TC-32B

NC PROGRAMMING MANUAL

Please read this manual carefully before starting operation.

This manual describes the NC-Programming of the TC-32B.
The tapping centre is able to perform drilling, tapping, and facing.
We shall not bear any responsibility for accidents caused by user's special handling or handling deviating from the generally recognized safe operation.

The relation between the manuals is as follows.

- OPERATION MANUAL

This manual describes the operations of the machine.

- INSTALLATION MANUAL

This manual describes the installation of the machine.

- PROGRAMMING MANUAL

This manual describes the programming of the machine.

Keep this manual for future reference.
Please include this manual when reselling this product.
When this manual or labels are lost or damaged, please replace them (charged) from your nearest agency.

This manual is printed by using paper obtained from farmed trees.

INTRODUCTION

Congratulations on your purchase of the Brother CNC tapping center. Correct usage of the machine is of most importance to assure the expected machine capabilities and functions as well as operator's safety. Read this Manual thoroughly before starting operation.

HOW TO USE THE MANUAL

This Instruction Manual consists of the following elements:

(1) **General description** Is an outline of the description given in the section.

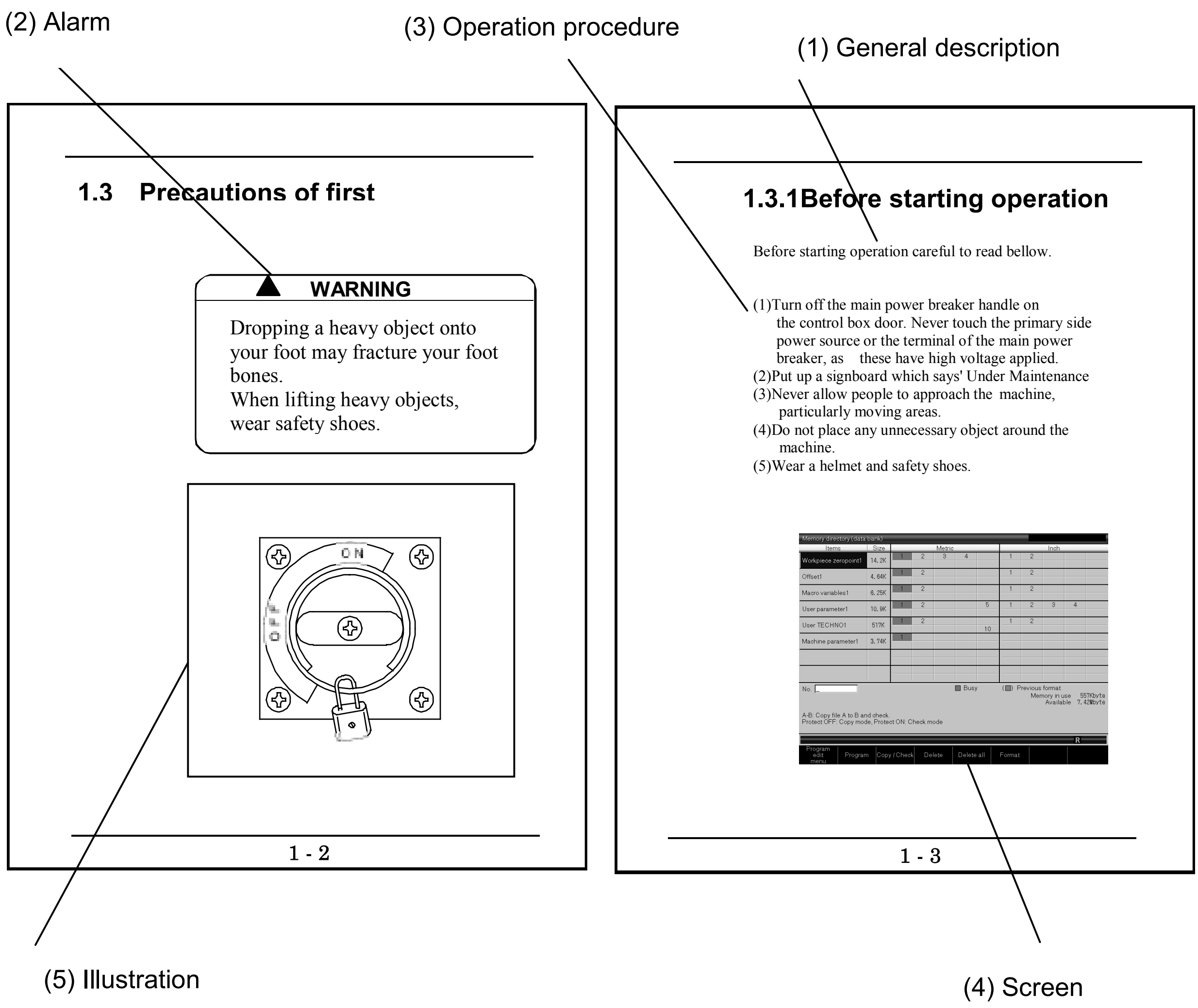
(2) **Alarm** Is a alert given against a danger which may cause serious damage or death to human being or may damage the machine.
The hazards are explained in this order:
degree of danger,
subject of danger,
expected damage,
preventive measure,

(3) **Operation procedure** Is a procedure of activating a function.

(4) **Screen** Is given to describe important points of a procedure given.

NOTE: This screen is only a representation of the information displayed on the actual screen and therefore differs somewhat from the actual screen layout and screen fonts.

(5) **Illustration** Is a sketch, figure, view, etc. indicating dimensions, position or zone, given in the points where it is necessary to provide complementary information to the text description.



(This page is a blank.)

(This page is a blank.)

CHAPTER 1

PROGRAM COMPOSITION

1.1 Types and Composition of Program

The program is divided into the main program and the subprogram.

(1) Main program

The main program is for machining one workpiece. While the main program is in use, a subprogram can be called to use the program more efficiently.
Command M02 (or M30) to finish the main program.

Main program

N0001 G92X100;
N0002 G00Z30
: **:** **:**
M02;

(2) Subprogram

A subprogram is used by calling it from the main program or other subprograms.
Command M99 to finish the subprogram.

Subprogram

N0100 G91X10;
: **:** **:**
M99;

1.2 Composition of Block

The program is composed of several commands. One command is called a block. A block is composed of one or more words. One block is discriminated from another block by an end of block code (EOB).
This manual expresses the end of block code by the symbol ";".

...	;	**N0001 G92X100**	;	...	;	**M02**	;	
		←Block→				←Block→		

(Note 1) **The end of block code**
ISO code : [LF] 0A(hexadecimal)
EIA code : [CR] 80(hexadecimal)

(Note 2) **One block has maximum 128 characters.**

1.3 Compositiom of Word

A word is composed of an address and some digit of figures as shown below.
(Algebraic sign + or - may added before a numerical value.)

X -1000

Address numerical value

Word

(Note 1) The address uses one of the alphabetical letters.
(Note 2) The address "O" can not be used except for comments.

1.4 Numerical Values

(1) Decimal point programming

Numerical values can be input in the following two ways and set by the user parameter (Switch 1).

Command type 1 (Standard)

Programmed command	Commanded axis	Actual amount (mm)	Actual amount (inch)
1	Feed axis	1mm	1 inch
	Rotation axis	1 deg	1 deg
1.	Rotation axis	1 mm	1 inch
	Rotation axis	1 deg	1 deg

Command type 2 (Minimum)

Programmed command	Commanded axis	Actual amount (mm)	Actual amount (inch)
1	Feed axis	0.001 mm	0.0001 inch
	Rotation axis	0.001 deg	0.001 deg
1.	Rotation axis	1 mm	1 inch
	Rotation axis	1 deg	1 deg

(Note) User parameter : Refer to Instruction manual.

(2) Programmable range of address

The maximum number of digits is 9.
The digits less than the minimum range are ignored.

1.5 Sequence Number

A sequence number (1~99999) can be used following the address N for each block.

Command format **N *****;**

i) A sequence number is used following the address N.
ii) A sequence number can be specified with up to 5-digit number.

(Note 1) The sequence number "N0" should not be used.
(Note 2) It is used at the head of a block.
Ex.) N0100 G90X100;
When a block has a slash (/) code at the head of block (the optional block skip is commanded), a sequence number can be used either before or after it.
Ex.) N0100/ G90X100; or /N0100 G90X100;

(Note 3)
The order of sequence numbers is arbitary and need not be consecutive.
(Note 4)
The sequence number is recognized as numerical values. Therefore such numerical values as 0001, 001, 01 and 1 are regarded as the same number.

1.6 Optional Block Skip

When a block has a slash (/) code at the start and **[BLOCK SKIP]** key on the operation panel is turned ON, all information in the block with the slash code is ignored during the automatic operation.
If the **[BLOCK SKIP]** key is OFF, information in the block with the slash code is effective.
That is, the block with a slash code can selectively be skipped.

..... ; / N0100 G00X100 ..... ; N0101 .....

Ignore these words

(Note 1)
A slash (/) code must be put at the start of a block. If it is placed elsewhere in the block, an alarm is generated.
This code can be also put right after a sequence number.
(Note 2)
In the single block mode during automatic operation, when the [BLOCK SKIP] key is ON the operation does not stop at a block with a slash code, but stops at the next block.

1.7 Control Out/In Function

For a easier look at the program, comments can be inserted in the program.
The comment is discriminated from operation by "(" and ")" at the start and the end.

(.............)

(Ex.) N1000 G00X200 (PRO-1);

(Note)
A comment including the control out and in codes should not be longer than one block.

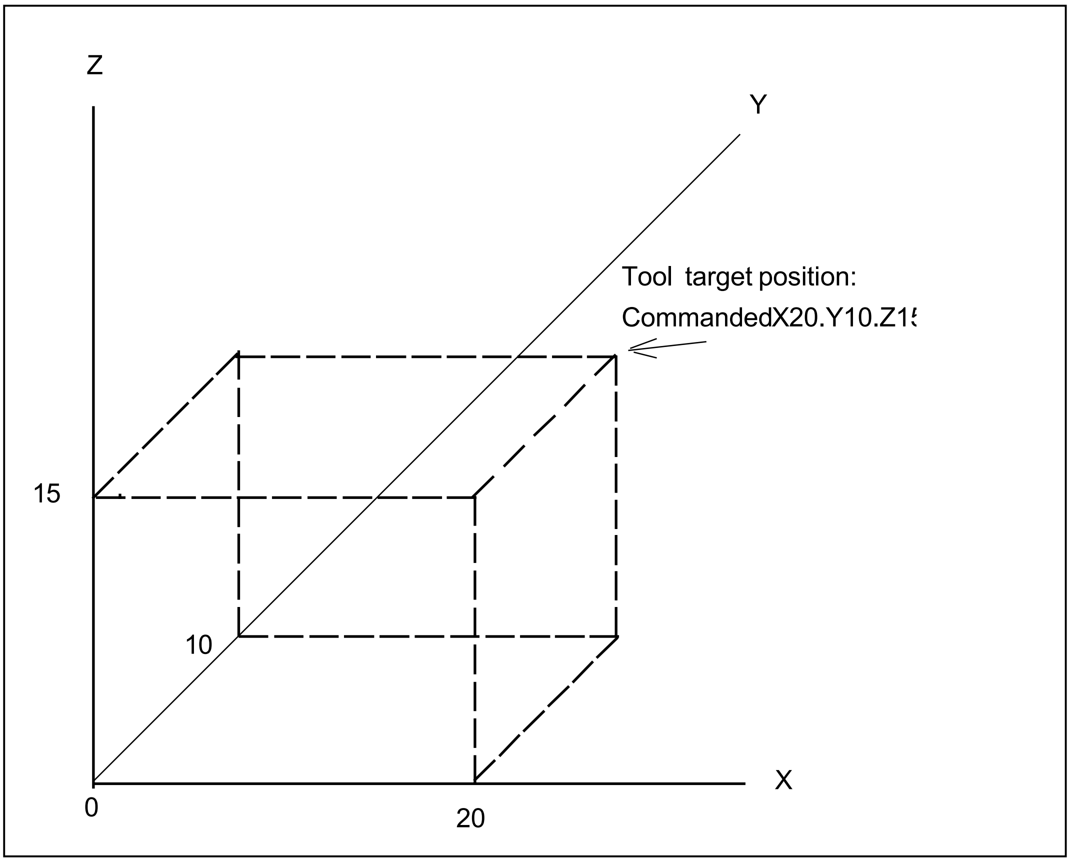
2

CHAPTER 2

COORDINATE COMMAND

2.1 Coordinate system and coordinate value

Coordinate values should be set in one coordinate system to specify a tool movement.
There are two types of coordinate systems.
(i) Machine coordinate system
(ii) Working coordinate system
The coordinate values are expressed by each component of the program axes (X, Y and Z for this unit).

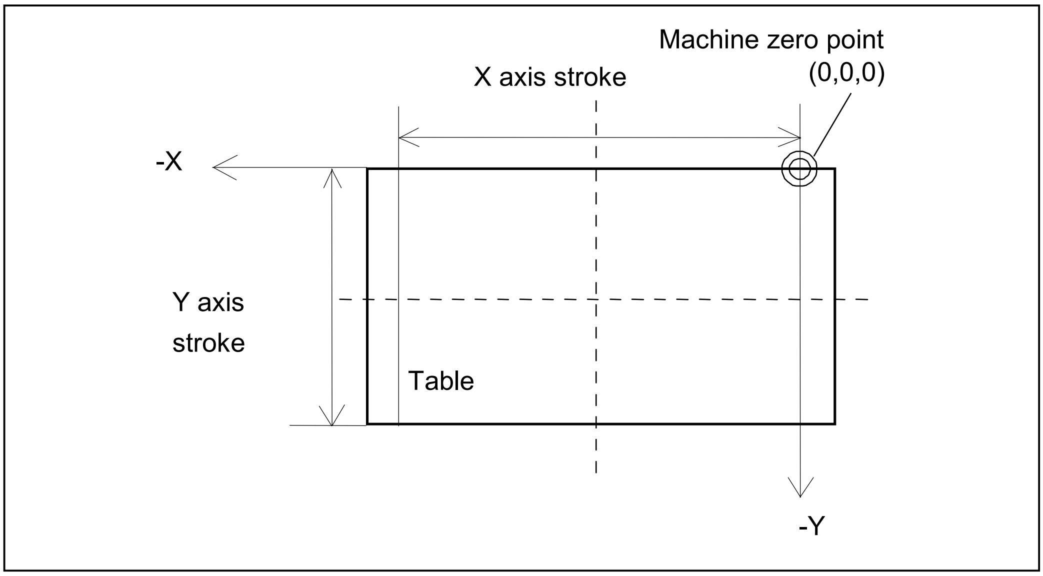


eNCPR2.01.ai

2.2 Machine Zero Point and Machine Coordinate System

(1) Machine zero point
The machine zero point is the reference point on the machine.

(2) Machine coordinate system
The coordinate systen with the machine zero point as its reference point is called the machine coordinate system. Each machine has its own coordinate system.



eNCPR2.02.ai

2.3 Working Coordinate System
The working coordinate system is used to specify a tool motion for each workpiece.
A coordinate system previously set in the "Data Bank" is once selected, programming afterward can be easily done by specifying that coordinate system.
Each coordinate system is set by using an offset amount from the machine zero point to the working zero position.
(Note) Data Bank : Refer to Instruction manual.

(This page is a blank.)

CHAPTER 3

PREPARATION FUNCTION

3.1 Outline of G code

Within 3-digit number following the address G determines the meaning of the command of the block concerned.

The G codes are divided into the following two types.

Type	Meaning
Modal	The G code is effective until another G code in the same group is commanded.
One-shot	The G code is effective only at the block in which it is specified.

3

Group	G cord	Contents	Modal
	G00*	Positioning	Modal
	G01	Linear interpolation	
	G02	Circular/ helical interpolation (CW)	
	G03	Circular / helical interpolation (CCW)	
	G102	XZ Circular interpolation (CW)	
	G103	XZ Circular interpolation (CCW)	
	G202	YZ Circular interpolation (CW)	
	G203	YZ Circular interpolation (CCW)	
	G04	Dwell	One-shot
	G09	Exact stop check	One-shot
	G10	Programmable data input	One-shot
	G13	Circular cutting CCW	One-shot
	G17*	XY plane selection	Modal
	G18	YZ plane selection	
	G19	ZX plane selection	
	G22*	Programmable stroke limit on	Modal
	G23	Programmable stroke limit cancel	
	G28	Return to the reference point	One-shot
	G29	Return from the reference point	
	G30	Return to the 2nd /3rd/4th reference point	
	G31	Skip function	One-shot
	G36	Coordinate calculation function (Bolt hole circle)	One-shot
	G37	Coordinate calculation function (Line-angle)	
	G38	Coordinate calculation function (Line-angle)	
	G39	Coordinate calculation function (Grid)	
	G40*	Tool dia offset cancel	Modal
	G41	Tool dia offset left	
	G42	Tool dia offset right	

The G codes with * mark indicates the modal status when the power is turned ON.
(Note1) Details of coordinate calculation functions are described in " Chapter 6 ".
(Note2) Details of tool dia offset are described in " Chapter 4 ".

Group	G cord	Contents	Modal
	G43	Tool length offset +	Modal
	G44	Tool length offset -	
	G49*	Tool length offset cancel	
	G50*	Scaling cancel	Modal
	G51	Scaling	
	G50.1	Mirror image cancel	Modal
	G51.1	Mirror image	
	G52	Local coordinate system	One-shot
	G53	Machine coordinate system selection	
	G54*	Working coordinate system selection 1	Modal
	G55	Working coordinate system selection 2	
	G56	Working coordinate system selection 3	
	G57	Working coordinate system selection 4	
	G58	Working coordinate system selection 5	
	G59	Working coordinate system selection 6	
	G54.1	Extended working coordinate system selection	
	G60	Single direction positioning	One-shot
	G61	Exact stop mode	Modal
	G64*	Cutting mode	
	G65	Macro call	One-shot
	G66	Macro modal call	Modal
	G67*	Cancel macro modal call	
	G68	Coordinate rotation function	Modal
	G69*	Coordinate rotation function cancel	
	G168	Coordinate rotation using measured results	

Group	G cord	Contents	Modal
	G90*	Absolute command	Modal
	G91	Incremental command	
	G92	Working coordinate system setting	One-shot
	G94	Feed rate per minute	
	G98*	Return to the initial point level	Modal
	G99	Return to the R point level	
	G73	Canned cycle (High-speed peck drilling cycle)	Modal
	G74	Canned cycle (Reverse tapping cycle)	
	G76	Canned cycle (Fine boring cycle)	
	G77	Canned cycle (Tapping cycle, synchro mode)	
	G78	Canned cycle (Reverse tapping cycle, synchro mode)	
	G80*	Canned cycle cancel	
	G81	Canned cycle (Drill, spot drilling cycle)	
	G82	Canned cycle (Drill, spot drilling cycle)	
	G83	Canned cycle (Peck drilling cycle)	
	G84	Canned cycle (Tapping cycle)	
	G85	Canned cycle (Boring cycle)	
	G86	Canned cycle (Boring cycle)	
	G87	Canned cycle (Back boring cycle)	
	G89	Canned cycle (Boring cycle)	

Group	G cord	Contents	Modal
	G173	Canned cycle (High-speed peck drilling cycle)	One-shot
	G177	Canned cycle (End mill tap cycle)	Modal
	G178	Canned cycle (End mill tap cycle)	
	G181	Canned cycle (Double drilling cycle)	Modal
	G182	Canned cycle (Double drilling cycle)	
	G183	Canned cycle cancel (Peck drilling cycle)	One-shot
	G185	Canned cycle (Double boring cycle)	Modal
	G186	Canned cycle (Double boring cycle)	
	G189	Canned cycle (Double drilling cycle)	
	G100	Non-stop automatic tool change	One-shot

Group	G cord	Contents	Modal
	G120	Positioning to the measuring point	One-shot
	G121	Automatic measurement Corner (Boss)	One-shot
	G122	Automatic measurement Parallel (Groove)	
	G123	Automatic measurement Parallel (Boss)	
	G124	Automatic measurement Circle center (Hole, 3 points)	
	G125	Automatic measurement Circle center (Boss, 3 points)	
	G126	Automatic measurement Circle center (Hole, 4 points)	
	G127	Automatic measurement Circle center (Boss, 4 points)	
	G128	Automatic measurement Z-axis height	
	G129	Automatic measurement Corner (Groove)	
	G31	Measurement feed	One-shot
	G131	Measurement feed	
	G132	Measurement feed	
	G133	Changeover of tap twisting direction (CW)	One-shot
	G134	Changeover of tap twisting direction (CCW)	

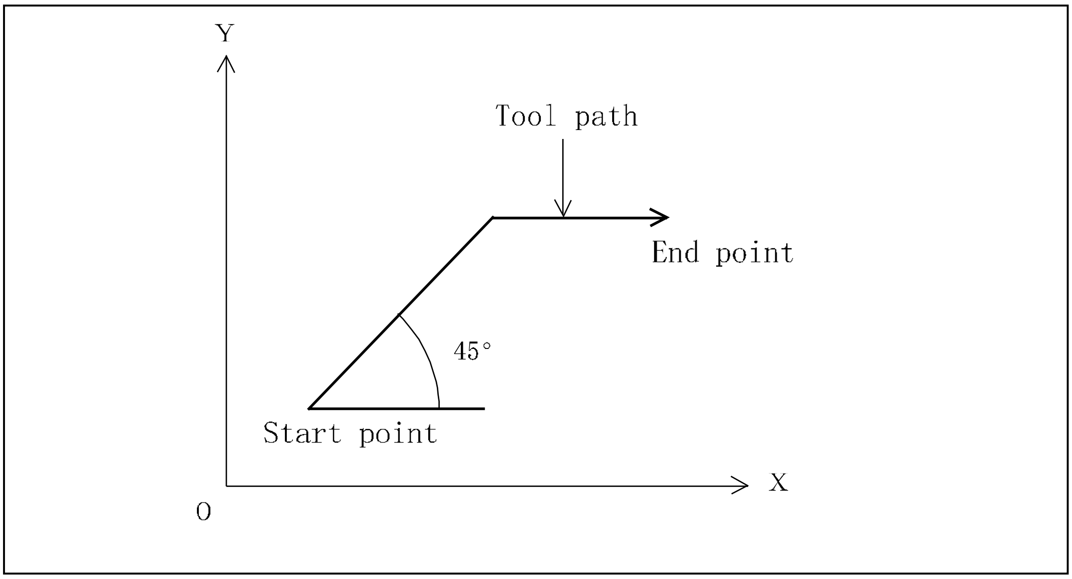
(Note)
Commands G120 to G129 are described in detail in " Option, Automatic Measurement " in the instruction manual.

3.2 Positioning (G00)

A tool moves from its current position to the end point at the rapid traverse rate in each axis direction independently. Therefore, a tool path is not always a linear line.

Command format	**G00 X_Y_Z_A_B_C_ ;**

When the additional axis is commanded and the optional additional axis is not installed, an alarm will occur.
In the positioning mode actuated by the G00 code, the execution proceeds to the next block after confirming the in-position check. (Note 1)



eNCPR3.01.ai

(Note 1)
In-position check is to confirm that the machine detecting position is within the specified range around the target (end) point.
(This range is set by the machine parameter for each axis.)
(Note 2)
The rapid traverse rate is set by the machine parameter for each axis.
Accordingly, rapid traverse rate cannot be specified by the F command.

3.3 Linear interpolation (G01)

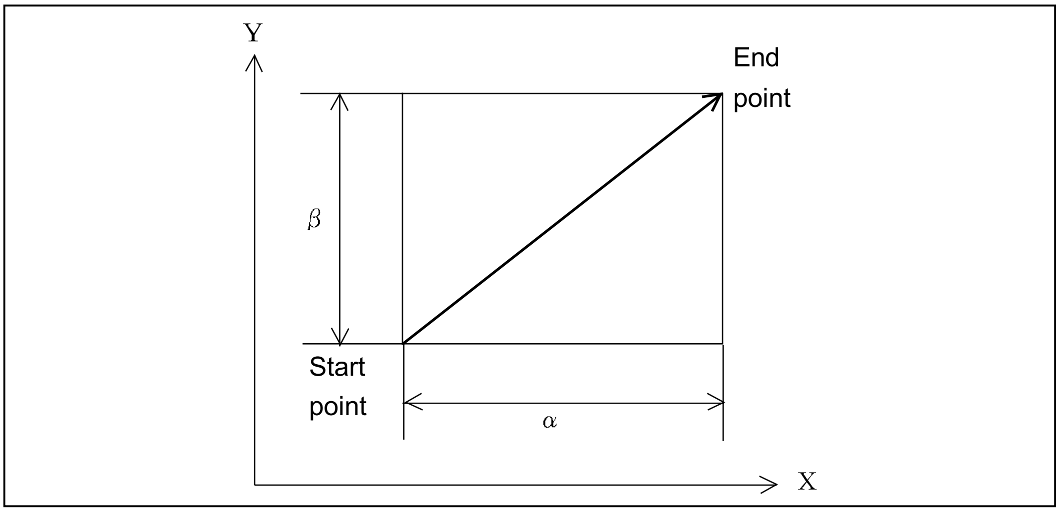
Linear interpolation moves a tool linearly from the current position to the target position at the specified feedrate.

Command format	G01 X_Y_F_ ;

Up to X,Y,X and one additional axis can be controlled simultaneously.
When the additional axis is commanded and the optional additional axis is not installed, an alarm will occur.

3

The feedrate is commanded by the address F. Once the feedrate is commanded, it is effective until another value is specified.
When the X, Y, and Z axes are commanded, the feedrate is determined by the value entered to mm / min.
When the additional axis is commanded, the feedrate is determined by the value entered to ᵒ /min.



eNCPR3.2.ai

(Note 1) Feedrate along each axis is as follows:
When "G01 G91 Xα Yβ Ff ;" is programmed:

Feedrate along X axis: $Fx = \frac{\alpha}{L} \bullet f$

Feedrate along Y axis: $Fy = \frac{\beta}{L} \bullet f$

$(L = \sqrt{\alpha^2 + \beta^2})$

(Note2)
The example below shows linear interpolation of linear axis α and rotation axis β.

When "G01 G91 Xα Bβ Ff ;" is programmed:

Time taken for B-axis movement: $Tb = \dfrac{\sqrt{\alpha^2+\beta^2}}{f}$

Feedrate along B axis: $Fb = \dfrac{\beta}{Tb}$

Feedrate along X axis: $Fx = \dfrac{\alpha}{L} \cdot f$

3.3.1 Chamfering to desired angle and cornering C

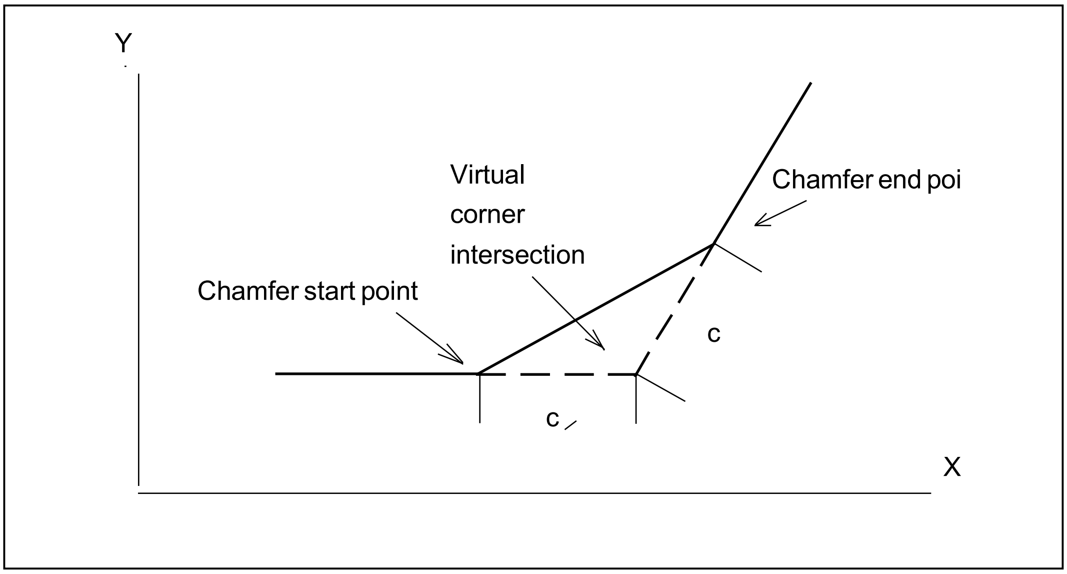
Chamfering to the desired angle or rounding can be performed between interpolation commands.

Chamfering

Command format

G01 X_Y_, C_ ;

C: Distance from virtual corner to the chamfer start point and send point.



eNCPR3.03.ai

(1) The corner chamfering command block and subsequent block must contain the interpolation command (G01-G03).
When the subsequent block does not contain an interpolation or movement command, an alarm will occur.

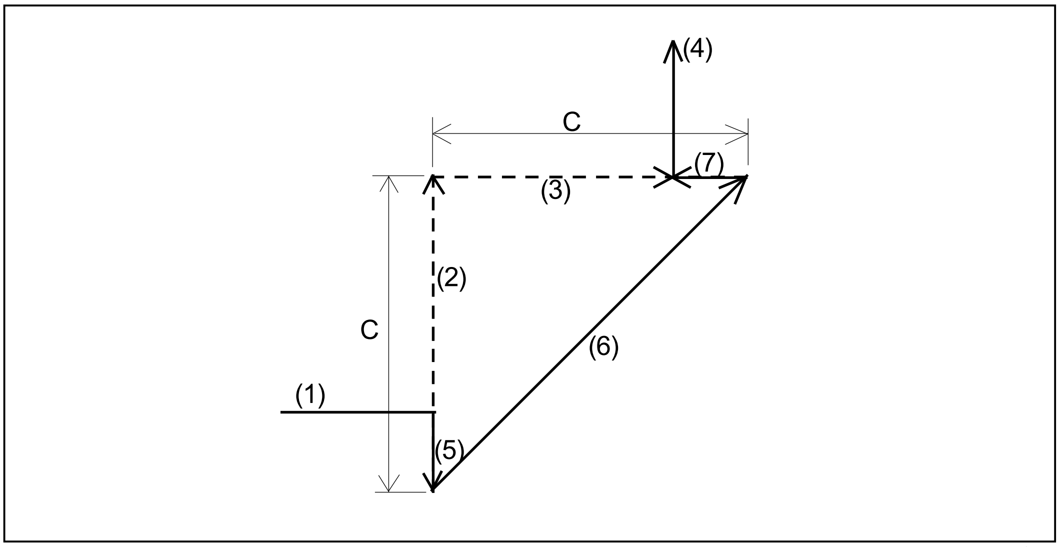
(2) The inserted block belongs to the corner chamfering command block. Thus, if the feed rate differs from the corner chamfering command block and the subsequent block , the inserted block moves at the feed rate of the corner
chamfering command block. Further, the program does not stop before the inserted block occurs even during single block operation. (It stops after the inserted block occurs.)

(3) Tool diameter compensation applies to the configuration after corner chamfering is performed.

3

(4) When the chamfering amount is longer than the chamfering command block and feeding quantity of the subsequent block, set extended point from each blocks as "chamfer start point" and "chamfer end point".

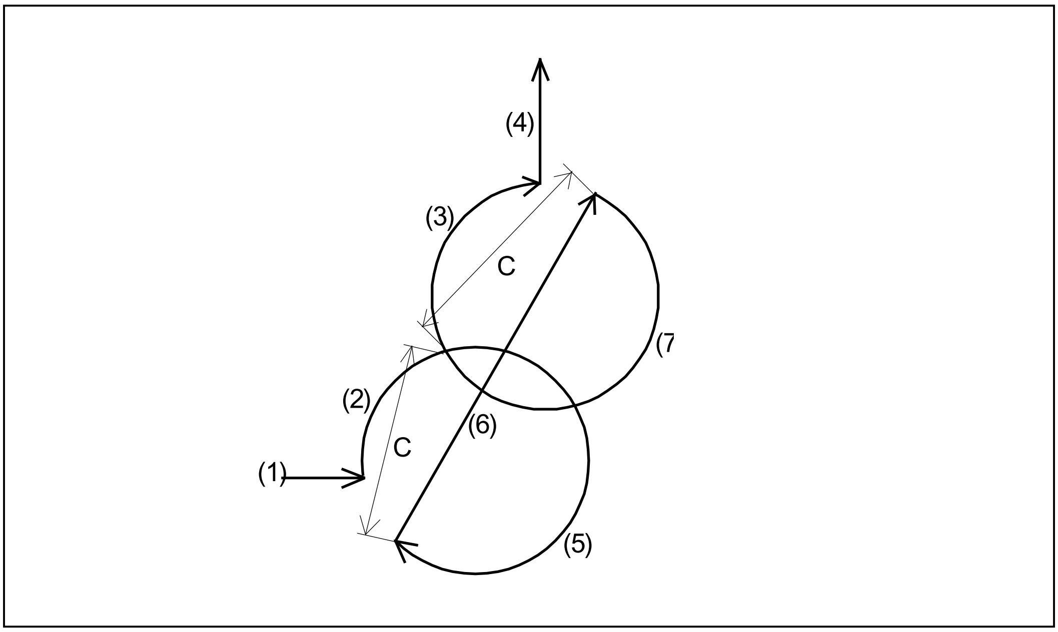
Example.1: Liner cutting



eNCPR3.04.ai

When set the programmed path to (1.2.3.4.) and the block C as (2), operate to 1-5-6-7-4.

Example.2: Circular cutting



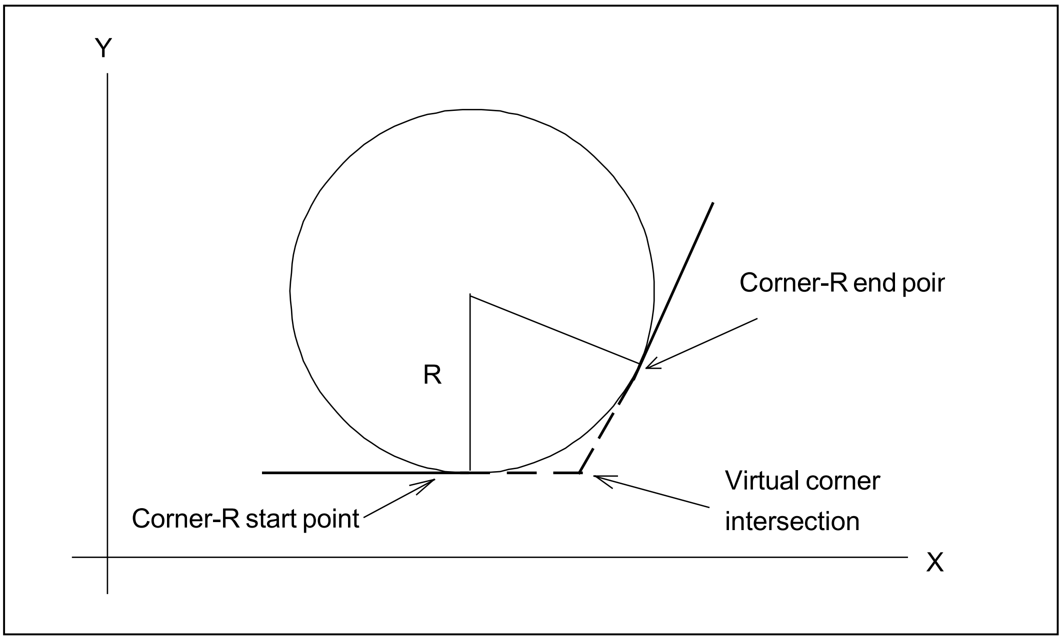
eNCPR3.05.ai

When set the programmed path to(1.2.3.4.)and the block C as (2), operate to 1-5-6-7-4.

Cornering

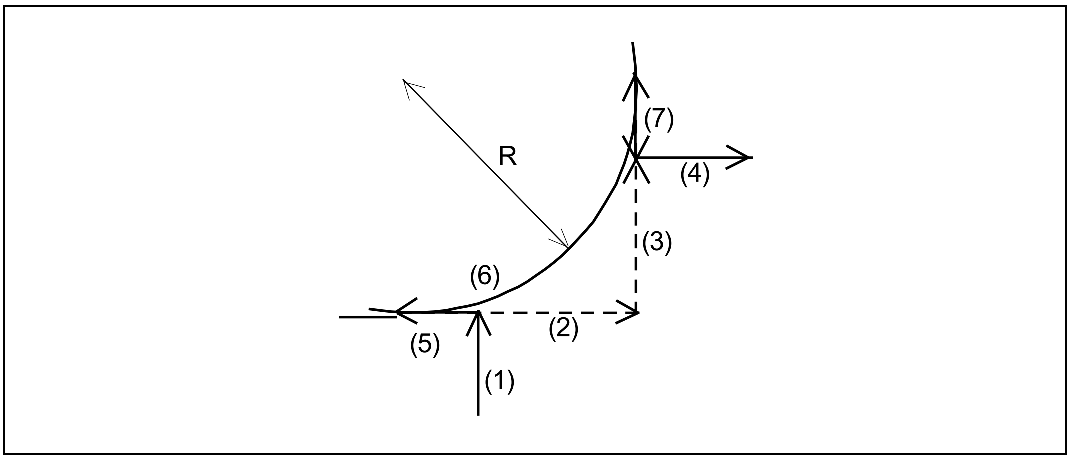
Command format | **G01 X_Y_, R_ ;** |

R : Radius of cornering



(1) The cornering command block and the subsequent block must contain the interpolation command (G01-G03).
When the subsequent block does not contain an interpolation or movement command, an alarm will occur.
(2) The inserted block belongs to the cornering command block. Thus, if the feed rate differs from the cornering command block and the subsequent block , the inserted block moves at the feed rate of the cornering command block. Further, the program does not stop before the inserted block occurs even during single block operation. (It stops after the inserted block occurs.)
(3) Tool diameter compensation applies to the configuration after cornering is performed.
(4) When the radius is longer than the corner R command block and the subsequent command block, set extended point from each blocks as "chamfer start point" and "chamfer end point".

Example.1: Liner cutting



eNCPR3.07.ai

When set the programmed path to(1.2.3.4.)and the block R as (2), operate to 1-5-6-7-4.

3.4 Circular/Helical Interpolation (G02, G03)

3.4.1 Circular interpolation

Circular interpolation moves a tool along a circular arc from the current position to the end point at the specified feedrate.

3.4.1.1 Circular interpolation

Command format

X-Y plane

```
G17G02 X_ Y_  { I_ J_ }  F_;
              { R_    }
G17G03 X_ Y_  { I_ J_ }  F_;
              { R_    }
```

Z-X plane

```
G18G02 Z_ X_  { K_ I_ }  F_;
              { R_    }
G18G03 Z_ X_  { K_ I_ }  F_;
              { R_    }
```

Y-Z plane

```
G19G02 Y_ Z_  { J_ K_ }  F_;
              { R_    }
G19G03 Y_ Z_  { J_ K_ }  F_;
              { R_    }
```

The commands are gives in the following format:

Rotation direction		G 02	Clockwise (CW).
		G 03	Counterclockwise (CCW).
End point	G90 mode	X,Y,Z	End point in the working coordinate system.
	G91 mode	X	Distance from the start point to the end point in the X direction.
		Y	Distance from the start point to the end point in the Y direction.
		Z	Distance from the start point to the end point in the Z direction.
Distance between start point and arc center		I	Distance from the start point to the center of arc in the X direction.
		J	Distance from the start point to the center of arc in the Y direction.
		K	Distance from the start point to the center of arc in the Z direction.
Arc radius		R	Arc radius
Feedrate		F	Feedrate in the tangential direction of circular arc.

Clockwise and counterclockwise are the rotation direction viewed from the positive direction to the negative direction on the Z axis of the plus direction.

3.4.1.2 XZ Circular interpolation

Command format	G102 / G103	X_ Y_	I_ J / R_	F_;

The commands are given in the following format:

<table>
<tr><td colspan="2" rowspan="2">Rotation direction</td><td>G 102</td><td>Clockwise (CW).</td></tr>
<tr><td>G103</td><td>Counterclockwise (CCW).</td></tr>
<tr><td rowspan="3">End point</td><td>G90 mode</td><td>X,Y</td><td>End point in the working coordinate system.</td></tr>
<tr><td rowspan="2">G91 mode</td><td>X</td><td>Distance from the start point to the end point in the X direction.</td></tr>
<tr><td>Y</td><td>Distance from the start point to the end point in the Y direction.</td></tr>
<tr><td colspan="2" rowspan="2">Distance between start point and arc center</td><td>I</td><td>Distance from the start point to the center of arc in the X direction.</td></tr>
<tr><td>J</td><td>Distance from the start point to the center of arc in the Y direction.</td></tr>
<tr><td colspan="2">Arc radius</td><td>R</td><td>Arc radius</td></tr>
<tr><td colspan="2">Feedrate</td><td>F</td><td>Feedrate in the tangential direction of circular arc.</td></tr>
</table>

Clockwise and counterclockwise are the rotation direction viewed from the positive direction to the negative direction on the Y axis of the X-Z plane.

(Note 1)
In contrast to the XY arc case, an error occurs when the diameter compensation command (G41, G42) or coordinate rotation command (G68, G168) is used, and the machine stops operation.

3

3.4.1.3 XZ Circular interpolation

Command format

$$\begin{pmatrix} G202 \\ G203 \end{pmatrix} \quad X\_\ Y\_ \quad \begin{pmatrix} I\_\ J \\ R\_ \end{pmatrix} \quad F\_;$$

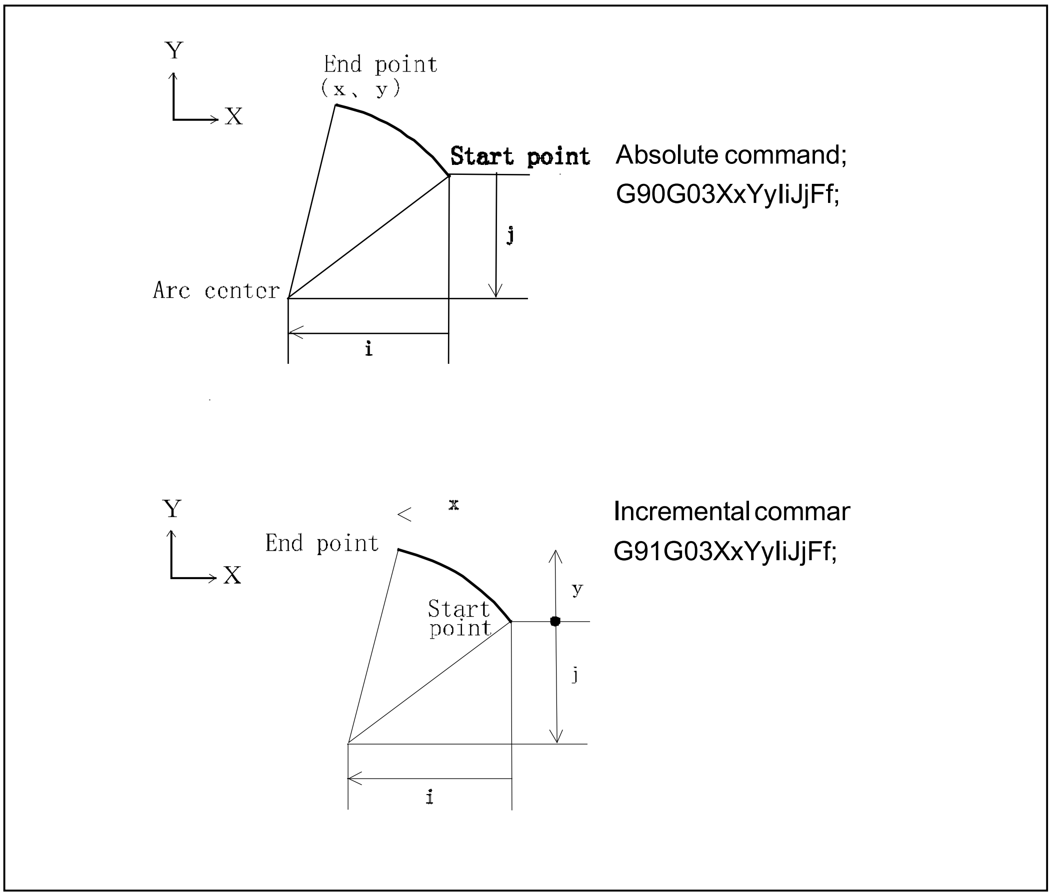
The commands are given in the following format:

Rotation direction		G202	Clockwise (CW).
		G203	Counterclockwise (CCW).
End point	G90 mode	X,Y	End point in the working coordinate system.
	G91 mode	X	Distance from the start point to the end point in the X direction.
		Y	Distance from the start point to the end point in the Y direction.
Distance between start point and arc center		I	Distance from the start point to the center of arc in the X direction.
		J	Distance from the start point to the center of arc in the Y direction.
Arc radius		R	Arc radius
Feedrate		F	Feedrate in the tangential direction of circular arc.

Clockwise and counterclockwise are the rotation direction viewed from the positive direction to the negative direction on the X axis of the Y-Z plane.

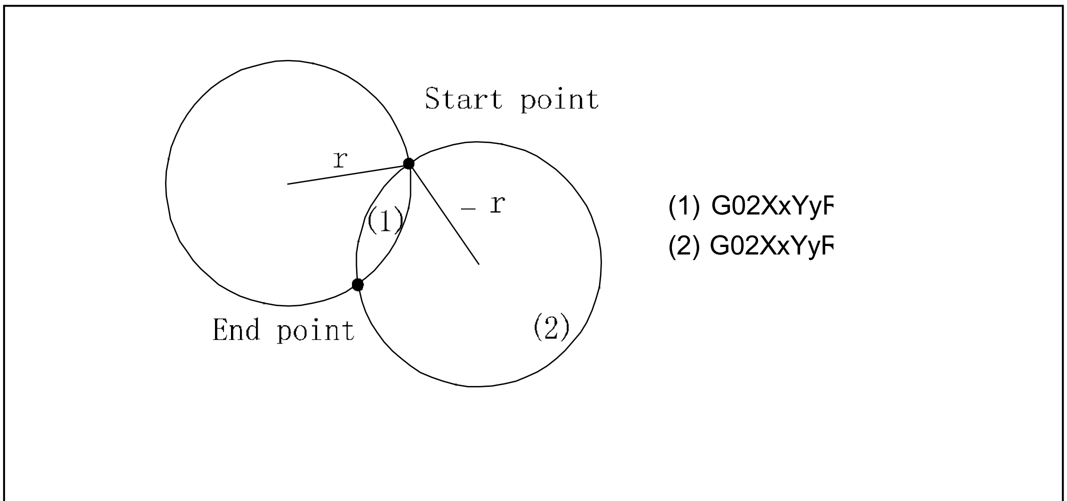
(Note 1)
In contrast to the XY arc case, an error occurs when the diameter compensation command (G41, G42) or coordinate rotation command (G68, G168) is used, and the machine stops operation.

The end point of the circular arc takes either the absolute value or the incremental value according to G90 or G91. The incremental value commands the distance from the circular arc start point to the end point.
The circular arc center is commanded by both I,J and K according to X,Y and Z axes. I,J and K form a vector component when viewed from the circular arc start point to the center.
It is commanded by the incremental value regardless of G90 or G91.



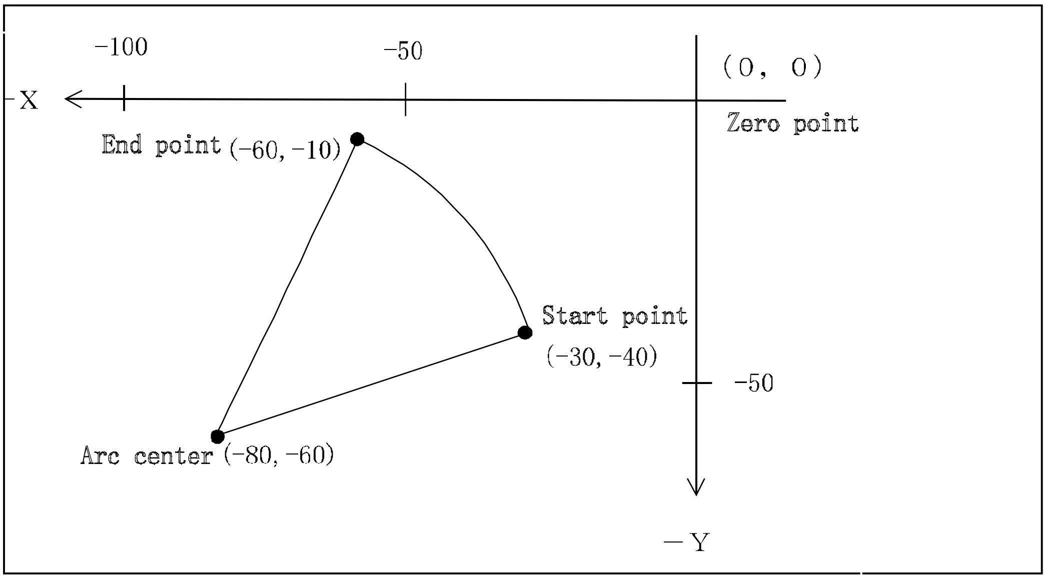
eNCPR3.08.ai

Instead of commanding I, J and K to specify the center of arc, the radius of arc can be used.
There are two types of circular arcs (one is less than 180° and the other is more than 180°).
When commanding a circular arc of more than 180°, put the algebraic mark "-" before the value for the radius.



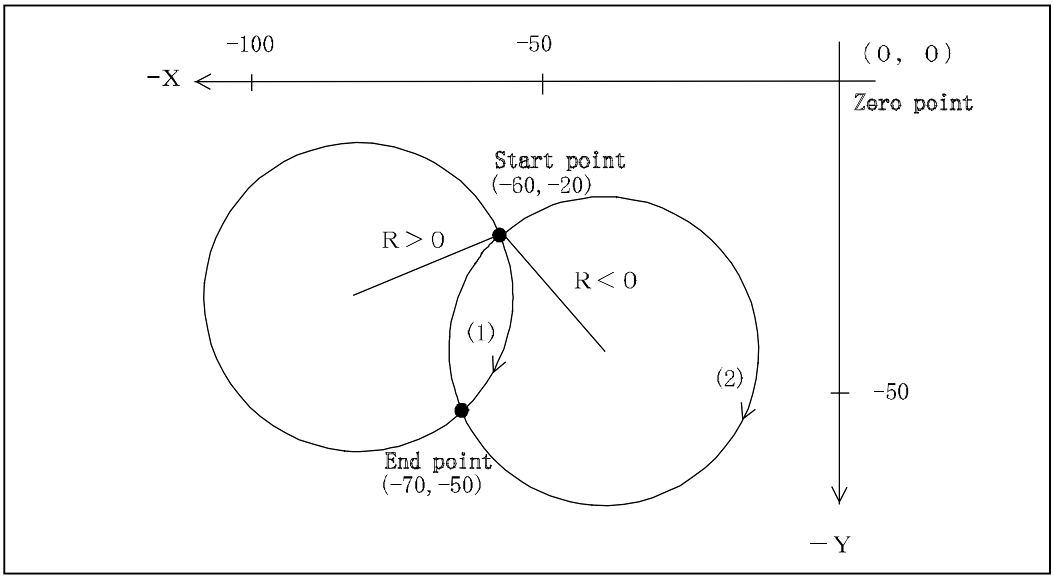
eNCPR3.09.ai

3



eNCPR3.10.ai

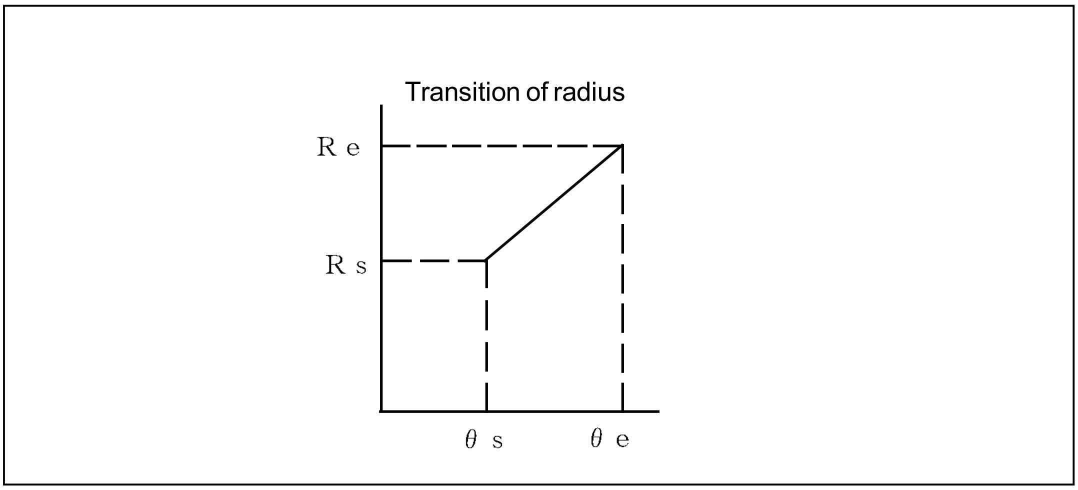
Absolute command;
G03X-60. Y-10. I-50. J-20. F1000 ;
Incremental command;
G03X-30. Y30. I-50. J-20. F1000 ;



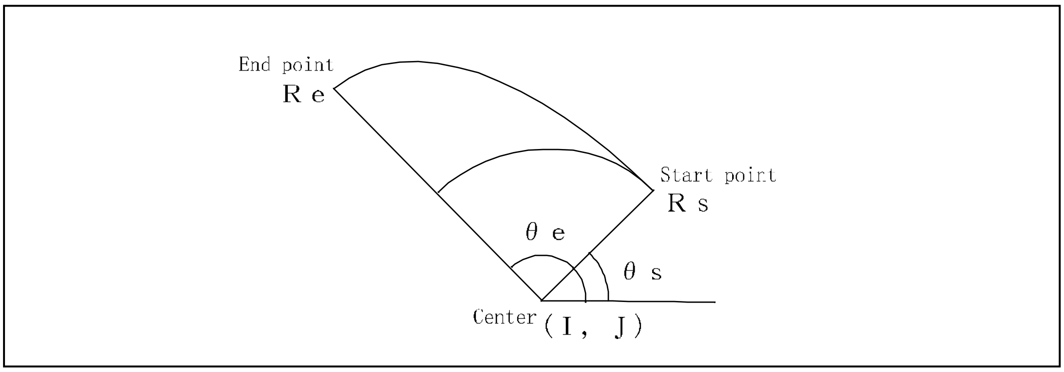
eNCPR3.11.ai

(1) G02X-70. Y-50. R25. F1000 ;
(2) G02X-70. Y-50. R-25. F1000 ;

(Note 1) When either I, J or K is omitted, it is regarded zero.
(Note 2) The circular arc, when its radius is zero, cannot be commanded.
(Note 3) When both X,Y and Z are omitted, the end point and the start point are regarded identical, and:
i) 360°arc (full circle) is assumed to be commanded when the arc center is programmed using the address I,J and K.
ii) When the address R is used, an alarm occurred.
(Note 4) The address R and "I, J and K" cannot be commanded simultaneously.
(Note 5) When the end point is not on the arc specified by start point and arc radius, the tool moves as shown below.



eNCPR3.15.ai



eNCPR3.14.ai

(Note 6) **If the ending radius is extremely larger than that of the starting radius, an alarm will occur.**

(Note 7) **The G36~G39 codes cannot be commanded in the circular arc mode.**

3.4.2 Helical interpolation

Putting the other than selected plane axis command in the circular arc block permits a helical cutting.

Command format

X-Y plane:

G17G02 X_Y_Z_	$\begin{Bmatrix} I\_\ J\_ \\ R\_ \end{Bmatrix}$	**F_;**
G17G03 X_Y_Z_	$\begin{Bmatrix} I\_\ J\_ \\ R\_ \end{Bmatrix}$	**F_;**

Z-Y plane:

G18G02 Z_X_Y_	$\begin{Bmatrix} K\_I\_ \\ R\_ \end{Bmatrix}$	**F_;**
G18G03 Z_X_Y_	$\begin{Bmatrix} K\_I\_ \\ R\_ \end{Bmatrix}$	**F_;**

Y-Z plane:

G19G02 Y_Z_X_	$\begin{Bmatrix} J\_K\_ \\ R\_ \end{Bmatrix}$	**F_;**
G19G03 Y_Z_X_	$\begin{Bmatrix} J\_K\_ \\ R\_ \end{Bmatrix}$	**F_;**

The F code commands the feedrate in the circular interpolation axis..

If the value of F is larger than the MAXIMUM CUTTING SPEED or the FEEDRATE SPEED set by the machine parameter, an alarm is generated.

The feedrate in the other than selected plane axis is determined by the values of "feedrate" in the circular interpolation axis, "end point X", "end point Y" and "end point Z". It can be calculated as follows:

$$F_Z = \frac{180 \times L}{\pi \times R \times \theta} \times F$$

F: Command speed (X, Y axes)
R: Radius (Start point, center)
θ: Angle
Fz: Other than selected plane of feeedrate speed.
L: Other than selected plane of feed distance.

Ex.)

Setting following values:
F=500 (mm/min), R=10 (mm), θ=360 (°), L=2 (mm)

Fz = (180×2×500)/(π×10×360) ≒ 15.9 (mm/min)

If the other than selected plane axis feedrate is larger than the MAXIMUM CUTTING SPEED or FEEDRATE SPEED set by the machine parameter , an alarm is generated.

When tool dia offset command is given, an offset is applied to the selected plane.

3.4.3 Spiral interpolation (G02, G03)

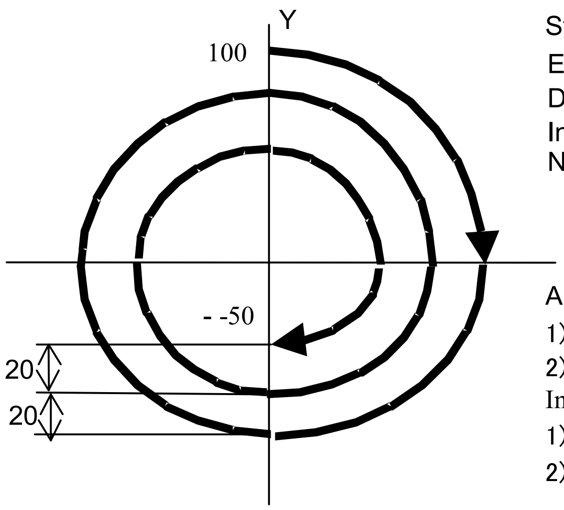
An increment or decrement per rotation is specified for the circular interpolation command to perform spiral interpolation.

Command format

X-Y plane:
{G17}G02X_Y_I_J_Q_L_F_;
{G17}G03X_Y_I_J_Q_L_F_;
Z-Y plane:
{G18}G02Z_X_K_I_Q_L_F_;
{G18}G03Z_X_K_I_Q_L_F_;
Y-Z plane:
{G19}G02Y_Z_J_K_Q_L_F_;
{G19}G03Y_Z_J_K_Q_L_F_;

G02	:	Clockwise cutting direction
G03	:	Counterclockwise cutting direction
XYZ	:	Coordinates of end point
L	:	Number of rotations (positive value, decimal numbers are rounded up to the nearest whole number). Add a decimal point, rounded off. Example: Set "L6" for five and 1/4 rotations (5.25 rotations).
Q	:	Increment or decrement in radius per rotation Setting a positive value increases the radius for each rotation. Setting a negative value decreases the radius for each rotation.
IJK	:	Vector (distance and direction) from the start point to the center (the same as circular interpolation)
F	:	Cutting speed

(Note)
Either L (number of rotations) or Q (increment/decrement in radius) can be omitted.
If there is a discrepancy between "L" and "Q" when used together, "Q" is used.



Start point (0,100)
End point (X,Y)(0,-50.)
Distance to the center (I,J)(0,-100.)
Increment/decrement in radius Q –20.0
No. of rotations L 3

Absolute command
1) G90G02X0.Y-50.I0J-100.Q-20. ;
2) G90G02X0.Y-50.I0J-100.L4;
Incremental command
1) G91G02X0Y-150.I0J-100.Q-20.;
2) G91G02X0Y-150.I0J-100.L4

Setting either 1) or 2) is acceptable.

Cutter compensation can be performed only in offset mode. An alarm will occur when this is attempted in startup or cancel mode.

The setting for [Cutter compensation] is applied relative to the start point and target point specified in the program during cutter compensation.

An alarm will occur when the programmed conical interpolation tool path or the tool path after cutter compensation intersects or makes contact with the spiral center.

An alarm will occur when the spiral defined that over the circle radius fudge factor limit point by the increment or decrement in radius per rotation does not match the end point.

An alarm will occur when corner CR is specified in the block immediately before a block that performs spiral interpolation.

Automatic corner override is not possible for the blocks immediately before and after a block that performs spiral interpolation.

Corner CR cannot be specified for spiral interpolation.

An alarm will occur when the radius is zero (0) or less (including negative values) as a result of setting an increment/decrement in the radius per rotation and the number of rotations.

An alarm will occur when the radius is specified using command "R."

An alarm will occur when the increment or decrement in radius is zero (0).

An alarm will occur when setting value of selected flat as below.

(1) Start point radius = End point radius
(2) Start point = Center
(3) End point = Center

Not commanded when mirror image is effective.
Not commanded when scaling image is effective.

When a cutter compensation cancel command is included in the block immediately after a block that performs spiral interpolation and cutter compensation, the end point of the spiral interpolation will be the position given by the vertical vector from the end point of spiral interpolation.

An in-position check is performed between the blocks immediately before and after a block that performs spiral interpolation.

3.4.4 Conical interpolation (G02, G03)

The travel command of another axis in addition to the spiral interpolation command is added and an increment and decrement is specified for that axis per spiral rotation to perform conical interpolation.

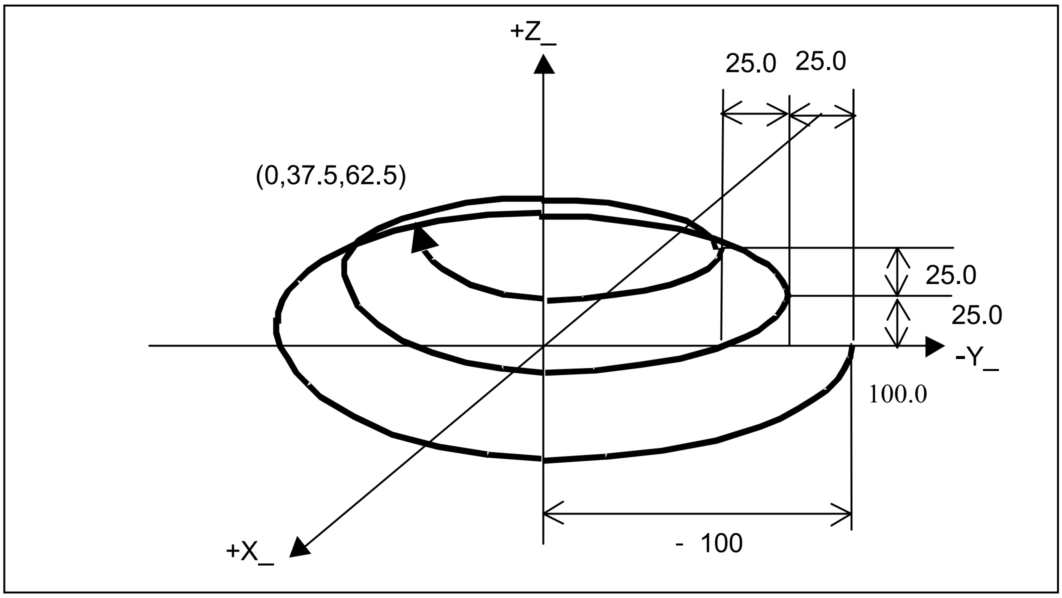
Command format

X-Y plane:
{G17}G02X_Y_Z_I_J_K_Q_L_F_;
{G17}G03X_Y_Z_I_J_K_Q_L_F_;
Z-X plane:
{G18}G02Z_X_Y_K_I_J_Q_L_F_;
{G18}G03Z_X_Y_K_I_J_Q_L_F_;
Y-Z plane:
{G19}G02Y_Z_X_J_K_I_Q_L_F_;
{G19}G03Y_Z_X_J_K_I_Q_L_F_;

G02 : Clockwise cutting direction
G03 : Counterclockwise cutting direction
XYZ : Coordinates of end point
L : Number of rotations (positive value, decimal numbers are rounded up to the nearest whole number). Add a decimal point, rounded off.
Example: Set "L6" for five and 1/4 rotations (5.25 rotations).
Q : Increment or decrement in radius per rotation
Setting a positive value increases the radius for each rotation.
Setting a negative value decreases the radius for each rotation.
IJK : Set a vector from the start point to the center for two axes and the increment/decrement in height per spiral rotation used for conical interpolation for the remaining axis.*

Plane to be set	Vector from start point to center	Increment and decrement in height per spiral rotation
G17 X-Y plane	I, J	K
G18 Z-X plane	K, I	J
G19 Y-Z plane	J, K	I

F : Cutting speed
*) As long as one of IJK, L, and Q (increment/decrement in height, number of rotations, increment/decrement in radius) is set, setting the remaining two items can be omitted.
If there is a discrepancy between "L" and "Q," the latter is used.
If there is a discrepancy between "L" and the increment/decrement in height, the latter is used.
If there is a discrepancy between "Q" and the increment/decrement in height, the former is used.
Priority Higher ← "Q" > Increment/decrement in height > "L" → Lower



Example of program

Start point	(0.,100.,0.)
End point	(0.,-37.5,62.5)
Distance to the center	(0.,-100.)
Increment/decrement in radius	-25.
Increment/decrement in height	25.
No. of rotations	3

Example of program

Start point	(0.,100.,0.)
End point	(0.,-37.5,62.5)
Distance to the center	(0.,-100.)
Increment/decrement in radius	-25.
Increment/decrement in height	25.
No. of rotations	3

Absolute command **G90G02X0 Y-37. 5Z62.5I0.J -100. (K25. / Q25. / L3) F300.;**

Incremental command **G90G02X0 Y-137. 5Z62.5I0.J -100. (K25. / Q25. / L3) F300.;**

Cutter compensation can be performed only in offset mode. An alarm will occur when this is attempted in startup or cancel mode.

The setting for [Cutter compensation] is applied to the selected plane during cutter compensation, relative to the start point and target point specified in the program.

An alarm will occur when the programmed conical interpolation tool path or the tool path after cutter compensation intersects or makes contact with the conical center.

An alarm will occur when the circular cone defined that over the circle radius fudge factor limit point by the increment or decrement in radius per rotation does not match the end point.

An alarm will occur when corner CR is specified in the block immediately before a block that performs conical interpolation.

Automatic corner override is not possible for the blocks immediately before and after a block that performs conical interpolation.

Corner CR cannot be specified for conical interpolation.

An alarm will occur when the cutter compensation direction (G41, G42) is changed between the blocks immediately before and after a block that performs conical interpolation.

An alarm will occur when the radius is specified using command "R."

An alarm will occur when the increment or decrement in radius is zero (0).

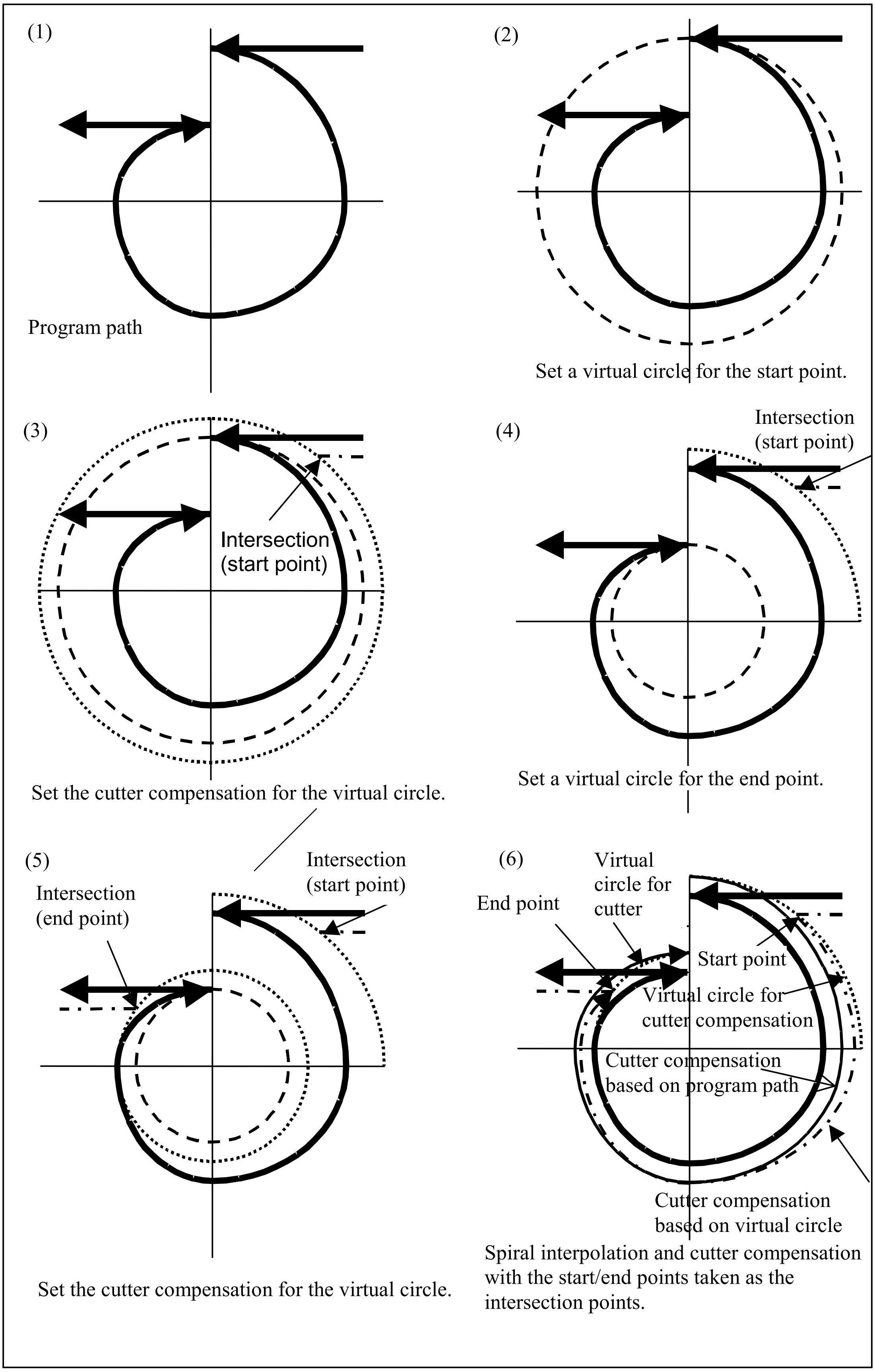
An alarm will occur when setting value of selected flat as below.
(1) Start point radius = End point radius
(2) Start point = Center
(3) End point = Center
Not commanded when mirror image is effective.
Not commanded when scaling image is effective.

