TC-31A
TC-31AN
TC-22A
TC-S2A
TC-32A
TC-R2A
TC-20A
TC-S2B

NC-PROGRAMMING MANUAL

Please read this manual carefully before starting operation.

brother®

This manual describes the NC-Programming of the TC-31A, TC-31AN, TC-22A, TC-S2A, TC-32A, TC-R2A, TC-20A and TC-S2B.
(The description of TC-31AN is same as that of TC-31A.)
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.

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.

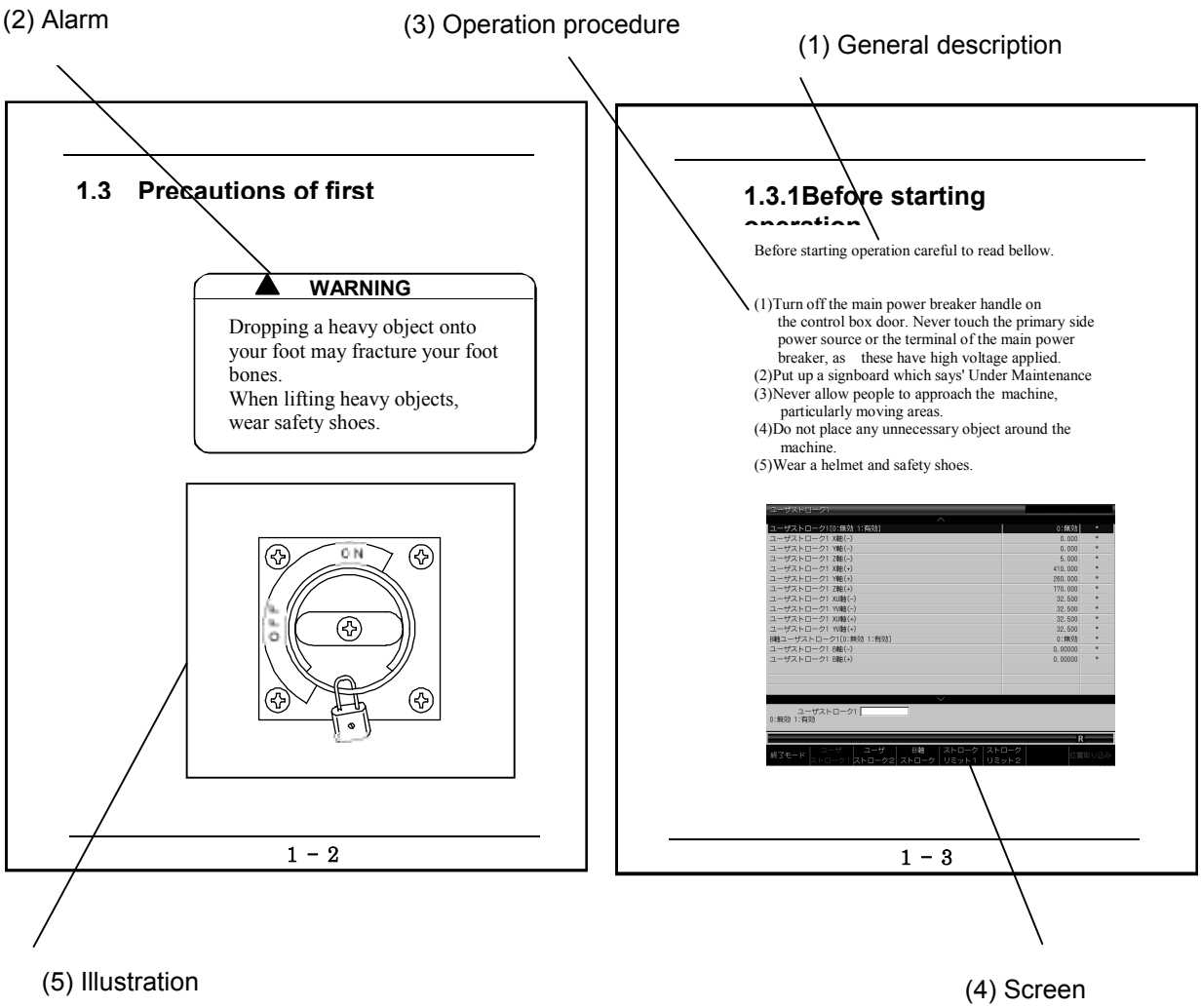
(2) **Alarm** Is a alert given against a danger which may cause serious damage or death to human being or may damage the machine. The hazards are explained in this order:
degree of danger,
subject of danger,
expected damage,
preventive measure,

(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 ia a blank.)

QUICK INDEX

CHAPTER 1

PROGRAM COMPOSITION

1.1 Types and composition of program

1

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

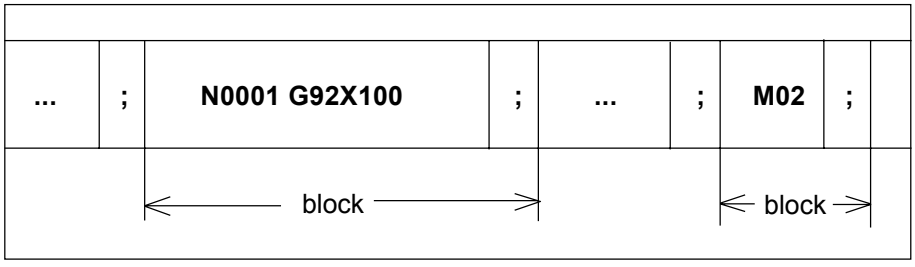
A subprogram is used by calling it from the main program or other subprograms.
Command M99 to finish the subprogram.

Sub program

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 ";".

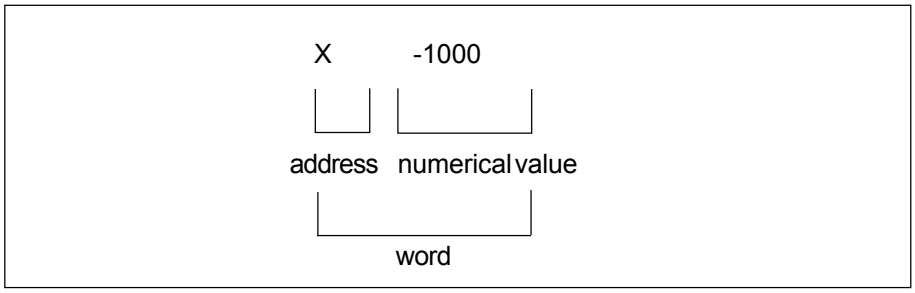


(Note 1) **The end of block code**
ISO code : [LF] 0A(hexadecimal)
EIA code : [CR] 80(hexadecimal)

(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.)



(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	1inch
	Rotation axis	1deg	1deg
1.	Feed axis	1mm	1inch
	Rotation axis	1deg	1deg

Command type 2 (Minimum)

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

(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

1.5 Sequence number

A sequence number (1~9999) can be used following the address N for each block.
There are such functions as a sequence search and restart.

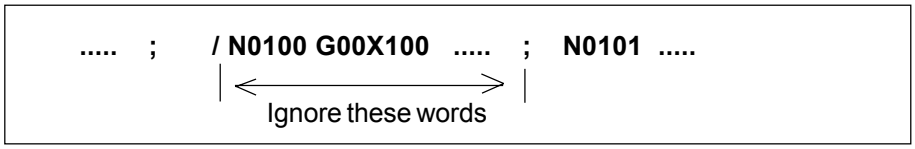
Command format	**N****;**

i) A sequence number is used following the address N.
ii) A sequence number can be specified with up to 4-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.



(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 sequence number search function, all the blocks with a slash code can be searched regardless of the [BLOCK SKIP] key ON or OFF status.
(Note 3) 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.

(.....)

control out code comment control in code

(Ex.) Ｎ１０００ Ｇ００Ｘ２００ （ＰＲＯ－１）；

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

1

(This page is a blank.)

CHAPTER 2

COORDINATE COMMAND

2.1 Coordinate system and coordinate value

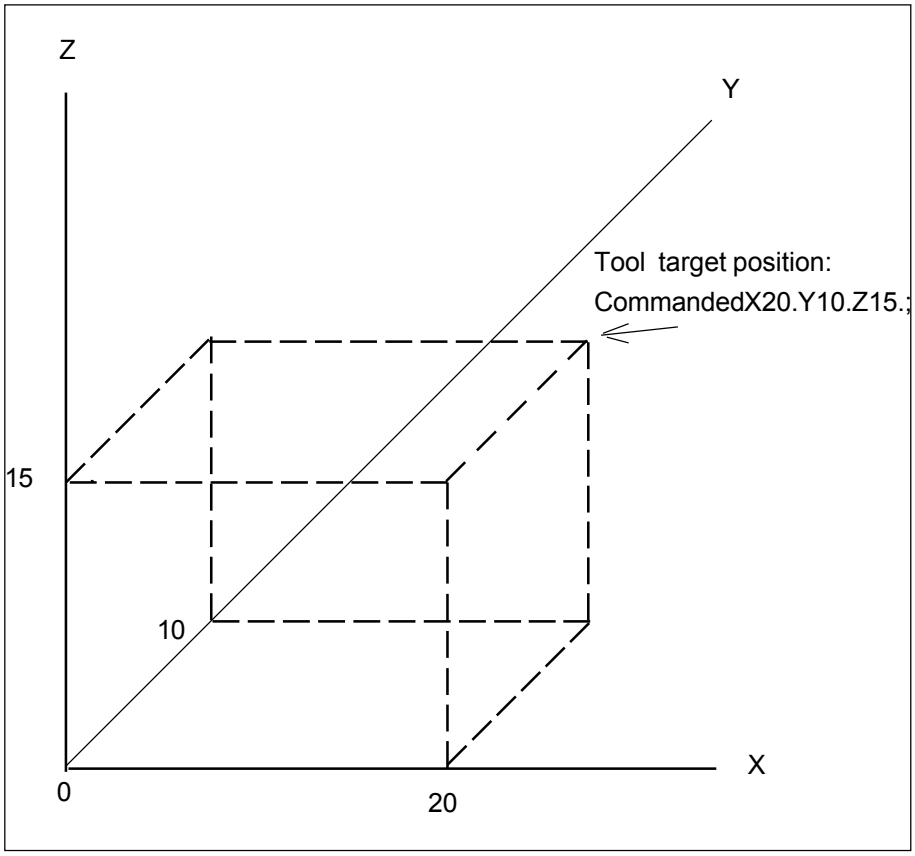
Coordinate values should be set in one coordinate system to specify a tool movement.

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).



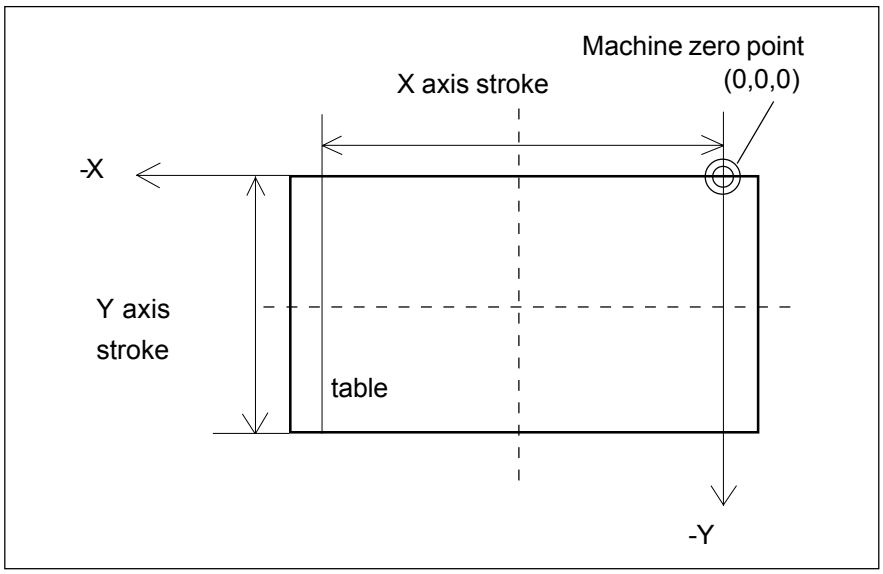
2.2 Machine zero point and machine coordinate system

(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.



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 .)

3

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.

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
	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	
(Note1)	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)	
(Note2)	G40*	Tool dia offset cancel	Modal
	G41	Tool dia offset left	
	G42	Tool dia offset right	
0303-2.tbl			

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

Group	G code	Contents	Modal/ One-shot
(Note1)	G43	Tool length offset +	Modal
	G44	Tool length offset -	
	G49*	Tool length offset cancel	
	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	
0304-2			

The G codes with * mark indicade the modal status when the power is turned ON.
(Note1) Details of tool length offset are described in " Chapter 4 ".

Group	G cord	Contents	Modal / one-shot
	G90*	Absolute command	Modal
	G91	Incremental command	
	G92	Working coordinate system setting	One-shot
	G94	Feedrate 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)	
TR2A/3042 TBL			

Group	G cord	Contents	Modal/ One-shot
	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 boring cycle)	
	G100	Non-stop Automatic Tool Change	One-shot
MTR2A/3043 TBL			

Group	G cord	Contents	Modal / one-shot
	G120	Positing 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)	
TR2A/3044 TBL			

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

3.2 Positioning (G00)

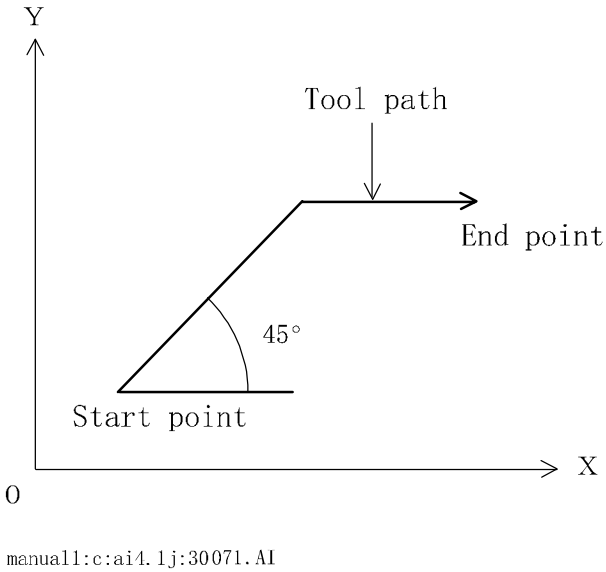
A tool moves from its current position to the end point at the rapid traverse rate in each axis direction independently. Therefore, a tool path is not always a linear line.

Command fromat

```
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)



(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)

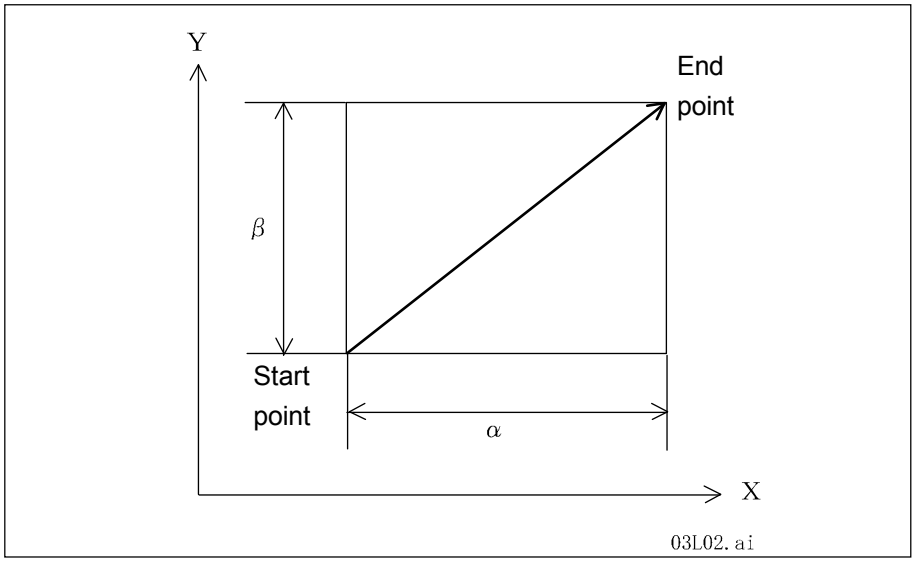
Linear interpolation moves a tool linearly from the current position to the target position at the specified feedrate.

Command fromat

```
G01 X_Y_F_ ;
```

3

Up to two of the X,Y,Z,A,B or C axes can be controlled simultaneously.
Linear interpolation can not be operated between the additional axes.
However, the X ,Y,and Z axes can be operated simultaneously.
When the additional axis is commanded and the optional additional axis is not installed, an alarm will occur.
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 °/min.



(Note 1) Feedrate along each axis is as follows:

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

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

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

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

(Note2)

The example below shows linear interpolation of linear axis α and rotation axis β as follows.

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

Feedrate along X axis: $Fx = \frac{\sqrt{\alpha^2+\beta^2}}{f}$

Feedrate along B axis: $Fb = \frac{\beta}{Fx}$

3.3.1 Chamfering to desired angle and cornering C

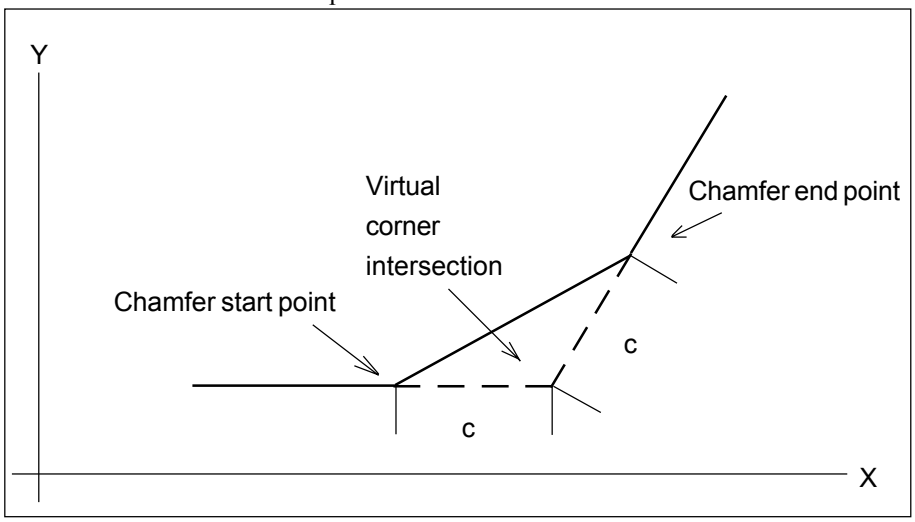
Chamfering to the desired angle or rounding can be performed between interpolation commands.

Chamfering

Command fromat

G01 X_Y_, C_ ;

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



(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.

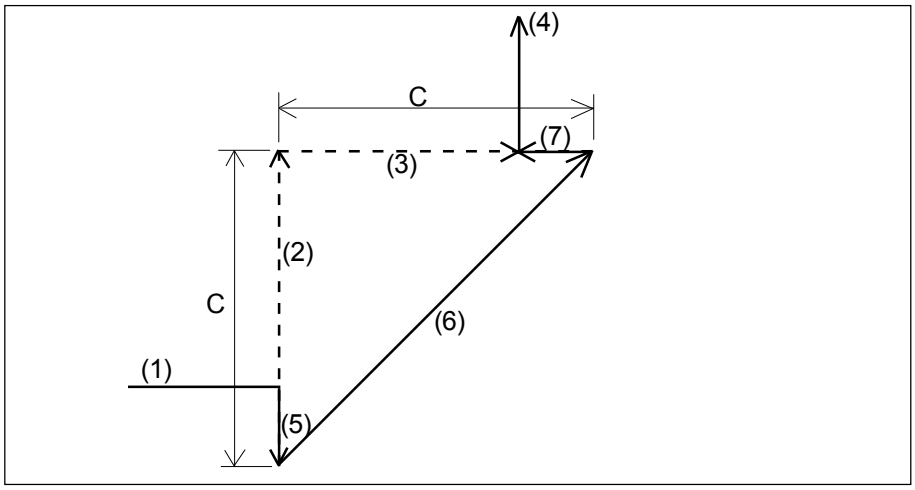
(2) The inserted block belongs to the corner chamfering command block. Thus, if the feed rate differs from the corner chamfering command block and the subsequent block , the inserted block moves at the feed rate of the corner chamfering command block. Further, the program does not stop before the inserted 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.

(4) The corner chamfering command can be commanded only on the XY (2-D)plane.

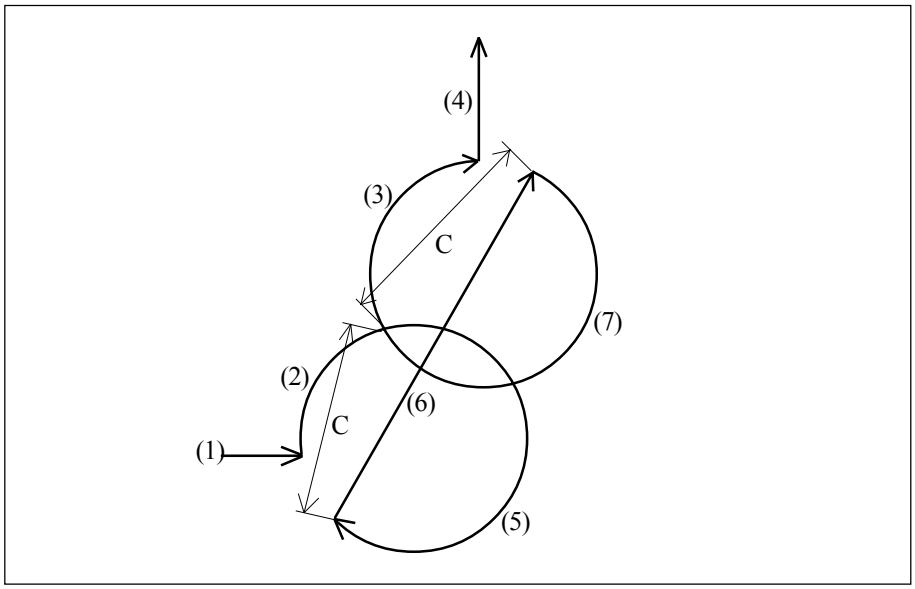
(5) When the chamfering amount is longer than the chamfering command block and feeding quantity of the subsequent block, set extended point from each blocks as "chamfer start point" and "chamfer end point".
(But the path becomes abnormal in cutter compensation mode.
Do not specify a chamfering amount longer than travel amount in cutter compensation mode.)

Example.1: Liner cutting



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



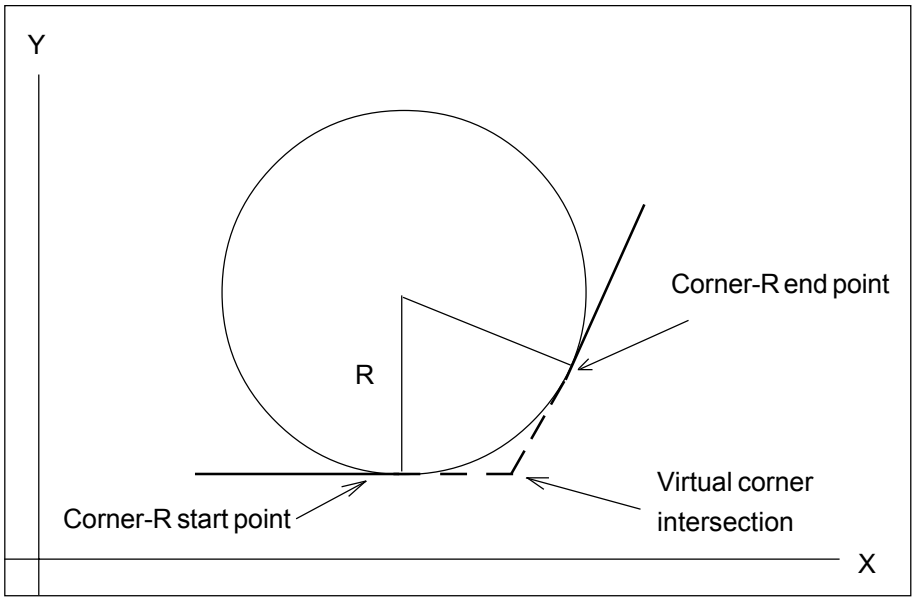
When set the programmed path to(1.2.3.4.)and the block C as (2), operate to 1-5-6-7-4.

Cornering

Command fromat

