, however, cannot be performed on "Setting Screen". Use "Parameter Screen".

Further, the parameter to set the program is also cleared on completion of comparison/stop or on resetting.

20.3.3 Associated Parameters

No.3473   Program number for sequence number comparison/stop.
No.3474   Sequence number for sequence number comparison/stop.

20.4   Manual Absolute ON/OFF

If manual absolute is turned on, the stroke by manual operation during program operation is added to the program coordinate values (work coordinates, machine coordinates, relative coordinates).

The then manual intervention amount is processed in next program operation, and thereafter, the tool takes the sane path as when there was no manual intervention.

If manual absolute is turned off, the stroke by manual operation during program operation is not added to the program coordinate values. In the subsequent program operation, the tool path will be shifted by the manual intervention amount from the original program.

(Note) *In the SEICOS-L III, manual absolute is fixed to ON.*
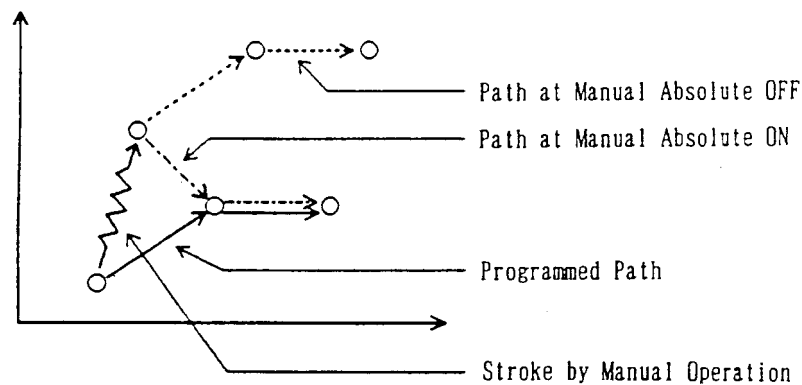
20.4.1   Sample Program for Manual Absolute ON

```
N1 G98 G01 X200. Z100. F500 ;
N2 X300. Z200. ;
N3 U100. Z300. ;
N4 Z400. ;
N5 X500. ;
```
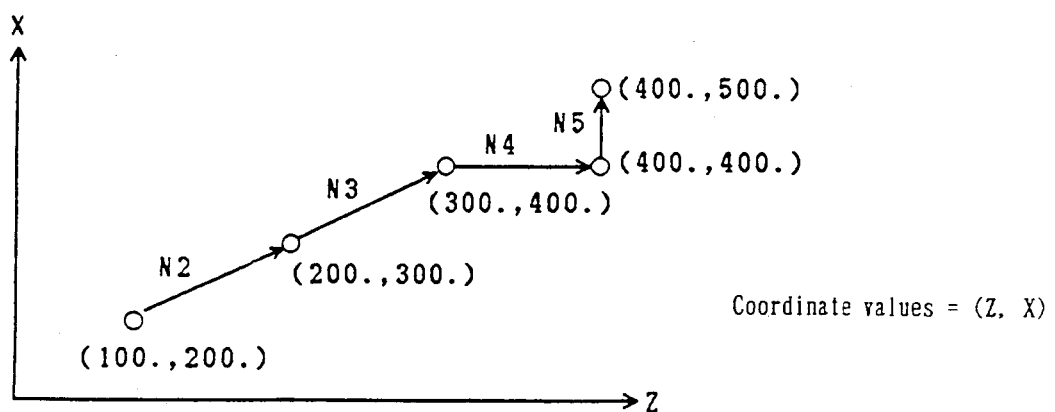
(1) When a single block stop is applied upon completion of the N1 block, and the N2 block is executed after moving the X axis by +150 through manual intervention.
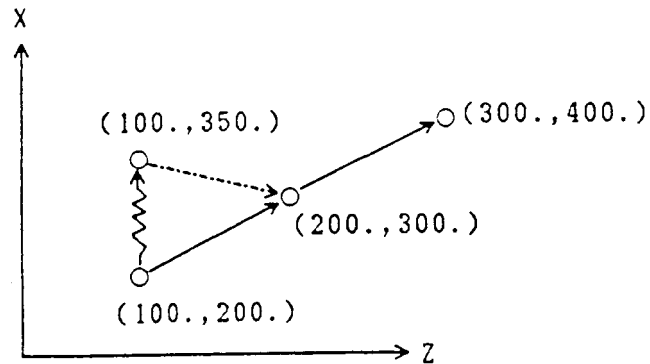
(2) When the FEED HOLD switch is pressed during execution of the N2 block, and execution of the N2 block is restarted after moving the X axis by +150 through manual intervention.
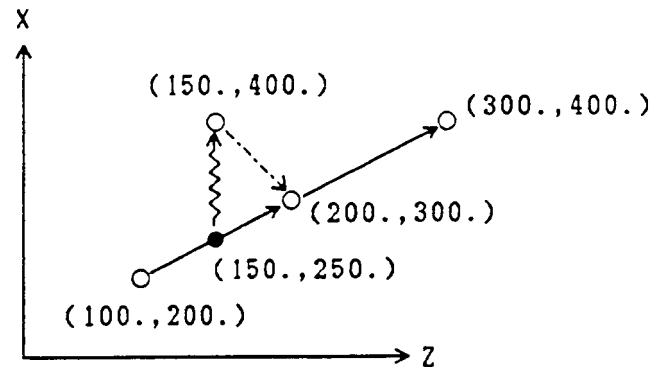
(3) When a single block stop is applied upon completion of the N2 block, and the N3 block is executed after moving the X axis by +200 through manual intervention.
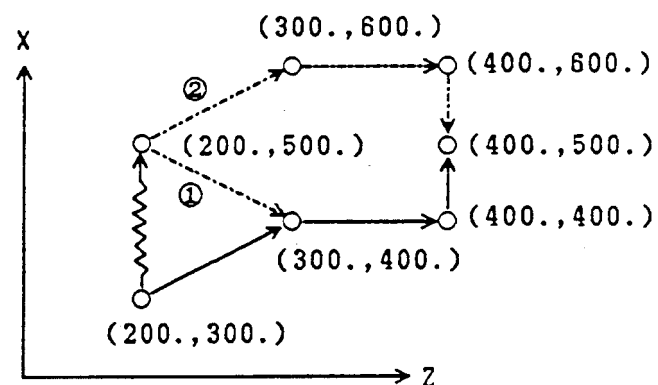
[1] When offsetting with an incremental command
[2] When offsetting with an incremental command

20.4.2 Cautions

(1) When there is an incremental command in the block to be executed after manual intervention, you select with a parameter whether to offset the manual intervention amount.
If it is not offset, do so in the subsequent block containing an absolute command.

(2) Offset the manual intervention amount in the block which has a move command for the manually intervened axis.
For example, when a single block stop is applied upon completion of the N3 block, and the N4 block is executed after moving the X and Z axes through manual intervention, the Z axis is offset in the N4 block, and the X axis in the N5 block.

(3) Through parameter setting, the block under Item (2) can be changed into the one with the axial command (whether or not shifting exists).

(4) In the arc interpolation/thread cutting block, no compensation of manual intervention is performed. It is offset in the subsequent block.

20.4.3 Associated Parameters

No.3403, #4 =0   An incremental command in the block next to manua1 intervention takes the same path as an absolute command.

      =1   Same path as at manual absolute OFF

No.3403, #6   Compensation in the manual intervention amount with manual absolute ON is:

      =0   Performed in the shift block of the axis. (Command + shifting amount)

      =1   Performed in the specified block of the axis.

20.5   Reset (Reset Associated with Automatic Operation)

Reset operation (pressing the RESET button or inputting an external reset signal) places the NC unit in the reset state.

The NC unit does the following;

(a) Deletes the preread buffer.

(b) Initializes the execution buffer.

(c) Cancels various compensations; tool nose radius compensation, cutter compensation, groove width compensation, too multiple offset.
(Does not perform offset operation)

(d) Resets an alarm.

(e) Stops an axis move.

20.5.1   Description of Initialization of the Execution Buffer

(1) Initialization of the G codes

    * Group 01 : G00 or G01 (parameter setting)

    * Group 02 : G18

    * Group 03 : G90 or G91 (parameter setting)

    Group 04 : G22 or G23 (status hold)

    * Group 05 : G98 or G99 (parameter setting)

    Group 06 : G20 or G21 (status hold)

    Group 07 : G40

    Group 09 : G80

    Group 10 : G198

    Group 13 : G64

    Group 14 : G67

    Group 15 : G501

    Group 16 : G150

    Group 17 : G96 or G97 (status hold)

    Group 18 : G131 (with option)/G130 (without option)

    Group 22 : G120

    Group 23 : G25 or G26 (status hold)

    Group 25 : G170 or G171 (status hold)

    Group 27 : G143

(Note 1) *The G codes of other than the above G code groups are deleted.*

(Note 2) *G code groups marked with "*" will not be initialized when Parameter No. 3402, #7 = 1. (The G code specified last will remain.)*
However, when the C code of the 01 group initializes G00 or G01 when the G code specified last is other than G00, G01, G02, G03, G32 or G34.

(Note 3) *The G code of 03 group will be effective only when the G code system is B or C.*

(2) The numeric values of addresses O, N, F will be maintained.

20.5.2 Related Parameters

No.3402, #0 = 0   Reset status is G00

         = 1   G01

No.3402, #7 = 0   G91

         = 1   G90   Only for B, C systems

No.3402, #4 = 0   G98

         = 1   G99

No.3402, #7 = 0   Initializes the commands given below at reset.

         = 1   Does not initialize

                (G code group: 01, 02, 03, 05)

No.3402, #0 = 0   Reset status is G00

21. CUSTOM MACROS

21.1   Program Call

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 call command.

```
      Main Program                          Subprogram

                                         O8000 ;
   M98  P8000 ;                             ₹
                                         M99 ;
```
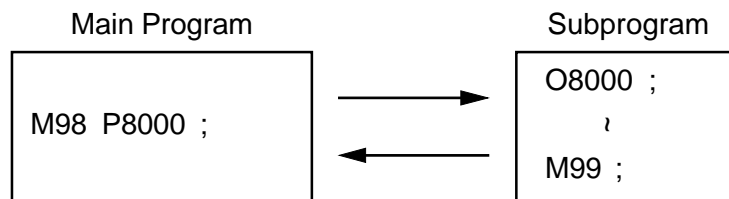
A pattern, which has part of the program replaced with variables, is registered in the memory as a macro program in advance. That registered macro program is called with a representative instruction and can be executed with real numbers assigned to variables. This representative instruction is referred to as a macro call instruction.

```
      Main Program                          Subprogram

   G65  P9000                           O9000 ;          Real number is
   <Argent> ;                           X#1 ;            assinged when
                                         M99 ;            executed
```
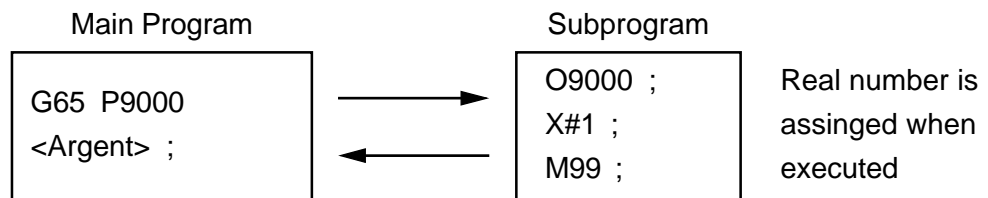
It is possible to call another program from a called program. This is called multi-call. When a certain program is calling another program, that called program is referred to a child program.
When a certain program is being called from another program, that calling program is referred to a parent program.

```
   Main program          Sub-program           Macro program
                   Child                 Child
                   Parent                Parent
                        Single                 Double
```
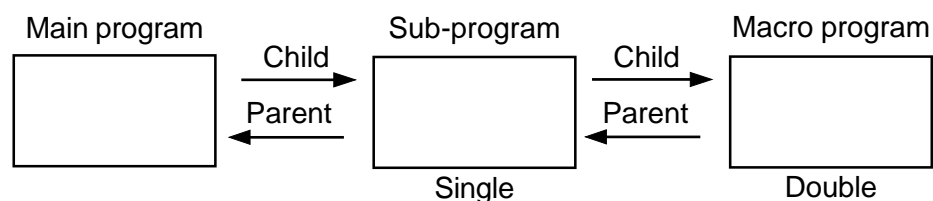
The following table shows the program call and return commands.

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

21.1.1   Subprogram Call

M98 P.... Q.... L.... ;

With this command, the program which begins with the sequence number Q of the program number specified by P is called and executed L times.