When a cutter compensation cancel command is included in the block immediately after a block that performs conical interpolation and cutter compensation, the end point of the conical interpolation will be the position given by the vertical vector from the end point of conical interpolation on the selected plane.

An in-position check is performed between the blocks immediately before and after a block that performs conical interpolation.

3.4.5 Cutter compensation procedure for spiral interpolation and conical interpolation (G02, G03)

Assuming a virtual circle with the center of the spiral interpolation as the center for the start point and end point of the block, cutter compensation is performed for the virtual circle and then spiral interpolation is performed based on the result of cutter compensation.



3.5 Circle Cutting (G12, G13)

Starting from the center of the circle, the tool cuts the inner side of the circle and returns to the center of the circle.

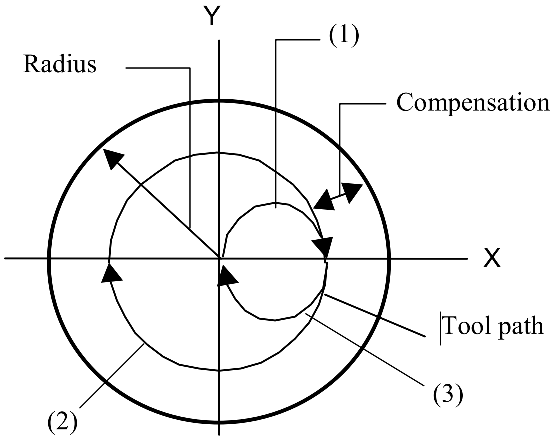
Command format

```
G12I_D_F_;
G13I_D_F_;
```

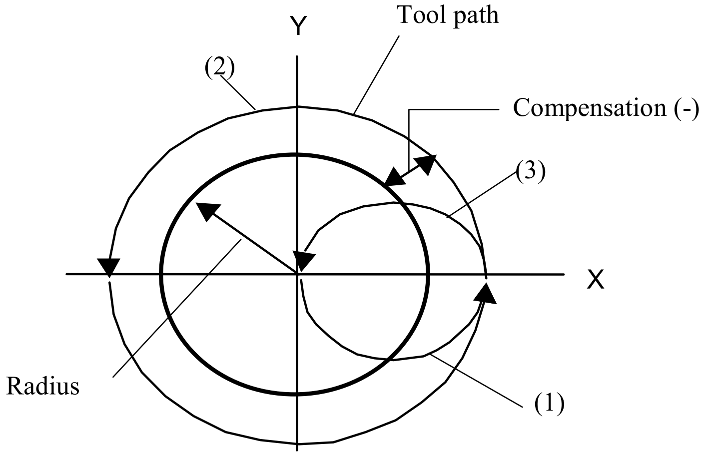
G12	:	Clockwise cutting direction
G13	:	Counterclockwise cutting direction
I	:	Radius of circle + and - symbols are ignored, and the value is always regarded as + (positive).
D	:	Compensation. Set the tool number for compensation. When compensation value is a plus (+), the inner side of the radius specified by command "I" is cut. When compensation value is a minus (-), the outer side of the radius specified by command "I" is cut.
F	:	Cutting speed

[Motion (When X, Y plane selected)]

The tool moves in a circle half the distance from the center of the circle in the X-axis direction.
The rotation direction is specified to G12 or G13.
The tool completes one rotation in the rotation direction specified by G12 or G13 from start point.
It then moves in a circle half the distance from the end point of circle cutting to the center of the circle in the rotation direction specified by G12 or G13.



When G12 is used and the compensation is a positive value.



When G13 is used and the compensation is a negative value.

An alarm will occur when command "D" is omitted.

An alarm will occur when the product of the radius (command "I") minus compensation is zero (0) or a negative value.

An alarm will occur when the circle cutting command (G12, G13) is specified together with the cutter compensation command (G40, G41, G42) (startup or cancel mode).

Corner CR cannot be set for a block that contains the circle cutting command and the block immediately before that block.

An alarm will occur when the radius after cutter compensation is smaller than the tool diameter.

Circle cutting is performed on the plane currently selected (G17, G18, G19).

The start point and end point are the same for circle cutting.

When circle cutting (G12, G13) is executed during cutter compensation (G41, G42), cutter compensation is valid for the path compensated by command "D."

3.6 Plane Selection (G17, G18, G19)

Refer to “3.4. Circular/Helical Interpolation (G02, G03)” for more detail.

3.7 Dwell (G04)

Upon completion of the previous block and in-position check, some time elapses before executing the next block.

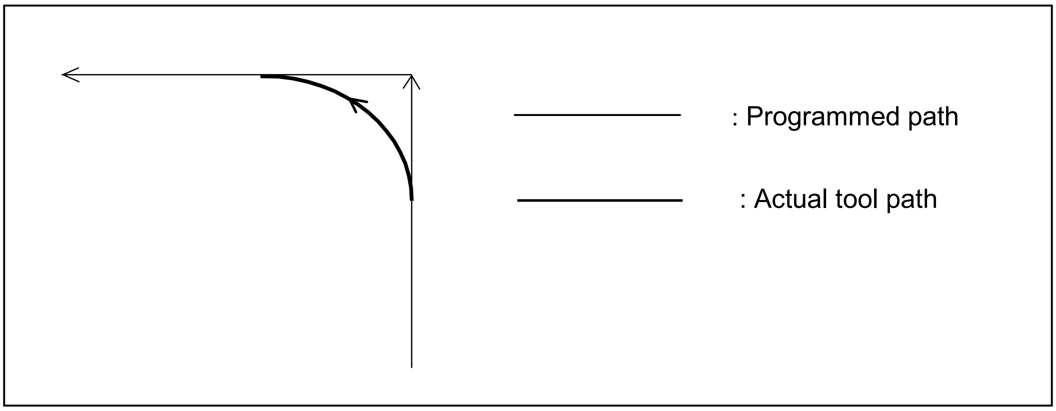
Command format **G04 P_ ;**

G04 X_ ;

P,X : Dwelling time (sec)

3.8 Exact Stop Check (G09, G61, G64)

Since acceleration and deceleration is applied independently to each axis, the actual tool path comes inside the programmed path if each axis speed changes greatly between the former block and the new block in the cutting feed.
The exact stop check is used to solve this problem.



(1) Exact stop check (G09)

Command format **G09 ;**

This command executes an in-position check at the end of a block before proceeding to the next block.

(Note 1) G09 is effective only in the commanded block.
(Note 2) In the positioning mode (G00) the exact stop check function is effective regardless of this command.

(2) Exact stop check mode (G61)

Command format **G61 ;**

After this command is given, the exact stop check function is effective at the end of each block until the cutting mode (G64) is commanded.

(3) Cutting mode (G64)

Command format **G64 ;**

When this command is given, the execution proceeds to the next block without slowing down between the continuing two blocks. This command is effective until G61 is commanded.

(Note 1) Even during the cutting mode (G64), the exact stop check is executed in the blocks in the positioning mode (G00) or in the exact stop check mode (G09), or in the disconnected cutting feed block.

(Note 2)

Old block / New block		Cutting feed	No traveling
Positioning	×	×	×
Cutting feed	×	○	×
No traveling	×	×	×

○ Cutting mode
× Exact stop check mode

When the old block is clamped while the additional axis is traveling, exact stop check is executed. When the new block is unclamped while the additional axis is traveling, exact stop check is executed.

:

3.9 Programmable Data Input (G10)

(1) Input of working zero position

Command format

G10L2Pn X_ Y_ Z_ A_ B_ C_ ;

n=1	**:**	**G54**
n=2	**:**	**G55**
n=3	**:**	**G56**
n=4	**:**	**G57**
n=5	**:**	**G58**
n=6	**:**	**G59**

When the G90 mode (absolute command) is selected, the commanded offset amount becomes newly effective.
When the G91 mode (incremental command) is selected, the commanded offset amount is added to the currently set offset amount to become a renewed offset amount.

When the additional axis is commanded while an optional additional axis is not installed, an alarm will occur.
(Note) Working zero position … “Refer to “Manual Chapter 10”.

(2) Input of tool data

Tool length offset data

G10L10 P_ R_ ;

Tool dia offset data

G10L12 P_ R_ ;

P: offset number
R: offset amount

When the G90 mode (absolute command) is selected, the commanded offset amount becomes newly effective.
When the G91 mode (incremental command) is selected, the commanded offset amount is added to the currently set offset amount to become a renewed offset amount.
(Note) Tool data … Refer to “Manual Chapter 10”.

3

(3) Input of tool fine offset value

When tool length /Tool diameter compensation command is issued using the program, the data of the fine offset number corresponding to the commanded offset number is automatically reflected in operation.

Change of tool fine offset data in program

Command format

```
G10L11   P_ R_    ;
```

```
G10L13   P_ R_    ;
```

L11	:	Fine offset of tool length
L13	:	Fine compensation of tool diameter
P	:	Fine offset No.
		Range : 1~99
R	:	Fine offset amount

The commanded value is added to the compensation amount in absolute mode (G90) nd the preset value in incremental mode (G91).
Setting range +/- 99.999 mm +/- 9.9999 inch

(4) Input of measured working coordinate zero point data.

Command format

```
G10L99 Pn   X_   Y_   Z_   Q_    ;
```

n=1	**:**	**G54**
n=2	**:**	**G55**
n=3	**:**	**G56**
n=4	**:**	**G57**
n=5	**:**	**G58**
n=6	**:**	**G59**
Q	**:**	**The number that stores the measured results.**

After automatic measurement (G121 to G129), set the coordinate system based on the measured position.

Input of additional working coordinate

Command format

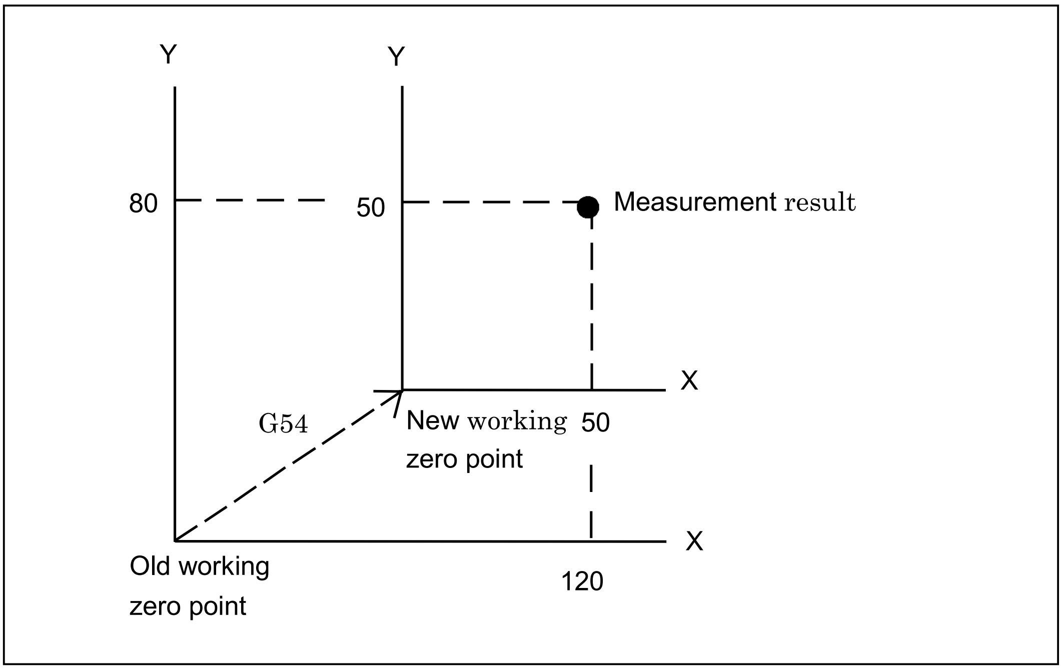
```
G10L99 Pn   X_   Y_   Z_   Q_    ;
```

n	:	Additional working coordinate system (1 to 48).
Q	:	The number that stores the measured results.

Ex.) Assume that automatic measurement is carried out on the G54 coordinate system and the measurement result turned out to be (120, 80). Set the coordinate system that this position will be (50, 50).

(Program)

```
      :
      :
G54 G121 X100. Y100. I20. J20. Z-10. R10. ; (Corner measurement)
G10 L99 X50. Y50. ;
      :
      :
```



eNCPR3.16.ai

(5) Input of tool life.

Command format

G10L97 P_ Q_ R_ W_ ;

P	:	Tool No.
Q	:	Life category
		1 Non counting
		2 Time (Minutes)
		3 Count of hole machining (Hole)
		4 Programs (Turns)
R	:	Life time
W	:	preliminary notice of life time

(Note)
If the G10 code is commanded during the tool dia offset, the tool moves to the point where a vertical vector is formed to the last movement command of X and Y.

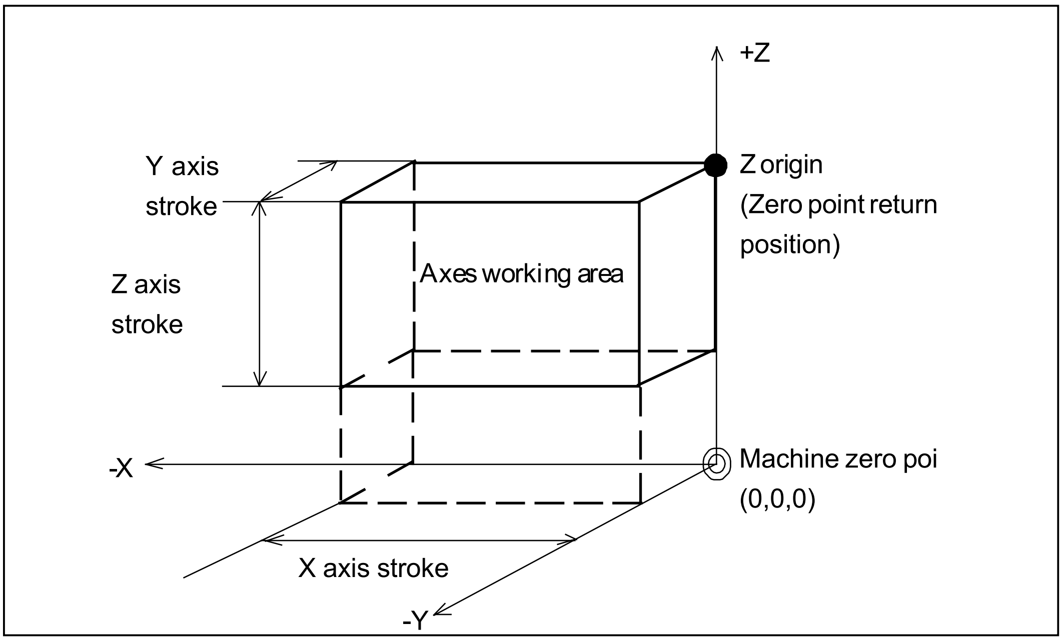
3.10 Soft Limit

The allowable area of the tool motions can be specified in the following three ways.

(1) Stroke setting by the parameter 2
(2) Stroke limit setting by the parameter 1
(3) Programmable stroke limit setting by the G22 code

3.10.1 Stroke

The maximum machine stroke is set by the parameter 2.
This should not be changed by the user



eNCPR3.17.ai

(Note) Z origin is set by the machine parameter.

3.10.2 Stroke limit

The allowable area of the tool motions in each axis of the X, Y and Z is set by the user parameter.

3.10.3 Programmable stroke limit (G22)

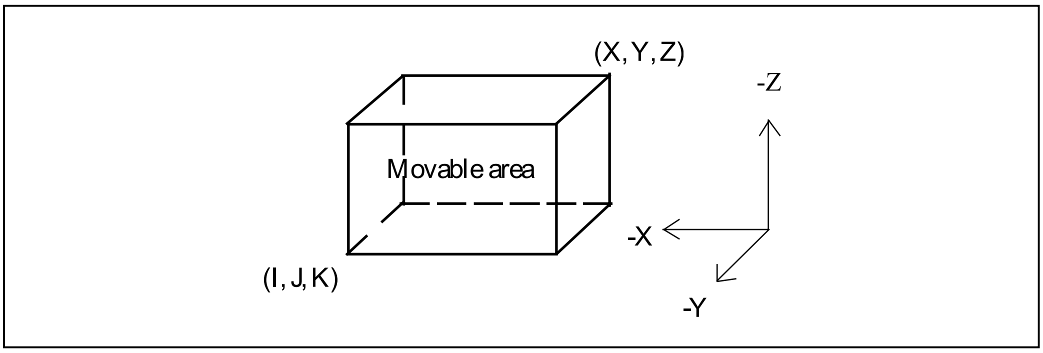
The allowable area of the tool motions is commanded by the program.

Command format

```
G22 X_Y_Z_I_J_K_ ;
```

X : Programmable stroke limit on + direction of X axis.
Y : Programmable stroke limit on + direction of Y axis.
Z : Programmable stroke limit on + direction of Z axis.
I : Programmable stroke limit on - direction of X axis.
J : Programmable stroke limit on - direction of Y axis.
K : Programmable stroke limit on - direction of Z axis.

These are commanded with the coordinate values in the machine coordinate system.
The command is done by the absolute values regardless of the G90 and G91 codes.



eNCPR3.18.ai

(Note 1) **The programmable stroke or the stroke is used as the soft limit in the following ways.**
G22: The programmable stroke is checked as the soft limit.
G23: The stroke is checked as the soft limit.

(Note 2) **Right after turning ON the power, the stroke limit set by the user parameter becomes effective.**
After that, the setting by changing the user parameter or the G22 command whichever is done later becomes effective.
As for the axis which is not specified by the G22 command, the stroke limit set by the user parameter recognized as the command value.
If the stroke limit by the user parameter is changed, however, all the axes which are not changed become as specified by the user parameter.

(Note 3) **The stroke set by the machine parameter is always effective.**

3.11 Return to the Reference Point (G28)

Command format **G28X_Y_Z_A_B_C_;**

This command provides an automatic return to the reference point through an intermediate point for commanded axes. Positioning to the reference point is made through an intermediate point as specified by X_Y_Z_A_B_C_.

It can be 3.12 Selection of machine coordinate system (G53) commanded by either the absolute command (G90) or the incremental command (G91).

The coordinate values of the intermediate point commanded in this block are memorized.

All the commanded axes are moved to the reference point at the rapid traverse rate by way of intermediate point.

(Note 1) As for the coordinate value of the intermediate point, only the values commanded by this G28 block are newly memorized.
The coordinate value of axis not commanded by this G28 block is regarded as that of previous G28 block.

(Note 2) The reference point is set by the user parameter.

(Note 3) A tool motion to the intermediate point or the reference point is done by positioning, and interpolation is not available.

(Note 4) During the single block operation, the block stops at the intermediate point.

(Note 5) The coordinate value of the intermediate point is memorized by the absolute value in the working coordinate system. Therefore, if the working coordinate system is changed after the G28 is commanded, the intermediate point is also changed to the new coordinate system.

(Note 6) When the additional axis is commanded while an optional additional axis is not installed, an alarm will occur.

3.12 Return from the Reference Point (G29)

Command format

G29X_Y_Z_A_B_C_;

This command provides positioning to the commanded position through an intermediate point for commanded axes. At an incremental command, an incremental distance from the intermediate point must be commanded.
The commanded axes are moved to the intermediate point at the rapid traverse rate, then positioned at the commanded point.

(Note 1) **A tool motion to the intermediate point or the commanded point is done by positioning, and interpolation is not available.**
(Note 2) **The tool goes through the intermediate point commanded by the G28 or G30 whichever is given later.**
(Note 3) **During the single block operation, the block stops at the intermediate point.**
(Note 4) **For axes whose intermediate point is not memorized using G28 or G30, the current position is regarded as the center point.**
(Note 5) **When the additional axis is commanded while an optional additional axis is not installed, an alarm will occur.**

3.13 Return to the 2nd to 6th reference point (G30)

Command format

G30P_X_Y_Z_A_B_C_;

P2 : Return to the 2nd reference point
P3 : Return to the 3rd reference point
P4 : Return to the 4th reference point
P5 : Return to the 5 reference point
P6 : Return to the 6th reference point

This command moves the axes to the 2nd, to 6th reference point in the same way as commanded by G28.
The G29 code can be used as the same way as G28.

(Note 1) **The 2nd to 6th reference points are set by the user parameter.**
(Note 2) **When P_ is omitted, return to the 2nd reference point is automatically selected.**
(Note 3) **When the additional axis is commanded while an optional additional axis is not installed, an alarm will occur.**

3.14 Selection of machine coordinate system (G53)

The coordinate values in the machine coordinate system can be commanded in the following ways.

Command format

G53 ;

The coordinate values commanded in the same block as G53 is recognized in the machine coordinate system.
(Note) **When the incremental mode (G91) is selected, the G53 command is ignored.**

3

3.15 Selection of working coordinate system (G54~G59)

When 6 sets of the coordinate systems for each workpiece are set in the data previously, necessary coordinates system can be selected by commanding the G54 through G59 codes.

Command format

```
G54 ┐
 .  │
 .  │        ;
 .  │
G59 ┘
```

G54 : working coordinate system 1
G55 : working coordinate system 2
G56 : working coordinate system 3
G57 : working coordinate system 4
G58 : working coordinate system 5
G59 : working coordinate system 6

3.16 Additional working coordinate system selection (G54.1)

Command format

G54.1 Pn ;

Pn : Specification code for additional working coordinate system.
n : 1~48

The working coordinate system can be selected from 48pairs using the above command.
G54 provides this function instead of G54.1.
Data setting method
1) The data can be confirmed or set on the working coordinate origin screen.
2) The data can be set by commanding G10 in the program.

Command format

G10 L20 Pn X_Y_Z_ ;

Pn : Specification code for additional working coordinate system.
n : 1~48
X,Y,Z :Setting value of workpiece origin offset value

When the absolute mode (G90) is selected, the commanded value is considered the offset value. When the incremental mode (G91) is selected, the commanded value is added to the preset offset value.

3.17 Scaling (G50, G51)

The programmed shape can be enlarged or reduced by the desired scaling factor.
Scaling is possible using the same ratio for all axes or a different ratio for each axis.

Scaling using the same ratio for all axes

Command format **G51X_Y_Z_P_;**

X, Y, Z	: Scaling center coordinate axes (workpiece coordinates)
P	: Scaling factor

Scaling using a different ratio for each axis

Command format **G51X_Y_Z_I_J_K_;**

X, Y, Z	: Scaling center coordinate axes (workpiece coordinates)
IJK	: Scaling factor of XYZ axes

Scaling / Cancel

Command format **G50;**

3

(Note 1) **Do not use other GM codes in a block where G51 is used, or an alarm will occur.**

(Note 2) **Set the scaling type (scaling using the same ratio for all axes or scaling using a different ratio for each axis) for the user parameter.**

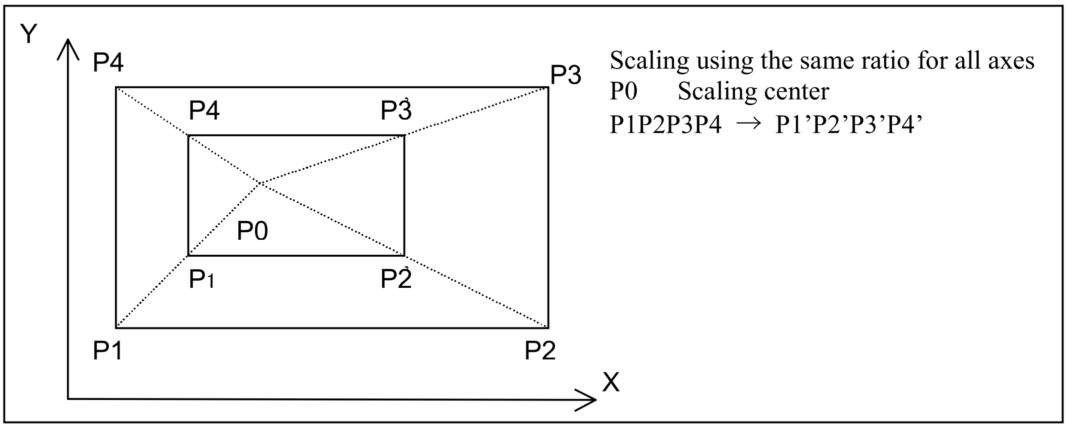
(Note 3) **When the scaling factor command (P or IJK) is omitted, the scaling parameter setting (user parameter 1) is used.**

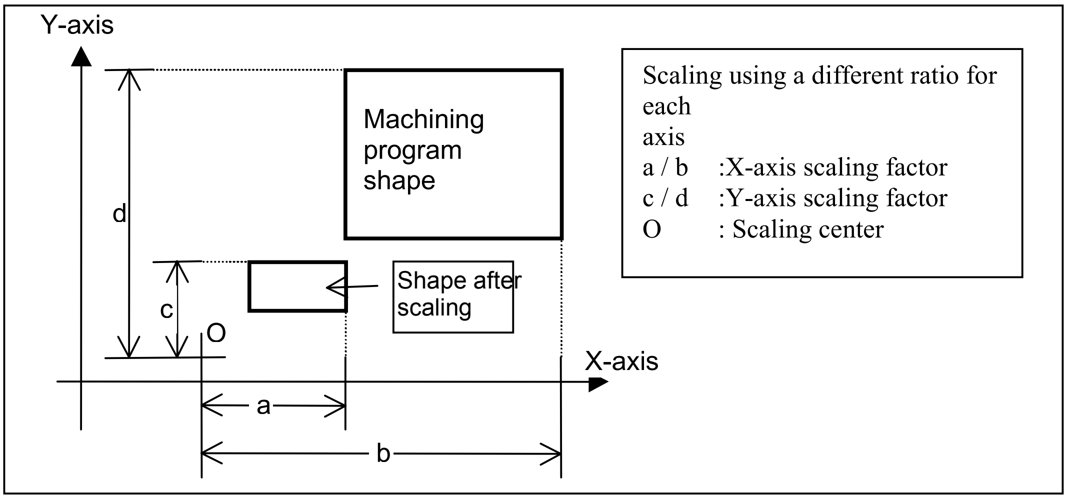
(Note 4) **When the scaling center coordinates (XYZ) are omitted, the tool position when G51 is used is regarded as the center coordinates.**

(Note 5) **Set the scaling factor unit (0.001 or 0.00001) for the parameter.
The valid range of the scaling factor command (P or IJK) or scaling factor parameter is ±1 to ±999999.
Accordingly, the valid scaling range is ±0.001 to ±999.999 or ±0.00001 to ±9.99999.**

(Note 6) **The axis does not travel when scaling start (G51) or scaling cancel (G50) is used.**

Example of scaling using the same ratio for all axes





(Note 1) **An alarm will occur when scaling is used for an axis that has scaling turned off for the parameter.**

(Note 2) **An alarm will occur when circle cutting is specified while a different scaling ratio is set for each axis.**

(Note 3) **Setting a different scaling ratio for each axis in circular interpolation mode does not result in elliptical interpolation.**

(Note 4) **When a different scaling ratio is set for each axis and the radius (R) of the arc is specified in circular interpolation mode, the larger scaling factor of the axes forming the plane on which the arc is drawn is applied to the radius.**

E.g.) Arc using command "R": The left and right command formats are equivalent.

G90 G00X0.Y100.;		G90G00X0.Y100.;
G51X0.Y0.Z0.I2000J1000;	=	
G02X100.Y0.R100.F500;		G02X200.Y0.R200.F500;

(Note 5) **When a different scaling ratio is set for each axis and the center (I, J) of the arc is specified in circular interpolation mode, the distance from the start point to the center (I, J) is not subject to scaling.**

E.g.) Arc using commands "I" "J": The left and right command formats are equivalent.

G90 G00X0.Y100.;		G90 G00X0.Y100.;
G51X0.Y0.I2000J1000;	=	
G02X100.Y0.I0.J-100.F500;		G02X200.Y0.I0.J-100.F500;

Precautions for use of scaling function:

(Note 1) When scaling is invalid

Tool offset set for [Cutter compensation] and [Tool length offset] is not subject to scaling.
Additional axes are not subject to scaling.
An alarm will occur when coordinate transformation (rotational transformation, scaling, programmable mirror image) is performed while the additional axis is selected by the plane selection command (G17, 18, 19).

Axis travel to the R point and ATC zero point specified by the ATC command (G100, M6) is not subject to scaling.
Overrun of single direction positioning (G60) is not subject to scaling.

Scaling is not performed for travel amounts generated through manual intervention.

The following are not subject to scaling in a canned cycle:
Infeed amount "Q" and relief amount "d" of deep hole cycle (G83, G73, G173, G183) XY-axes shift "Q" of fine balling (G76) and back balling (G87).
However, an alarm will occur when the canned cycle is performed while the Z-axis is set for scaling.

(Note 2) Traveling axes when performing scaling or programmable mirror image

When using the scaling or programmable mirror image function, the axis not specified travels according to the specified axis or coordinates.
As a result, the following may occur:

1. **The machine is not operable because the lock signal check is input for an axis not specified.**
2. **The Z-axis travels because the dry run offset is automatically applied.**
3. **An alarm occurs because the specified axis cannot be used.**

(Note 3) Cases when an alarm will occur

An alarm will occur when any reference position return related command (G28 to G30) is used during scaling.

An alarm will occur when any coordinate change command (G10L2/20/98/99, G22 to G23, G52 to G59, G92, G92.1) (external workpiece zero offset) is used during scaling.

An alarm will occur when any automatic workpiece measurement command (G120 to G129) is used during scaling.

An alarm will occur when any of the following is performed during scaling:
Tool change, XY or YZ circular arc (G102/103, 202/203), circular cutting spiral interpolation or conical interpolation

An alarm will occur when a canned cycle is performed while the Z-axis is set for scaling.

An alarm will occur when the amount of travel becomes 0 as a result of scaling.

An alarm will occur when the corner C or R command is used during scaling.

An alarm will occur when scaling is specified in MDI operation.

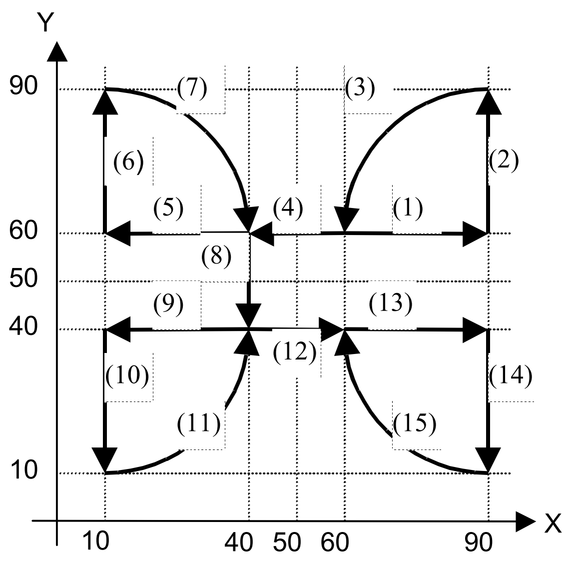
(Note 4) Scaling is cancelled when M02 or M03 is used or operation is reset.

3

Program example of mirror image using scaling function

When a negative number is specified for the scaling factor, programmable mirror image is applied. When a negative value is specified for the scaling factor and there is only one scaling axis, CW and CCW of circular travel will be reversed.

Program example of mirror image using scaling function



Sub program
O9000;
G00G90X60.Y60.;
G01X100.F100;
G01Y100.;
G03X60.Y60.I-30.J-30.;
M99;

Main program
N10 G00G90;
N20 M98P9000;
N30 G51X50.Y50.I-1000J1000;
N40 M98P9000;
N50 G51X50.Y50.I-1000J1000;
N60 M98P9000;
N70 G51X50.Y50.I1000J-1000;
N80 M98P9000;
N90 G50;

Mirror image is applied to scaling center coordinates and programmed path while the mirror image (G51.1) is valid.

3.18 Programmable Mirror Image (G50.1, G51.1)

Mirror image is applied to the program commands for the axes specified in the program.

Mirror image

Command format **G51.1X_Y_Z_;**

Mirror image cancel

Command format **G50.1X_Y_Z_;**

Mirror image setting can be applied simultaneously for the 1st to 3rd axes.

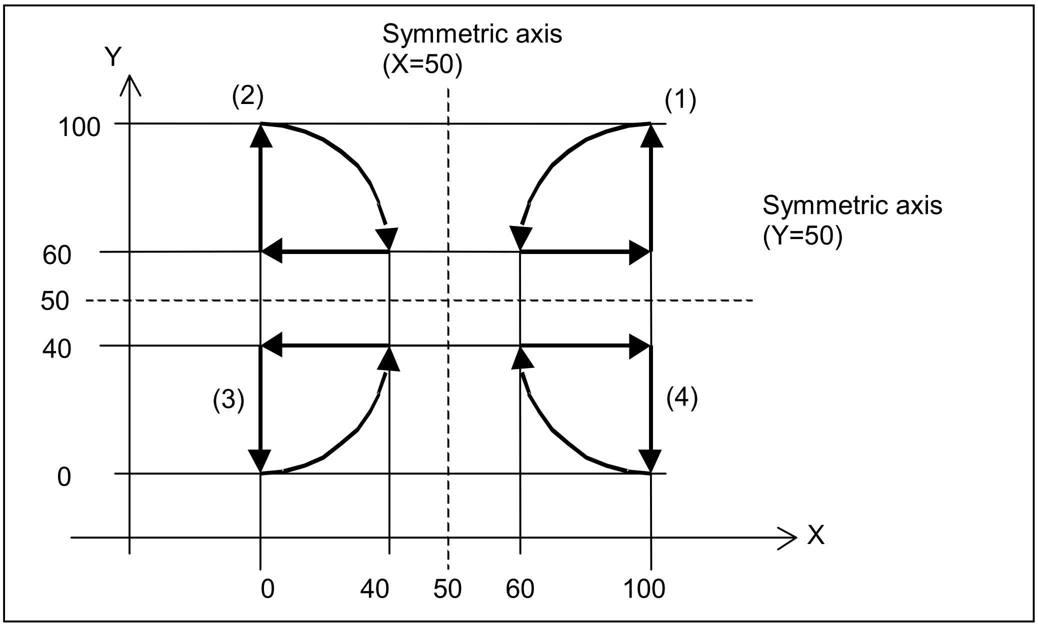
Set the mirror image axis. Omit this for axes about which a mirror image is not created.

Set the mirror image axis in workpiece coordinates.

Using G51.1 command is valid while setting a mirror image. It is regarded as an addition of mirror axes or a change of the mirror axis coordinates.

Set the axis for canceling mirror image to cancel mirror image. Set the coordinates using numerical values.

An alarm will occur when a mirror image is canceled for an axis where mirror image is not set.



(1) Original program command
(2) When mirror axis is set for position X50.
(3) When mirror axis is set for position X50. Y50.
(4) When mirror axis is set for position Y50.

Precautions for use of programmable mirror image:

(Note 1) When programmable mirror image is invalid
Mirror image is not applied to the positioning direction for single direction positioning (G60).

Tool length offset is not subject to mirror image setting compensation.

The spindle rotation direction does not change during mirror image setting.

The thread cutting direction does not change during mirror image setting.

Manual intervention allows the axis travel while ignoring the mirror image setting.
However, when there is a manual interruption during mirror processing, the axis travels according to the path (tool path) after mirror processing.

(Note 2) Traveling axes when performing scaling or programmable mirror image
When using the scaling or programmable mirror image function, the axis not specified travels according to the specified axis or coordinates. As a result, the following may occur:
1. The machine is not operable because the lock signal check is input for an axis not specified.
2. The Z-axis travels because the dry run offset is automatically applied.
3. An alarm occurs because the specified axis cannot be used.

(Note 3) Cases when an alarm will occur
An alarm will occur when mirror image (G50.1 or G51.1) is used during scaling or rotational transformation.

An alarm will occur when coordinate transformation (rotational transformation, scaling, programmable mirror image) is performed while the additional axis is selected by the plane selection function (G17, 18, 19).

An alarm will occur when any reference position return related command (G28 to G30) is used during mirror image setting.

An alarm will occur when any coordinate change command (G10L2/20/98/99, G22 to G23, G52 to G59, G92, G92.1) (external workpiece zero offset) is used during mirror image setting.

An alarm will occur when any automatic workpiece measurement command (G120 to G129, etc.) is used during mirror image setting.

An alarm will occur when any of the following is performed during mirror image setting:
Tool change, XY or YZ circular arc (G102/103, 202/203), circular cutting spiral interpolation or conical interpolation

An alarm will occur when a canned cycle is performed while the Z-axis is set for mirror image.

An alarm will occur when mirror image is specified in MDI operation.

(Note 4) Mirror image is cancelled when M02 or M30 is used or operation is reset.

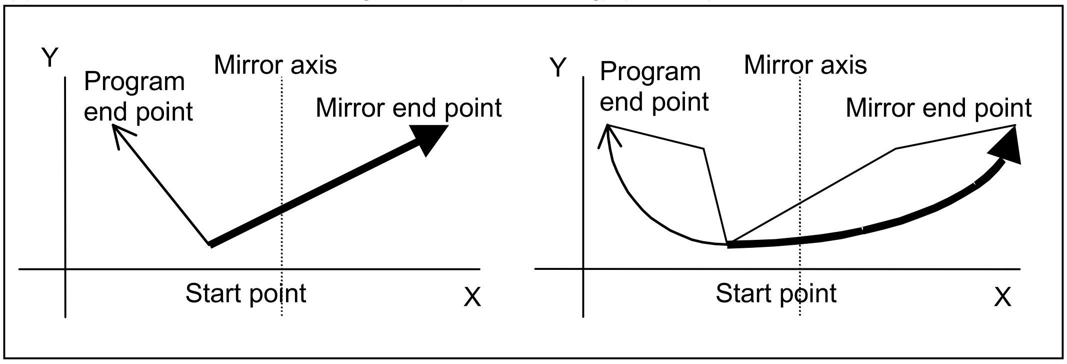
Coordinates are calculated according to the following sequence: mirror, scaling, and then rotational transformation. Accordingly, set these in this order in a program. Set these in the reverse order to cancel previous settings. An alarm will occur when the specified sequence is not followed.

When mirror image is set for only one axis on the selected plane, change the following commands:
Circular interpolation : Rotation direction
Cutter compensation : Compensation direction
Rotational transformation : Rotation direction
Circle cutting : Rotation direction

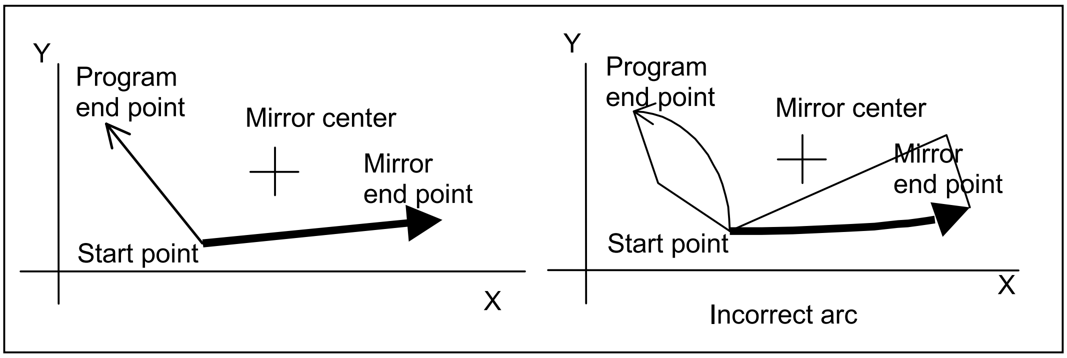
While the mirror image function is enabled, the stroke limit is checked using the coordinates after the mirror image is created.

The axis does not travel while setting or canceling a mirror image.

(1) Axis motion when mirror image is set (1 axis setting) (G90/91)



(2) Axis motion when mirror image is set (2 axes setting) (G90/G91)



3

3.19 Rotational Transformation Function (G68, G69)

The shape specified in the program is rotated.

Rotational transformation

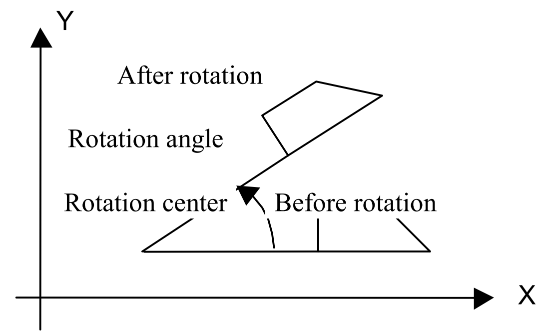
Command format

$$\left\{\begin{matrix}\textbf{G17}\\ \textbf{G18}\\ \textbf{G19}\end{matrix}\right\}\ \textbf{G68}\,\boldsymbol{\alpha}\_\,\boldsymbol{\beta}\_\textbf{R}\_;$$

Rotational transformation cancel

Command format **G69;**

αβ	:	Rotation center coordinates
R	:	Rotation angle (based on CCW)



Plane section command can be omitted. The plane currently selected is valid when it is omitted.

Relationship between selected plane and αβ.

Selected plane	α	β
G17	X	Y
G18	Z	X
G19	Y	Z

Rotation angle (R) is specified within the range of -360.000 to 360.0000 programming mode.

The rotation angle in incremental programming mode is determined in reference to the angle after the previous rotational transformation, and in reference to the α axis when it is the first rotational transformation.

An alarm will occur when any reference position return related command (G27, G28, G29, G30) is used during rotational transformation.

An alarm will occur when ccommand (G52 or G92) during rotational transformation.

An alarm will occur when any automatic workpiece measurement command (G131, G132, G120 to G129) is used during rotational transformation.

An alarm will occur when any plane selection command (G17, G18, G19) is used during rotational transformation.

An alarm will occur when the axes forming the selected plane do not match the axis specified for the rotation transformation center.

An alarm will occur when the rotational transformation command is used during MDI operation.

An alarm will occur when the linear axis (X, Y, Z) and rotation axis (A, B, C) simultaneous interpolation command is used during rotational transformation.

Command "R" cannot be omitted. An alarm will occur when it is omitted.

When the rotational transformation command is used while the mirror image and scaling functions are valid, calculation is performed according to the following sequence:

1. Change of rotational transformation center coordinates due to mirror image function
2. Change of rotation angle direction for rotational transformation when there is only one mirror axis
3. Change of rotational transformation center coordinates due to scaling function

The rotation angle of the rotational transformation is not subject to scaling.

When a rotational transformation is performed in incremental mode, the current tool position is regarded as the start coordinates.

Rotational transformation is cancelled when M02 or M03 is used or operation is reset.

When the center coordinates are omitted for rotational transformation, the coordinates of the spindle’s current position are regarded as the rotation center coordinates.

Even if the rotation center and angle are changed during rotational transformation, rotational transformation using the changed center and angle can be performed without canceling this mode.

Coordinates are calculated according to the following sequence: mirror image, scaling, and then rotational transformation. Accordingly, set them in this order in a program. Set these in the reverse order to cancel previous settings. An alarm will occur when the specified sequence is not followed.

3.20 Coordinate rotation using measured results (G168)

Command format

```
G168 X_Y_Q_;
```

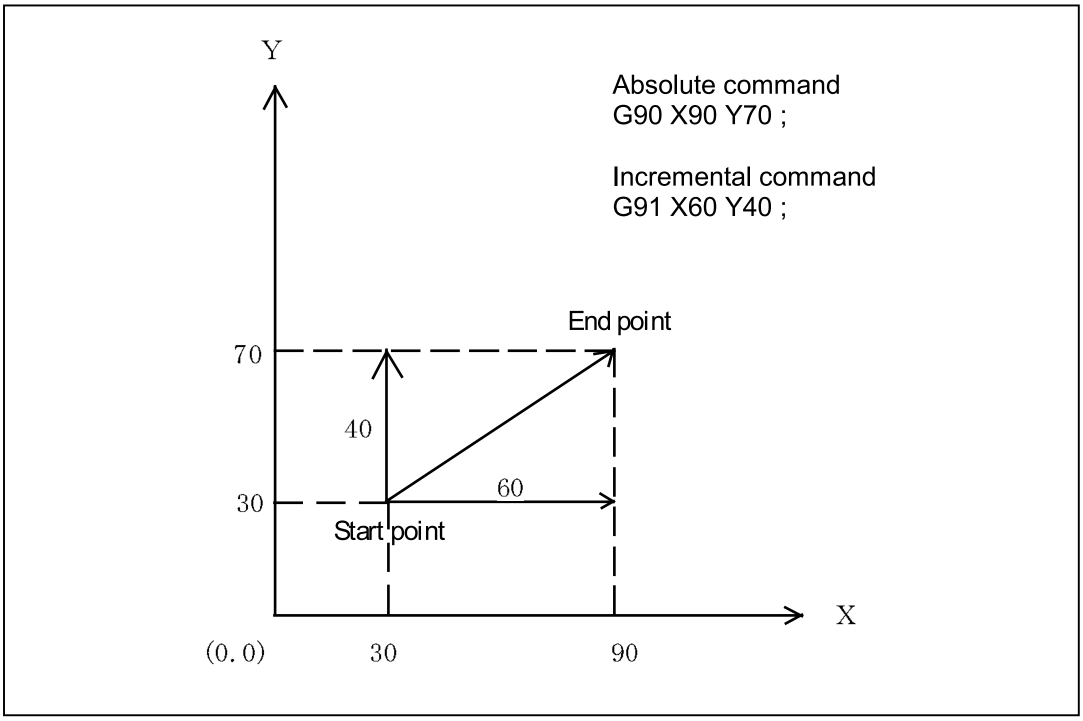
X,Y : Rotation center coordinate value.
Q : Selects the desired measured result by setting "1" to "4".
When the selection is omitted, the setting is considered to be "1".

The coordinate system commanded in the absolute value is always recognized.
When this setting is omitted, the position in which the block has shifted from G69 to G168 (or G68) is considered the center.

The coordinate is rotated using the angle obtained from the measurement.
Other features are the same as those for the coordinate rotation function.

3.21 Absolute command and incremental command (G90, G91)

The axis movement amount can be specified by either the absolute command or the incremental command.
(1) Absolute command (G90)
This is commanded by the G90 code. It specifies an end point of the block in the working coordinate system.
(2) Incremental command (G91)
This is commanded by the G91 code. It specifies a distance from the start point to the end point in the block.



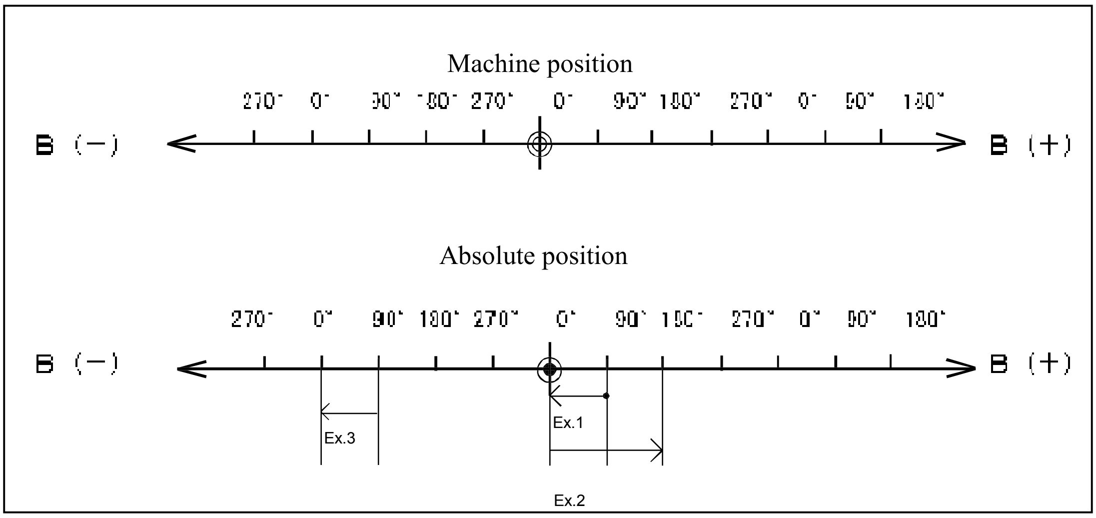
eNCPR3.19.ai

(3) When additional axis is commanded

1. Absolute command (e.g., B axis)

- When B STROKE of user parameter is set to 1: YES, the B axis rotates to the commanded angle.
- When B STROKE of user parameter is set to 0: NO, the B axis rotates in the direction closer to the commanded angle.
  When the commanded angle is the same both in the positive and negative directions (e.g. 180 degrees.), the B axis rotates in the positive direction.
- When B STROKE of user parameter is set to 0: NO, even a larger angle than 360 degrees is commanded, this is handled within 360 degrees.

When B STROKE is set to 0: NO



eNCPR3.20.ai

(ex.1) When B0.000 is entered, the axis rotates 90 degrees in the negative direction
(ex.2) When B180.000 is entered, the axis rotates 180 degrees in the positive direction
(ex.3) When B0.000 is entered, the axis rotates 90 degrees in the negative direction

◎ B-axis machine zero point
◉ B-axis work zero point (Set to 90 degrees in this example)
· B-axis current position before traveling (Angle)

2. Incremental command

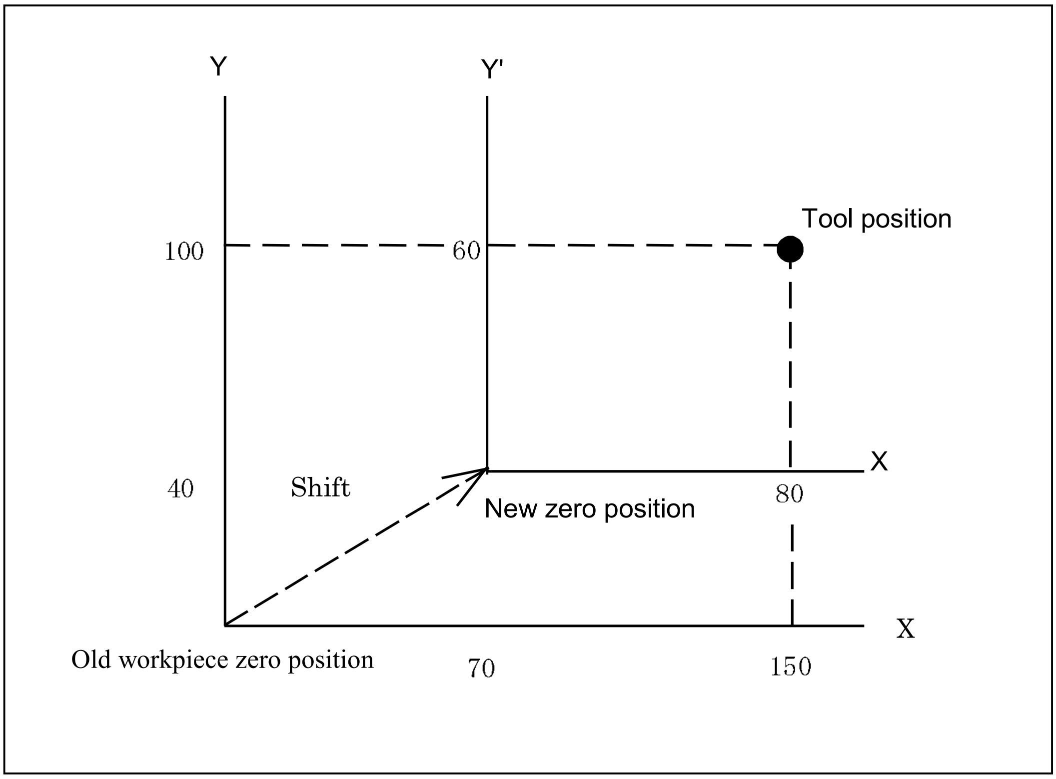
Regardless of the setting of B STROKE (1: YES or 0: NO) of user parameter , the axis rotates for the commanded angle.
However, when B STROKE of user parameter is set to 1: YES, STROKE OVER or LIMIT OVER alarm may occur due to stroke and stroke limit control.

3.22 Change of workpiece coordinate system (G92)

Change of workpiece zero position can be commanded as follows:

Command format	**G92X_Y_Z_A_B_C_;**

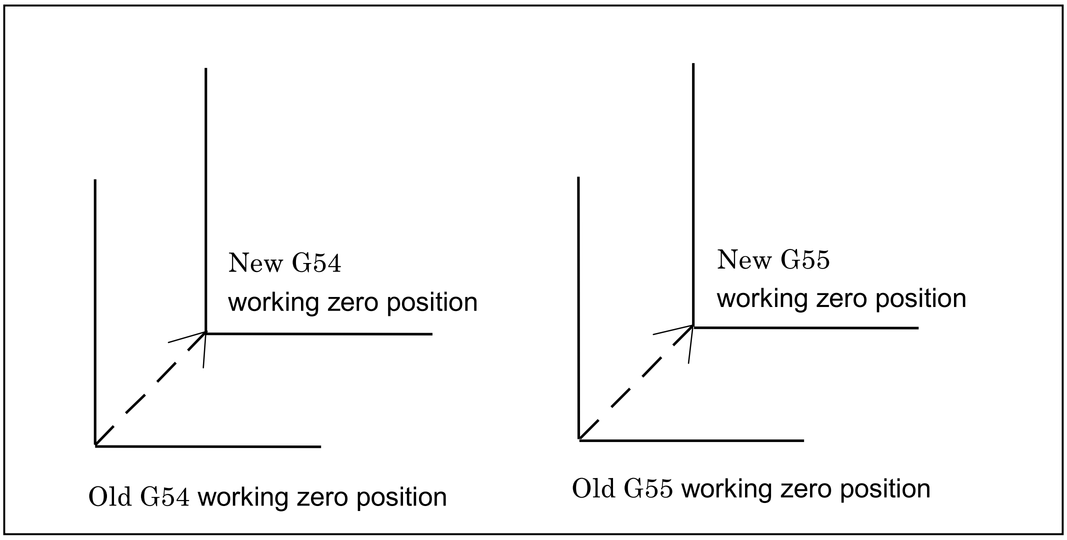
This command shifts the zero position in the working coordinate system so that the current tool position becomes to the commanded coordinate values.



eNCPR3.21.ai

Ex.) The absolute coordinate of the tool position changes to (80, 60) from the current position (150, 100) as commanded "G92 X80. Y60.;"

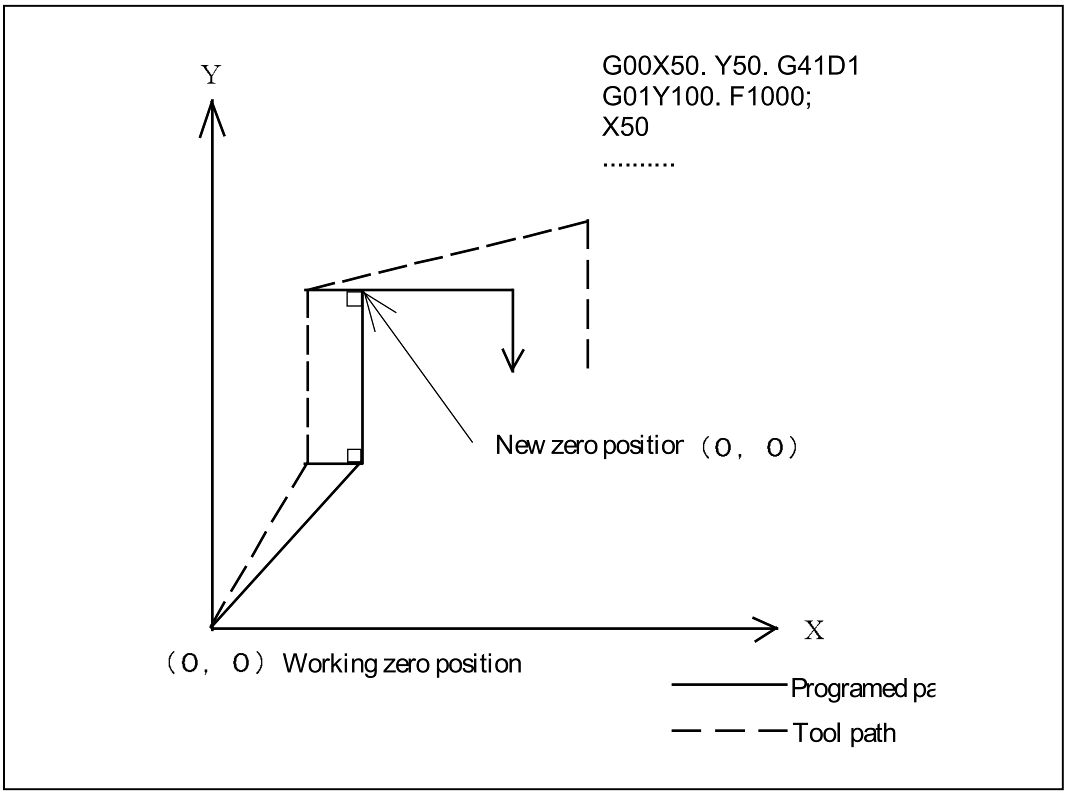
(Note 1) **The commanded coordinate values are always absolute regardless of G90 and G91.**
(Note 2) **The working coordinate values of the not commanded axes do not change.**
(Note 3) **The current working zero position shifts when G92 is executed, and other working zero positions also shift the same amount accordingly.**



eNCPR3.22.ai

In the above figures, G92 is commanded in the coordinate system of G54.
When the working zero position of G54 shifts, the other working zero positions of G55 through G59 also shift the same amount as G54.

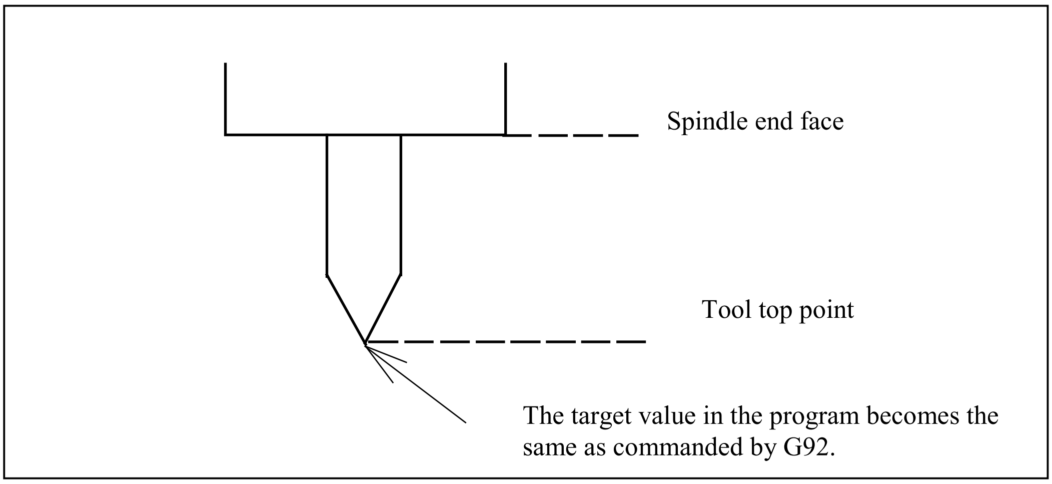
(Note 4) **When G92 is commanded during the tool dia offset, the tool moves to the position where the offset vector is formed vertically to the X/Y movement direction. And the working coordinate system is created with the current position in the program as commanded by G92.**



eNCPR3.23.ai

(Note 5) **When G92 is commanded during the tool length offset, the working coordinate system is created so that the target value of the programmed Z axis becomes the same as commanded by G92.**

3



(Note 6) When the additional axis is commanded while an optional additional axis is not installed, an alarm will occur.

3.23 Skip function (G31,G131,G132)

The tool moves linearly (linear interpolation) at the specified feedrate from the current position to the target position or until the detection signal turns ON.

Command format

```
G31  X_Y_Z_F_  ;
G131 X_Y_Z_F_  ;
G132 X_Y_Z_F_  ;
```

Up to three axes (X,Y,Z) can be controlled simultaneously.
The feedrate is set by address F. Once the feedrate is set, it is effective until another value is specified.

For G131, the SENSOR SIGNAL OFF alarm occurs when the tool has moved to the target position without the detection signal turning ON.
For G31, G132, an alarm does not occur.

As the coordinate value when detective signal turns ON is stored in system variables(#5061~#5063) of the custom macro, it can be used in the custom macro.

Note 1: An alarm occurs when tool dia offset mode is selected.
Note 2: The tool does not move during a dry run state.
Note 3: The tool moves to the target position during a machine lock state.
Note 4: When the detection signal is already ON, the tool stops at the current position.

3.24 Continuous skip function (G31)

The tool moves linearly (linear interpolation) at the specified feedrate from the current position to the target position. If the detection signal turns ON in the meantime, the coordinate value when the detective signal turns ON is stored in the system variables(#5061~#5063) of custom macro.

Command format

```
G31P90X_F_ ;
G31P90Y_F_ ;
G31P90Z_F_ ;
```

Note 1: An alarm occurs when tool dia offset mode is selected.
Note 2: The tool does not move during a dry run state.
Note 3: The tool moves to the target position during a machine lock state. 0.

3.25 Change of tap twisting direction (G133,G134)

Command format

$$\begin{pmatrix} \text{G134} \\ \text{G133} \end{pmatrix} \quad \text{Z\_} \quad \begin{pmatrix} \text{I\_} \\ \text{J\_} \end{pmatrix} \quad \text{S\_;}$$

Commanding G133 and G134 rotates the spindle clockwise and counterclockwise respectively.

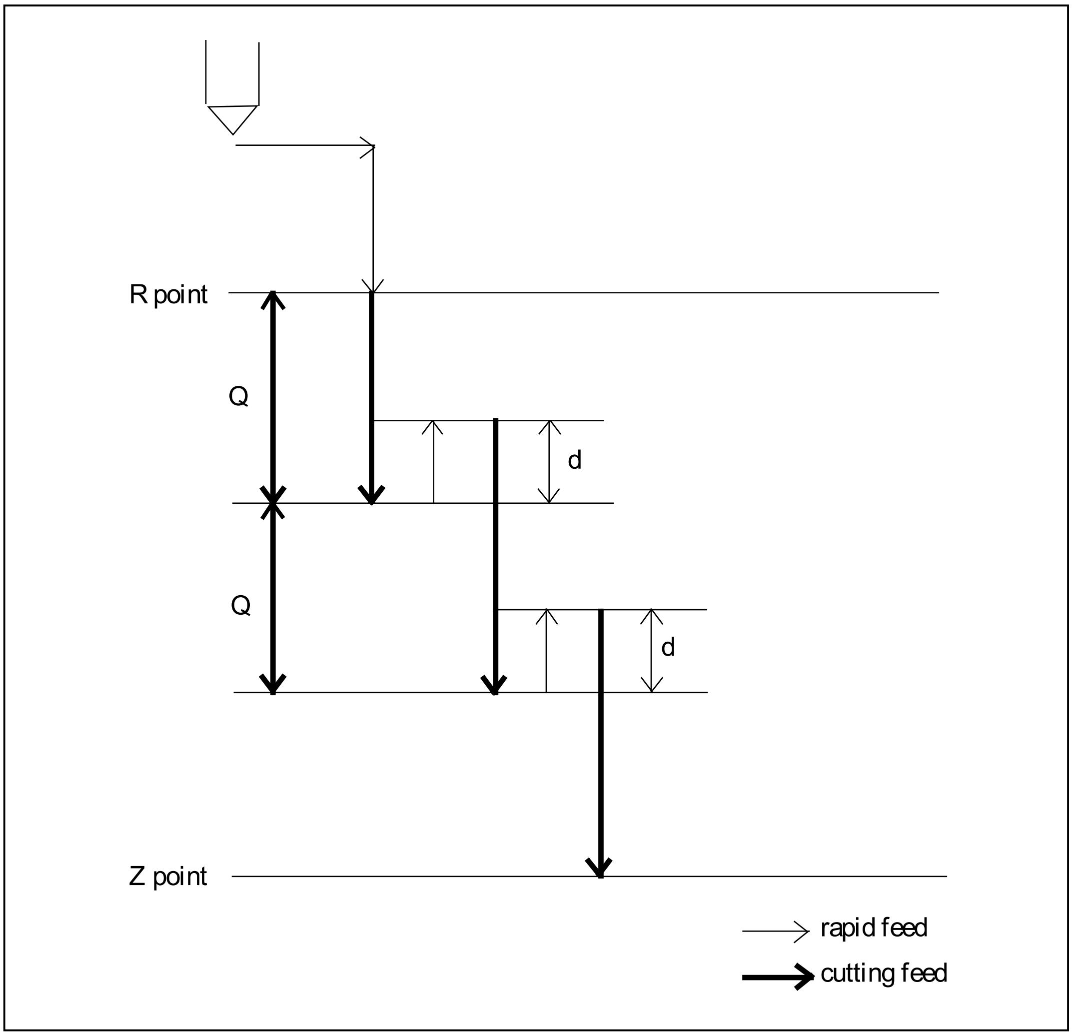
Z:	Z axis target position. Conforms to G90/G91 mode.
I:	Thread pitch
J:	No, of thread
S:	Spindle speed

The Z axis is moved synchronousy with the spindle.
These are one shot G codes.
Command G133/G134 each time even for continuous operation.

3

3.26 High speed peck drilling cycle (G173)

Command format	G173 X _ Y _ Z _ R _ Q _ F _ ;



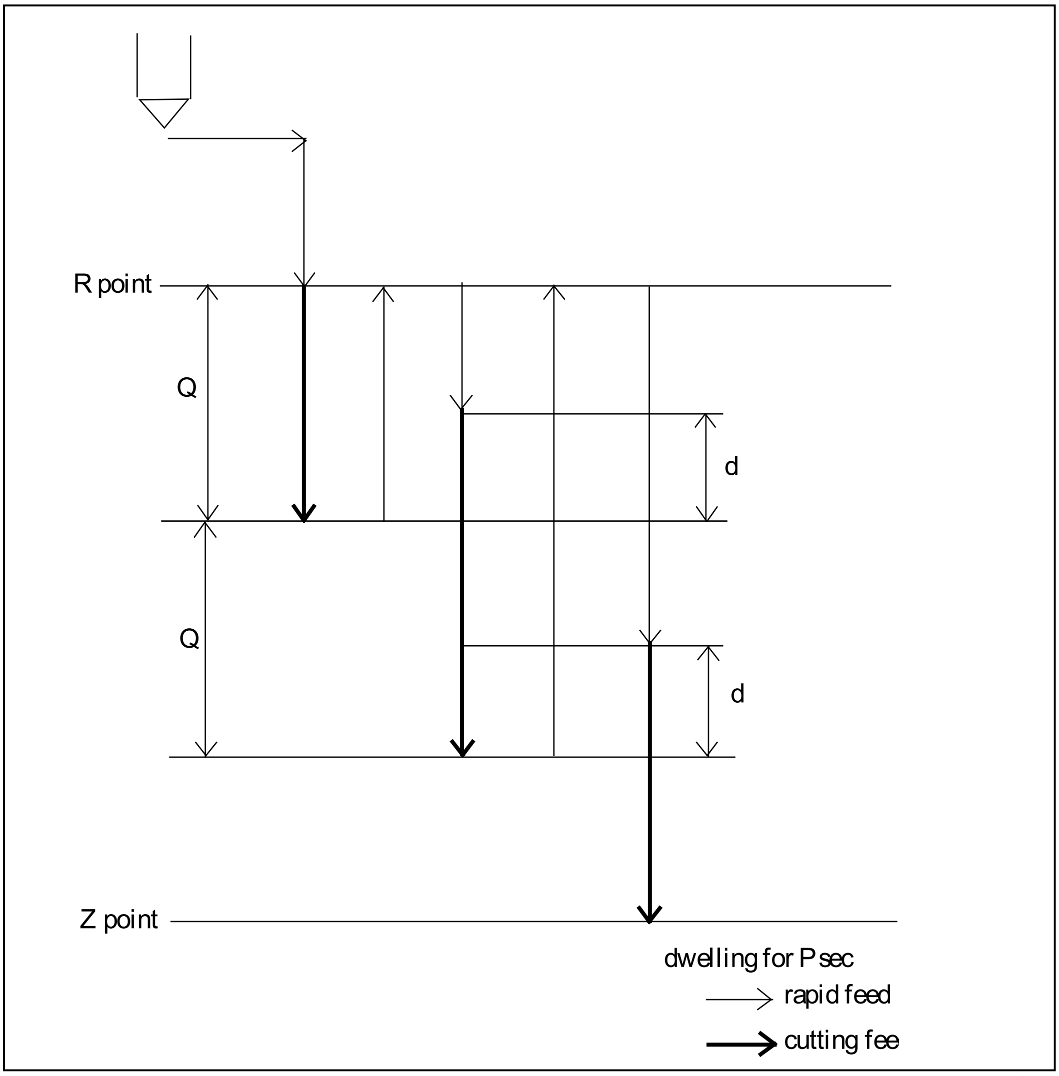
eNCPR5.19.ai

3.27 Peck drilling cycle (G183)

Command format

G183 X _ Y _ Z _ R _ Q _ F _ ;

This is cycle where return operation is removed from G83.



eNCPR5.23.ai

3.28 Local coordinate system function (G52)

Command format

G52 X_ Y_ Z_ A_ B_ C_ ;

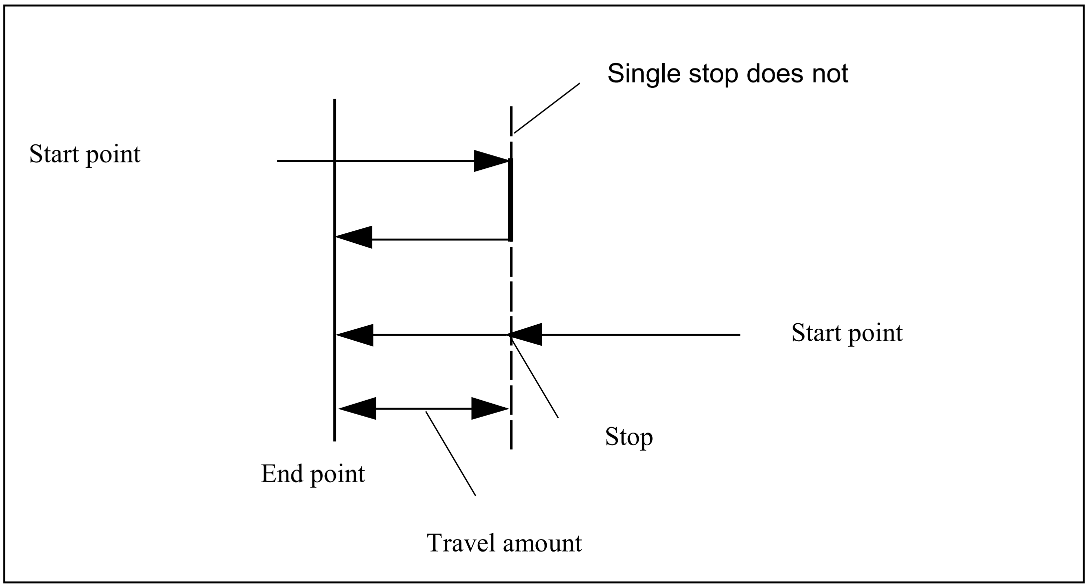
X, Y, Z, A, B, C: Amount of shift from workpiece coordinate zero point
Operation will be the same regardless of G90 or G91.
Amount of shift is applied only to the specified axis.