```
G01  X_Y_,  R_ ;
```

R : Radius of cornering



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

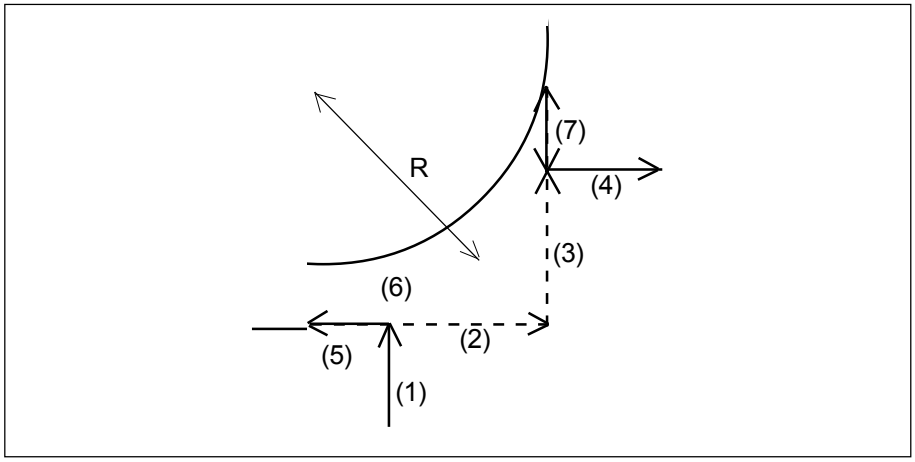
(2) The inserted block belongs to the cornering command block. Thus, if the feed rate differs from the cornering command block and the subsequent block , the inserted block moves at the feed rate of the cornering command block. Further, the program does not stop before the inserted block occurs even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

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

(4) The corner-R command can be commanded only on the XY (2-D)plane.

(5) 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".
(But the path becomes abnormal in cutter compensation mode. Do not specify diameter longer than travel amount in cutter compensation mode.)

Example.1: Liner cutting



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

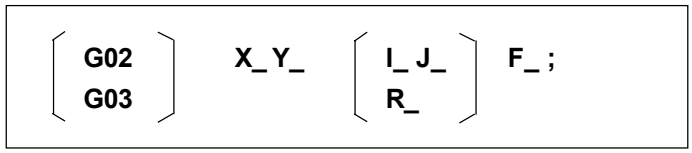
3.4 Circular/helical interpolation (G02, G03)

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 XY Circular interpolation

Command fromat



The commands are given in the following format:

Rotation direction		G 02	Clockwise(CW).
		G 03	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 Z axis of the X-Y plane.

3.4.1.2 XZ Circular interpolation

Command fromat

[G102 / G103]	X_ Z_	[I_ K_ / R_]	F_ ;

The commands are given in the following format:

Rotation direction		G102	Clockwise(CW).
		G103	Counterclockwise(CCW).
End point	G90mode	X,Z	End point in the working coordinate system.
	G91mode	X	Distance from the start point to the end point in the X 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.
		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 Y axis of the X-Z plane.

(Note 1)

In contrast to the XY arc case, an error occurs when the diameter compensation command (G40, G41, G42) or coordinate rotation command (G68, G69, G168) is used, and the machine stops operation.

3.4.1.3 YZ Circular interpolation

Command fromat

{ G202 / G203 }	Y_Z_	{ J_ K_ / R_ }	F_ ;

The commands are given in the following format:

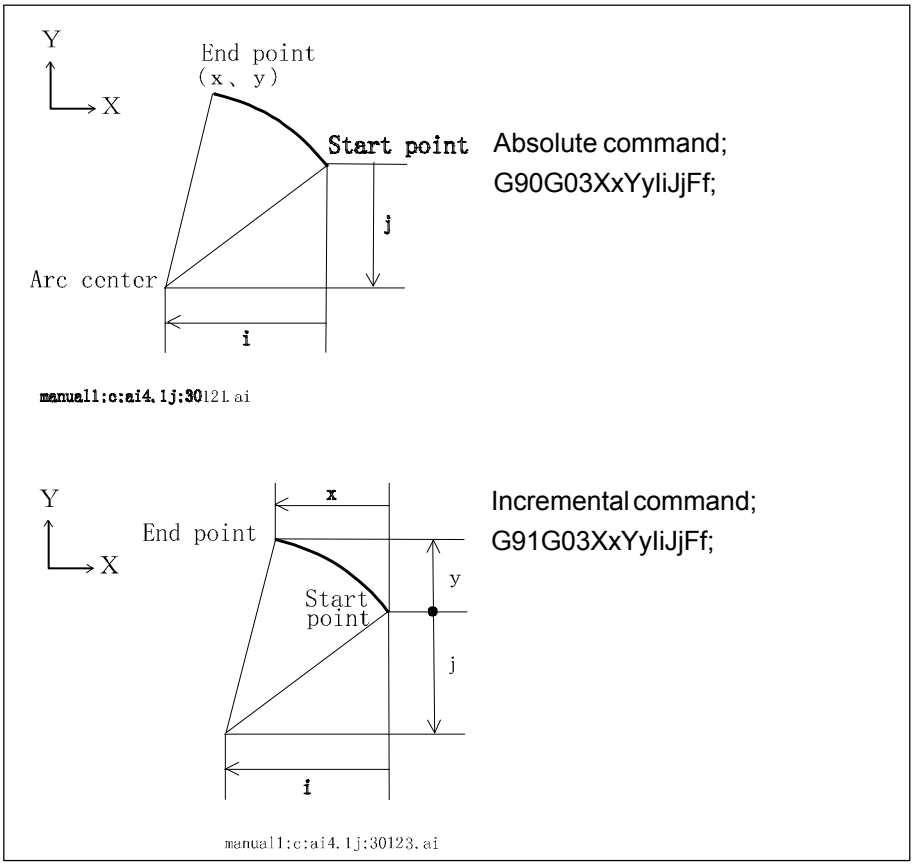
Rotation direction		G202	Clockwise(CW).
		G203	Counterclockwise(CCW).
End point	G90mode	Y,Z	End point in the working coordinate system.
	G91mode	Y	Distance from the start point to the end point in the Y direction.
		Z	Distance from the start point to the end point in the Z direction.
Distance between start point and arc center		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 X axis of the Y-Z plane.

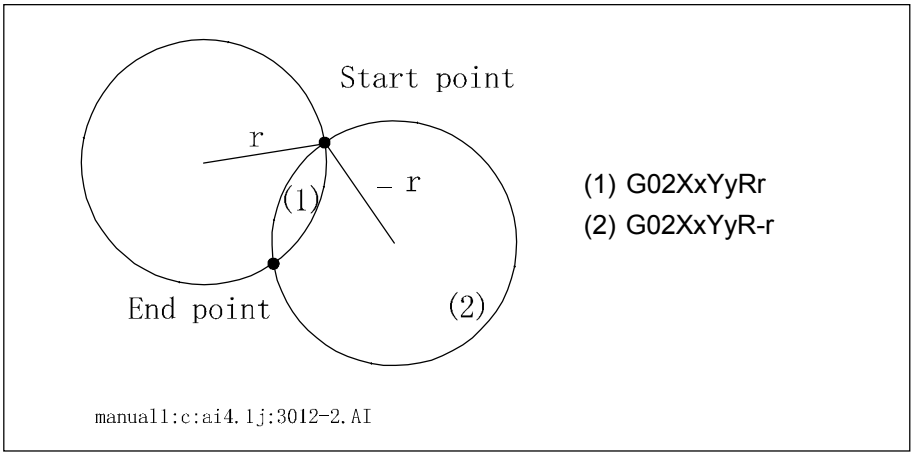
(Note 1)

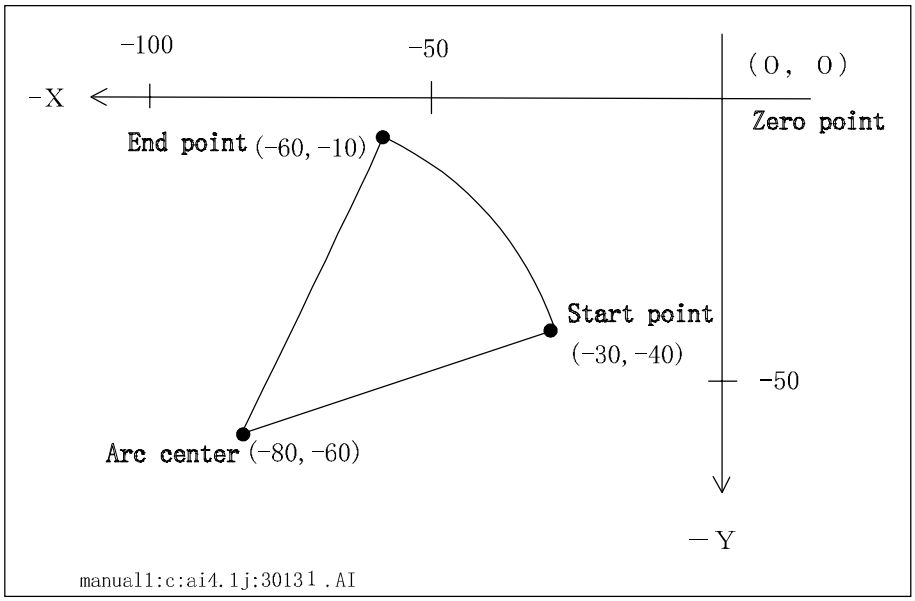
In contrast to the XY arc case, an error occurs when the diameter compensation command (G40, G41, G42) or coordinate rotation command (G68, G69, G168) is used, and the machine stops operation.

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.



Insted 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 algebrac mark "-" before the value for the radius.



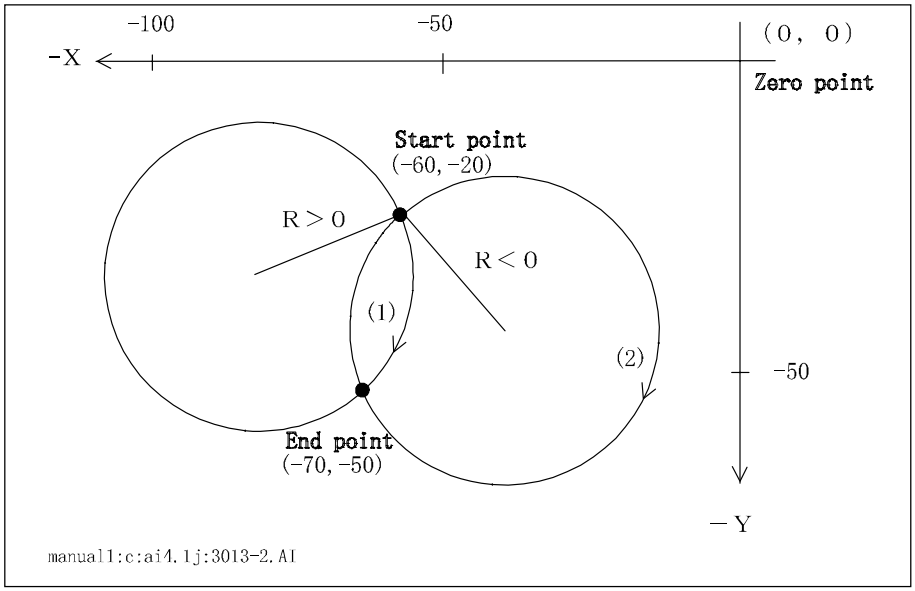


Absolute command;

G03X-60. Y-10. I-50. J-20. F1000 ;

Incremental command;

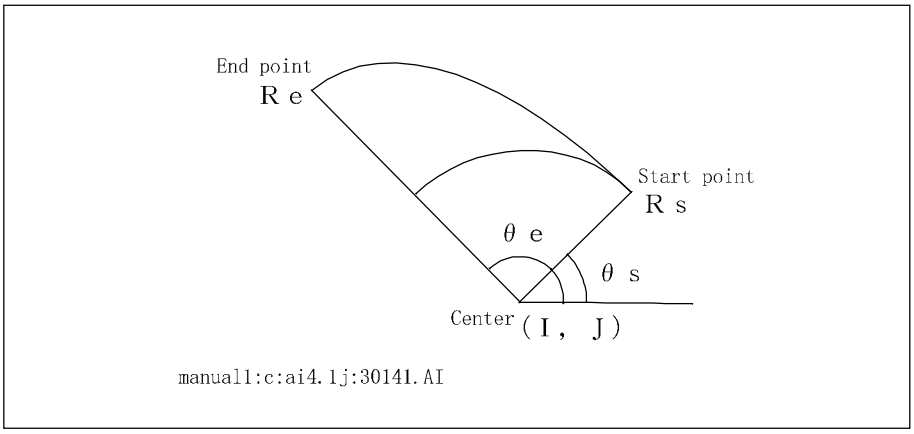
G03X-30. Y30. I-50. J-20. F1000 ;

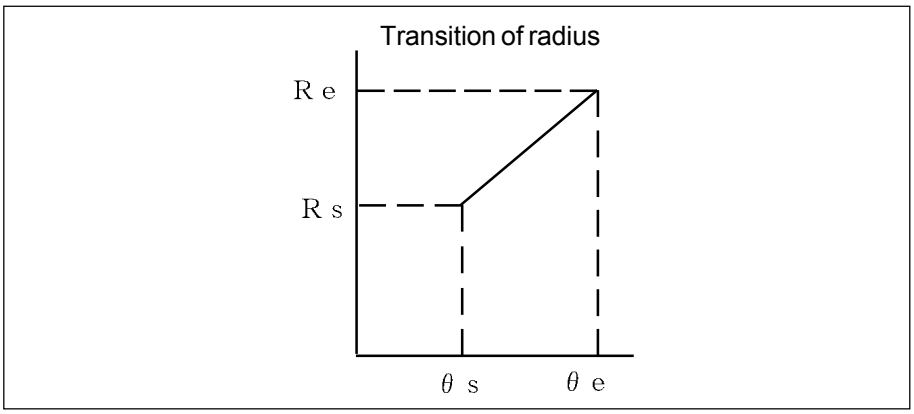


(1) G02X-70. Y-50. R25. F1000 ;

(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) a 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, a circle of zero degree is assumed and the tool does not move.
(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.





(Note 6) If the ending radius is extremel 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.

(Note) Helical interpolation is optional function.
If helical interpolation is commanded without using the optional function, an alarm is generated.

3.4.2 Helical interpolation

Putting the Z axis command in the circular arc block permits a helical cutting.

Command fromat

G02 / G03	X_ Y_ Z_	I_ J_ / R_	F_ ;

The F code commands the feedrate in the X-Y plane.

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

The feedrare in the Z axis direction is determined by the values of "feedrate" in the X, Y axes direction, "end point X", "end point Y" and "end point Z". It can be calculated as follows:

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

Fz : Z axis feedrate
L : Z travel distance
R : Radius(start point - center)
θ : Angle
F : Commanded feedrate(XY axes)

Ex.)

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

$$Fz = (180 \times 2 \times 500) / (\pi \times 10 \times 360)$$
$$\fallingdotseq 15.9 (mm/min)$$

If the Z axis feedrate is larger than the Z MAXIMUM CUTTING SPEED or Z MAXIMUM RAPID SPEED set by the machine parameter , an alarm is generated.

When tool dia offset command is given, an offset is applied to the circular arc (XY axes).

3.5 Dwell (G04)

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

Command fromat

```
G04 P_ ;
```

or

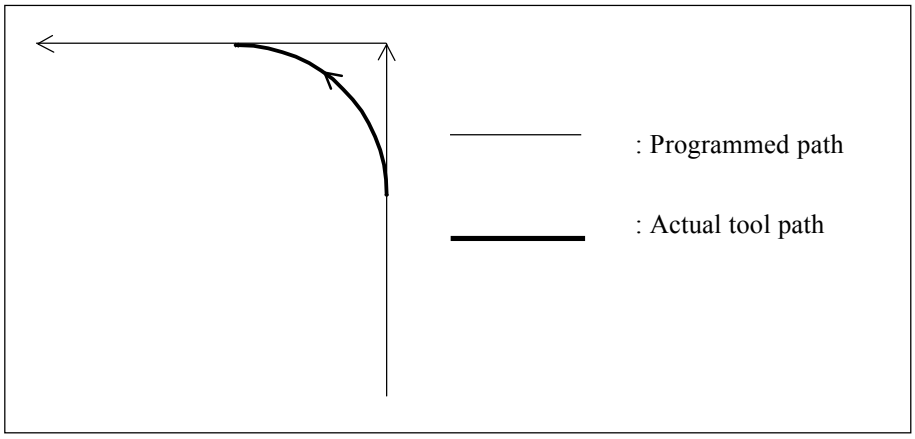
```
G04 X_ ;
```

P,X : Dwelling time (sec)

3.6 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.

3



(1) Exact stop check (G09)

Command fromat | **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 fromat | **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 fromat

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.

3.7 Programmable data input (G10)

(1) Input of workpiece zero position

Command fromat

```
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) Workpiece zero position.....Refer to “Data bank” in the operation manual.

(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 “Data bank” in the operation manual.

(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 fromat

```
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) and the preset value in inclremental mode (G91).
Setting range +/- 99.999 mm +/- 9.9999 inch

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

Command fromat

```
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 fromat

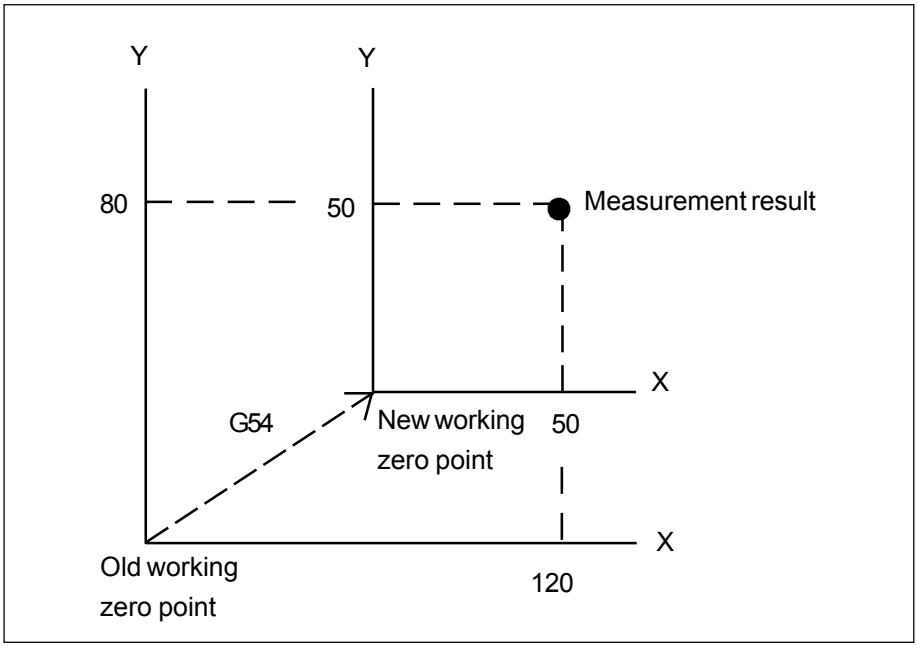
```
G10L98 Pn X_ Y_ Z_ Q_ ;
```

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. ;
.
.