If P is omitted, the program which begins with the sequence number Q of the current program number will be called.

If Q is omitted, the program which starts from the head of the program number specified by P will be called.

If L is omitted, the program will be executed once. No arguments can be specified.

21.1.2   Macro Simple Call

G65 P.... L.... <argument> ;

This command calls the 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.

21.1.3 Macro Modal Call

    G66 P.... L.... <argument> ;

This command on 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.... ;   The macro is called in the block containing a move in this portion.

        Z .... ;

    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.

21.1.4 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 codes can be set for 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		O 9010
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 parameter No.6030=-100, O9010 is called by G100, that is model call.

21.1.5 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 availables as an argument; G: #10, L: #12, P: #16.

In this case, any 10 sets of M codes can be set for 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 ;
PRA6050		O 9020
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.

21.1.6 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 for the parameters out of M01 through M999999.

The MF and M codes are not sent out.

Parameter		M code calling ;
PRA6040		O 9000
6041		9001
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.

21.1.7 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 can not be specified.

The TF and T codes are not sent out.

Parameter

									#0
PRA6000									TCS

TCS = 0 : Does not call the subprogram by the T code.

= 1 : Calls the subprogram by the T code.

21.1.8 Subprogram Call by 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 can not be specified.

The SF and S codes are not sent out.

Parameter

								#1	
PRA6000								SCS	

SCS = 0 : Does not call the subprogram by the S code.

= 1 : Calls the subprogram by the S code.

21.1.9 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 for the parameter No.1020.

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 than 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

(Note) *For G code system A; U/V/W cannot be made the addresses for 2nd miscellaneous function.*

Parameter

PRA6000

					#2 BCS			

BCS = 0 : Does not call the subprogram by the 2nd miscellaneous function code.
= 1 : Calls the subprogram by the 2nd miscellaneous function code.

21.1.10 Return from 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 Lxxxx ;

This 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.

21.2　Multi-Call

21.2.1　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.

21.2.2　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.
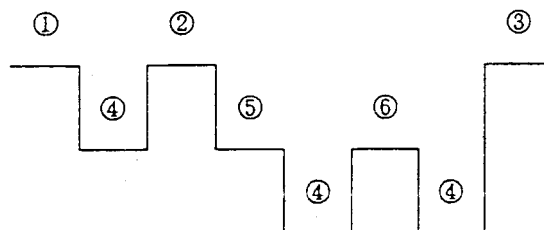
21.2.3 J macro Multiplicity and Local Variable

If the macro is called, macro multiplicity(level) increases by one.
The local variable level also increases by one, accompanying it.

Main Program (Level 0)	Macro (Level 1)	Macro (Level 2)
G65 P2000 <Argument> ;	O2000 ; G65 P4000 <Argument> ; M99 ;	O4000 ; M99 ;

Local Variable (Level 0)	(Level 1)	(Level 2)
#1 #2 ~ #33	#1 #2 ~ #33	#1 #2 ~ #33
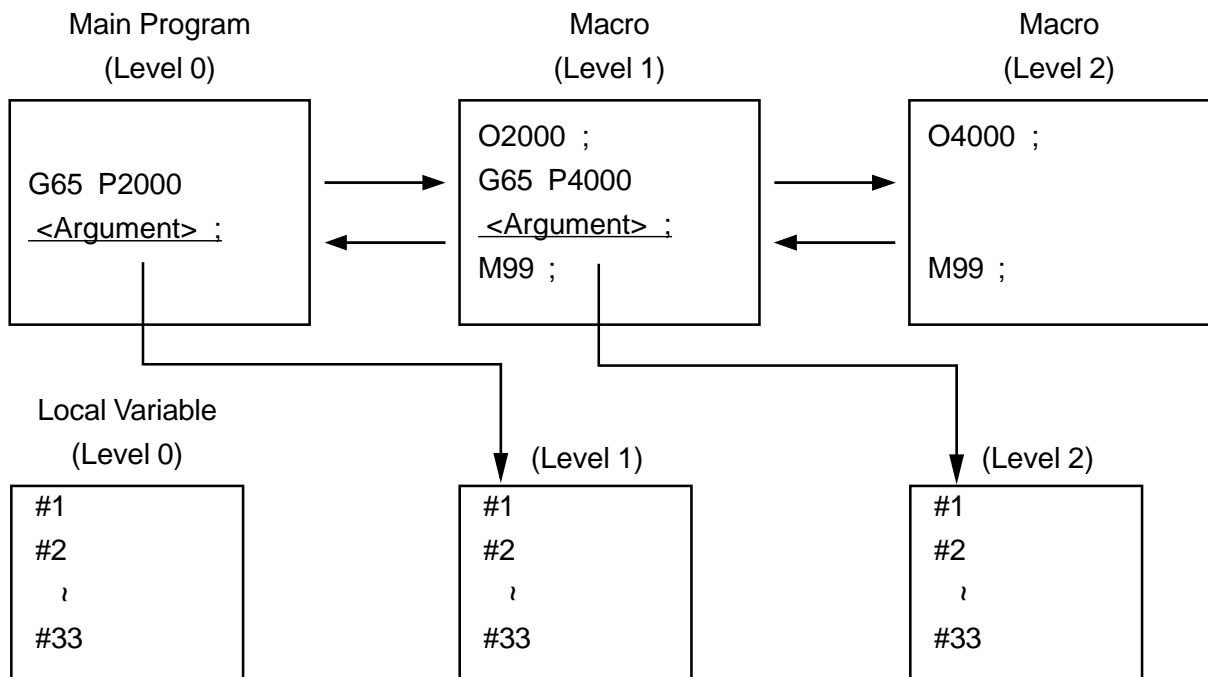
(1) If the macro is called, the local variable of the parent program is stored, and that for the child program is newly provided.
The argument can be transferred to this by specifying the argument.

(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.

21.2.4　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 G66 P1000 A1.0 ;	#1 = 0	Argument transfer	
	#1 = 0		
		O1000 ;	#1 = 0
Z1000 ;			
		#1 = #1 + 1 ; M99;	#1 = 2
	#1 = 0		Succeeded
		O1000 ;	#1 = 2
Z2000 ;			
		#1 = #1 + 1 ; M99;	
	#1 = 0		#1 = 3
		Argument transfer	✕Not succeeded
G66 P1000 A1.0 ;	#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 6000

		#5 LVCON				

0 :Succeeds to the local variable by modal call.

1 :Does not succeeds to the local variable by modal call.

21 - 9

21.2.5 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 Miscellaneous 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 Subprogram	○	○	○	×	×	○
T Code Subprogram	○	○	×	○	○	○
S Code Subprogram	○	×	○	○	○	○
In 2nd Miscellaneous Function Subprogram	×	○	○	○	○	○

Since this prohibitive condition is set with the parameter, it can be cancelled with the parameter.

Parameters

	#7	#6	#5	#4	#3	#2	#1	#0	
No.6012									Arbitrary G Code Macro Call
6013									M Code Macro Call
6014									M Code Subprogram Call
6015									T Code Subprogram Call
6016									S Code Subprogram Call
6017									2nd Miscellaneous Function Code Subprogram Call

In G Code Macro
In M Code Macro
In M Code Subprogram
In T Code Subprogram
In S Code Subprogram
In 2nd Miscellaneous Function Code Subprogram

In the parameters above, when each bit is;

0 : Enables a call
1 : Disables a call

21.3 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.

21.3.1 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

21.3.2 Argument Designation II

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

21.3.3 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)	0
A, B, C, U, V, W, X, Y, Z (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 the reference 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 those addresses.
E, F	Conforms to the least input increment for the speed command. (Refer to Subtable b.)
D, G, H, L, M, P, S, T	0

[Subtable a] Decimal Point Position in Minimum Set Unit of Axial Address

	Metric (G21)	Inch (G20)	Rotary Axis
IS-B	3(2)	4(3)	3(2)
IS-C	4(3)	5(4)	4(3)

- The values in parentheses are supplied when IPR is "1".
- All are "0" in case of pocket calculator type decimal point programming.

[Subtable b] Decimal Point Position in Minimum Set Unit of Speed Command

	Metric (G21)	Inch (G20)
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 1 IR4=1 4
Thread cutting (G33)	MS6=0 5 IS7=0 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 programming.

Parameter

No.1003

#7		#1	
IPR		ISC	

ISC = 0 : IS-B
= 1 : IS-C

Parameter

No.1010

	#0
	ROT

ROT = 0 : Linear axis
= 1 : Rotary axis

Parameter

No.3400

	#1	#0
	F61	DPI

DPI = 0 : Conventional decimal point programming
= 1 : Pocket calculator type decimal point pogramming

Parameter

No.3401

	#5	#4	#3	#2	#1	#0
	MS6	IS7	MM1	IM2	MR3	IR4

21.3.4 Cautions

(1) Argument designations I and II can be mixed for use. When the identical variable is doubly specified as an argument, the latter one becomes effective.

(2) For both argument designation I and II specify only the addresses I, J, and K in the alphabetic order.

(3) In the custom macro call commands, specify a call code before all argument designations.

21.4 Variables

If you specify variables instead of assigning direct values to specific addresses in the macro program, you can get the values of the variables as address values by invoking the variables during execution.

21.4.1 Variable Expression

Variables are expressed in variable numbers each preceded by #.

  #i (i=1, 2, 3......)

   #1, #2, #3

Or, they are expressed in <equations>.

  # [<equation>]

  # [100], # [500+1] , # [#20/2]

In the description below, #i can be substituted by #[<equation>].

21.4.2 Variable Types

(1) Local variable (#1 ~ #33)

This is a variable used locally for each macro. That is, each time a macro is invoked, a variable area is maintained independently, a value is stored by argument specification, or this variable enables the results calculated within the macro to be stored.

This local variable is reset each time a macro is invoked and registered by argument specification but it becomes an unspecified variable (null).

It will be reset to <null> when the power is switched on or the operation is reset.

(2) Common variable (#100 - #199, #500 - #999)

This is a variable that can be used in common for all the macros and all the multiplicity macros.

[1] #100 - #199 : Will be reset to < null > when the power is switched on.

[2] #500 - #999 : Will not be reset to < null > when the power is switched on.

(3) System variable

This is a variable whose system application is fixed.

(4) Character string variable

By adding $ at the beginning of an alphameric character string, you can use it as a variable.

Substituting a value for a character string variable is called creation of a variable, which is used in the description of <equations>

hereinafter.

   $STR1 =123.0 ;

   $STR2 = $STR1*2 ;

To replace a numeral following an address by variables, the variables needs to be put in a parenthesis.

   X [$STR1]   Y [$STR2] ;

(Note) Maximum character string useable consists of 22 characters including $.

     The variables are useable common to all macro.

     With resetting, they are all erased.

21.4.3 Variable Citation

Numerals following an address are replaceable by variables.

With <Address> #i or <Address> -#i being commanded, the variable value or the complement is taken as the command value of the address.

When #11 is 20.0, X#11 corresponds to X20.0 command.

(Note 1) *No variable citation can be performed for Address/,O,N,.*

(Note 2) *A variable number cannot be replaced by variables.*

> *##20 .... Wrong*
>
> *#[#20] .... Correct*

(Note 3) *When variables are used as the address data, the variables below the effective digits of the address are rounded.*

(Note 4) *No commanding is possible when the maximum command value is exceeded for an address.*

(Note 5) *Numerals following an address are replaceable by <equation>.*
With (Address> [<equation>] or <address> - [<equation>] being commanded, either the <equation> value or the complement is taken as the command value of the address.

(Note 6) *A constant without a decimal point used in [ ] is regarded to have a decimal point at its end.*
When the minimum set unit is IS-B in metric,

> *X1000 is 1.000mm, whereas*
>
> *X[1000] is 1000mm.*

21.4.4 Undefined Variable

A variable value for a variable not yet defined is called <empty>.

#0 is used as a variable which is constantly <empty>.

A undefined variable is produced in the following cases:

1) Local variable for which argument has not been assigned in macro call command.

2) Common variable on supply of power   #100 ~ #1XX

3) Variable in which <empty> has been loaded in manual operation

(1) Variable Citation

When an undefined variable has been cited, the address itself is also ignored.

When #1 is <empty>, X100 Y#1; is equal to X100;.

The complement of <empty> is <empty> itself.

With #1 is <empty>, X100 Y-#1; is equal to X100;.

(2) Operation

In coupling with an operator, it is treated in the same way as Constant 0.