1) Executing this command creates a local coordinate system in all coordinate systems from G54 to G59.
2) The workpiece coordinate system does not vary even when this command is executed.
3) The local coordinate system of the specified axis is canceled when G92 command is executed.
4) An error will occur when this command is executed during coordinate rotation.
5) When this command is executed during tool compensation, the tool moves to the position where the offset equivalent to the tool diameter is vertically applied to the end point of the previous block.
6) The local coordinate system is canceled when any of the following operations are performed:
   G52 is used to instruct for the command value of the axis.
   G92 is used
   M02 (M30) is used.

3.29 Single direction positioning function (G60)

Command format **G60 X_Y_Z_A_B_C_ ;**

X, Y, Z, A, B, C: Command value of the axis for which single direction positioning is performed.
Coordinate of end point for G90 and travel amount for G91



eNCPR3.25.ai

Operation is reset.
When the above command is executed, the axis moves from the end point for the preset travel amount, and then moves to the end point.
G60 is a one shot command and the axis travel path is the same as that for G00.
The travel amount is set for the user parameter.

1) Single direction positioning is not performed for the Z-axis during a canned cycle, or the XY-axes when they are moving for the preset amount of shift in the G76 and G87 cycles.
2) Single direction positioning is not performed for any axis that does not have the travel amount set for the parameter.
3) Single direction positioning is performed even when 0 is specified for the travel amount.
4) An error will occur when G60 is used during tool compensation.

3

3.30 G code priority

(1) Executed correctly.
(2) Error
(3) The last G command is effective.
(4) One-shot is executed and the modal is updated.
(5) One-shot is executed and the modal is updated, but an error occurs when circle arc is commanded.
(6) Executed when the modal is G0 or G01, but an error occurs when circle arc is commanded.
(7) G22 is executed when G22 is commanded and the model for G0 group is updated.
Both are executed when G23 is commanded.
(8) An error occurs when circular command is output.
(9) An error occurs while circular arc mode is selected.
(10) The one commanded after the block is executed.
When G80 group is executed, the model for G00 group is updated.
When G0 group is executed, G80 group is canceled.
(11) An error occurs, but both are executed when commanded with G80.
(12) One shot execution, modal cancellation.
(13) Executed correctly except when the XZ or YZ arc command is executed.
(14) An error occurs, but both are executed when commanded with G69.
(15) G00 group is executed. G80 is model cancelled.
(16) One shot is executed and the model is updated, but an error occurs when G54P is used.
(17) Both are executed when the G0 group and model updated are simultaneously with G80.
An error occurs when used simultaneously with G54P.
(18) An error occurs when G54P is used.
(19) An error occurs when G102, G103, G202, G203 are used.
(20) Only effect for G17.
(21) An error occurs when G102~G203, without G17 of XY flat selection.
(22) An error occurs when already set to the measurement rotation mode.
Only G17 is able to command for G168.
(23) An error occurs when Z axis is mirror mode.
(24) An error occurs when changing for during the measurement.
(25) G68 is effective. G168 is error.

Command the same block (Model-Model)

	G0 G1	G2 G3	G17	G18 G19	G22	G23	G40	G41 G42	G43 G44	G49	G50	G51	G50.1	G51.1	G54	G54 P	G61	G66	G67	G68 G168	G69	G73	G80	G90	G94	G98	G177
G0,G1	3	3	1	1	7	1	1	1	1	1	2	2	4	4	1	1	1	2	2	4	1	10	10	1	1	1	10
G2,G3	3	3	21	19	7	1	1	1	8	8	2	2	4	19	1	1	1	2	2	19	1	10	10	1	1	1	10
G17			3	3	2	1	1	2	1	1	2	2	2	2	1	1	1	2	2	22	1	1	1	1	1	1	1
G18,G19			3	3	2	1	1	2	1	1	2	2	2	2	1	1	1	2	2	22	1	2	1	1	1	1	2
G22					3	3	2	2	2	2	2	2	2	2	1	1	1	2	2	2	1	2	1	1	1	1	2
G23					3	3	2	2	2	2	2	2	2	2	1	1	1	2	2	2	1	2	1	1	1	1	2
G40							3	3	1	1	2	2	2	2	1	1	1	2	2	2	1	2	1	1	1	1	2
G41,G42							3	3	1	1	2	2	2	2	1	1	1	2	2	2	2	2	1	1	1	1	2
G43,G44									3	3	2	2	2	2	1	1	1	2	2	2	2	1	1	1	1	1	1
G49									3	3	2	2	2	2	1	1	1	2	2	2	1	1	1	1	1	1	1
G50											3	3	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
G51											3	3	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
G50.1													3	3	2	2	1	2	2	2	2	2	2	1	1	1	2
G51.1													3	3	2	2	1	2	2	2	2	2	2	1	1	1	2
G54															3	3	1	2	2	1	1	1	1	1	1	1	1
G54 P															3	3	1	2	2	1	1	2	1	1	1	1	2
G61																	3	2	2	1	1	1	1	1	1	1	1
G66																		3	2	2	2	2	2	2	2	2	2
G67																			3	2	2	2	2	2	2	2	2
G68,G168																				3	3	2	1	1	1	1	2
G69																				3	3	1	1	1	1	1	1
G73																						3	3	1	1	1	3
G80																						3	3	1	1	1	3
G90																								3	1	1	1
G94																									3	1	1
G98																										3	1
G177																											3

3

3

Command the same block (Model-One-shot)

	G0 G1	G2 G3	G17	G18 G19	G22	G23	G40	G41 G42	G43 G44	G49	G50	G51	G50.1	G51.1	G54	G54 P	G61	G66	G67	G68 G168	G69	G73	G80	G90	G94	G98	G177
G4	4	4	2	2	2	2	2	2	2	2	2	2	2	2	1	2	1	2	2	2	1	2	1	1	1	1	2
G9	1	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	1	2	2	1	1	1	1	1	1	1	1
G10	4	4	2	2	2	2	2	2	2	2	2	2	2	2	1	2	1	2	2	2	1	2	1	1	1	1	2
G12	4	4	2	2	2	2	2	2	2	2	2	2	2	2	1	1	1	2	2	1	1	2	1	1	1	1	2
G28	4	4	2	2	2	2	2	2	1	1	2	2	2	2	1	2	1	2	2	2	1	2	4	1	1	1	2
G30	4	4	2	2	2	2	2	2	1	1	2	2	2	2	1	2	1	2	2	2	1	2	4	1	1	1	2
G31	4	4	2	2	2	2	2	2	1	1	2	2	2	2	1	2	1	2	2	2	1	2	4	1	1	1	2
G36	4	2	20	2	2	2	2	2	1	1	2	2	2	2	1	2	1	2	2	2	1	2	4	1	1	1	2
G52	4	4	2	2	2	2	2	2	2	2	2	2	2	2	1	1	1	2	2	2	1	2	4	1	1	1	2
G53	1	1	1	1	1	1	1	1	1	1	2	2	2	2	4	4	1	2	2	1	1	1	1	1	1	1	1
G60	4	4	1	1	2	2	2	2	1	1	2	2	2	2	1	1	1	2	2	2	1	1	4	1	1	1	1
G65	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2	2
G92	4	4	2	2	2	2	2	2	2	2	2	2	2	2	1	1	1	2	2	2	1	2	4	1	1	1	2
G100	4	4	2	2	2	2	1	20	1	1	2	2	2	2	1	1	1	2	2	2	1	2	4	1	1	1	2
G120	4	4	2	2	2	2	2	2	1	1	2	2	2	2	1	1	1	2	2	2	1	2	4	1	1	1	2
G121	4	2	20	2	2	2	2	2	1	1	2	2	2	2	1	1	1	2	2	2	1	2	4	1	1	1	2
G131	4	4	2	2	2	2	2	2	1	1	2	2	2	2	1	1	1	2	2	2	1	2	4	1	1	1	2
G133	4	4	20	2	2	2	2	2	2	2	2	2	2	2	1	1	1	2	2	2	1	2	4	1	1	1	2
G173	10	10	20	2	2	2	2	2	1	1	2	2	2	2	1	2	1	2	2	2	1	3	3	1	1	1	3

Command the same block (One-shot -One-shot)

	G4	G9	G10	G12	G28	G30	G31	G36	G52	G53	G60	G65	G92	G100	G120	G121	G131	G133	G173
G4	3	1	2	2	2	2	2	2	2	1	2	2	2	2	2	2	2	2	2
G9		3	1	1	1	1	1	1	1	1	1	2	1	1	1	1	1	1	1
G10			3	2	2	2	2	2	2	1	2	2	2	2	2	2	2	2	2
G12				3	2	2	2	2	2	1	2	2	2	2	2	2	2	2	2
G28					3	3	2	2	2	1	2	2	2	2	2	2	2	2	2
G30					3	3	2	2	2	1	2	2	2	2	2	2	2	2	2
G31							3	2	2	1	2	2	2	2	2	2	3	2	2
G36								3	2	1	1	2	2	2	2	2	2	2	2
G52									3	1	2	2	2	2	2	2	2	2	2
G53										3	1	2	1	1	1	1	1	1	1
G60											3	2	2	2	2	2	2	2	1
G65												3	2	2	2	2	2	2	2
G92													3	2	2	2	2	2	2
G100														3	2	2	2	2	2
G120															3	2	2	2	2
G121																3	2	2	2
G131																	3	2	2
G133																		3	2
G173																			3

3

Command during Model

	G0 G1	G2 G3	G17	G18 G19	G22	G23	G40	G41 G42	G43 G44	G49	G50	G51	G50.1	G51.1	G54	G54.1	G61	G66	G67	G68 G168	G69	G73	G80	G90	G94	G98	G177
G0,G1			1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	15	1	1	1	1	15
G2,G3			1	19	1	1	1	19	1	1	1	19	1	19	1	1	1	1	1	19	1	15	1	1	1	1	15
G17	1	1			1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	2	1	1	1	1	1	1	1
G18,G19	1	1			1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	2	1	2	1	1	1	1	2
G22	1	1	1	1			1	1	1	1	1	2	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1
G23	1	1	1	1			1	1	1	1	1	2	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1
G40	1	1	1	1	1	1			1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
G41,G42	1	1	1	1	1	1			1	1	1	1	1	1	1	1	1	2	1	1	1	2	1	1	1	1	2
G43,G44	1	1	1	1	1	1	1	1			1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
G49	1	1	1	1	1	1	1	1			1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
G50	1	1	1	1	1	1	1	1	1	1			1	1	1	1	1	1	1	2	1	1	1	1	1	1	1
G51	1	1	1	1	1	1	1	1	1	1			1	1	1	1	1	1	1	2	1	1	1	1	1	1	1
G50.1	1	1	1	1	1	1	1	1	1	1	1	2			1	1	1	1	1	2	1	1	1	1	1	1	1
G51.1	1	1	1	1	1	1	1	1	1	1	1	2			1	1	1	1	1	2	1	1	1	1	1	1	1
G54	1	1	1	1	1	1	1	1	1	1	1	2	1	2			1	1	1	1	1	1	1	1	1	1	1
G54.1	1	1	1	1	1	1	1	1	1	1	1	2	1	2			1	1	1	1	1	1	1	1	1	1	1
G61	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1		1	1	1	1	1	1	1	1	1	1
G66	1	1	1	1	1	1	1	2	1	1	1	1	1	1	1	1	1			1	1	1	1	1	1	1	1
G67	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1			1	1	1	1	1	1	1	1
G68,G168	1	1	1	25	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1			1	1	1	1	1	1
G69	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1			1	1	1	1	1	1
G73	1	1	1	2	1	1	1	2	1	1	1	1	1	23	1	1	1	1	1	1	1			1	1	1	
G80	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1			1	1	1	
G90	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1		1	1	1
G94	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1		1	1
G98	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1		1
G177	1	1	1	2	1	1	1	2	1	1	1	1	1	23	1	1	1	1	1	1	1			1	1	1	
G4	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
G9	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
G10	1	1	1	1	1	1	1	1	1	1	1	24	1	24	1	1	1	1	1	1	1	1	1	1	1	1	1
G12	1	1	1	1	1	1	1	1	1	1	1	2	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1
G28	1	1	1	1	1	1	1	2	1	1	1	2	1	2	1	1	1	1	1	2	1	1	1	1	1	1	1
G30	1	1	1	1	1	1	1	2	1	1	1	2	1	2	1	1	1	1	1	2	1	1	1	1	1	1	1
G31	1	1	1	1	1	1	1	2	1	1	1	2	1	2	1	1	1	1	1	2	1	2	1	1	1	1	2
G36	1	2	1	2	1	1	1	2	1	1	1	2	1	2	1	1	1	1	1	2	1	1	1	1	1	1	1
G52	1	1	1	1	1	1	1	1	1	1	1	2	1	2	1	1	1	1	1	2	1	1	1	1	1	1	1
G53	1	1	1	1	1	1	1	1	1	1	1	2	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1
G60	1	1	1	1	1	1	1	1	1	1	1	2	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1
G65	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
G92	1	1	1	1	1	1	1	1	1	1	1	2	1	2	1	1	1	1	1	2	1	1	1	1	1	1	1
G100	1	1	1	1	1	1	1	1	1	1	1	2	1	2	1	1	1	1	1	1	1	12	1	1	1	1	12
G120	1	1	1	1	1	1	1	2	1	1	1	2	1	2	1	1	1	1	1	2	1	1	1	1	1	1	1
G121	1	2	1	2	1	1	1	2	1	1	1	2	1	2	1	1	1	1	1	2	1	1	1	1	1	1	1
G131	1	1	1	1	1	1	1	2	1	1	1	2	1	2	1	1	1	1	1	2	1	2	1	1	1	1	2
G133	1	1	1	2	1	1	1	2	1	1	1	1	1	23	1	1	1	1	1	1	1	1	1	1	1	1	1
G173	1	1	1	2	1	1	1	2	1	1	1	1	1	23	1	1	1	1	1	1	1	1	1	1	1	1	1

CHAPTER 4

PREPARATOION FUNCTION (TOOL OFFSET FUNCTION)

4

4.1 Tool dia offset (G40, G41, G42)

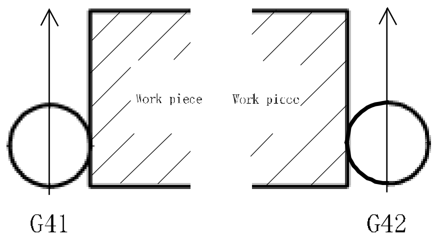
4.1.1 Tool dia offset function

Programming is done according to the actual workpiece form, but this function enables the tool to move along the path with an offset from actual workpiece form, which is equivalent to the used tool radius.

Command format

$$\left\{\begin{matrix} \text{G41} \\ \text{G42} \end{matrix}\right\} \quad \text{Dn;}$$

G codes and D code used for tool dia offset
G40 : Tool dia offset cancel (Effective at power ON)
G41 : Left offset along tool path
G42 : Right offset along tool path
G41 and G42 command an offset mode, while G40 commands a cancel of the offset mode.



eNCPR4.01.ai

Dn : Tool offset number (n=0~99)
The offset amount of D0 is always zero.
The offset amount is set on the tool data setting screen.

(Note1) **Refer to "Chapter 10" in the Instruction Manual for details of the tool data setting screen.**

(Note2) **When a command without X and Y axis travel of more than three blocks or a command with a travel amount of zero (0) is given in tool dia. offset mode, excessive cutting or insufficient cutting may occur, respectively.**

4.1.1.1 Tool dia fine compensation

When G41and G42 are commanded in the program, the tool dlameter fine compensation value corresponding to the commanded tool number is added tothetool diameter compensation value. The tooldiameter fine compensation value is placed on the tool list screen.

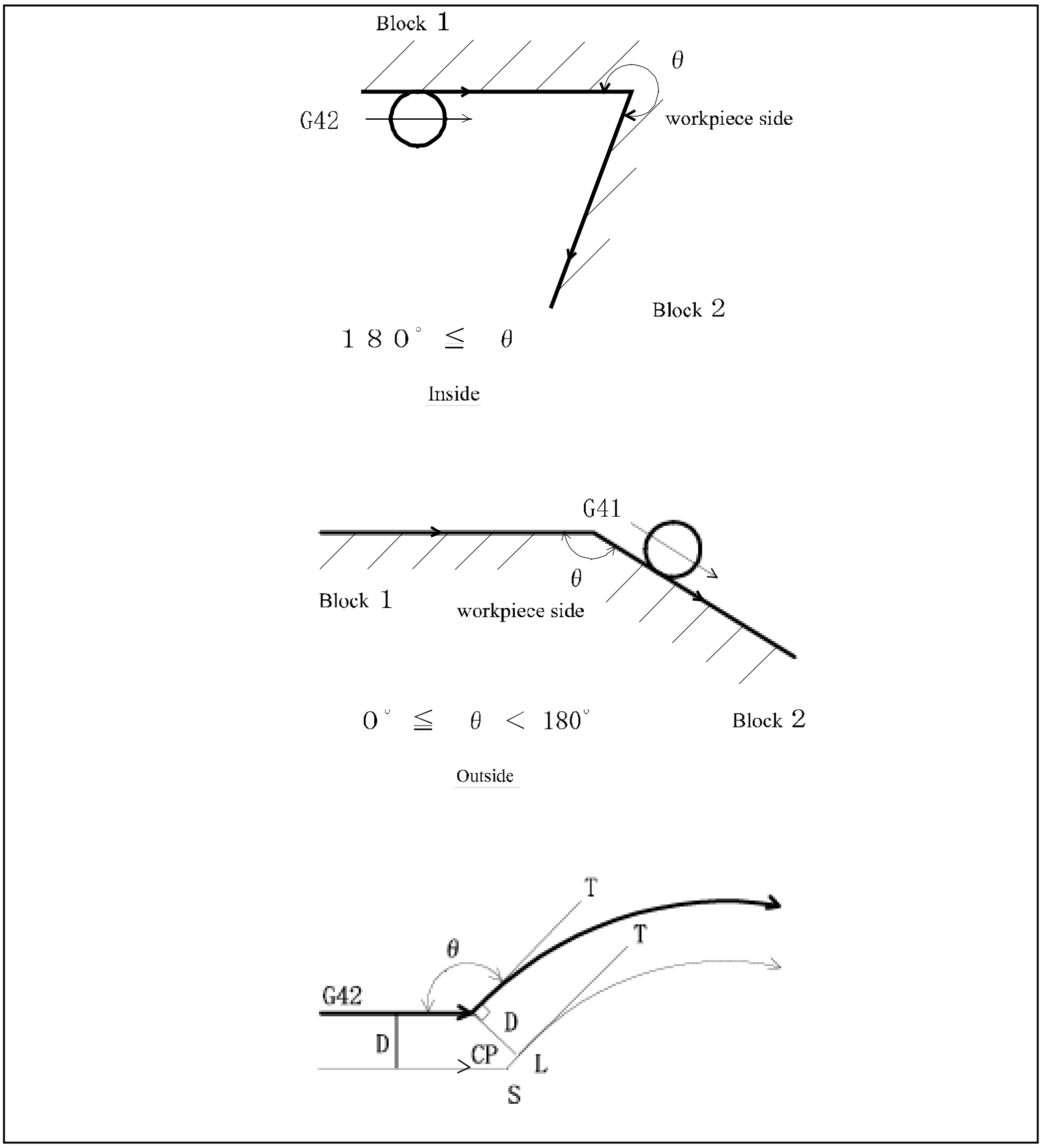
4.1.2 Cancel mode

The system enters the cancel mode right after the power is turned ON or the **[RESET]** key is pressed.
In the cancel mode, the path of the tool center coincides with the programmed path.

Terms and symboles for tool dia offset

1. Inside and outside

If the angle measured on workpiece side is larger than 180 −, it is called "Inside".
If the angle measured on workpiece side is smaller than 180 −, it is called "Outside".



eNCPR4.02.ai

	:	Programmed path
	:	Tool center path
	:	Auxiliary line
L	:	Linear lin
C	:	Circular line
D	:	Tool dia offset amount
θ	:	Tool dia offset angle
T	:	Circular tangent
CP	:	Cross point
S	:	Single block stop point

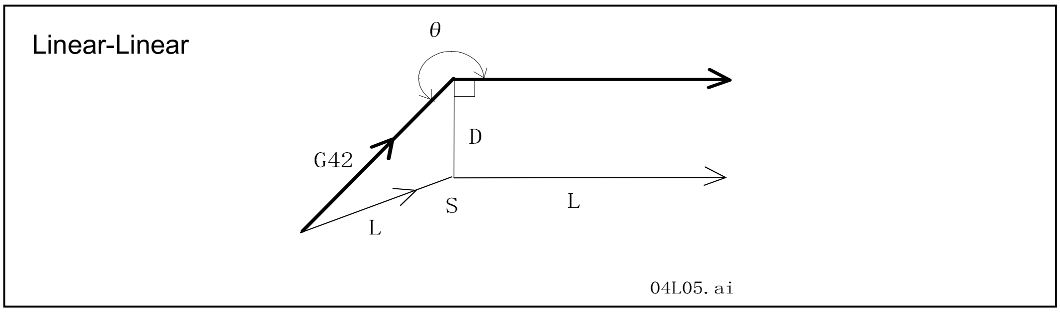
4.1.3 Start-up

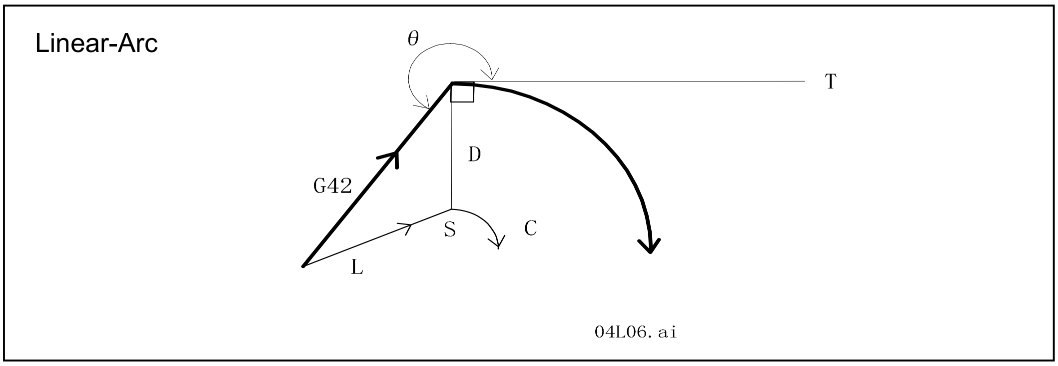
When a block which satisfies all the following conditions is executed in the cancel mode, the system enters the offset mode. The control in this operation is called the start-up.

a) G41 or G42 is commanded.
b) The tool offset number is not zero.
b) The movement command other than circular arc (G02 or G03) is given on the
X-Y plane,and the movement distance is not zero.

(Note 1) In the case of circular arc command, an alarm is generated.
(Note 2) Command the G0, G1, G2, or G3 first before command the G41/G42.

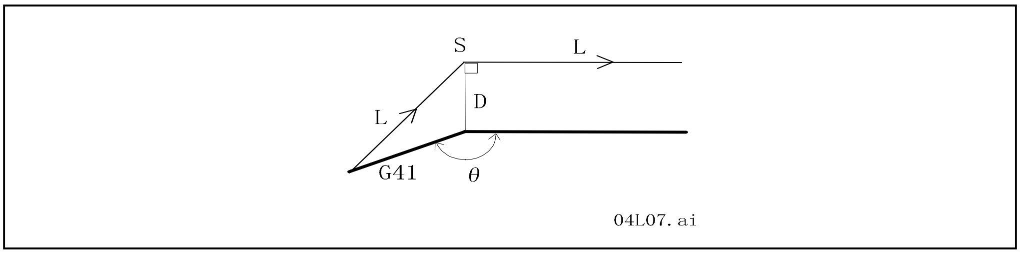
4.1.3.1 Inside cutting (180 ≤ θ)



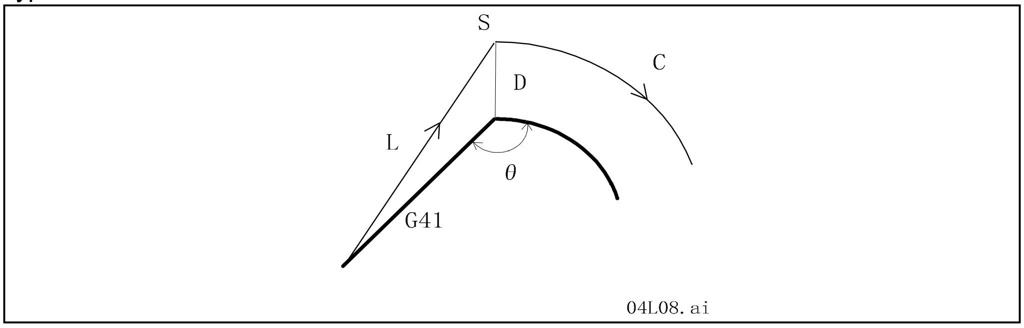


4.1.3.2 Outside cutting

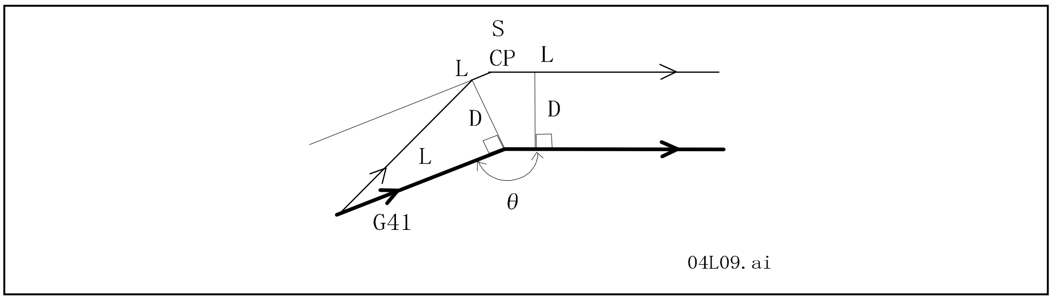
(a)Type 1 : Linear - Linear



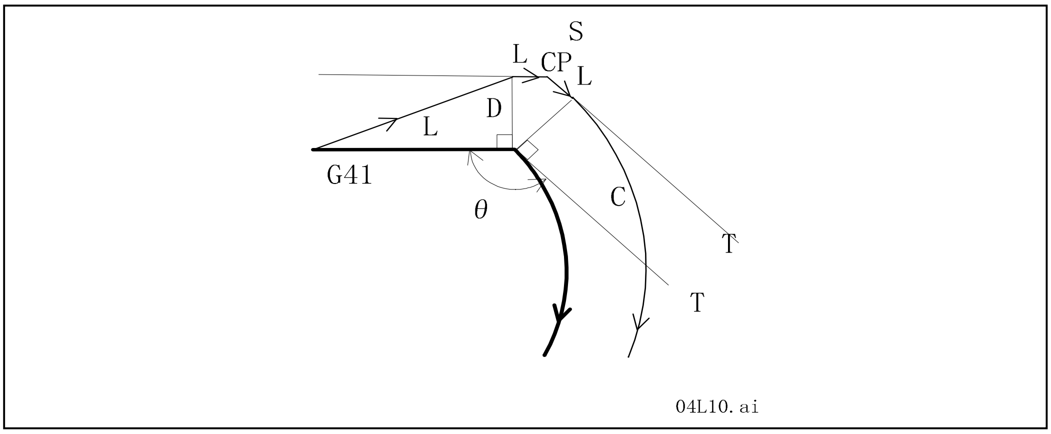
Type 1 : Linear - Arc



(b)Type 2 : Linear - Linear



Type 2 : Linear - Arc



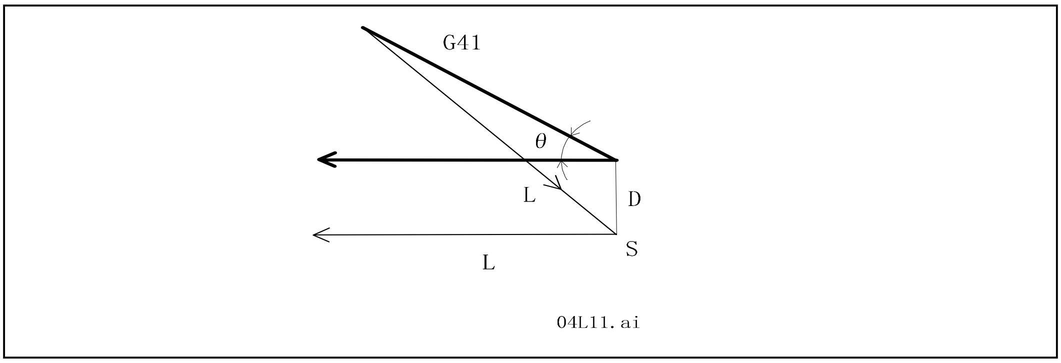
(Note 1) **Type 1 and 2 can be selected in parameter 1 for start-up and cancel motions.**

(Note 2) **If the angle is close to 180˚ (179˚ ≤ θ < 180˚) while type 2 is being selected,actual movement will be type 1.**

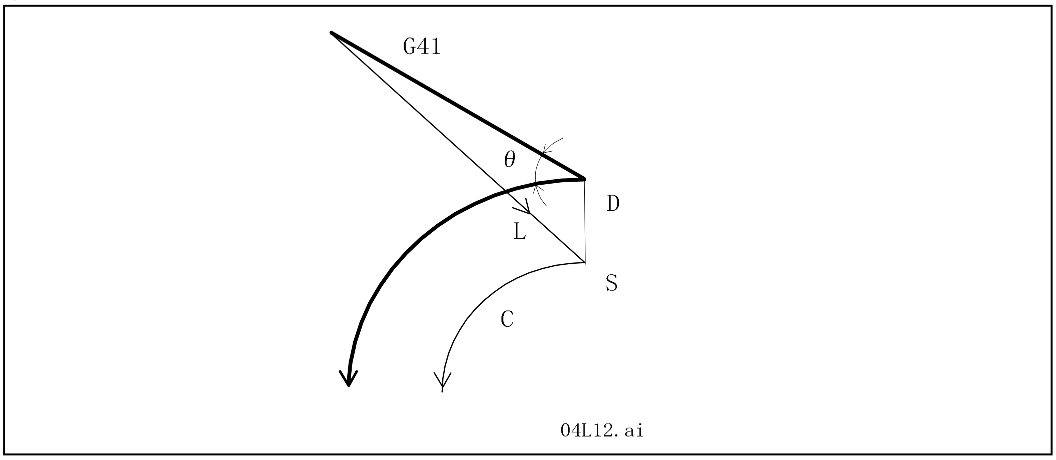
4

4.1.3.3 Outside cutting (θ < 90°)

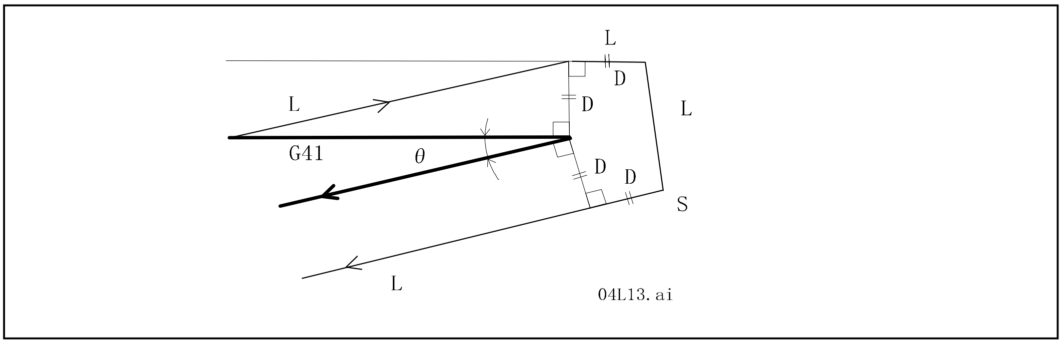
(a)Type 1 : Linear - Linear



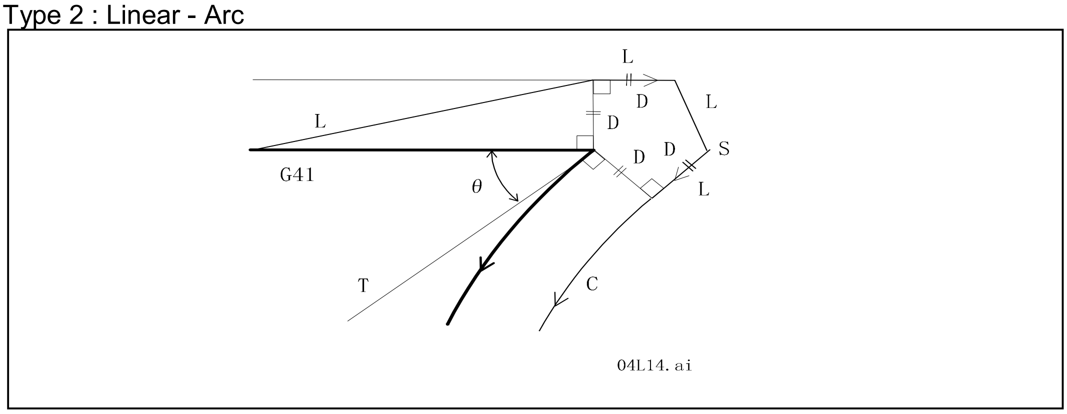
Type 1 : Linear - Arc



(b)Type 2 : Linear - Linear



Type 2 : Linear - Arc



(Note 1) **Type 1 and 2 can be selected in parameter 1 for start-up and cancel motions.**

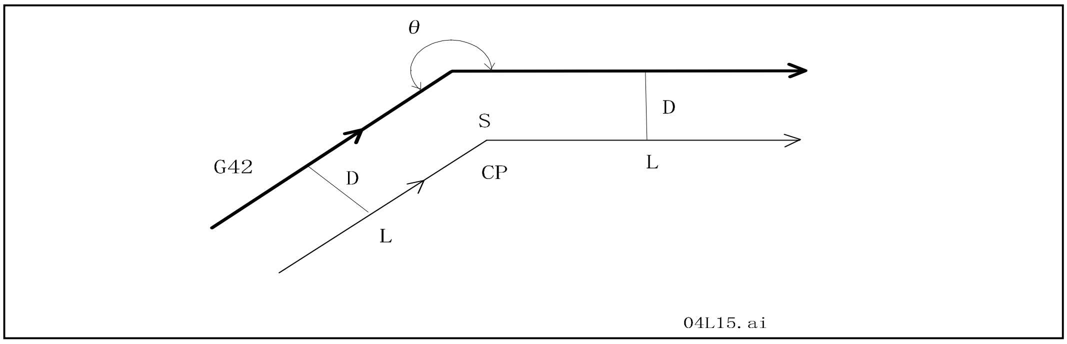
(Note 2) **If the angle is close to 1°(θ≤1°) while type 2 is being selected, actualmovement will be type 1.**

4.1.4 Offset mode

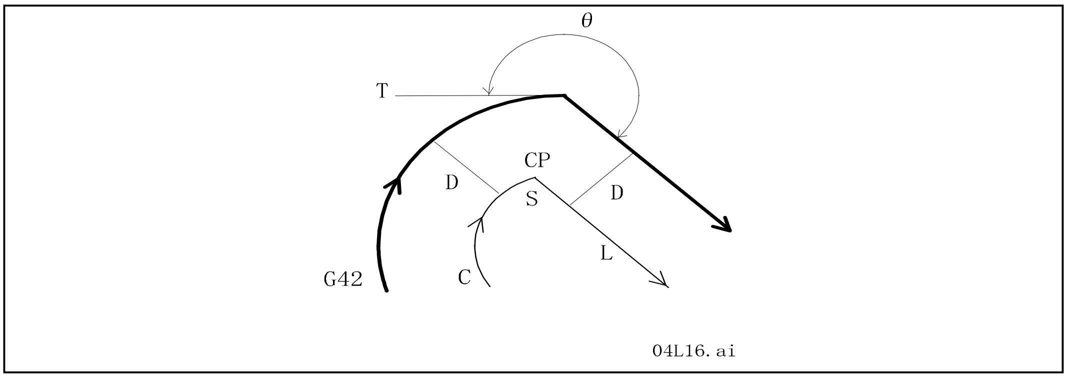
A tool movement command in the offset mode includes a positioning, a linear interpolation, a circular interpolation and a helical interpolation.

4.1.4.1 Inside cutting

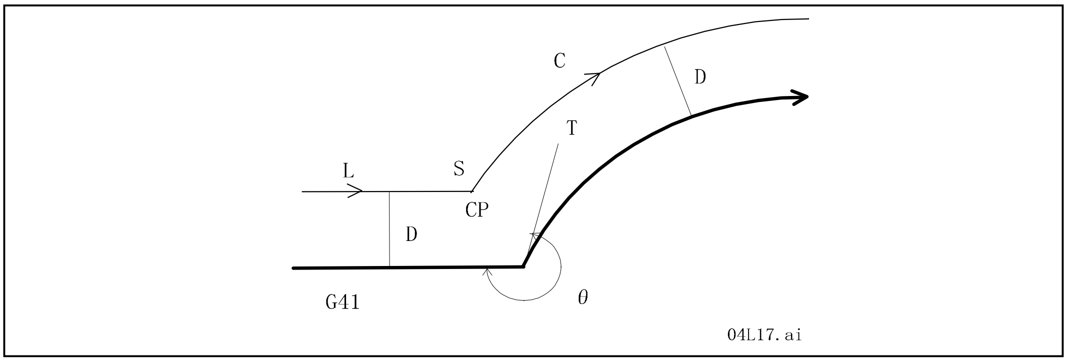
Linear - Linear



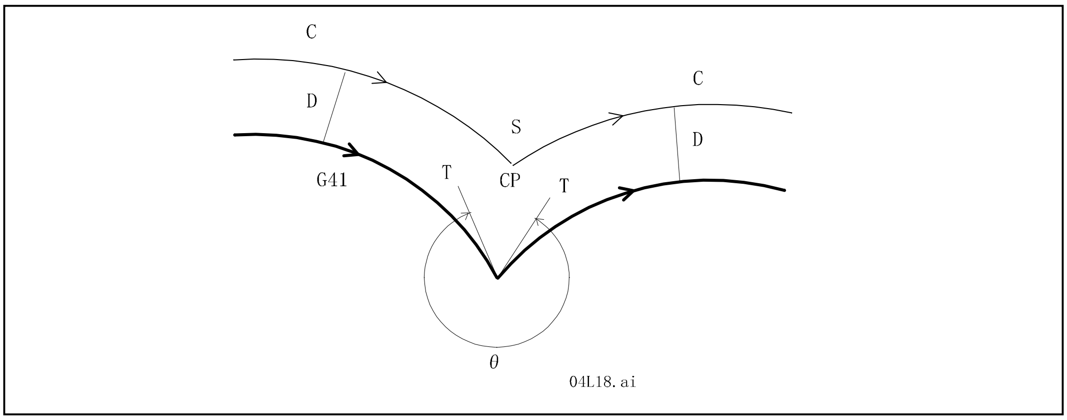
Arc - Linear



Linear - Arc

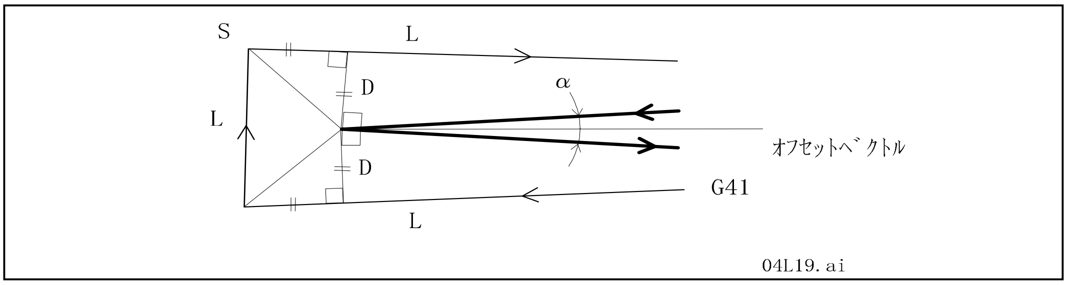


Arc – Arc

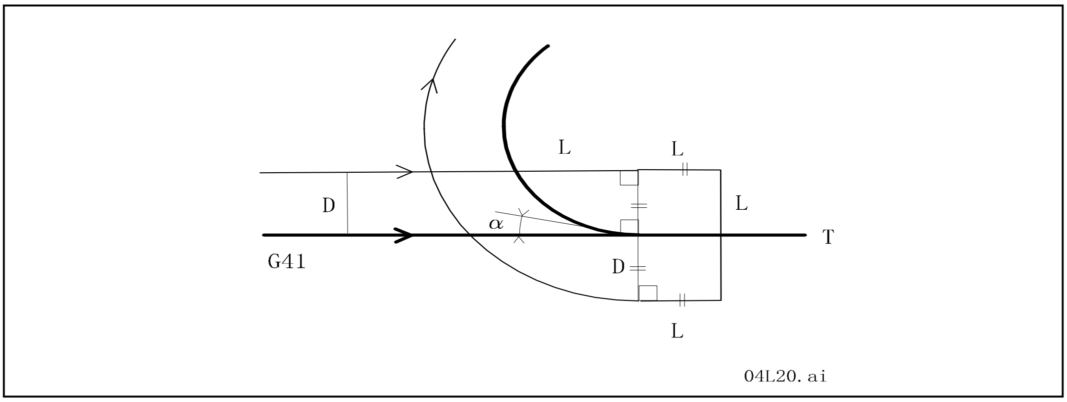


(Note 1) When going around at a narrow angle (there is α < 1°) no cross point of 2 perpendicular lines from programme lines, so that tool center path will be exceptionally as follows;

Linear - linear



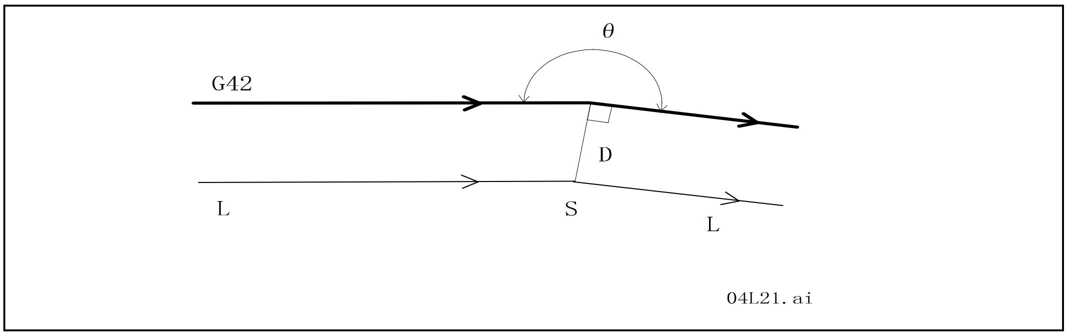
Linear –Arc



It will be processed in the same procedure as above in case of Arc-Linear and Arc-Arc.

(Note 2) When (180˚ ≤ θ< 181˚), tool center path will be as follows;

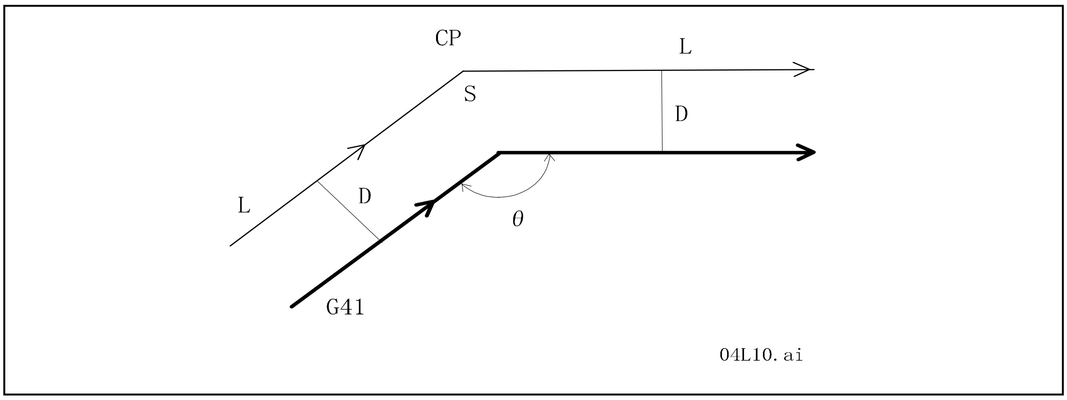
Linear - linear



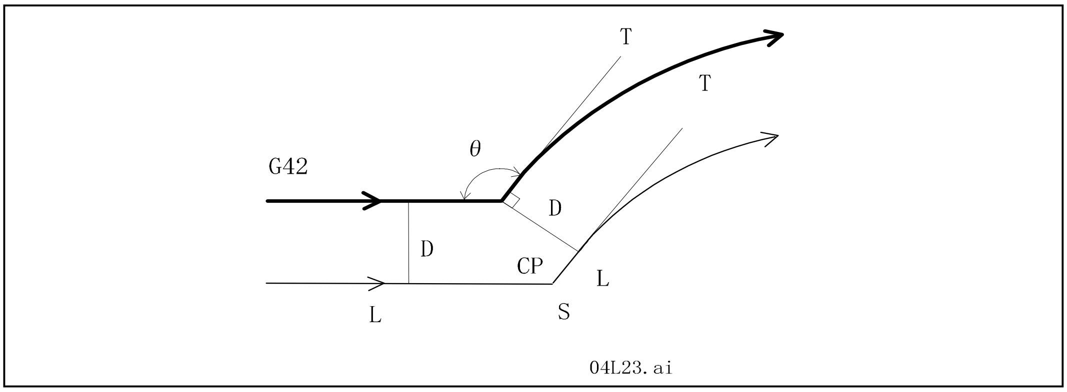
It will be processed in the same procedure as above in case of Arc-Linear, Linear-Arc and Arc-Arc.

4.1.4.2 Outside cutting (90°≤θ<180°)

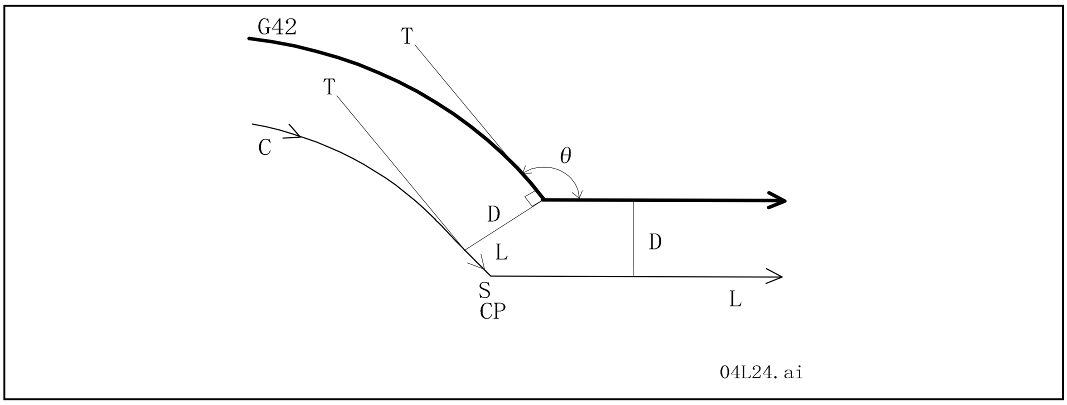
Linear - Linear



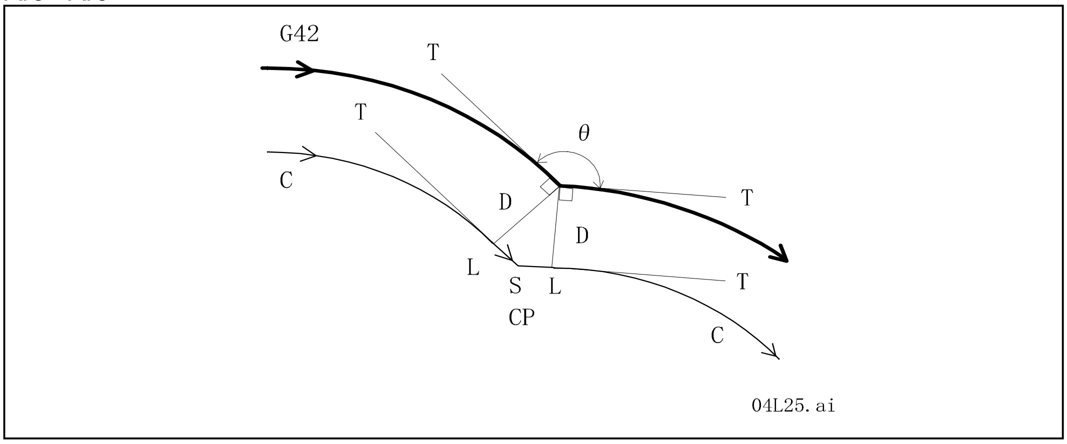
Linear - Arc



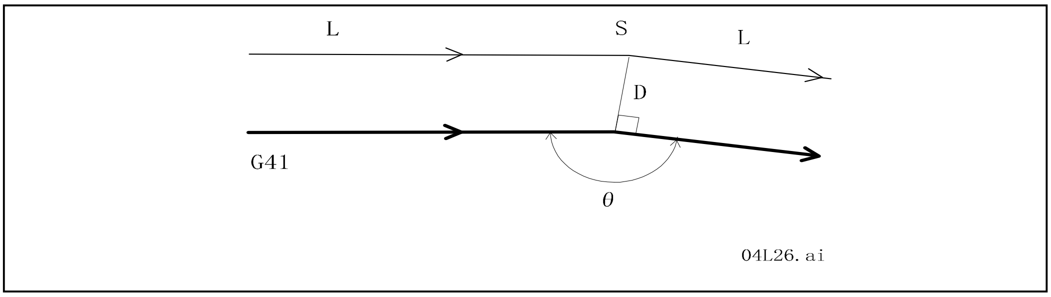
Arc - Linear



Arc - Arc



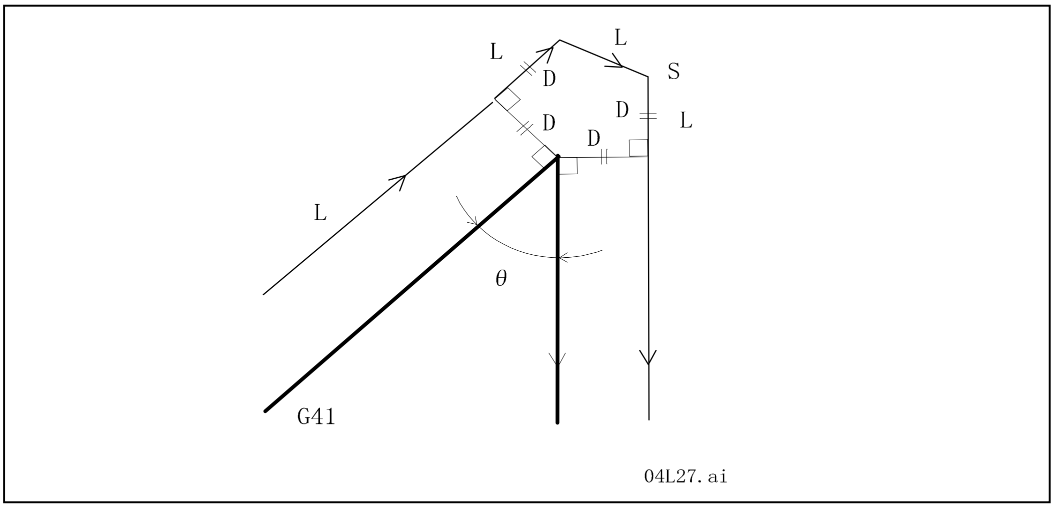
(Note 1) When 179˚ <θ<180˚, tool center path will be as follows;
Linear -Linear



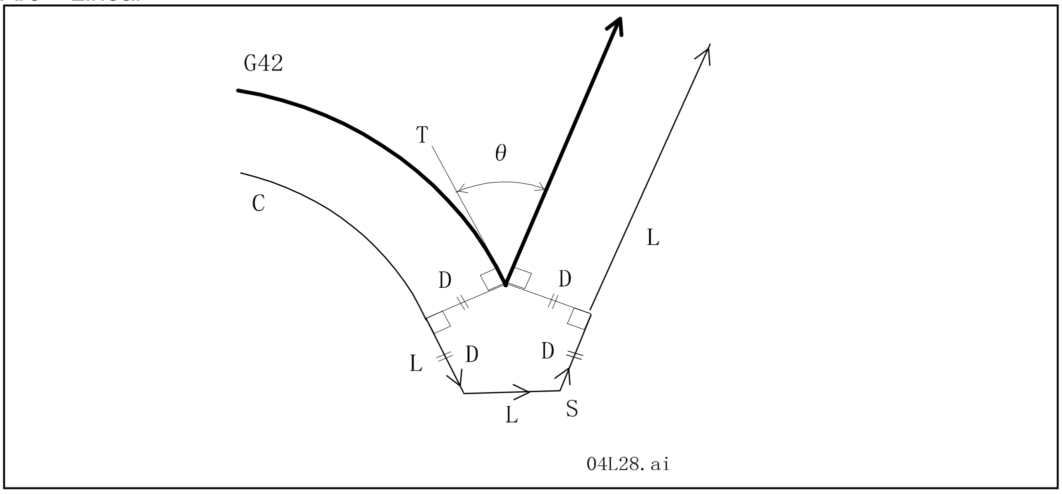
It will be processed in the same procedure as above in case of Arc - Linear, Linear - Arc and Arc - Arc.

4.1.4.3 Outside cutting (θ<90˚)

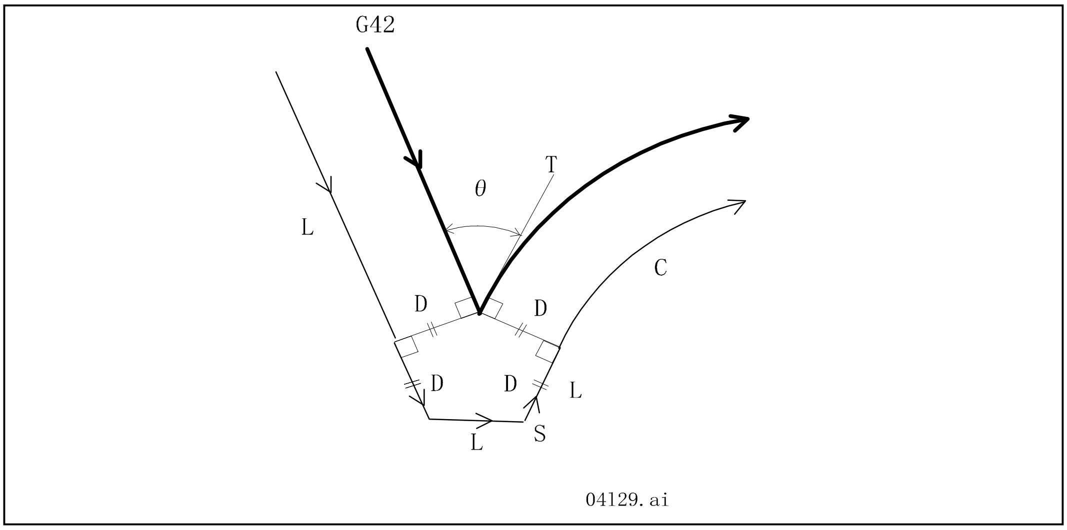
Linear – Linear



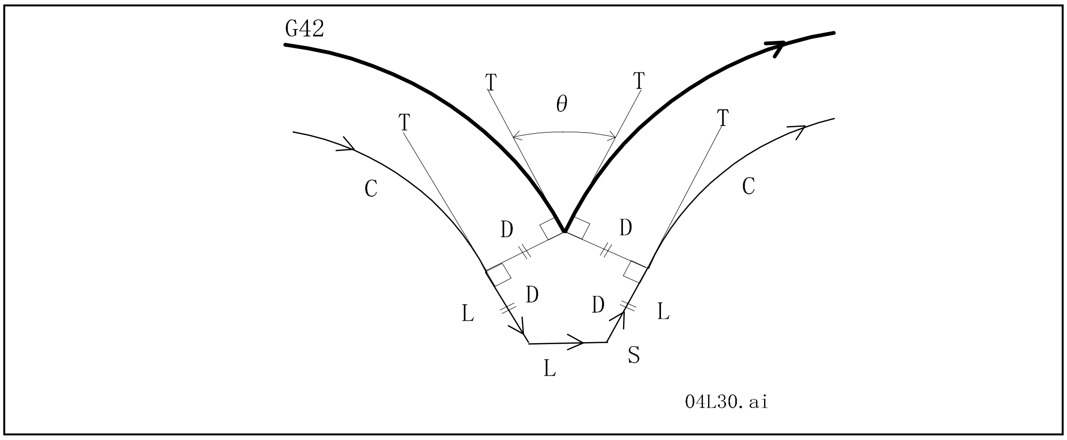
Arc – Linear



Linear - Arc

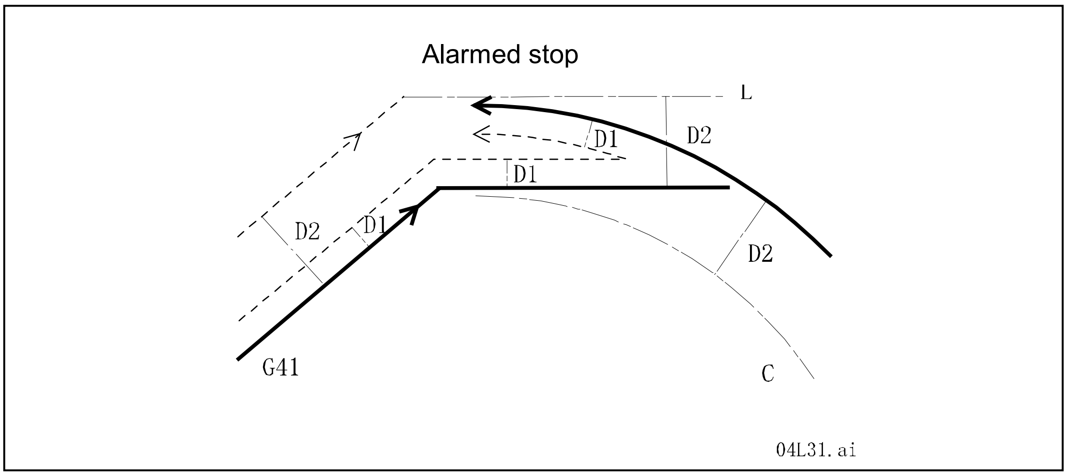


Arc - Arc



4.1.4.4 Exceptional case

There is no cross point at inside cutting.



As above figure shows, the cross point of the arcs is present if the offset value is small, but it may be disappear if the offset value becomes large.
In this case, alarm occurs in the preceding block, and the machine stops.

4

4.1.5 Offset cancel

When the command satisfying all the conditions as shown below is executed in the offset mode, the offset cancel mode becomes effective.
The tool motion in this status is called an offset cancel.

a) G40 is commanded.

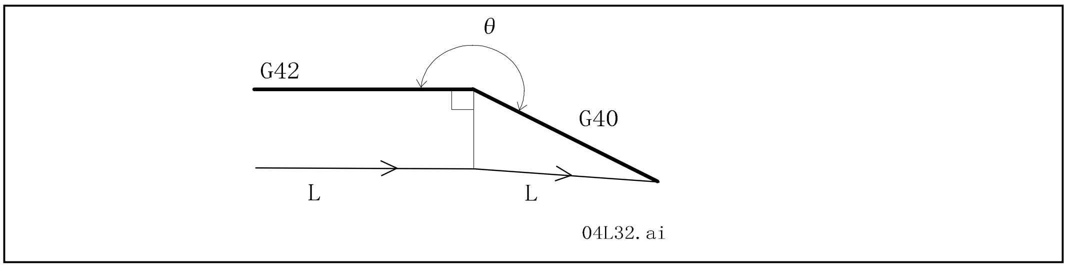
Command format	**G40 ;**

b) The movement commanded by G0 or G1.

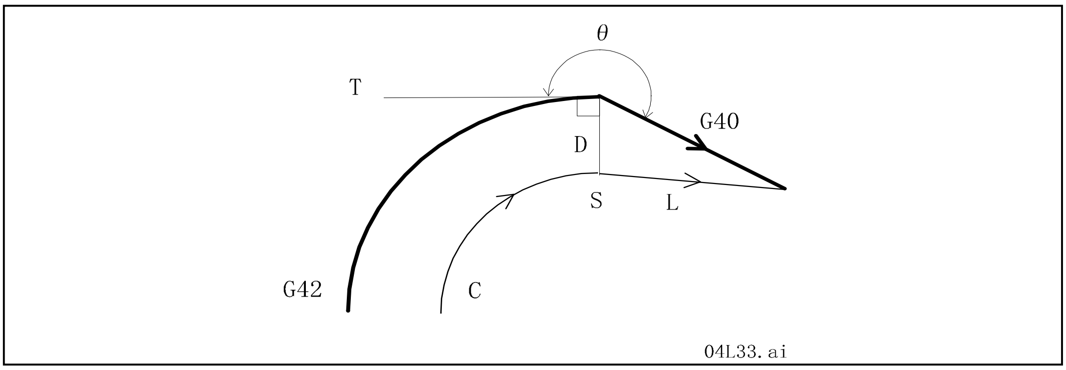
(Note 1) In the case of other command, an alarm is generated.

4.1.5.1 Inside cutting (180° ≤ θ)

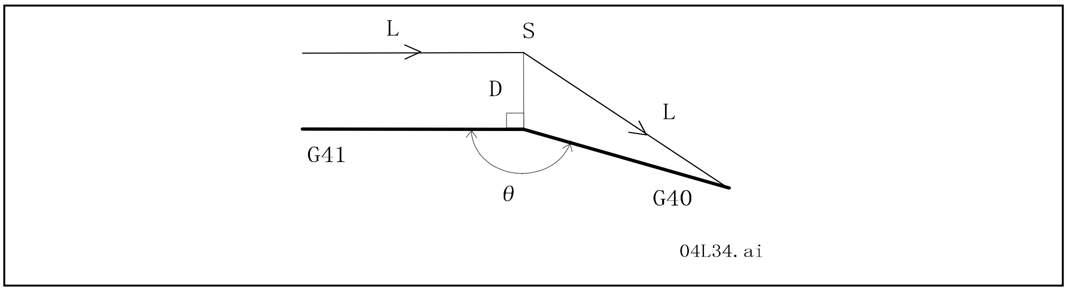
Linear - Linear



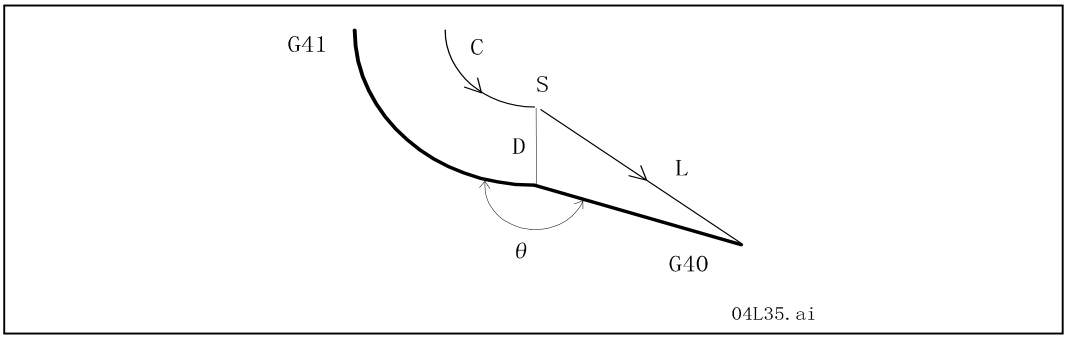
Arc - Linear



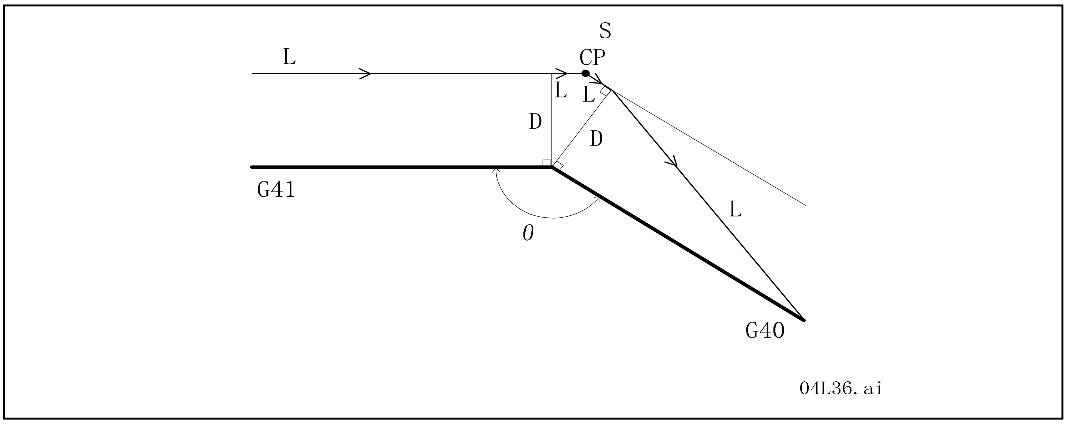
4.1.5.2 Outside cutting (90° ≤ θ ≤ 180°)



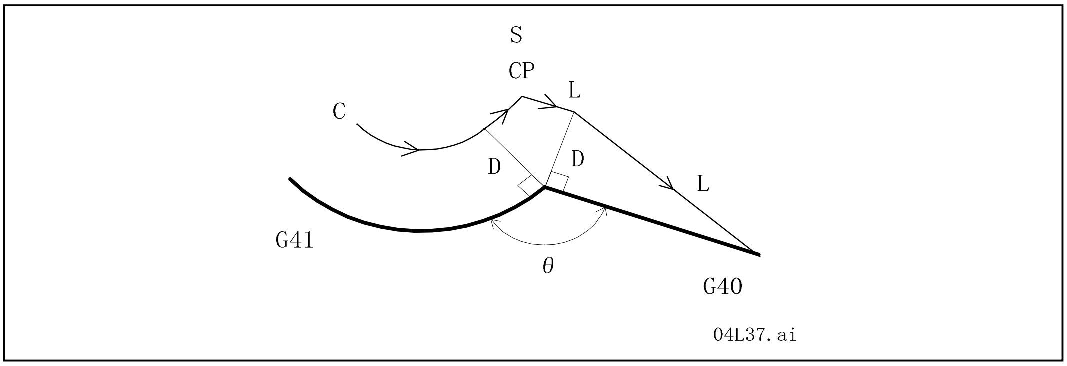
Type 1:Arc-Linear



Type 2:Arc-Linear



Type 2:Linear-Linear

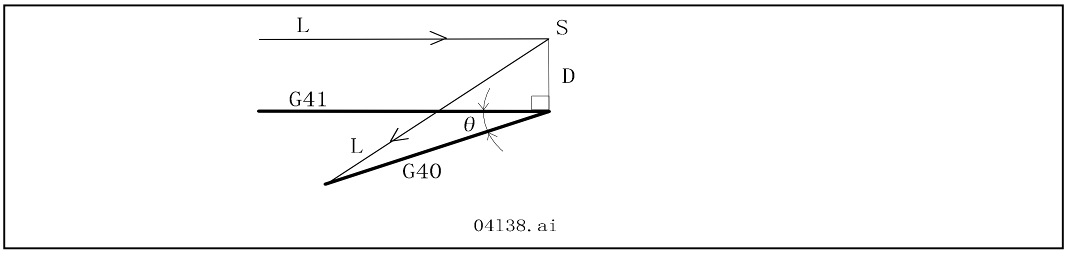


(Note 1) **Type 1 and 2 can be selected in parameter 1 for start-up and cancel motions.**

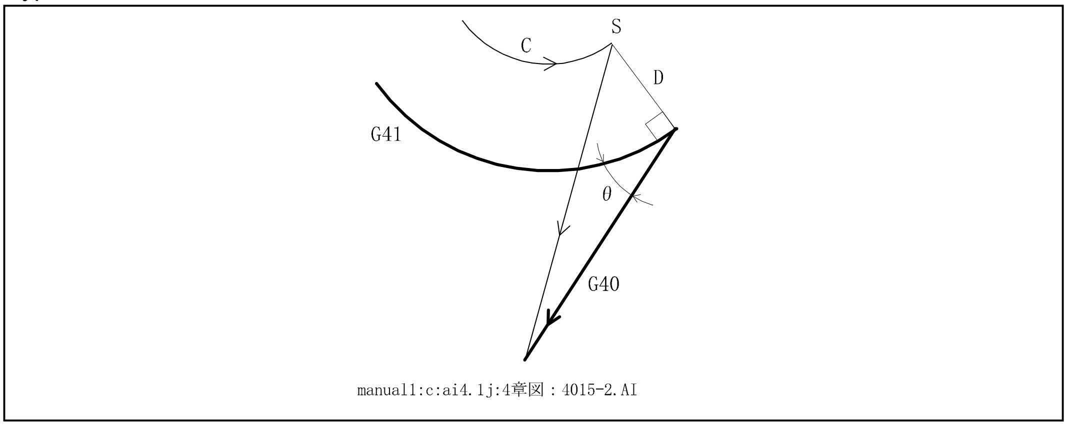
(Note 2) **If the angle is close to 180˚(79˚ ≤ θ< 180˚) while type 2 is being selected,actual movement will be type 1.**

4.1.5.3 Outside cutting （θ＜90˚）

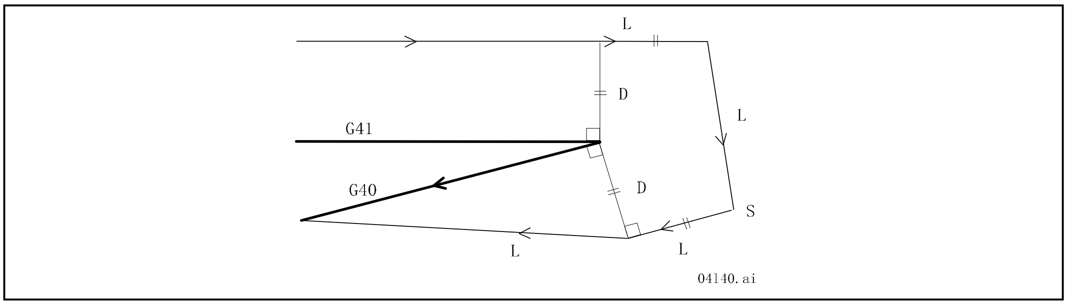
Type 1:Linear-Linear



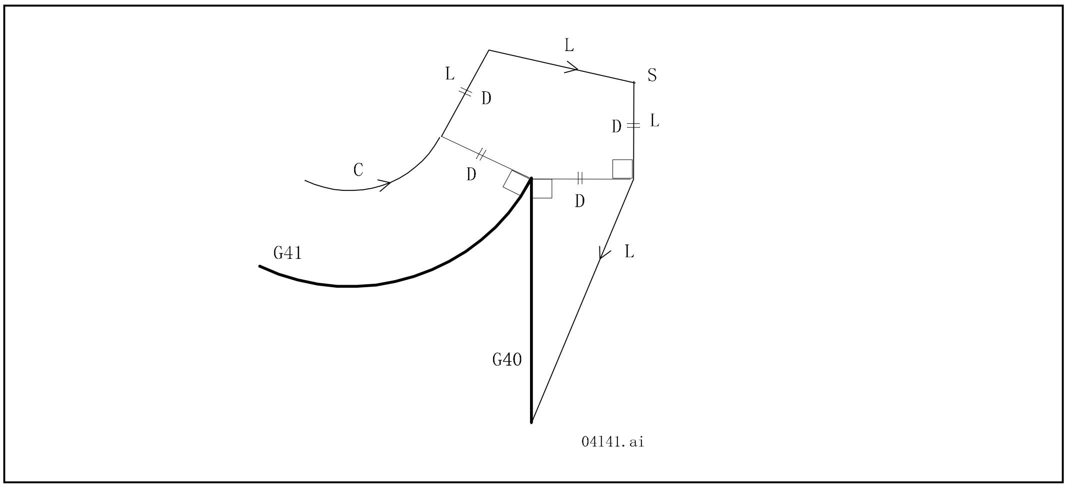
Type 1:Arc-Linear



Type 2:Linear-Linear



Type 2:Arc-Linear

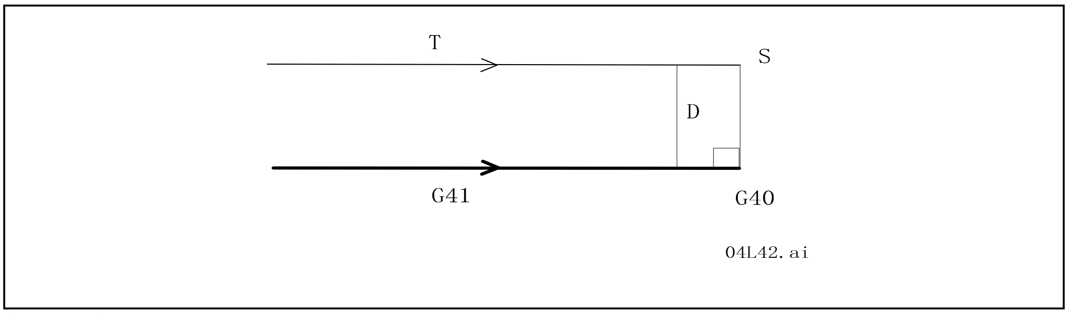


4

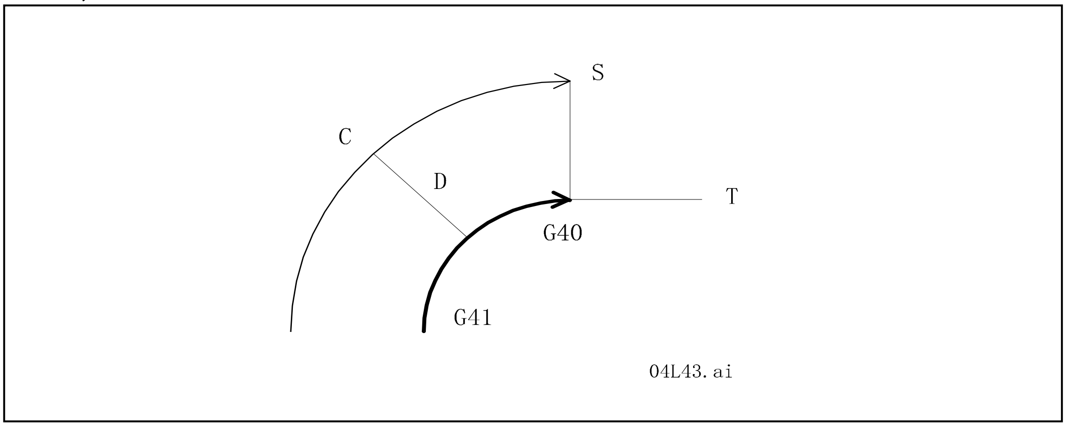
4.1.6 G40 single command

When G40 is specified independently, the tool moves to the position offset perpendicularly in the preceding block and stops.