(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.

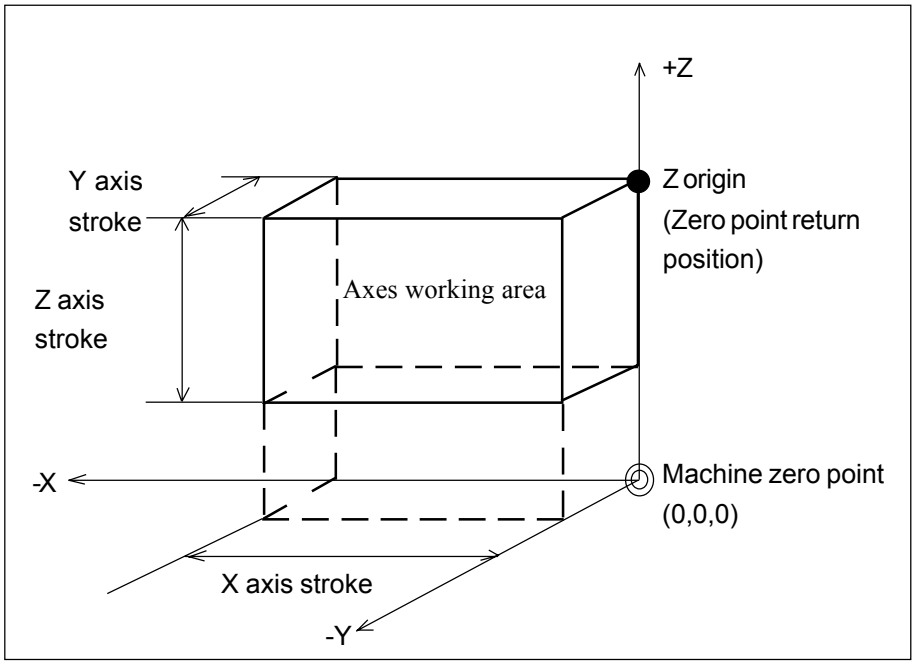
3.8 Soft limit

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

3.8.1 Stroke

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



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

3.8.2 Stroke limit

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

3.8.3 Programmable stroke limit (G22)

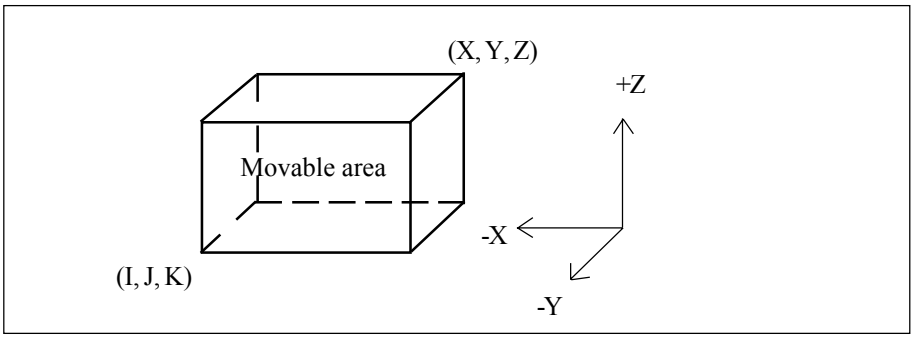
The allowable area of the tool motions is commanded by the program.

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.



(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.

3.9 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 coodinate 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 preivous 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.

3.10 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.

3.11 Return to the 2nd/3rd/4th 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

This command moves the axes to the 2nd, 3rd or 4th 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, 3rd and 4th 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.

3.12 Selection of machine coodinate 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.

3.13 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

3.14 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.

3.15 Coordinate rotation function (G68,G69)

Command format

```
G68 X__Y__R__;
```

X,Y,Z: Rotation center coordinate value

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

R : Rotation angle

Conforms to G90/G91 ; the counterclockwise direction is considered the positive direction.

When this setting is omitted,an alarm will occur.

Commanded range : -360.000~360.000

1) The tool diameter is compensated for after coordinate rotation.
2) An alarm will occur when commanded in the MDI mode.
3) An alarm will occur when following commands is operated during coordinate rotation.
   G28~G30,G36,G131,G92,G120,G121
4) Coordinate rotation doesn't apply to G53.
5) An alarm will occur when G01X_Z_; G01 Y_Z_; is commanded during c oordinate rotation.
6) The rotation center coordinate value is converted to the machine coordinate value and stored inside the NC unit.
   Thus, even if the coordinate system is changed after coordinate rotation has been commanded, the rotation centeer coordinate value will not change.

Command format

```
G69 ;
```

Coordinate rotation is canceled

Coordinate rotation can be canceled even by the following operations:
CommandM02(M30).
Reset operation

3.16 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 slection 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 whitch the block has shifted from G69 to G168 (or G68) is considered the center.

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

3.17 Absolute command and incremental command (G90, G91)

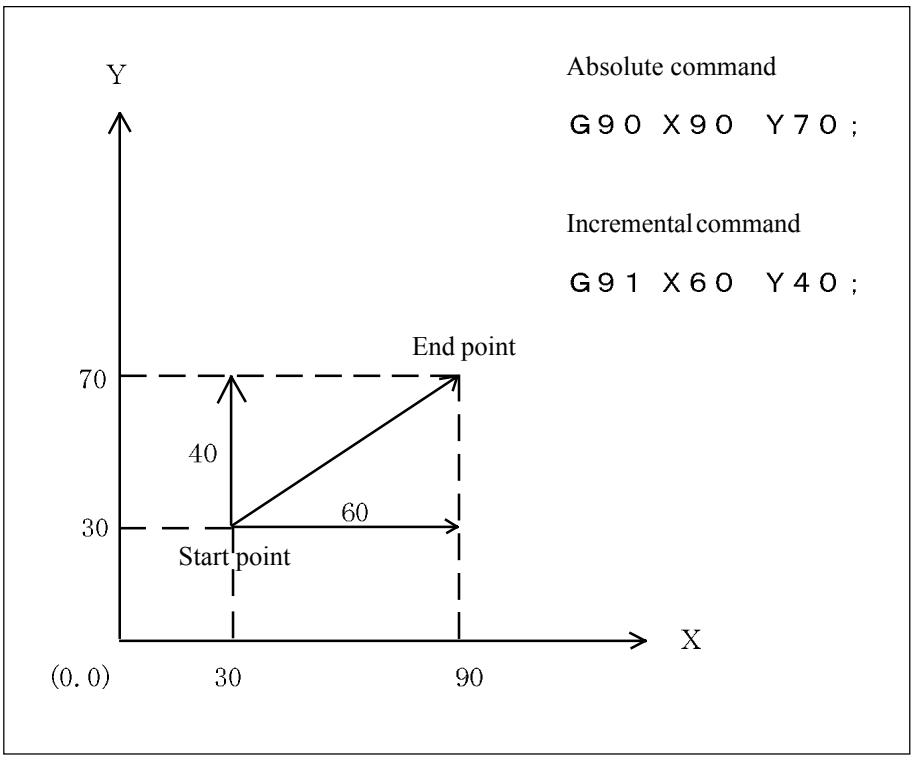
The axis movement amount can be specified by either the absolute command or the incremental command.

(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.

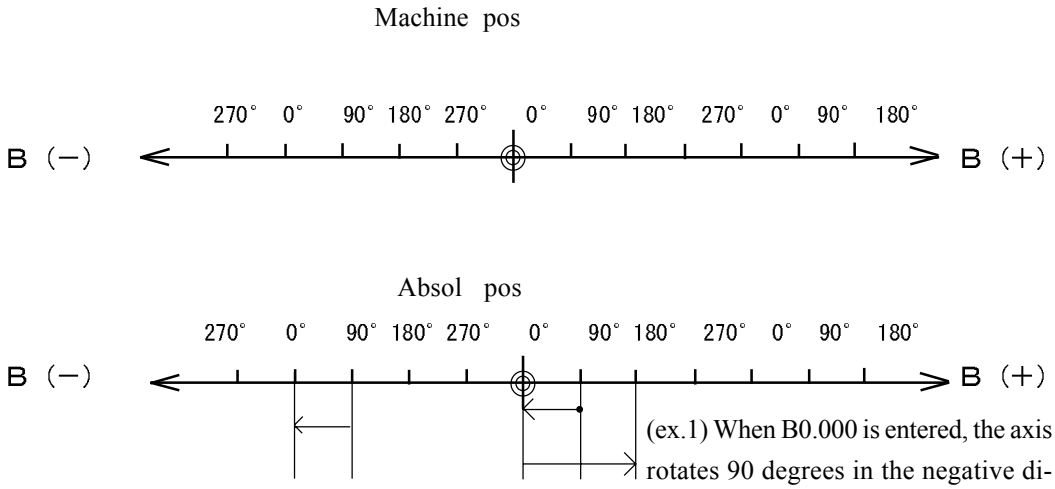


(3) When additional axis is commanded

1. Absolute command (e.g., Baxis)

- 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 deg.), 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



(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 carrent position before travering (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.

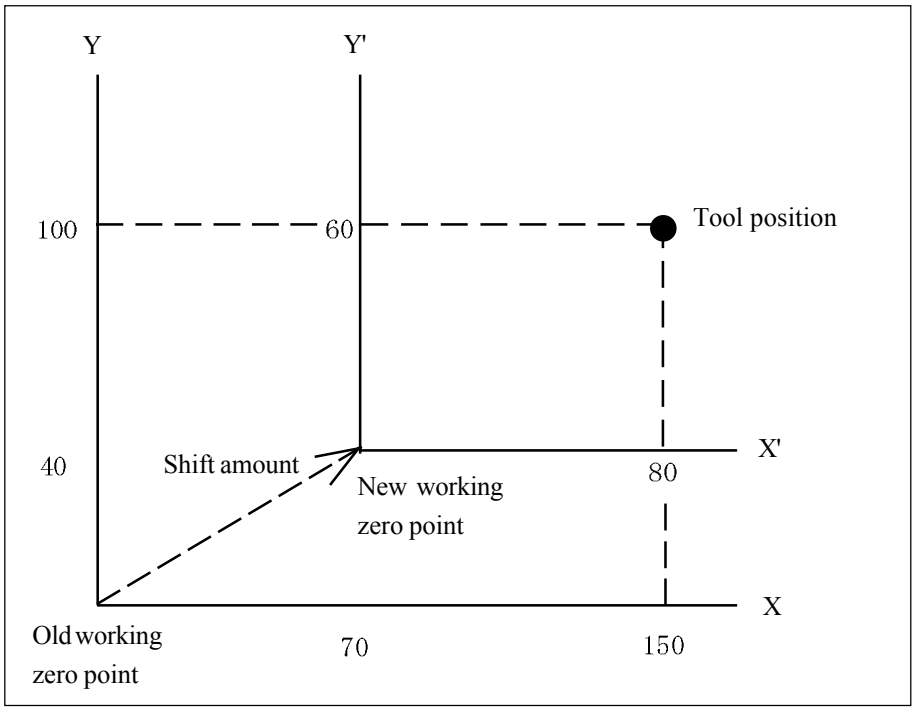
3.18 Change of working coordinate system (G92)

Change of working 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 sytem so that the current tool position becomes to the commanded coordinate values.

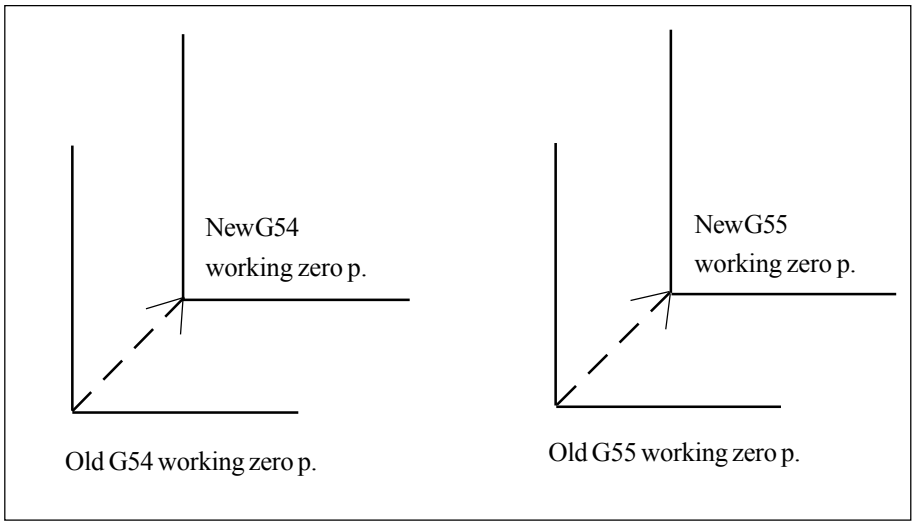


Ex.) The absolute coordinate of the tool position changes to (80, 60) from the current position (150, 100) as commanded "G92 X80. Y60.;"

(Note 1) The commanded coordinate values are always absolute regardless of G90 and G91.

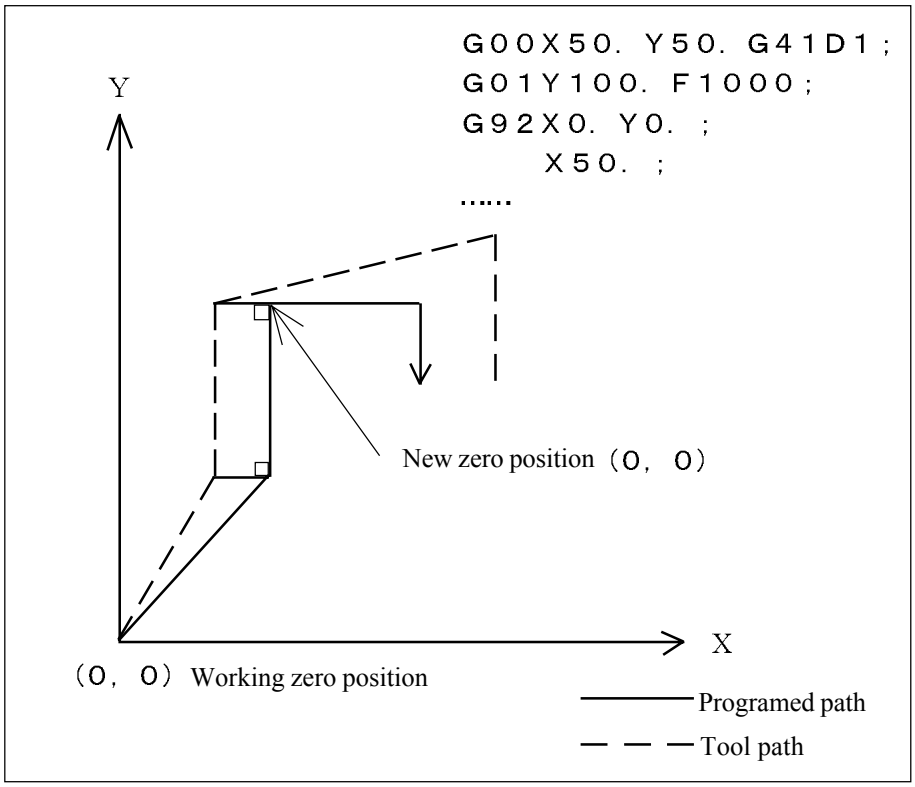
(Note 2) The working coordinate values of the uncommanded axes do not change.

(Note 3) The current working zero position shifts when G92 is executed, and other working zero positions also shift the same amount accordingly.

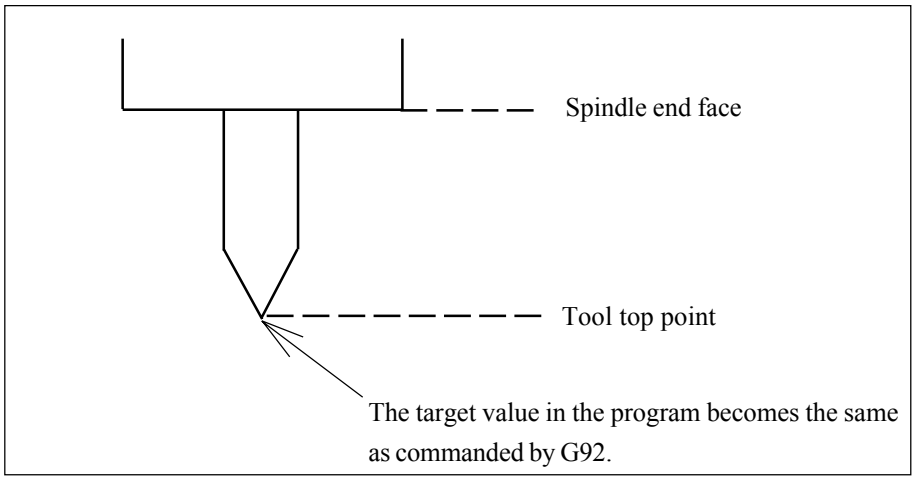


In the above figures, G92 is commanded in the coordinate system of G54.
When the working zero position of G54 shifts, the other working zero positions of G55 through G59 also shift the same amount as G54.

(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.



(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.



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

3.19 Measurement feed(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

```
G131 X__ Y__ Z__ F__ ;
G132
```

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 G132, an alarm does not occur.

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 posision during a machine lock state.
Note 4: When the G131/G132 are used in a restarting program, the tool returns to the target position.
Note 5: When the detection signal is already ON, the tool stops at the current position when attempting to execute G131/G132.

3.20 Change of tap twisting direction (G133,G134)

Command format

G134 G133	Z__	I__ J__	S__ ;

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

Z : Z axis target position.
Comforms 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.

3.21 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 “0” for the command value of the axis.
   G92 is used
   M02 (M30) is used.

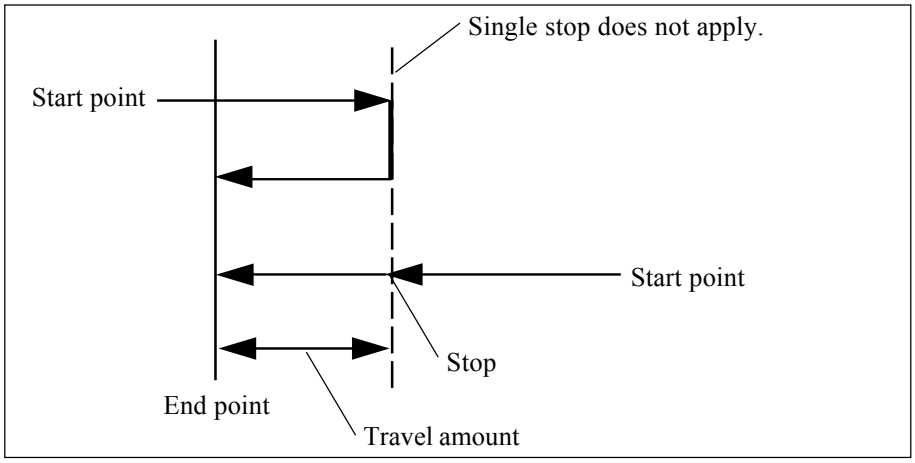
3.22 Single direction positioning function (G60)

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



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.

3.23 G code priority

How to see the table

G code / Command / G code	G00 ~ G03
G00~G03 G102,G103 G202,G203	(3)
	(1)
G04	(4)
	(1)

In the above table,(4) is the case when G01 (G00 to G03) group and G04 are commanded in the same block. The modal for G01 is updated and G04 is executed.
(1) is the case when G04 is commanded while G01 group is modal.
G04 is executed correctly.

(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 G00 or G01, but an error occurs when circle arc is commanded.
(7) G22 is executed when G22 is commanded and the model for G00 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 G00 group is executed, G80 group is canceled (G80).
(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 group is cancelled (G80).
(16) One shot is executed and the model is updated, but an error occurs when G54P is used.
(17) Both are executed when used simultaneously with G80.
An error occurs when used simultaneously with G54P.
(18) An error occurs when G54P is used.