When #1 is <empty>,

#2=#1;	#2 gets <empty>.
#2=#1+1;	#2 gets 1.
#2=#1*5;	#2 gets 0.
#2=#[#1]	#2 gets <empty>.
#2=#[#1+#1]	#2 gets <empty>.

(3) Comparison Operation

Only when EQ is NE, <empty> and 0 are different.

When #1 is <empty>,

#1EQ#0 is true.

#1EQ0 is false.

#1NE#0 is false.

#1NE0 is true.

#1GE#0 is true.

#1GE0 is true.

21.4.5   Naming Command

A part of a common variable can be provided with a name with max. 12 characters.
With SETVNn (Name 1, Name 2, Name 3, ...) ; being commanded, Variable is provided
with Name 1, Variable n+1 with Name 2, and Variable n+2 with Name 3 ....
Characters used for a name include numerals, alphabets, decimal point,
  +, -, *, /, $, ?, #, [,], !, ,&, and @,.
Common variables which can be named are those within a scope selected optionally by
#500.
These names are not erased even with power turned OFF.

21.5 System Variable

21.5.1 The Interface Input Signal (#1000 - #1031, #1032, #1032 - #1035)

The status of the 32 point input signals used specially for the macro program can be known by reading the interface input signal #1000 - #1031.

System variable	Number of points	Interface input signal		System variable	Number of 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 signal
1.0	Contact closed
0.0	Contact open

The input signals of 32 points can be read at a time by reading #1032 - #1035.

System variable	Number of points	Interface input signal
#1032	32	UI000 ~ UI031
#1033	32	UI100 ~ UI131
#1034	32	UI200 ~ UI231
#1035	32	UI300 ~ UI331

$$\#1032 = \sum_{1=0}^{30} \{\# [1000 + i] \times 2^1 - \#1031 \times 2^{31}\}$$
$$\# [1032 + n] = \sum_{1=0}^{30} \{2^1 \times Vi\} - 2^{31} \times V_{31}$$

However, Vi = 0 when UIni is [0]

Vi = 1 when Uini is [1]

n = 0 ~ 3

21.5.2 The Interface Output Signal (#1100 - #1131, #1132, #1133 - #1135)

On or off signal can be sent to each of the 32 point output signals specially meant for the macro program by substituting values in #1100 - #1131.

System variable	Number of points	Interface input signal		System variable	Number of points	Interface input signal	
#1100	1	2^0	UI000	#1116	1	2^{16}	UI016
#1101	1	2^1	001	#1117	1	2^{17}	017
#1102	1	2^2	002	#1118	1	2^{18}	018
#1103	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 signal
1.0	Contact closed
0.0	Contact open

The input signals of 32 points can be sent at a time by substituting values in #1132 - #1135.

System variable	Number of points	Interface input signal
#1132	32	UI000 ~ UI031
#1133	32	UI100 ~ UI131
#1134	32	UI200 ~ UI231
#1135	32	UI300 ~ UI331

$$\#1132 = \sum_{1=0}^{30} \{\#[1100 + i] \times 2^1 - \#1131 \times 2^{31}\}$$

$$\#[1132 + n] = \sum_{1=0}^{30} \{2^1 \times Vi\} - 2^{31} \times V_{31}$$

However, Vi = 0 when UIni is [0]

        Vi = 1 when Uini is [1]

        n = 0 ~ 3

(Note) *Taken as 0 when < null >.*
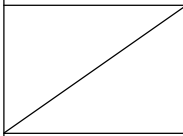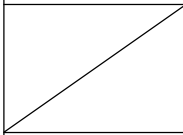
        Taken as 1 when other than < null > or 0.

21.5.3 Tool Offset Amount (#2001 through #2999, #3101 through #3199)

< With Y axis >

Item	Offset No.	Variable No.
1st axis profile (X)	1 2 ⟩ 99	#2001 #2002 ⟩ #2099
2nd axis profile (Y)	1 2 ⟩ 99	#2101 #2102 ⟩ #2199
3rd axis profile (Z)	1 2 ⟩ 99	#2201 #2202 ⟩ #2299
Tool nose radius profile (R)	1 2 ⟩ 99	#2301 #2302 ⟩ #2399
Virtual tool nose number (T)	1 2 ⟩ 99	#2401 #2402 ⟩ #2499
Grooving too width profile (H)	1 2 ⟩ 99	#2501 #2502 ⟩ #2599
1st axis wear amount (U)	1 2 ⟩ 99	#2601 #2602 ⟩ #2699
2nd axis wear amount (V)	1 2 ⟩ 99	#2701 #2702 ⟩ #2799
3rd axis wear amount (W)	1 2 ⟩ 99	#2801 #2802 ⟩ #2899
Tool nose radius wear amount (Q)	1 2 ⟩ 99	#2901 #2902 ⟩ #2999
Grooving too width wear amount (J)	1 2 ⟩ 99	#3101 #3102 ⟩ #3199

< Without Y axis >

Item	Offset No.	Variable No.
1st axis profile (X)	1 2 ⟩ 99	#2001 #2002 ⟩ #2099
2nd axis profile (Z)	1 2 ⟩ 99	#2101 #2102 ⟩ #2199
Tip R shape (R)	1 2 ⟩ 99	#2201 #2202 ⟩ #2299
Virtual tip number (T)	1 2 ⟩ 99	#2301 #2302 ⟩ #2399
	1 2 ⟩ 99	#2401 #2402 ⟩ #2499
Grooving too width profile (H)	1 2 ⟩ 99	#2501 #2502 ⟩ #2599
1st axis wear amount (U)	1 2 ⟩ 99	#2601 #2602 ⟩ #2699
2nd axis wear amount (W)	1 2 ⟩ 99	#2701 #2702 ⟩ #2799
Virtual tip wear (Q)	1 2 ⟩ 99	#2801 #2802 ⟩ #2899
	1 2 ⟩ 99	#2901 #2902 ⟩ #2999
Grooving too width wear amount (J)	1 2 ⟩ 99	#3101 #3102 ⟩ #3199

(Note 1) *The address within the brackets ( ) of each item is the address displayed on the tool compensation screen when the axis number and axis name are set as shown below.*

- *With Y axis .... First axis: X axis, Second axis: Y axis, Third axis: Z axis*
- *No Y axis .... First axis: X axis, Second axis: Z axis*

(Note 2) *Even if the Y-axis is not provided, you can set the parameter to use "Z OFFSET AMOUNT/TOOL NOSE RADIUS/TOOL NOSE POINT" with the same variable number as when the Y-axis is attached.*

Parameter

YST = 0   System variable for the tool offset amount without the Y-axis is Type I
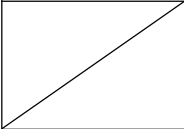
    = 1   System variable for the tool offset amount without the Y-axis is Type II (Same as when the Y-axis is attached)

System variables when the numbe of tool sets is 100 or more are as follows:
(This applies irrespective of availability of Y axis.)

Item	Offset No.	Variable No.	Item	Offset No.	Variable No.
1st axis profile (X)	1 2 ι 200	#20001 #20002 ι #20200	1st axis wear amount (U)	1 2 ι 200	#26001 #26002 ι #26200
2nd axis profile (Y)	1 2 ι 200	#21001 #21002 ι #21200	2nd axis wear amount (V)	1 2 ι 200	#27001 #27002 ι #27200
3rd axis profile (Z)	1 2 ι 200	#22001 #22002 ι #22200	3rd axis wear amount (W)	1 2 ι 200	#28001 #28002 ι #28200
Tool nose R profile (Z)	1 2 ι 200	#23001 #23002 ι #23200	Tool nose R wear amount (Q)	1 2 ι 200	#29001 #29002 ι #29200
Virtual tool nose profile (T)	1 2 ι 200	#24001 #24002 ι #24200			
Grooving tool width profile (H)	1 2 ι 200	#25001 #25002 ι #25200	Grooving tool width wear amount (J)	1 2 ι 200	#31001 #31002 ι #31200

Compensation amount up to offset No.99 can be referred with #2000 ~ #3199.

21.5.4　Alarm (#3000)

You can set the unit in the alarm status to output an alarm when the conditions for the alarm are generated during the program.

#3000 = n (<Alarm message>) ;

(n ≦ 4095)

Specify the alarm number n with the alarm message within 32 characters in "(" , ")"

(Note) Use an alarm number that has not been specified previously for the unit.

If you use alarm number that has been specified already for the unit, the #3000 alarm message will be displayed, and this alarm will remain effective until it is reset.

21.5.5　Clock (#3001, #3001)

You can know the tine on the clock by reading the value of #3000 and #3001. You can preset the time by substituting values to its variables.

Type	System variable	Unit	At power ON	Count condition
Clock 1	#3001	Mill-seconds	0	Always
Clock 2	#3002	Hour	Saved	When STL signal is ON

(Note) The clock accuracy is 10 milliseconds. It overflows at 4294967295 milliseconds and returns to 0.

21.5.6　Single Block Stop Control, Auxiliary Function Completion Signal Hold Control (#3003)

Controls as shown in the table below, are possible by substituting values in #3003.

#3003	Single block stop	Auxiliary function completion signal
0	No control	Wait
1	Control	Wait
2	No control	No wait
3	Control	No wait

(Note) Distribution completion signal cannot be sent if there is no wait for the auxiliary completion signal.

0 clear is possible by resetting #3003.

21.5.7 Rendering Feed Hold, Feed Rate Override, Exact Stop Check Ineffective(#3004)

Controls as shown in the table below are possible by substituting values in #3004.

#3004	Feed hold	Feed hold override	Exact stop check
0	Effective	Effective	Effective
1	Ineffective	Effective	Effective
2	Effective	Ineffective	Effective
3	Ineffective	Ineffective	Effective
4	Effective	Effective	Ineffective
5	Ineffective	Effective	Ineffective
6	Effective	Ineffective	Ineffective
7	Ineffective	Ineffective	Ineffective

(Note) If the feed hold button is pressed during execution of a block for which feed hold is ineffective, then the results will be as described below.

(a) If the feed hold button is pressed continuously, single block stop occurs. However, if the single block stop has been suppressed, no stop occurs.

(b) If the feed hold button is pressed and then released, the feed hold lamp flickers but no stop occurs. Stop occurs at the first block where feed hold became effective.

0 clear is possible by resetting #3004.

21.5.8 Mirror Image Condition (#3007)

The mirror image condition for each axis can be known at each instant by reading #3007. The axis corresponds to each bit as shown in the table below, when the binary digit is dsplayed.

	#7	#6	#5	#4	#3	#2	#1	#0
#3007	8	7	6	5	4	3	2	1axis

Indication for each bit

0 : Mirror image is ineffective
1 : Mirror image is effective.

21.5.9 Operating Status Information (#3009, #3010)

The operating status at each instant can be known by reading #3009, #3010.
The status corresponds to each bit as shown below, when the binary digit is displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
#3009	8	7	6	5	4	3	2	1axis

Indication for each bit

0 : Machine lock of each axis ineffective
1 : Machine lock of each axis effective.

```
       #7    #6    #5    #4    #3    #2    #1    #0
#3010 ┌─────┬─────┬─────┬─────┬─────┬─────┬─────┬─────┐
      │     │     │     │     │     │     │     │     │
      └─────┴─────┴─────┴─────┴─────┴─────┴─────┴─────┘
                                                  └──── Single block
                                            └────────── Program restart
                                      └──────────────── Dry run
                                └──────────────────────── Auxiliary function lock
                          └────────────────────────────── All axes machine
                                                           locked
                    └────────────────────────────────────── Safety guard
```
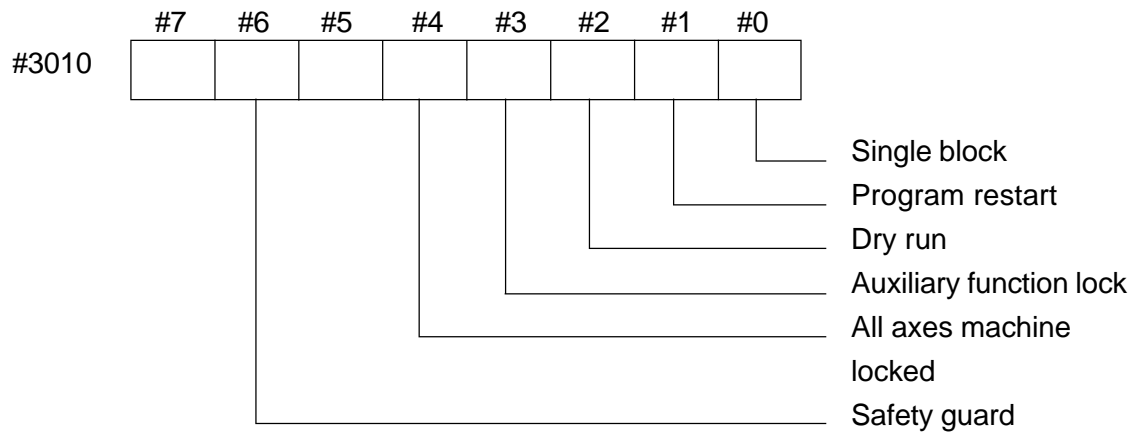
Indication for each bit

0 : Ineffective

1 : Effective

21.5.10 Data and Time (#3011, #3012)

The date and time can be known by reading #3011, #3012.