Linear – Linear
G41 X_Y_D_;
G40 ;

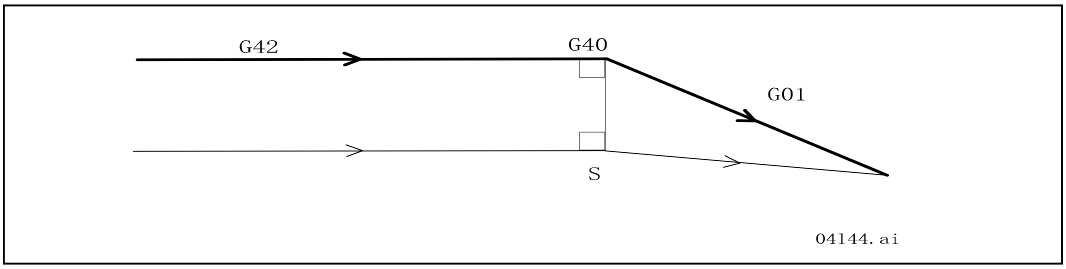


Arc – Linear
G41 X_Y_D_;
G40 ;



(Note) **Offset amount is cancelled by the axial movement command in the following block.**

G42 X Y D ;
G40 ;
G01 X Y F ;



4.1.7 Change of offset direction in offset mode

By commanding G41 or G42, or converting the algebraic sign (+, -) of the offset amount, the offset direction can be changed even in the offset mode.
The block not to be changed the next of start block.
As same as miller (Single axis commanding) and the case of changing the offset directon when D adress position changed.

G code \ Offset amount sign	+	-
G41	Left side offset	Right side offset
G42	Right side offset	Left side offset

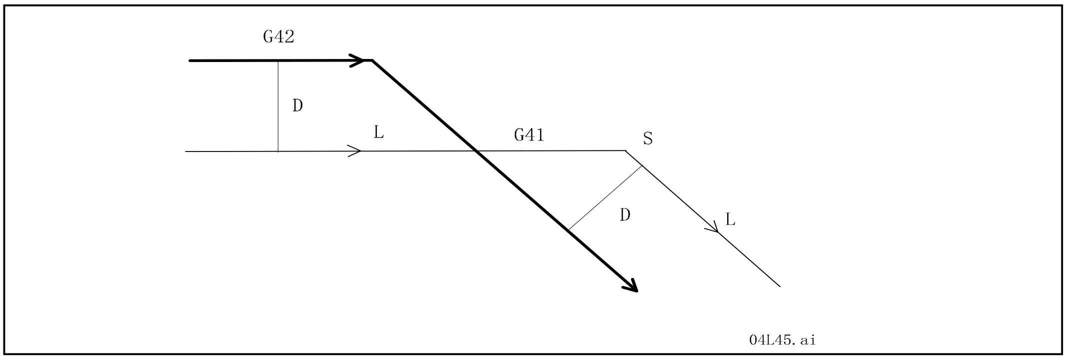
Conditions of execution

<table>
<tr><th>Offset mode</th><th>command</th><th>Linear-Linear</th><th>Linear-Arc</th><th>Arc-Linear</th><th>Arc-Arc</th></tr>
<tr><td>G41</td><td>G41</td><td rowspan="2">Perform</td><td colspan="3" rowspan="2">(Where the offset equivalent to the tool dia is vertically applied to the end point of the previous block)</td></tr>
<tr><td>G42</td><td>G42</td></tr>
<tr><td>G41</td><td>G42</td><td colspan="2" rowspan="2">Perform</td><td colspan="2" rowspan="2">Perform</td></tr>
<tr><td>G42</td><td>G41</td></tr>
</table>

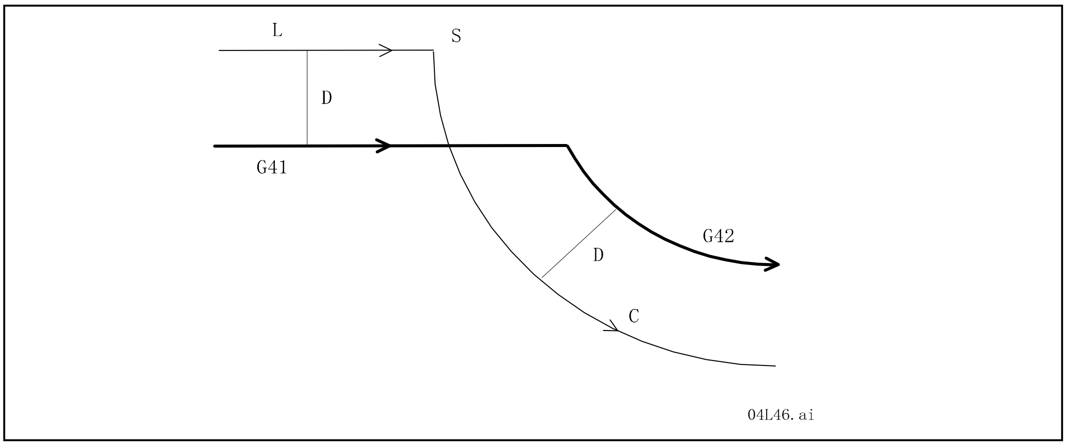
When the offset direction is changed, the "inside" and "outside" cuttings are not discriminated. But whether there is a cross point or not discriminates those cuttings.
The offset amount described hereafter has a positive value.

4.1.8 Change of offset direction in offset mode
4.1.8.1 When there is a cross point

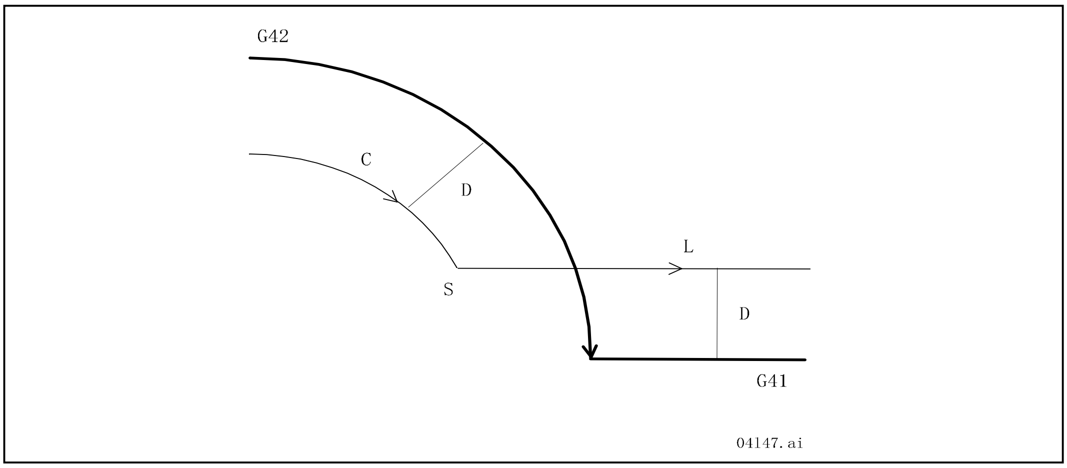
Linear - Linear



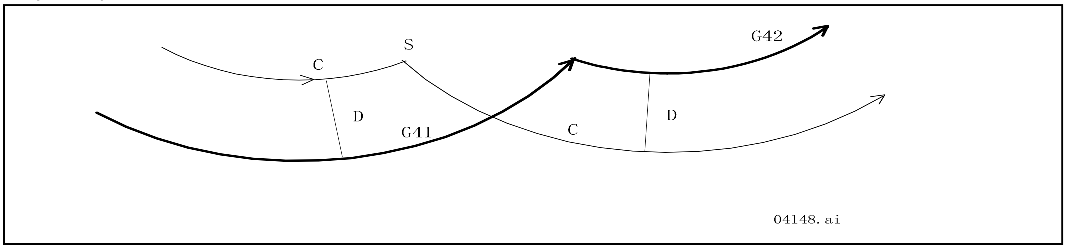
Linear - Arc



Arc - Linear



Arc - Arc

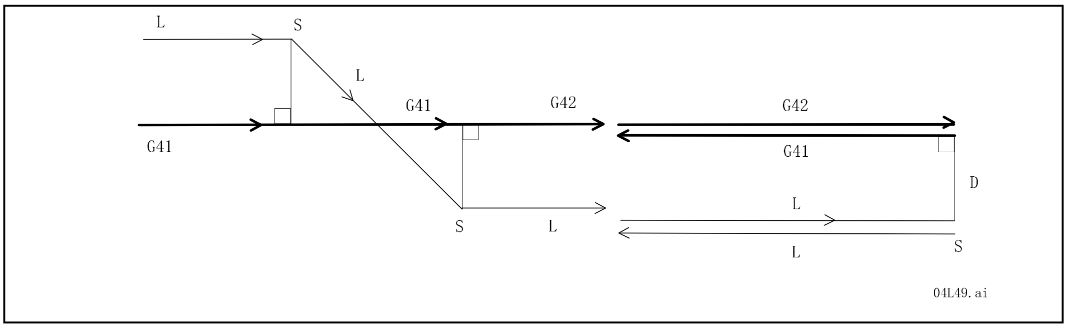


4

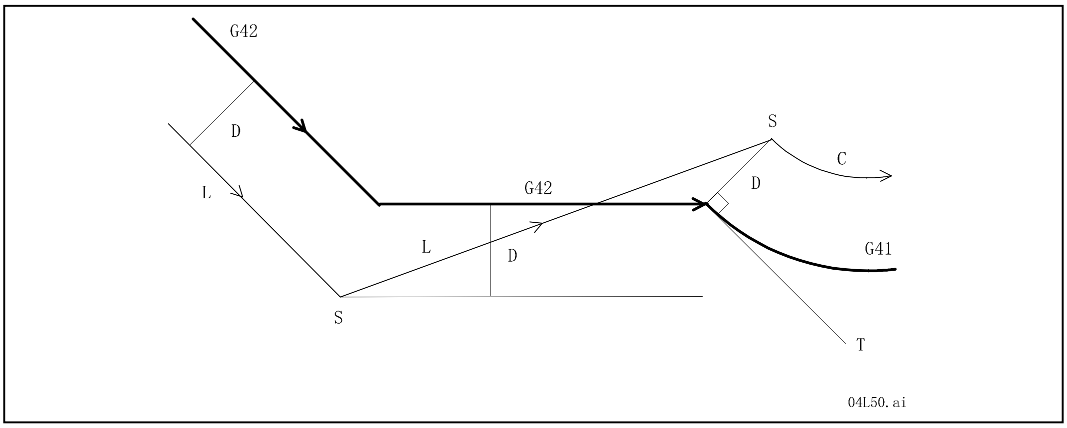
4

4.1.8.2 When there is no cross point

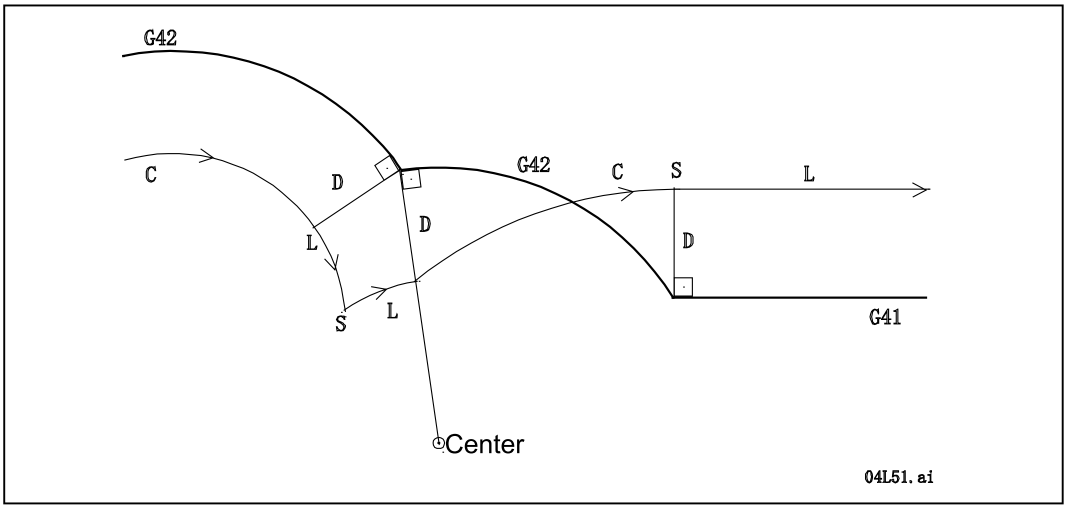
Linear – Linear



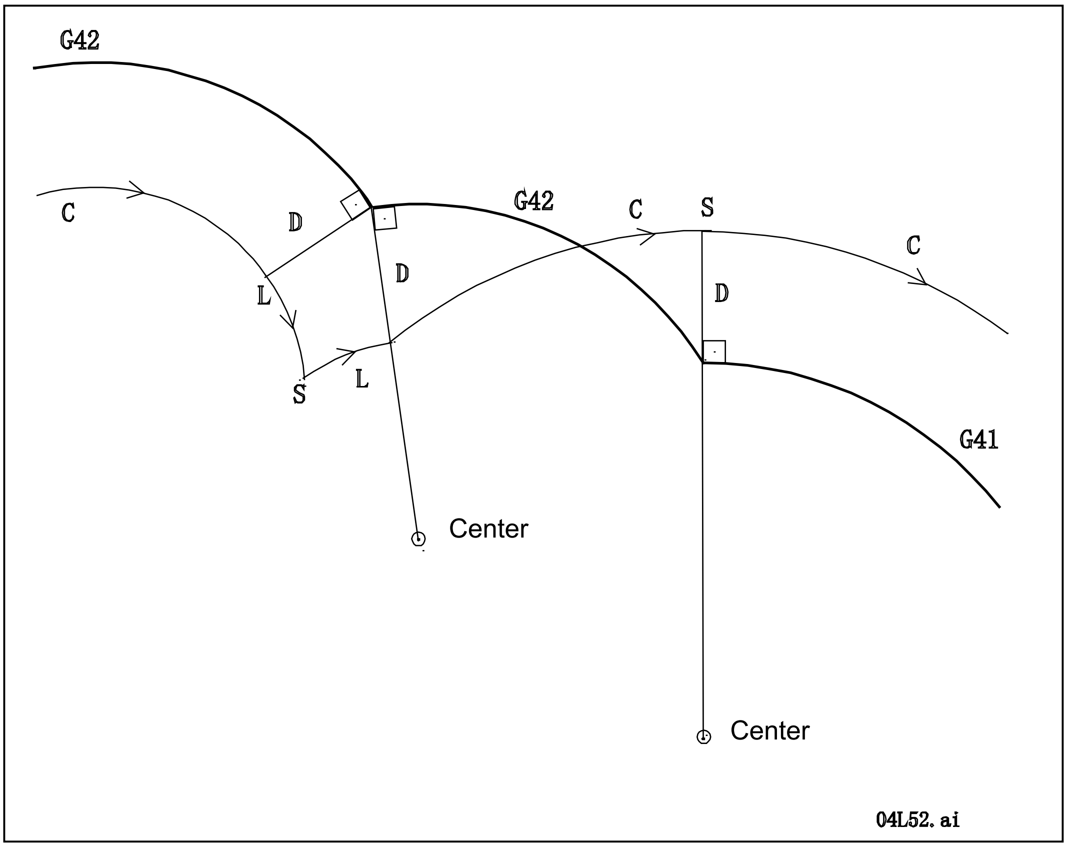
Linear – Arc



Arc – Linear

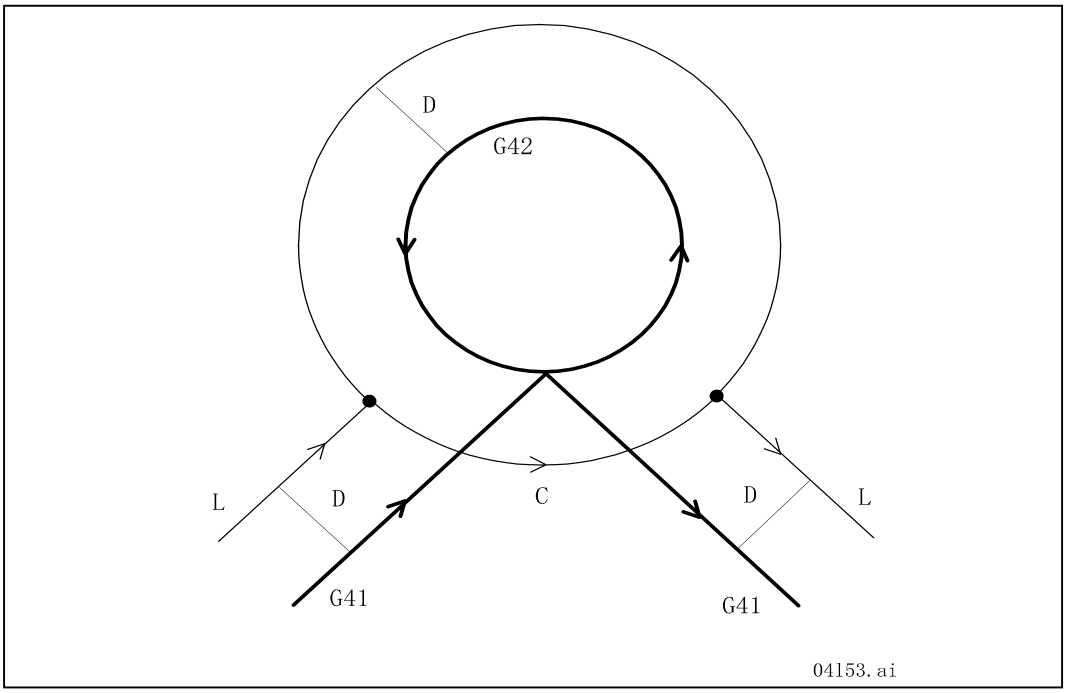


Arc – Arc



4.1.8.3 When offset path becomes more than a circle

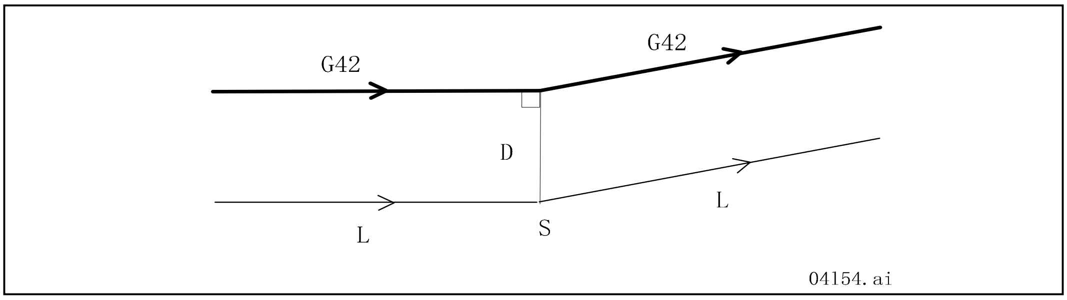
By changing offset direction offset path becomes more than a circle, but actual offset path is short cutted as shown below.
In this case the circle must be specified in segments.



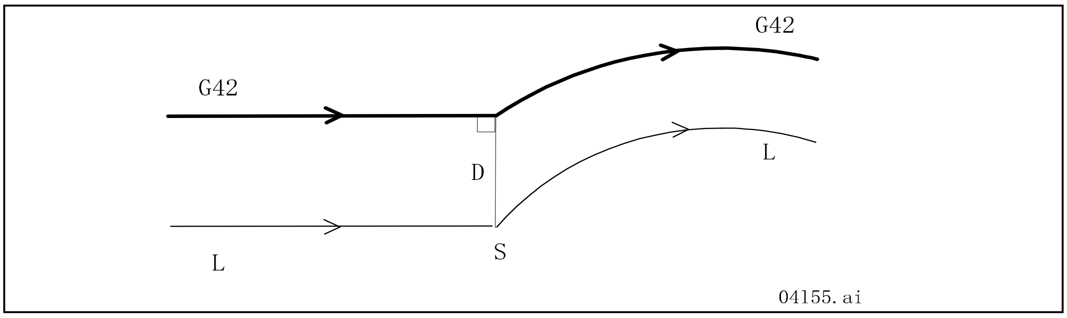
4

4.1.9 G code command for tool dia offset in offset mode

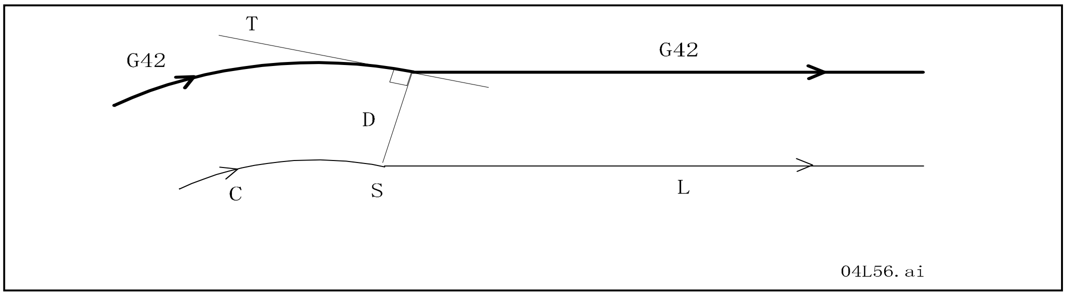
Linear - Linear



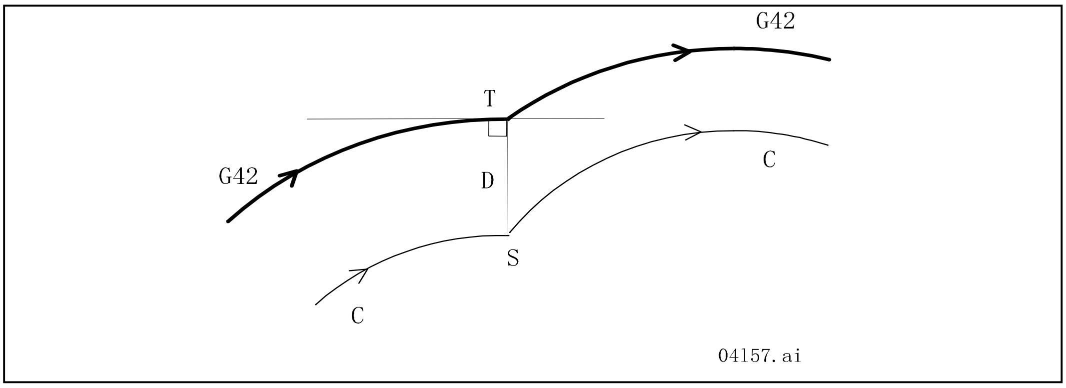
Linear - Arc



Arc - Linear



Arc - Arc

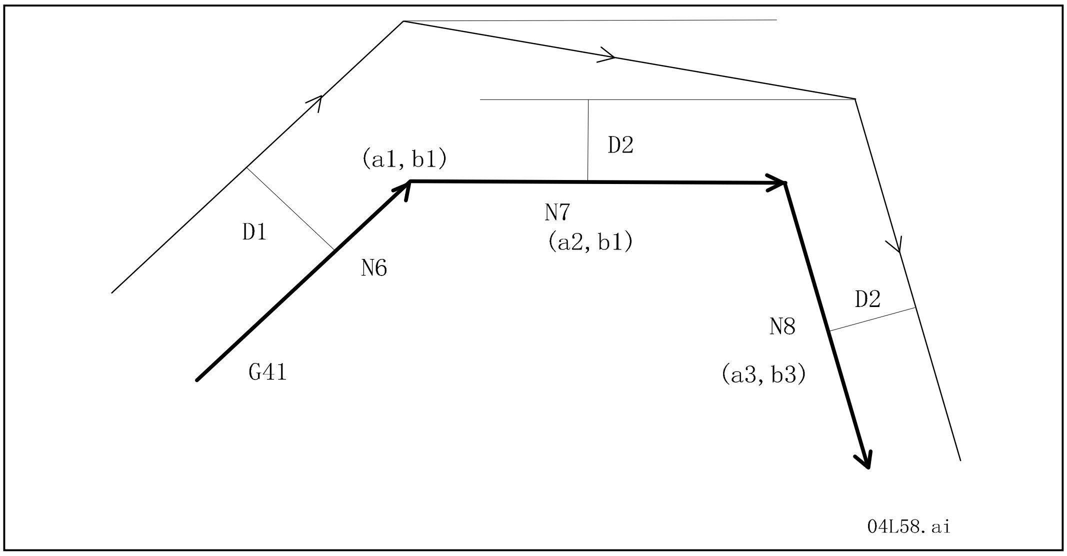


4.1.10 Notes on tool dia offset

(1) Command of tool dia offset amount
The offset amount is commanded by the number of the D command.
When G41 or G42 is commanded, the offset amount is commanded in the same block.
If it is omitted, the number of D command previously used becomes effective.

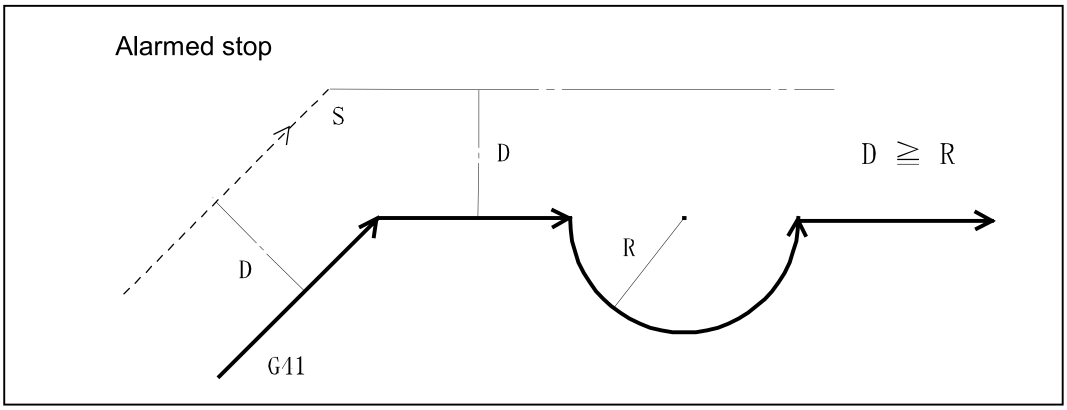
(2) Change of tool dia offset amount
When the offset amount is changed in the offset mode, the offset amount is changed at the end point of the block.

N1 G41 X_Y_D1;
N6 Xa1 Yb1;
N7 Xa2 D2;....Change of offset amount
N8 Xa3 Yb3;



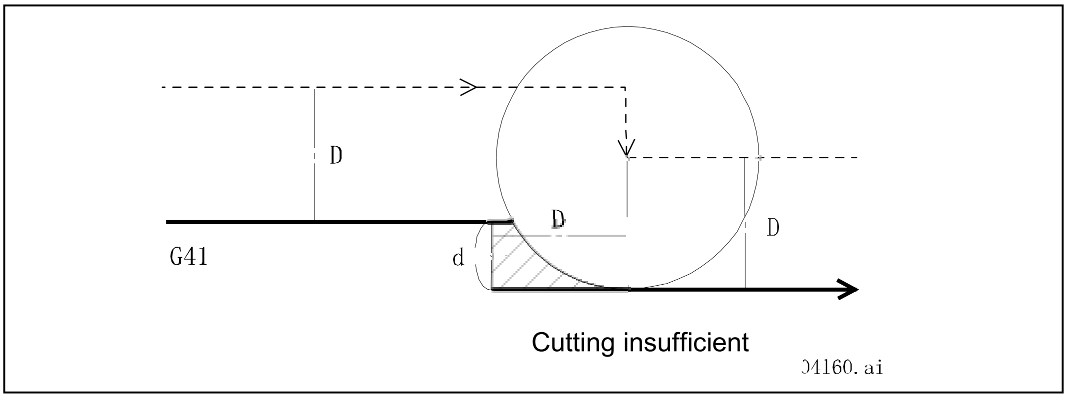
(3) Current position display
The current position display corresponds to the tool center position

(4) Machining of inner wall of circular arc smaller than the used tool radius



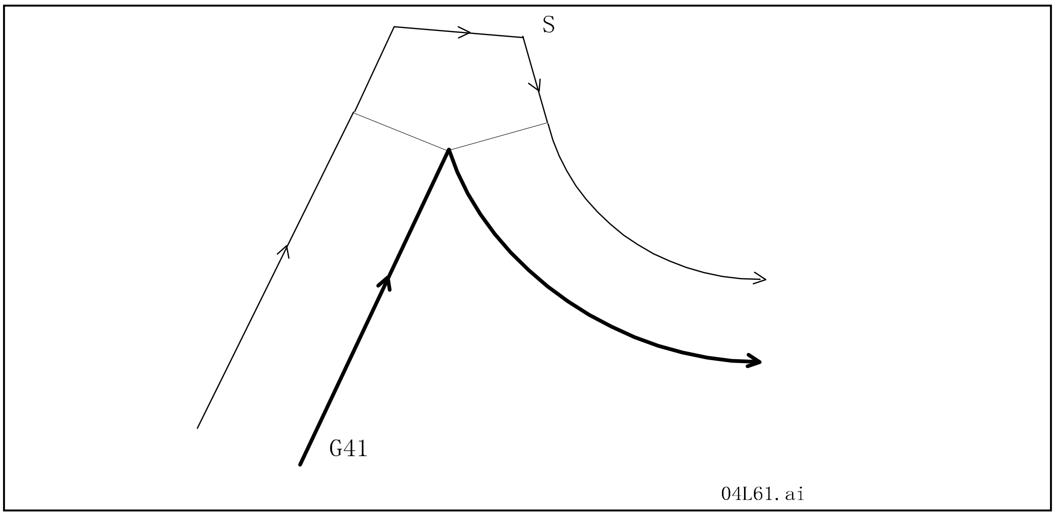
Since cutting is not available, the operation stops at the end point of the previous block and an alarm is generated.

(5) Cutting insufficient
This problem occurs in the case of a program containing a step smaller than the tool radius.



(6) Corner movement

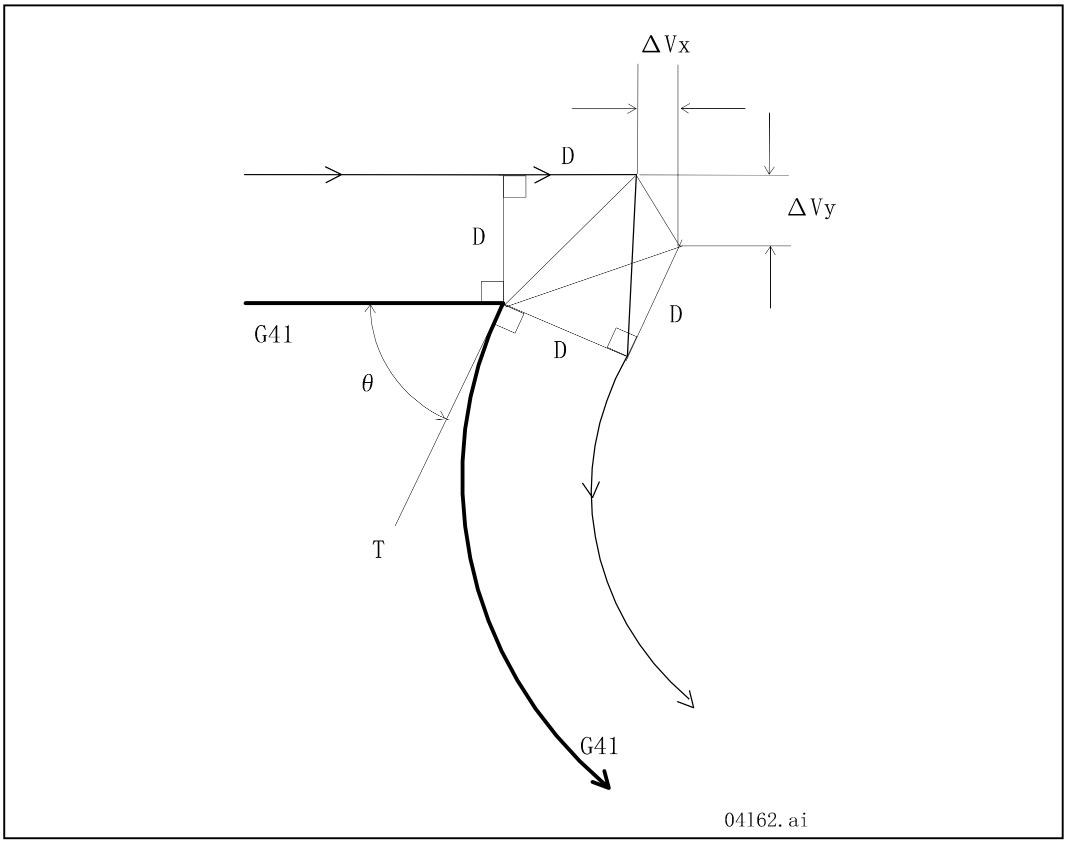
When cutting the outer side, the tool moves around the corner from different angles.The movement mode and the feedrate up to the single block stop point are as specified in the current block.



When the movement amount around the corner is small as shown in the figure below and the following conditions are satisfied, the movement is ignored.

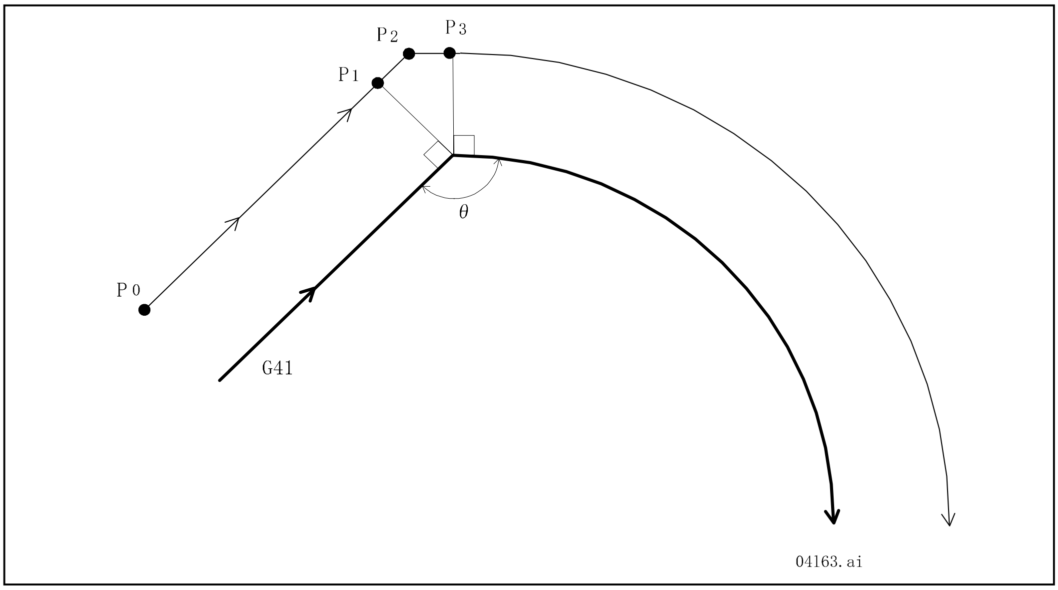
$\Delta VX \leq \Delta V$ and $\Delta VY \leq \Delta V$

The value ΔV is set by the parameter 1.



Thus, extremely small movements around the corner can be reduced.

This function is not available if the following block is a full circular arc.



The original movement in the above figure are:

P0-P1-P2	Linear movement
P2-P3	Linear movement

The tool moves once around the circular arc afterwards with P3 as the target.
If this small movement function is used, the movement from P2 to P3 is ignored and executed as follows:

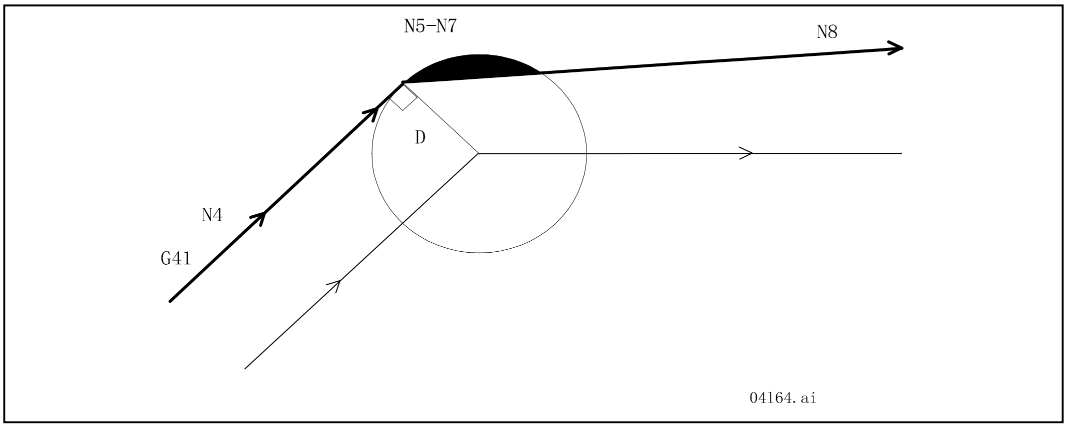
P0-P1-P2	Linear movement
P2-P3	(small) circular movement

A full circular is ignored and the movement from P2 to P3 becomes a small circular movement, thus ignoring this function.

(7) Block without movement

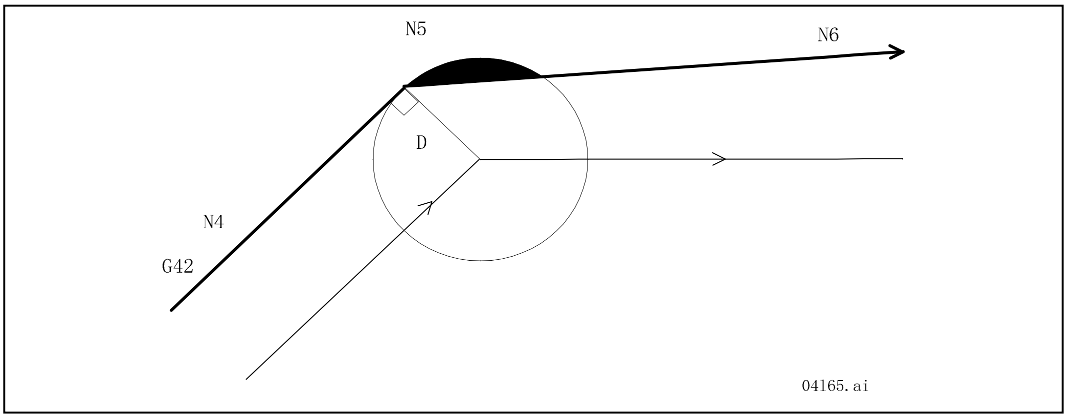
When the command without any X/Y movement is given for more than 3 blocks during the tool dia offset mode, the movement is as shown below and overcutting or undercutting occurs.

N4 X_Y_;
N5 X0;
N6 X_;
N7 Z_;
N8 X;



(Note 1) The block with the movement amount zero will result in the same as above.

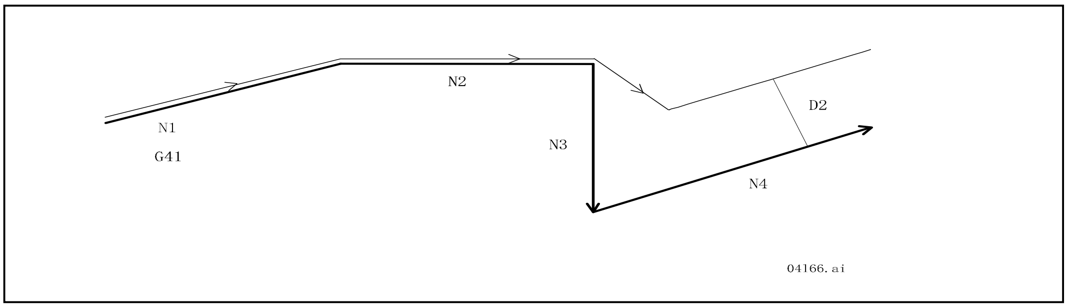
N4 G91 X_Y_;
N5 X0;
N6 X_;



(Note 2) Even if the X/Y movement command is not given at the start-up, the start-up motion begins at the time the X or Y movement command is executed.

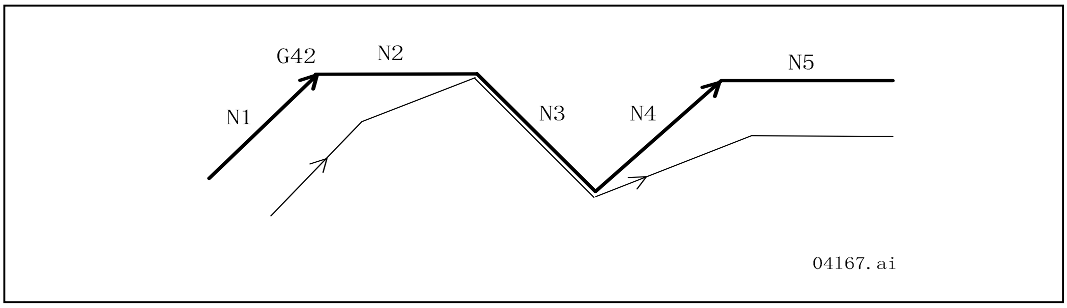
(8) Tool movement in case of tool dia offset amount zero
a) Start-up
When G41 or G42 is commanded in the cancel mode, the offset mode becomes effective but the start-up motion is not available as the offset amount is zero.
When another offset number is specified and the offset amount is not zero, the motion becomes thesame as the case of changing the tool dia offset amount as described in (2).

N1 X_Y_D1; (D1=0)
N2 X_;
N3 Y_D2; (D2≠0)
N4 X_Y_;



b) During offset mode
If the offset amount is changed to zero in the offset mode, the cancel mode is not available any longer. The motion becomes the same as the case of changing the tool dia offset amount as described in (2).
Even if the offset number is changed again and the offset amount is not zero, the motion becomes the same as the case of changing the tool dia offset amount as described in (2).

N1 X_Y_;
N2 X_D1; (D1=0)
N3 X_Y_;
N4 X_Y_D2 (D2≠0)
N5 X_;



(9) Exceptional case or alarm-generating command

1.Command to produce the vertical vector

G10 : Programmable data input
G52 : Local coordinate system
G92 : Coordinate system setting
#3000: Alarm display
#3006: Message display & stop

When the above command is given, the tool moves to the point which is offset as much as the tool dia as specified by the last X/Y movement command.

2.Forcible tool dia offset cancel

M06 : Tool change
G100 : Non-stop ATC

When the above command is given, G40 (tool dia offset cancel) becomes automatically effective and the tool moves to the point which is offset as much as the tool dia as specified by the last X/Y movement command.

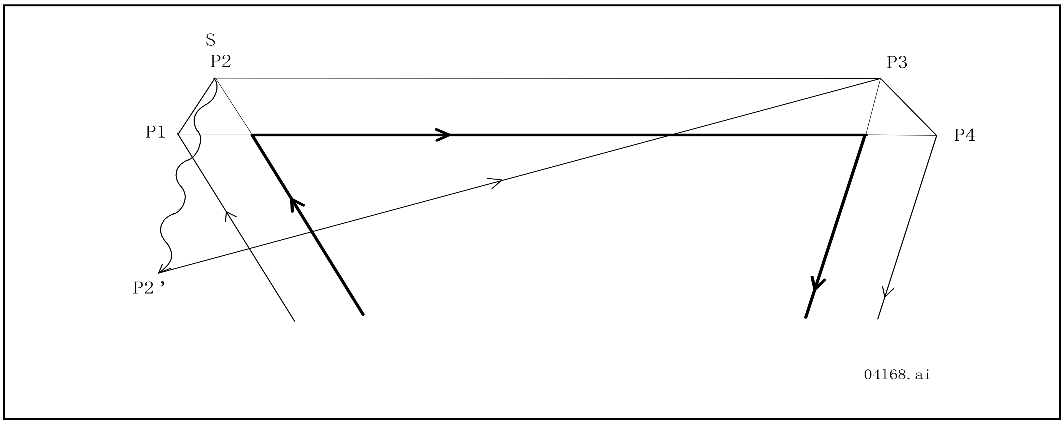
3.Alarm-generating command

G28 : Return to reference point
G29 : Return from reference point
G30 : Return to 2nd, 3rd and 4th reference point
G36 ~ G39 : Coordinate calculation
G60 : Single direction positioning
G66 : Macro
G73 ~ G89, G173 ~ G189 : Canned cycle
M200, M201 : Tool breakage detection
M120 : TOUCH signal check
G120 : Positioning to the measuring point
G121 ~ G128 : Automatic measurement
G131, G132 : Measurement feed
G133, G134 : Changeover of tap twisting direction
M203 : Tool breakage detection judgement
M206 : Thermal watch condition judgement
M207 : Thermal measurement motion
M410, M411 : Index of the pallet

(10) Input command in MDI operation

When inputting the tool dia offset command (G40, G41, G42) in the MDI operation, an alarm is generated.

(11) Manual intervention
When moving the tool by manual operation in the offset mode and starting the memory operation again, the corrected offset path starts from two blocks ahead.



04L68.ai

When moving the tool by manual operation after it stopped at the block end point P2, the tool movs from P2' to P3, then follows the corrected offset path after P3.

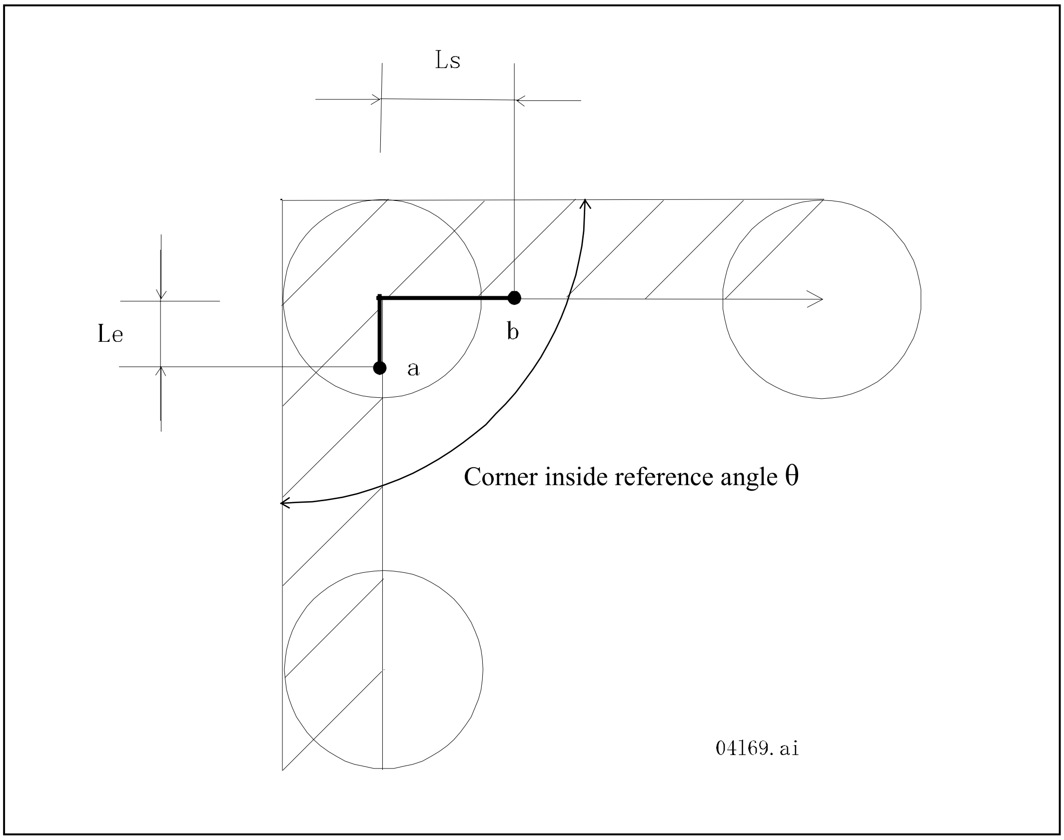
4.1.11 Override function related to tool dia offset
4.1.11.1 Automatic corner override

When the blocks before and after the inner corner satisfy the following conditions, an automatic override is applied to reduce the load on the tool.

1. Movement is commanded G01, G02 or G03. (Volute/without circular cone interpolation)
2. The offset mode is selected and the offset amount is not zero.
3. The corner is inside and its angle is less than the AUTO.CORNER OVER RIDE ANGLE set by the parameter 1.
4. The block does not contain G41, G42 and G40.
5. There is no change in the offset direction.

The parameter 1 has the following setting values.

1)AUTO.CORNER OVERRIDE LEN1
: Deceleration stroke at corner end point :Le
2)AUTO.CORNER OVERRIDE LEN1
: Deceleration stroke at corner start point :Ls
3)AUTO.CORNER OVERRIDE RATIO :Reduction ratio (%) :Y
4)AUTO.CORNER OVERRIDE ANGLE :Corner inside reference angle : θ



An override is applied to the thick lined area between a and b.

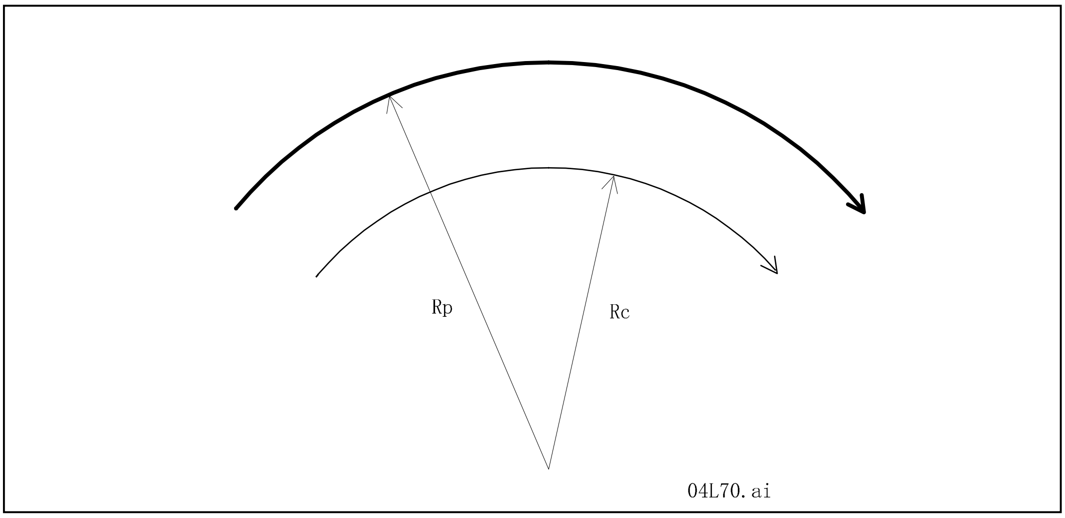
Actual feedrate = Commanded feedrate $\dfrac{\text{Deceleration}}{100}$

4

4.1.11.2 Override of the inside circular cutting

When cutting along the circular arc whitch is offset inside during the offset mode, the actual feedrate is calculated by multiplying the commanded feedrate by Rc/Rp.

$$\text{Actual feedrate} = \text{Commanded feedrate} \times \frac{Rc}{Rp}$$



(Note 1)
When the value of Rc/Rp becomes smaller than the OVERRIDE LMT IN INSIDE ARC set by the parameter 1, multiply that parameter value instead of Rc/Rp.

$$\text{Actual feedrate} = \text{Commanded feedrate} \quad \frac{\text{OVERRIDE LMT IN INSIDE ARC}}{100}$$

4.2 Tool Length Offset (G43, G44, G49)

This function corrects the tool position so that the tool nose comes to the programmed position. In either the absolute command or the incremental command, the end point in the programmed Z-axis move command is offset as specified by H code to become the actual end point.

(1) Tool length offset (+)

Command format

G43 Hn

Hn: Tool length offset No. (n=0~99)

(Note)
The offset amount of H0 is always zero.
The offset amount is set on the tool data setting screen.
The tool length offset is done in the Z-axis direction.

(2) Tool length offset (-)

Command format

G44 Hn ;

Hn: Tool length offset No. (n=0~99)

(3) Tool length offset cancel

Command format

G49

(Note 1)
The tool length offset can be cancelled by commanding G49 or specifying zero for the tool length offset number.
(Note 2)
The tool length offset is cancelled by M06 (tool change) or G100 (non-stop ATC).
(Note 3)
If the Z-axis command is not given to the block of "G43H_;" or "G44H_;", it isregarded that the Z-axis command is given to the current position and Z axismoves by the offset amount specified by the H code.
In the same way, if the Z-axis command is not given to the block of "G49;", it is regarded that the Z-axis command is given to the current position and Z axis moves by the offset amount specified by the final H code.
(Note 4)
If the Z-axis command of the reference point return (G28) or the 2nd, 3rd,4th reference point return (G30) is given, the tool moves while the tool length offset is applied to the intermediate point and the tool returns to the reference point by cancelling the tool length offset tentatively.
The Z-axis movement afterwards is executed with the tool length offset.
If the incremental mode is selected at this time, the tool movement is regarded to start from the reference point.
(Note 5)
When G53Z_; is commanded during tool length offset, tool length offset is canceled temporarily and the Z axis moves to a certain point.

4.2.1 Tool length fine offset

When G43 and G44 are commanded in the program, the tool length fine offset value corresponding to the commanded tool No. is added to the tool length offset value.
Offset value to be reflected = Tool length offset value + Tool length fine offset value.

The tool length fine offset value is placed on the tool list screen.

(This page is a blank.)

CHAPTER 5

PREPARATION FUNCTION (CANNED CYCLE)

5 CANNED CYCLE

For repetitive machining, a series of paths that is usually specified in a few blocks can be specified in one block.

5.1 List of Canned Cycle Function

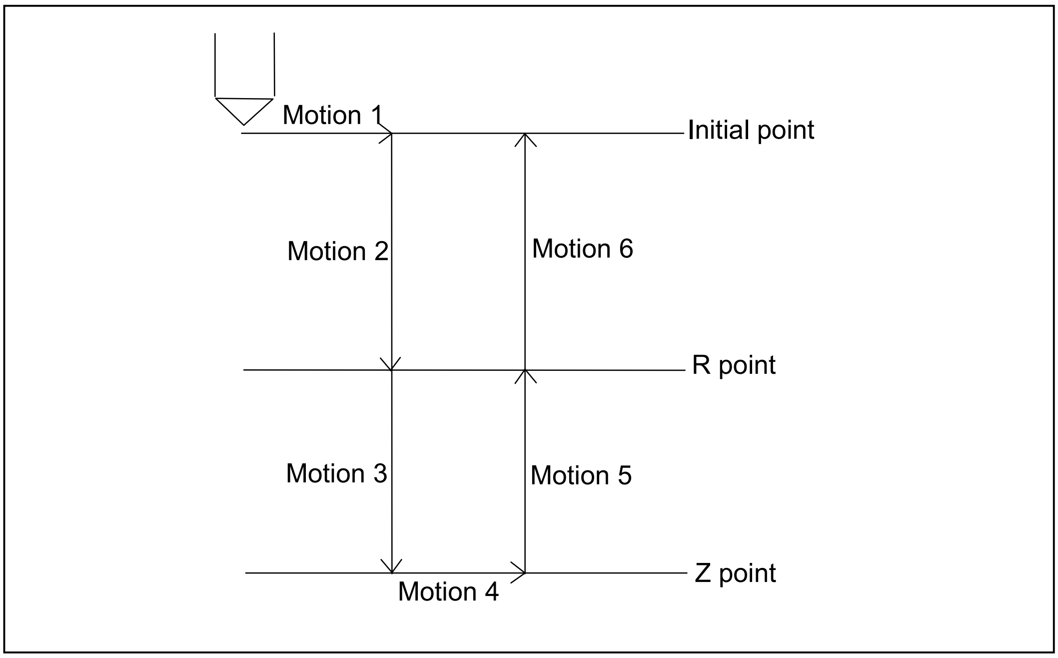
Table 5-1 List of canned cycle function

G code	Content	Feeding type at matching	Spindle motion at bottom of hole	Retracting motion	Spindle at return point
G73	High speed peckdrilling	intermittent feed	dwell	rapid feed	
G74	Reverse tapping	cutting feed	dwell → CW	cutting feed	stop
G76	Fine boring	cutting feed	dwell → orientation rapid	rapid feed	CW
G77	Tapping (synchro mode)	intermittent feed	CCW	cutting feed	stop
G78	Reverse tapping (synchro mode)	intermittent feed	CW	cutting feed	stop
G80	Cancel	~	~	~	~
G81	Drilling	cutting feed	dwell	rapid feed	
G82	Drilling	cutting feed	dwell	rapid feed	
G83	Peck drilling	intermittent feed	dwell	rapid feed	
G84	Tapping	cutting feed	dwell CCW	cutting feed	stop
G85	Boring	cutting feed	dwell	cutting feed	
G86	Boring	cutting feed	dwell → stop	rapid feed	CW
G87	Back boring	cutting feed	dwell → orientation	rapid feed	CW
G89	Boring	cutting feed	dwell	cutting feed	
G177	End mill tap cycle	cutting feed	CCW	cutting feed	stop
G178	End mill tap cycle	cutting feed	CW	cutting feed	stop
G181	Double drilling cycle	cutting feed	dwell	rapid feed	
G182	Double drilling cycle	cutting feed	dwell	rapid feed	
G185	Double boring cycle	cutting feed	dwell	cutting feed rapid feed	
G186	Double boring cycle	cutting feed	dwell → stop	rapid feed	CW
G189	Double boring cycle	cutting feed	dwell	cutting feed rapid feed	

5.2 Basic Motions in Canned Cycle

In general, the canned cycle is composed of the following six motions.

Motion 1 : Positioning (at rapid feed) to the drilling position (X/Y)
Motion 2 : Positioning to R point (at rapid feed)
Motion 3 : Hole machining (cutting feed)
Motion 4 : Machining at the bottom of hole
Motion 5 : Relief to R point (at rapid feed/cutting feed)
Motion 6 : Positioning to initial point (at rapid feed)



eNCPR5.01.ai

The system stops upon completion of the motion 1, 2 or 6 in the single block operation.

(Note) Temporary stop range in tapping cycle (G74, G77, G78, G84, G177, G178).

(1) During the motion 1, 2 or 6 in the tapping cycle, a temporary stop is available.

(2) During the motions 3 through 5 in the tapping cycle, a temporary stop is forbidden.
If such a stop is forcibly done (by pressing the HOLD switch or changing to the manual mode), the system stops upon completion of the motion 5.
If the RESET key is pressed during the motion 3 through 5, the system also stops after completion of the motion 5.

5.3 General description of canned cycle

5.3.1 Command related to canned cycle motions

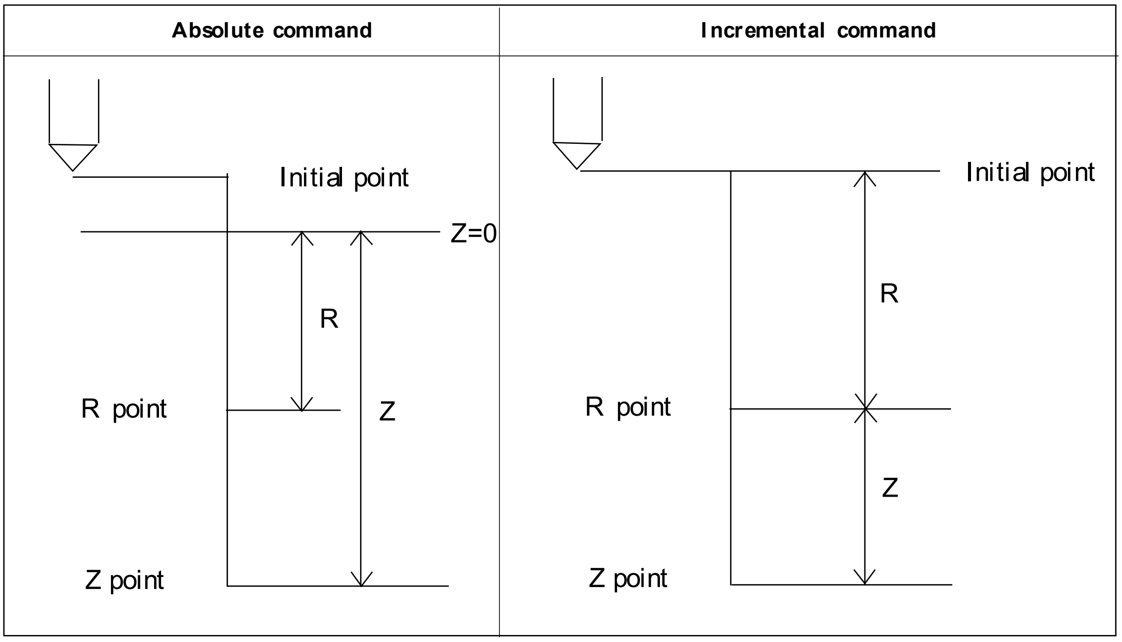
(1) Data format
- G90 Absolute command
- G91 Incremental command

(2) Return level
- G98 Initial point level return
- G99 R point level return

(3) Drilling mode
- G73 , G74
- G76 ~ G78
- G80 ~ G87
- G89
- G173
- G177 , G178
- G181 ~ G183
- G185 , G186
- G189

Refer to table 5-1(P5-2)

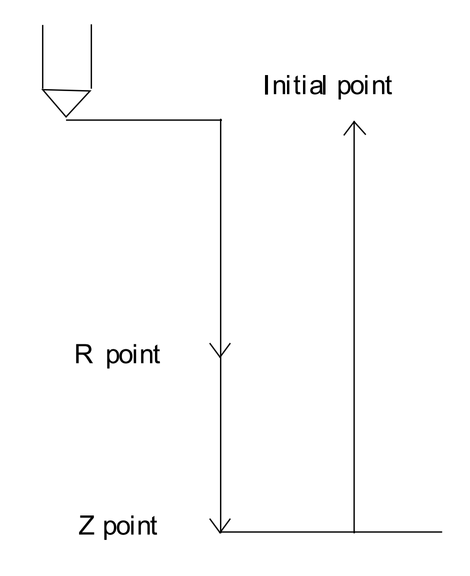
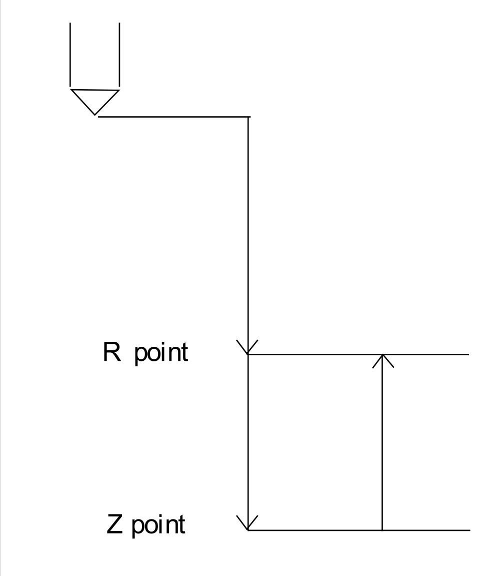
5.3.2 Setting of data in absolute/incremental command



eNCPR5.02.ai

5.3.3 Types of return point (G98, G99)

There are two types of return points - initial point level return (G98) and R point level return (G99) - when the canned cycle motions are finished.

Initial point level return (G98)	R point level return (G99)
	

eNCPR5.03.ai

(Note 1)
G98 and G99 are modal commands. G98 is effective when the power is turned ON.
(Note 2)
If there is no Z-axis movement even if the tool length offset mode is selected, the initial point is memorized without offset.
(Note 3)
The Z-axis machine coordinate value becomes the initial point when the canned cycle cancel mode changes to the canned cycle mode.

5.3.4 Canned cycle motion conditions

The canned cycle motions are available when the following commands are given.
(1) The drilling mode (G73, G74, G76~G78, G81~G87, G89, G173, G177, G178, G181~G183, G185, G186, G189) command blocks contain any of X, Y, Z, R, A, B or C.
(2) The blocks after the drilling mode command block through the canned cycle cancel block contain any of X, Y, Z, R, A, B or C.
(Note)
If there is no X, Y, Z, R, A, B or C in the block during the canned cycle, and the drilling data other than those are commanded, the drilling data only are memorized.

5.3.5 Machining data of canned cycle

Command format

G * *	X _ Y _	Z _ R _ Q _ P _ F _ S _	K _ ;
G code	Position data	Machining data	Repeat number

G code G73, G74, G76~G78, G81~G87, G89, G173, G177, G178, G181~G183, G185, G186and G189.
The G codes of the canned cycle are all modal.

X, Y : Drilling positon.
The tool motion to the drilling position is done at a rapid feed.

Z : Bottom position
When the incremental mode is selected, the distance from the R point to the bottom of hole is specified.

R : R point position.
When the incremental mode is selected, the distance from the point before the canned cycle becomes effective to the R point is specified.