G code priority list (1)

Upper step : When commanded in the same block
Lower step : When commanded in another block

G code / Command / G code	G00, G03, G102, G103, G202, G203	G22, G23	G40 ~G42	G43, G44, G49	G54 ~G59	G54.1	G61, G64	G66, G67	G68, G69, G168	G73 ~G89	G90, G91	G94	G98, G99	G177 ~G189
G00~G03 G102,G103 G202,G203	(3)	(7)	(8)	(8)	(1)	(1)	(1)	(2)	(4)	(10)	(1)	(1)	(1)	(10)
	(1)	(1)	(13)	(1)	(1)	(1)	(1)	(1)	(13)	(15)	(1)	(1)	(1)	(15)
G04	(4)	(2)	(2)	(2)	(16)	(2)	(1)	(2)	(14)	(11)	(1)	(1)	(1)	(2)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G09	(1)	(1)	(1)	(1)	(1)	(1)	(4)	(2)	(1)	(1)	(1)	(1)	(1)	(1)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G10	(4)	(2)	(2)	(2)	(16)	(2)	(1)	(2)	(14)	(11)	(1)	(1)	(1)	(2)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G22,G23	(7)	(3)	(2)	(2)	(1)	(1)	(1)	(2)	(14)	(11)	(1)	(1)	(1)	(2)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G28~G30	(4)	(2)	(2)	(1)	(16)	(2)	(1)	(2)	(14)	(11)	(1)	(1)	(1)	(2)
	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)
G36~G39	(5)	(2)	(2)	(1)	(16)	(2)	(1)	(2)	(14)	(11)	(1)	(1)	(1)	(2)
	(6)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)
G40~G42	(8)	(2)	(3)	(1)	(1)	(1)	(1)	(2)	(14)	(11)	(1)	(1)	(1)	(2)
	(9)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(1)	(2)
G43,G44 G49	(8)	(2)	(1)	(3)	(1)	(1)	(1)	(2)	(14)	(1)	(1)	(1)	(1)	(1)
	(9)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G52	(4)	(2)	(2)	(2)	(1)	(1)	(1)	(2)	(14)	(11)	(1)	(1)	(1)	(2)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)
G53	(1)	(1)	(1)	(1)	(4)	(4)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(1)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G54~G59	(1)	(1)	(1)	(1)	(3)	(3)	(1)	(2)	(1)	(17)	(1)	(1)	(1)	(17)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(18)	(1)	(1)	(1)	(18)
G54.1	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(2)	(1)	(1)	(1)	(2)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(1)	(2)
G60	(4)	(2)	(2)	(1)	(1)	(1)	(1)	(2)	(14)	(1)	(1)	(1)	(1)	(1)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
0307-3.tbl														

3

G code priority list (2)

Upper step : When commanded in the same block
Lower step : When commanded in another block

G code / Command G code	G00~G03 G102, G103, G202, G203	G22, G23	G40 ~ G42	G43, G44, G49	G54 ~G59	G54.1	G61, G64	G66, G67	G68, G69, G168	G73 ~ G89	G90, G91	G94	G98, G99	G177 ~ G189
G61,G64	(1)	(1)	(1)	(1)	(1)	(1)	(3)	(2)	(1)	(1)	(1)	(1)	(1)	(1)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G65	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G66,G67	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(3)	(2)	(2)	(2)	(2)	(2)	(2)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G68,G69, G168	(13)	(2)	(2)	(2)	(1)	(1)	(1)	(2)	(3)	(11) (14)	(1)	(1)	(1)	(2)
	(13)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G73~G89	(10)	(11)	(11)	(1)	(17)	(2)	(1)	(2)	(11) (14)	(3)	(1)	(1)	(1)	(3)
	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G90,G91	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(3)	(1)	(1)	(1)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G92	(4)	(2)	(2)	(2)	(1)	(1)	(1)	(2)	(2)	(11)	(1)	(1)	(1)	(2)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)
G94	(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)	(1)	(1)	(1)
G98,G99	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(3)	(1)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G100	(4)	(2)	(1)	(1)	(1)	(1)	(1)	(2)	(14)	(11)	(1)	(1)	(1)	(2)
	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(12)	(1)	(1)	(1)	(12)
G120	(4)	(2)	(2)	(1)	(1)	(1)	(1)	(2)	(14)	(11)	(1)	(1)	(1)	(2)
	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)
G121~G129	(8)	(2)	(2)	(1)	(1)	(1)	(1)	(2)	(14)	(11)	(1)	(1)	(1)	(2)
	(9)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)
0308-1.tbl														

G code priority list (3)

Upper step : When commanded in the same block
Lower step : When commanded in another block

G code / Command G code	G00~G03 G102, G103, G202, G203	G22, G23	G40 ~G42	G43, G44, G49	G54 ~G59	G54.1	G61, G64	G66, G67	G68, G69, G168	G73 ~G89	G90, G91	G94	G98, G99	G177 ~G182, G185~ G189
G131,G132	(4)	(2)	(2)	(1)	(1)	(1)	(1)	(2)	(14)	(11)	(1)	(1)	(1)	(2)
	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(2)	(2)	(1)	(1)	(1)	(2)
G133,G134	(4)	(2)	(2)	(2)	(1)	(1)	(1)	(2)	(14)	(11)	(1)	(1)	(1)	(2)
	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G173,G183	(10)	(2)	(2)	(1)	(17)	(2)	(1)	(2)	(14)	(3)	(1)	(1)	(1)	(3)
	(1)	(1)	(2)	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G177~G182, G185~G189	(10)	(2)	(2)	(1)	(17)	(2)	(1)	(2)	(14)	(3)	(1)	(1)	(1)	(3)
	(1)	(1)	(2)	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
0308-5.tbl														

G code priority list (4)

Upper step : When commanded in the same block
Lower step : When commanded in another block

G code Command G code	G04	G09	G10	G28~ G30	G36~ G39	G52	G53	G60	G65	G92	G100	G120	G121 ~ G129	G31, G131, G132	G133, G134	G173, G183
G00~G03 G102,G103 G202,G203	(4)	(1)	(4)	(4)	(5)	(4)	(1)	(4)	(2)	(4)	(4)	(4)	(8)	(4)	(4)	(10)
G04	(1)	(1)	(2)	(2)	(2)	(2)	(1)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)
G09	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G10	(2)	(1)	(1)	(2)	(2)	(2)	(1)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)
G22, G23	(2)	(1)	(2)	(2)	(2)	(2)	(1)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)
G28 ~G30	(2)	(1)	(2)	(3)	(2)	(2)	(1)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)
G36~G39	(2)	(1)	(2)	(2)	(3)	(2)	(1)	(1)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)
G40~G42	(2)	(1)	(2)	(2)	(2)	(2)	(1)	(2)	(2)	(2)	(1)	(2)	(2)	(2)	(2)	(2)
G43, G44 G49	(2)	(1)	(2)	(1)	(1)	(2)	(1)	(1)	(2)	(2)	(1)	(1)	(1)	(1)	(1)	(1)
G52	(2)	(1)	(2)	(2)	(2)	(1)	(1)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)
G53	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G54~G59	(16)	(1)	(1)	(16)	(16)	(1)	(4)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(1)	(17)
G54.1	(2)	(1)	(2)	(2)	(2)	(1)	(4)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(1)	(2)
G60	(2)	(1)	(2)	(2)	(1)	(2)	(1)	(1)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(1)
0310-1-2																

G code priority list (5)

Upper step : When commanded in the same block
Lower step : When commanded in another block

G code Command G code	G04	G09	G10	G28 ~ G30	G36 ~ G39	G52	G53	G60	G65	G92	G100	G120	G121 ~ G129	G31 G131 G132	G133 G134	G173 ~ G183
G61, G64	(1)	(4)	(1)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G65	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(1)	(2)	(2)	(2)	(2)	(2)	(2)	(2)
G66, G67	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(2)
G68, G69, G168	(14)	(1)	(14)	(14)	(14)	(14)	(1)	(14)	(2)	(14)	(14)	(14)	(14)	(14)	(14)	(14)
G73~G89	(11)	(1)	(11)	(11)	(11)	(11)	(1)	(1)	(2)	(11)	(11)	(11)	(11)	(11)	(11)	(3)
G90, G91	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G92	(2)	(1)	(2)	(2)	(2)	(2)	(1)	(2)	(2)	(1)	(2)	(2)	(2)	(2)	(2)	(2)
G94	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G98, G99	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(1)	(2)	(1)	(1)	(1)	(1)	(1)	(1)	(1)
G100	(2)	(1)	(2)	(2)	(2)	(2)	(1)	(2)	(2)	(2)	(1)	(2)	(2)	(2)	(2)	(2)
G120	(2)	(1)	(2)	(2)	(2)	(2)	(1)	(2)	(2)	(2)	(2)	(1)	(2)	(2)	(2)	(2)
G121~ G129	(2)	(1)	(2)	(2)	(2)	(2)	(1)	(2)	(2)	(2)	(2)	(2)	(3)	(2)	(2)	(2)

0310-1.tbl

G code priority list (6)

Upper step : When commanded in the same block
Lower step : When commanded in another block

G code Command G code	G04	G09	G10	G28~ G30	G36~ G39	G52	G53	G60	G65	G92	G100	G120	G121~ G129	G131, G132	G133, G134	G173, G183
G131, G132	(2)	(1)	(2)	(2)	(2)	(2)	(1)	(2)	(2)	(2)	(2)	(2)	(2)	(3)	(2)	(2)
G133, G134	(2)	(1)	(2)	(2)	(2)	(2)	(1)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(3)	(2)
G173, G183	(2)	(1)	(2)	(2)	(2)	(2)	(1)	(1)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(3)
G177~G182	(2)	(1)	(2)	(2)	(2)	(2)	(1)	(1)	(2)	(2)	(2)	(2)	(2)	(2)	(2)	(3)
G185~G189																
0311-1.tbl																

CHAPTER 4

PREPARATION FUNCTION (TOOL OFFSET FUNCTION)

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.

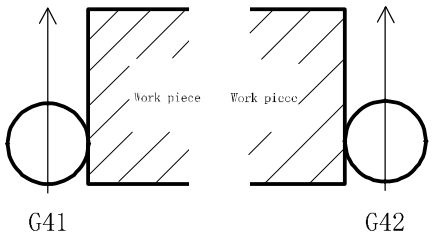
Command format

(G41 / G42)		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.



manual1:c:ai4.1j:4章図：4002-1.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.

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

(Note)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.

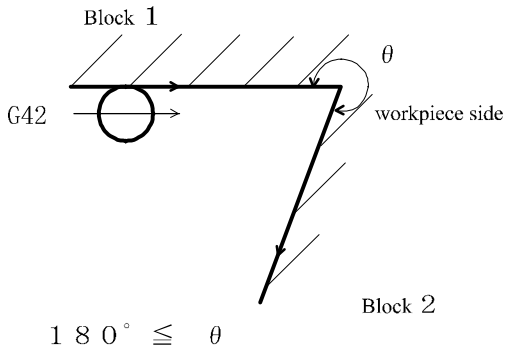
In the cancel mode, the path of the tool center coincides with the programmed path.

Terms and symboles for tool dia offset

1. Inside and outside

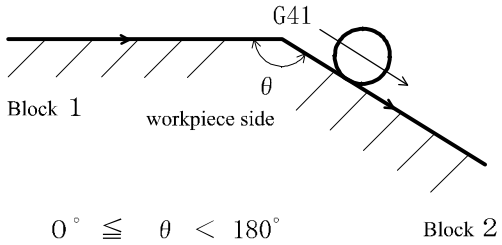
If the angle measured on workpiece side is larger than 180°, it is called "Inside".

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



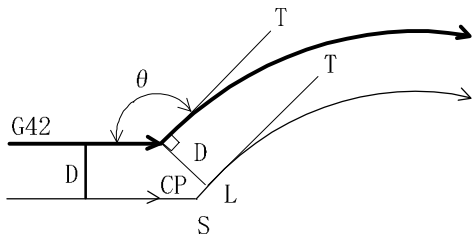
Inside

manual1:c:ai4.1j:chapter4: 4003-1.AI



Outside

manual1:c:ai4.1j:chapter4: 4003-2.AI



manual1:c:ai4.1j:chapter4:4003-3.AI

2. Symbols in figure

———	:	Programmed path
———	:	Tool center path
———	:	Auxiliary line
L	:	Linear line
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.

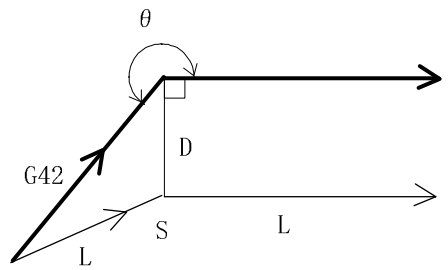
a) G41 or G42 is commanded.
b) The tool offset number is not zero.
b) The movement command other than circular arc (G02 or G03) is given on the X-Y plane,and the movement distance is not zero.

(Note 1) In the case of circular arc command, an alarm is generated.
(Note 2) In case of the linear interpolation XZ or YZ axes command, an alarm is generated.

4

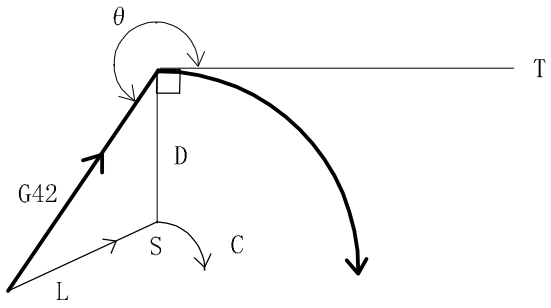
4.1.3.1 Inside cutting （１８０° ≦ θ）

Linear-Linear



manual1:c:ai4.1j:chapter4:4005-1.AI

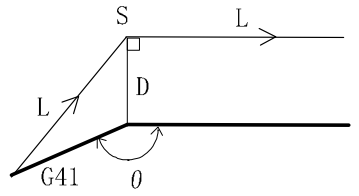
Linear-Arc



manual1:c:ai4.1j:chapter4:4005-2.AI

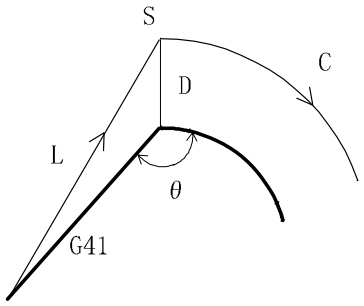
4.1.3.2 Outside cutting （９０° ≦ θ ＜ １８０°）

(a)Type 1 : Linear - Linear



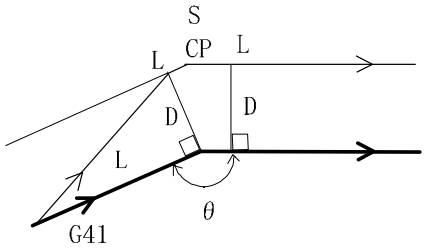
manual1:c:ai4.1j:chapter4:4006-1.AI

Type 1 : Linear - Arc



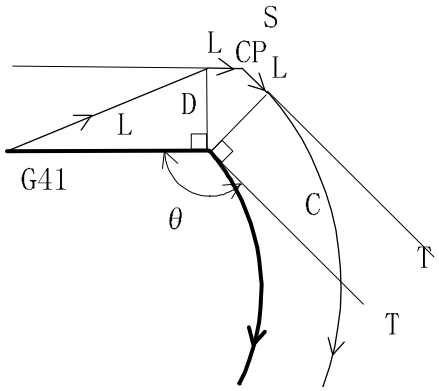
manual1:c:ai4.1j:chapter4:4006-2.AI

(b)Type 2 : Linear - Linear



manual1:c:ai4.1j:chapter4:4006-3.AI

Type 2 : Linear - Arc



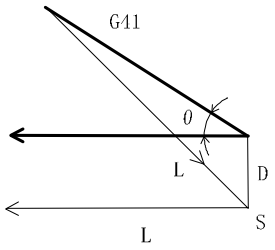
manual1:c:ai4.1j:chapter4:4006-4.AI

(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.

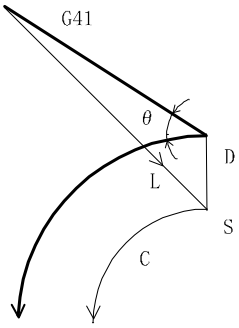
4.1.3.3 Outside cutting （θ ＜ ９０°）

(a)Type 1 : Linear - Linear



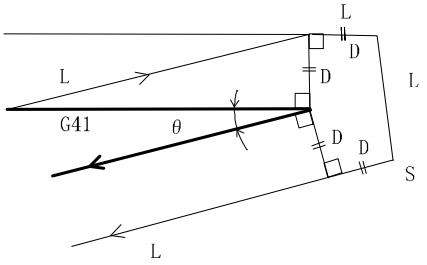
manual1:c:ai4.1j:4章図：4007-1.AI

Type 1 : Linear - Arc



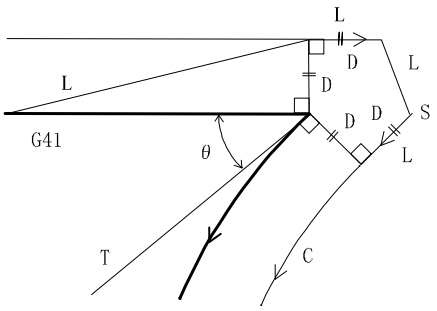
(b)Type 2 : Linear - Linear

manual1:c:ai4.1j:4章図：4007-2.AI



manual1:c:ai4.1j:4章図：4007-3.AI

Type 2 : Linear - Arc



manual1:c:ai4.1j:4章図：4007-4.AI

(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.

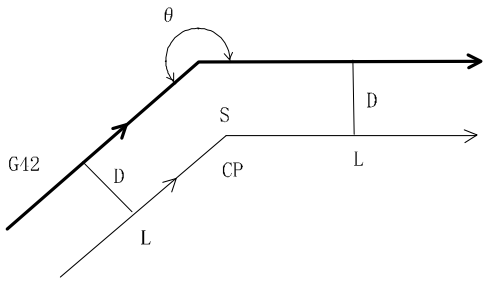
4.1.4 Offset mode

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

(Note) The linear interpolation XZ or YZ axes command is excluded. When this command is given, an alarm is generated.

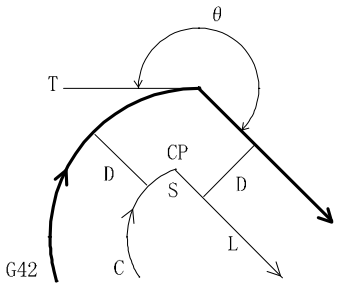
4.1.4.1 Inside cutting （ １８０° ≦ θ ）

Linear - Linear



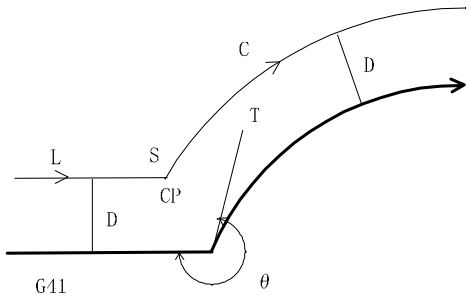
manual1:c:ai4.1j:4章図：4008-1.AI

Arc - Linear



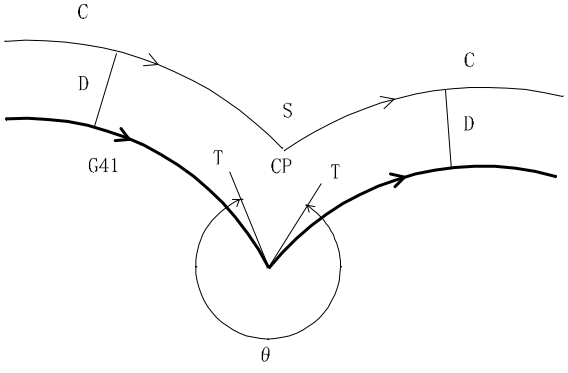
manual1:c:ai4.1j:4章図：4008-2.AI

Linear - Arc



manual1:c:ai4.1j:4章図：4008-3.AI

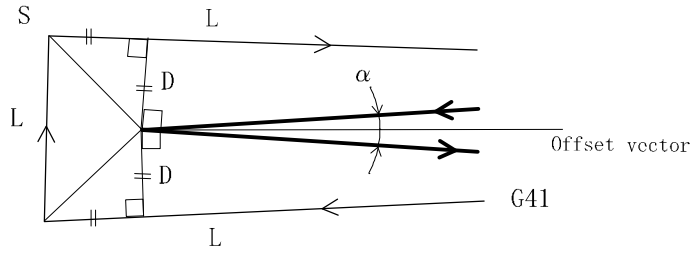
Arc - Arc



manual1:c:ai4.1j:4章図：4008-4.AI

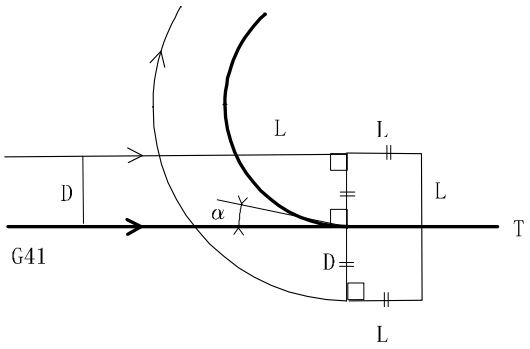
(Note 1) When going around at a narrow angle （ $\alpha < 1°$ ）, there is no crosspoint of 2 perpendicular lines from programmed lines, so that tool centerpath will be exceptionally as follows;