Type	System variable
Year/month/day	#3011
Hour/minute/second	#3012

(Example) : Indication for 11:07:05 on January 1, 1989

 #3011 = 19890123

 #3012 = 110705

21.5.11 Operating Time Timer (#3022, #3023)

You can read the three types of integral time indicating the operating status of the NC unit.

Type	System variable	Units	Ap power ON	Count condition
Automatic operation time	#3022	Hour	Saved	During automatic operation
Cutting time	#3033	Hours	Saved	During cutting feed

(Note) Accuracy of the clock is in milliseconds.

21.5.12 Modal Data (#4001 ~ #4120, #4201 ~ #4330)

By reading the values of #4001 ~ #4120, the modal data specified up to the immediately prior block car be found.

By reading the values of #4201 ~ #4330, the nodal command for the currently executed block can be found.

Unit will be the one used when specified.

System Variable	Modal information of the immediately prior	System Variable	Modal information of the block currently executed data
#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	Sequential number	#4314	Sequential number
#4115	Program number	#4315	Program number
#4119	S code	#4319	S code
#4120	T code	#4320	T code

(Note) *#4120/#4320 will be as follows depending on a parameter:*

No. 6003 #1 In #4120/#4320, preparatory tools are

0 : Reflected

1 : Not reflected

21.5.13 Positional Information During a Fixed Cycle (#4500 ~ #4504)

By reading #4500~ #4504 during a fixed cycle, the next positional information and machining data can be found.

System Variable	Positional Information and machining data
#4500	R point
#4501	Initial point
#4502	Z point
#4503	Q data
#4504	P data

21.5.14 Position Data (#5001 ~ #5108)

You can know various types of position data by reading the values of #5001 - #5108.

System variable	Position data		Reading during operation
#5001	Last position in the 1st axis block	(ABSIO)	Possible
#5002	Last position in the 2nd axis block	(ABSIO)	Possible
#5003	Last position in the 3rd axis block	(ABSIO)	Possible
≀	≀		
#5008	Last position in the 8th axis block	(ABSIO)	Possible
#5021	Current position in the 1st axis	(ABSMT)	Not possible
#5022	Current position in the 2nd axis	(ABSNT)	Not possible
#5023	Current position in the 3rd axis	(ABSMT)	Not possible
≀	≀		
#5028	Current position in the 8th axis	(ABSMT)	Not possible
#5041	Current position in the 1st axis	(ABSOT)	Not possible
#5042	Current position in the 2nd axis	(ABSOT)	Not possible
#5043	Current position in the 3rd axis	(ABSOT)	Not possible
≀	≀		
#5048	Current position in the 8th axis	(ABSOT)	Not possible
#5061	1st axis skip signal position	(ABSKP)	Possible
#5062	2nd axis skip signal position	(ABSKP)	Possible
#5063	3rd axis skip signal position	(ABSKP)	Possible
≀	≀		
#5068	8th axis skip signal position	(ABSKP)	Possible
#5101	Deviation in the 1st axis servo position		Not possible
#5102	Deviation in the 2nd axis servo position		Not possible
#5103	Deviation in the 3rd axis servo position		Not possible
≀	≀		
#5108	Deviation in the 4th axis servo position		Not possible

Name	ABSIO	ABSMT	ABSOT	ABSKP
Meaning	Last position of previous block	Current specified position	Current specified position	Position where the skip signal became ON in the G31 block
Coordinate system	Work coordinate system	Machine coordinate system	Work coordinate system	Work coordinate system
Tool position, tool length Tool system compensation	Not considered Tool tip position	Considered Tool reference point position	Considered Tool reference point position	Considered Tool reference point position

21.5.15 Shift Amount (#5401 - #5411)

Shift amount	Variable number
1st axis work coordinate shift amount	#5401
2nd axis work coordinate shift amount	#5402
3rd axis work coordinate shift amount	#5403
4th axis work coordinate shift amount	#5404
5th axis work coordinate shift amount	#5405
6th axis work coordinate shift amount	#5406
7th axis work coordinate shift amount	#5407
8th axis work coordinate shift amount	#5408
Work length shift amount K	#5411

21.5.16 Lift Management Data (#6001 - #7000)

You can know the tool life management data by reading #6001 - #7000.

Also, you can rewrite the data by substituting values.

	Tool number	Variable	Remarks		
Tool type	1 2 ≀ 199 200	#6001 #6002 ≀ #6199 #6200	No ATC ATC-A 1: Lathe turning 5: Rotation X 2: Rotation X 6: Fixed Z 3: Rotation Z 7: Fixed special 4: Fixed turn 8: Measurement	ATC-C 1: Lathe turning 5: Rotation X 2: Rotation X 6: Fixed Z 3: Rotation Z 4: Fixed turn	ATC-E 1: Lathe turning 2: Rotary tool 3: Rotation Z 4: Fixed rotation
Setting units	1 2 ≀ 199 200	#6201 #6202 ≀ #6399 #6400	0: Minutes 1: Rotation 2: Length 3: Hole		
Set data	1 2 ≀ 199 200	#6401 #6402 ≀ #6599 #6600			
Used data	1 2 ≀ 199 200	#6601 #6602 ≀ #6799 #6800			
Status data	1 2 ≀ 199 200	#6801 #6802 ≀ #6999 #7000	0: Unused 1: Used 2: Monitoring used 3: 4: Life 5: Wear life 6: Skip 7: 8: Tip measurement ng 9: Work measurement ng 10: Overload fault 11: No load fault 12. External fault		

21.5.17 Axis Names (#3041 to #3048)

Each axis name can be learned by reading #3041 to #3048.

System Variable	Description	Interface Input Signal		
#3041	1st axis name	88 : X	67 : C	98 : ZB
#3042	2nd axis name	89 : Y	85 : U	99 : ZC
#3043	3rd axis name	90 : Z	86 : V	
ⅼ	ⅼ	65 : A	87 : W	
#3043	8th axis name	66 : B	97 : CB	

21.5.18 Axis Numbers (#3061 to #3072)

Each axis number can be learned by reading #3061 to #3072.

System Variable	Description
#3061	X-axis number
#3062	Y-axis number
#3063	Z-axis number
#3064	A-axis number
#3065	Baxis number
#3066	C-axis number
#3067	U-axis number
#3068	V-axis number
#3069	W-axis number
#3070	CB-axis number
#3071	ZB-axis number
#3072	ZC-axis number

21.5.19 Double Speed Cutting Software Spindle Speed Change Rate Command Value (#32001 to #32200)

The double speed cutting software's spindle speed change rate command value corresponding to the tool number can be learned by reading #32001 to #32200.

System Variable	Tool Number	Description
#32001	1	Spindle speed change rate command value (50 to 200)
#32002	2	
ⅼ	ⅼ	
#32199	199	
#32200	200	

21.5.20 Work Counter (#3901 to #3903)

The work counter's value can be learned by reading #3901 to #3903.

System Variable	Description
#3901	Machined count
#3902	Set count
#3903	Total count

21.5.21 Super HiCELL Information (#3081 to #3084)

(1) Tool reference shift amount (#3081 to #3083)

The tool reference shift amount for each axis can be read by reading #3081 to #3083.

System Variable	Description
#3081	1st axis tool reference shift amount
#3082	2nd axis tool reference shift amount
#3083	3rd axis tool reference shift amount

(2) Turret head indexing angle (#3084)

The turret head indexing angle can be read by reading #3084. A data range is from 0 to 180 degrees.

21.5.22 NC Chucking Information (#8900 to #8901)

(1) Angle between the Cs-axis motor zero point and pin-in position (#8900)

The angle between the Cs-axis motor zero point and pin-in position can be read by reading #8900. The least input increment for data is 0.001 degree.

This data refers to the NC chuck system parameter No. 80.

(2) Angle between the jaw vertical position and Cs-axis motor zero point (#8901)

The angle between the jaw vertical position and Cs-axis motor zero point can be read by reading #8901. The least input increment for data is 0.001 degree.

This data refers to the NC chuck system parameter No. 81.

21.5.23 Scheduler Information

The scheduler information can be learned by reading the following system variables. The information can be rewritten by substituting a numeral value as to Program No., Set Value, Use Value, Setup Stop, and Status.

Description	Scheduler No.	System Variable	Remarks
Executed Program No.		#8000	1~99999999
Running Program No.		#8001	1~99999999
Discharge Counter		#8002	0~65535
Program No.	1 2 ι 30	#8101 #8102 ι #8130	1~99999999
Set Value (Quantity)	1 2 ι 30	#8201 #8202 ι #8230	0~65535
Usage (Quantity)	1 2 ι 30	#8301 #8302 ι #8330	0~65535
Setup Stop	1 2 ι 30	#8401 #8402 ι #8430	None
Status	1 2 ι 30	#8501 #8502 ι #8530	0: Unmachined 1: Machining 2: Machining Complete

21.6   Expression and Computation

The expression refers to a general numerical expression where constants and variables are combined by operators, or simple numerical values or variables.

In the following description, the constants may be used instead of #i and #j.

21.6.1   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)

21.6.2   Multiplication Type Computation

#i*#j .... Multiplication

#i/#j .... Division

#iAND#j .... AND (for each bit of 32 bits)

#iMOD#j .... Remainder

21.6.3   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

#iNE#j .... Judgment of #i ≠ #i

#iGT#j .... Judgment of #i > #j

#iGE#j .... Judgment of #i ≧ #j

#iLT#j .... Judgment of #i < #j

#iLE#j .... Judgment of #i ≦ #j

21.6.4 Function

SIN	[#i]	sine (unit in degree)
COS	[#i]	cosine (unit in degree)
TAN	[#i]	tangent (unit in degree)
ASIN	[#i]	inverse sine (unit in degree)
ACOS	[#i]	inverse cosine (unit in degree)
ATAN	[#i]/[#j]	inverse tangent (unit in degree)
ABS	[#i]	absolute value
SQRT	[#i]	square root
EXF	[#i]	exponent at the base e
LN	[#i]	natural logarithm
ROUND	[#i]	rounded integer
FIX	[#i]	omit decimals
FUP	[#i]	raise decimals to a unit
BIN	[#i]	convert from BCD
BCD	[#i]	convert to BCD
ADF	[#i]	add a decimal point
SPA	[#i]	a tool number to be called next
SPB	[#i]	a tool number currently used
SPC	[#i]	a tool condition during SPC command
TLT	[#i]	a tool number in a turret

(1) About ADP function

By executing ADP [#n] (n=1 ~ 33), a macro program can add a decimal point to a parameter passed without any decimal point. At the custom macro side called by

   G65 Pnnnn X10 ;

the value of ADP[I/24] will be equal to the parameter with a decimal point at the end, namely 10. which is used when unit setting is not desired to be conscious of at the macro program side.

When a number is substituted for a local variable at the custom macro, ADP function will not work on the variable subsequently.

```
G65 P100 X10 Y20        ; (metric)
   └──▶  O100 ;
          #1=#24           ;  #1 is 0.01
          #1=ADP [#24]  ;  #1 is 10.
          #1=#25           ;  #1 is 0.02
          #1=ADP [#25]  ;  #1 is 20.
          #24=ADP[#24]  ;  #24 is 10.
          #1=ADP [#24]  ;  #1 is 10.
          #25=0.05         ;  #25 is 0.05
          #1=ADP [(#25]  ;  #1 is 0.05
```

ADP function can be inhibited by a parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
PRA6000				CAV				

0 : ADP function is valid
1 : ADP function is invalid
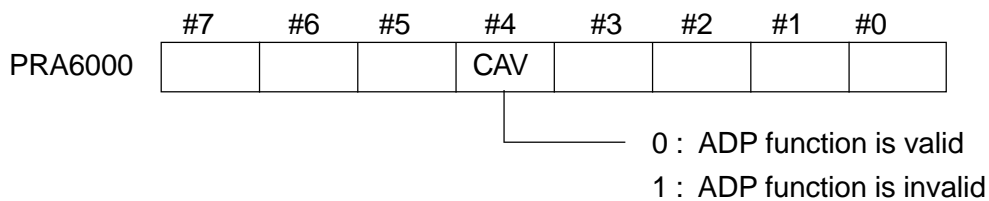
In macro at G65 P___ ___ X10 ;

	CVA=0	CVA=1
#24	0.01	0.01
ADP[#24]	10.0	0.01

(Note) *If ADP function is used for <NULL>, <NULL> will be returned.*

(2) Functions corresponding to tool life control functions
When a tool ends its life while tool life control functions are utilized, the tool number specified by a program becomes inconsistent with the number for the tool actually used. In such a case, by using these functions, a tool number actual called or used can be found out.