Q : Cutting amount, shift amount, distance to feeding speed changeover point.
(1) Each cutting amount commanded by G73, G83, G173 or G183.
(2) Each cutting amount commanded by G77 or G78.
(3) Each shift amount commanded by G76 or G87.
(4) The distance to the feeding change point by G177or G178.
(5) Orientation angle by G86, G186

P : Dwelling time. (The unit is the same as specified by G04.)

F : Cutting feedrate.

S : Spindle speed.

K : Repeat number of canned cycle.

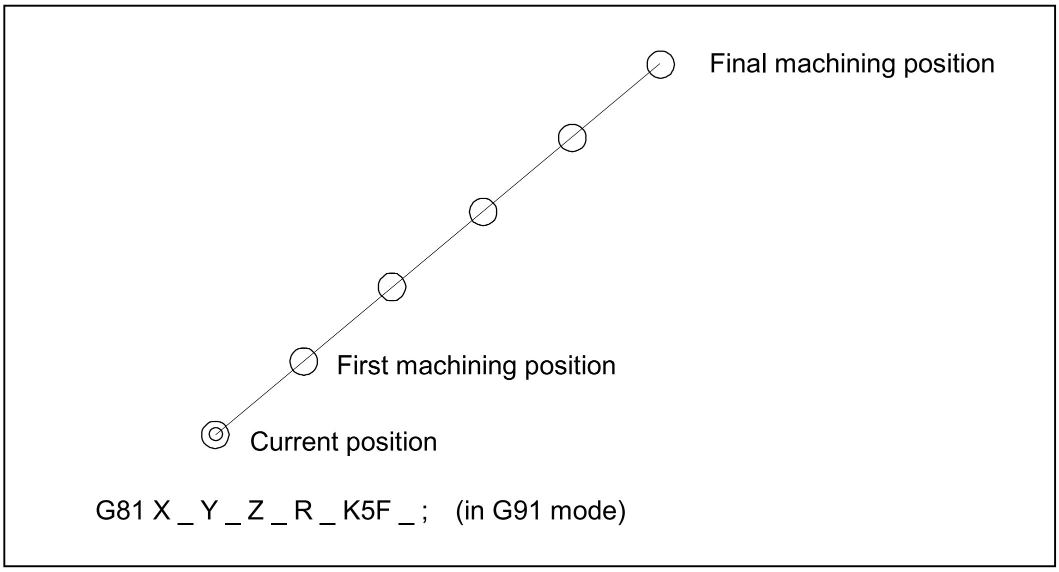
5.3.6 Repeat number of canned cycle

When drilling at an equal interval is repeated in the same canned cycle, use the address K and specify the repeat number.

The command range of K is 0 - 9999.
K is effective only in the specified block.
If K is not specified, the value of K is regarded 1.
When K0 is specified, drilling is not executed. Then the specified drilling data are memorized and the X and Y command are given, these axes move accordingly.

The programming of "X_Y_" commands the initial drilling position in incremental mode (G91).
If the absolute command (G90) is given, drilling is repeted at the same position.

(Example)



eNCPR5.04.ai

G81 X _ Y _ Z _ R _ K5F _ ; (in G91 mode)

5

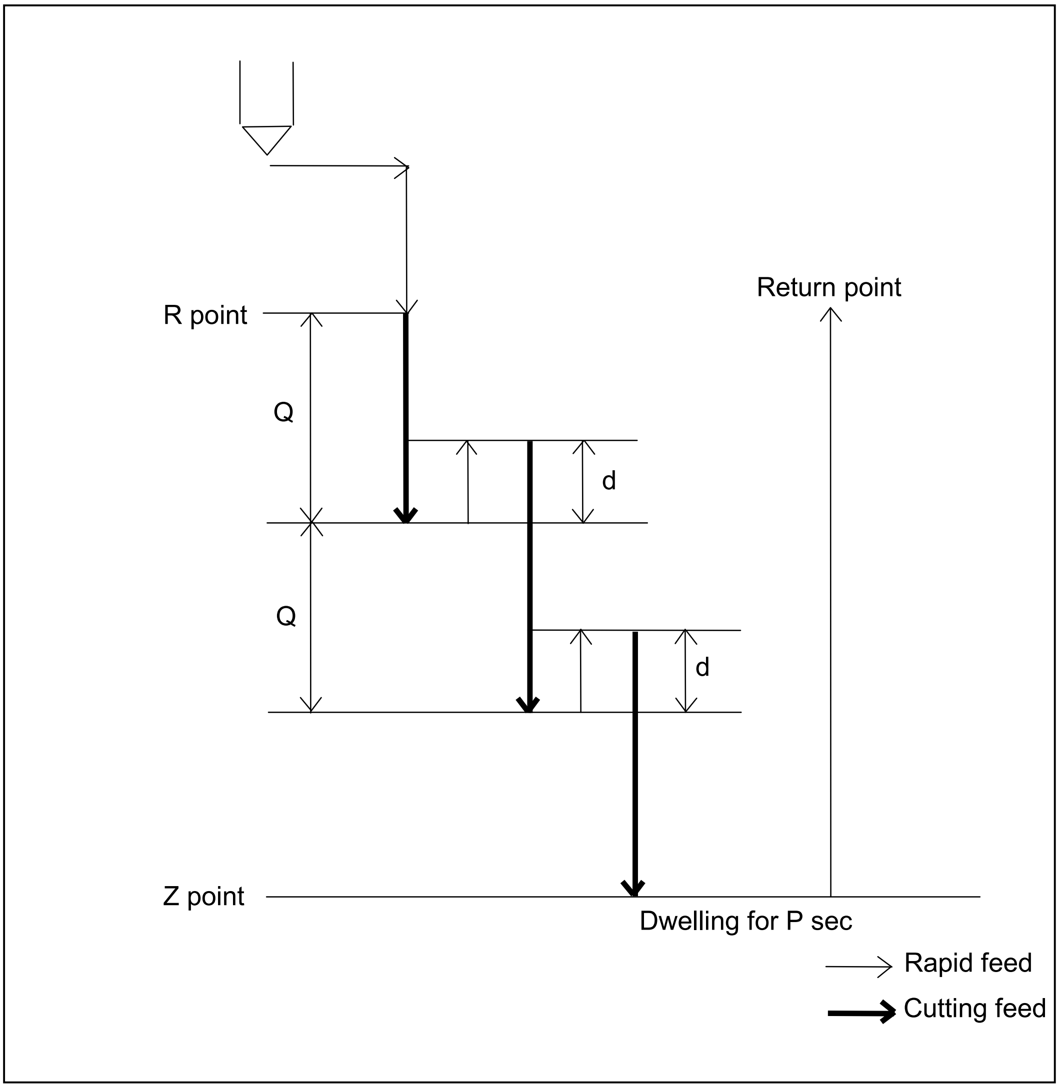
5

5.4 Details of canned cycle

5.4.1 High-speed peck drilling cycle (G73)

Command format

G73 X _ Y _ Z _ R _ P _ Q _ F _ ;

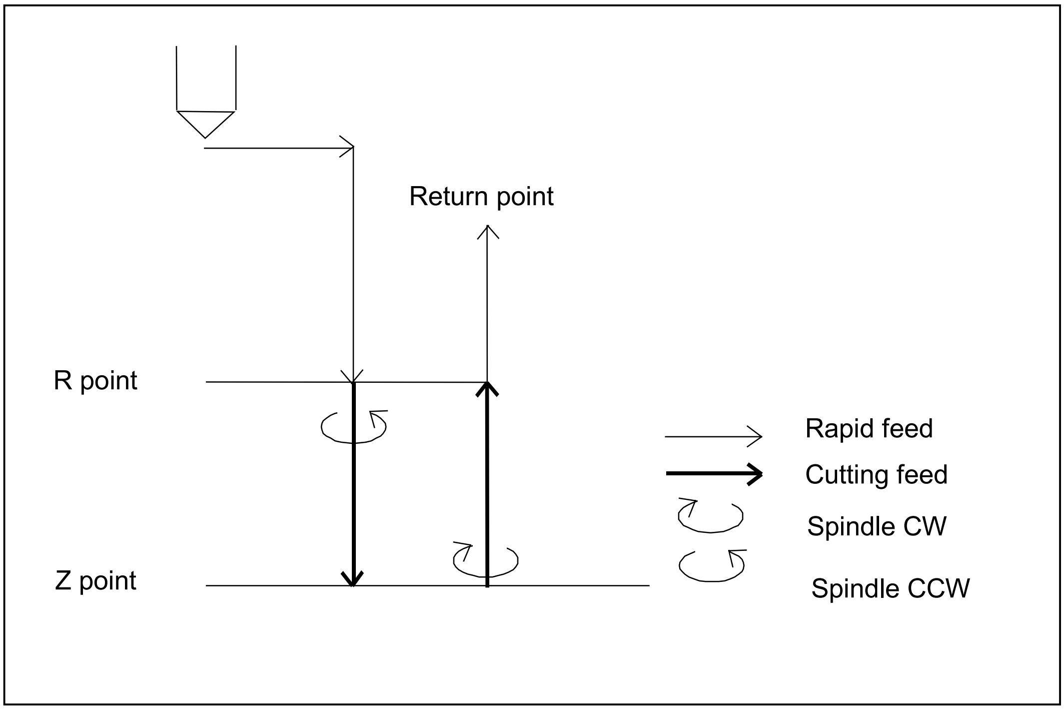


eNCPR5.05.ai

- The relief amount d is set by the parameter 1.
- If the minus value is commanded for the cutting amount Q, the algebraic mark (-) is ignored.

5.4.2 Reverse tapping cycle (G74)

Command format	G74 X _ Y _ Z _ R _ P _ F _ S _ ;



eNCPR5.06.ai

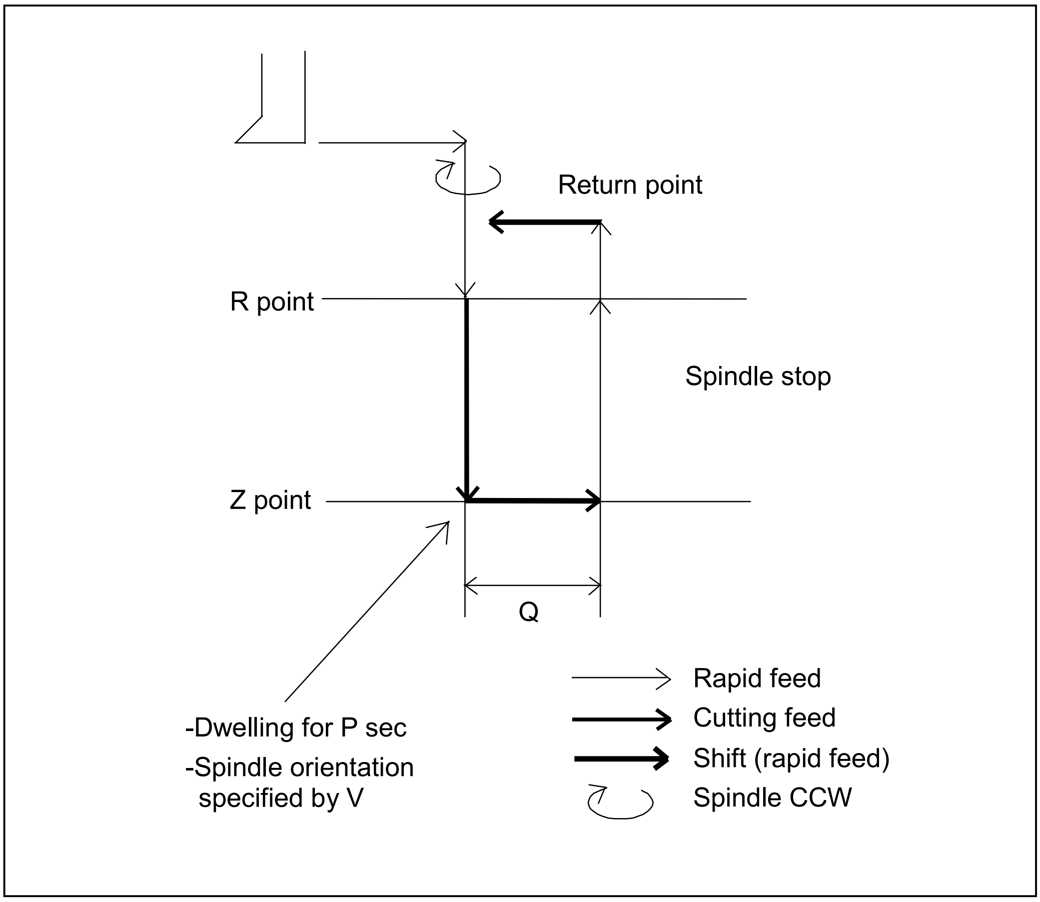
Spindle rotation stops at Z point.
After dwelling for P sec, spindle rotates CW.

- When a temporary stop is applied on the way from the R point to the Z point or the R point, the tool stops after returning to the R point.
- If the address S exceeds the max, spindle speed in tapping, an alarm is generated.

5

5.4.3 Fine boring cycle (G76)

Command format **G76 X _ Y _ Z _ R _ Q _ P _ F _ S _V _;**



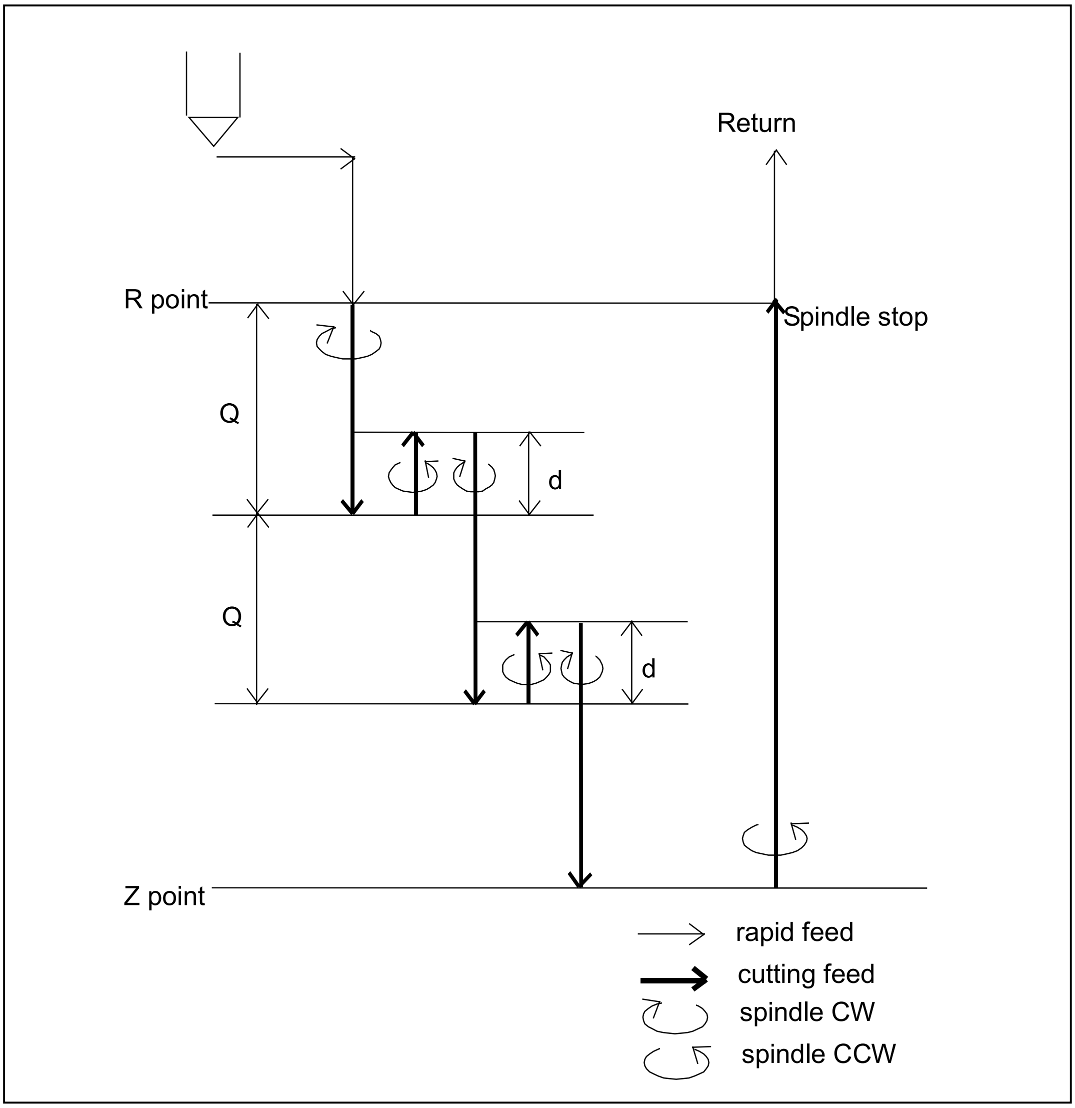
eNCPR5.07.ai

- If the minus value is commanded for the shift amount Q, the algebraic mark (-) is ignored.
- The shift direction is selected from +X, -X, +Y and -Y set by the parameter 1 in advance.
- The shift direction can be selected only from the above four. Therefore, the tool should be mounted so that the tool nose faces in one of the specified direction when the spindle orientation executes.
- When V is omitted, 0° is considered to be commanded.

5.4.4 Tapping cycle (G77)

Command format

G77 X _ Y _ Z _ R _ (I_ / J_) Q _ S _ ;



eNCPR5.08.ai

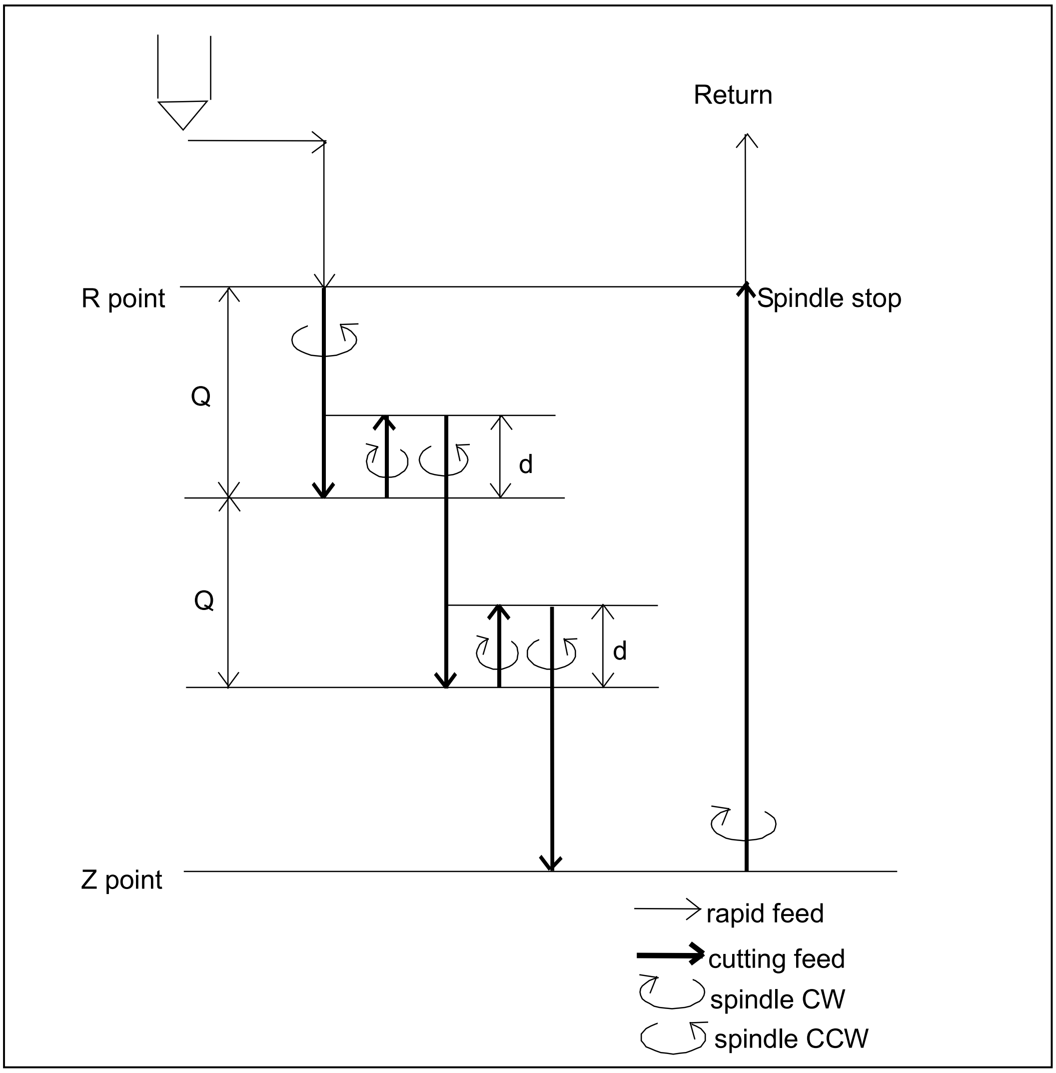
- The relief amount d is set by the parameter 1.
- If the minus value is commanded for the cutting amount Q, the algebraic mark (-) is ignored.
- When a temporary stop is applied on the way from the R point to the Z point or the R point, the tool stops after returning to the R point.
- A thread pitch or number of threads should be specified.
- Set the data on a thread pitch following the address I, and the data on a number of threads following the address J.
- If the address S exceeds the max, spindle speed in tapping, an alarm is generated.

5

5.4.5 Reverse tapping cycle (synchro mode) (G78)

Command format

G78 X _ Y _ Z _ R _ { I_ / J_ } Q _ S _ ;



- When a temporary stop is applied on the way from the R point to the Z point or the R point, the tool stops after returning to the R point.
- A thread pitch or number of threads should be specified.
  Set the data on a thread pitch following the address “I”, and the data on a number of threads following the address “J”
- When I and J commanded the same block, movement follow the “I” command value.
- If the address S exceeds the max, spindle speed in tapping, an alarm is generated.

- Tapping high-speed return
  The spindle speed at a return of synchro tapping (G77 or G78) is variable.

Command format

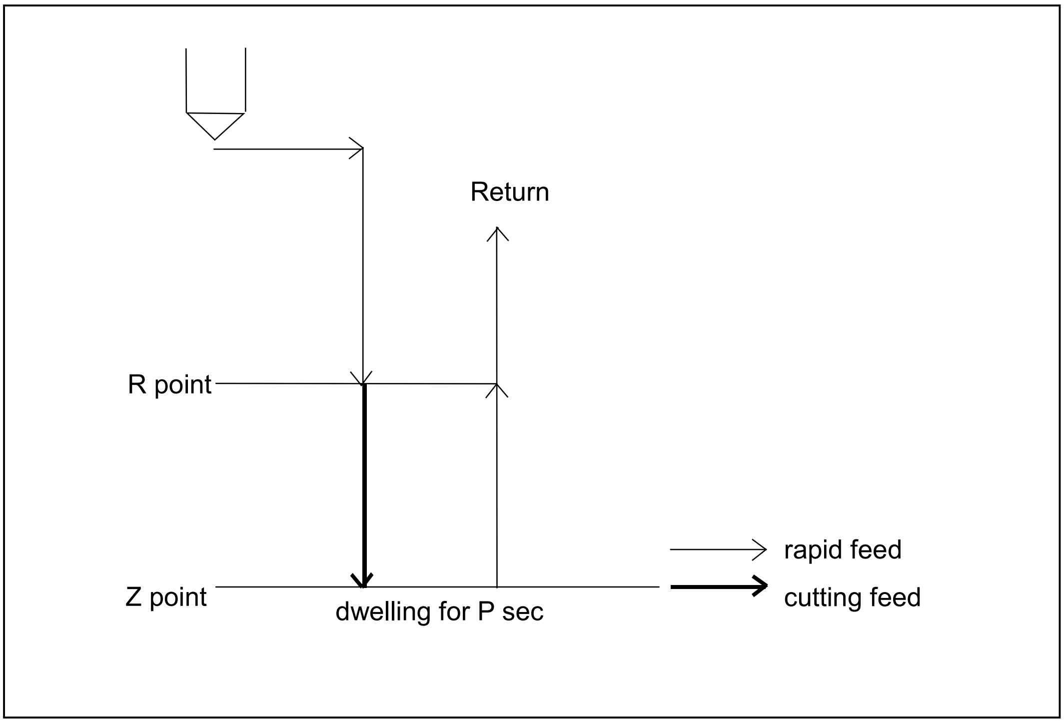
G77 / G78	X _ Y _ Z _ R _ Q _	I_ / J_	S _ L _ ;

- The address L commands the spindle speed during the return motion.
- When the address L is omitted, the spindle speeds at cutting and return motion become identical.
- The address L, once commanded, is regarded as modal in the canned cycle mode.
- If the address L command value is larger than the max, spindle speed in tapping, an alarm is generated and the tool stops at the R point.
- When the address L command value is smaller than the address S command value, the spindle rotates according to the address S command value.

5.4.6 Drilling cycle (G81, G82)

Command format

G81 / G82	X _ Y _ Z _ R _ P _ F _ ;



eNCPR5.10.ai

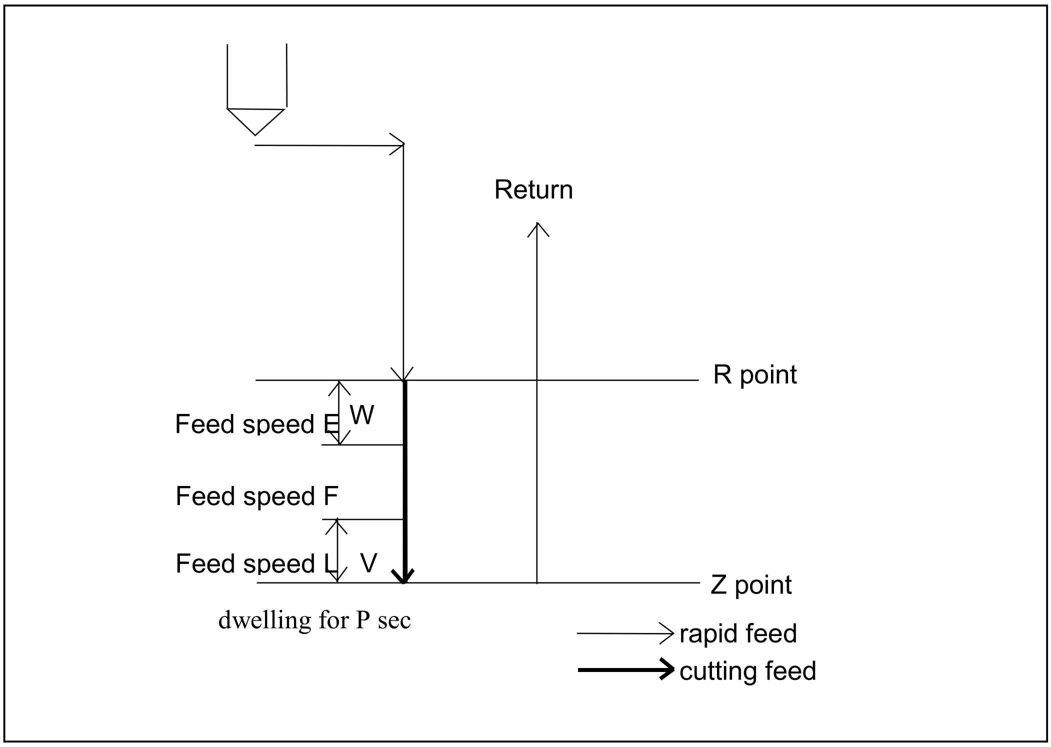
5

High speed cycle

Feed speed at start and end of drilling cycle (G81 or G82) is variable.

Command format $\begin{pmatrix} \textbf{G81} \\ \textbf{G82} \end{pmatrix}$ **X _ Y _ Z _ R _ W _ V _ F _ E _ L _ P _;**

W : Speed changeover point
Distance from point "R", regardless of absolute mode (G90) or incremental mode (G91)
E : Feed speed from point "R" to point specified by "W"
V : Speed changeover point
Distance from point "Z", regardless of absolute mode (G90) or incremental mode (G91)
L : Feed speed from point "Z" to point specified by "V"

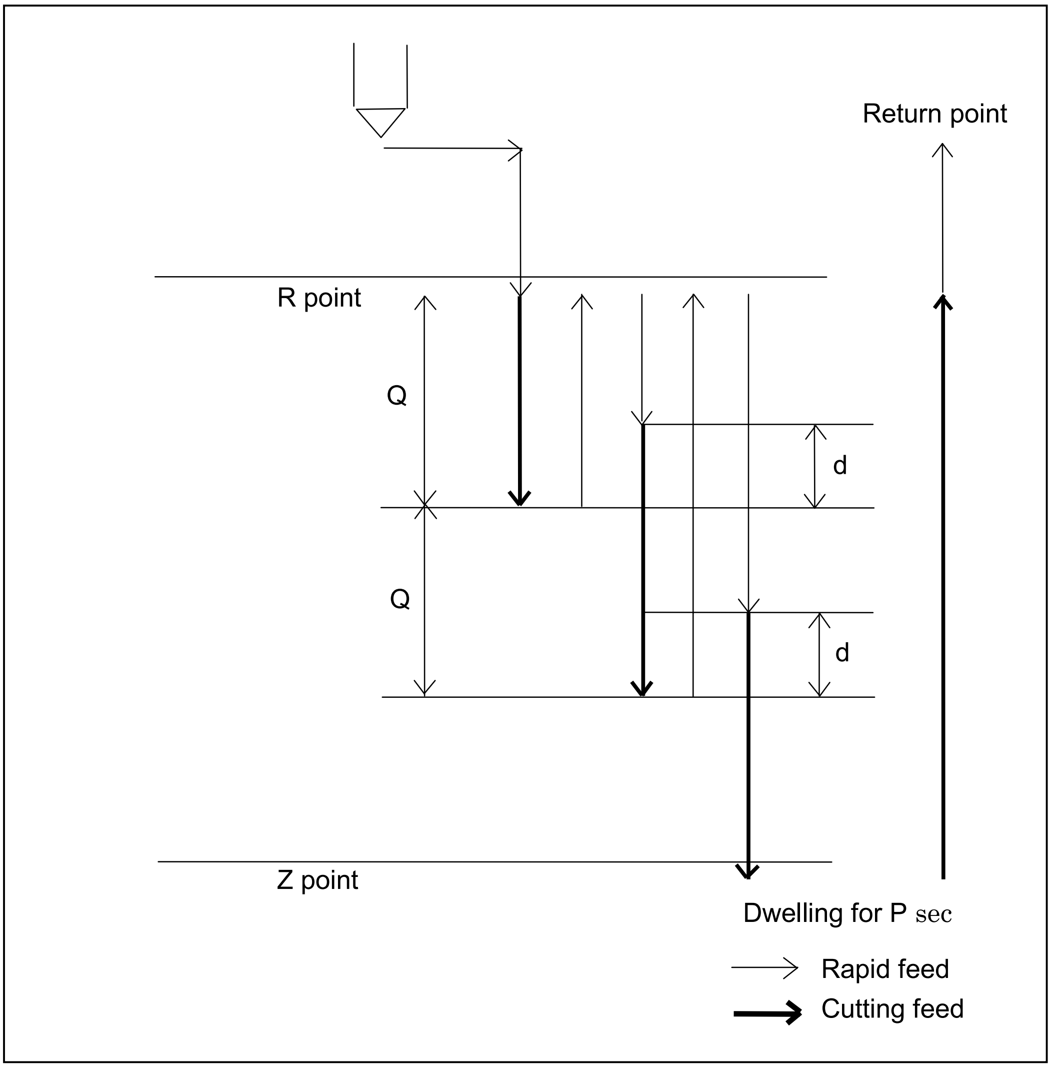


eNCPR5.11.ai

5.4.7 Peck drilling cycle (G83)

Command format

G83 X _ Y _ Z _ R _ P _ Q _ F _ ;



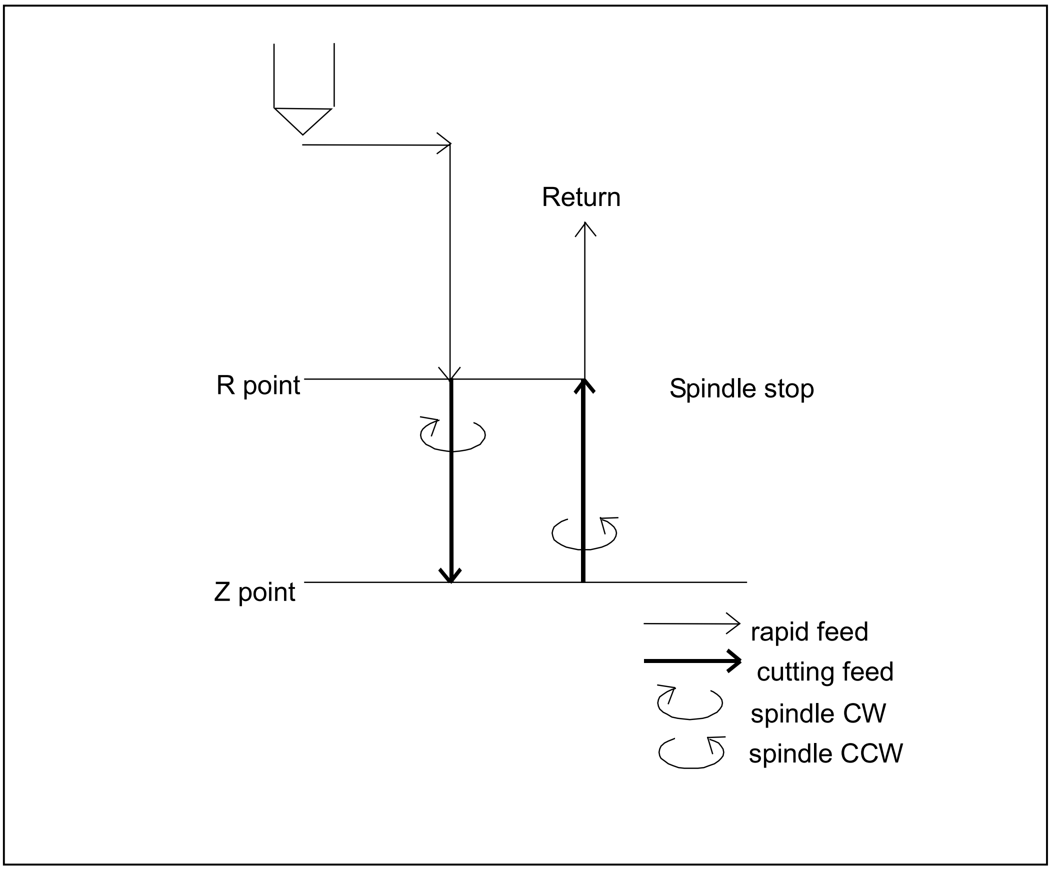
eNCPR5.12.ai

- The cutting start point d is set by the parameter.
- If the minus value is commanded for the cutting amount Q, the algebraic mark (-) is ignored.

5

5.4.8 Tapping cycle (G84)

Command format **G84 X _ Y _ Z _ R _ P _ F _ S _ ;**



eNCPR5.13.ai

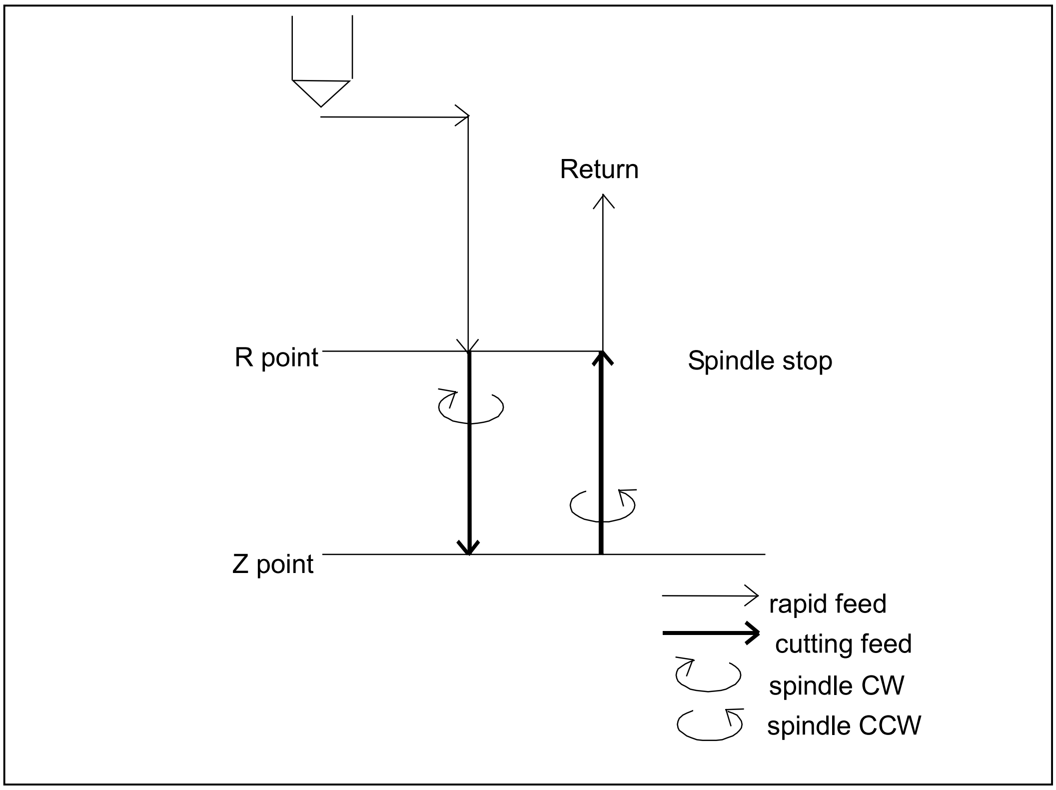
Spindle rotation stops at Z point.
After dwelling for P sec, spindle rotates CW.

- When a temporary stop is applied on the way from the R point to the Z point or the R point, the tool stops after returning to the R point.
- If the address S exceeds the max, spindle speed in tapping, an alarm is generated.

5.4.9 Boring cycle (G85, G89)

Command format

G85 G89	X _ Y _ Z _ R_ P_ F_ ;



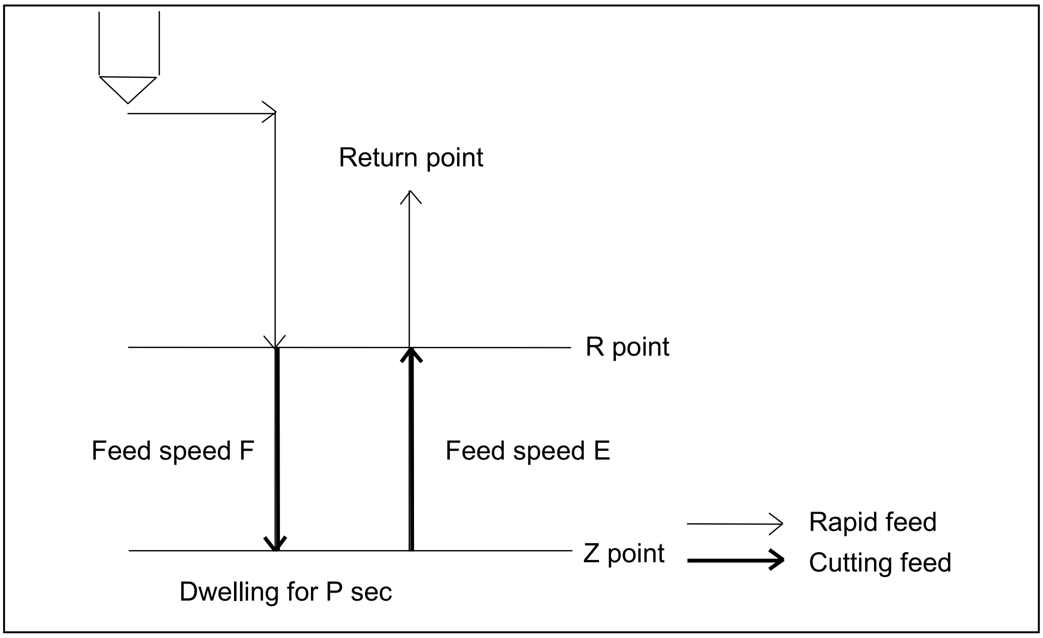
eNCPR5.14.ai

High speed cycle

Free speed at return of boring cycle (G85 or G89) is variable.

Command format

G85 G89	X _ Y _ Z _ R_ P_ F_ E_ P_ ;

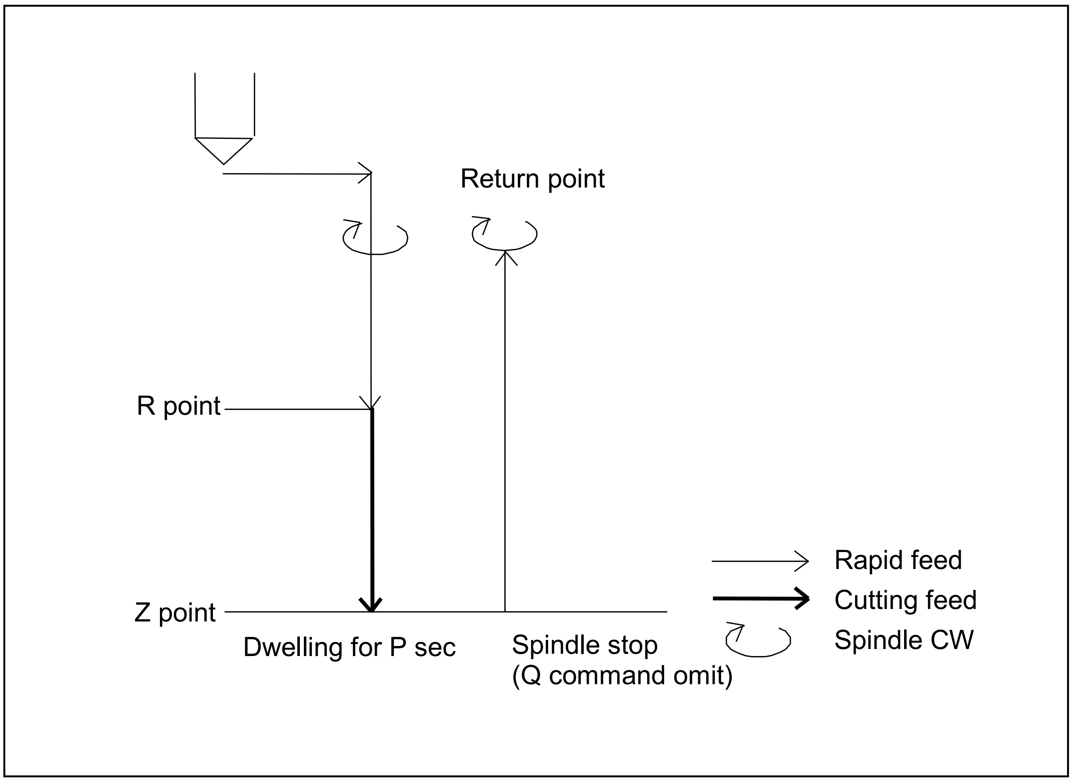


eNCPR5.15.ai

5.4.10 Boring cycle (G86)

Command format

G86	X _ Y _ Z _ R _ P _ F _ S_ Q_;



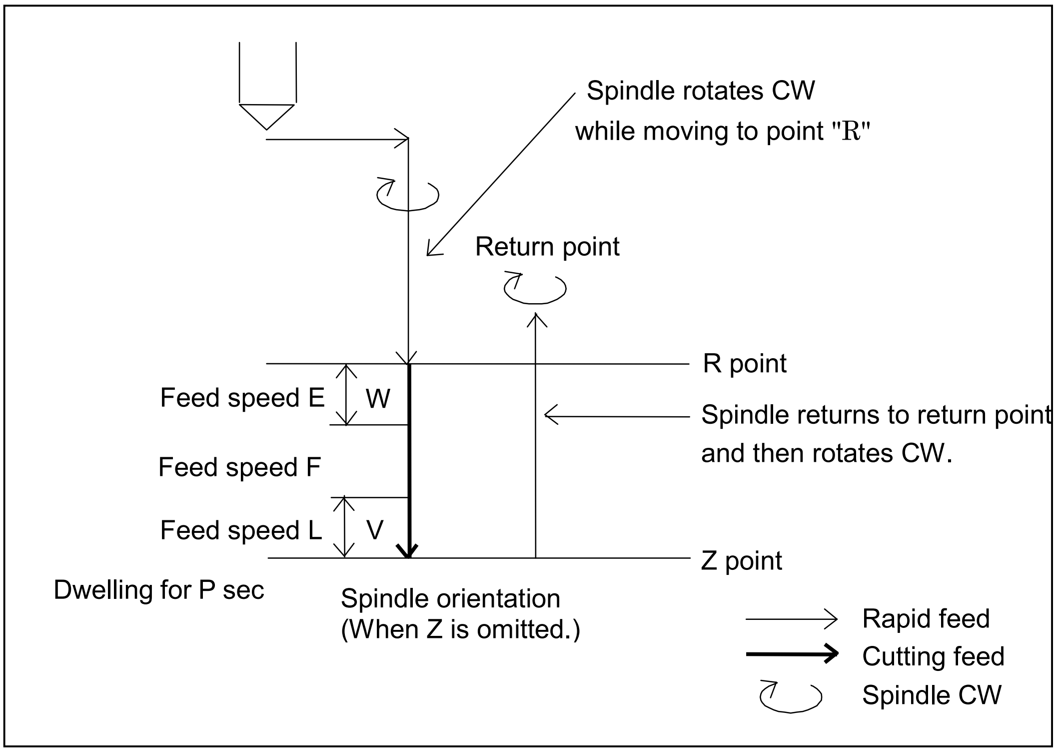
eNCPR5.16.ai

High speed cycle

Feed speed at start and end of boring cycle (G86) is variable.

Command format	**G86 X _ Y _ Z _ R _ W _ V _ F _ E _ L _ P_ Q_ ;**

W	:	Speed changeover point Distance from point "R", regardless of absolute mode (G90) or incremental mode (G91)
E	:	Feed speed from point "R" to point specified by "W"
V	:	Speed changeover point Distance from point "Z", regardless of absolute mode (G90) or incremental mode (G91)
L	:	Feed speed from point "Z" to point specified by "V"
Q	:	The angle of the spindle when orienting spindle at the fixed rotation point. When this command is omitted, only spindle orientation is performed.

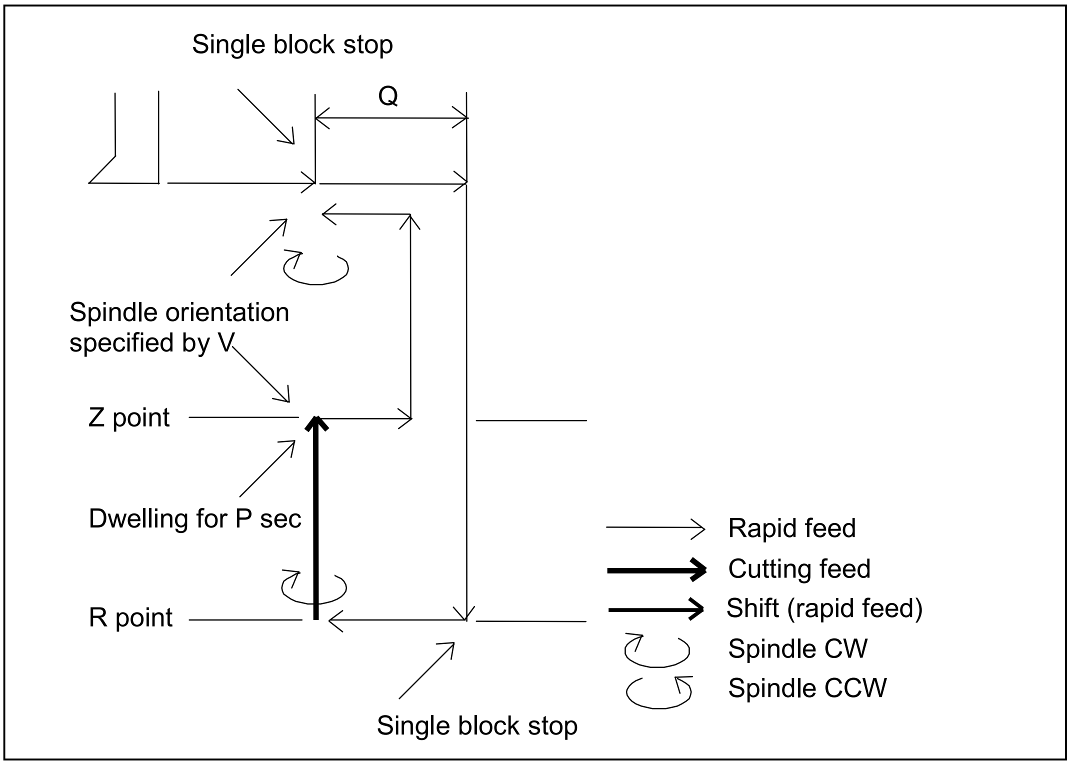


eNCPR5.17.ai

5.4.11 Back boring cycle (G87)

Command format

G87 X _ Y _ Z _ R _ Q _ P _ F _ S_ V_ ;



eNCPR5.18.ai

- If the minus value is commanded for the shift amount Q, the algebraic mark (-) is ignored.
- The shift direction is selected from +X, -X, +Y and -Y set by the parameter 1 in advance.
- The shift direction can be selected only from the above four. Therefore, the tool should be mounted so that the tool nose faces in one of the specified direction when the spindle orientation executes.
- G99 (R point level return) is unused.
- When V is omitted, 0° is considered to be commanded.

5.4.12 End mill tap cycle (G177)

Command format

G177 X _ Y _ Z _ R _ (I_ / J_) S_ L _ Q _E _ ;

Q : Feeding speed changeover point.
Distance from point "R", regardless of absolute mode (G90) or incremental mode (G91).
Start the tapping operation from this position.

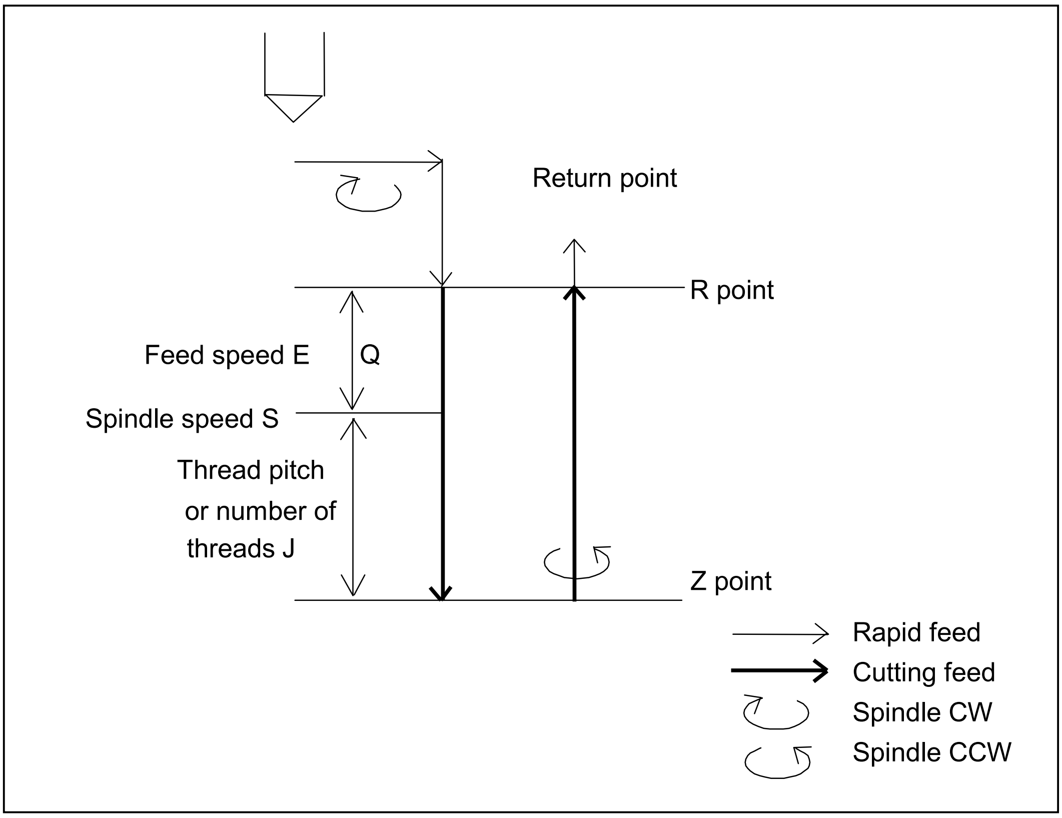
E : Feeding speed in zone "Q"

L : Spindle speed when returning from point 'Z' topoint 'R' when not specified, spindle returns at speed specified by 'S'.

I : Thread pitch in tap zone.

J : Number of threads pitch in tap zone.

S : Spindle speed.
Start spindle with XY axes moving.



eNCPR5.20.ai

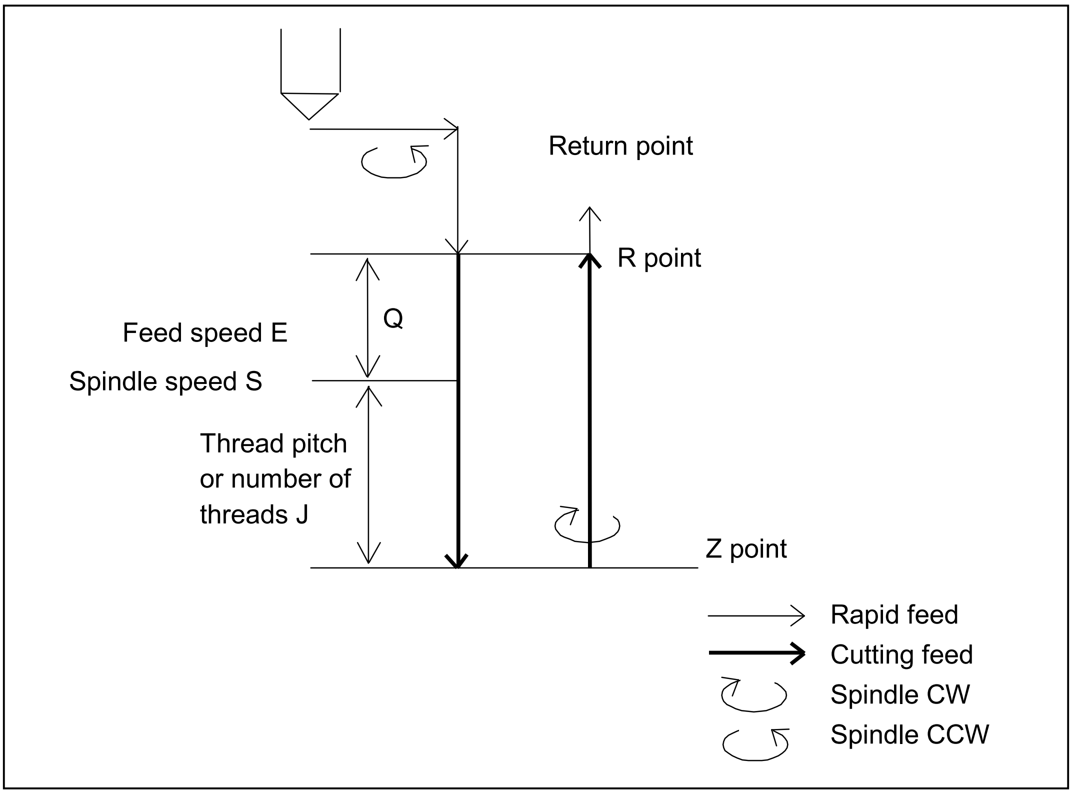
5

5.4.13 End mill tap cycle (G178)

Command format

G178 X _ Y _ Z _ R _ (I_ / J_) S_ L _ Q _E _ ;

Q : Feeding speed changeover point.
Distance from point "R", regardless of absolute mode (G90) or incremental mode (G91).
Start the tapping operation from this position.
E : Feeding speed in zone "Q"
L : Spindle speed when returning from point 'Z' topoint 'R' when not specified, spindle returns at speed specified by 'S'.
I : Thread pitch in tap zone.
J : Number of threads pitch in tap zone.
S : Spindle speed.
Start spindle with XY axes moving.

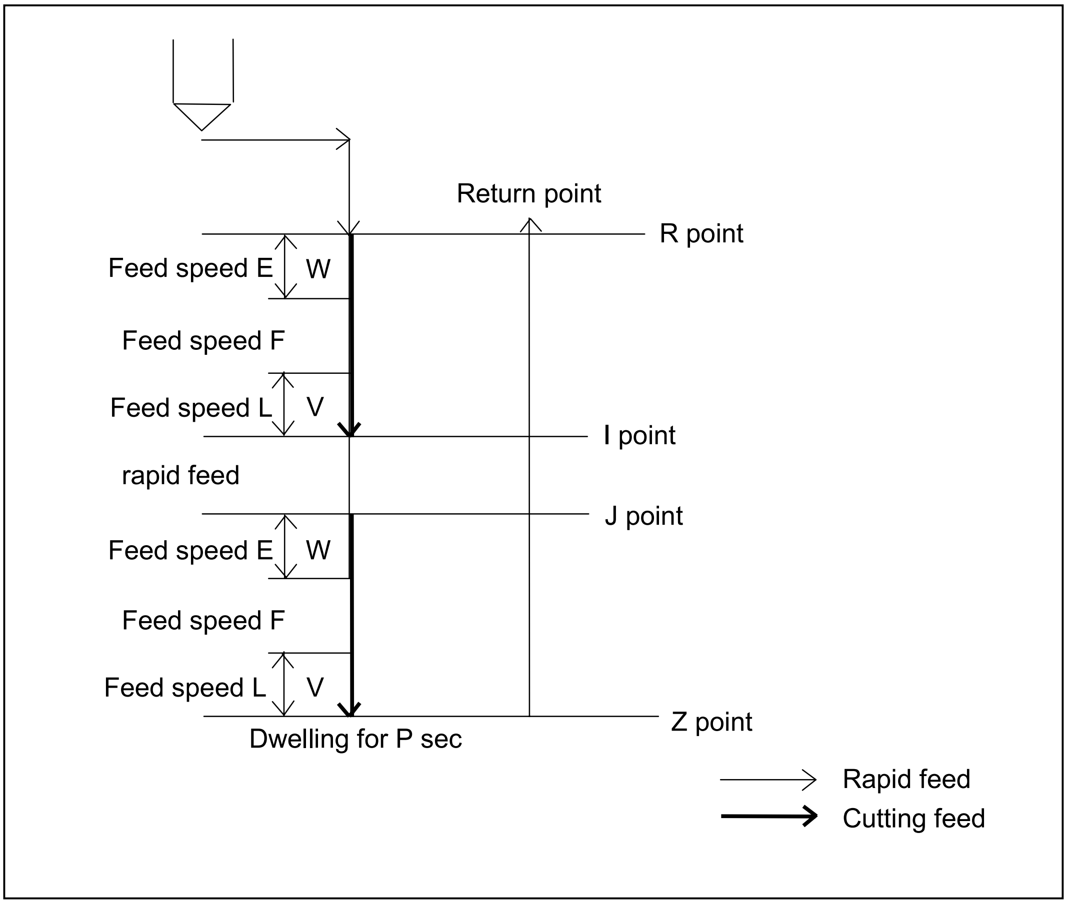


eNCPR5.21.ai

5.4.14 Double drilling cycle (G181,G182)

Command format	G181 / G182 X _ Y _ Z _ R _ I _ J _ W _ V _ F _ E _ L _P_ ;

I : Double rapid feed start point (follow G90/G91)
Distance from point "R" when incremental mode is specified
J : Double cutting feed start point (follow G90/G91)
Distance from point "I" when incremental mode is specified
W: Speed changeover point
Incremental regardless of absolute mode (G90) or incremental mode (G91)
E : Feed speed within range specified by "W"
V : Speed changeover point
Incremental regardless of absolute mode (G90) or incremental mode (G91)
L : Feed speed within range specified by "V".



eNCPR5.22.ai

5

5.4.15 Double boring cycle (G185,G189)

Command format

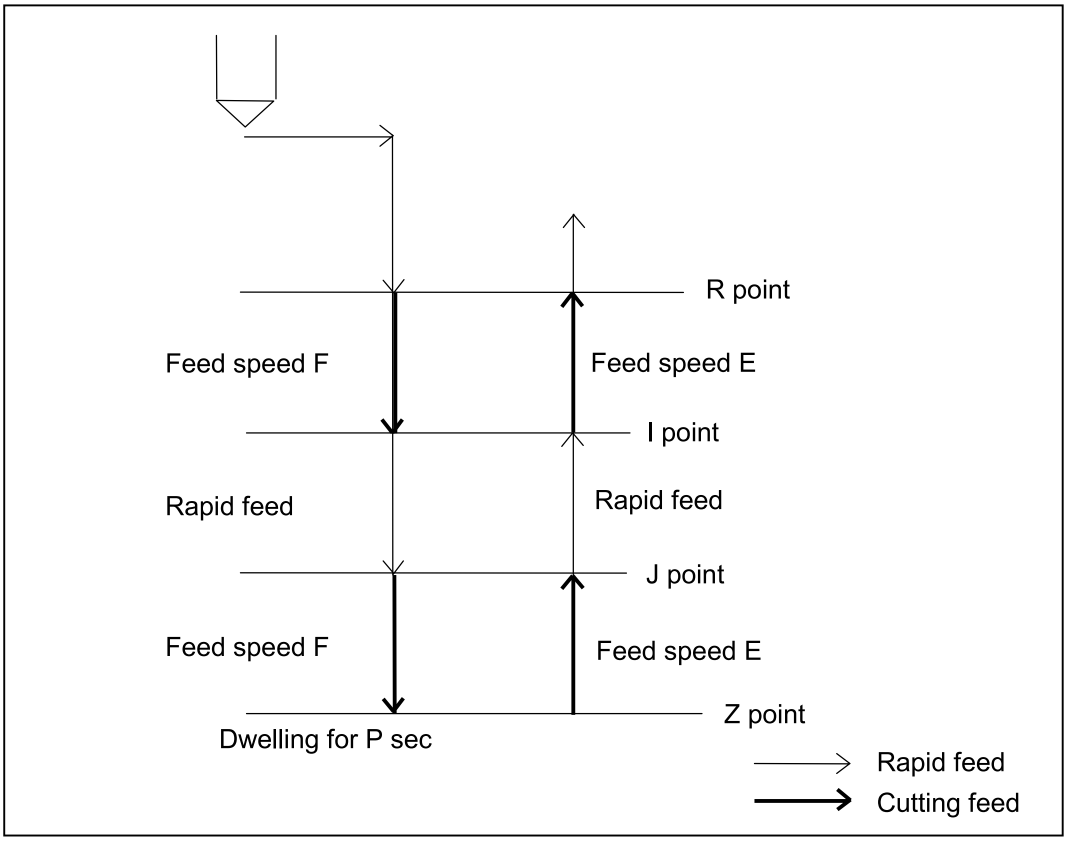
G185 / G189	X _ Y _ Z _ R _ I _ J _ F _ E _ P _ ;

I : Double rapid feed start point (follow G90/G91)
Distance from point "R" when incremental mode is specified

J : Double cutting feed start point (follow G90/G91)
Distance from point "I" when incremental mode is specified

F : Cutting feed speed from point "R" to point "Z"

E : Cutting feed speed from point "Z" to point "R"

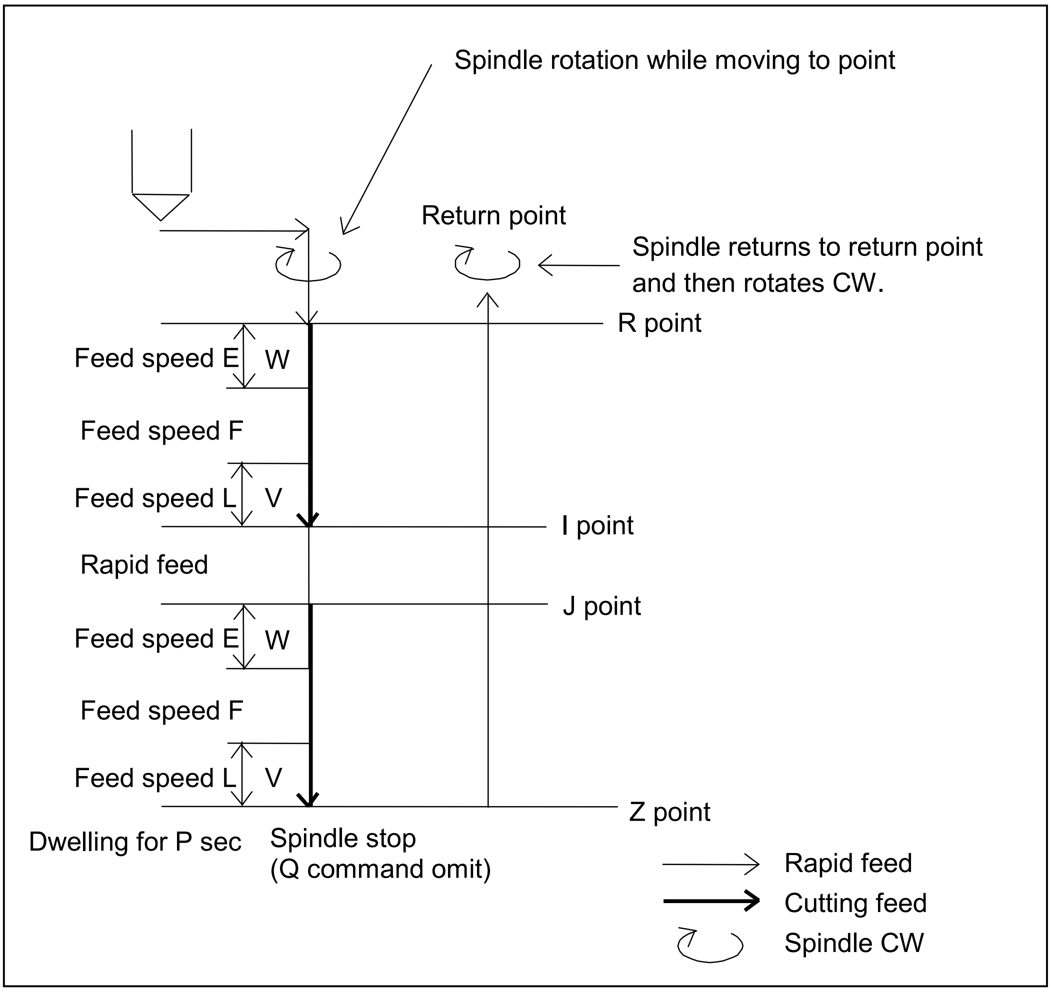


eNCPR5.24.ai

5.4.16 Double boring cycle (G186)

Command format **G186 X _ Y _ Z _ R _ I _ J _ W _ V _ F _ E _ L _ P _ Q _ S _ ;**

I : Double rapid feed start point (follow G90/G91)
Distance from point "R" when incremental mode is specified
J : Double cutting feed start point (follow G90/G91)
Distance from point "I" when incremental mode is specified
W : Speed changeover point
Incremental regardless of absolute mode (G90) or incremental mode (G91)
E : Feed speed within range specified by "W"
V : Speed changeover point
Incremental regardless of absolute mode (G90) or incremental mode (G91)
L : Feed speed within range specified by "V"
Q : Set angle for Z position to stop spindle at the constant rotate position.
When omit it, just stop the spindle.



eNCPR5.25.ai

5

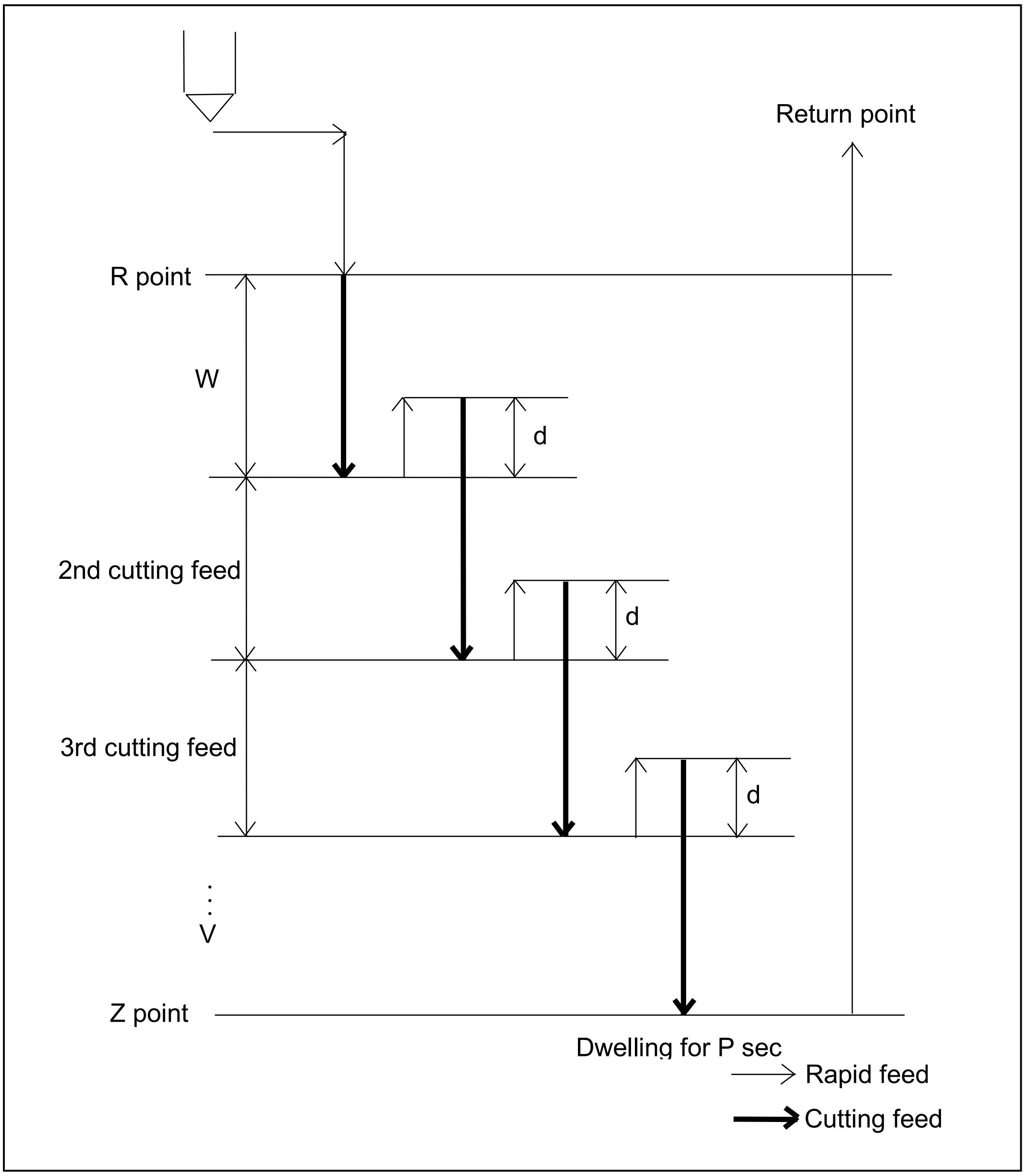
5.4.17 Canned cycle of reducing step

For G73, G77, G78, G83, G173 and G183 fixed cycles, reducing step is available which reduces the cutting feed depth gradually.