Linear - linear



manual1:c:ai4.1j:4章図：4009-1.AI

Linear -Arc

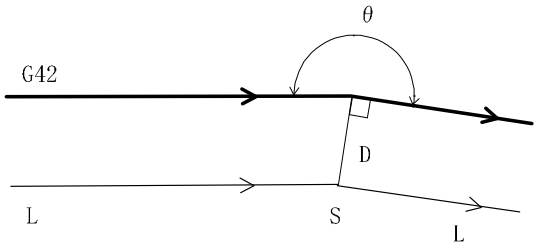


manual1:c:ai4.1j:4章図：4009-2.AI

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

(Note 2) When($180° \leqq \theta < 181°$), tool center path will be as follows;

Linear - linear

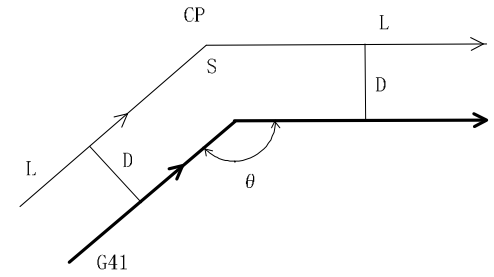


manual1:c:ai4.1j:4章図：4009-3.AI

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

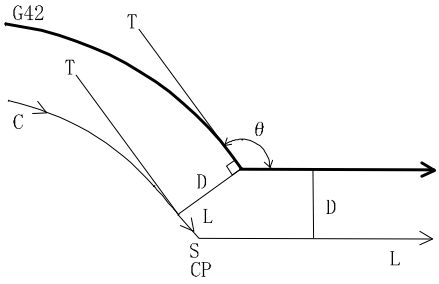
4.1.4.2 Outside cutting （ ９０° ≦ θ ＜ １８０° ）

Linear - Linear



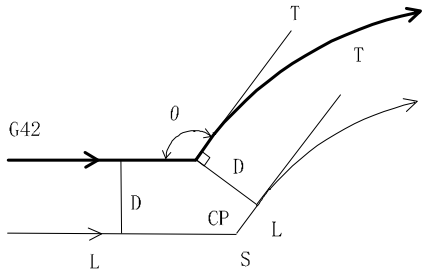
manual1:c:ai4.1j:4章図：4010-1.AI

Arc - Linear



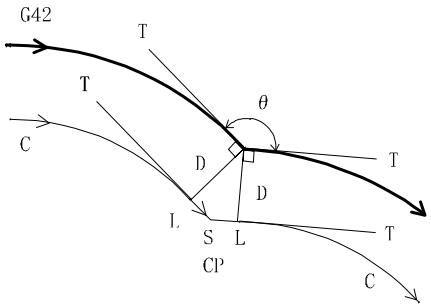
manual1:c:ai4.1j:4章図：4010-2.AI

Linear - Arc



manual1:c:ai4.1j:4章図：4010-3.AI

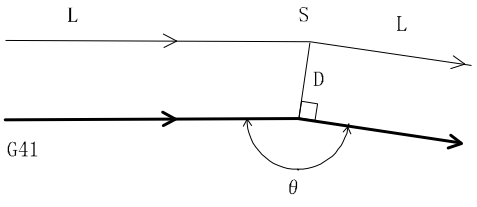
Arc - Arc



manual1:c:ai4.1j:4章図：4010-4.AI

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

Linear -Linear

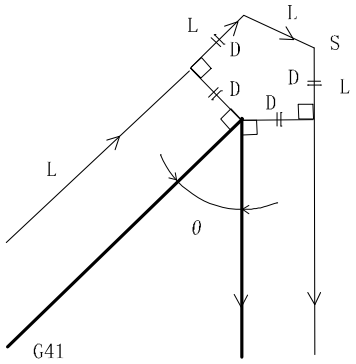


manual1:c:ai4.1j:4章図：4010-5.AI

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

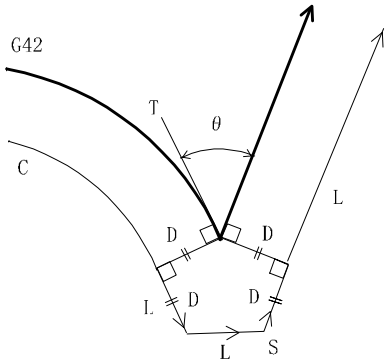
4.1.4.3 Outside cutting

Linear - Linear



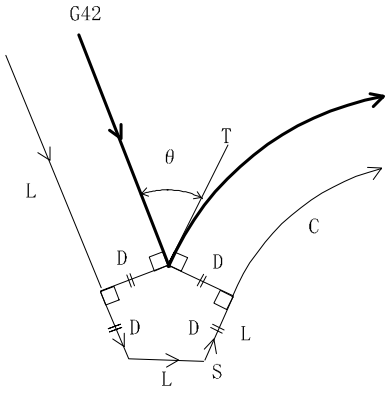
manual1:c:ai4.1j:4章図：4011-1.AI

Arc - Linear



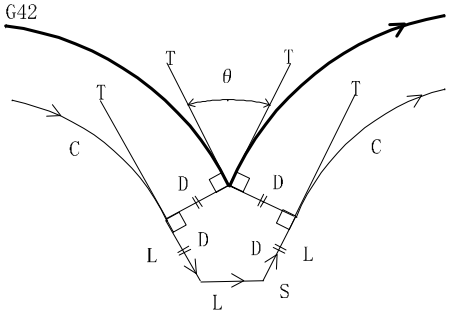
manual1:c:ai4.1j:4章図：4011-2.AI

Linear - Arc



manual1:c:ai4.1j:1章図：4011-3.AI

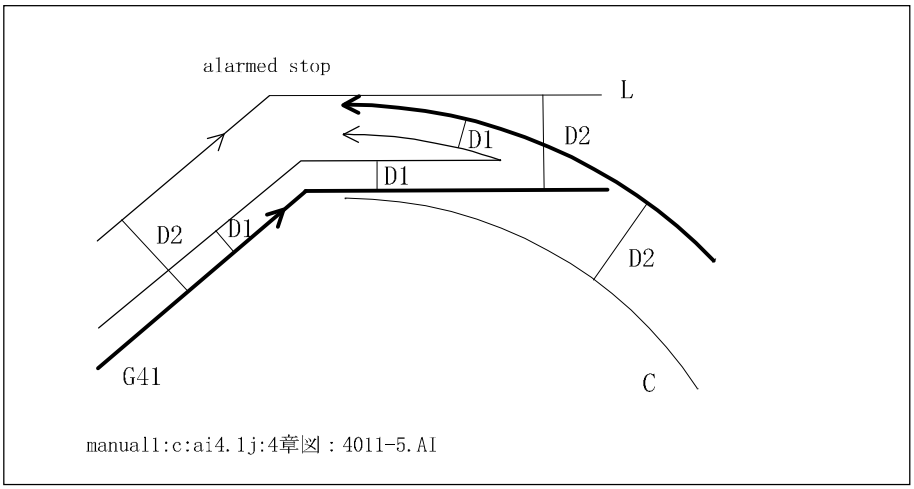
Arc - Arc



manual1:c:ai4.1j:4章図：4011-4.AI

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.

4.1.5 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 ;**

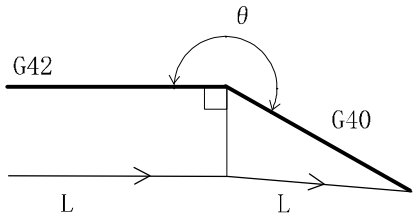
b) The movement command other than circular arc (G02 or G03) is given on the X-Y plane.

(Note 1) In the case of a circular arc command, an alarm is generated.

(Note 2) In case of the linear interpolation XZ or YZ axes command, an alram is generated.

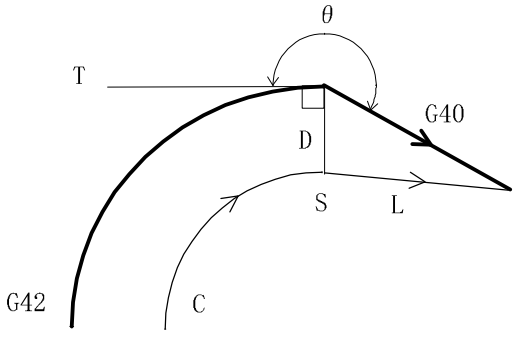
4.1.5.1 Inside cutting （１８０° ≦ θ）

Linear - Linear



manual1:c:ai4.1j:4章図：4013-1.AI

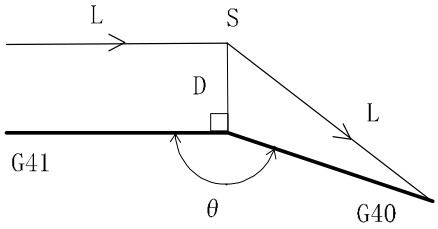
Arc - Linear



manual1:c:ai4.1j:4章図：4013-2.AI

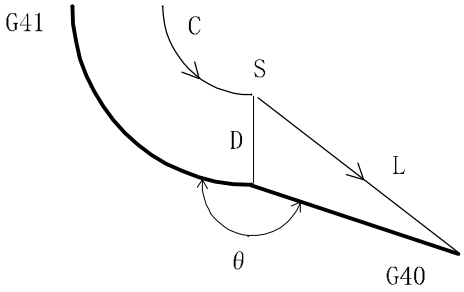
4.1.5.2 Outside cutting （９０° ≦ θ ＜ １８０°）

Type 1:Linear-Linear



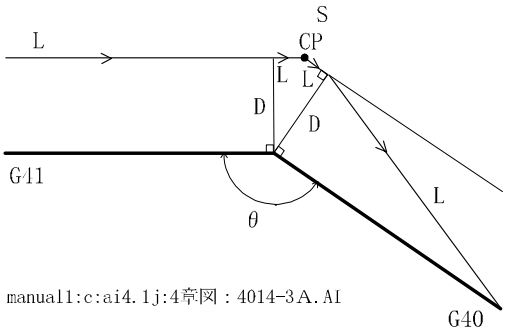
manual1:c:ai4.1j:4章図：4014-1.AI

Type 1:Arc-Linear



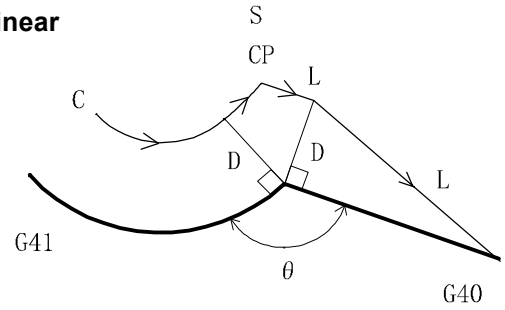
manual1:c:ai4.1j:4章図：4014-2.AI

Type 2:Arc-Linear



manual1:c:ai4.1j:4章図：4014-3A.AI

Type 2:Linear-Linear



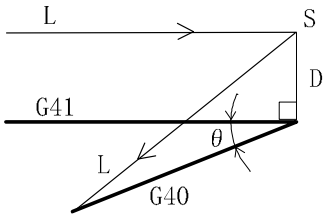
manual1:c:ai4.1j:4章図：4014-4.AI

(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.

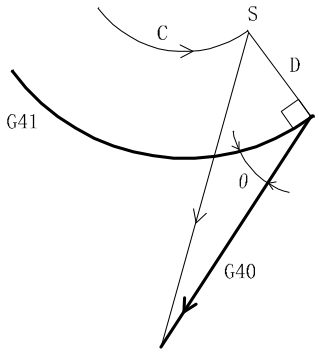
4.1.5.3 Outside cutting （θ ＜ ９０°）

Type 1:Linear-Linear



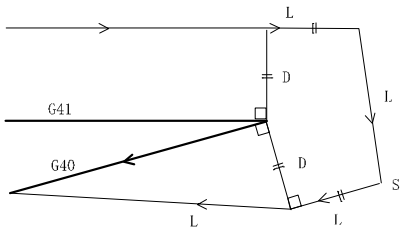
manual1:c:ai4.1j:4章図：4015-1.AI

Type 1:Arc-Linear



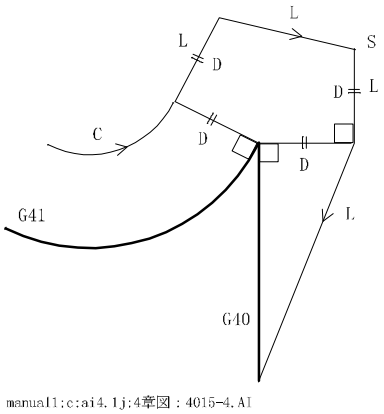
manual1:c:ai4.1j:4章図：4015-2.AI

Type 2:Linear-Linear



manual1:c:ai4.1j:4章図：4015-3.AI

Type 2:Arc-Linear



manual1:c:ai4.1j:4章図：4015-4.AI

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

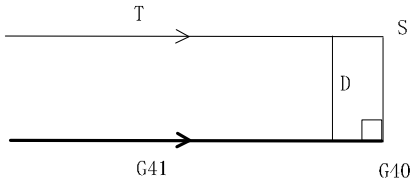
(Note 2) If the angle is close to 1(θ ≦ 1°) while type 2 is being selected, actual movement will be type 1.

4.1.6 G40 single command

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

Linear - Linear

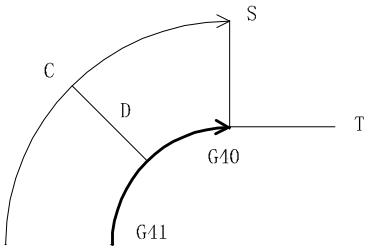
G４１ X__Y__D__；
G４０ ；



manual1:c:ai4.1j:4章図：4016-1.AI

Arc - Linear

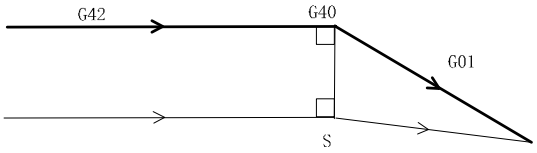
G４１ X__Y__D__；
G４０ ；



manual1:c:ai4.1j:4章図：4016-2.AI

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

G４２ X__Y__D__；
G４０ ；
G０１ X__Y__F__；



manual1:c:ai4.1j:4章図：4016-3.AI

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.

Offset amount sign / G code	+	−
G41	Left side offset	Right side offset
G42	Right side offset	Left side offset

4

Conditions of execution

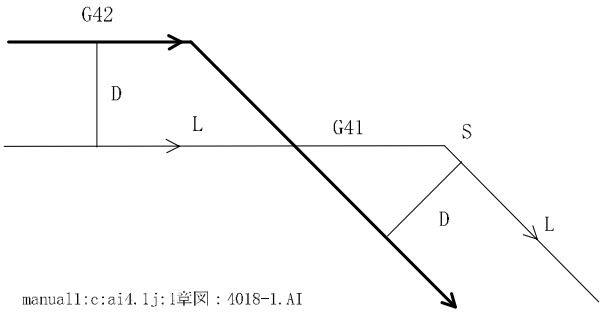
Offsetmode	command	Linear-Linear	Linear-Arc	Arc-Linear	Arc-Arc
G41	G41	Perform 〔Where the offset equivalent to the tool dia is vertically applied to the end point of the previous block becomes the stop point.〕			
G42	G42				
G41	G42	Perform		Perform	
G42	G41				