[1] SPA [x]
      x : a tool number (the last four digits)
    When a tool number x is called by T command, a tool number to be actually called will be returned.

[2] SPB [x]

    x : a tool number (the last four digits)

A tool number actually called by T command will be returned.

<Example>

<Program>	<Life control setting>					
01000 ;	Basic tool				Preparatory tool	
#1=SPA[0100] ;	0100				0200	
T0100 ;						
M12 ;		Use	Set	Unit		State
#2=SPB[0100] ;	T01	9	10	number of times		in use
M99 ;	T02	0	10	number of times		

<Result>

	The 2st	The 2nd
#1	100,000	200,000
#2	100,000	200,000

[3] About SPC function

    SPC [T___ ___ ___ ___ T___ ___ ___ ___ T___ ___ ___ ___] ;

T___ ___ ___ ___ : an arbitary tool number, up to 10 pcs

Upon the execution, the state of a tool with a number specified (when it is a preparatory tool under the tool life control function, the state of a tool to be actually called) will be saved. By SPC function, existence of unused tools can be found through the state of a tool at that time.

SPC[0]   If there is any unused tool among tool numbers specified by SPG command, 1 will be returned. If not, 0 will be returned.

SPC[x]   x : any arbitary tool number among tool numbers specified by SPG command

          If it is an unused tool when SPG command is executed, 1 will be returned. If not, 0 will be returned.

SPC[x]   will return 0 if other tool numbers than specified by SPG command are found.

(3) TLT[x]

x : a turret number

It reads a tool layout information and returns tool numbers within a turret plane.

21.6.5 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.

```
#iEQ#j+#k*SIN[#1]
            └────┘
              ①
        └──────┘
            ②
      └──────┘
          ③
    └──────┘
        ④
```
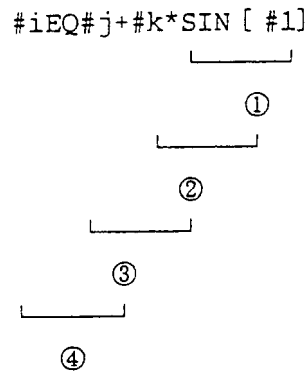
21.6.6 Change in Operation Order by [ ]

The part whose operation wants to be preceded in order is put in a [ ].

```
SIN[ [ [#i+#j] *#k+#1] *#m]
          └───┘
            ①
        └──────┘
            ②
            └──────┘
              ③
              └──────────┘
                  ④
    └──────────────────┘
            ⑤
```
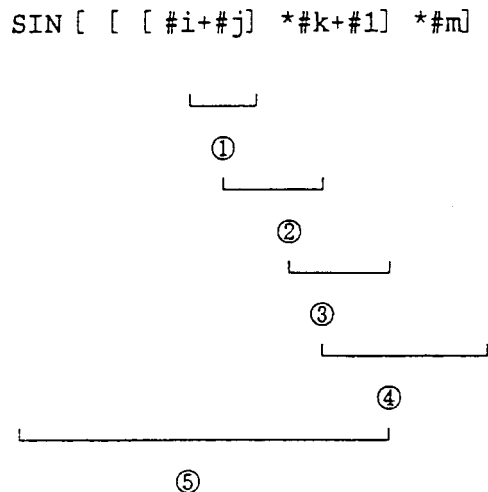
21.6.7 Substitution Command

By specifying #1=<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;

21.7　Control Command

21. 7 .1　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.

```
O1000;
   N1 ;                [2]
    ⁊
   N100;
    ⁊
   GOTOX X X ;
    ⁊
   N200;               [1]
    ⁊
   N300;
    ⁊
   % (EOR)
```
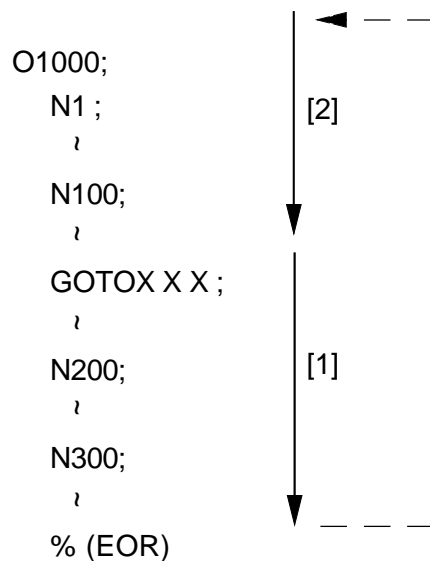
(Note 1)　On the program, backward branching takes a longer executing time than forward branching.

(Note 2)　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.

21.7.2 IF Command

IF (expression) THEN ... ELSE ...

Execute with a condition or branch off according to the evaluation of the conditional expression.

(1) One line format

IF <expression> THEN [1] ELSE [2]
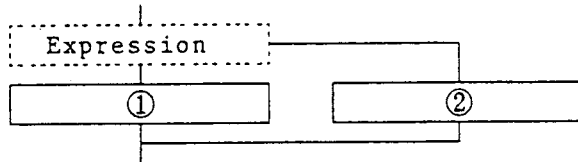
When the value of the <expression> is true (not 0), [1] will be executed, if false (0), then true will be executed. There is no restriction on the command to be executed. Commands after THEN or ELSE can be omitted.

IF [#1LT0] THEN #1 =-#1 ;

IF [#1EQ4] GOTO4 ;

(2) Block format

IF<expression 1> THEN ;

   [1]

ELSE IF<expression 2> THEN ;

   [2]

ELSE ;

   [3]

ENDIF ;
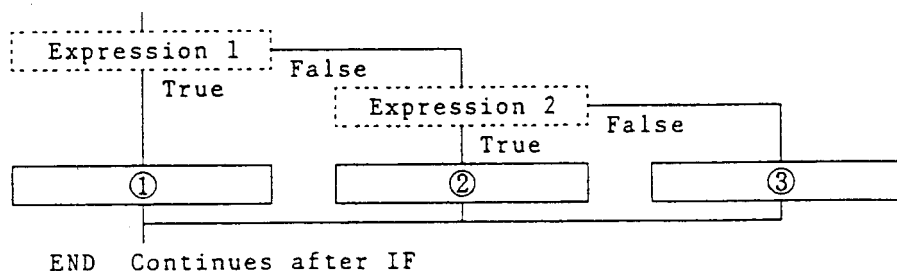
If the value of <expression 1> is true, then (1) is executed and the control branches off to the block after ENDIF. If the value is false (0), the control branches off to the next ELSE statement, and if an IF command exists after this, then the next block will be executed according to the conditional expression or it branches off to ELSE. Commands after ELSE and until ENDIF can be omitted.

21.7.3 Repeat Command

DO m ;   (m=1, 2, or 3)

END m ;

By specifying as above, the blocks between DOm and ENDm are repeatedly executed.

(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.
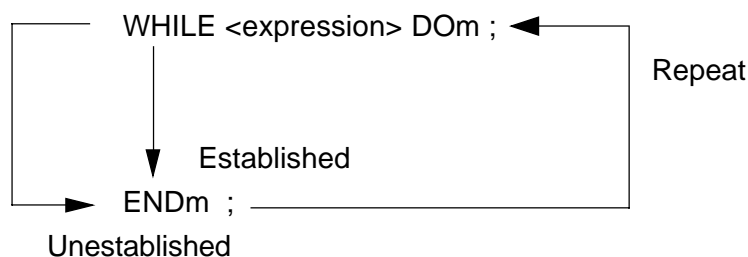
(2) Cautions for the repeat command

    (a) Specify DOm before ENDm.

    (b) For 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) Muptiplicity of the DO loop is allowed up to 3 levels.

(3) Program resulting in error

    (a) DO1 ;

        DO1 ;   ◄————————   There is no corresponding END.

        END1 ;

    (b) DO1 ;

        END1 ;   ◄————————   There is no corresponding DO.

        END1 ;

    (c) DC1 ;

        DO2 ;

        END1 ;   The loop cannot intersect.

        END2 ;

    (d) DO1 ;

        N100 ; ◄

        END1 ;

        GOTO100 ;

21.8 External Output Command

You can output messages or NC data to external devices through the R5232C data input interface. If the external device is a printer, you can print out data.

Use commands in the sequence given below.

[1] Open command: POPEN
                    Linking with I/O device

[2] Data output command: PRINT, BPRNT, DPRNT
                    Output data

[3] Close command: PCLOS
                    Cancel link with I/O device

21.8.1 Open Command (POPEN)

After the RS232 port is opened, the control code of DC2 is output from the NC side.

21.8.2 Data Output Command 1 (PRINT)

PRINT (ABC...) P<expression> D<expression>
                                      — Numeric data
                            — Specify number of digits
                  — Character data

With PRINT, the characters and numerical values by variable digit are output by ISO, EIA, and ASCII codes.

Which code to be used is set with Parameter No.100, #2(ISP)/#3 (EIA).

(1) The characters within the range enclosed by '(' and ')' are output as they are. The following alphanumeric characters can be output:

'+', '-', '*', '/', '=', '&', '#', '[', ']', '$', '?', ',' '@', ' ' , '{', '}', Character data can be omitted.

(2) The number of digits before the decimal point and after the decimal point of the numeric value to be output, are specified in the address P. The method is to specify the number of digits after the decimal point at the 'unit' position in the P data, and the number of digits before the decimal point at the 'ten' position. Data in the 'hundred' position will be ignored. If the specification is P13, then the number of digits after the decimal point is 3, and the number of digits before the decimal point is 1.

The specification of number of digits is effective until a new P specification is made in the same block.

PRINT   P12 D#1 D#2 P23 D#3
            P12 effective   P23 effective

Specification of number of digits may be omitted. In this case, the number of digits will be as follows:

		Number of digits before the decimal point	Number of digits after the decimal point
Metric	(IS-B)	4	3
Inch	(IS-B)	3	4
Metric	(IS-C)	3	4
Inch	(IS-C)	2	5

(3) The <expression> to be output is specified after D. The value of the <expression> will be output for each digit.

Suppose the number data consists of a maximum of 8 lines. When the first digit is 0, with Parameter PRT (No.6000, #7)= 1, no output is made and, with PRT=0, the space code is output. If the number of digits after the decimal point is any value other than 0, the numeric value after the decimal point will be output always. If the number of digits after the decimal point is 0, the decimal point will not be output. If the code for the <expression> is correct, and parameter PRT = 0, then space code will be output but, if PRT = 1, there will be no output.

If the value of the <expression> is <null>, and parameter PRT = 0, then the space code will be output for the number of significant digits, but if PRT = 1, there will be no output.

PRINT P43 D#0;
If PRT = 0,

```
   □   □   □   □   □   □   □   □   □      is output (□ : Space)
Code     ‾‾‾‾‾‾‾‾‾‾       ‾‾‾‾‾‾‾‾
         After decimal    Before decimal
         point            point
```

If the <expression> overflows, ' * ' will be output for the number of significant digits.PRINT P53 D#1 ;
When #1 overflows on the positive side,

```
   ＋ ＊ ＊ ＊ ＊ ＊ ＊ ＊   is output
Code ‾‾‾‾‾   ‾‾‾‾‾‾‾
       |         |
       |       Before decimal point
   After decimal point
```

When #1 overflows on the negative side,

```
   － ＊ ＊ ＊ ＊ ＊ ＊ ＊   is output
```

If there is an underflow of the <expression>, it will be taken as 0.

(4) Following command data output, the EOB code is output. The punch format at this stage is determined according to Parameter 100, #4 (ICR1)/#5 (ICR2).

(5) If there is a ' , ' immediately after EOB, EOB code will not be output.

PRINT (ABCDEFG), ;

EOB will not be output. However, if the ' , ' occurs within the brackets
'(   )' then it will be output.

PRINT (ABCDEFG,);

EOB will be output.

This function does not exist in BPRNT and DPRNT.