(1) High-speed peck drilling cycle (G73) (Reducing step)

Command format	G73 X _ Y _ Z _ R _ P _ W _ V _ F _ ;

W : 1st cutting feed
V : Minimum cutting feed



eNCPR5.26.ai

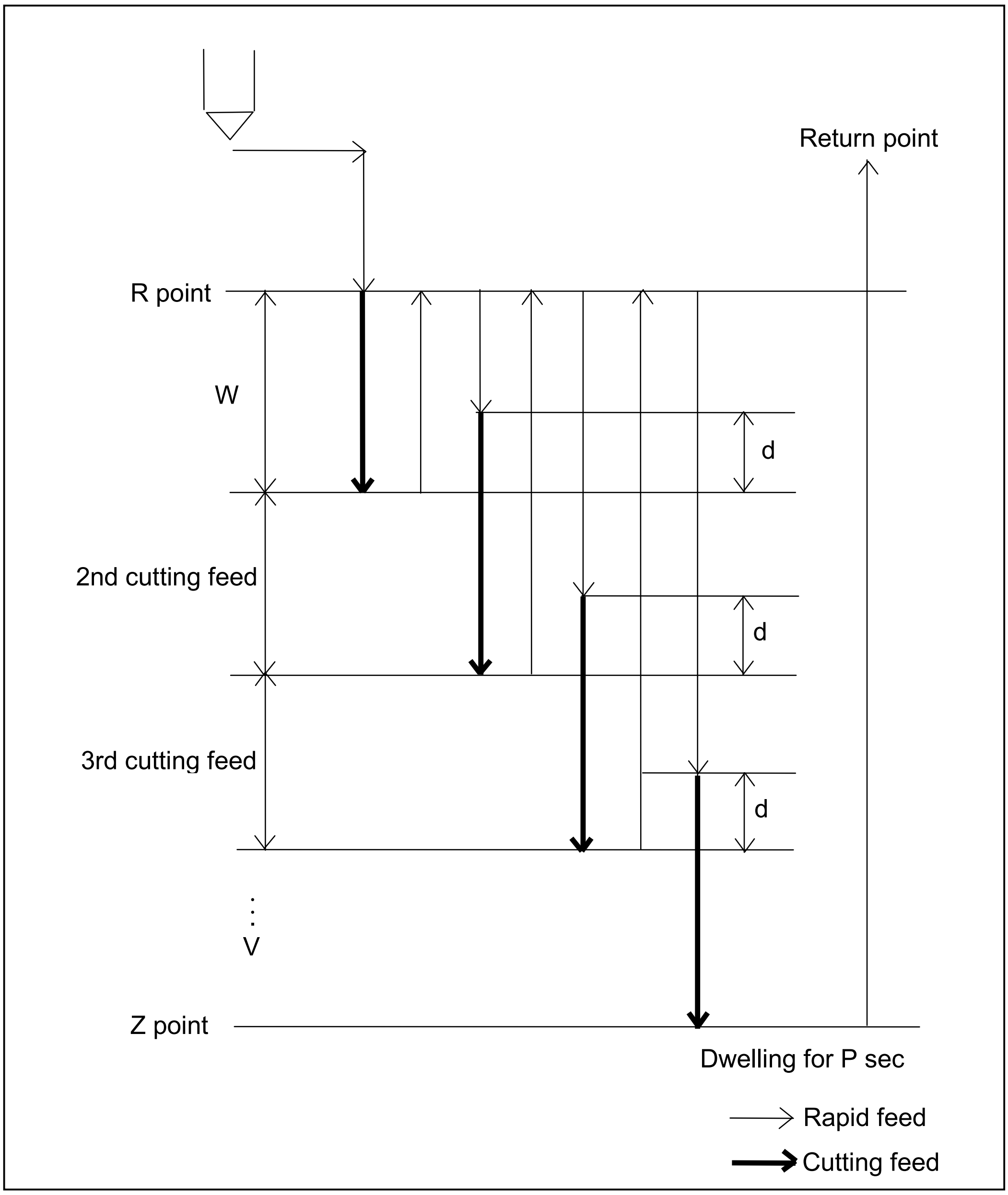
- The relief amount d is set by the parameter 1.
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.

(2) Peck drilling cycle (G83) (Reducing step)

Command format

G83 X _ Y _ Z _ R _ P _ W _ V _ F _ ;

W : 1st cutting feed
V : Minimum cutting feed



eNCPR5.27.ai

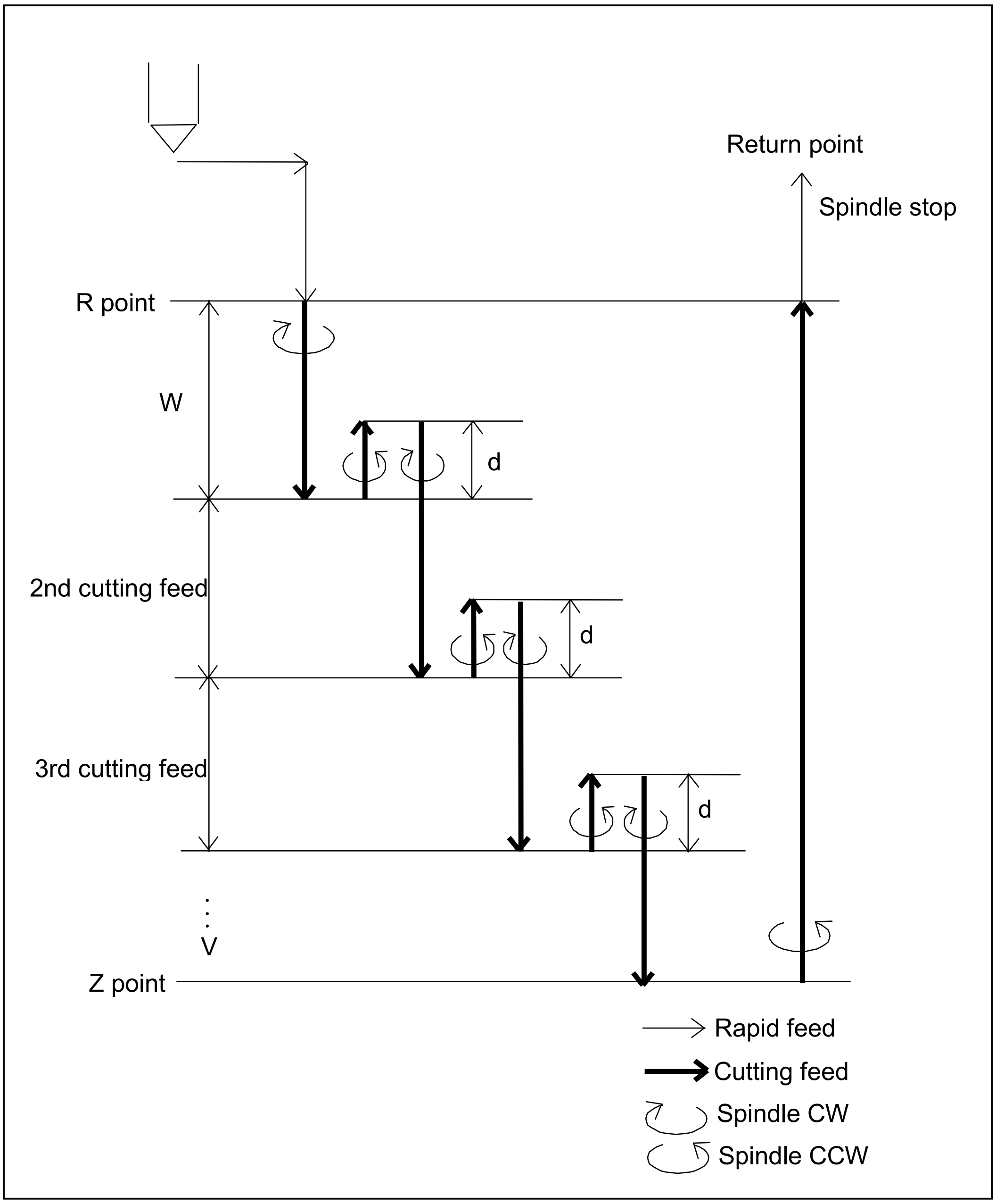
- The relief amount d is set by the parameter 1.
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.

(3) Tapping cycle (synchro mode)(G77) (Reducing step)

Command format

G77	X _ Y _ Z _ R _	I_ / J_	W _ V _ S;

W : 1st cutting feed
V : Minimum cutting feed



eNCPR5.28.ai

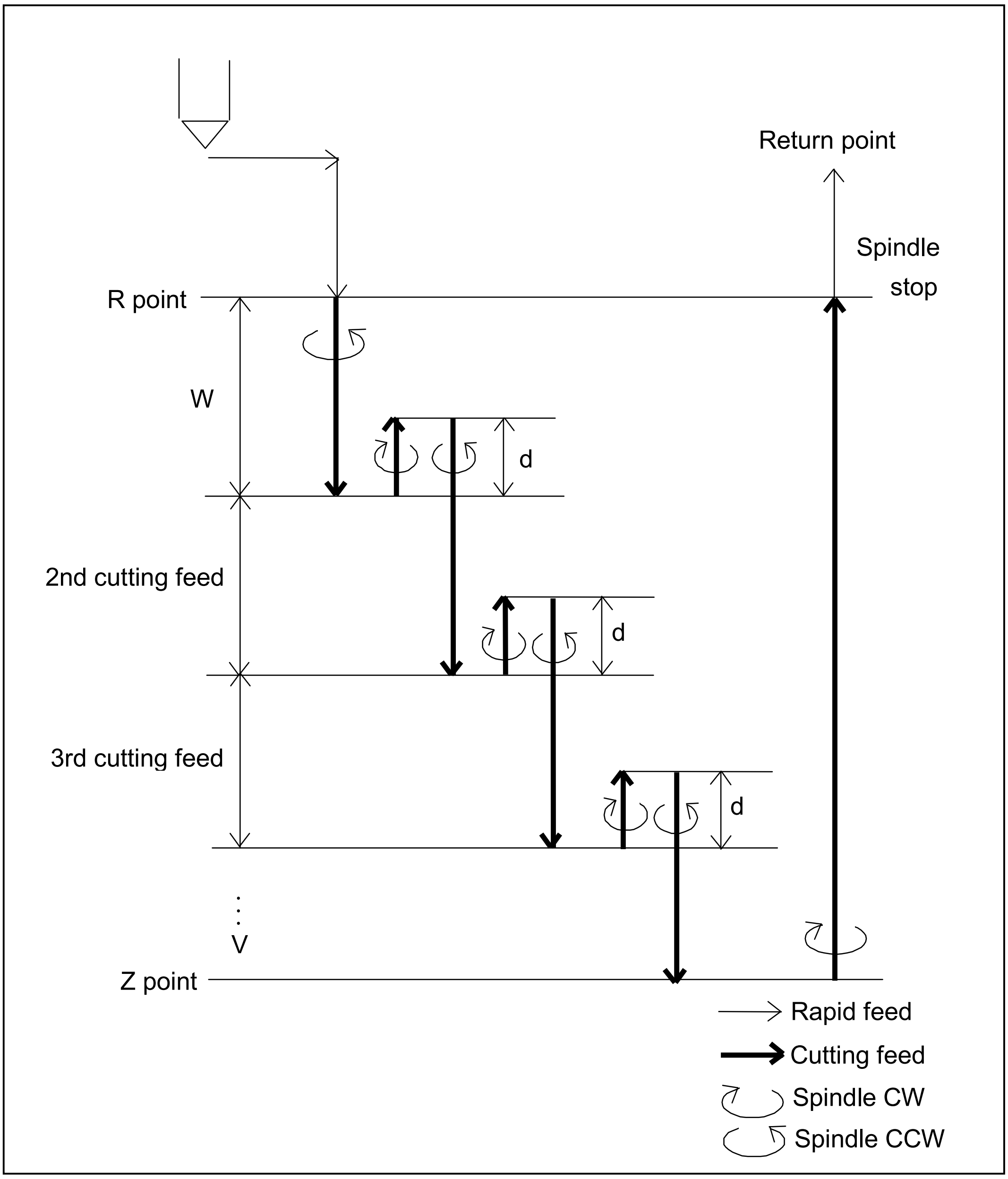
- The relief amount d is set by the parameter 1.
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.
- When a temporary stop is applied on the way from the R point to the Z point or the R point,
- the tool stops after returning to the R point.
- A thread pitch or number of threads should be specified.
- Set the data on a thread pitch following the address I, and the data on a number of threads following the address J.
- If the address S exceeds the max, spindle speed in tapping, an alarm is generated.

(4) Reverse tapping cycle (synchro mode)(G78) (Reducing step)

Command format

G78	X _ Y _ Z _ R_	I_ / J_	W _ V _ S _	;

W : 1st cutting feed
V : Minimum cutting feed



eNCPR5.29.ai

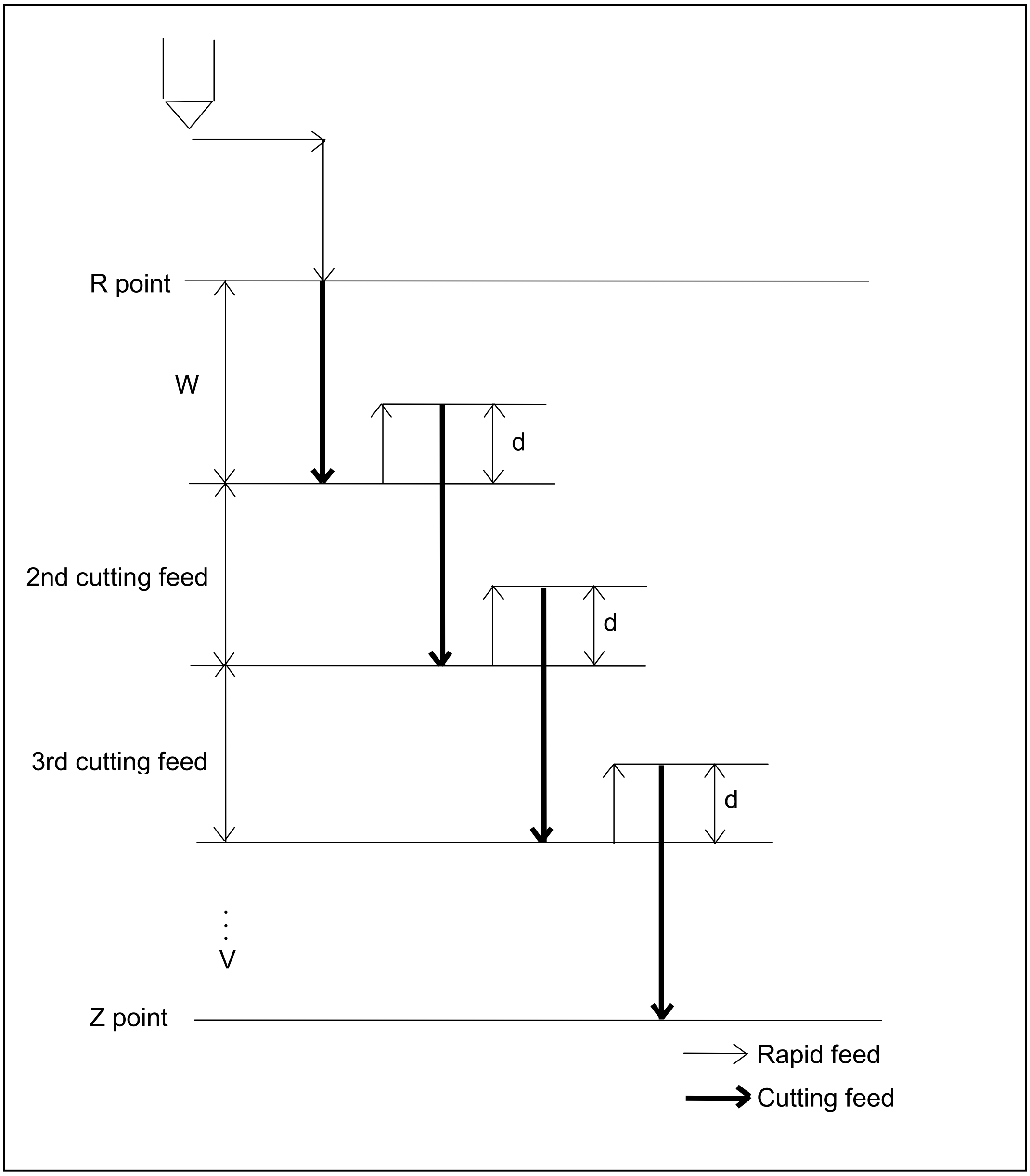
- The relief amount d is set by the parameter 1.
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.
- When a temporary stop is applied on the way from the R point to the Z point or the R point, the tool stops after returning to the R point.
- A thread pitch or number of threads should be specified.
- Set the data on a thread pitch following the address I, and the data on a number of threads following the address J.
- If the address S exceeds the max, spindle speed in tapping, an alarm is generated.

5

(5) High-speed peck drilling cycle (G173) (Reducing step)

Command format **G173 X _ Y _ Z _ R _ W _ V _ F _ ;**

W : 1st cutting feed
V : Minimum cutting feed



eNCPR5.30.ai

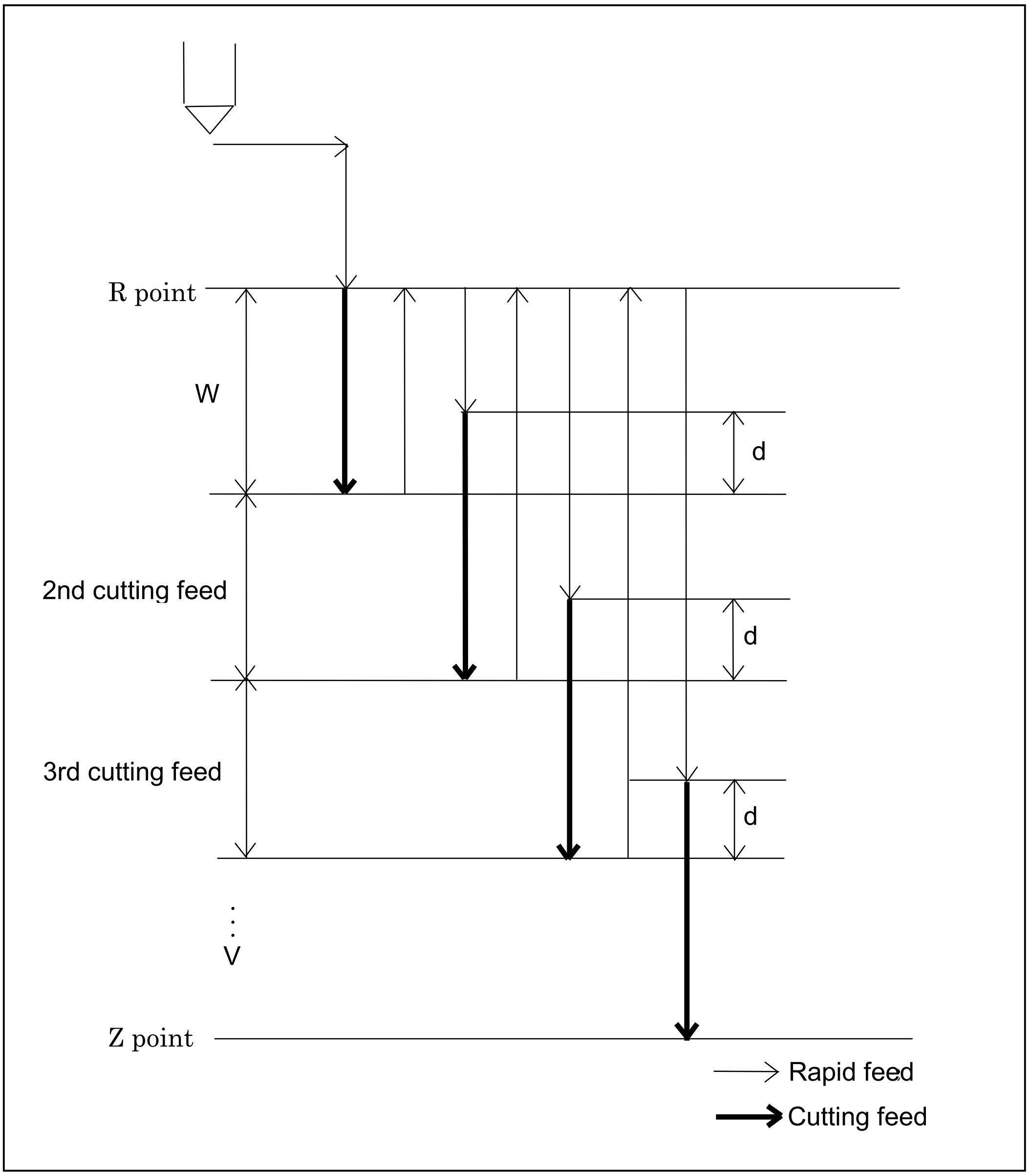
- The relief amount d is set by the parameter 1.
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.

(6) Peck drilling cycle (G183) (Reducing step)

Command format

G183 X _ Y _ Z _ R _ W _ V _ F _ ;

W : 1st cutting feed
V : Minimum cutting feed



eNCPR5.31.ai

- The relief amount d is set by the parameter 1.
- If a negative value is entered for the cutting amount V and W, the algebraic symbol (-) is ignored.

(7) For G73, G83, G173 and G183 fixed cycles, the cutting feed after the second time will be as below.
Cutting feed depth = Coefficient × 1st cutting feed (W)

Time of cutting	2	3	4	5	6	7	8	9	10
Coefficient	0.825	0.675	0.525	0.425	0.35	0.3	0.225	0.175	0.15

11	12	13	14
0.1	0.1	0.075	0.075

- Coefficient of 14th cutting is applied to 15th cutting and after.
- When the cutting feed has become smaller than the V value (minimum cutting feed), the V value is applied for cutting.

(8) For G77 and G78 fixed cycles, the cutting feed after the second time will be as below.
Cutting feed depth = Coefficient × 1st cutting feed (W)

Time of cutting	2	3	4	5	6	7	8	9	10
Coefficient	0.85	0.65	0.55	0.4	0.35	0.3	0.2	0.2	0.15

11	12	13	14
0.1	0.1	0.05	0.05

- Coefficient of 14th cutting is applied to 15th cutting and after.
- When the cutting feed has become smaller than the V value (minimum cutting feed), the V value is applied for cutting.

(Note 1) When switching between reducing step feed and fixed step feed, use the W and Q commands.
(Note 2) When the W and Q commands are specified to the same block, the W command has a priority.
(Note 3) When the W or Q command is not given, or 0 is given, cutting is carried out once.
When the V command is not given, or 0 is given, V=0.001 (metric) or V=0.0001(inch) is used.

5.4.18 Canned cycle cancel (G80)

The canned cycle (G73, G74, G76 to G78, G81 to 87, G89, G177, G178, G181 to G182, G185, G186, G189) is cancelled and ordinary motion becomes effective afterwards.
The R point, Z point, and other data on drilling are all cancelled.

Command format

G80 ;

(Note 1) The canned cycle is cancelled not only by G80 but also by G00-G03. The tool change command G100 and M06 can cancel the canned cycle after the tool change motion.
(Note 2) When the axis move command is given in the same block as G80, the axis movement starts after the canned cycle is cancelled.

5.4.19 Notes on canned cycle

(1) When commanding the canned cycle (G73, G81 to G83, G85, G89, G173,G181 to G183, G185, G189) which does not control the spindle rotation, the spindle should be rotated in advance by the M code.

(2) When the M code is commanded in the same block as the canned cycle, the M code is executed at the same time as the initial X/Y positioning or after that.
When the repetition is specified by K code, the M code is executed only at the first time and not executed any longer.

(3) When M00 or M01 is commanded in the same block as the canned cycle, the spindle and coolant stop after the X/Y positioning and automatic resetting is not available.
If resetting is necessary, command it in the manual operation mode or the MDI operation mode.

(4) When G00 to G03 are commanded in the same block as that of the canned cycle, the motion will be as below.

G00 G81 X_ Y_ Z_ R_ P_ F_ ;
In this case, the modal for G00 is updated and the canned cycle by G81 is executed.

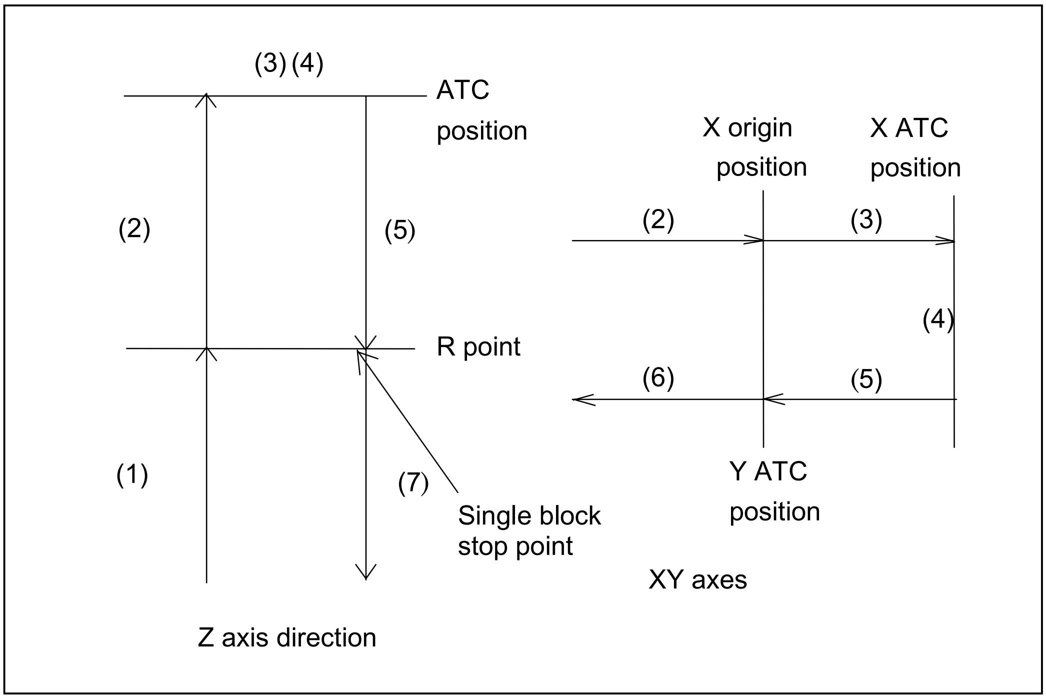
G81 G00 X_ Y_ Z_ R_ P_ F_ ;
In this case, X, Y, and Z axes move in accordance with G00 and the canned cycle is not executed.

(5) M200, M201, and M120 cannot be commanded in the same block as that of the canned cycle.

5.5 Canned cycle for tool change (non-stop ATC) (G100)

(1) When TC-32B.

Command format **G100 T _ X _ Y _ Z _ R _ A _ B _ L _ ;**



eNCPR5.34.ai

T _ _	:	Tool number.
T1 _ _	:	Pot number.
T9 _ _	:	Group number.
A, B	:	Target value when moving A and B axes at in rapid feed simultaneously with tool change motion.
X,Y	:	Target value of movement (6) in rapid feed.
Z	:	Target value of movement (7) in rapid feed.
R	:	Retern position before tool change motion. (Operated with tool length compensation applied) When "R" is not commanded, operation is performed using [ATC SYNCHRONOUS START POS] of USER PARAMETER 1 as "R" command value.
L	:	Specify the tool number, pot (magazine) number, and group number after “L” The pot with the corresponding tool attached is indexed by operation (4) after ATC. (Next tool preparation)

Operations

(1) Tool moves to the Z-axis point "R" while performing 0-degree spindle orientation.
When "T" is commanded, magazine swivels.
(2) Tool movement to theZ-axis ATC origin, to the X axis origin point and to Y axis ATC position, A, B axes movement to the the commanded value and also maga zine cover opening occur simultaneously.
(3) Tool moves to theX-axis ATC position.
(4) Arm swivels to change tool.
Tool change motion differs depending on setting on **[MAGAZINE TOOL]** screen.
Refer to the "Tool Change Motion" for detailes.
(5) Tool movement to the X axis orgin point and Z axis point "R" and magazine cover closing occurs stimultaneously.
(6) XY axes moves to the commanded value.
(7) Tool moves to Z-axis commanded position.

Caution

- **When performing cycle operation, tool moves in cutting mode between operations (1) and (6). And tool moves in position check mode.
For XY positioning at operation (2) and (3) (from X:origin Y: ATC position to X axis positioning (ATC position)) ,tool moves in cutting mode when Y axis positions at the ATC position before X axis does.**

- **When [RESET] key or [STOP SWITCH] key is pressed at operations (2) , machine stops after Z axis movement to ATC origin point.
For X, Y, A, and B axes movement, machine stops immediately.**

- **When [RESET] key or [STOP SWITCH] key is pressed between operations (3) and (5) , machine stops after motion (5) is completed.
For the operation (2), A, and B axes movement, machine stops immediately.**

- **Machine does not stop if single block occurs between operations (1) and (6).**

- **For operation (5), machine goes to the next opeation after closing the magazine cover without checking if the magazine cover is closed.**

- **Mode can not be switched between operation (2) and (6).**

- **All data other than G100 can be omitted. However, code "T" must be commanded once by operator before G100 is commanded.**

- **Tool offset is canceled when G100 is commanded.
Further, tool length compensation is canceled from operation (2).**

- **When tool offset (G41 or G42) and X and Y axes movement are commanded to block G100, tool offset begins when X and Y axes movement (6) commences.
Start-up is performed in format 1, regardless of parameters.**

- **When tool offset is commanded to block G100, tool offset becomes valid from operation (5) .**

- **When G100 is commanded while tool length compensation (G49) is canceled, operation (1) is performed subject to [ATC REFERENCE TOOL LENGTH] selection for user parameter.
When tool offset is not commanded to block G100, operations (5) are performed subject to [ATC REFERENCE TOOL LENGTH] selection for user parameter.**

- **Alarm appears when code M other than spindle command is commanded to block G100.
Alarm appears when axis A or B is commanded while optional A and B axes are not in use.**

5

- **When point "R" command position (5) is lower than Z axis command position (7), tool moves to Z-axis commanded position, and operation (7) is not performed.**

Tool change motion

Tool change motion differs depending on tool type set on **[MAGAZINE TOOL]** screen.

(When large tool is not set on **[MAGAZINE TOOL]**)

The following sequence is performed:
(1) Pot raises.
(2) Magazine swivels.
(3) Pot lowers.
(4) Arm swivels at high speed.

(When large tool is set on **[MAGAZINE TOOL]**)

Tool change from standard tool to standard tool, and large tool to large tool.

The following sequence is performed:
(1) Pot raises.
(2) Magazine swivels.
(3) Pot lowers.
(4) Arm swivels.
(5) Pot raises.

When pot is already at upper limit before tool change motion commences, sequence starts from (2).

When ((4) arm swivels) changing from large tool to large tool, arm swivels at low speed.
When ((4) arm swivels) changing from standard tool to standard tool, arm swivels at high speed.

Tool change from large tool to standard tool or vice versa.

The following sequence is performed:
(1) Pot raises.
(2) Magazine swivels. (Empty pot is indexed.)
(3) Pot lowers.
(4) Arm swivels.
(5) Pot raises.
(6) Magazine swivels. (Specified pot is indexed.)
(7) Pot lowers.
(8) Arm swivels.
(9) Pot raises.

When pot is already at upper or lower limit before tool change motion commences, sequence starts from (3) or (4), respectively.

For operation (2), empty pot is indexed appropriately for large tool and standard tool.

Arm swivels ((4) or (8)) at low speed when large tool is to be changed, and at high speed when standard tool is to be changed.

(Note1) **If empty pot for large tool is not available when change from large tool to standard tool is attempted, [*NO EMPTY POT] alarm appears.**
(Note2) **If empty pot for standard tool is not available when change from standard tool to large tool is attempted, [*NO EMPTY POT] alarm appears.**

Next tool preparation

Next tool preparation is performed after the arm has swiveled or the pot has risen after the arm swivels in the ATC sequence.
When ATC is not performed, only next tool preparation is performed.
Next tool preparation is not performed when the next tool is already indexed.

The next tool preparation sequence differs depending on the type of tool set on the magazine tool screen.

When large tool is not set on **[MAGAZINE TOOL]**
(1) Pot rises
(2) Magazine swivels
(3) Pot lowers
Next tool preparation ends when the above sequence is completed.
The sequence starts from (2) when the pot is already at the upper limit before next tool preparation commences.

When large tool is set on **[MAGAZINE TOOL]**
Both spindle tool and next tool are standard or large
(1) Pot rises
(2) Magazine swivels
(3) Pot lowers
Next tool preparation ends when the above sequence is completed.
The sequence starts from (2) when the pot is already at the upper limit before next tool preparation commences.

Spindle tool is standard and next tool is large or vice versa
(1) Pot rises
(2) Magazine swivels (empty pot is indexed)
(3) Pot lowers
Next tool preparation ends when the above sequence is completed.
The sequence starts from (2) when the pot is already at the upper limit before next tool preparation commences.

(Note 1) When the next tool is changed from a large tool to a standard tool and there is no empty pot for a large tool, the [NO EMPTY POT] error occurs.
(Note 2) When the next tool is changed from a standard tool to a large tool and there is no empty pot for a standard tool, the [NO EMPTY POT] error occurs.

5

(This page is a blank.)

CHAPTER 6

PREPARATION FUNCTION (COORDINATE CALCULATION)

6 Coordinate calculation function

This function is for calculating the point group coordinates in one block.
Point groups are such as on a linear line, on a grid and on a circular arc.
By combining such functions as the canned cycle etc., drilling at each point group is available by one command.

6.1 List of Coordinate Calculation Function

Table 6-1 List of coordinate calculation function

G code	Name	Function
G36	Bolt hole circle	Coordinate calculation of point group on circular arc.
G37	Linear (angle)	Coordinate calculation of point group on linear line by angle designation.
G38	Linear (X,Y)	Coordinate calculation of point group of linear line by coordinate designation.
G39	Grid	Coordinate calculation of point group on grid.

6.2 Coordinate Calculation Parameter

Command format

```
G36
 ·
 ·    X_Y_I_J_K_P_Q_     ;
 ·
G39
```

X,Y: Reference point of coordinate calculation
I,J,K,P,Q: Coordinate calculation parameter

(1) Reference point coordinate (X,Y)
This should be commanded by the working coordinate system.
When this is omitted, the current position becomes the reference point.

(2) Coordinate calculation parameter (I,J,K,P,Q)
This should be commanded in the same block as the G36~G39.
This parameter is effective only in the block, and automatically cleared after calculation is finished.

Relation of each calculation function and parameter is as shown below.

Function	G code	Parameter				
		I	J	K	P	Q
Bolt hole circle	G36	●	●	●	●	
Linear(angle)	G37	●	●	○		
Linear(X,Y)	G38	●	●	○		
Grid	G39	●	●	●	●	●

● : Be sure to specify. Otherwise, alarm occurs.
○ : If the command is omitted, it is regarded as 1.
Brank : It is no need to enter any data. Any data will be ignored.

6.3 Details of Coordinate Calculation Function

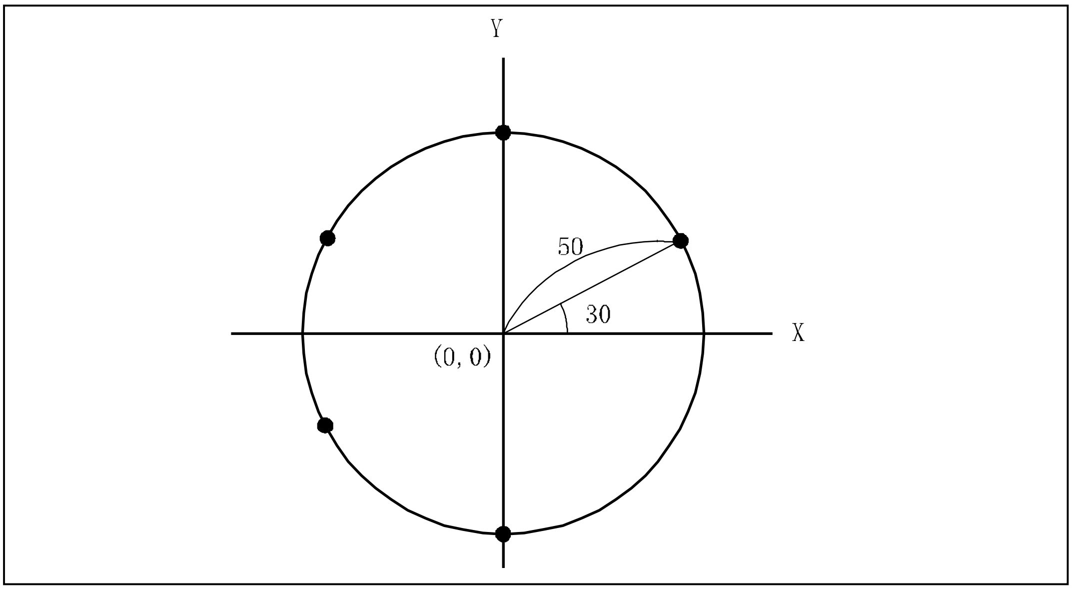
6.3.1 Bolt hole circle

While setting the commanded coordinate value as a circular center, the coordinate value equally dividing a circular arc from a certain point is calculated.

Command format **G36 X_ Y_ I_ J_ K_ P_ ;**

X,Y : Coordinate of the circular arc center
I : Circular arc radius
J : Angle of the start point to the X axis
K : No. of machining (Max. 999)
P : No. of divisions (Max. 999.999)

(Ex.) G36 X0 Y0 I50 J30 K5 P6 ;



eNCPR6.05.ai

(Note) Coordinate values are calculated from the start point in the CCW direction.

6

6

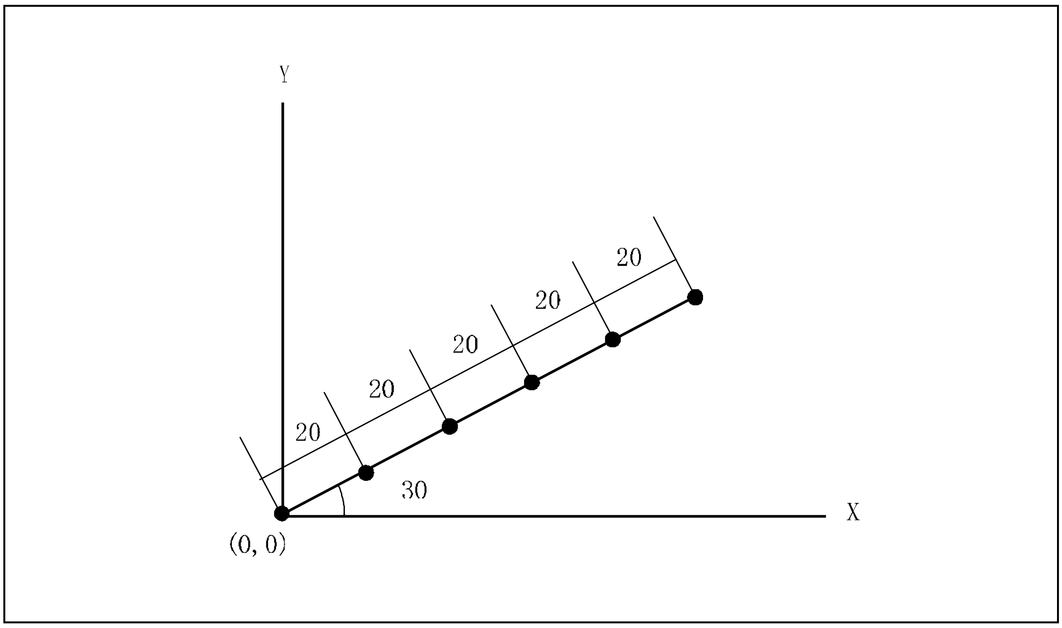
6.3.2 Linear (angle)

With the reference point at the commanded coordinate, the coordinate values along the linear line at the angle (θ°) formed by the X axis are calculated.

Command format

G37 X_ Y_ I_ J_ K_ ;

X,Y: Coordinate of reference point
I : Interval between machning points
J : Angle intersecting the X axis
K : No. of machining (Max. 999)



eNCPR6.01.ai

(Note 1) When K is omitted, it is regarded as 1.
(Note 2) The reference point becomes the first machining point.

6.3.3 Linear (X, Y)

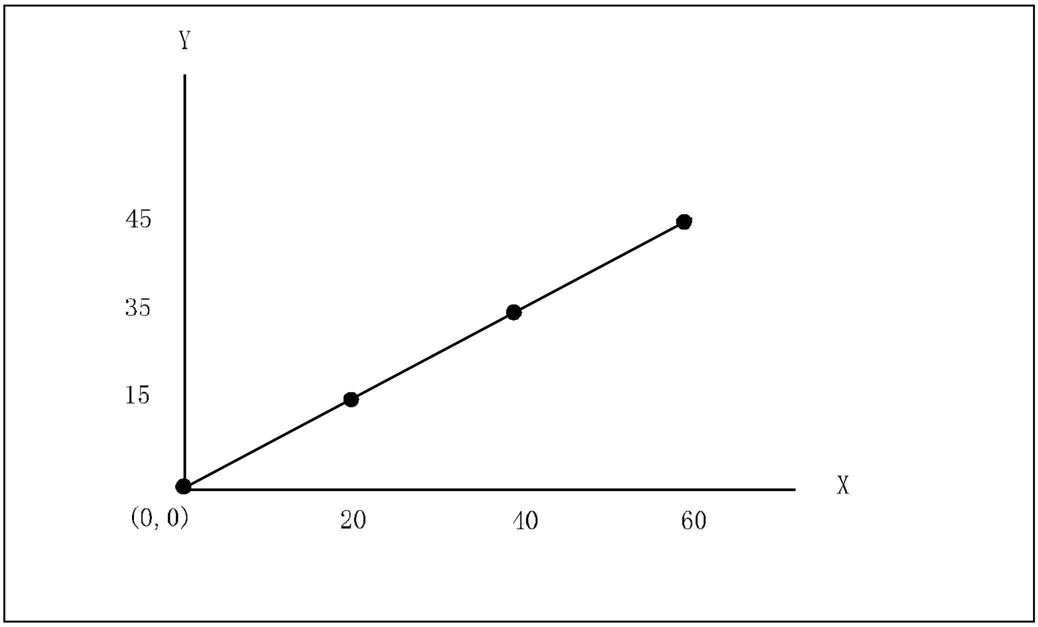
With the reference point at the commanded coordinate, the coordinate values added in the X and Y directions respectively are calculated.

Command format

G38 X_ Y_ I_ J_ K_ ;

X,Y : Coordinate of reference point
I : Interval on the X axis
J : Interval on the Y axis
K : No. of machining (Max. 999)

(Ex.) G38 X0 Y0 I20 J15 K4 ;



eNCPR6.02.ai

(Note 1) When K is omitted, it is regarded as 1.
(Note 2) The reference point becomes the first machining point.

6

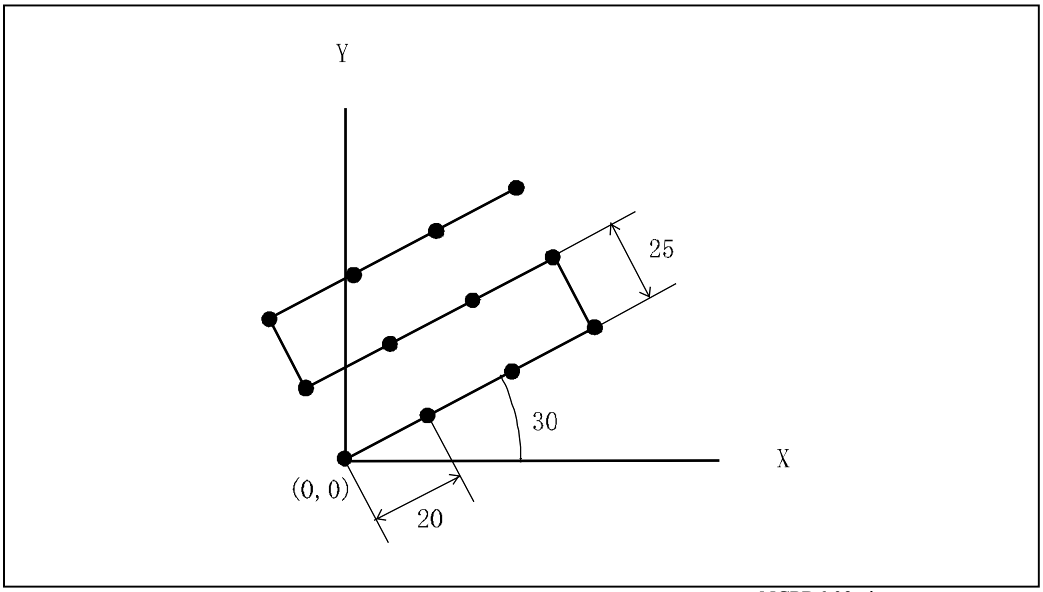
6.3.4 Grid

With the reference point at the commanded coordinate, the coordinate values of the grid composed of points arranged at an even interval and parallel to both the X-axis and the vertical axis are calculated.
By specifying an angle to the X axis, the total form can be inclined.

Command format **G39 X_ Y_ I_ J_ K_ P_ Q_ ;**

X,Y : Coordinate of reference point
I : Interval on the X axis
J : Interval on the Y axis
K : No. of machining point in the X direction (Max. 999)
P : No. of machining point in the Y direction (Max. 999)
Q : Angle intersecting the X axis

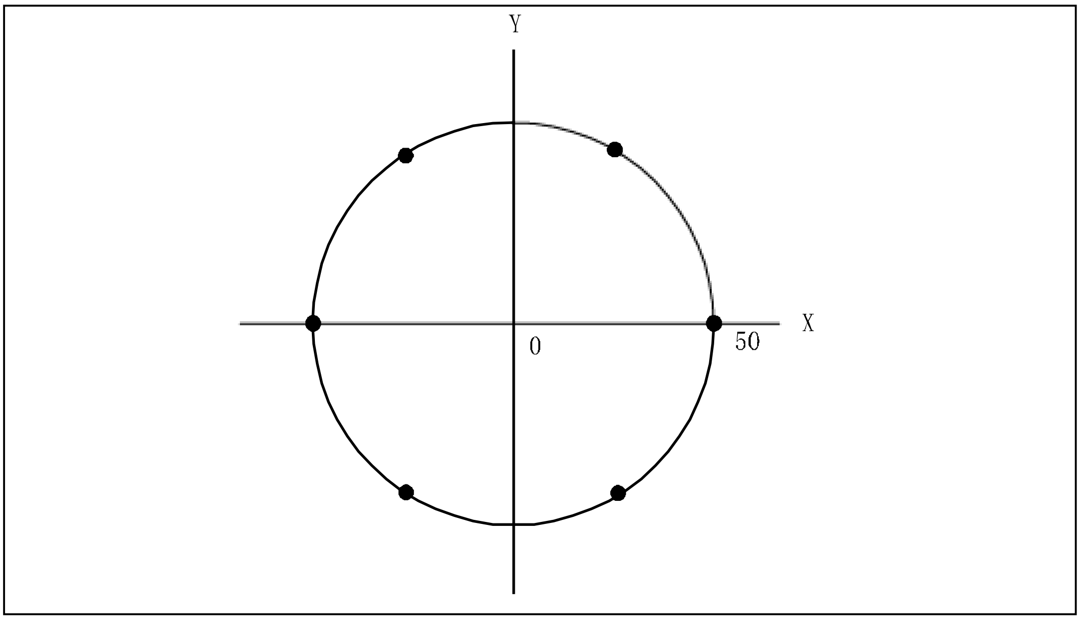
(Ex.) G39 X0 Y0 I20 J25 K4 P3 Q30 ;



eNCPR6.03.ai

(Note 1) The reference point becomes the first machining point.
(Note 2) The coordinate is calulated in the X direction from the reference point.

6.4 Usage of Coordinate Calculation Function



eNCPR6.04.ai

When drilling at 6 points along a circular arc of radius 50 as shown above:
:
N100G81R2.Z-10.F1000K0 ;
N105G36X0.Y0.I50.J0.K6P6 ;
:
The canned cycle data are stored by N100 and the coordinates are calculated by N105, thus executing drilling at each point.

CHAPTER 7

MACRO

7

7.1 What is a Macro?

A "macro" has four main functions: variable function, calculation function, control function (condition branch), and call function (performs the same operation repeatedly). Using these macro functions allows you to create original canned cycles or more flexible programs.
Details of these functions are explained in the following sections:

7.2 Variable function
7.3 Calculation function
7.4 Control function
7.5 Call function

Examples of how these functions are combined are shown below in example 1 and on the following page in example 2.
The program creation procedure will be explained in the subsequent sections.

e.g.1 Tool breakage detection is performed once when machine program has been executed ten times.

```
N01G90G0G54. . . . .;
N02;[WORKING PROGRAM]

N50#100=#100+1  ;         (Count up)
N511IF[#100LT10] GOTO 55     ;
        (Proceeds to N55 when contents of #100 are less than 10)
N53M200;            (Tool breakage detection)
N54#100=0  ;        (Counter reset)
N55M30      ;
```

e.g.2 Program that performs arc cutting by designating center, radius, and angle.

X: CENTER X	Y: CENTER Y	R: RADIUS	Z: CUT POS Z
W: STOP BEFORE WORK		U: CUT START ANGLE	
V: CUT END ANGLE		F: FEEDRATE	

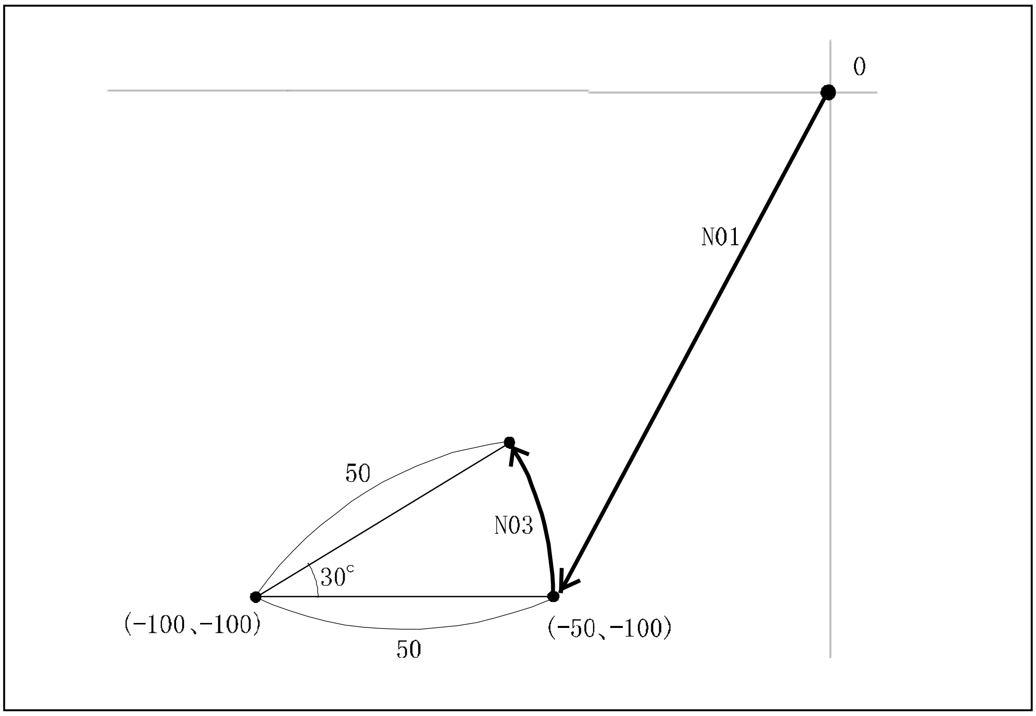
Main program

```
N01G90G54G0Z30.   ;
N02G65P0042X-100.   Y-100.   R50.   Z-3.
W2. UO. V30. F1000;
```

Macro program O0042

```
N01 G90G0X[#24+COS[#21]*#18]
Y[#25+SIN[#21]*#18];
N02 Z#23;
N03 G1Z#26F#9;
N03 G3X[#24+COS[#22]*#18]
Y[#25+SIN[#22]*#18 R#18;
N04 G0Z#23;
N05 M99;
```

7



eNCPR7.01.ai

7

7.2 Variable Function

7.2.1 Outline of variable function

For normal programs, commands are given by directly designating a numerical value (e.g. G90, X200). Using the macro's variable function allows you to use the value stored in the variable for G and X commands.
The value stored in the variable can be changed in PROGRAM mode or in MDI mode.

7.2.2 Expression of variable

The variable number is designated after "#" .
[Example 1] #100
Using brackets [ ] when designating the variable number allows you to use the value stored in the variable or to use a formula.

[Example 2] #100 = #[100+10]
This formula specifies that the value stored in variable #110 is written to #100.

[Example 3] When #1 = 9, #9 = 20, and #20 = 30,
#5 = # [# [#1]] is equal to #5 = 30.
A variable can be used instead of designating a numerical value.

[Example 4] #3=#2+10;
G01X#3Y10;
This formula specifies the X coordinate as the value stored in variable 2 with 10 added to it.
(If #2 is 40, X#3 is 50.)
When a variable is used as address data as shown in example 4, however, the values below the significant digits used as address data are rounded off.
Note that an alarm will occur if the value rounded off to the nearest significant digit exceeds the maximum address command value.
[Example] Assume that the G00X#1; command is used for a machine with a significant digit setting of 1/1000.
When #1 is 12.345678, the command format will be G00X12.346.
Variable cannot be quoted in address N.

[Example 5] N#20 cannot be used.

7.2.3 Undefined variable

When the value of the variable is not defined, it is called a "NULL" variable.
#0 is always a "NULL" variable. This can be read but cannot be write.

e.g.1 When #1 is <Blank>.
G01X#1Y100. → G01Y100.
G01X [1+10.]Y100. → G01X10. Y100.

e.g.2 Calculation
#0 + #0 → 0
#0*5 → 0

e.g.3 Condition formula

When #1 is <Blank>	When #1 is 0 (zero)
#1 EQ #0 → Established #1 NE 0 → Established #1 GE #0 → Established #1 GT 0 → Not established	#1 EQ #0 → Not established #1 NE 0 → Not established #1 GE #0 → Established #1 GT 0 → Not established

For EQ and NE, <Blank> is considered not equal to 0 (zero).

7.2.4 Types of variables

There are two types of variables:

1. Local variable (#1 ~ #26)
2. Common variable (#100 ~ #199, #500 ~ #599)

Local variables are provided for each call level of the macro program. When a macro program is called, the local variables of the called macro level are stored, and a new local variable area is created for the called macro program.
Details on local variables and levels will be explained in 7.5 Call Function.

Common variables can be read from and written to any programs between any levels.

The table below shows the detailed specifications.

Variable No.	Variable type	Function
#0	Constantly NULL	This variable is constantly kept NULL; thus no value can be entered.
#1 to #26	Local variable	These variables can be used for each macro level. When power is turned off, these variables are cleared to be blank. Substitute the variable for these range bellow. -1.0 by 1099 to -1.0 by 10-99 , 0, 1.0 by 10-99 to 1.0 by 1099.
#100 to #199	Common variable	These variables can be used among variables macro programs. When power is turned off, these variables are cleared to be blank. Substitute the variable for these range bellow. -1.0 by 1099 to -1.0 by 10-99 , 0, 1.0 by 10-99 to1.0 by 1099. Caution: Not display all of the significant figures, when the number in a range. Display number is rounded number. Therefore, not always same the number on display and the correct conference of the variable. Refer to the "Macro display and setting".
#500 to #599		These variables can be used among variables macro programs. Keep the data if turn off the power. Substitute the variable for these range and significant figures Meter: -999999.999 to 999999.999 (Integer: six figures, Decimal: Three figures) Inch:-99999.9999 to 99999.9999 (Integer: five figures, Decimal: four figures) When substitute from the number of many decimal figures to these variable, to be upper figures by round. Alarm occurred when substitute to the over range of numbers.

7.2.5 Variable display and setting

Variables are displayed and manually set on the data bank screen.
Press the **[3]** and **[ENT]** keys at the data bank menu screen, or shift the cursor to the menu No.3 and press the **[ENT]** key. The following items are displayed on the screen.

Common variables (#100~#199, #500~#999) and local variables can be referred to or changed.

Number display

Numbers are displayed only when value of common variables for #100 to #199 or local variables is in a range bellow.
The display changes to [***********], if not in a range.

Meter:-999999.999 to 999999.999
(Integer: six figures, Decimal: three figures)
Inch:-99999.9999 to 99999.9999
(Integer: five figures, Decimal: four figures)

Not display all of the significant figures, when the number in a range.
Display number is rounded number.
Therefore, not always same the number on display and the correct conference of the variable.

7.2.6 System variable

1. Interface input/output signal

Signal input	#1000 ~#1015	R
Signal output	#1100 ~ #1115	R/W
Signal batch reading (16 bits)	#1032	R
Signal batch writing (16 bits)	#1132	R/W

[Example of use]
A signal is output from the program to external output port 103.
- Assign #1100 to port N103 using the user parameter (external output signal).

- A signal is output to port 103 when the command is used in the program below.

```
:
#1100 = 1;
:
```

2. Workpiece coordinate origin
The workpiece coordinate origin is read and written.

Workpiece coordinate (External)	#5201~ #5206	R/W
(G54)	#5221~ #5226	R/W
(G55)	#5241 ~#5246	R/W
.	.	
.	.	
(G59)	#5321~#5326	R/W
(G54,1P1)	#7001~#7006	R/W
(G54,1P2)	#7021~#7026	R/W
.	.	
.	.	
(G54.1P48)	#7941~#7946	R/W

3. Tool data
The tool length and tool dia. is read and written.

TL OFFSET	#11001~#11099	R/W
FINE OFFSET	#10001~#10099	R/W
TL ∅ COMP	#13001~#13099	R/W
FINE ∅ COMP	#12001~#12099	R/W

4. Alarm indication

#3000 = n (ALARM MESSAGE)
Alarm number 9000 + n (n: 0 ~200) occurs, and the alarm message in the brackets (the first 20 characters, reset by the [RESET] key) is displayed.
Only alphanumerical characters are used in the brackets and registered in the alarm log.
[Example of use]
When the following block is executed:
#3000 = 6 (ABCD);
"9006 *ABCD" alarm occurs.

(Note) **Command during tool radius compensation, feed to the position that offset vector standing position be vertical to feed directin before.**

5. Message display and stop

#3006 = (MESSAGE)
After execution of the previous block is stopped, a message up to 20 characters is displayed in brackets. When the message contains 21 characters or more, only the first 20 characters are displayed.
The alarm number is fixed at 9300 (stop level 1, reset level 1).

(Note) **Command during tool radius compensation, feed to the position that offset vector standing position be vertical to feed directin before.**

6. Time

Time 1	#3001	R/W	This timer counts the operation time in increments of 10 msec. The timer is reset to zero when the counted time reaches 42949672.96 seconds (approx. 497 days) or when power is turned on. After resetting to zero, it starts counting.
Time 2	#3002	R/W	This timer counts the time when the start LED is lit (STL) in increments of 10 msec. The timer is reset to zero when the counted time reaches 42949672.96 seconds (approx. 497 days). The value is stored even when the power is turned off.

7. Operation control

Operation control	#3003	R/W	MFIN 0: WAIT 2: NOT WAIT
Operation control	#3004	R/W	FEED OVERRIDE EXACT STOP 0: VALID 2: INVALID

#3003
- "0" is set when power is turned on.
- "0" is set when the [RESET] button is pressed or M30 is used.
- When MFIN is set to [NOT WAIT], the program proceeds to the next block without waiting for MFIN. In addition, MFIN is not confirmed to be OFF before the M signal is output.
- When MFIN is set to [NOT WAIT], the next M signal is output after the time set for [EXT SIGNAL OUTPUT TIME (MFIN OFF)] (user parameter - switch 1) has elapsed.

#3004
- "0" is set when power is turned on.
- "0" is set when the [RESET] button is pressed or M30 is used.
- When [SPINDLE OVERRIDE] is set to [INVALID], override is fixed at 100%, regardless of the position of the [OVERRIDE] switch on the operation panel.
- Spindle override and rapid feed override are also fixed at 100%.

8. Mirror image

Mirror images of each axis
Numerical value is converted from binary number to decimal number.

Mirror image	#3007 bit 0: X axis bit 0: Y axis bit 0: Z axis	R	0: INVALID 1:VALID

9. Modal information

The called modal information can be read.

No.	G code
#4001	G00~G03, G102, G103, G202, G203
#4002	G17
#4003	G90, G91
#4004	G22, G23
#4005	G94
#4006	Inch 20, Metric 21
#4007	G40, G41, G42
#4008	G43, G44, G49
#4009	G73~G89, G177~G189
#4010	G98, G99
#4011	G50, G51
#4012	G66, G67
#4014	G54~G59
#4015	G61, G64
#4016	G68, G69, G168
#4022	G50.1, G51.1
#4107	D code
#4109	F code
#4111	H code
#4113	M code
#4114	Sequence number
#4115	Program number
#4119	S code
#4120	T code
#4130	P code (Additional workpiece coordinate system number currently selected.) (This value is 0 when no additional workpiece coordinate system is selected.)

#4113
- M code returns the M number executed last.

#4114
- Sequence number returns the N number executed last, instead of the number of the block currently being executed.

N90 #100 = 0;
N100 #100 = #4114:
When the above is used, 100 is set for #100.

#4115
- Program number returns the sub program number if the sub program is being executed.

7

10. Current position

No.	Contents	Coordinate System	Tool offset	Read while traveling
#5001~ #5008	End point coordinate	Workpiece coordinate system	Not included	Possible
#5021~ #5028	Current position	Machine coordinate system	Included	Not possible
#5041~ #5048	Current position	Workpiece coordinate system	Included	Not possible
#5061~ #5068	Skip coordinate	Workpiece coordinate system	Included	Possible
#5081~ #5088	Tool length offset			Not possible
#5101~ #5108	Servo deviation			Not possible
#5201~ #5208	External workpiece coordinate zero			

"Not possible" is listed in the "Read while traveling "column for "Current position" tool length offset" and "servo deviation". This means that the value is not guaranteed because the value is set when "Interpretation" is performed.
In the program below, the macro command is executed in the block during interpretation while the axis travel block is being executed, so the position during travel is read, instead of the position traveled to in the previous block.

X-10.;
X-10.;
X-10.;
#100=#5021;

11. ATC tool

Read the tool number that set to the ATC tool screen.

#3700	Spindle	R
#3701	Pot 1	R
#3702	Pot 2	R
.	.	R
.	.	R
#3750	Pot 50	R

0 : Cap is specified.
1~99 : Tool number set by NC language mode.
1001~1099 : Tool number set by conversation mode
Blank : Non setting

12. Workpiece counter

Read setting value of the workpiece counter screen and write.

#3801	Workpiece counter 1 count number	R/W
#3802	Workpiece counter 1 present	R/W
#3803	Workpiece counter 1 finish	R/W
#3804	Workpiece counter 1 finish announcement	R/W
#3811	Workpiece counter 2 count number	R/W
#3812	Workpiece counter 2 present	R/W
#3813	Workpiece counter 2 finish	R/W
#3814	Workpiece counter 2 finish announcement	R/W
#3821	Workpiece counter 3 count number	R/W
#3822	Workpiece counter 3 present	R/W
#3823	Workpiece counter 3 finish	R/W
#3824	Workpiece counter 3 finish announcement	R/W
#3831	Workpiece counter 4 count number	R/W
#3832	Workpiece counter 4 present	R/W
#3833	Workpiece counter 4 finish	R/W
#3834	Workpiece counter 4 finish announcement	R/W

Blank : Non setting

7.3 Calculation Function

7.3.1 Calculation type

Calculations such as those below are possible for variables and numerical values.

[Supplementary explanation]

- Numerical values are entered for i, j, and k of #i, #j, and #k (e.g. #10), indicating they are macro variables.
- Instead of #j and #k, a constant can also be used for the right side of the equation.

Variable definition and replacement	#i = #j	Definition and replacement
Addition	#i = #j + #K #i = #j -#k #i = #j OR #k #i = #j XOR #k	Addition Subtraction Logical OR Exclusive OR
Multiplication	#i = #j * #k #i = #j /#k #i = #j AND #k	Multiplication Division AND
Function	#i = SIN [#k] #i = COS [#k] #i = TAN [#k] #i = ATAN [#k] #i = SQRT [#k] #i = ABS [#k] #i = BIN [#k] #i = BCD [#k] #i = ROUND [#k] #i = FIX [#k] #i = FUP [#k]	Sine Cosine Tangent Reverse tangent Square root Absolute value BCD to BIN conversion BIN to BCD conversion Rounding off Rounding down to nearest whole number Rounding off to nearest whole number

7.3.2 Calculation order

The order that an expression is evaluated is as shown below.

1. Function
2. Multiplication
3. Addition

To replace the above order, use brackets [ ].
Brackets [ ] can be used up to five times, including the brackets for the function.

7.3.3 Precautions for calculation

(Note 1) Formula

The right side of the equation can be connected using a constant, variable, function, or operator. When using a constant, any value without a decimal point is regarded as having a decimal point at its end.
[Example] #1 =12;, #1 is regarded as 12.000.

(Note 2) Angle calculation

The unit for SIN, COS, TAN, and ATAN functions is “degree”.
For example, 90 degrees and 30 minutes is designated as 90.5 degrees.

(Note 3) Logical calculation

Logical sum (OR), logical product (AND), and exclusive-or (XOR) perform the following calculation for each bit of the integral number. Decimal places are regarded as “as”.

(Note 4)

Calculation object	Result for AND	Result of OR	Result of XOR
0 and 0	0	0	0
0 and 1	0	1	1
1 and 0	0	1	1
1 and 1	1	1	0

Conversion between BCD and BIN
BIN indicates binary number.
BCD indicates binary-coded decimal number.
The value of each digit in a decimal number is expressed as a 4-bit binary number.
[Example] 12 = 0001 (4 bits) 0010 (4 bits)
The binary number 00010010 is the decimal number 18.
When 12 is converted from BIN to BCD, the result will be 18.
Conversion from BCD to BIN is performed in the reverse of the above.
Decimal places in the conversion source number are always regarded as zero (0).

(Note 5) Range of constant

The range of constants that can be used in the formula is as below.
-999999999 ~ -0.000000010
+0.00000001 ~ +999999999
The maximum number of digits that can be designated is 9 for a decimal number.

(Note 6) Calculation accuracy

When calculation is performed for the macro statement, small calculation errors occur and these accumulate as the calculation is repeated. However, the data is retained internally using up to approximately 15 significant digits in floating-point format, thus ensuring calculation accuracy.

7.4 Control Function

The control function allows you to change the flow of the program in the middle of the program by designating certain conditions.
The control function has the following three types:

1. GOTO statement (Unconditional branch)
2. IF statement (Conditional branch)
3. WHILE statement (Repetition)

Possible controls using these statements will be described below.

7.4.1 GOTO Statement (unconditional branch)

The program is branched unconditionally to sequence number “n” (n: 1 ~ 9999).

Command format	**GOTO n;**	**(n: sequence number)**

An alarm will occur when the sequence number “n” is not within the range 1 to 9999 or there is no corresponding sequence number.
The sequence number can also be designated by a formula.

[Example]

```
N1 GOTO 3;
N2 GOTO #10;
N3 ;
```

N2 (sequence number 2) is skipped unconditionally.
If N2 is executed, the program skips to the sequence number of the value stored in #10.

The sequence number can be skipped by GOTO only within the same program. When GOTO is designated, a search is started toward the end of the program, and the first sequence number found is enabled.
When the search reaches the end of the program, it starts again from the top of the program.

7.4.2 IF statement (conditional branch)

IF is followed by a condition formula.

Command format	**IF [condition formula] GOTO n; (n: 1 ~ 9999)**

When the condition formula is satisfied, the program is branched to the sequence number “n”.
When not satisfied, the next block is executed.
The following condition formulas are available:

Types of condition formula

# iEQ # j	# i is EQ # j
# iNE # j	# 1 is not equal to # j
# i GT # j	# i is greater than # j
# i LT # j	# i is less than # j
# i GE # j	# i is # j or more
# i LE # j	# i is # j or less

(Note 1) Use square brackets for condition formula.
(Note 2) The range of numerical values that can be used in the conditional expression is -2147483647 to 2147483647.
If a value not within this range is used, an alarm will occur.

[Example]

```
               :
               :
(1)  IF[#100 EQ 50] GOTO 123;
               :
               :
(2)  IF[#101 GT 102] GOTO 123;

               :
               :
(3)  GOTO 124;
               :
               :
(4)   N123;
               :
               :
(5)   N124;
               :
               :
```

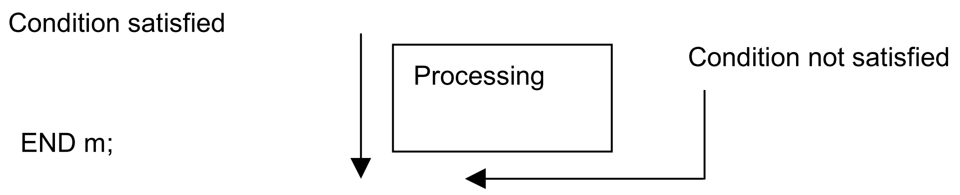
At (1) above,
If variable #100 is 50, the program skips to (4), where the sequence number is 123.
If it is not 50, the program proceeds to the next block (2).
At (2) above
If #101 is larger than 102 (#101 > 102), the program skips to (4), where the sequence number is 123.
If #101 is less than 102 (#101 」 102), the program proceeds to the next block (3).
At (3) above, the program unconditionally skips to (5) via the GOTO statement.