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

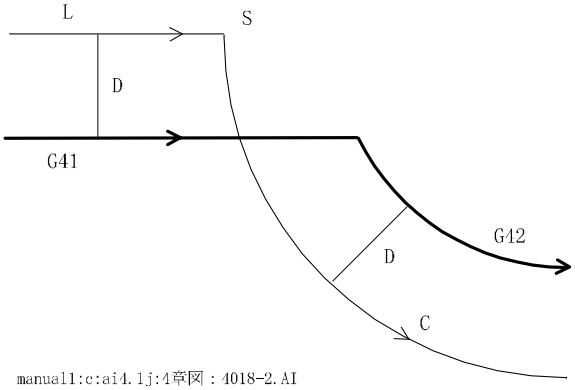
4.1.8.1 When there is a cross point

Linear - Linear



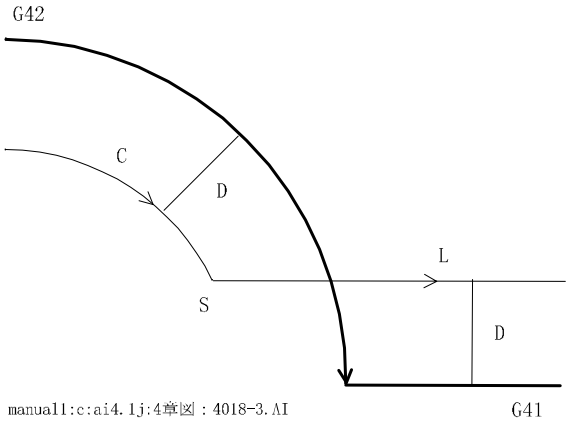
manual1:c:ai4.1j:1章図：4018-1.AI

Linear - Arc



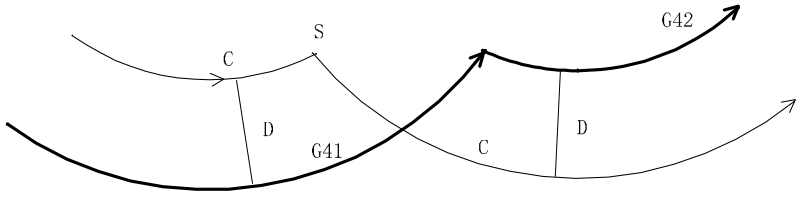
manual1:c:ai4.1j:4章図：4018-2.AI

Arc - Linear



manual1:c:ai4.1j:4章図：4018-3.AI

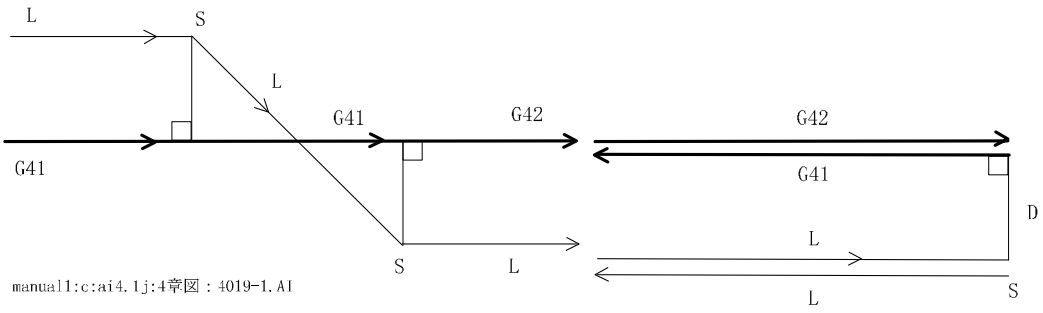
Arc - Arc



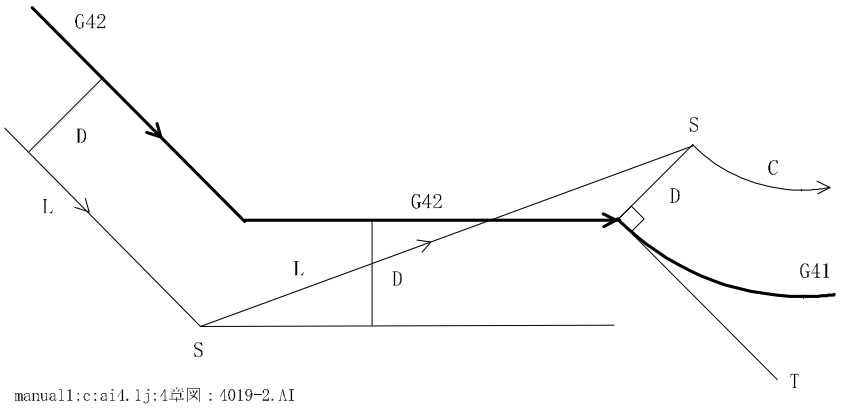
manual1:c:ai4.1j:4章図：4018-4.AI

4.1.8.2 When there is no cross point

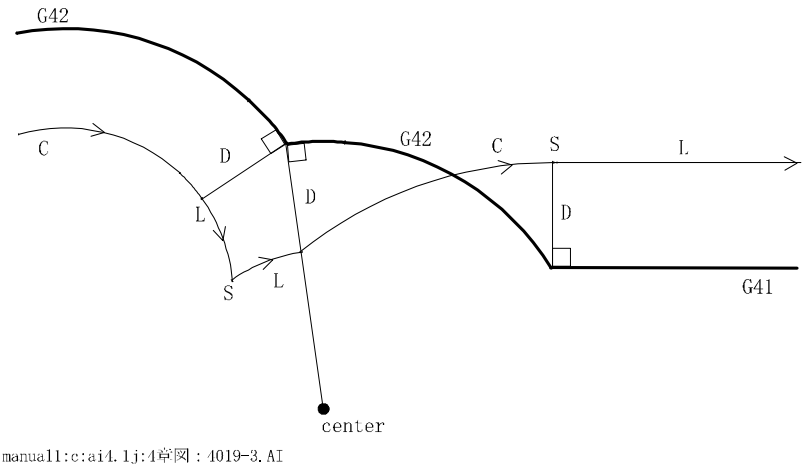
Linear - Linear

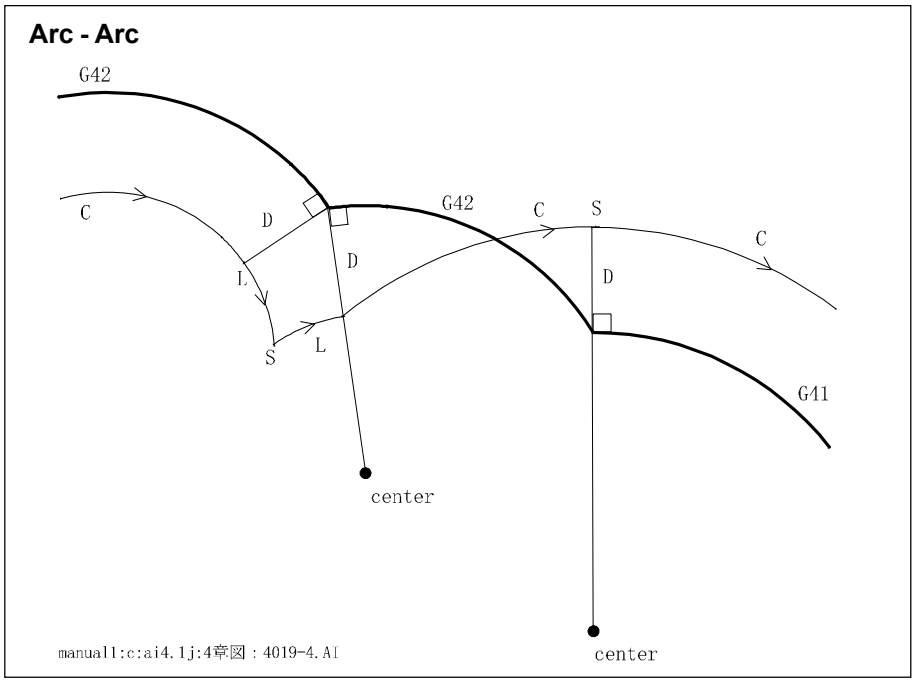


Linear - Arc



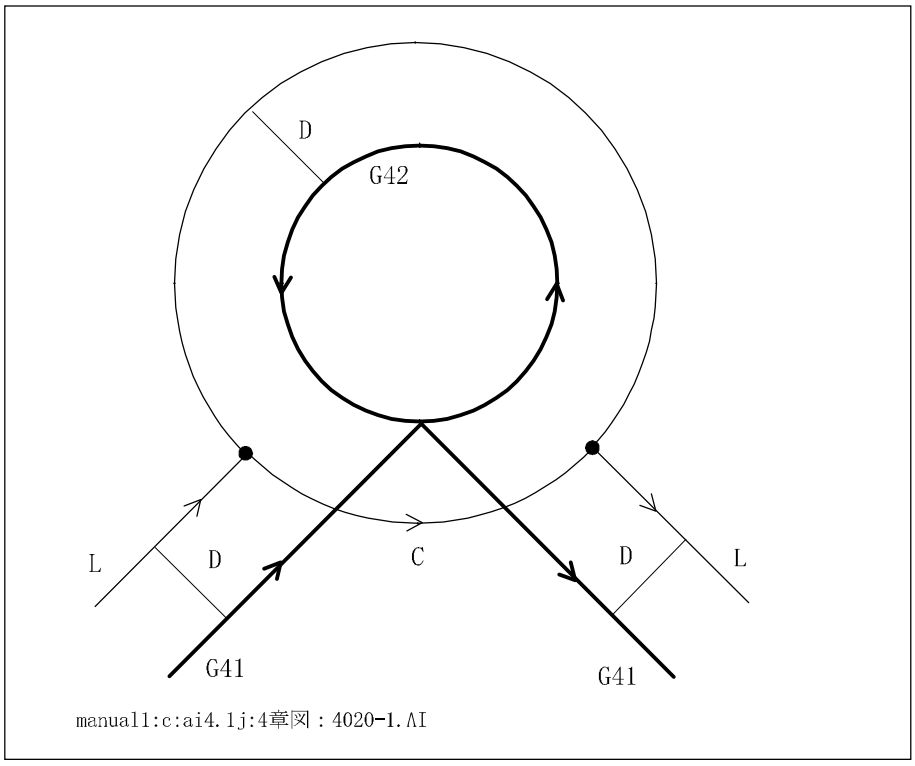
Arc - Linear





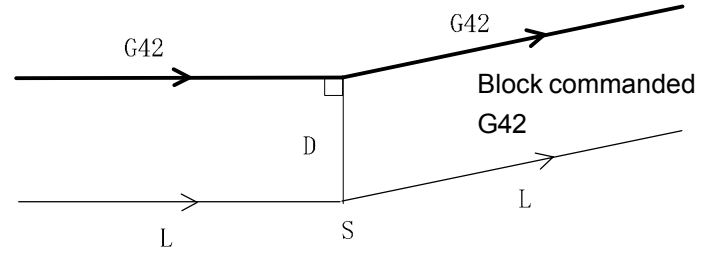
4.1.8.3 When offset path becomes more than a circle

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.



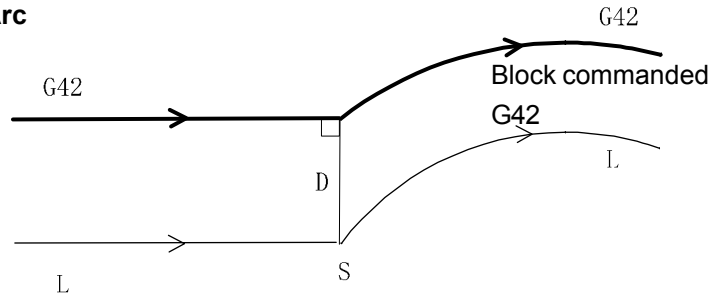
4.1.9 G code command for tool dia offset in offset mode

Linear - Linear



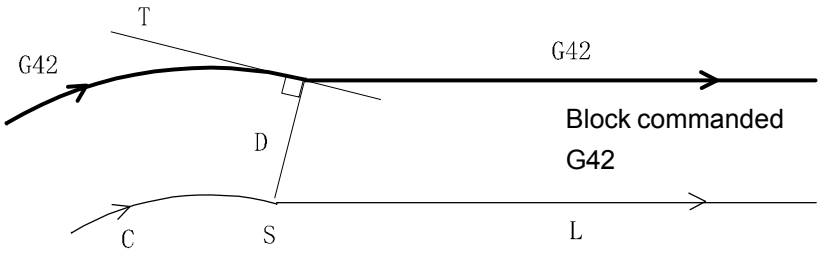
manual1:c:ai4.1j:4章図：4021-1.AI

Linear - Arc



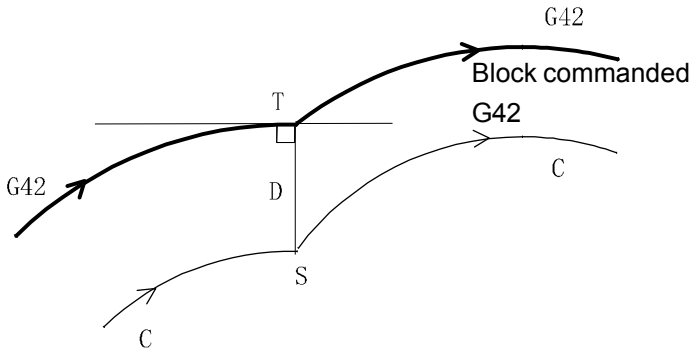
manual1:c:ai4.1j:4章図：4021-2.AI

Arc - Linear



manual1:c:ai4.1j:4章図：4021-3.AI

Arc - Arc



manual1:c:ai4.1j:4章図：4021-4.AI

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.

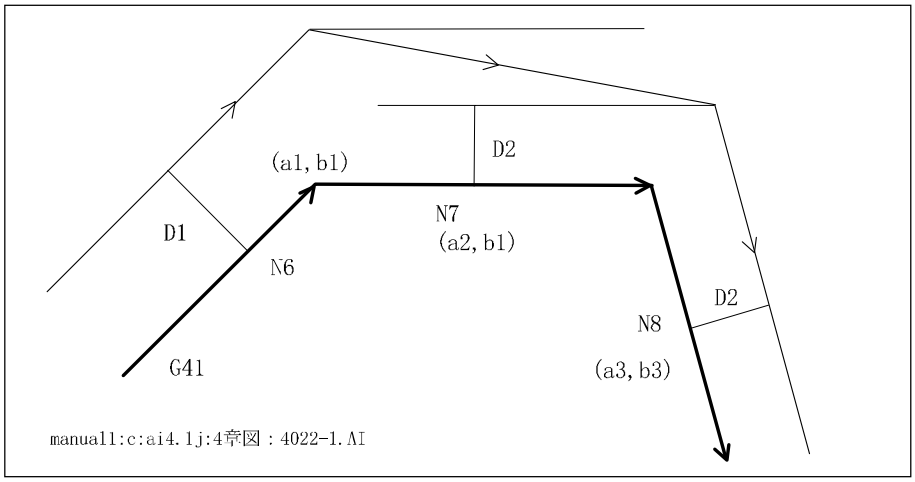
When G41 or G42 is commanded, the offset amount is commanded in the same block.

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.

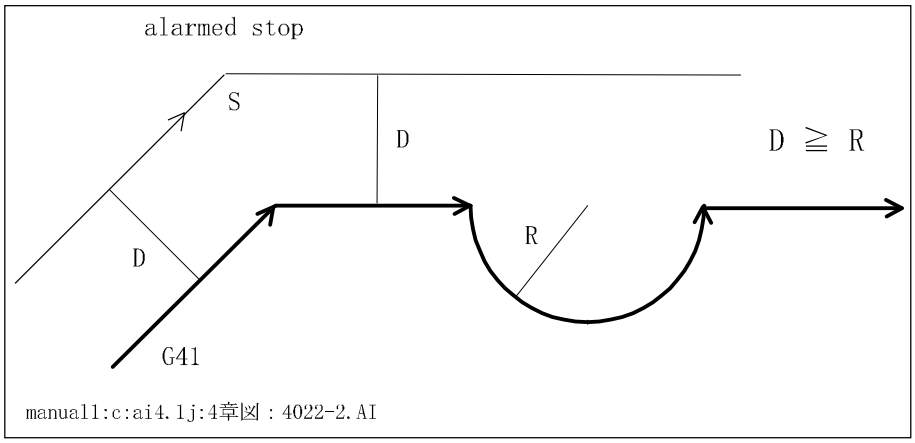
N1 G41 X_Y_D1;
N6 Xa1 Yb1;
N7 Xa2 D2;....Change of offset amount
N8 Xa3 Yb3;



(3) Current position display

The current position display corresponds to the tool center position

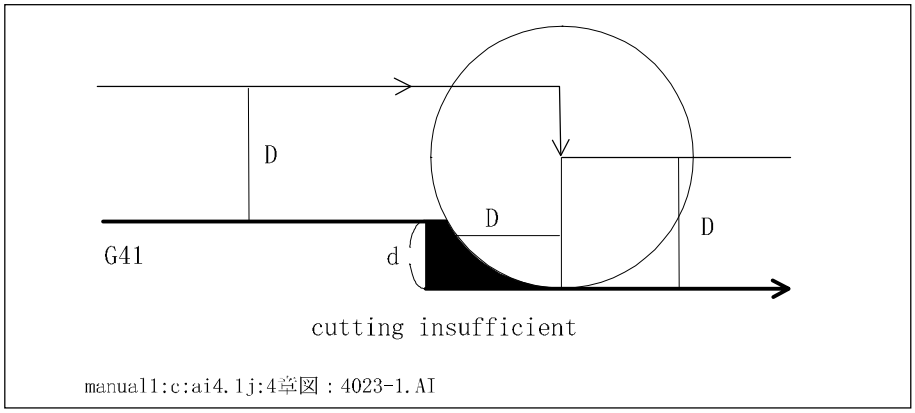
(4) Machining of inner wall of circular arc smaller than the used tool radius



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

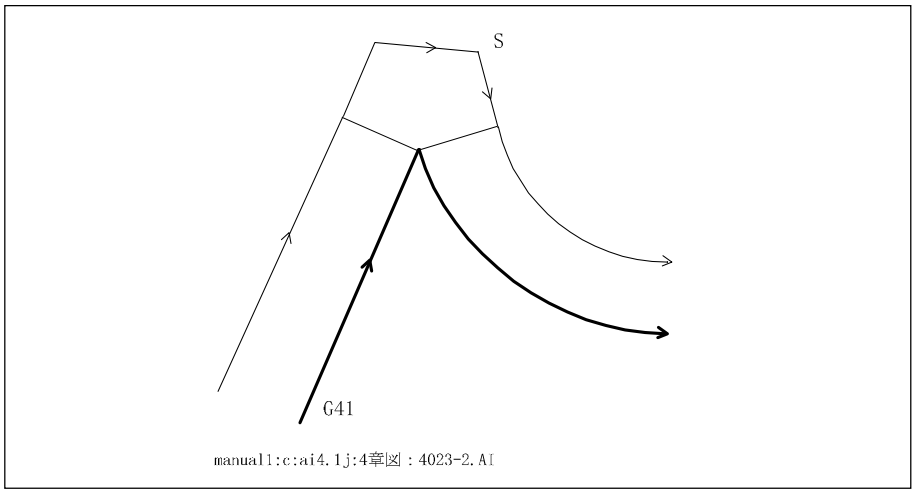
(5) Cutting insufficient

This problem occurs in the case of a program containing a step smaller than the tool radius.



(6) Corner movement

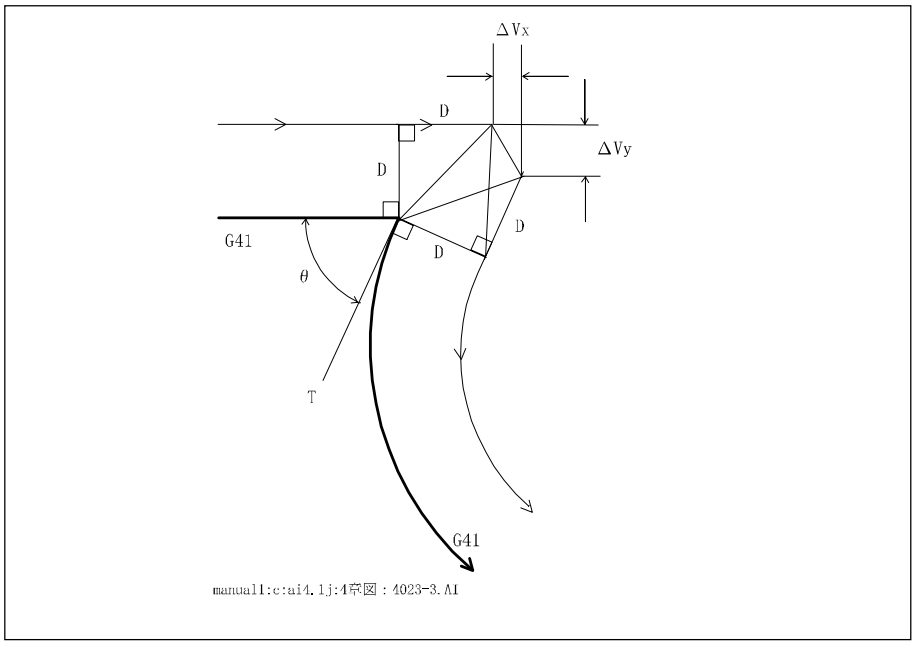
When cutting the outer side, the tool moves around the corner from different angles.The movement mode and the feedrate up to the single block stop point are as specified in the current block.



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.

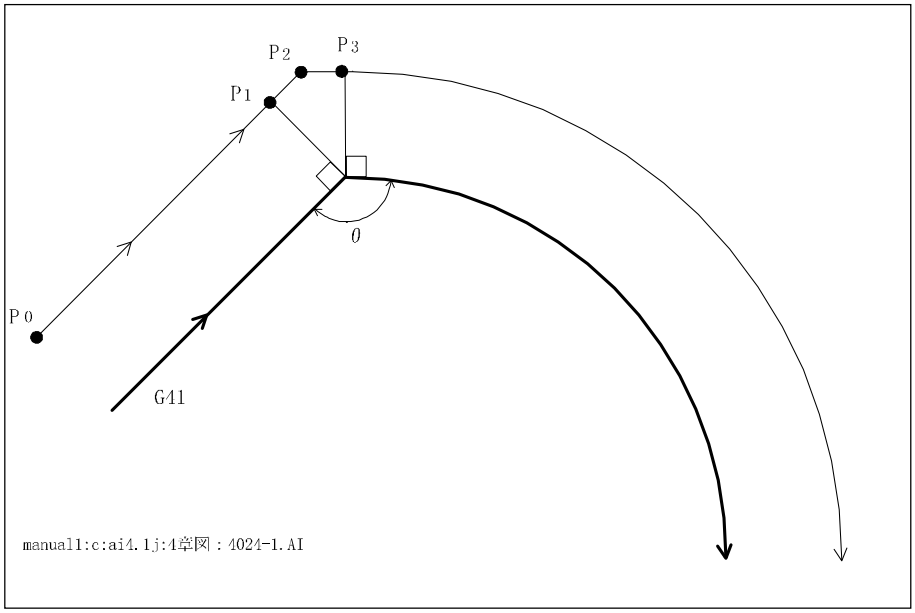
$\Delta VX \leqq \triangle V$ and $\triangle VY \leqq \triangle V$

The value $\triangle V$ 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

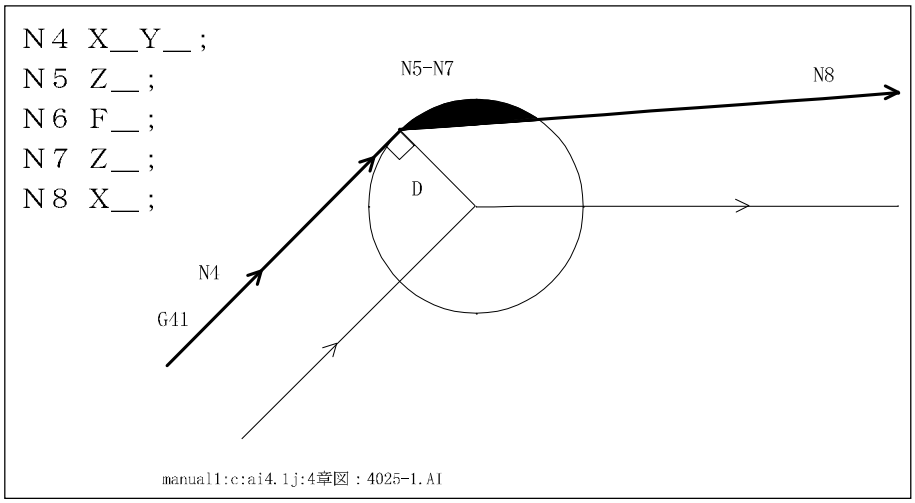
The tool moves once around the circular arc afterwards with P3 as the target.
If this small movement function is used, the movement from P2 to P3 is ignored and executed as follows:

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

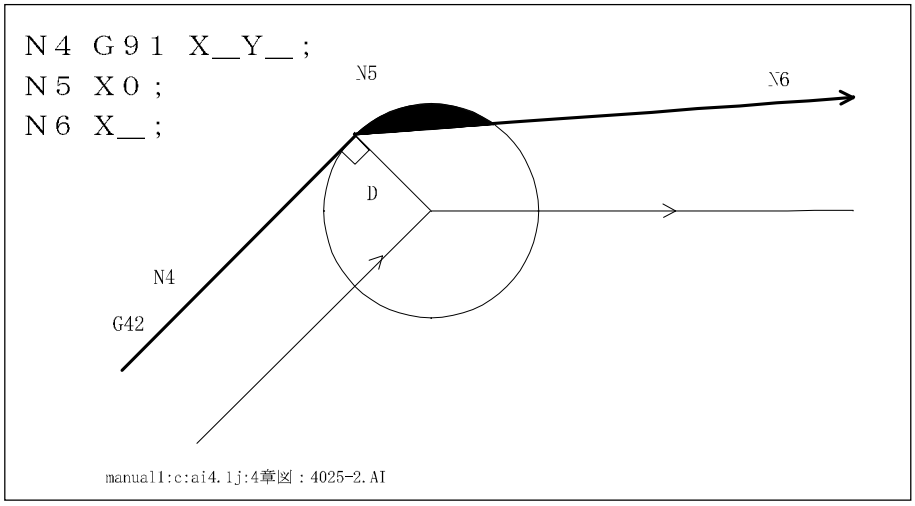
A full circular is ignored and the movement from P2 to P3 becomes a small circular movement, thus ignoring this function.

(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.