(example) PRINT P34 (X=) D#1 (Y=) D#2 P25 (Z=) D#3;

        #1 = 738.196451

        #2 = -48.8

        #3 = 338.417

[1] If parameter PRT = 0

```
D8   BD   A0   B7   33   B8   2E   B1   39   36   35
      |     |_____|                           □738.1965
      |_____ X=
59   BD   2D   A0   B4   B8   2E   B8   30   30   30
      |     |_____|                                 -□48.8000
      |_____ Y=
5A   BD   2B   AA   AA   AA   AA   AA   AA   AA   AA   0A
      |     |_____ LF
      |                                                        +********
      |_____ Z=
```

Output will be as shown above.

[2] If parameter PRT = 1

```
D8   BD   B7   33   B8   2B   B1   39   36   35
      |     |_____|                        738.1965
      |_____ X=
59   BD   2D   B4   2E   B8   30   30   30
      |     |_____|                             -48.8000
      |_____ Y=
5A   BD   2B   AA   AA   AA   AA   AA   AA   AA   AA   0A
      |     |_____ LF
      |                                                +  ********
      |_____ Z=
```
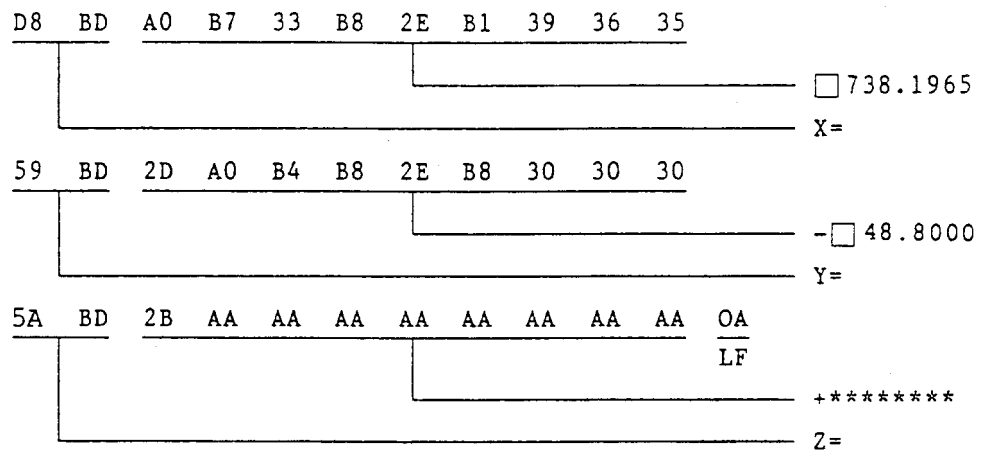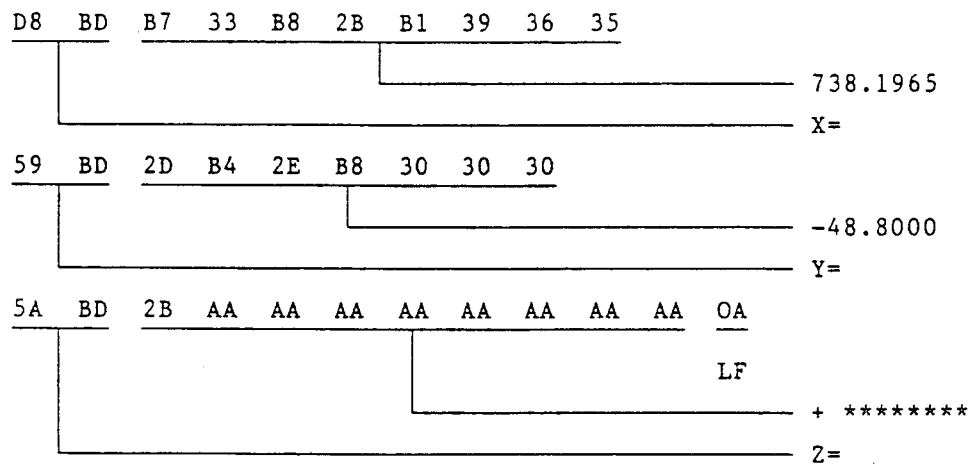
Output will be as shown above.

21.8.3   Data Output Command 2 (BPRNT)

BPRNT [A #1 [3] ......] ;
                              Number of significant digits after the decimal point
                              Variable
                              Character
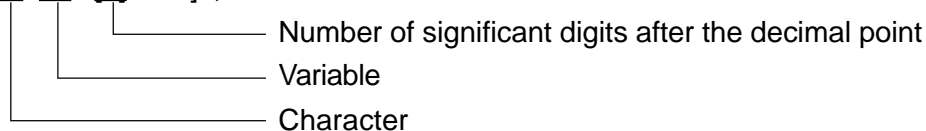
Binary output of variable values and character output occurs in BPRNT.

(1) In character output, the specified character is output as it is.
     The characters that can be specified are:
     • Alphabet (A to Z)
     • Number
     • Special characters (*, /, +, -).
     However, * will be output by space code.
(2) The number of significant digits after the decimal point is specified by enclosing it in
     brackets after the variable specification. The value of the variable will be treated as
     2-word data (32 bit) considering the lines after the decimal point, arid output as upper
     and binary as it is.
(3) Following command data output, the EOB code is output. The punch format at this
     stage is determined according to Parameter 100, #4 (ICR1)/#5 (ICR2).
(4) Same as the explanations in section (5) of PRINT specification.
     (example)   BPRINT [C**X#100 [3] Y#10 [3] M#10 [0]] ;
                 #100 = 0.40956
                 #101 = -1638.4
                 #10 = 12.34

```
43   A0   A0   D8   00   00   01   9A   57   FF   F7   00   00
                                                        -1638400
                                                        Y
                                                        410
                                                        ◁□□ X

AD   00   00   00   0C   0A
                    LF
                                                        12
                                                        M
```
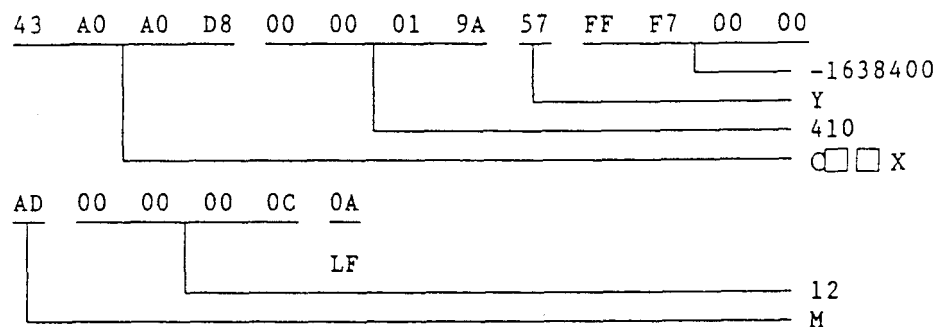
Output will be as shown above.

21.8.4   Data Output Command 3 (DPRNT)

DPRNT [B #2 [43] ......]

- Number of digits after the decimal point
- Number of digits before the decimal point
- Variable
- Character

In DPRNT, the characters and numerical values by variable digit are output by ISO, EIA, and ASCII codes.

Which code to be used is set with Parameter No.100, #2(ISP)/#3(EIA).

(1) Explanations are the same as (1), (2), (3) of BPRNT.

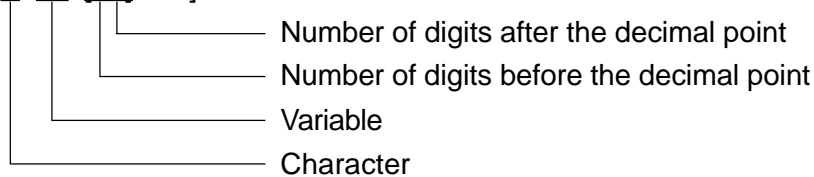(2) The variable, number is specified after # during output of vales of variable. Also, the number of digits before and after the decimal point are specified by enclosing them in brackets. The value of the variable will be output by number of digits specified, from the first digit for all digits in ISO or ASCII code. The same is the case for the decimal point.

Assume that the value of the variable consists of a numeric value of 8 digits maximum. If the first digit is 0, and if parameter PRT(No.6000, #7)=1, there is no output; if PRT=0 space code is output. If the number of digits after the decimal point is 0, the numeric value after the decimal point will be output always. If the number of digits after the decimal point is a value other than 0, the decimal point will not be output.

If the code of the variable valtre is correct and if parameter PRT = 0, space code is output but if PRT = 1, there is no output.If the value of the variable is <null>, it will be treated as 0.

(example)   DPRNT [C#[53]E#2[53]N#3[20]]

#1=299.79246

#2=-16 .02

#3=602 .205

Example)   Output will be as shown above.

[1]   If parameter PRT = 0

```
C3   A0   A0   A0   B2   39   39   2E   B7   39   B2
```
- □□□ 299.792
- C

```
C5   2D   A0   A0   A0   B1   36   2E   30   B2   30
```
- -□□□ 16.020
- E

```
4A   A0   30   B2   0A
```
- LF
- □ 02
- N

Output will be as shown above.

[2] If parameter PRT = 1

```
C3    2D   39   39   2E   B7   37   B2
 |         └──────────┐
 |                    └─────────────────────────── 299.792
 └──────────────────────────────────────────── C

C5    2D   B1   36   2E   30   B2   30
 |         └──────────┐
 |                    └─────────────────────────── -16.020
 └──────────────────────────────────────────── E

4A    30   B2   0A
 |         |         LF
 |         └─────────────────────────────────── 02
 └──────────────────────────────────────────── N
```
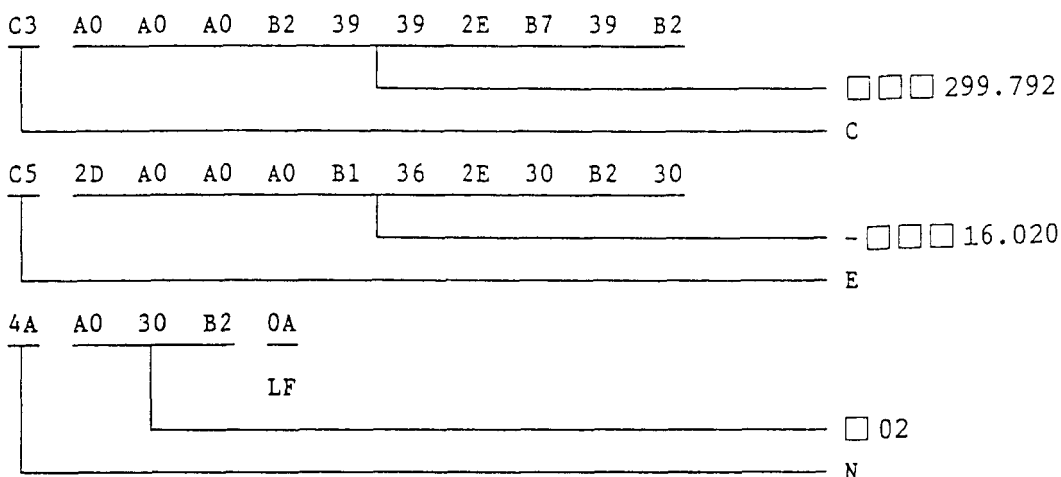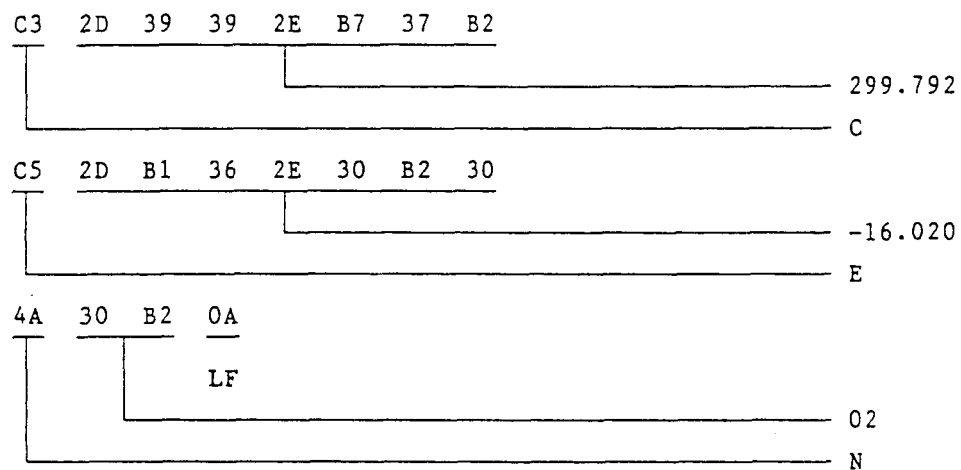
Output will be as shown above.