7.4.3 WHILE statement (repetition)

Command format **WHILE [condition formula] DOm ~ ENDm; (m = 1 ~ 4)**

WHILE is followed by a condition formula.
When the condition formula is satisfied, the program between DO and END is executed.
When not satisfied, the program proceeds to the block after END.
When WHILE (condition formula) is omitted, the program between DOm and END is repeated endlessly.
Use square brackets for the condition formula.
[Example 1] WHILE statement]
WHILE [condition formula] DO m; (m = 1 ~ 4)

(Note1) **The range of numerical values that can be used in the conditional expression is -2147483647 to 2147483647.**
If a value not within this range is used, an alarm will occur.



7.4.4 Precautions for control function

(Note1) **DOm to ENDm must correspond one to one in the WHILE statement.**
If not, an alarm will occur.

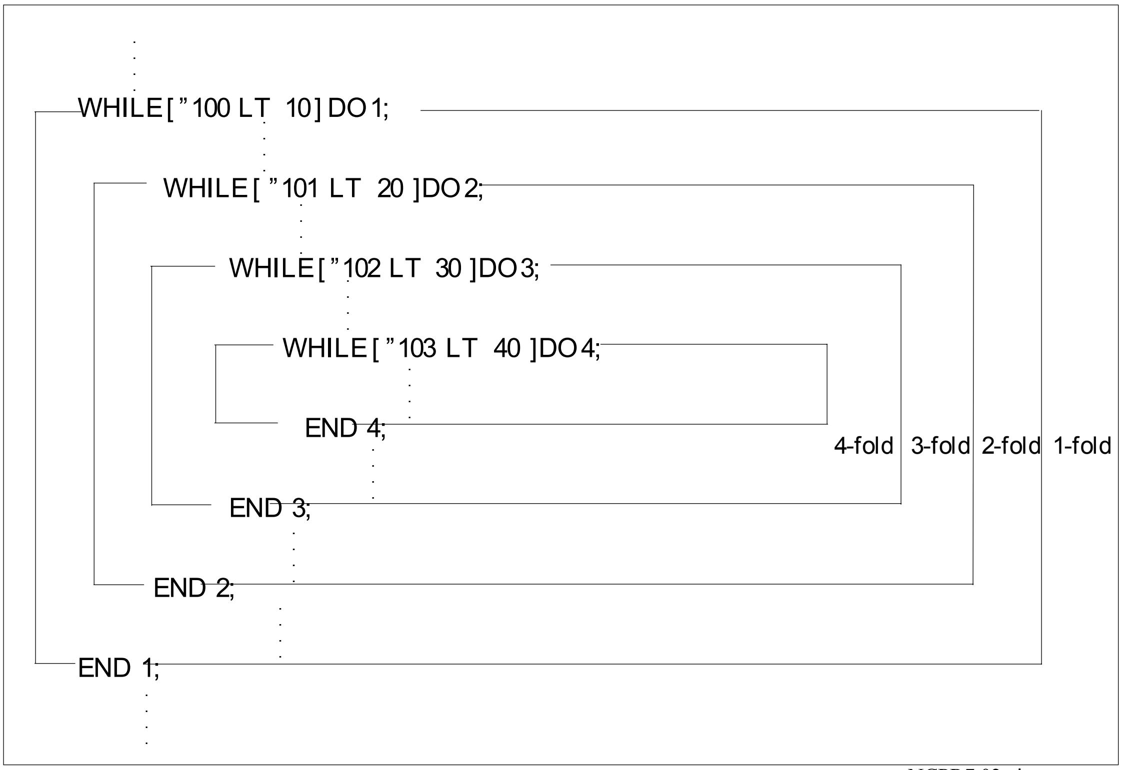
```
      :
WHILE [#100 LT 10]DO 1;
      :
      :
WHILE [#101 EQ 50]DO 1;
      :
      :
END 1;
      :
```

Identifier "m" can be used multiple times as long as the above condition is met.

```
      :
WHILE [#100 LT 10]DO 1;
      :
      :
END 1;
      :
      :
WHILE [#101 EQ 50]DO 1;
      :
      :
END 1;
      :
```

(Note 2) DOm ~ ENDm should not be overlapped in the WHILE statement.

```
      :
WHILE [#100 LT 10]DO 1;
      :
      :
   WHILE [#101 EQ 50]DO 2;
      :
      :
   END 1;
      :
      :
   END 2;
      :
           :
```



eNCPR7.02.ai

(Note 4) IF statement and WHILE statement
IF ~ GOTO cannot be branched to a section between WHILE and END.

```
IF[#100  LT  10]GOTO  123;
:
:
WHILE [#101 EQ 0] DO 1;
:
:
N123;
:
:
END 1;
```

(Note 5) **IF statement and WHILE statement**
IF ~ GOTO within WHILE ~ END cannot be branched to a section outside WHILE ~ END.

```
WHILE [#101 EQ 0] DO 1;
:
:
 IF[#101 EQ 10]GOTO 123;
:
:
END 1;
:
:
N123
```

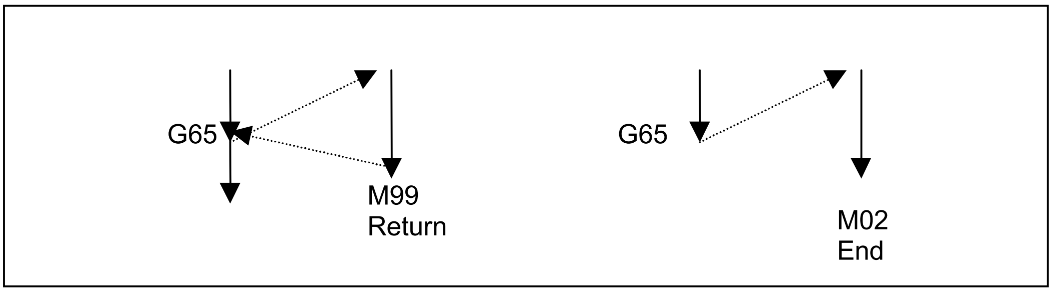
7.5 Call Function

Using G65 and G66 (described in a later section), another program can be called and executed. This function is called a “macro call function”. The called program is called a “macro program”.

A unique canned cycle can be easily created using this function when performing the same operation repeatedly.

A macro program call (G65, G66) can be executed in MEM mode but cannot be executed in MDI mode.
The called macro program is returned to the call source by executing M99. When M02 or M30 (end of program) is executed, the macro program ends without returning to the calling source. (Memory operation ends.)

<Difference between M99 and M02>



0604.doc

In addition, one macro program can call another macro program using G65 and G66. This parent/child relationship is possible for up to four generations. (This state is described by the phrase “the depth of nesting for macro program is up to 4-fold.”This is called a “multiple nesting call”.)

A macro program call function enables the designated value to be transferred from the call source program to the call destination program using arguments. (See 7.5.3 for details on arguments.)

M98 (sub program call) (see chapter 10 for details) is a function similar to a macro program call (G65, G66). The difference between these functions is explained in section 7.5.3.

Each type of macro call function is explained in the following sections.

7.5.1 Simple call function

G65 is generally used to call a macro program.

Command format

G65 P_L_(Argument);

P : Macro program number to be called
L : Number of calls to be repeated (up to 9999)
If “L” is omitted, “1”is automatically selected.

(Argument): Data transferred to macro. Can be omitted.
See 7.5.3 for argument.

[Example 1]
G65 P200;
This format specifies that a program between Nos. 200 and 299 is called once.

[Example 2]
G65 P200L2;
and
G65 P200;
G65 P200;
This format specifies that a program between Nos. 200 and 299 is called twice.

7.5.2 Modal call function

When a macro program is automatically called each time an axis movement command is given once registered, it is called a "modal call function".

Use G66 to register a modal call and G67 to cancel registration. When a modal call is registered, the macro program is executed after each axis movement.

Command format

G66 P_L_(Argument);

P : Macro program number to be called
L : Number of calls to be repeated (up to 9999)
If "L" is omitted, "1" is automatically selected.
(See descriptions for G65).
(Argument) :Data transferred to macro. Can be omitted.
See 7.5.3 for argument.

To cancel a modal call, use the following command.

Command format

G67;

[Example 1]

G66 P10:	(1) Register call for program number 10.
G01X10.0Y10.0;	(2) Call program number 10 after execution of.
G01X1.0Y1.0;	(3) Call program number 10 after execution of .
G67;	(4) Cancel registration.
G01X10.0Y10.0;	(5) Does not call any program.
G01X1.0Y1.0;	(6) Does not call any program.

Program number 10 is called once after (2) is executed. Program number 10 is also called once after (3) is executed.
Programs are not called after (5) and (6) are executed.

**(Note 1) G67 must be designated by a program besides a call program.
G66 mode can also be canceled using M30.**

(Note 2) G66 cannot be designated in G66 mode.

**(Note 3) G66 command is only used to register macro program call.
Macro programs cannot be called with this command.**

7.5.3 Macro call argument

Argument(s) must be declared when it is necessary to pass local variables to the macro.

Format 1

Augments can be declared for all addresses, excluding G, L, N, O, and P.

Addresses with argument specified	Macro variables
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
G	-
H	#11
I	#4
J	#5
K	#6
L	-
M	#13
N	-
O	-
P	-
Q	#17
R	#18
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

Format 2

A, B, and C, and repeating I, J, K can be designated.

Addresses with argument specified	Nth repeat	Macro variables
A	1	#1
B	1	#2
C	1	#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

(Note 1) **Addresses that do not require setting can be omitted.**
(Note 2) **Local variables corresponding to non-designated addresses are null.**
(Note 3) **Conversion between format 1 and 2 is performed according to the addresses to be used. However, there are cases where both formats are mixed. In this case, the value designated later is valid.**
(Note 4) **Command argument after G65 and G66.**

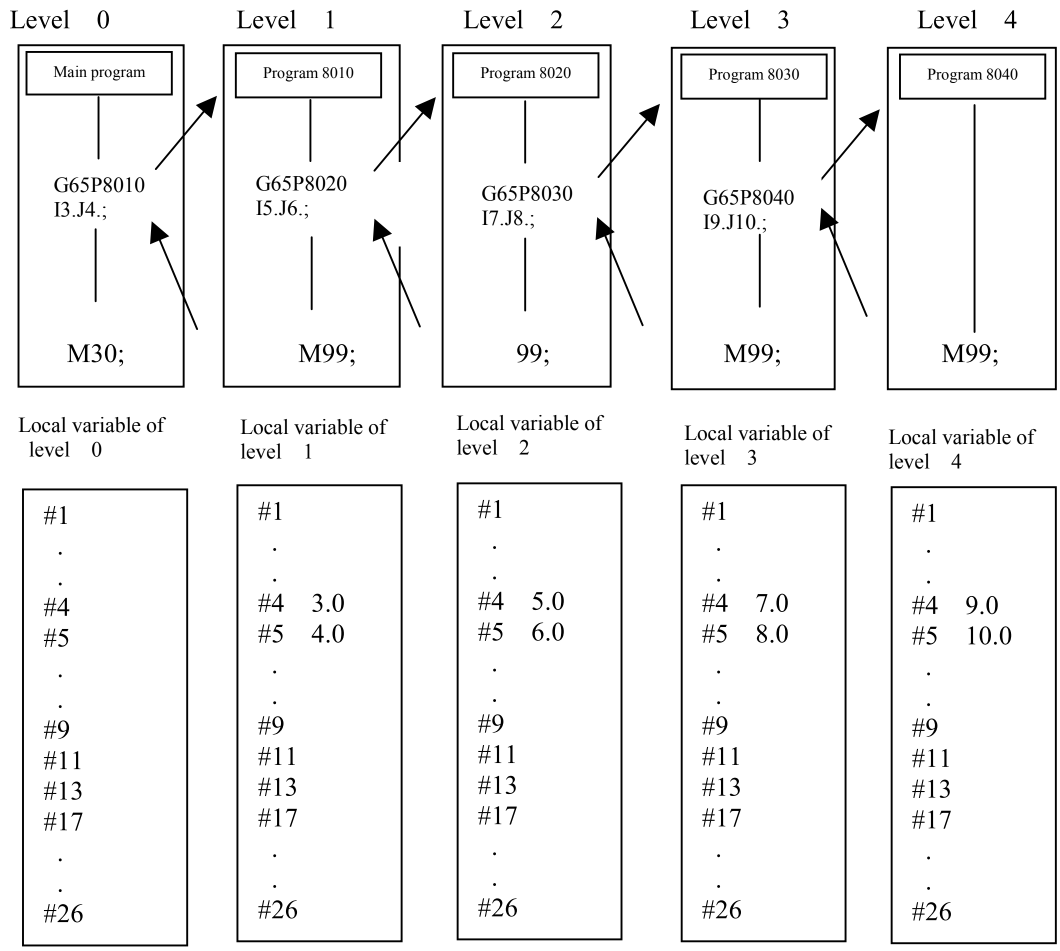
(Example)
G65 D1.0 E2.0 I3.0 J4.0 K5.0 I6.0 J7.0 K8.0 F9.0
(1) #7 ← 1.0
(2) #8 ← 2.0
(3) #4 ← 3.0
(4) #5 ← 4.0
(5) #6 ← 5.0
(6) #7 ← 6.0 (Value designated by "D" is invalid).
(7) #8 ← 7.0 (Value designated by "E" is invalid).
(8) #9 ← 8.0
(9) #9 ← 9.0 (Value designated second time with "K" is invalid).

7.5.4 Difference between G65 and M98

1. Arguments can be designated for G65, but cannot be designated for M98.
2. Local variables are available for G65 depending on the depth of nesting, but are not available for M98.
3. Calling depth of nesting for G65 is up to 4-fold.
   When combined with that for M98, up to 8-fold are possible.

7.5.5 Multiple nesting call

Macro calling depth of nesting is up to 4 -fold.
Local variables (#1 ~ #26)are provided for each macro level. When macro is called by G65, local variable of called macro level is stored once, and new local variable of called macro program is prepared. When M99 is executed, stored local variable becomes valid.
Common variables can be read and written even between different macro levels.



Common variables : Commonly read and written from any macro level.

#100 ~ #199
#500 ~ #599

7

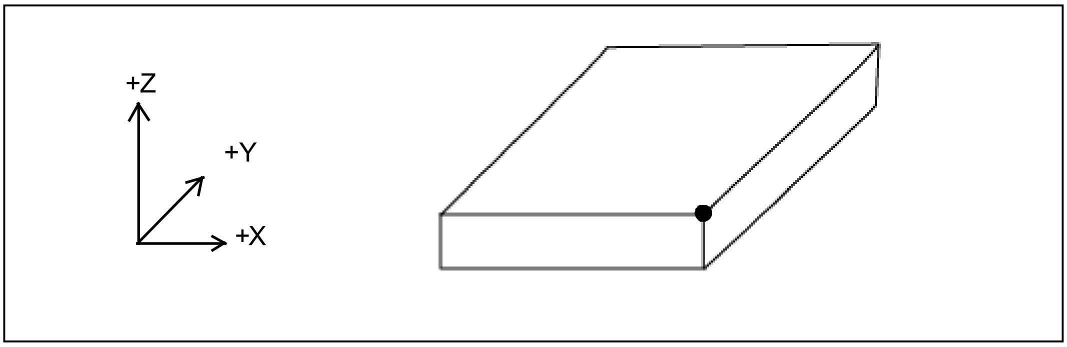
CHAPTER 8

AUTOMATIC WORK MEASUREMENT

8 Automatic Work Measurement

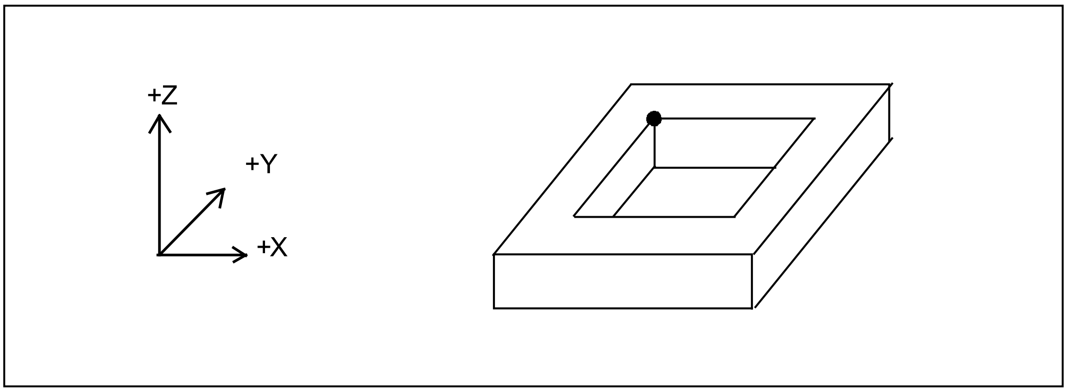
Automatic work measuring functions

1. G121 -- X and Y coordinates of a corner



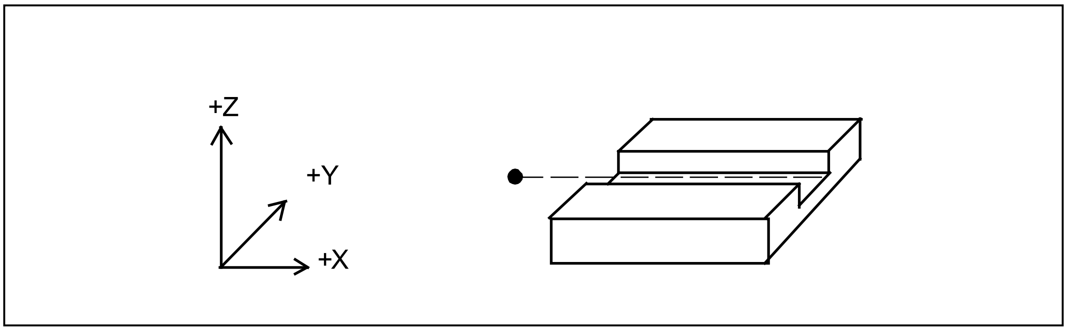
eNCPR9.01.ai

2. G129 -- X and Y coordinates of a groove



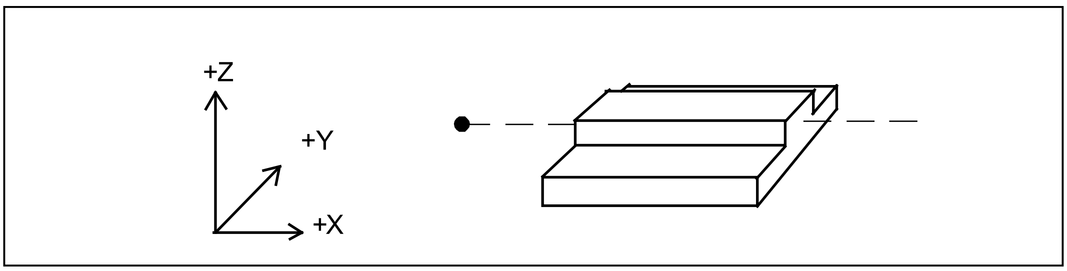
eNCPR9.02.ai

3. G122 -- X and Y coordinates of the axis of parallel groove



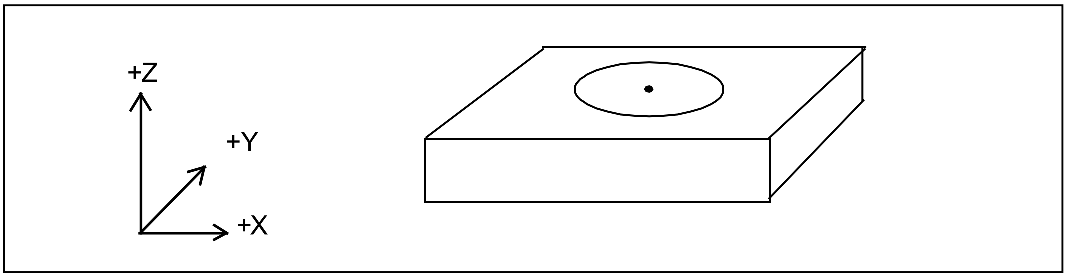
eNCPR9.03.ai

4. G123 -- X and Y coordinates of the axis of parallel bos



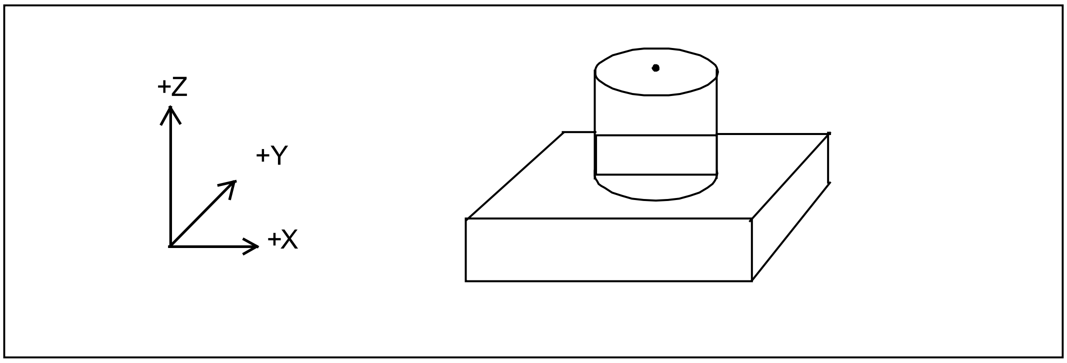
eNCPR9.04.ai

5. G124,G126 -- X and Y coordinates of the center of a hole

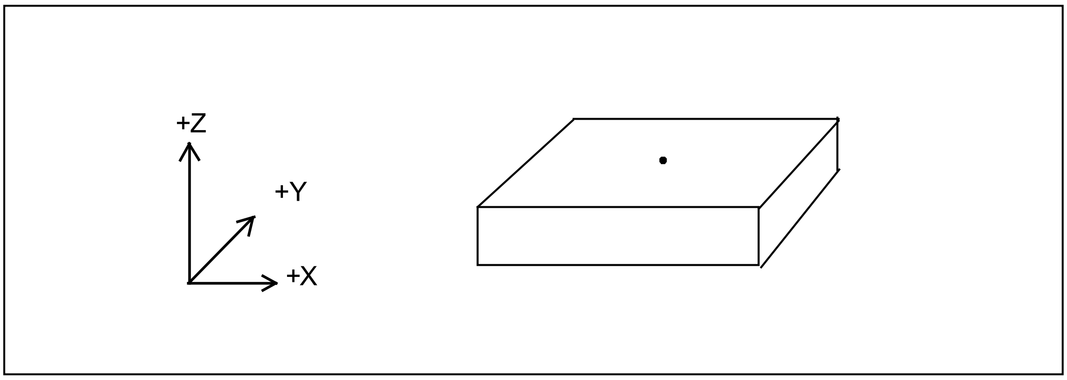


eNCPR9.05.ai

6. G125,G127 -- X and Y coordinates of the center of a boss



7. G128 -- Z coordinate of the top surface of a workpiece

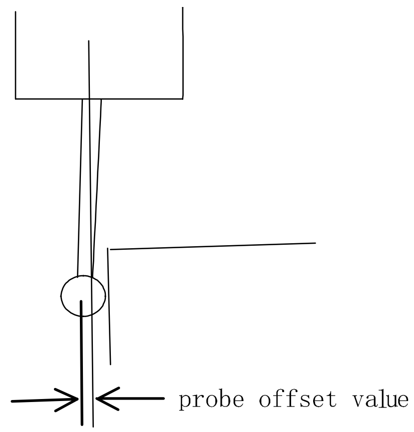


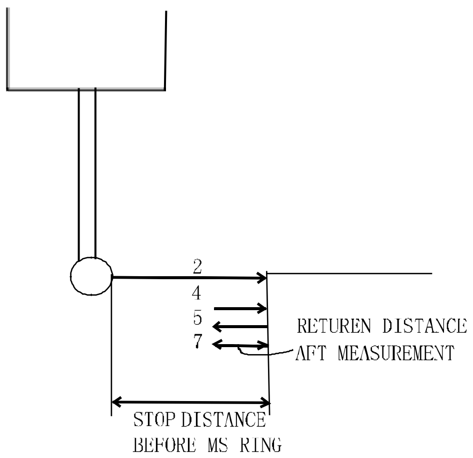
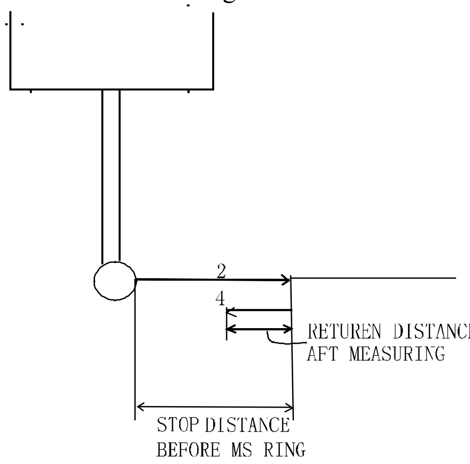
8.1 Before Automatic Work Measurement

Set the necessary parameters of User Parameter 7 (ZERO MEASUREMENT).
Unless the parameters are set correctly, the probe may be damaged.

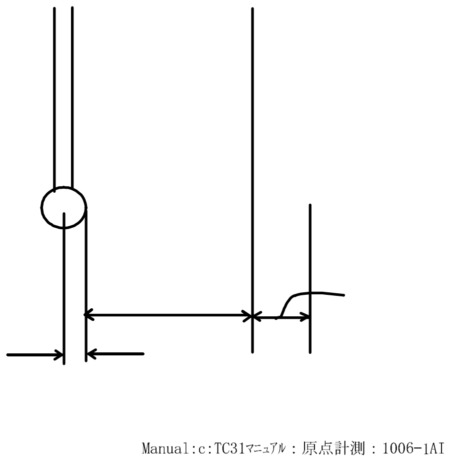
8.2 Setting of Data on Automatic Work Measurement

User Parameter (AUTOMATIC WORK MEASUREMENT)

Item	Description
PROBE OFFSET VALUE 1 PROBE OFFSET VALUE 2	Sets the difference between the stylus tip ball center and the spindle center when the detection signal is turned on while the touch probe is attached to the spindle. X component and Y component of the difference are prob offset 1 and 2 respectively. To set the difference, carry out 6.PROBE OFFSET(G121/G129) of automatic centering.  Manual1:c:TC31ﾏﾆｭｱﾙ：原点計測 :09L04 Setting range: -9.999 ~ 9.999 mm -0.9999 ~ 0.9999 inch
PROBE OFFSET VALUE 3 PROBE OFFSET VALUE 4	Sets the difference of the center of the circle obtained by the three-point measurement(G124,G125) and the actual circle center. X component and Y component of the difference are probe offset 3 and 4 respectively. T set the difference, carry out 7.PROBE OFFSET(G124/G125) of automatic centering. Setting range: -9.999 ~ 9.999 mm -0.9999 ~ 0.9999 inch

Item	Description
MEASURING MOTION (0:TYPE1 1:TYPE2)	(0:TYPE1) 1) It is checked that the detection signal is off. 2) The probe moves in the specified axis direction at the speed preset to MEASURING SPEED 1. 3) When the detection signal has turned on, the axis stops traveling. 4) The probe moves forward for the RETURN DISTANCE AFT MEASURNG. 5) The probe moves in the specified axis direction at the speed preset to MEASURING SPEED 2. 6) When the detection signal has turned on, the axis stopstraveling. 7) The probe returns to the position of 4). 8) It is checked that the detection signal is off.  eNCPR9.09.ai (1:TYPE2) 1) It is checked that the detection signal is off. 2) The probe moves in the specified axis direction at the speed preset to MEASURERING SPEED 2. 3) When the detection signal has turned on, the axis raveling. 4) The probe moves forward for the RETURN DISTANCE AFT MEASURNG. 5) It is checked that the detection signal is off.  eNCPR9.10.ai

8

Item	Description
MEASURING SPEED 1	Sets the first measuring speed for MEASURING MOTION(TYPE1). *Relief amount of probe = L (mm) *SKIP FEED TIME CONSTANT 1 = t (msec) *MEASURING SPEED 1 = F1 (mm/min) *Delay in control system = td (msec) = 12 (msec) $L \geq \underbrace{((F1\times td)\div(60\times1000))}_{\text{Over travel amount due to delay in control system}} + \underbrace{((F1\times t/2)\div(60\times1000))}_{\text{Over travel amount}}$ $\boxed{F1 \leq (120000L\div(24+t))\div\underbrace{1.2}_{\text{Safety rate}}}$ Setting range:1~5000 mm/min 0.1~196.8 inch/min
MEASURING SPEED 2	Sets the second measuring speed for MEASURING MOTION(TYPE1) and the measuring speed for MEASURING MOTION(TYPE2). *Allowable error in control system = E (mm) *MEASURING SPEED 2 = F2 (mm/min) *Delay in control system = td (msec) = 0.5 (msec) $E \geq (F2\times td)\div60 \longrightarrow \boxed{F2 \leq 120\times E}$ Setting range:1~5000 mm/min 0.1~196.8 inch/min
STOP DISTANCE BEFORE	Sets the distance between the probe end face at the MSRNG measurement start point and the estimated workpiece surface. When measuring has been skipped for the values preset to STOP DISTANCE BEFORE MSRING and MEASURING TRAVEL LMT DISTANCE , SENSOR SIGNAL OFF alarm will occur.  Setting range: 0.000~99.999 mm 0.0000~9.9999 inch

Item	Description
MEASURING TRAVEL LMT DISTANCE	Sets the amount of overtravel when the measuring skip has exceeded the estimated value(program command value). Setting range: 0.000~99.999 mm 0.0000~9.9999 inch
MEASUREMENT TOLERANCE 1	When the difference between the measured value and the estimated value (program command value) has exceeded the preset value, MEASD VAL ERR LRG(1) will occur. When 0 is set,the value error check is not carried out. Setting range:0.000~99.999 mm 0.0000~9.9999 inch
MEASUREMENT TOLERANCE 2	If the difference between the current measurement result and the previous measurement result exceeds the setting range, MEASD VAL ERR LRG(2) occurs. However, the difference is not checked in the following cases: *Zero (0) is set. *1st automatic work measurement is performed after the power is turned on (1st measurement for each of measurement results 1 to 4). *No previous measurement result exists. *Current G code differs from the previous G code. Setting range: 0.000~99.999 mm 0.0000~9.9999 inch
MEASURNG RETURN DISTANCE AFT	Sets the amount that the measuring probe retracts where it has contacted the non measuring object during automatic measurement. *RETURN DISTANCE AFT MEASURNG = Lb (mm) *SKIP FEED TIME CONSTANT 1 = t (msec) *MEASURING SPEED 2 = F2 (mm/min) $\boxed{Lb \geq MAX(1.0,\ F2 \times t/6000)}$ Setting range: 0.000~99.999 mm 0.0000~9.9999 inch

8

(Note)

- **The setting values of MEASURING SPEED, RETURN DISTANCE AFT MEASURNG and so on differ according to the probe mounted. Consult the probe maker and set the values.**

- **During automatic work measurement, the speed that the probe moves to the measurement start point or returns from the measured point conforms to the modal of G00 and G01.**
  G00... Rapid traverse rate
  G01... F command value
  G02/G03/G102/G103/G202/G203... IN ARC MODE alarm will occur

- **Perform automatic work measurement with the tool offset function ON or the probe may be damaged.**
- **Check that the chips are not stuck to the measuring probe tip and the surface of the workpiece to be measured. We shall not bear responsibility for the decreased measurement accuracy due to the above cause.**

- **The mode cannot be changed during automatic work measurement.**
  If mode selection is attempted, DURING AUTOMATIC WORK MEASURMENT alarm will occur.

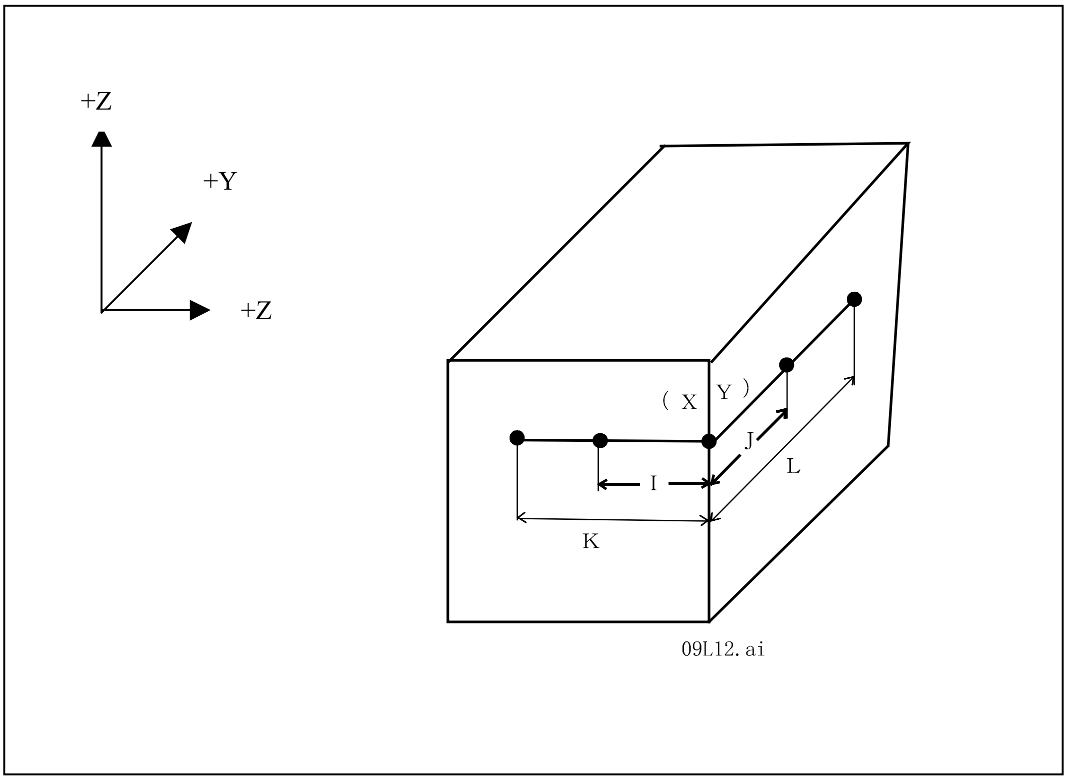
- **To prevent the probe from being damaged, press the [SINGL] key for the first automatic work measurement and check the condition every single operation.**
- **Tool length offset set first for automatic work measurement.**

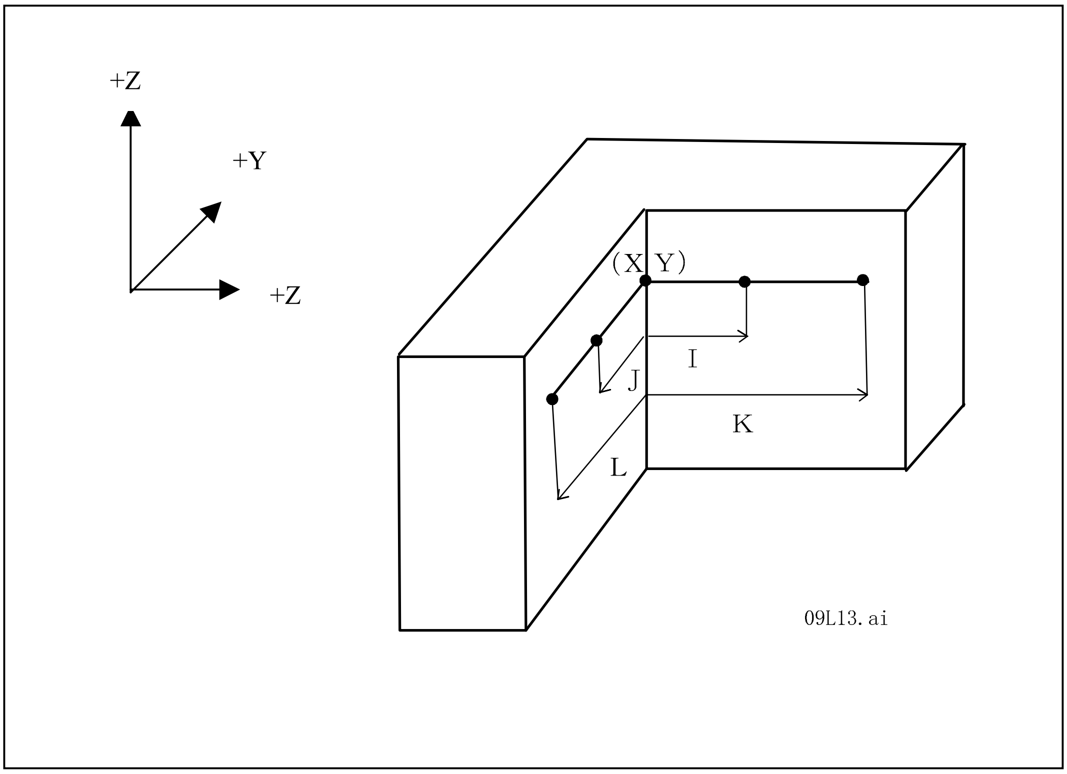
8.3 Operation of Automatic Work Measurement

8.3.1 Corner

Command format

Boss	**G121 X_ Y_ I_ J_ K_L_D_Z_ R_Q_**
Groove	**G129 X_Y_ I_ J_ K_ L_D_Z_R_Q_ ;**





X,Y ...Estimated corner value
I ,K ...X-axis position when measuring in the Y direction, offset value from (X, Y)
J ,L ...Y-axis position when measuring in the Y direction, offset value from (X, Y)
D ...Tool offset number
Z ...Z coordinate during measurement
R ...Z coordinate of return point when the Z axis has traveled from one measurement point to the other measurement point or when the movement has com pleted.
Q ...Register No. that stores the measured results ("1" when omitted)

(Note 1)
When either “I”・or “J” is not commanded , when “I” is zero (o) , or when “J” is zero (0), an alarm occurs.

(Note 2)
When "K" or "L" is commanded, the tilt angle (the angle used for coordinate rotation) is calculated should the work piece become tilted. An [ZERO MESR ADRS ERR] error will occur when both "K" and "L" are commanded or zero (0) is commanded.

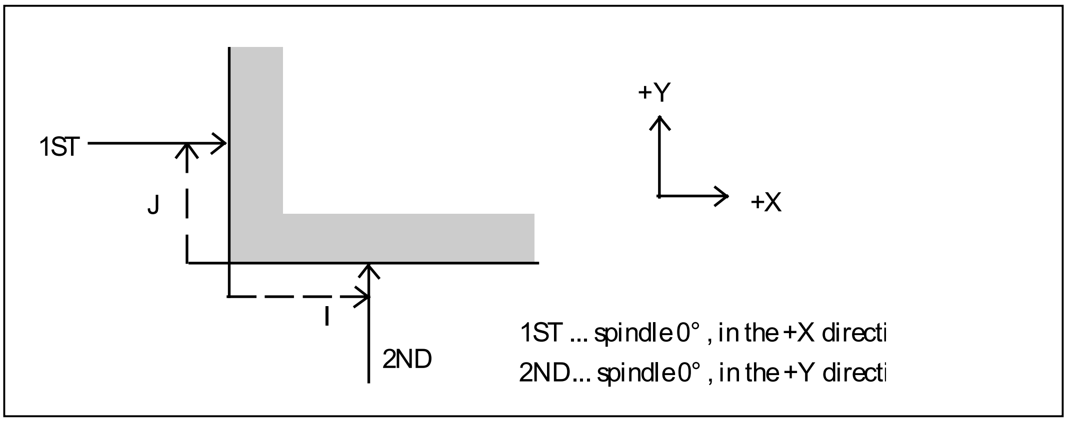
(Note 3)
Before executing G121/ G129, execute PROBE OFFSET VALUE (G121/G129) of CENTER ALIGNMENT and set the offset value to PROBE OFFSET VALUE 1 and PROBE OFFSET VALUE 2 of parameter 7.

Measurment pattern

1. The spindle is oriented. The probe moves to the first measurement start point of the X and Y axes.
2. The probe moves to the Z axis measurement height.
3. The first measurement is carried out (Position "J").
4. Measured by position "L" when "L" is commanded
5. The probe moves to the Z axis return point.
6. The spindle is oriented. The probe moves to the second measurement start point of the X and Y axes.
7. The probe moves to the Z axis measurement height.
8. The second measurement is carried out.
9. Measured by position "K" when "K" is commanded
10. The probe moves to the Z axis return point.

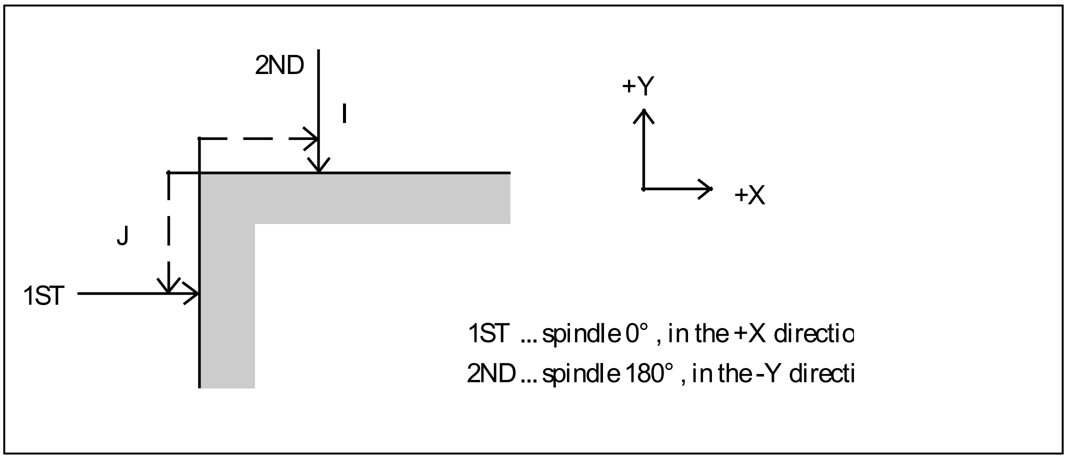
I and J symbols, spindle orientation, and direction of the measuring skip movement.

a) I>0, J>0



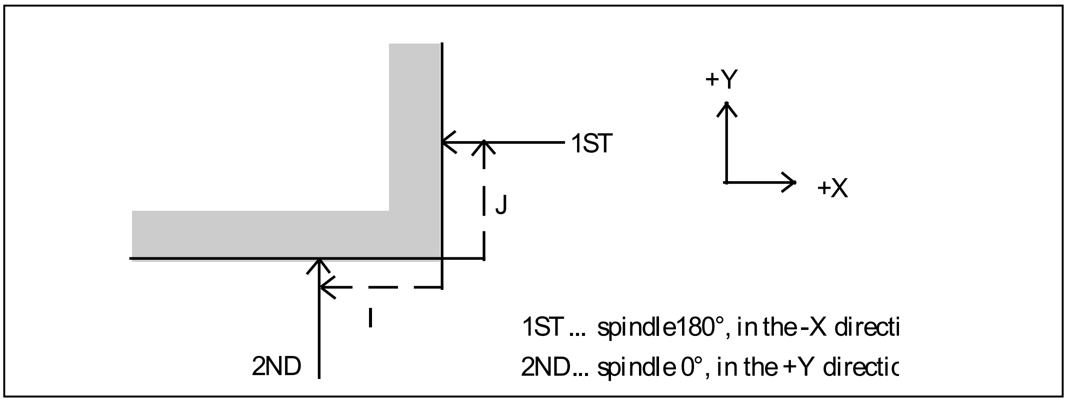
eNCPR9.11.ai

b) I>0, J<0



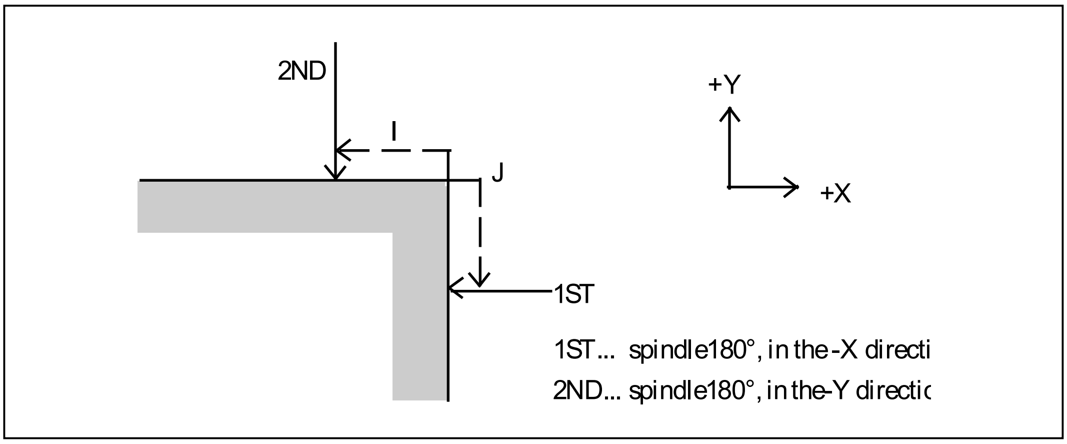
eNCPR9.12.ai

c) I>0, J<0



eNCPR9.13.ai

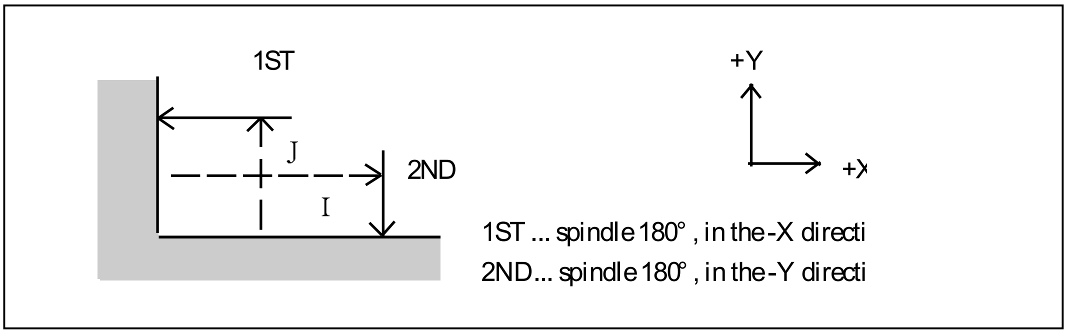
d) I<0, J<0



eNCPR9.14.ai

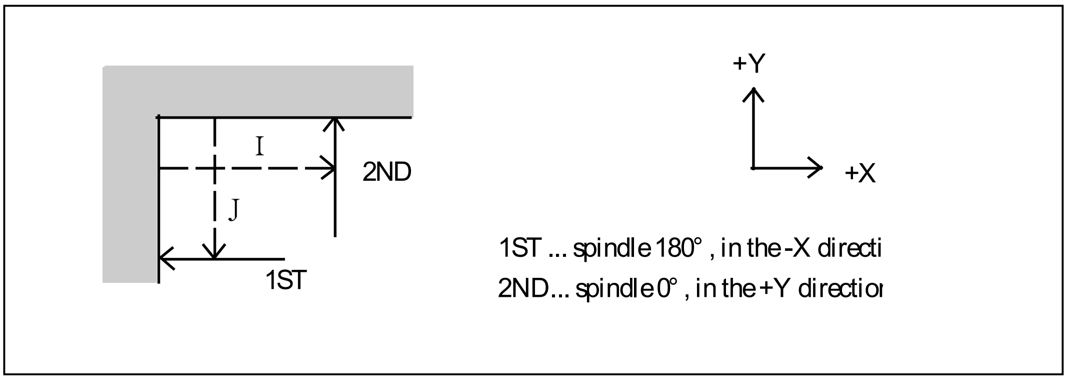
Measurement pattern
1. The spindle is oriented. The probe moves to the first measurement start point of the X and Y axes.
2. The probe moves to the Z axis measurement height.
3. The first measurement is carried out (Position "J").
4. Measured by position "L" when "L" is commanded
5. The spindle is oriented. The probe moves to the second measurement start point of the X and Y axes.
6. The second measurement is carried out.
7. Measured by position "K" when "K" is commanded
8. The probe moves to the Z axis return point.

a) I<0, J<0



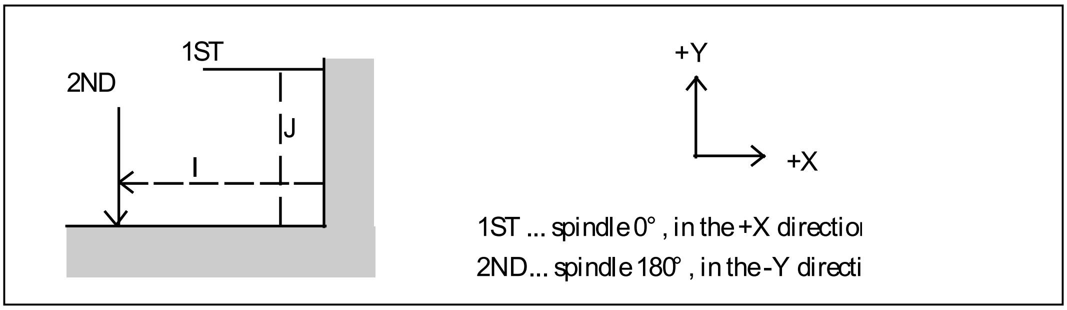
eNCPR9.15.ai

b) I<0, J>0



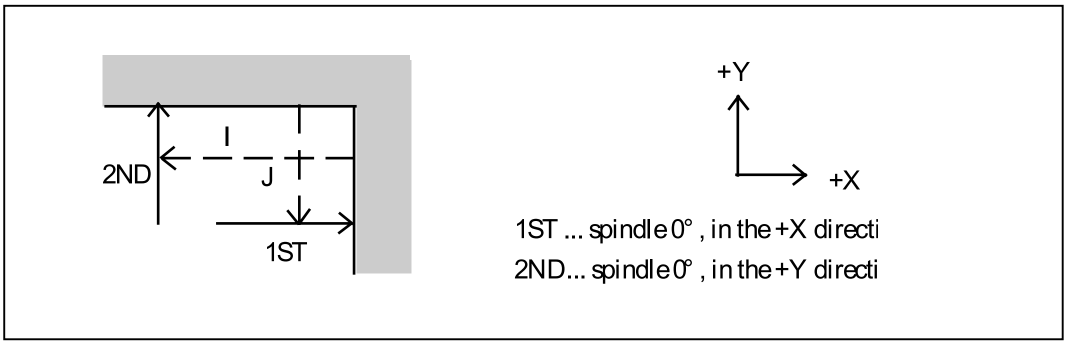
eNCPR9.16.ai

c) I<0, J<0



eNCPR9.17.ai

d) I>0 , J>0

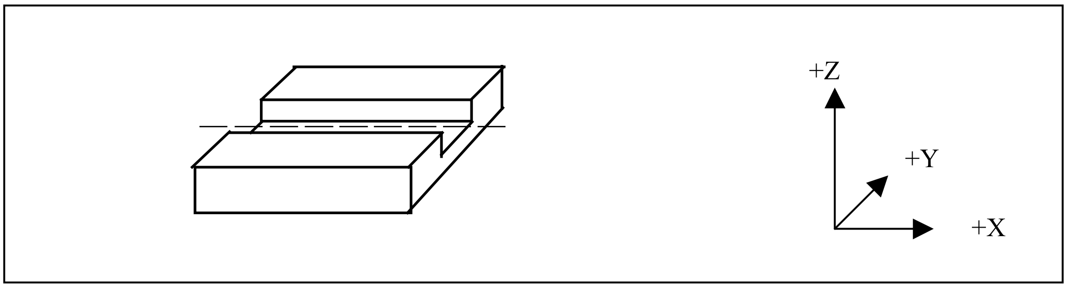


eNCPR9.18.ai

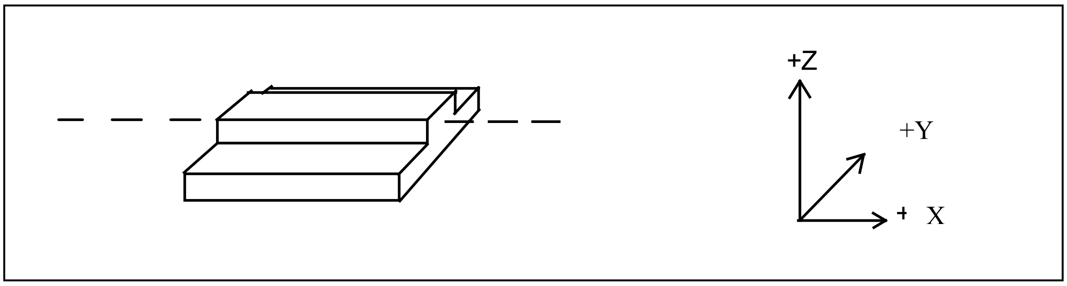
8.3.2 Parallel

Command format

groove	**G122 X_ Y_ I_(J_) D_ Z_ R_ Q_**
boss	**G123 X_ Y_ I_(J_) D_ Z_ R_ Q_**



eNCPR9.19.ai



eNCPR9.20.ai

X,Y ... Estimated groove (boss) center value
I,J ... Width of groove
I : Measures in the X direction.
J : Measures in the Y direction.

Note: I and J cannot be commanded at the same time.

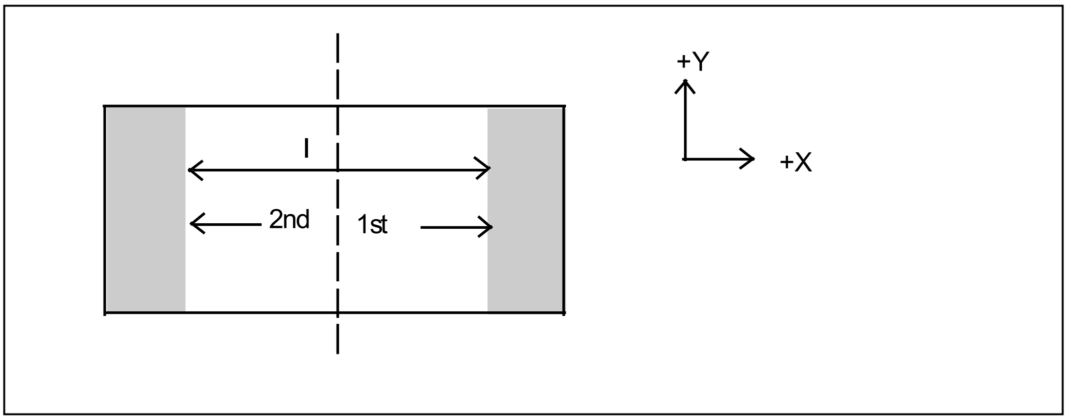
D ... Tool offset number
Z ... Z coordinate during measurement
R ... Z coordinate of return point when the Z axis has traveled from one measurement point to the other measurement point or when the movement has completed.
Q ... Register No. that stores the measured results ("1" when omitted)

Groove

Measurement pattern

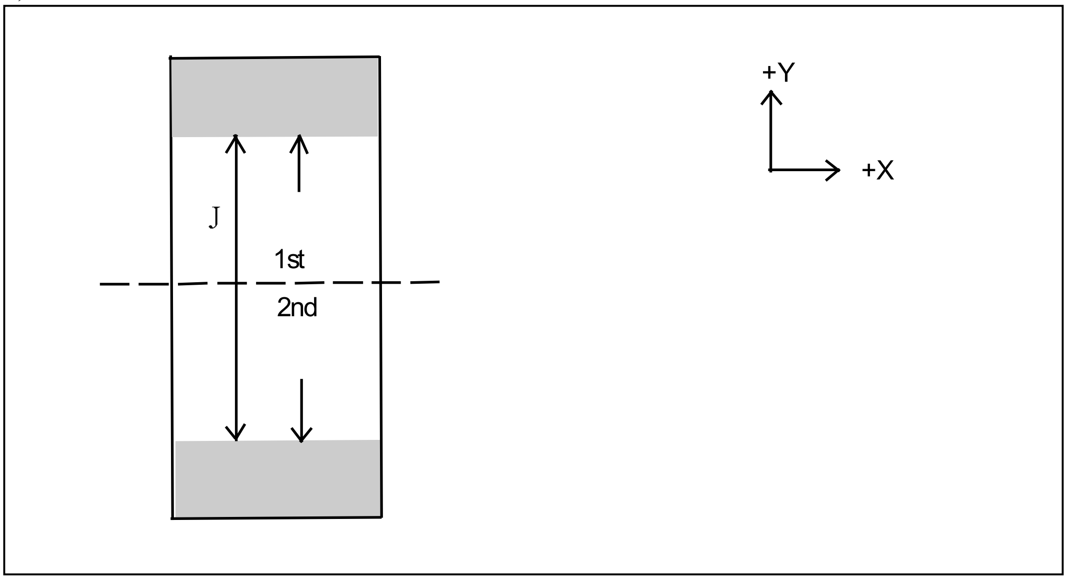
1. Spindle orientation 0 .
   The probe moves to the first measurement start point of the X and Y axes.
2. The probe moves to the Z axis measurement height.
3. The first measurement is carried out .
4. Spindle orientation 18 0. The probe moves to the second measurement start point of the X and Y axes.
5. The second measurement is carried out.
6. The probe moves to the Z axis return point.

a) When I is commanded



eNCPR9.21.ai

b) When J is commanded



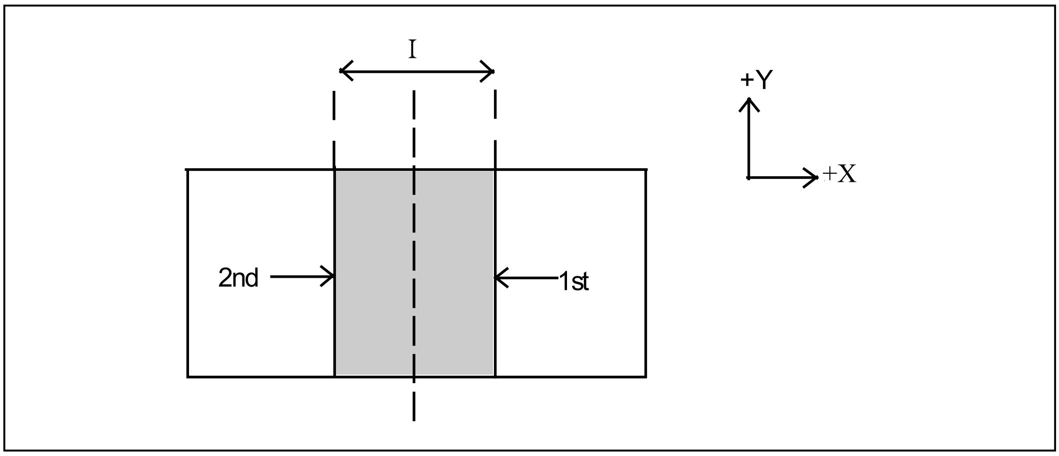
eNCPR9.22.ai

Boss

Measurement pattern

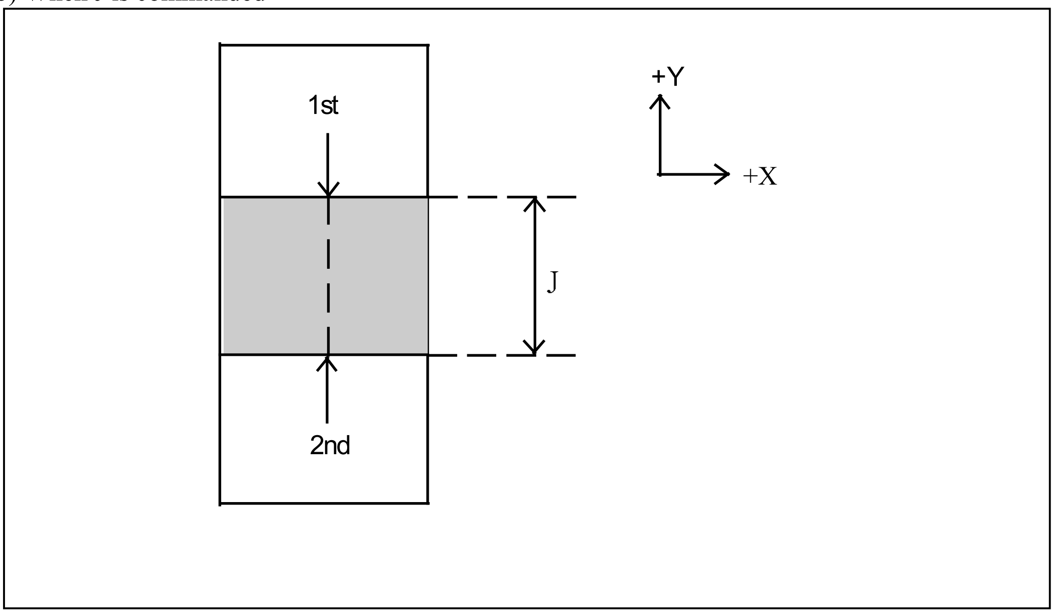
1. Spindle orientation 180°.
   The probe moves to the first measurement start point of the X and Y axes.
2. The probe moves to the Z axis measurement height.
3. The first measurement is carried out.
4. The probe moves to the Z axis return point.
5. Spindle orientation 0°.
   The probe moves to the second measurement start point of the X and Y axes.
6. The probe moves to the Z axis measurement height.
7. The second measurement is carried out.
8. The probe moves to the Z axis return point.

a) When I is commanded



eNCPR9.23.ai

b) When J is commanded



eNCPR9.24.ai

8.3.3 Circle

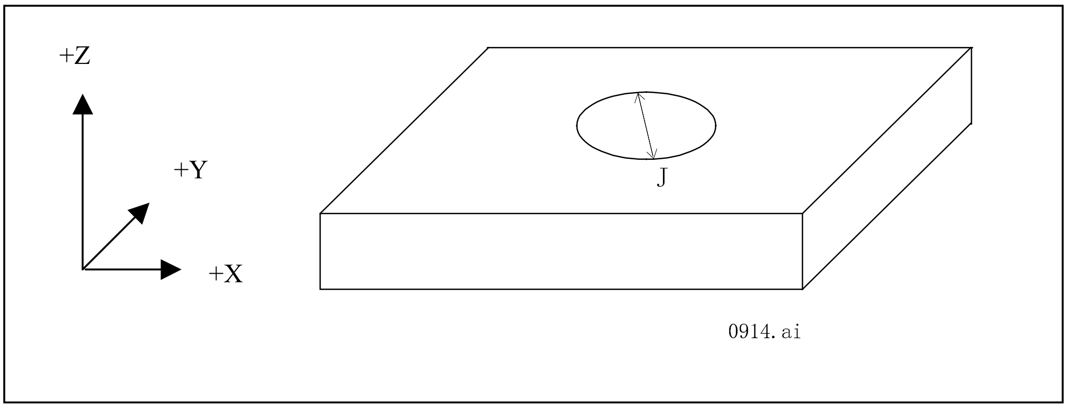
The circle center is calculated by measuring three points.

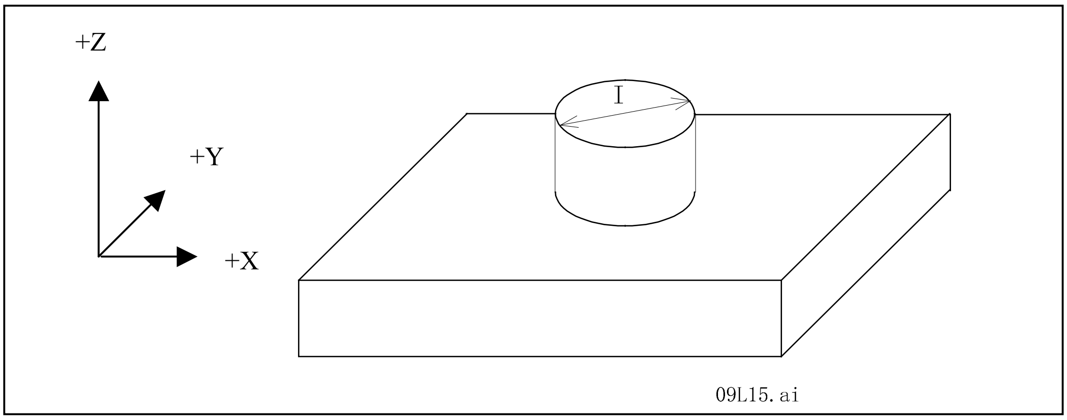
Command format	hole G124 X_ Y_ I_ D_Z_ R_Q_ boss G125 X_ Y_ I_ D_ Z_ R_Q_

The circle center is calcutaled by measuring four points.

Command format	hole G126 X_ Y_ I_D_ Z_ R_Q_ boss G127 X_ Y_ I_ D_Z_ R_Q_

X,Y … Estimated hole (boss) center value
I … Diameter of measuring circle
D … Tool offset number
Z … Z coordinate during measurement
R … Z coordinate of return point when the Z axis has traveled from one measurement point to the other measurement point or when the movement has completed.
Q … Register No. that stores the measured results ("1" when omitted)



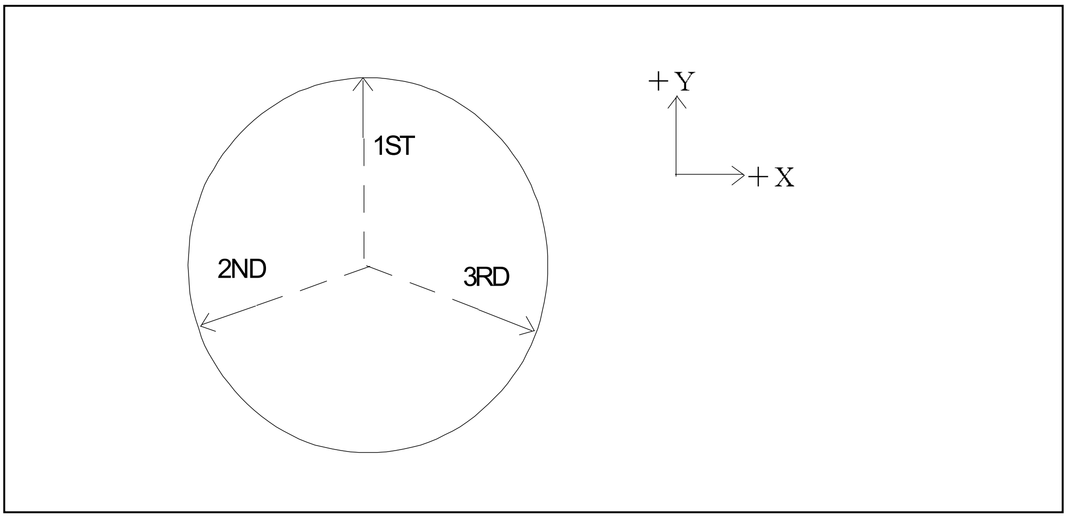


8

Measurment pattern

Hole ... Three-point measurement

1. Spindle orientation 0° . The probe moves to the first measurement start point of the X and Y axes.
2. The probe moves to the Z axis measurement height.
3. The first measurement is carried out (in the +Y direction).
4. Spindle orientation 0° . The probe moves to the second measurement start point of the X and Y axes.
5. The second measurement is carried out (in the direction of 120° from the first point).
6. Spindle orientation 0° . The probe moves to the third measurement start point of the X and Y axes.
7. The third measurement is carried out (in the direction of 240° from the first point).
8. The probe moves to the Z axis return point.

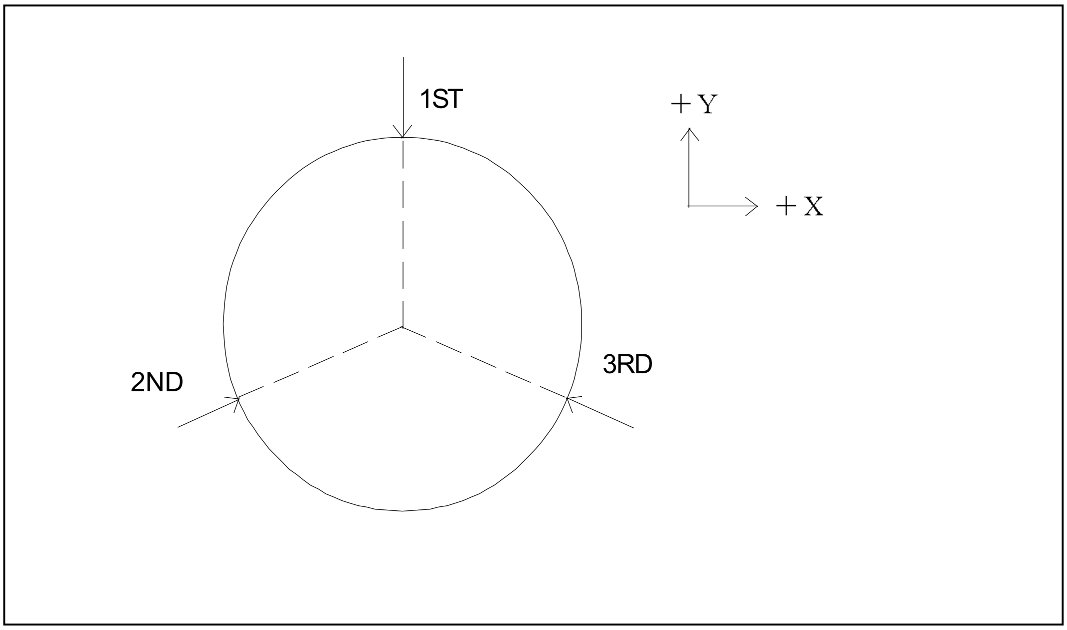


eNCPR9.25.ai

Boss … Three-point measurement

1. Spindle orientation 180° . The probe moves to the first measurement start point of the X and Y axes.
2. The probe moves to the Z axis measurement height.
3. The first measurement is carried out (in the -Y direction).
4. The probe moves to the Z axis return point.
5. Spindle orientation 180° . The probe moves to the second measurement start point of the X and Y axes.
6. The probe moves to the Z axis measurement height.
7. The second measurement is carried out (in the direction of 120° from the first point).
8. The probe moves to the Z axis return point.
9. Spindle orientation 180° . The probe moves to the third measurement start point of the X and Y axes.
10. The probe moves to the Z axis measurement height.
11. The third measurement is carried out (in the direction of 240° from the first point).
12. The probe moves to the Z axis return point.