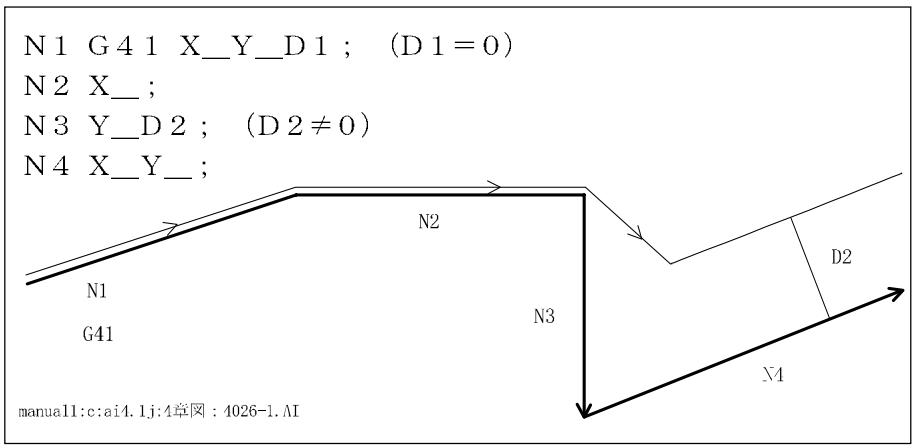
(Note 1) The block with the movement amount zero will result in the same as above.



(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 the same as the case of changing the tool dia offset amount as described in (2).

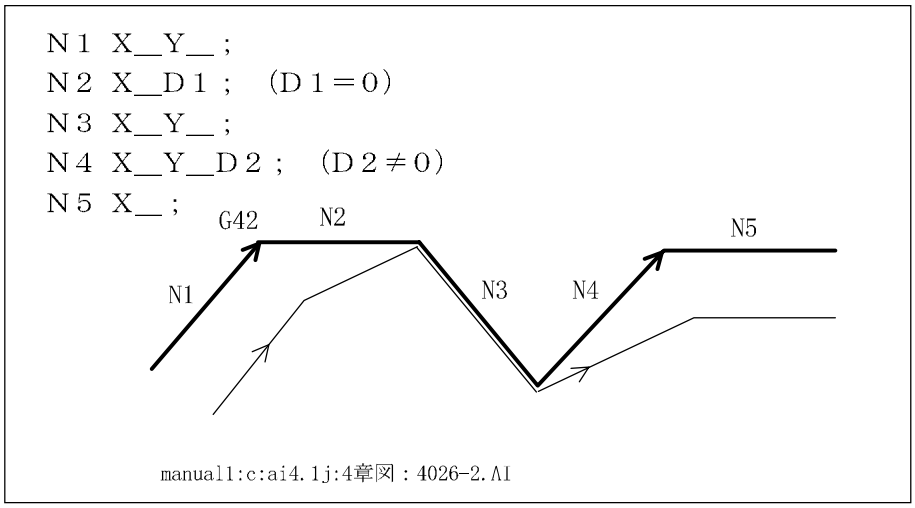


(Note 2) Even if the X/Y movement command is not given at the start-up, the start-up motion begins at the time the X or Y movement command is executed.

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).



(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

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 spcified 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

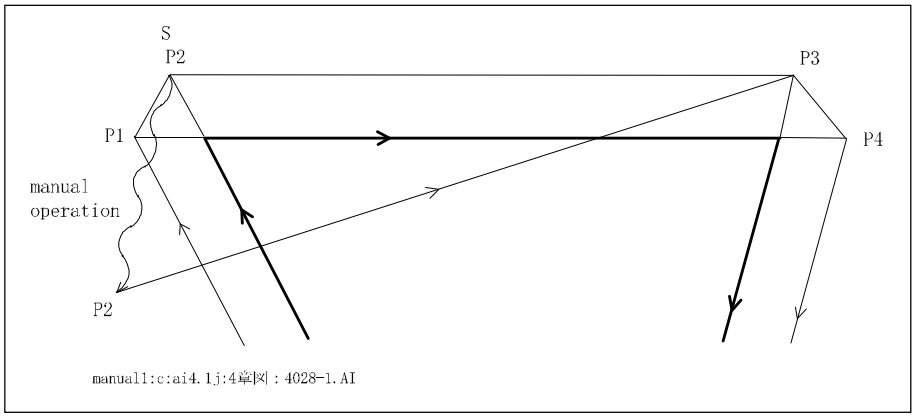
M206 : Thermal watch condition judgement

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.



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

4.1.11 Override function related to tool dia offset

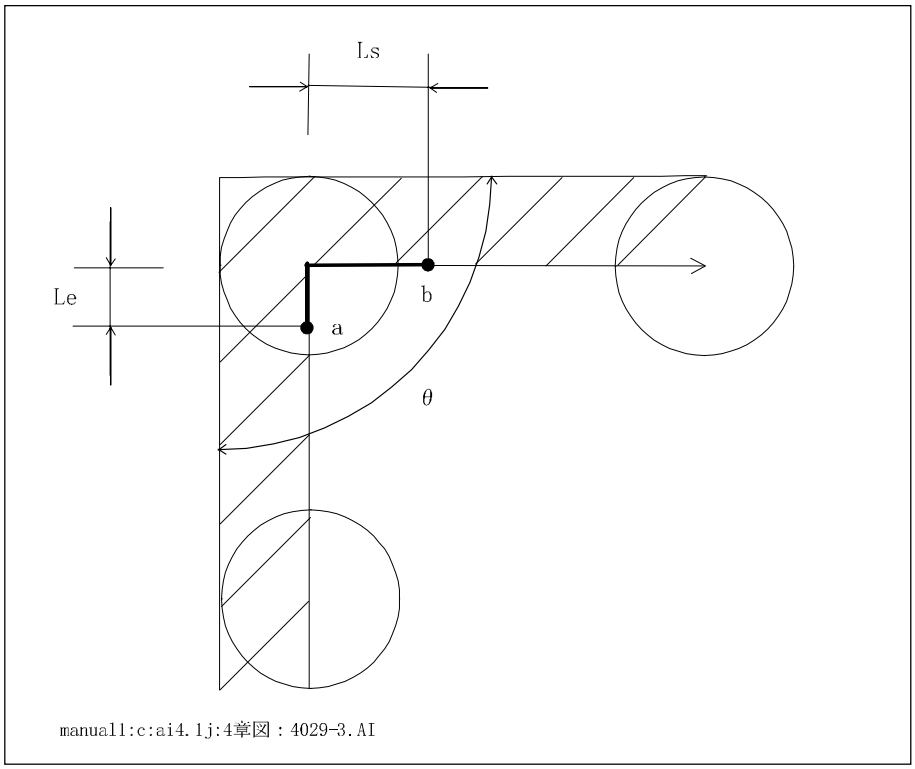
4.1.11.1 Automatic corner override

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 by G01, G02 or G03.
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 OVERRIDE 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.CORNEROVERRIDELEN1
: Deceleration stroke at corner end point :Le
2)AUTO.CORNEROVERRIDELEN1
: 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.

$$\text{Actual feedrate} = \text{Commanded feedrate} \times \frac{\text{Deceleration stroke}}{100}$$

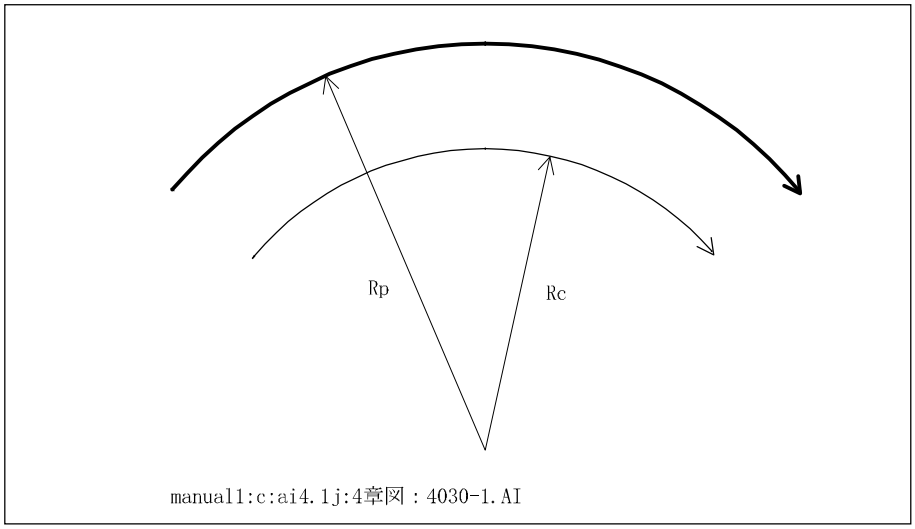
4.1.11.2 Override of the inside circular cutting

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.

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

Rp: Programmed radius

Rc: Tool center path radius



(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.

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

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.

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

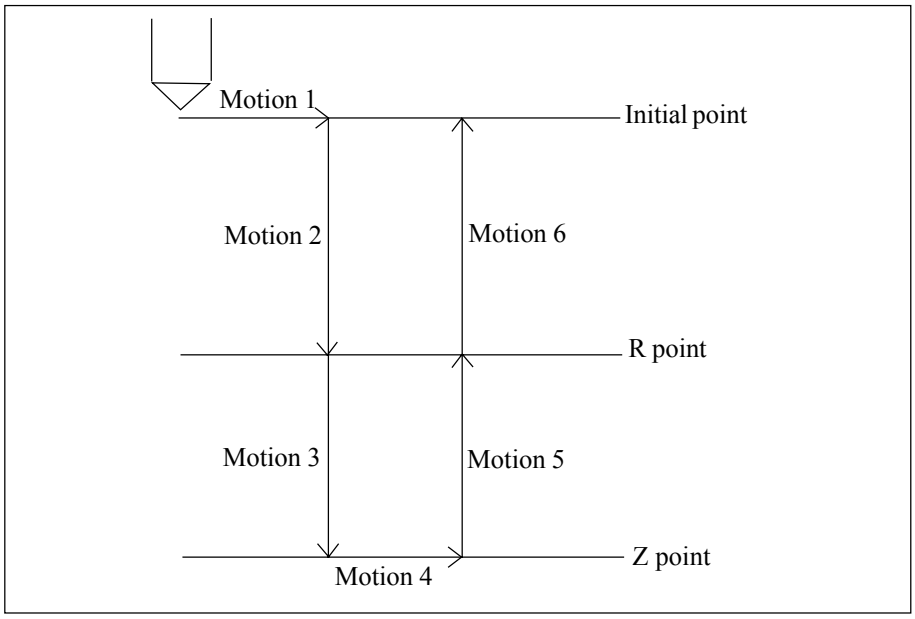
G code	Content	Feeding type at machining	Spindle motion at bottom of hole	Retracting motion	Spindle at return point
G73	High speed peckdrilling	intermittent feed	dwell	rapid feed	
G74	Reverse tapping	cutting feed	dwell→CW	cutting feed	stop
G76	Fine boring	cutting feed	dwell→orientation rapid	rapid feed	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	
G173	High speed peck drilling	Intermittent feed	dwell	none	_

Gcode	Content	Feeding type at machining	Spindle motion at bottom of hole	Retracting motion	Spindle at return point
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	
G183	Peck drilling	intermittent feed	dwell	none	_
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)



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

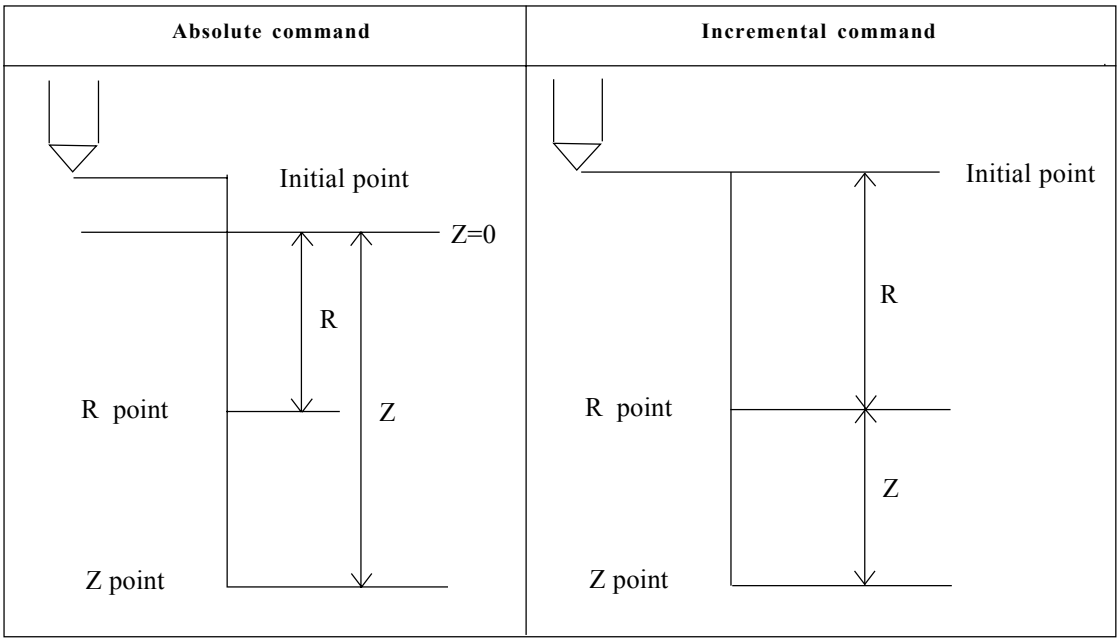
(2) Return level
- G98 Initial point level return
- G99 R point level return

(3) Drilling mode

G73 , G74
G76 ~ G78
G80 ~ G87
G89
G173
G177 , G178
G181 ~ G183
G185 , G186
G189

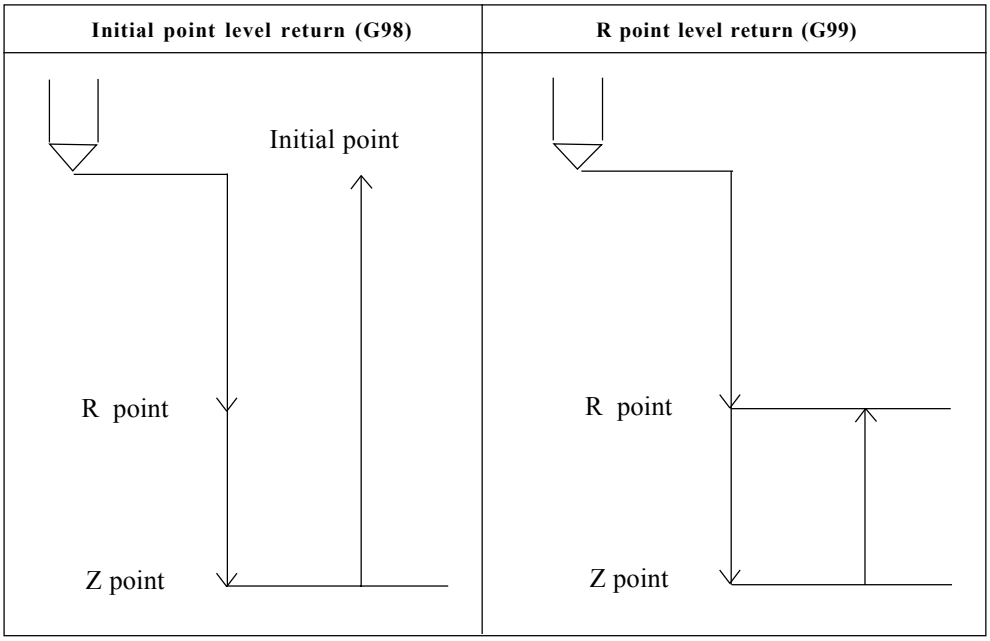
Refer to table 5-1(P5-2)

5.3.2 Setting of data in absolute/incremental command



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.



(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.

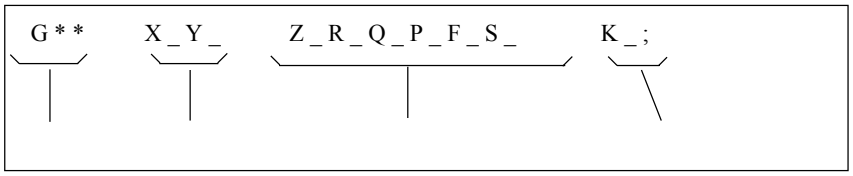
5.3.4 Canned cycle motion conditions

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 code Position data Machining data Repeat number

G codeG73,G74,G76~G78,G81~G87,G89,G173, G177, G178, G181~G183,G185,G186,G189.

The G codes of the canned cycle are all modal.(except G173,G183)

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.

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

F : Cutting feedrate.

S : Spindle speed.

K : Repeat number of canned cycle.

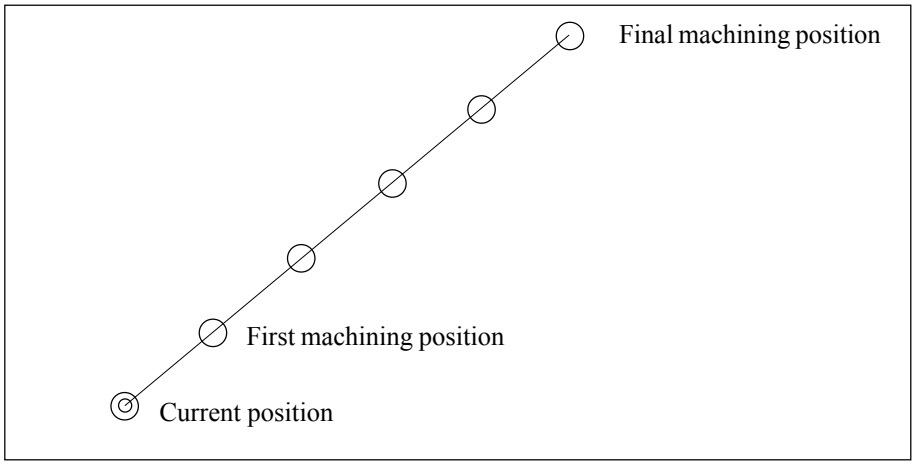
5.3.6 Repeat number of canned cycle

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)



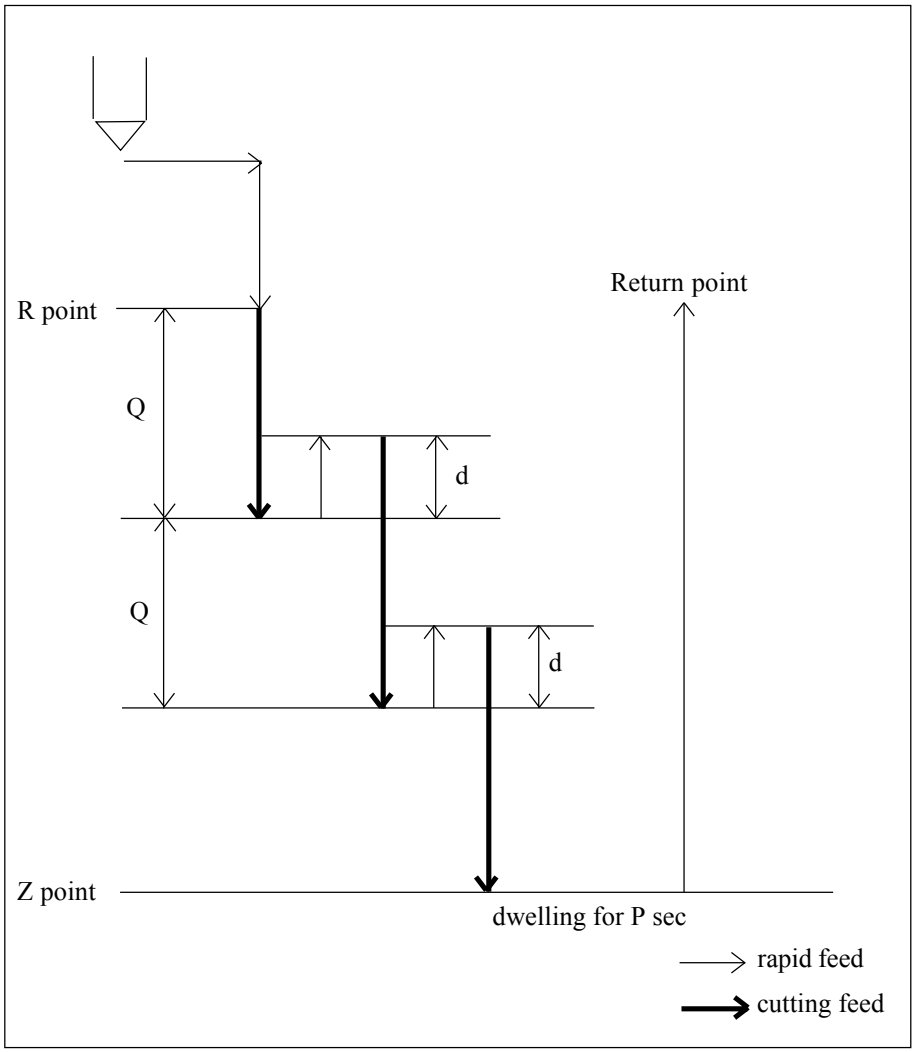
G81 X _ Y _ Z _ R _ K5F _ ; (in G91 mode)

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 _ ;

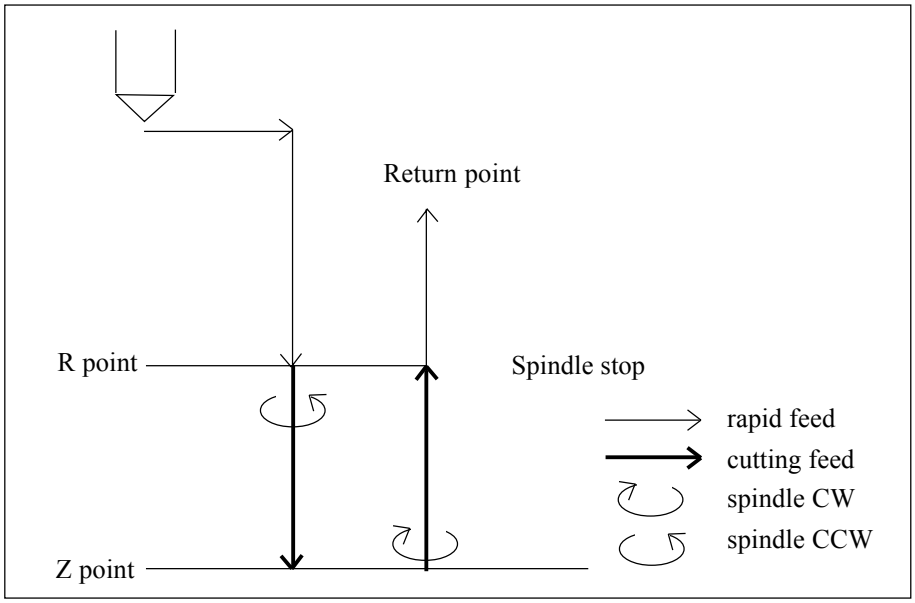


The relief amount d is set by the parameter 1.
If the minus value is commanded for the cutting amount Q, the algebraic mark (-) is ignored.