21.8.5   Close Command (PCLOS)

After the control code of DC4 is output from the NC side, the RS232 port is closed.
The "close" command is executed after all the data having been set in the NC internal
buffer by "print" command are output outside.

21.8.6 Resrtictions
(1) The timing fo.r output of data is the timing for execution of the above mentioned
     command in the program.
(2) The number of characters that can be output by the command in one program is 256
     bytes.

21.8.7   Associated Parameters

No.100, #3   The punch code is:
                    = 0 ISO code
                    = 1 EIA code
No.100, #2   The ISO code is:
                    = 0 With the parity bit (TH check ON)
                    = 1 Without the parity bit (ASCII code: TH check OFF)
No.100, #4 to #5   "EOB" Punch Format for ISO Code:

#5	#4	"EOB" Punch Format
0	0	LF - CR - CR
0	1	LF-CR
1	0	LF
1	1	

No.135	Equipment number for the custom macro external output
No.104 to 110	Setting of the top bits of Equipment No's 1 to 7
No.112 to 115	Setting of DC1to DC4 codes
No.116 to 122	Setting of the baud rates of Equipment No's 1 to 7
No.116 to 122	Setting of the port No's of Equipment No's 1 to 7

21.8.8   Associated Alarms

No.560	External output open error
(#001)	Without "open" command (POPEN) being executed, "output" command has been given.
(#002)	Setting of an equipment number is erroneous.
(#003)	Setting of a port number is erroneous.
(#004)	A port already in use has been assigned.
(#005)	"Open" has been failed. (The cable has not been connected.)
No.561	External output communication error
(#010)	DSR signals are OFF.
(#011)	Overrun error has occurred.
(#012)	Parity error has occurred.
(#013)	Framing error has occurred.

22. COMPATIBILITY WITH SEICOS-LII/LIII

[About S-LII]

SEICOS-Σ10L/Σ16T/Σ18T/Σ21L retains higher compatibility with S-LII, which is provided with a number of new functions.

Please note, however, that compatibility in commanding method has not been given to some of the functions concerning creation of an NC program.

(No functional problem, however, exists.)

Although the differences are outlined below, please see each relevant items in this manual for the details.

(1) Drilling Fixed Cycle

Through use of general-purpose functions, G codes are totally changed.

Through parameter setting, however, the fixed cycle same as S-LII (G code/operation) can also be executed. (See 22.1.)

(2) Work Length Changing

Changing of a G code. (G154 → G54)

In addition to absolute commanding, incremental commanding is made available.

(3) Shifting of threading starting angle (Multi-threading)

Changing of a command address of the shift angle. (B → Q)

(4) Polar Coordinate Interpolation

Changing a G code for cancellation of Polar Coordinate Interpolation mode. (G122 → G120)

No change takes place concerning G code for Polar Coordinate Interpolation mode.

[About S-LIII]

The SEICOS-Σ10L/Σ16T/Σ18T/Σ21L has almost perfect compatibility with the S-LIII.

22.1 Drilling Fixes Cycle (G80-G89)

Through parameter setting, SEICOS-LII G code is made useable.

22.1.1 G Codes

G80 : Cancel drilling fixed cycle
G83 : Drilling cycle, deep drilling cycle, high speed deep drilling cycle
G84 : Tapping cycle
G85 : Boring cycle
G87 : Drilling cycle, deep drilling cycle, high speed deep drilling cycle
G88 : Tapping cycle
G89 : Boring cycle

(Note 1) The drilling axis for G83 -G85 is the Z axis, for G87 - G89 is the X axis, and is fixed regardless of the plane selected (G17 - G19).

(Note 2) G83/G87, G84/G88, G85/G89 differ depending on whether the drilling axis is the Z axis or the X axis. Explanations about G83, G84, and G85 are given below, and explanations for G87, G88, G89, are shown in brackets [ ].

22.1.2 Command Format

$$\begin{bmatrix} G198 \\ G199 \end{bmatrix} \ G\_\_\_ \ X\_\_\_ \ [Z] \_\_\_ \ C\_\_\_ \ Z \ [X] \_\_\_ \ R\_\_\_ \ Q\_\_\_ \ P\_\_\_ \ L\_\_\_ \ F\_\_\_ \ ;$$

G198/G199 : Return point
G___ : Fixed cycle G code (G83 - G89)
X[Z] ___/C___ : Drilling position <Note 2>
Z[X] ___ : Position of point Z <Note 2>
R___ : Position of point R
Q___ : Cutting depth of G83 [G87]
P___ : Dwell time
L___ : Number of repetitions
F___ : Cutting feed rate

(Note 1) When the number of repetitions is omitted, L will be taken as 1.
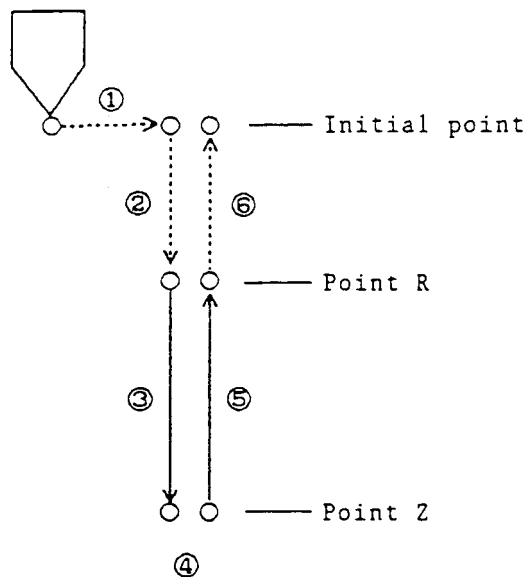If you specify L = 0, the tool will move to the drilling position, but it will not perform drilling.

(Note 2) Specify the position of point Z by the axis address of the drilling axis.

(Note 3) Point R, point Z, P and Q are modal during the fixed cycle.

22.1.3 Machining Cycle

The machining cycles of the fixed cycle generally consists of the operations from [1] to [6] given below.

[1] Positioning at the drilling position

[2] Rapid traverse up to point R

[3] Hole drilling up to point Z (feed rate Z)

[4] Operation at point Z

[5] Return up to point R

[6] Rapid traverse up to the initial point

```
In the figure, --> :  indicates rapid traverse
                →  :  indicates cutting feed
```

(Note) Stop will not occur in the single block between operation [3] to operation [5].

22.1.4 Return Point

The return point of the fixed cycle is specified by the G code give below.
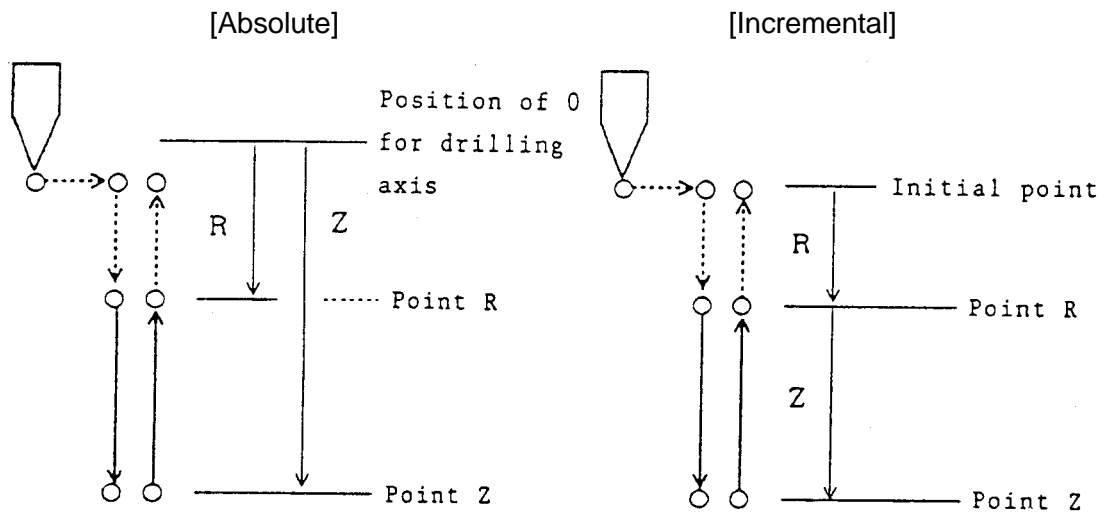
G198 : Initial point level return

G199 : Point R level return

(Note) Initial point is the position of the drilling axis when the status changes from the fixed cycle cancel status to the fixed cycle mode.

22.1.5 "Point R" and "Point Z"

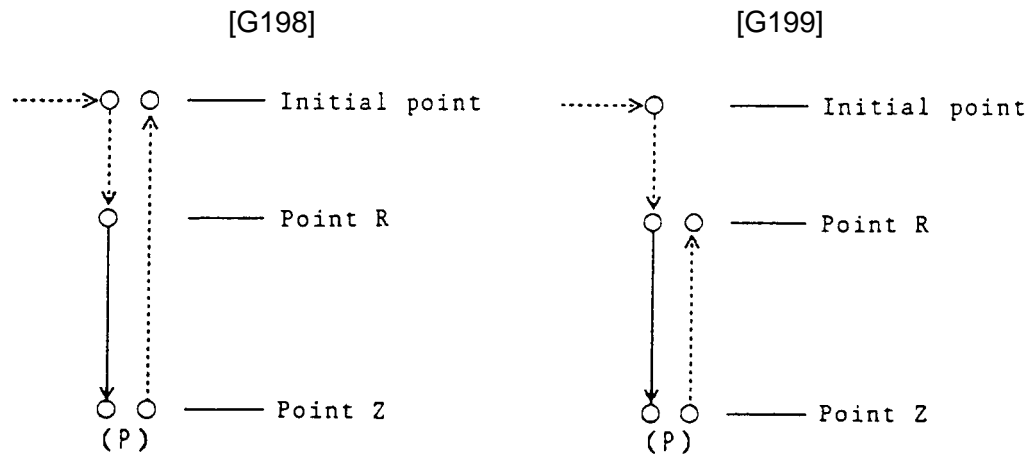Absolute command / incremental command is possible for point Z.
However, the incremental command is permanent for point R.

22.1.6 Explanations for Fixed Cycle Operation

(1) G83 [G87] (drill) for Q = 0

$\begin{bmatrix} \text{G198} \\ \text{G199} \end{bmatrix}$ G83 X [Z] ___ C ___ Z [X] ___ R ___ P ___ L ___ F ___ ;

(P): Dwell

(Note) *Dwell can be made effective in the P command by parameter setting.*

(2) G83 [G87] (High speed deep drilling) for Q not equal to 0

$\begin{bmatrix} \text{G198} \\ \text{G199} \end{bmatrix}$ G83 X [Z]___ C ___ Z [X]___ R ___ Q ___ P ___ L ___ F ___ ;