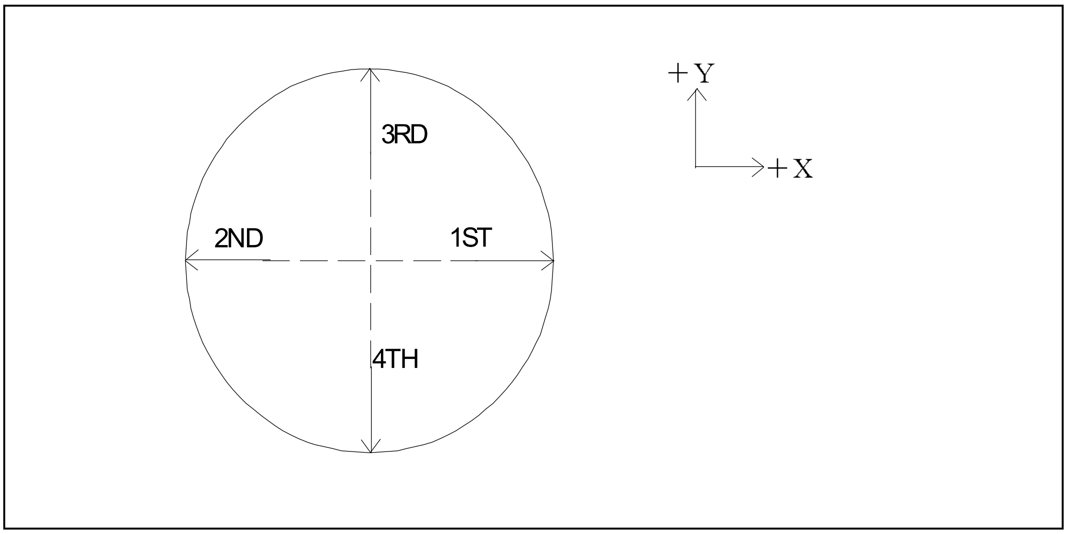
8



eNCPR9.26.ai

Hole ... Four-point measurement

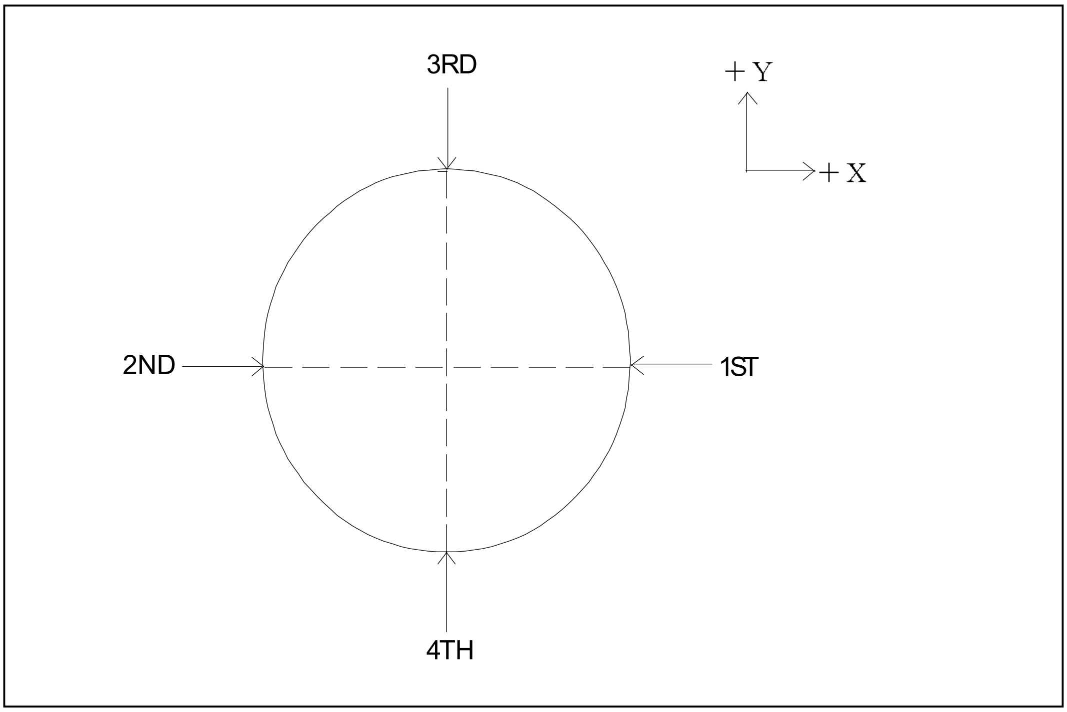
1. Spindle orientation 0°. The probe moves to the first measurement start point of the X and Y axes.
2. The probe moves to the Z axis measurement height.
3. The first measurement is carried out (in the +X direction).
4. Spindle orientation 180°. The probe moves to the second measurement start point of the X and Y axes.
5. The second measurement is carried out (in the -X direction).
6. Spindle orientation 0°. The probe moves to the third measurement start point of the X and Y axes.
7. The third measurement is carried out (in the +Y direction).
8. Spindle orientation 180°. The probe moves to the third measurement start point of the X and Y axes.
9. The fourth measurement is carried out (in the -Y direction).
10. The probe moves to the Z axis return point.



eNCPR9.27.ai

Boss ... Four-point measurement

1. Spindle orientation 180° . The probe moves to the first measurement start point of the X and Y axes.
2. The probe moves to the Z axis measurement height.
3. The first measurement is carried out (in the -X direction).
4. The probe moves to the Z axis return point.
5. Spindle orientation 0° . The probe moves to the second measurement start point of the X and Y axes.
6. The probe moves to the Z axis measurement height.
7. The second measurement is carried out (in the +X direction).
8. The probe moves to the Z axis return point.
9. Spindle orientation 180° . The probe moves to the third measurement start point of the X and Y axes.
10. The probe moves to the Z axis measurement height.
11. The third measurement is carried out (in the -Y direction).
12. The probe moves to the Z axis return point.
13. Spindle orientation 0° . The probe moves to the fourth measurement start point of the X and Y axes.
14. The probe moves to the Z axis measurement height.
15. The fourth measurement is carried out (in the +Y direction).
16. The probe moves to the Z axis return point.



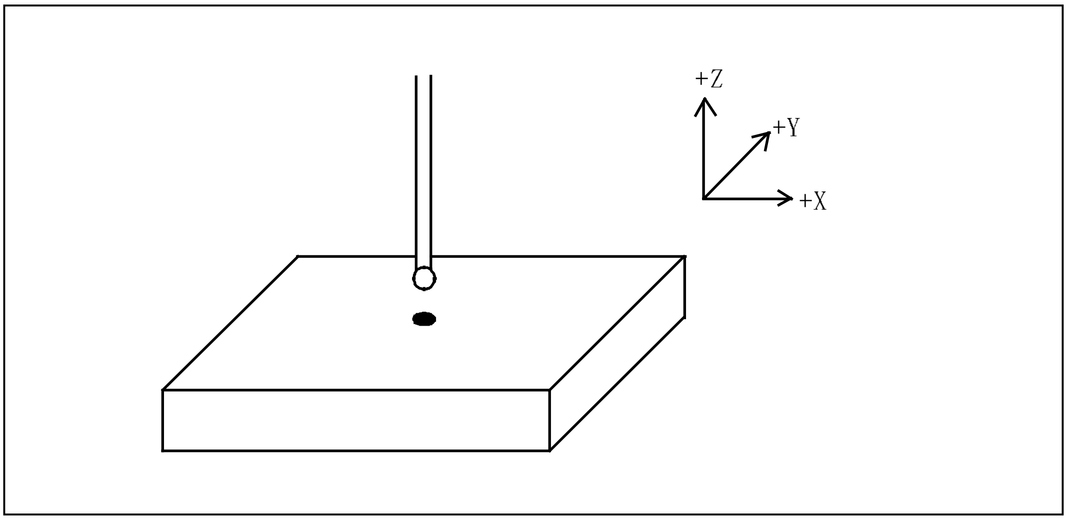
eNCPR9.28.ai

8.3.4 Z LEVEL

Command format

G128 X_ Y_ Z_Q_

X,Y ... X and Y coordinates of measuring point
Z ... Z coordinate of measuring start point
Q ... Register No. that stores the measured results ("1" when omitted)



eNCPR9.29.ai

Measurment pattern

1. Spindle orientation 0°.
   The probe moves to the first measurement start point of the X and Y axes.
2. The probe moves to the Z axis measurement height.
3. The first measurement is carried out (in the -Z direction).

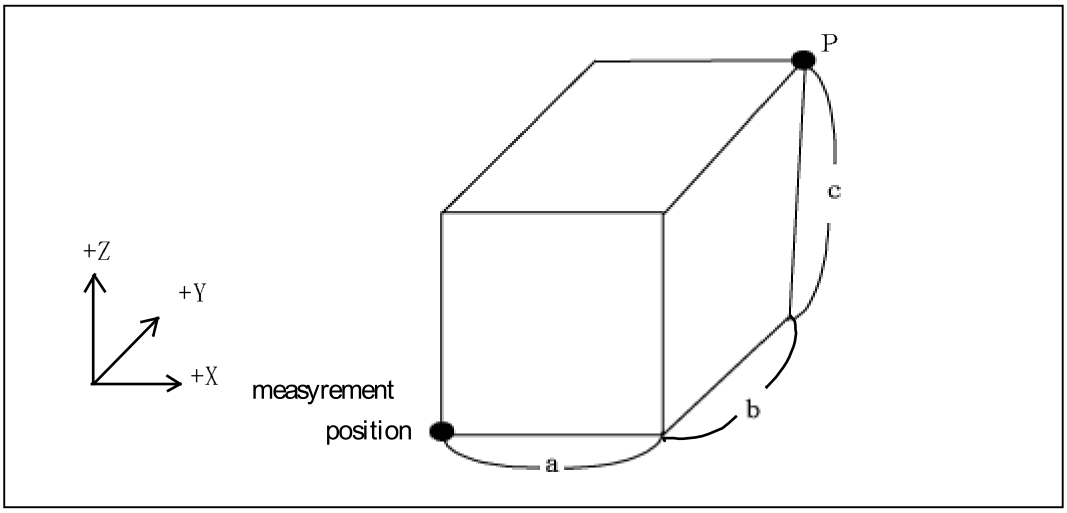
8

8.3.5 Positioning to the measurement position

Command format

G120 X_ Y_ Z_Q_

X,Y and Z... Incremental amount from the measurement position
Q ... Select the desired register No. that stores the measured results ("1" when omitted).



eNCPR9.30.ai

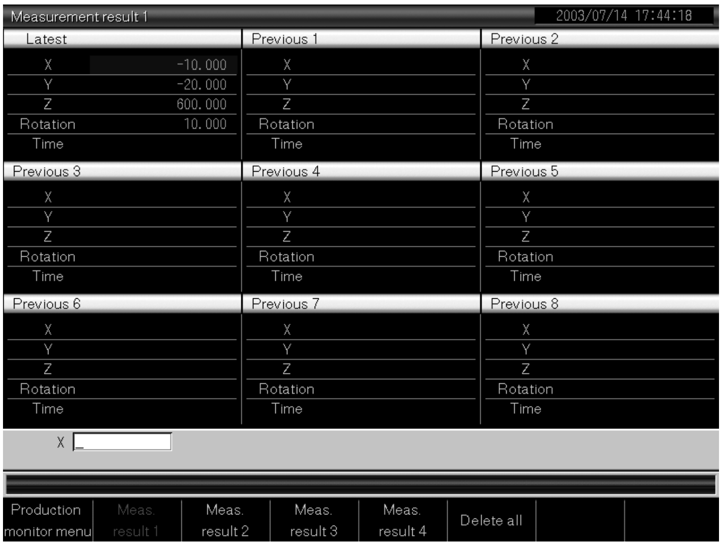
G120 Xa Yb Zc

When **[G120 Xa, Yb, and Zc]** is commanded, the probe moves to point P.
When the measurement data does not exist, NO MEASRUING DATA alarm will occur.

8.4 Handling of the measured results

8.4.1 Display of the measured results

The following screen appears when **[6]** is pressed on the <PRODUCTION MONITOR MENU> screen or the cursor is moved to **[6. MEASUREMENT RESULTS]** and **[ENT]** is pressed.



S07005u1.bmp

When you continue to another measurement, previous measurement results are displayed.

T07005u2.bmp

8.4.2 Reflection of measured results on workpiece coordinate system

The measured results are reflected in the workpiece coordinate system.

Command format

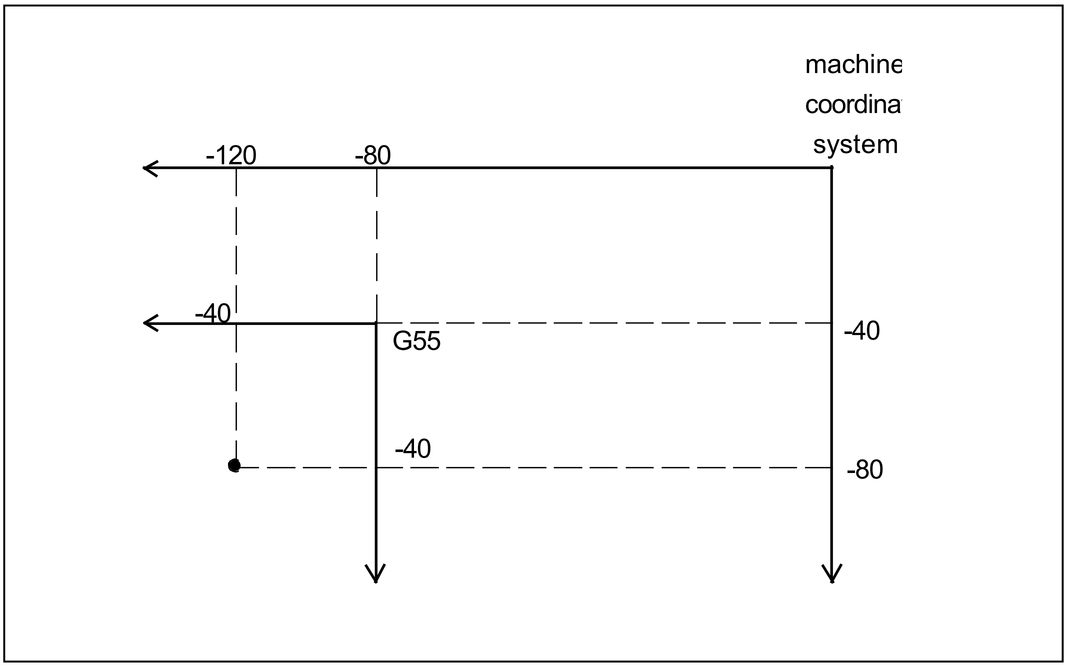
```
G10 L99    Pn          X_   Y_   Z_   Q_
Pn:   n=   1     G54
           2     G55
           3     G56
           4     G57
           5     G58
           6     G59
```

Command format

```
G10 L98    Pn          X_   Y_   Z_   Q_
      Pn:  n=1   G54.1 P1
           n=2   G54.1 P2
           .     .
           .     .
           .     .
           n=48  G54.1 P48
```

X, Y, Z .... Coordinate of the measured position
Q .... Measurement No. to be used

e.g. Assume that the value measured by measurement No. 2 is (X, Y) = (-80, -120) in the machine coordinate system. To assign this position to (-40, -40) of the absolute coordinate system G55, command **[G10, L99 P2 X-40 Y-40 Q2]**.



The working coordinate data G55 is changed to X = -40.000 and Y = -80.000.

8.5 Lock key operations

DRY RUN

The probe moves to the measurement start point, but measurement is not carried out. Measurement data is not transferred, either.

MACHINE LOCK

Axes are not moved. The coordinate value on the<POSITION> screen varies.

RESTART

When G121 to G129 are commanded during restarting, RESTART ERROR will occur.

CHAPTER 9

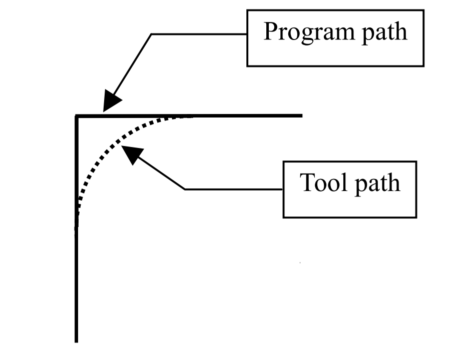
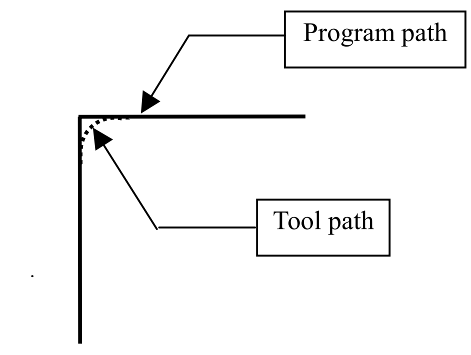
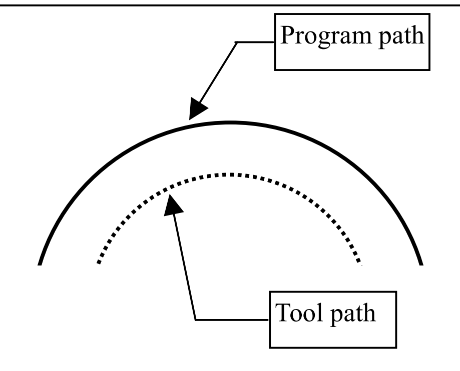
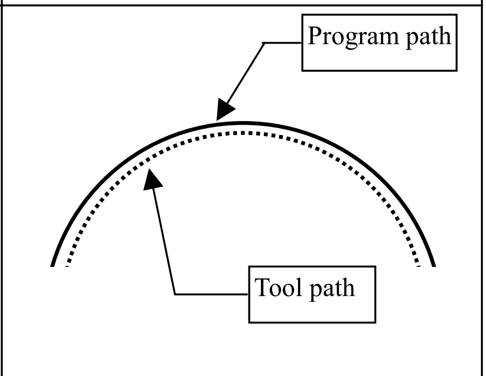
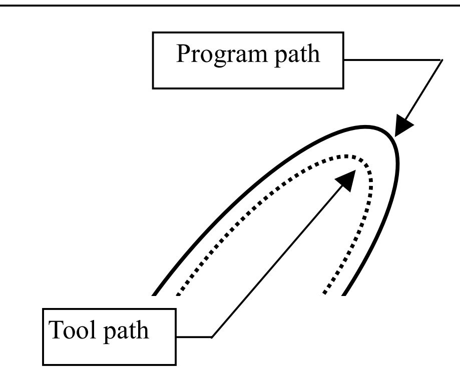
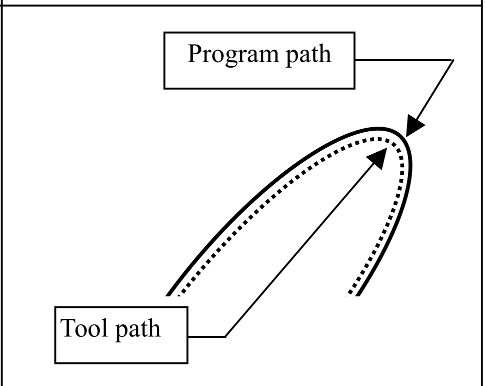
HIGH ACCURACY MODE A

9

9.1 Outline

High accuracy mode A is a function for highly accurate machining at high speed. It is ideal for contouring and 3D workpiece machining. This function enables you to machine workpieces at high speed yet maintain accuracy. It is also possible to change the shape accuracy where necessary by setting override settings for user parameters. Up to three override patterns can be set for parameters, and these can be changed in the NC program.
Functions available in high accuracy mode A are given below. See section 4 for details.

(1) Automatic corner deceleration function
(2) Automatic arc deceleration function
(3) Automatic curve approximation deceleration function

	When high accuracy mode A is OFF	When high accuracy mode A is ON
(1) Automatic corner deceleration function		
(2) Automatic arc deceleration function		
(3)Automatic curve approximation deceleration function		

9.2 Usage

9.2.1 User parameter setting

High accuracy mode A has three deceleration functions. Adjusting the settings for user parameters (Corner deceleration override, Arc deceleration override, Curve approximation deceleration override) alters the shape accuracy. In addition to this, up to three patterns (levels 1 to 3) can be set for parameters, and these can be changed in the NC program. Each level is controlled in reference to the corresponding settings shown below.

M code	Level	User parameter (switch 1)
M260	1	Corner deceleration override 1
		Arc deceleration override 1
		Curve approximation deceleration override 1
M261	2	Corner deceleration override 2
		Arc deceleration override 2
		Curve approximation deceleration override 2
M262	3	Corner deceleration override 3
		Arc deceleration override 3
		Curve approximation deceleration override 3

9.2.2 User parameter description

Parameter name	Descriptions	Setting range (%)
Corner deceleration override 1	Set the automatic corner deceleration override for level 1 (M260) in high accuracy mode A. When 100 is set, automatic corner deceleration is performed using the machine's unique deceleration rate. When a value higher than 100 is set, the deceleration rate decreases and machining time is shortened. When a value lower than 100 is set, accuracy improves. When 0 is set, automatic corner deceleration is not performed.	10 ~ 1000
Arc deceleration override 1	Set the automatic arc deceleration override for level 1 (M260) in high accuracy mode A. When 100 is set, automatic arc deceleration is performed using the machine's unique deceleration rate. When a value higher than 100 is set, the deceleration rate decreases and machining time is shortened. When a value lower than 100 is set, accuracy improves. When 0 is set, automatic arc deceleration is not performed.	10 ~ 1000
Curve approximation deceleration override 1	Set the automatic curve approximation deceleration override for level 1 (M260) in high accuracy mode A. When 100 is set, automatic curve approximation deceleration is performed using the machine's unique deceleration rate. When a value higher than 100 is set, the deceleration rate decreases and machining time is shortened. When a value lower than 100 is set, accuracy improves. When 0 is set, automatic curve approximation deceleration is not performed.	10 ~ 1000
Corner deceleration override 2	Set the automatic corner deceleration override for level 2 (M261) in high accuracy mode A. The function is the same as level 1.	10 ~ 1000
Arc deceleration override 2	Set the automatic arc deceleration override for level 2 (M261) in high accuracy mode A. The function is the same as level 1.	10 ~ 1000
Curve approximation deceleration override 2	Set the automatic curve approximation deceleration override for level 2 (M261) in high accuracy mode A. The function is the same as level 1.	10 ~ 1000
Corner deceleration override 3	Set the automatic corner deceleration override for level 3 (M262) in high accuracy mode A. The function is the same as level 1.	10 ~ 1000
Arc deceleration override 3	Set the automatic arc deceleration override for level 3 (M262) in high accuracy mode A. The function is the same as level 1.	10 ~ 1000
Curve approximation deceleration override 3	Set the automatic curve approximation deceleration override for level 3 (M262) in high accuracy mode A. The function is the same as level 1.	10 ~ 1000

9.2.3 Usage in a program

Use the following M codes to use high accuracy mode A.

```
M260：High accuracy mode A (level 1) on
M261：High accuracy mode A (level 2) on
M262：High accuracy mode A (level 3) on
M269：High accuracy mode A off
```

An example of use is shown below.

Example

```
(Program example)
NC program
G00 X0 Y0 Z0；
 ；
M260                   ←  High accuracy mode A (level 1) on
G01 X20. Y30. Z50.；  ┐
X40. Y20. Z30.；      ├  Executing high accuracy mode A (level 1)
 ；                    ┘
M269                   ←  High accuracy mode A (level 1) off
 ；
M261                   ←  High accuracy mode A (level 2) on
G01 X20. Y30. Z50.；  ┐
X40. Y20. Z30.；      ├  Executing high accuracy mode A (level 2)
 ；                    ┘
M269                   ←  High accuracy mode A (level 2) off
 ；
M262                   ←  High accuracy mode A (level 3) on
G01 X20. Y30. Z50.；  ┐
X40. Y20. Z30.；      ├  Executing high accuracy mode A (level 3)
 ；                    ┘
M269                   ←  High accuracy mode A (level 3) off
M30；
```

9

9.2.4 Conditions available

G code modal conditions*1) must be as below to use high accuracy mode A. The conditions below are current when the power is turned on.

G64：Cutting mode
G67：Macro cancel
G80：Canned cycle cancel

*1) See the Instruction Manual for check method of G code modal conditions.

9.2.5 Conditions where high accuracy mode A is released

High accuracy mode A is turned off if any of the following operations is performed while high accuracy mode A is on.

(1) Power is turned on.
(2) The [RST] key is pressed.
(3) Any operation that resets memory operation is performed.
(e.g. The [Z.RTN] key is pressed in manual mode.)
(4) End of program (M02, M03) is executed.

9

9.3 Restrictions

9.3.1 Functions available

Functions that can be used while high accuracy mode A is on are given below.

1. All M codes
2. C codes in the table below.

G code	Function	G code	Function
0	Positioning	51	Scaling
1	Linear interpolation	51.1	Mirror image
2	Arc (CW)	52	Local coord. system setting
3	Arc (CCW)	53	Machine coord. system selection
4	Dwell	54	Workpiece coord. system selection 1
9	Exact stop (one shot)	54.1 ~	Extended workpiece coord. system selection
10	Programmable data input	55	Workpiece coord. system selection 2
12	Circular cutting CW	56	Workpiece coord. system selection 3
13	Circular cutting CCW	57	Workpiece coord. system selection 4
17	XY plane selection	58	Workpiece coord. system selection 5
18	ZX plane selection	59	Workpiece coord. system selection 6
19	YZ plane selection	64	Cutting mode
22	Programmable stroke limit valid	65	Macro call
23	Programmable stroke limit invalid	68	Rotational transformation
28	Reference position return	69	Rotational transformation cancel
29	Return from reference position	80	Canned cycle cancel
30	2nd to 6th reference position return	90	Absolute command
36	Coordinate calculation function (bolt hole)	91	Incremental command
37	Coordinate calculation function (line (angle))	92	Workpiece coord. system setting
38	Coordinate calculation function (line (X, Y))	94	Initial position return
39	Coordinate calculation function (line (grid))	98	Minute feed
40	Cutter compensation cancel	99	R-point return
41	Cutter compensation left	100	Tool change
42	Cutter compensation right	102	XZ circular interpolation CW
43	Tool length offset +	103	XZ circular interpolation CCW
44	Tool length offset -	168	Rotation transformation using meas. result
49	Tool length offset cancel	202	YZ circular interpolation CW
50	Scaling cancel	203	YZ circular interpolation CCW
50.1	Mirror image cancel		

9.3.2 Additional axis travel command

When additional axis travel command is used while high accuracy mode A is on, high accuracy mode A is released temporarily during additional axis travel.

In addition, commands that move the feed axis and additional axis simultaneously by cutting feed (G01, G02, G03) cannot be used.

9

9.4 Effective Functions

The functions below are available while high accuracy mode A is on.

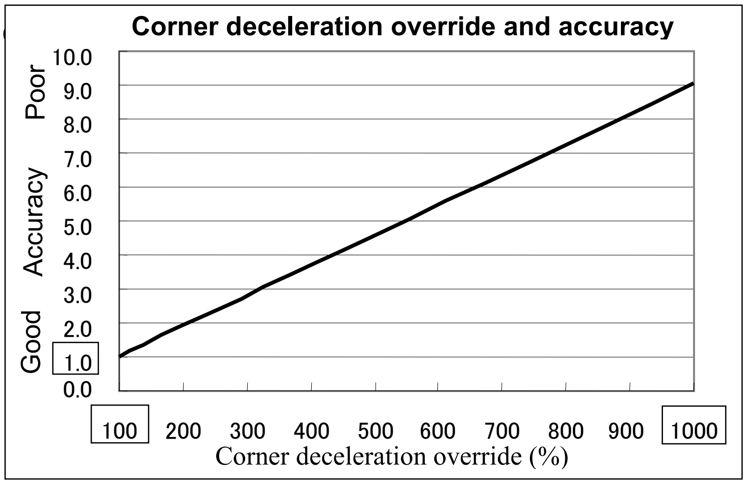
(1) Automatic corner deceleration function
(2) Automatic arc deceleration function
(3) Automatic curve approximation deceleration function

9.4.1 Automatic corner deceleration function

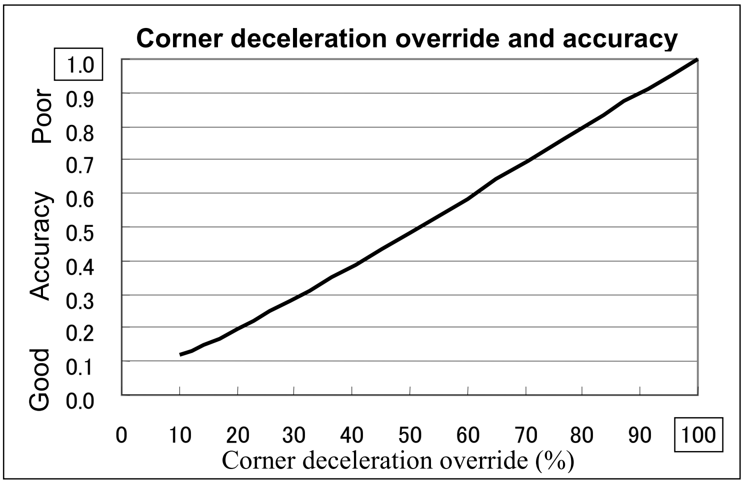
When machining a corner, the actual tool path gradually deviates from the program path as the tool approaches the corner. This results in an error. In addition to this, the error becomes larger as the feed rate increases at the corner.
This function automatically decelerates the feed rate only around the corner according to the corner deceleration override setting so that the corner shape accuracy specified by the NC program is maintained.
The setting range of the corner deceleration override is 0%, or 10% to 1000% (100% is the default).
The smaller the setting, the more effective the deceleration function, minimizing error and improving shape accuracy.
When 0% is entered, the automatic corner deceleration function is turned off.
The graph below shows the accuracy relative to the corner deceleration override setting.

How to read graph:
Accuracy at corner when 100% is set for corner deceleration override is "1".

[1] Corner deceleration override (100% ~ 1000%)



[2] Corner deceleration override (10% ~ 100%)

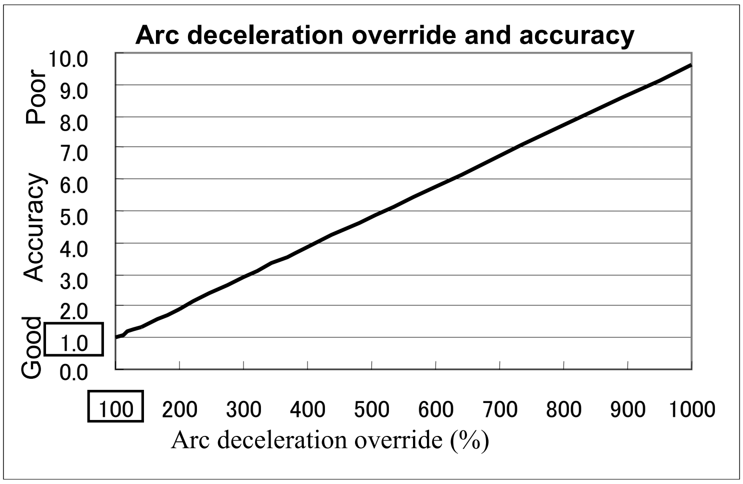


9.4.2 Automatic arc deceleration function

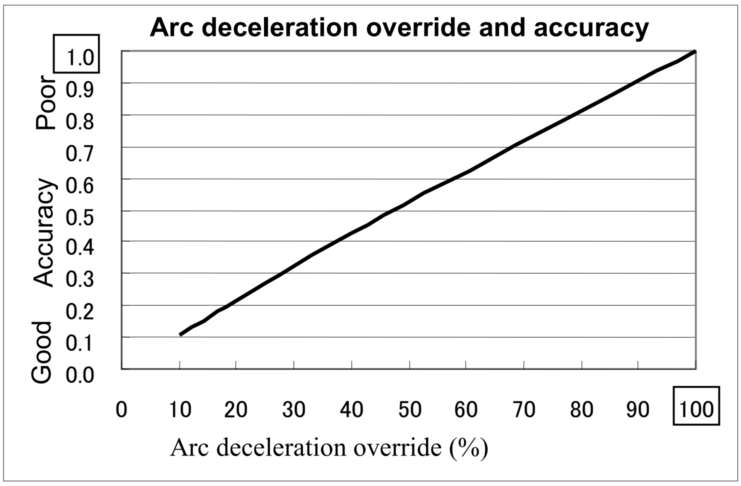
When performing circular interpolation, a radial error in the actual tool path occurs relative to the specified circular arc, and the arc radius decreases. In addition to this, the error becomes larger as the feed rate increases.
This function automatically decelerates the arc cutting feed rate according to the arc deceleration override setting so that the radial arc shape accuracy specified by the NC program is maintained.
The setting range of the arc deceleration override is 0%, or 10% to 1000% (100% is the default).
The lower the setting is, the more effective the deceleration function, minimizing error and improving shape accuracy.
When 0% is entered, the automatic arc deceleration function is turned off.
The graph below shows the accuracy relative to the arc deceleration override setting.

How to read graph:
Circular interpolation accuracy when 100% is set for arc deceleration override is "1".

[1] Arc deceleration override (100% ~ 1000%)



[2] Arc deceleration override (10% ~ 100%)



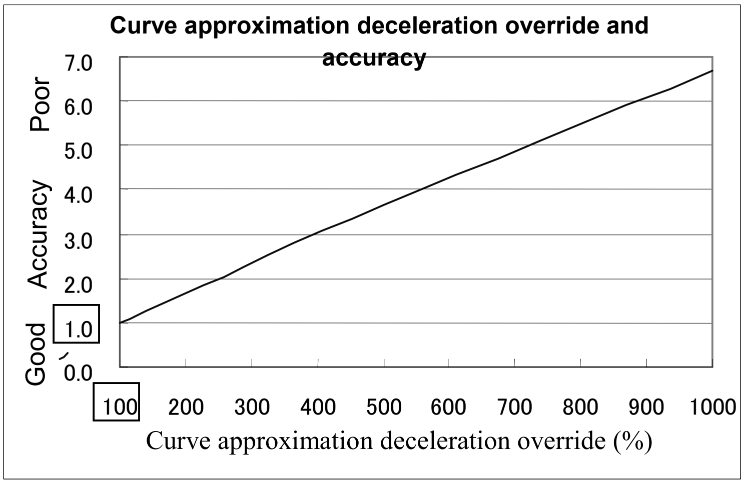
9

9.4.3 Automatic curve approximation deceleration

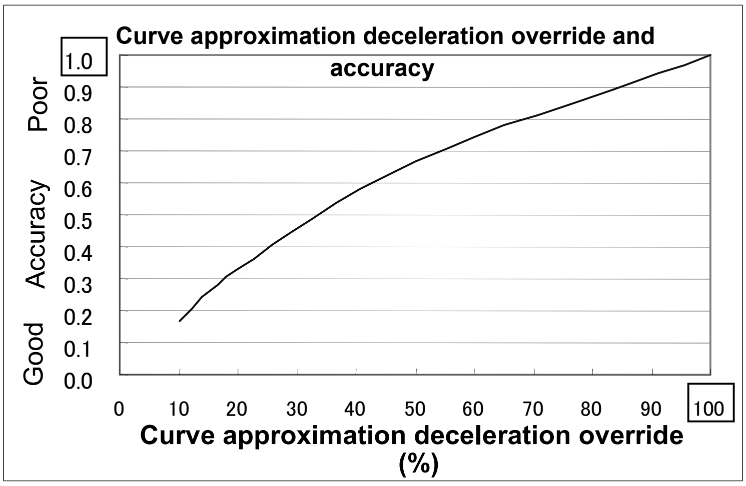
This function automatically decelerates the curve approximation block feed rate according to the curve approximation deceleration override setting so that the shape accuracy for the curve approximation block (curve composed of minute blocks) specified by the NC program is maintained.
The setting range of the curve approximation deceleration override is 0%, or 10% to 1000% (100% is the default).
The lower the setting is, the more effective the deceleration function, minimizing error and improving shape accuracy.
When 0% is entered, the automatic curve approximation deceleration function is turned off.
The graph below shows the accuracy relative to the curve approximation deceleration override setting.

How to read graph:
An error for curve approximation block when 100% is set for curve approximation deceleration override is "1".

[1] Curve approximation deceleration override (100% ~ 1000%)



[2] Curve approximation deceleration override (10% ~ 100%)



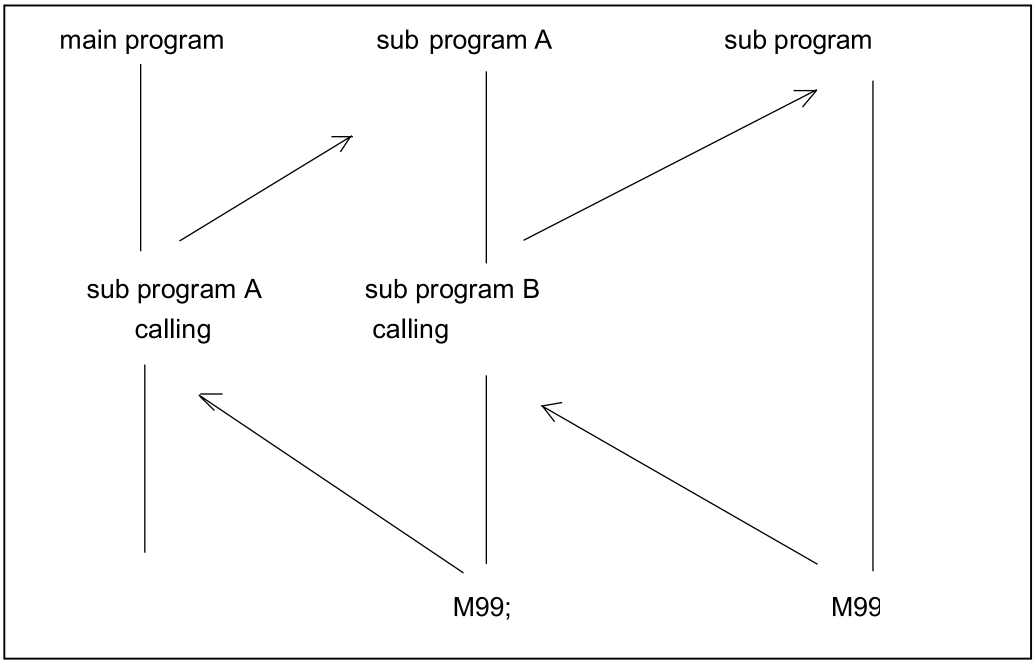
CHAPTER 10

SUBPROGRAM FUNCTION

10

10 Function of Subprogram

When a program contains fixed sequences or frequently repeated patterns, these sequences or patterns may be entered into the memory as a subprogram.
The subprogram can be called out in the memory operation mode.
A subprogram called by the main program can also call another subprograms.
Max 8 fold nesting is available.



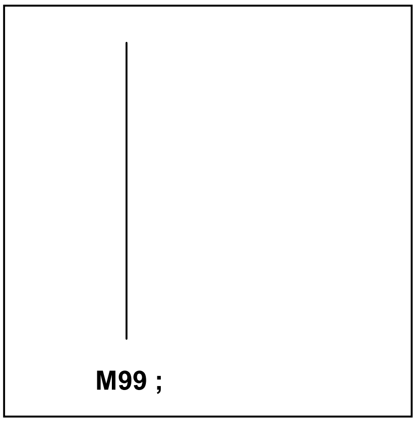
eNCPR11.01.ai

One calling command can call a subprogram repeatedly.

10.1 Making Subprogram

10

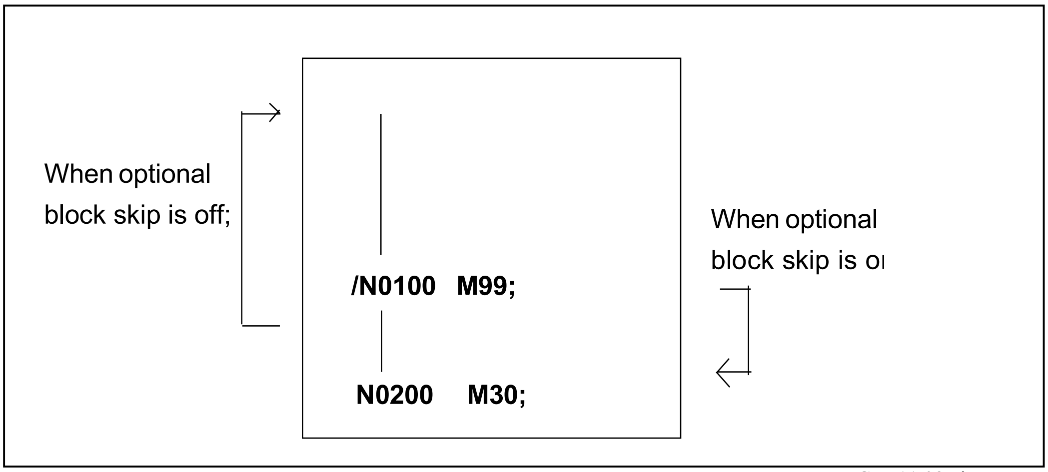
Generally, a subprogram is made by the following format.



(Note 1) Specify M99 at the last block of subprogram.
(Note 2) If another G or M code is commanded in the M99 block, an alarm is generated.
The axis movement is not available even by commanding X, Y or Z address.

Special uses of M99

If the M99 command is executed in the main program, the control returns to the start of the main program.



eNCPR11.02.ai

If the programming is done as shown above, and if the optional block skip is OFF, the control returns to the start of the main program and executes the program repeatedly up to N0100.
If the optional block skip is turned ON, the N0100 block is omitted and control goes to the next block.

10.2 Simple Call

A subprogram can be called out from the main program or another subprogram and executed accordingly.

Command format

```
M98 P_L_ ;
```

P : Subprogram number to be called.
L : Number of execution to be repeated. (Max. 9999)

If L is omitted, it is regarded as once.

(Note) **If another G or M code is commanded in the M98 block, an alarm is generated.**
The axis movement is not available even by commanding X, Y or Z address.

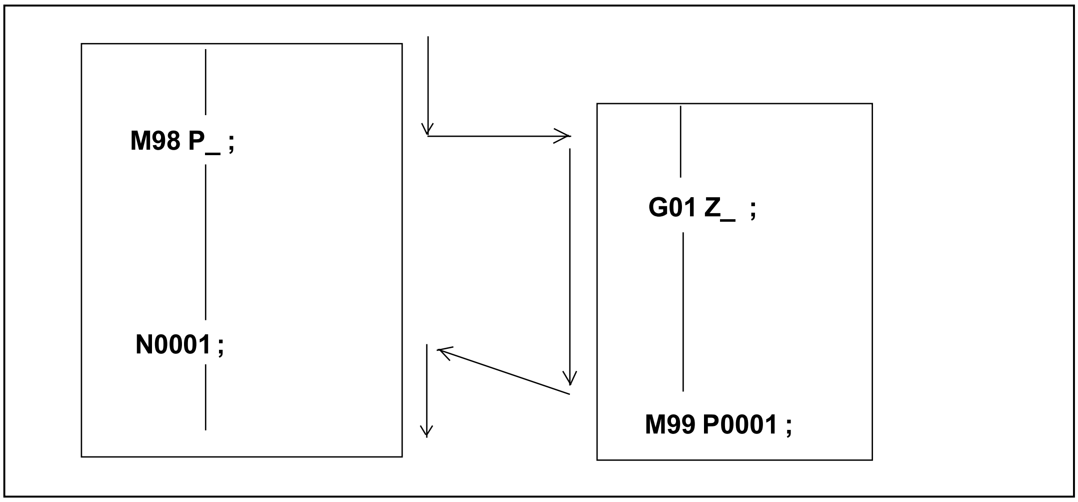
10.3 Return No. Designation from Program

Command format

M99 P_ ;

10.3.1 Command by sub program

(Program execution sequence)
When the command is executed, the program returns the commanded sequence No.of the parent program. The sequence No is serched from the top of the program,and the program returns to the block initially found.
An alarm will occur when the commanded sequence No.is not present.



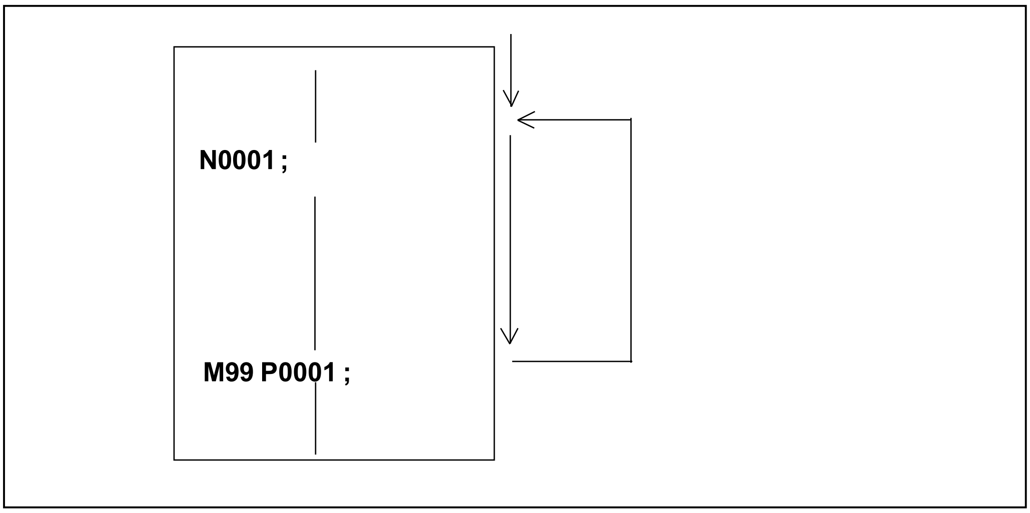
eNCPR11.03.ai

(Note) An alarm will occur unless the number of repeats commanded by M98 is "1".

10.3.2 Command by main program

When the command is executed the program to the sequence No. commanded within the main program. The sequence No.is searched from the top of the program ,and the program jump to the block initially found.
An alarm will occur when the commanded sequence No. is not present.

(Program execution sequence)



eNCPR11.04.ai

CHAPTER 11

FEED FUNCTION

11 Feed Function

Feedrate is specified by the number following address F.

(1) Command range

Metric system :	1~999999 mm/min
	1~999999° /min
Inch system :	0.1~99999.9 inch/min
	0.1~99999.9° /min

(2) Clamp

If the axis movement at a higher feedrate than the values specified by the machine parameter, an alarm is generated.

CHAPTER 12

S,T,M FUNCTION

12 S,T,M Function

By commanding the following functions, machine motions other than the axis movements are available.
S : Spindle speed command
T : Tool magazine number command
M : ON/OFF command of various solenoids of the machine

12.1 S Function

The S code is used for specifying the spindle speed.
The spindle speed is specified by the address S and a following within 5-digit number.

(Note 1)
The S command is not cancelled by the NC reset function, but it becomes zero when the power is turned ON.
(Note 2)
The S command should be always given before commanding the spindle rotation (M03,M04)
(Note 3)
When the S command is in the same block as that of axis movement, the S command becomes effective at the same time the axis movement starts.

12.2 T Function

When commanded, the magazine rotates to the corresponding pot.

Three commanded forms are provided as below.

12.2.1 Commanded by tool No.

Command format

T_ _ _;

Commanded by tool No. after “T” (T1-T99), and the pot with the corresponding tool attached is Indexed.

12.2.2 Commanding by pot No.(magazine No.)

Commanding by pot No.
Commanded the pot number (magazine number) using the two digit numerical values after “T1”, and the pot with the corresponding tool attached is indexed. (T101-T 1nn , “nn” indicates the maximum numbers of the attached pots.)

12.2.3 Commanded by group No.

Commanded the group number using the two digit numerical value after “T9” (T901-T930), and the pot with the corresponding tool attached is indexed.

12.3 M Function

The M codes are used for commanding ON/OFF of various solenoids of the machine.
It is commanded by address M and a following within 3-digit number.

When the M command is in the same block as that of the axis movement, the motion is devided following three types.

The M command becomes effective before the axis movement starts.
The M command becomes effective at the same time the axis movement starts.
The M command becomes effective after the axis movement is finished.

(Note)
The modal command is effective until it is cancelled by the next M code or changed.
The one-shot command is effective only in the commanded block.

List of M code (1)

Group	M code	Content	Operation order v. axis feed	Modal/ one-shot
	M00	Program stop	after	one-shot
	M01	Optional stop	after	one-shot
	M02	End of program	――――	
	M30	End of program		
	M03	Spindle CW	simultaneous	modal
	M04	Spindle CCW		
	M05*	Spindle stop		
	M19	Spindle orientation		
	M111	Spindle orientation (180°)		
	M08	Coolant pump ON	simultaneous	modal
	M09*	Coolant pump OFF	after	
	M06	Tool change	simultaneous	one-shot
	M98	Sub program call	――――	
	M99	Return from subprogram		

The code with * is already set when the power is turned on.

12

List of M code (2)

Group	M code	Content	Operation order vs. axis feec	Modal/ one-shot
	M400	M400 ON (Chip shower On)	simultaneous	modal
	M401*	M400 OFF (Chip shower Off)		
	M402	M402 ON	simultaneous	modal
	M403*	M402 ON		
	M404	M404 ON	simultaneous	modal
	M405*	M404 OFF		
	M406	M406 ON		
	M407*	M406 OFF		
	M408	M408 ON (Oil hole On)		
	M409*	M408 OFF (Oil hole Off)	after	modal
	M490	Center through coolant ON	before	modal
	M491	Center through coolant ON		
	M492	Center through coolant ON		
	M493	Center through coolant ON		
	M494	Center through coolant ON		
	M495*	Center through coolant OFF	after	modal
	M496	Back washing cycle ON	before	one-shot
	M211	Workpiece counter 1set	Simultaneous	modal
	M221*	Workpiece counter 1 cancel		
	M212	Workpiece counter 2set	Simultaneous	modal
	M222*	Workpiece counter 2 cancel		
	M213	Workpiece counter 3 set	Simultaneous	modal
	M223*	Workpiece counter 3 cancel		
	M214	Workpiece counter 4 set	Simultaneous	modal
	M224*	Workpiece counter 4 cancel		
	M230*	Tool life counter set	Simultaneous	modal
	M231	Tool life counter cancel		
	M800~ M899	Signal output * for sequence controller	Simultaneous	one-shot
	M900~ M999	Extend signal output	Simultaneous	one-shot

The code with * is already set when the power is turned on.

List of M code (3)

Group	M code	Content	Operation order vs. axis feed	Modal/ one-shot
	M120	TOUCH signal check	after	one-shot
	M121	TOUCH signal check	after	one-shot
	M200	Tool breakage detection (with return motion)	after	one-shot
	M201	Tool breakage detection	after	one-shot
	M320	Measurement device sensor ON confirmation	————	one-shot
	M321	Measurement device sensor OFF confirmation	————	one-shot
	M350	Thermal displacement compensation (X)	before	one-shot
	M351	Thermal displacement compensation (Y)	before	one-shot
	M352	Thermal displacement compensation (Z)	before	one-shot
	M353	Thermal displacement compensation (XYZ)	before	one-shot
	M438	Magazine cover open	simultaneous	one-shot
	M439	Magazine cover close		
	M251	Tap time constant control	simultaneous	modal
	M252*	Tap acceleration constant control		
	M260	High precision mode A ON (Level 1)	simultaneous	modal
	M261	High precision mode A ON (Level 2)		
	M262	High precision mode A ON (Level 3)		
	M269*	High precision mode A OFF		
	M270	Positioning finish check distance setting off	simultaneous	modal
	M271~ M279	Positioning finish check distance setting		
	M410	Index of the pallet 2 to the outside		one-shot
	M411	Index of the pallet 1 to the outside		
	M430	Pallet (C axis) unclamp		modal
	M431*	Pallet (C axis) clamp		
	M435*	Magazine rotation maximum speed	simultaneous	modal
	M436	Magazine rotation speed 1		
	M437	Magazine rotation speed 2		
	M501~ M599	Magazine rotation for tool setting position	simultaneous	one-shot

The code with * is already set when the power is turned on.

12

List of M code (4)

Group	M code	Content	Operation order vs. axis feed	Modal/ one-shot
	M440	Unclamping B axis	————	modal
	M441*	Clamping B axis		
	M442	Unclamping A axis	————	modal
	M443*	Clamping A axis		
	M450	One-shot output (Proceeds to the next block after the signal has turned off)	simultaneous	one-shot
	M451			
	M455	One-shot output (Proceeds to the next block without waiting until the signal has turned off.)		
	M456			
	M460	Waiting for M460 signal ON	simultaneous	one-shot
	M461	Waiting for M460 signal OFF		
	M462	Waiting for M462 signal ON		
	M463	Waiting for M462 signal OFF		
	M464	Waiting for M464 signal ON		
	M465	Waiting for M464 signal OFF		
	M466	Waiting for M466 signal ON		
	M467	Waiting for M466 signal OFF		
	M468	Waiting for M468 signal ON		
	M469	Waiting for M469 signal OFF		
	M480	M480 signal ON	simultaneous	modal
	M481*	M480 signal OFF		
	M482	M482 signal ON	simultaneous	
	M483*	M482 signal OFF		
	M484	M484 signal ON	simultaneous	
	M485*	M484 signal OFF		
	M486	M486 signal ON	simultaneous	
	M487*	M486 signal OFF		

The code with * is already set when the power is turned on.

12.3.1 Program stop (M00)

The spindle stops after the commanded motions in a block are all finished.
The coolant pump is turned OFF at this time.
Next sequence is started by pressing the START switch.

(Note)
When the spindle should be rotated in the blocks after the M00 command, command M03 or M04. The coolant pump ON should also be commanded if necessary.

12.3.2 Optional stop (M01)

When the [OPT STOP] key is set ON, similar to the M00, the automatic operation is stopped after a block which contains M01 is executed.

12.3.3 End of program (M02, M30)

This code shows the end of program. Executing this command takes the control return to the head of the program. The NC enters the reset status at this time.

(Note)
If another G or M code is commanded in the M02 or M30 block, an alarm is generated.
The axis movement is not available even by commanding X, Y or Z address.

12.3.4 Commands on the spindle (M03,M04,M05,M19,M111)

When the axis movement command is in the same block, that command and the spindle command are executed at the same time.

12.3.4.1 Spindle orientation to desired angle (M19)

Command format

M19 R_;

R: Spindle angle (-360Deg to 360Deg)

Orients the spindle to the angle commanded by "R".
Turns the spindle clockwise when the angle is commanded in the positive value and counter clockwise when the angle is commanded in the negative value.
After the Spindle is activated, the servo motor stays on.

12.3.5 M signal level output (M400 ~ M409)

External output terminal name	ON (low voltage)	OFF (high voltage)
M400	M400	M401
M402	M402	M403
M404	M404	M405
M406	M406	M407
M408	M408	M409

- Using M401 command turns off the chip shower when the time set for **[CHIP SHOWER DRAIN TIME]** (user parameter - switch 1) has elapsed.

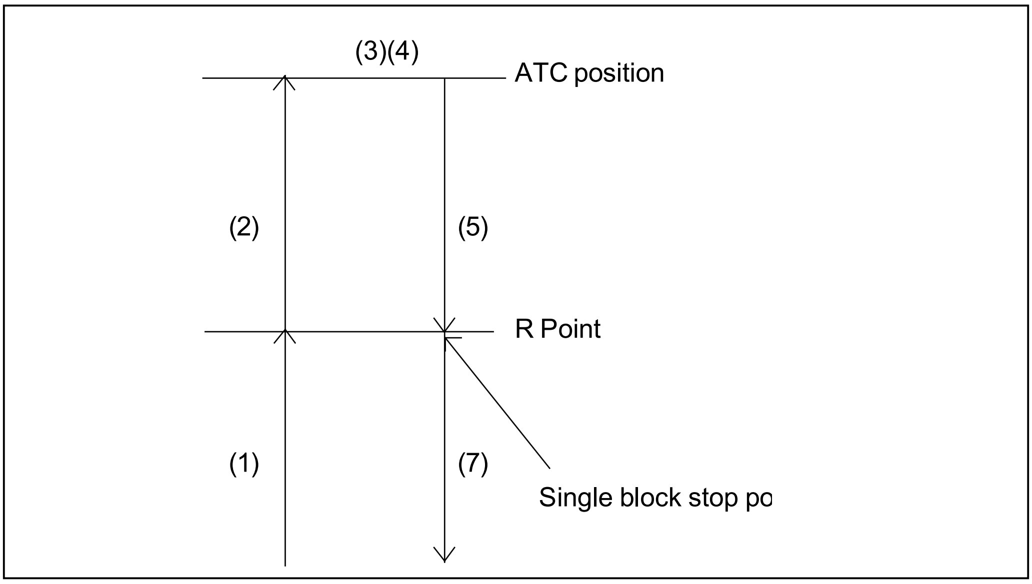
12

12.3.6 Tool change (M06)

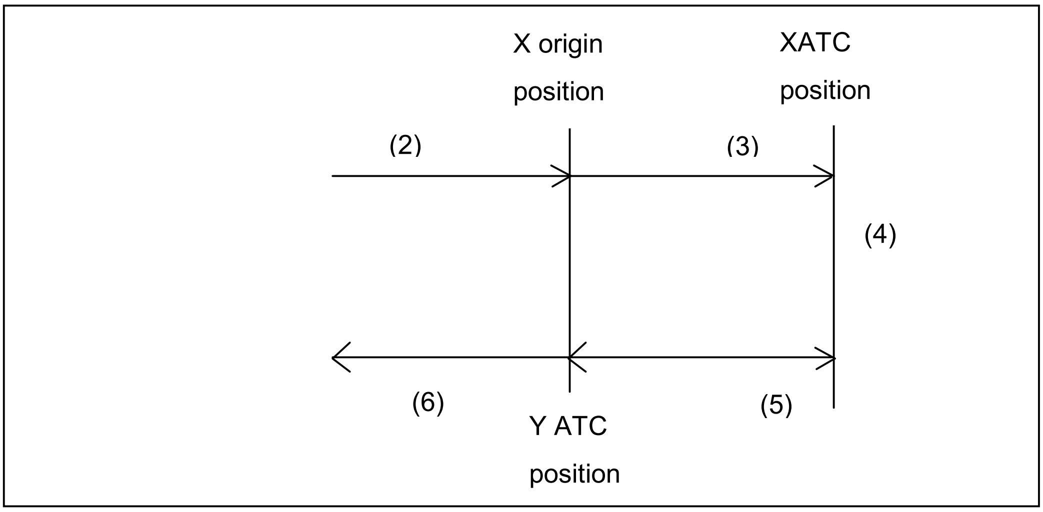
(1) When TC-32B

Command format

M06 T _ X _ Y _ Z _ R _ A _ B _ L _ ;



eNCPR13.03.ai



12

T _ _ : Tool number.
T1 _ _ : Pot number.
T9 _ _ : Group number.
A, B : Target value when moving A and B axes at in rapid feed simultaneously with tool change motion.
X,Y : Target value of movement(6) in rapid feed.
Z : Target value of movement(7) in rapid feed.
R : Retern position before tool change motion.
(operated with tool length compensation applied)
When "R" is not commanded, operation is performed using [ATC SYNCHRONOUS START POS] of USER PARAMETER 1 as "R" command value.
L : Specify the tool number, pot (magazine) number, and group number after “L”.
The pot with the corresponding tool attached is indexed by operation (4) after ATC. (Next tool preparation)

Operations

(1) Tool moves to the Z-axis point "R" while performing 0-degree spindle orientation. When "T" is commanded, magazine swivels.
(2) Tool movement to theZ-axis ATC origin, to the X axis origin point and to Y axis ATC position, A, B axes movement to the the commanded value and also maga zine cover opening occur simultaneously.
(3) Tool moves to theX-axis ATC position.
(4) Arm swivels to change tool.
Tool change motion differs depending on setting on [MAGAZINE TOOL] screen. Refer to the "Tool Change Motion" for detailes.
(5) Tool movement to the X axis orgin point and Z axis point "R" and magazine cover closing occurs stimultaneously.
(6) XY axes moves to the commanded value.
(7) Tool moves to Z-axis commanded position.

Caution

- **When performing cycle operation, tool moves in cutting mode between operations (1)and (6). And tool moves in position check mode.**
  For XY positioning at operation (2) and (3) (from X:origin Y: ATC to X axis positioning (ATC position)) ,tool moves in cutting mode when Y axis positions at the ATC position before X axis does.
- **When [RESET] key or [STOP SWITCH] key is pressed at operations (2), machine stops after Z axis movement to ATC origin point.**
  For X, Y, A, and B axes movement, machine stops immediately.
- **When [RESET] key or [STOP SWITCH] key is pressed between operations (3) and (5) , machine stops after motion (5) is completed.**
  For the operation (2), A, and B axes movement, machine stops immediately.
- **Machine does not stop if single block occurs between operations (2) and (6) .**
- **For operation (5), machine goes to the next opeation after closing the magazine cover witho6ut checking if the magazine cover is closed.**
- **Modes can not be switched between operation (1) and (6).**
- **All data other than M06 can be omitted. However, code "T" must be commanded once by operator before M06 is commanded.**
- **Tool offset is canceled when M06 is commanded. Further, tool length compensation is canceled from operation (2).**
- **When tool offset (G41 or G42) and X and Y axes movement are commanded to block M06, tool offset begins when X and Y axes movement (6 commences.**
  Start-up is performed in format 1, regardless of6 parameters.
- **When tool offset is commanded to block M06, tool offset becomes valid from operation (5) .**
- **When M06 is commanded while tool length compensation (G49) is canceled, operation (1) is performed subject to [ATC REFERENCE TOOL LENGTH] selection for user parameter.**
  When tool offset is not commanded to block M06, operations (5) are performed subject to [ATC REFERENCE TOOL LENGTH] selection for user parameter.
- **Alarm appears when code M other than spindle command is commanded to block M06.**
  Alarm appears when axis A or B is commanded while optional A and B axes are not in use.
- **When point "R" command position (5) is lower than Z axis command position (7), tool moves to Z-axis commanded position, and operation (7) is not performed.**

Tool change motion

Tool change motion differs depending on tool type set on [MAGAZINE TOOL] screen.
(When large tool is not set on [MAGAZINE TOOL])
The following sequence is performed:
(1) Pot raises.
(2) Magazine swivels.
(3) Pot lowers.
(4) Arm swivels at high speed.

(When large tool is set on [MAGAZINE TOOL])

Tool change from standard tool to standard tool, and large tool to large tool.
The following sequence is performed:
(1) Pot raises.
(2) Magazine swivels.
(3) Pot lowers.
(4) Arm swivels.
(5) Pot raises.
When pot is already at upper limit before tool change motion commences, sequence starts from (2).

When ((4) arm swivels) changing from large tool to large tool, arm swivels at low speed.
When ((4) arm swivels) changing from standard tool to standard tool, arm swivels at high speed.

Tool change from large tool to standard tool or vice versa.

The following sequence is performed:
(1) Pot raises.
(2) Magazine swivels. (Empty pot is indexed.)
(3) Pot lowers.
(4) Arm swivels.
(5) Pot raises.
(6) Magazine swivels. (Specified pot is indexed.)
(7) Pot lowers.
(8) Arm swivels.
(9) Pot raises.
When pot is already at upper or lower limit before tool change motion commences, sequence starts from (3) or (4), respectively.

For operation (2), empty pot is indexed appropriately for large tool and standard tool.

Arm swivels ((4) or (8)) at low speed when large tool is to be changed, and at high speed when standard tool is to be changed.

Notes

1) If empty pot for large tool is not available when change from large tool to standard tool is attempted, [*NO EMPTY POT] alarm appears.
2) If empty pot for standard tool is not available when change from standard tool to large tool is attempted, [*NO EMPTY POT] alarm appears.

Next tool preparation

Next tool preparation is performed after the arm has swiveled or the pot has risen after the arm swivels in the ATC sequence.
When ATC is not performed, only next tool preparation is performed.
Next tool preparation is not performed when the next tool is already indexed.

The next tool preparation sequence differs depending on the type of tool set on the magazine tool screen.

When large tool is not set on **[MAGAZINE TOOL].**
(1) Pot rises
(2) Magazine swivels
(3) Pot lowers
Next tool preparation ends when the above sequence is completed.
The sequence starts from (2) when the pot is already at the upper limit before next tool preparation commences.

When large tool is set on **[MAGAZINE TOOL].**
Both spindle tool and next tool are standard or large
(1) Pot rises
(2) Magazine swivels
(3) Pot lowers
Next tool preparation ends when the above sequence is completed.
The sequence starts from (2) when the pot is already at the upper limit before next tool preparation commences.

Spindle tool is standard and next tool is large or vice versa
(1) Pot rises
(2) Magazine swivels (empty pot is indexed)
(3) Pot lowers
Next tool preparation ends when the above sequence is completed.
The sequence starts from (2) when the pot is already at the upper limit before next tool preparation commences.

Note 1) When the next tool is changed from a large tool to a standard tool and there is no empty pot for a large tool, the [NO EMPTY POT] error occurs.
Note 2) When the next tool is changed from a standard tool to a large tool and there is no empty pot for a standard tool, the [NO EMPTY POT] error occurs.

12

12.3.7 Workpiece counter specification (M211~M214)

When M211~M214 are specified to the counter 1~4 respectively, and M211 ~ M214 are commanded in the memory operation, the commanded counter counts up by specified step at the execution of M02 or M30.
The counter specification is cancelled when the power is turned ON, the **[RESET]** key is pressed, M02 or M30 is executed, the operation is reset or M221~M224 (counter cancel) is commanded.
(Ex.) When the counter 1 counts 1 and the counter 2 counts 2, execution of M211 and M212 by the operation program makes the counter 1 count up 1, the counter 2 count up 2.
(Note) M211~ M214 can be commanded during MDI operation.

12.3.8 Workpiece counter cancel (M221~M224)

M221~M224 command cancellation of the counter 1~ 4 respectively.
When M221~M224 are commanded in the memory operation or MDI operation, relative counter specificaton is cancelled.

12.3.8.1 Tool life counter

Execute command M231 to interrupt counting the service life of the tool attached to the spindle.
Execute command M230 to resume counting.

12.3.9 Automatic corner deceleration (M232, M233)

- This function is valid for commands that perform continuous XY-axes cutting movement.
- Automatic corner deceleration does not apply during tool compensation.

12.3.10 Tool breakage detection (M120 and M121)

When the M120 command is given, the input signal TOUCH (touch sensor) is checked.
If this is ON, operation is ended. If this is OFF, a tool breakage error is generated and the TOOL (tool error) signal is output.
M121 command, if this is OFF for finishing the check. If this is ON, a tool breakage error is generated.

12.3.11 Tool breakage detection (M200 and M201)

Operation procedures are described in Option, Tool Breakage Detection Unit in the Instruction Manual.

12

12.3.12 Tap time constant selection (M241 to 250)

- When the M251 command is given, tap time stay constant when tapping of Z-axis speed .
- When the M251 command is given, tap acceleration stay constant when tapping of Z-axis speed.

12.3.13 Pallet related M codes (M410, M411, M430, and M431)

This function is available for QT mounted machine.

- When command M410 is executed, the Z-axis is returned to its origin and then the C-axis is indexed to -180° (pallet 2 is positioned outside).
- When command M411 is executed, the Z-axis is returned to its origin and then the C-axis is indexed to 0° (pallet 1 is positioned outside).
- When command M430 is executed, the C-axis unclamped by override.
- When command M431 is executed, the C-axis clamped by override.

12.3.14 Unclamping and clamping C axis (M430 and M431)

This function is available for QT mounted machine.

- When M430 is commanded, the C axis is unclamped. Unclamping and clamping C axis is not automatically controlled afterward.
- When M431 is commanded, the C axis is clamped. Unclamping and clamping C axis is automatically controlled afterward.

This command is valid when C is set to operate using the mashine paramater and the clamp merchanism is set to [1:Type2]or[2:Type3].
An alarm will occur in other cases.

12.3.15 Unclamping and clamping B axis (M440 and M441)

- When M440 is commanded, the B axis is unclamped. Unclamping and clamping B axis is not automatically controlled afterward.
- When M441 is commanded, the B axis is clamped. Unclamping and clamping B axis is automatically controlled afterward.
  This command is valid when B is set to operate using the mashine paramater and the clamp merchanism is set to [1:Type2]or[2:Type3].
  An alarm will occur in other cases.

12.3.16 Unclamping and clamping A axis (M442 and M443)

- When M442 is commanded, the A axis is unclamped. Unclamping and clamping A axis is not automatically controlled afterward.
- When M443 is commanded, the A axis is clamped. Unclamping and clamping A axis is automatically controlled afterward.

  This command is valid when A is set to operate using the mashine paramater and the clamp merchanism is set to [1:Type2]or[2:Type3].
  An alarm will occur in other cases.

12.3.17 One-shot output (M450, M451, M455, and M456)

- M450 and M451 commands proceed to the next block after output time has passed and the signal has turned off.
- M455 and M456 commands proceed to the next block without waiting until the signal turns off.
- The signal output time is set by user parameter 1.

12.3.18 Waiting until response is given (M460 to M469)

For example
- M460 command waits until M460 signal turns on.
- M461 command waits until M460 signal turns off
- M462 command waits until M462 signal turns on.
- M463 command waits until M462 signal turns off.
- M464 command waits until M464 signal turns on.
- M465 command waits until M464 signal turns off.
- M466 command waits until M466 signal turns on.
- M467 command waits until M466 signal turns off.
- M468 command waits until M468 signal turns on.
- M469 command waits until M468 signal turns off.

The maximum waiting time is set by MAXIMUM TIME OF EXT SIGNAL of user parameter 1.
If the actual time has exceeded the preset time, an alarm occurs.

12.3.19 Magazine rotate speed (M435 to M437)

Arm style of ATC machine is changeable the program for magazine rotate speed.
- M435 : Magazine rotate maximum speed.
- M436 : Magazine rotate speed 1.
- M437 : Magazine rotate speed 2.

Return to the maximum speed to follow the operation below.
Command M435.
Finish the program.
Reset the driving.
Tool changing.

M436, M437 rotate speed:
Set “Magazine rotate speed 1(M436).” “Magazine rotate speed 2 (M437).” in the user parameter.

12.3.20 Magazine rotate to tool setting position (M501 to M599)

- M5** : Rotate the magazine to set pot position that ** specified the tool number of setting position

12.3.21 Positioning finished check distance (M270 to M279)

- M270 : Positioning finished check distance, positioning finished angle non-specific.
- M271 : User parameter of “M271Positioning finished check distance” and “M271 Positioning finished angle” value to be the imposition width for positioning. X, Y, Z axes use the value of “Positioning finished check distance”. A, B, C axes use the value of “Positioning finished angle”.
- M272~M279 : As same as M271.

Return to the “Positioning finished check distance”, “Positioning finished angle non-specific” to follow the operation below.
Command M270.
Finish the program.
Reset the driving.

12

CHAPTER 13

OPTION

13.1 Programming Precautions When Using Rotation Axis (index table)

13.1 Programming Precautions When Using Rotation Axis (index table)

When using the QT table on the TC-31A, 32A, and R2A with the rotation axis installed, be sure to place the rotation axis positioning command before the cutting command in the program file.

6A3468001
0401(1)