(Note 1) *Select high speed deep drilling or deep drilling by parameter setting.*
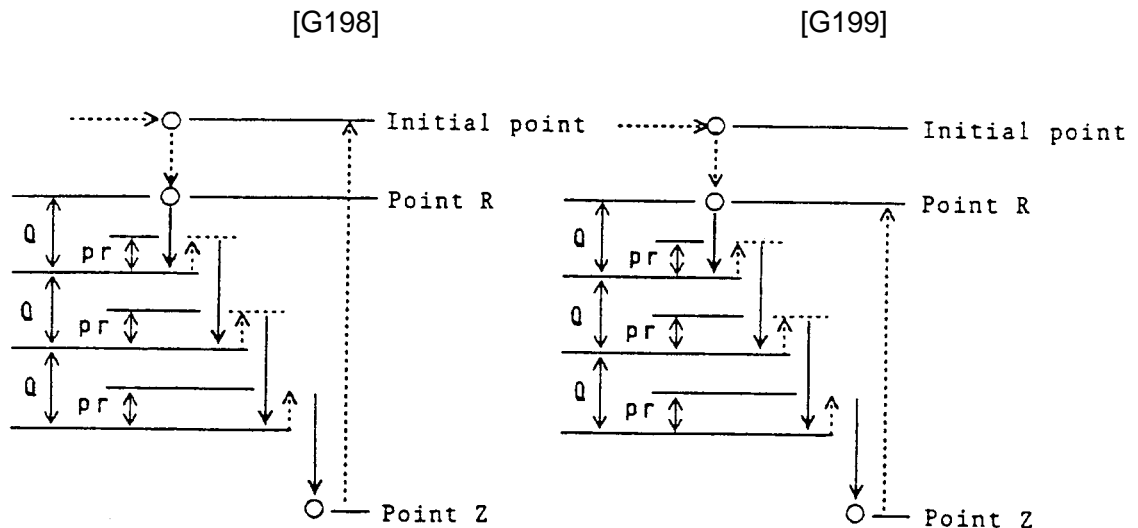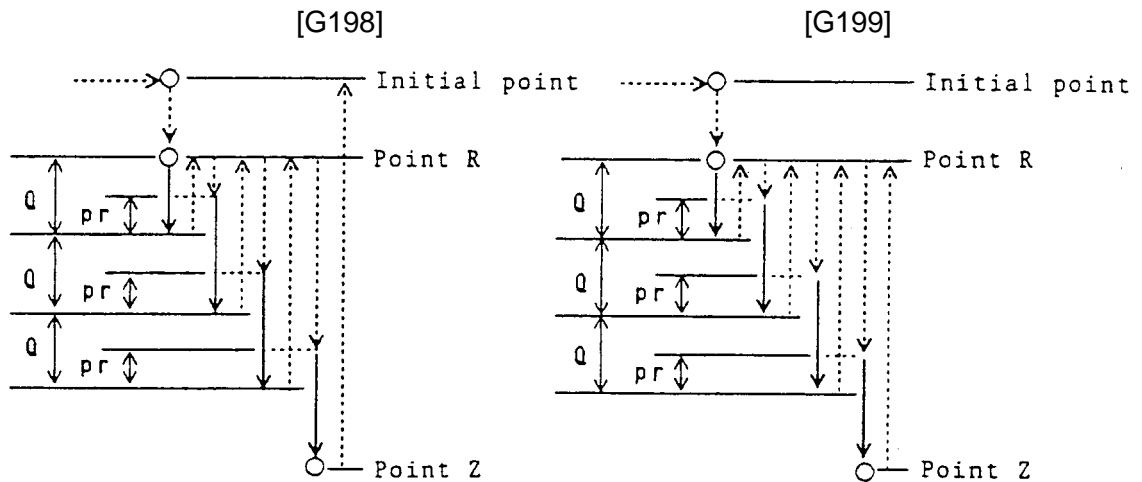*(Note 2)* *Dwell can be made effective in the P command by parameter setting.*

(3) G83 [G87] (Deep drilling) for Q not equal to 0

$$\begin{bmatrix} G198 \\ G199 \end{bmatrix}$$ G83 X [Z]___ C___ Z [X]___ R___ Q___ P___ L___ F___ ;

[G198]   [G199]

- - - ->O ————————∧— Initial point   - - - ->O ———————— Initial point

——————O ∧—∧—∧—∧— Point R   ——————O ∧—∧—∧—∧— Point R
Q p r   Q p r
Q p r   Q p r
Q p r   Q p r

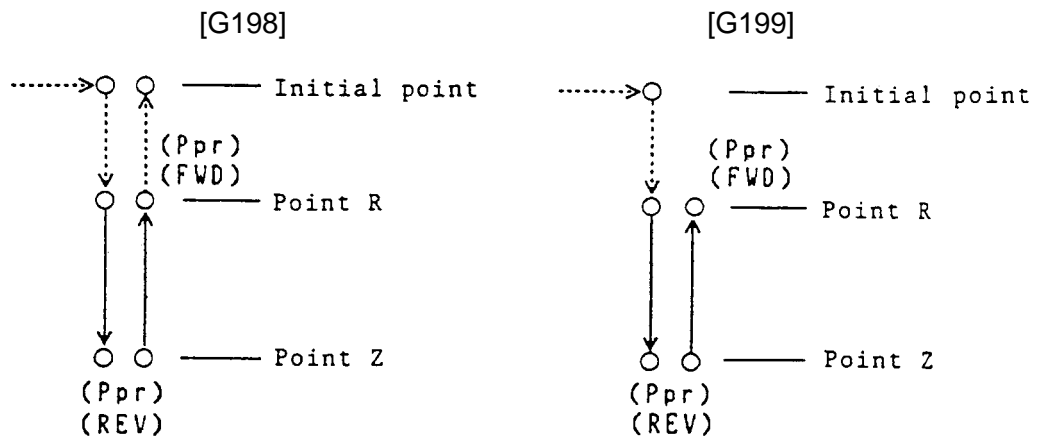   O — Point Z   O — Point Z

(Note 1) *Select high speed deep drilling or deep drilling by parameter setting.*
(Note 2) *Dwell can be made effective in the P command by parameter setting.*

(4) G84 [G88] (Tap)

$$\begin{bmatrix} G198 \\ G199 \end{bmatrix}$$ G84 X [Z]___ C___ Z [X]___ R___ P___ L___ F___ ;

[G198]   [G199]

- - - ->O O ———— Initial point   - - - ->O ———— Initial point
       (P p r)   (P p r)
       (FWD)   (FWD)
       O O ———— Point R   O O ———— Point R

       O O ———— Point Z   O O ———— Point Z
      (P p r)   (P p r)
      (REV)   (REV)
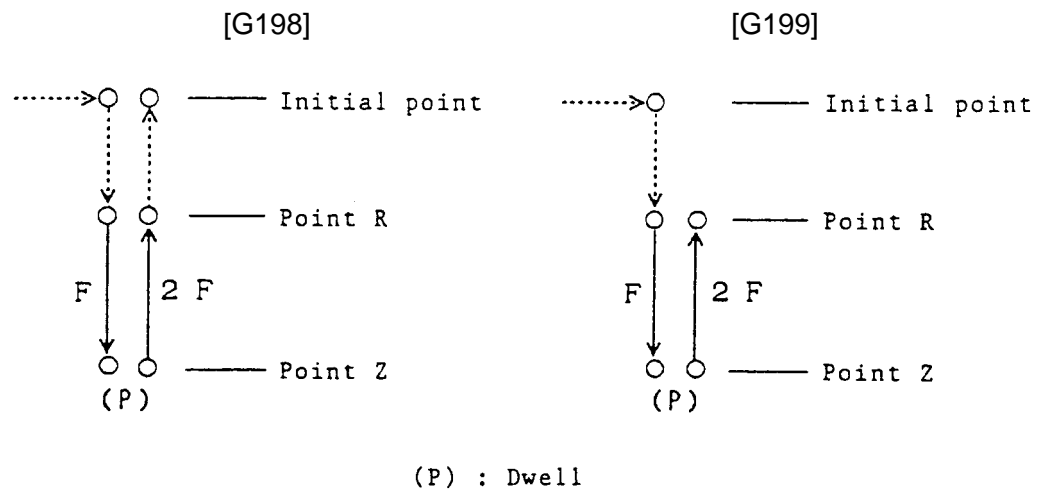
(FWD) : Tool forward rotation
(REV) : Tool reverse rotation
(Ppr) : Dwell (parameter setting)

(Note 1) *Dwell can be made ineffective in the P command by parameter setting.*
(Note 2) *Feed hold and override are ineffective during cutting.*

(5) G85 [G89] (Boring)

$$\begin{bmatrix} G198 \\ G199 \end{bmatrix}$$ G85 X[Z]___ C___ Z[X]___ R___ P___ L___ F___ ;

[G198]   [G199]

(P) : Dwell

(Note) *The return up to point R will be performed at a cutting feed rate that is twice the specified feed rate F.*

22.1.7 Precautions

(1) The tool will stop at the end point of operation [1], [2], [6] when the single block is ON. In this case, the feed hold lamp will become ON at the end point of operation [1] and [2], and at the end point of operation [6] when repetitions remain.
You can initiate single block stop between operations [3] to [5] of cycles other than the tap cycle (G84 [G88]) by parameter setting.

(2) If the [Stop] button has been pressed in operation [3] - [5] of G84 [G88], the feed hold lamp becomes ON immediately, and stops after the operation continues until [6].

(3) The feed rate override will be 100% fixed during operation [3] - [5] of G84 [G88]. You can make dry run effective/ineffective by parameter setting.

(4) Specify Q, P when there is an R or axis command.
In other blocks, P and Q will not be treated as fixed cycle data.
Also, even in blocks where the G code (except G09) of 00 group has been specified, Q and P will not be treated as fixed cycle data.

(5) By specifying the G code (G00, G01, ...) of 01 group or G80, the fixed cycle mode will be canceled.

(6) If you specify M, S in the block of the fixed cycle command, they will be sent at the first operation [1] (positioning of drilling position).
If the number of repetitions (L) has been specified, M and S will be sent only for the first time.

(7) Specify positive values for P, Q, L and F.

(8) Always specify Q when specifying the radius.
Specify R to suit the drilling axis specification.
If radius is specified for drilling axis .... Specify radius for R
If diameter is specified for drilling axis .... Specify diameter for R

22.1.8 Related Parameters

No.5101, #7 = 0   G code of drilling fixed cycle is MIII type

            = 1   G code of drilling fixed cycle is LIII type

No.5101, #6 = 0   For G code of LIII type, G83 is high speed deep drilling

            = 1   For G code of LIII type, G83 is high deep drilling

No.1401, #5 = 0   Dry run is invalid in the thread cutting/tapping command.

            = 1   Dry run is valid in the thread cutting/tapping command.

No.3407, #0 = 0   No single block stop during hole drilling (operation [3] - [5])

            = 1   Stops during hole drilling (operation [3] - [5])

                  (However, tap cycle is excluded.)

No.5100, #3 = 0   When direction of rotation of tool is changed, rotation stop is not output

            = 1   When direction of rotation of tool is changed, rotation stop is output

No.5100, #5 = 0   During G83, dwell addition by P command is ineffective

            = 1   During G83, dwell addition by P command is effective

No.5100, #6 = 0   During G84 [G88], dwell addition by P command is ineffective

            = 1   During G84 [G88], dwell addition by P command is effective

No.5150   Return volume of G83 (high-speed deep drilling)

No.5151   Clearance of G83 (deep drilling)

No.5131   M code of tool forward rotation (M13 for 0 setting)

No.5132   M code for tool reverse rotation (M14 for 0 setting)

No.5133   M code for tool stop (M05 for 0 setting)

No.5134   M code of tool orientation (M15 for 0 setting)

22.1.9 Related Alarm

No. 133   Error in the drilling fixed cycle command.

(#000)   In the first block of the fixed cycle, no Z point (hole bottom position) has been commanded,

23. MISCELLANEOUS

23.1 Preread Stop Command

23.1.1 Preread Stop G-codes

    (1) Group 00 G-codes

        G07 (Virtual axis interpolation)

        G08 (Precedence control)

        G10 (Programmable data setting)

        G11 (Programmable parameter setting mode cancel)

        G28 (Reference point return)

        G30 (2nd/3rd reference point return)

        G301 (Floating reference point return)

        G31 (Skip)

        G53 (Machine coordinate system selection)

        G194 (Machine external measurement compensation)

        G271 (Work coordinate system presetting)

        G921 (Work coordinate system preset)

    (2) Group 04 G-codes (Stored stroke check)

    (3) Group 06 G-codes (Inch/metric conversion)

    (4) Group 12 G-codes (Work length change)

    (5) Group 22 G-codes (Polar coordinate interpolation)

    (6) Group 24 G-codes (Three dimensional coordinate conversion)

    (7) Group 25 G-codes (Front/back machining mode selection)

23.1.2 Preread Stop M-codes

    (1) Special M-codes

        M00 (Program stop)

        M01 (Optional stop)

        M02 (Program end)

        M30 (Program end)

    (2) Preread stop parameter changeover M-codes

        The following M-codes can be designated as preread stop M-codes by parameter setting.

        M12 (Work count)

        M31 (No-workpiece check)

        M32 (Top cut check)

        M33 (Top cut reset)

(3) Parameter specified preread stop M-codes

The M-codes specified by the following parameters can be designated as preread stop M-codes.

Parameters No. 3434 to No. 3441 (Preread stop M-code)

Parameters No. 3442 to No. 3449 (Preread stop M-code group)

(4) Other preread stop M-codes

- Polygon machining start/end M-codes

23.1.3 Preread Stop T-codes

The T-codes other than multiple offset are available as preread stop T-codes.

23.1.4 Associated Parameters

No. 3404, #0 = 0   Disables canned preread stop M-code.

        = 1   Enables canned preread stop M-code.

No. 3434 to 3441   Preread stop M-code (Independent setting)

No. 3442 to 3449   Minimum/maximum value of the preread stop M-code group (Group setting)

No. 8678   M-code to start polygon machining (0 for M156)

No. 8679   M-code to end polygon machining (0 for M157)