5.4.2 Reverse tapping cycle (G74)

Command format

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

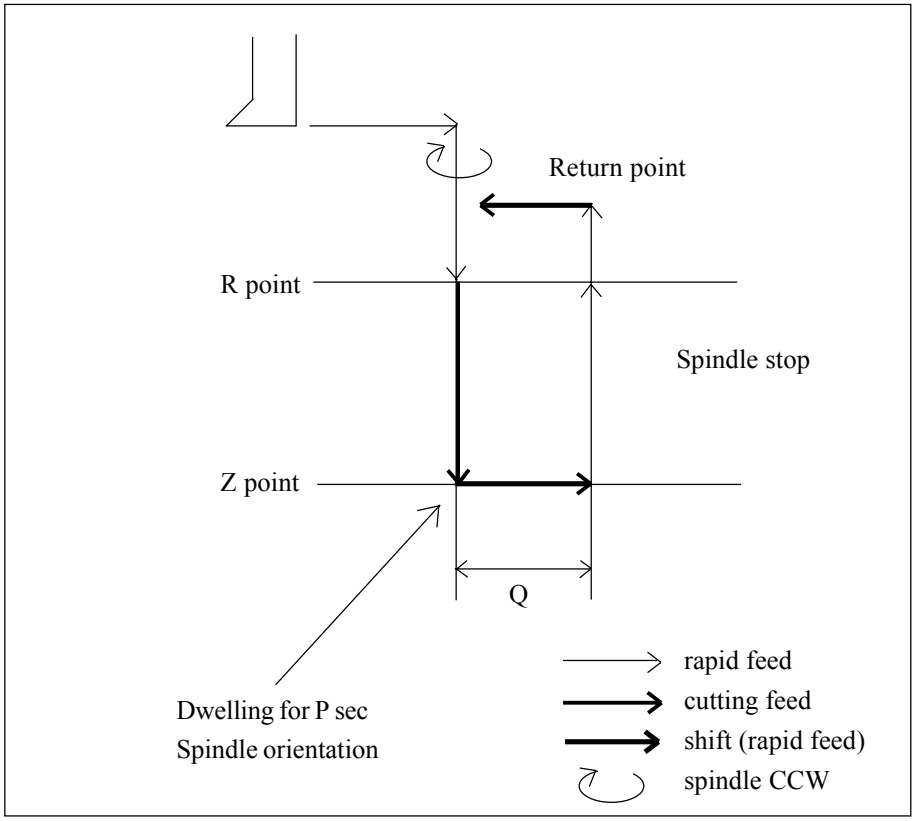


Spindle rotation stops at Z point.
After dwelling for P sec, spindle rotates CW.
When a temporary stop is applied on the way from the R point to the Z point or the R point, the tool stops after returning to the R point.
If the address S exceeds the max, spindle speed in tapping, an alarm is generated.

5.4.3 Fine boring cycle (G76)

Command format

G76 X _ Y _ Z _ R _ Q _ P _ F _ ;



If the minus value is commanded for the shift amount Q, the algebraic mark (-) is ignored.

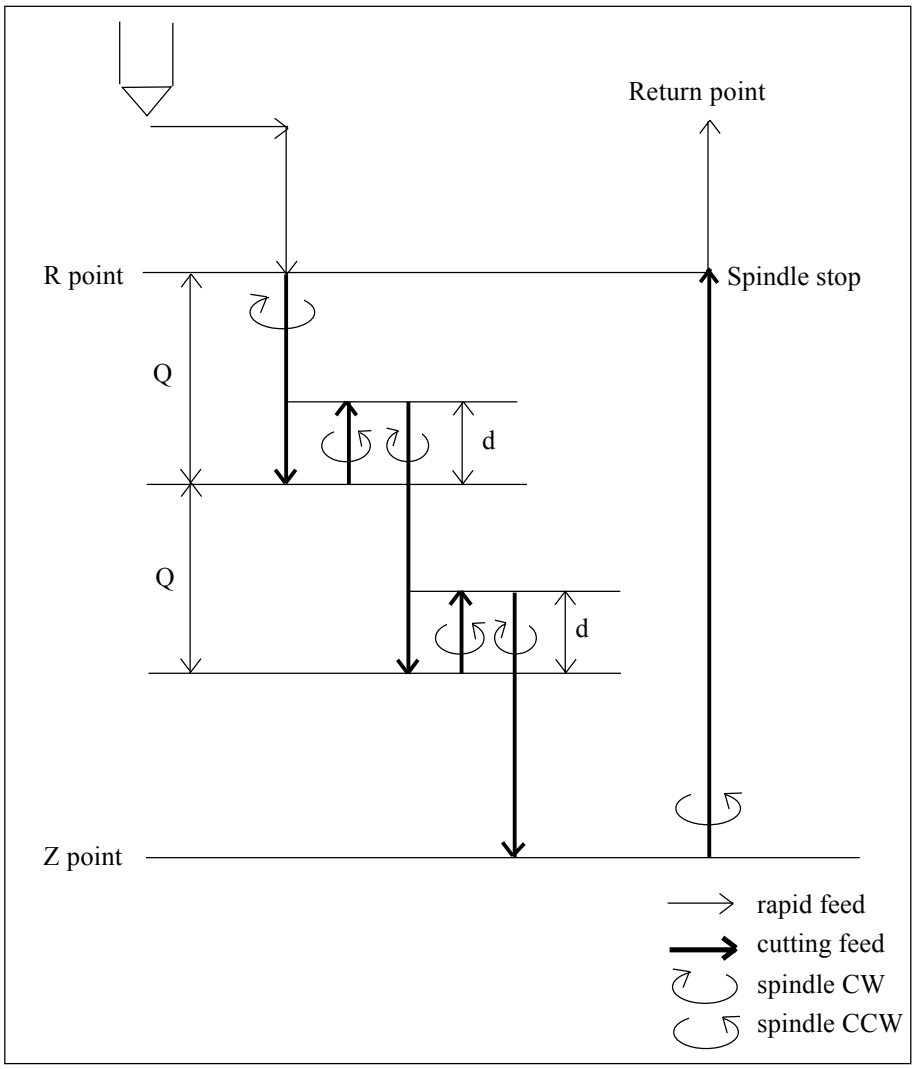
The shift direction is selected from +X, -X, +Y and -Y set by the parameter 1 in advance.

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.

5.4.4 Tapping cycle (G77)

Command format

G77 X _ Y _ Z _ R _ { I _ / J _ } Q _ S _ ;



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.

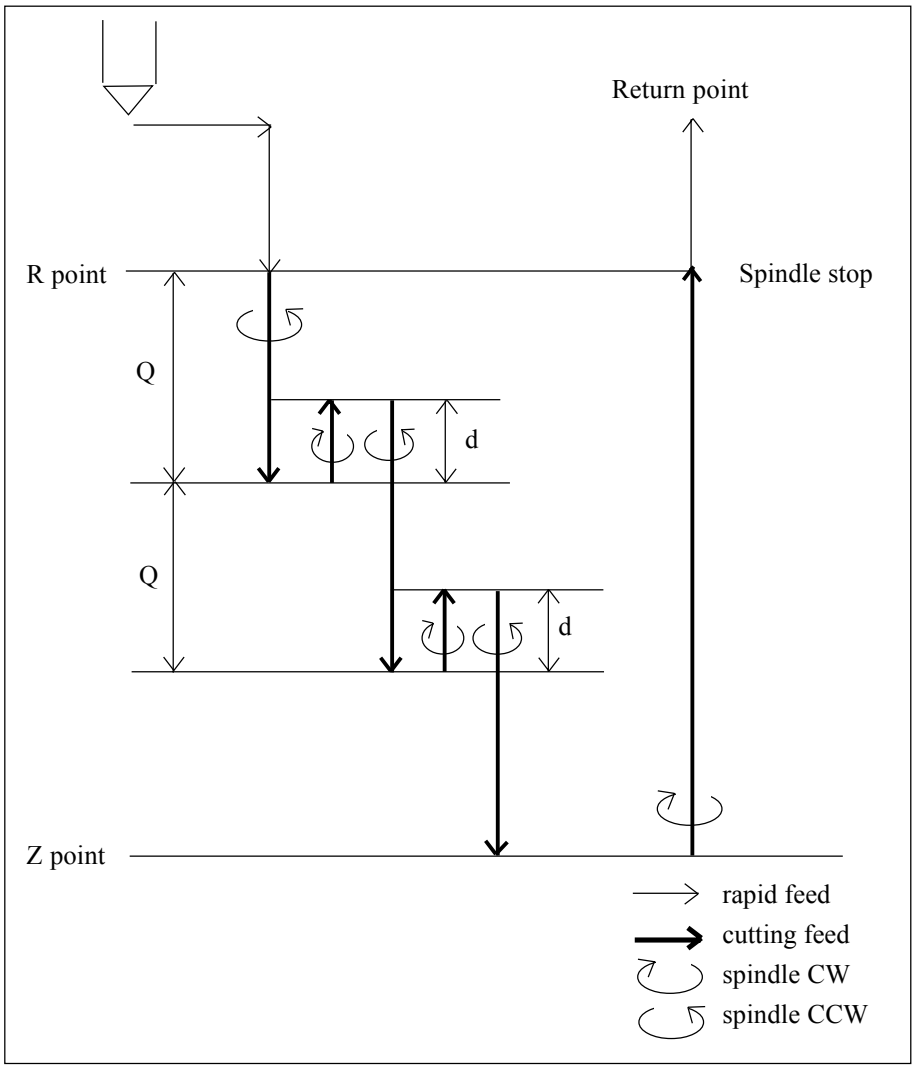
A thread pitch or number of threads should be specified.
Set the data on a thread pitch following the address I, and the data on a number of threads following the address J.

If the address S exceeds the max, spindle speed in tapping, an alarm is generated.

5.4.5 Reverse tapping cycle (synchro mode) (G78)

Command format

G78 X _ Y _ Z _ R _ { I _ / J _ } Q _ S _ ;



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.

Tapping high-speed return

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

Command format

```
{G77}                  {I _}
{   }  X _ Y _ Z _ R _ Q _{   }  S _ L _ ;
{G78}                  {J _}
```

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.

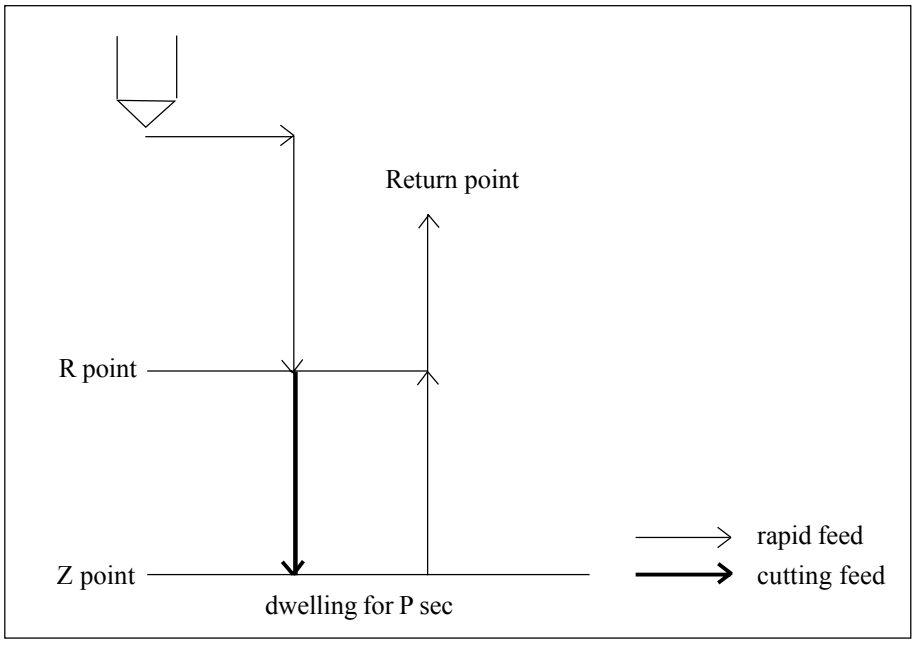
If the address L command value is larger than the max, spindle speed in tapping, an alarm is generated and the tool stops at the R point.

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}
{   }  X _ Y _ Z _ R _ P _ F _ ;
{G82}
```



High speed cycle

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

Command format

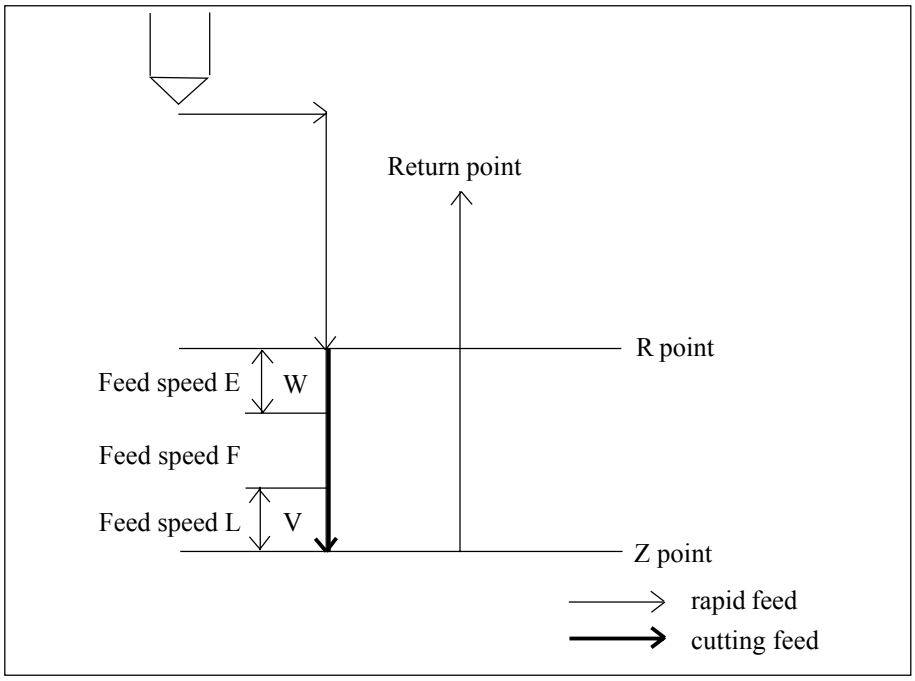
G81
G82
X _ Y _ Z _ R _ W _ V _ F _ E _ L _ ;

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)

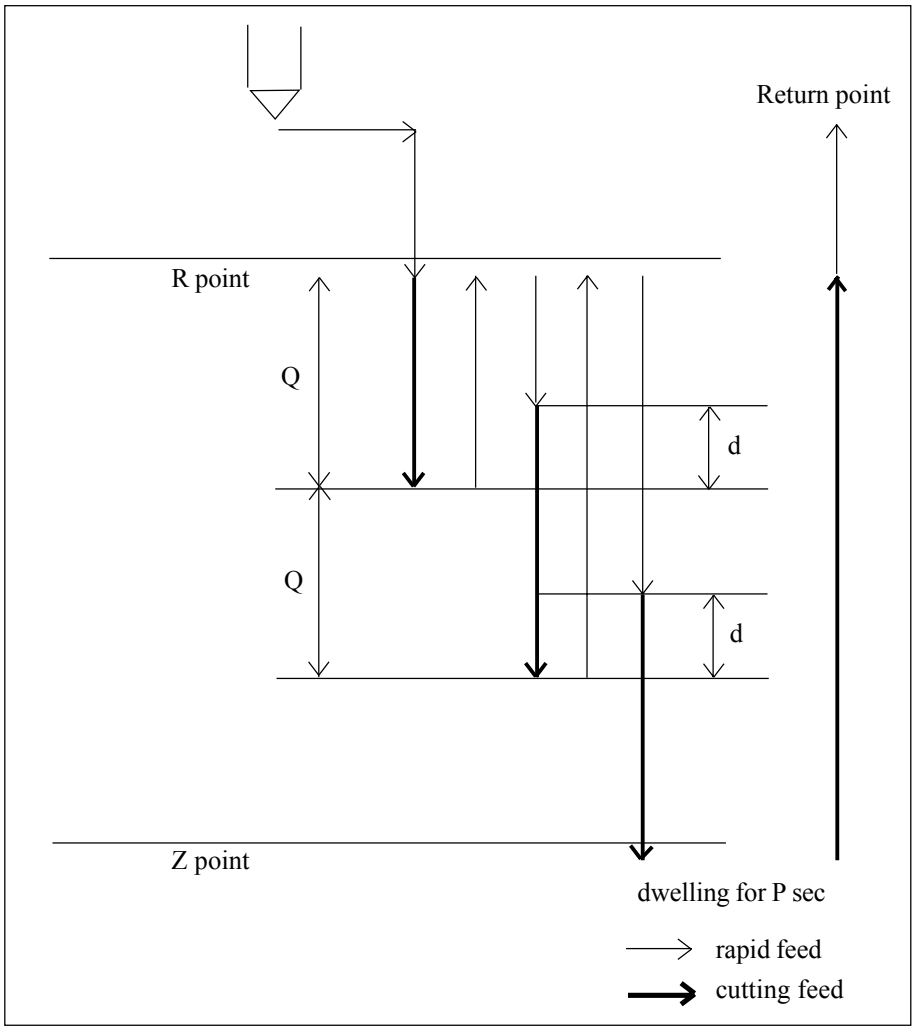
L : Feed speed from point "Z" to point specified by "V"



5.4.7 Peck drilling cycle (G83)

Command format

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



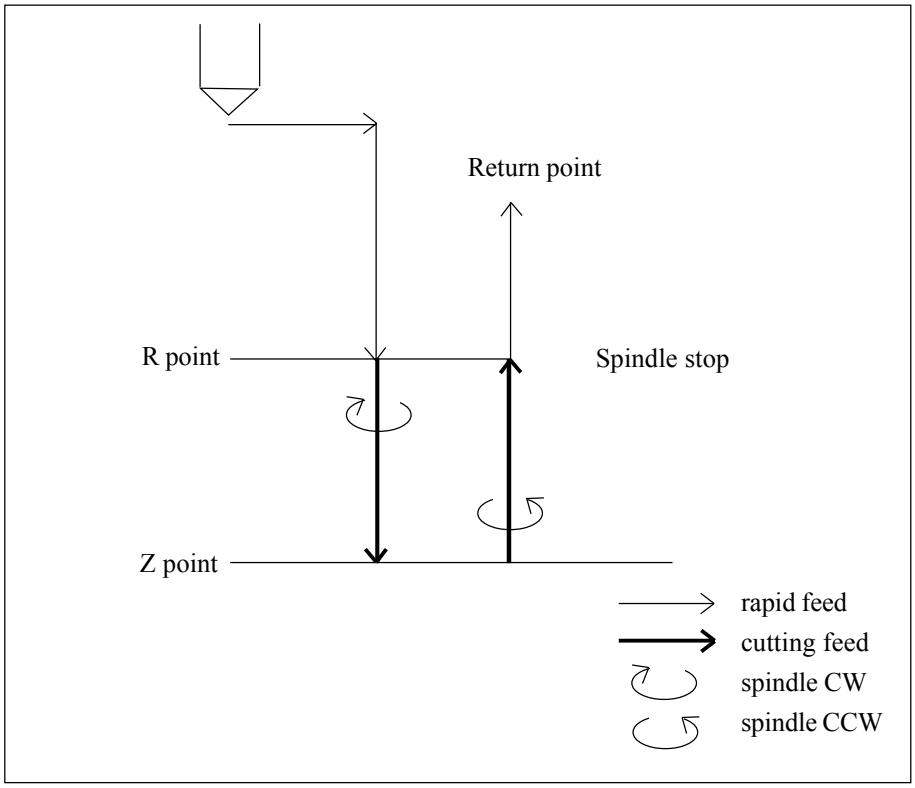
The cutting start point d is set by the parameter.

If the minus value is commanded for the cutting amount Q, the algebraic mark (-) is ignored.

5.4.8 Tapping cycle (G84)

Command format

G84 X _ Y _ Z _ R _ P _ F _ S _ ;



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.

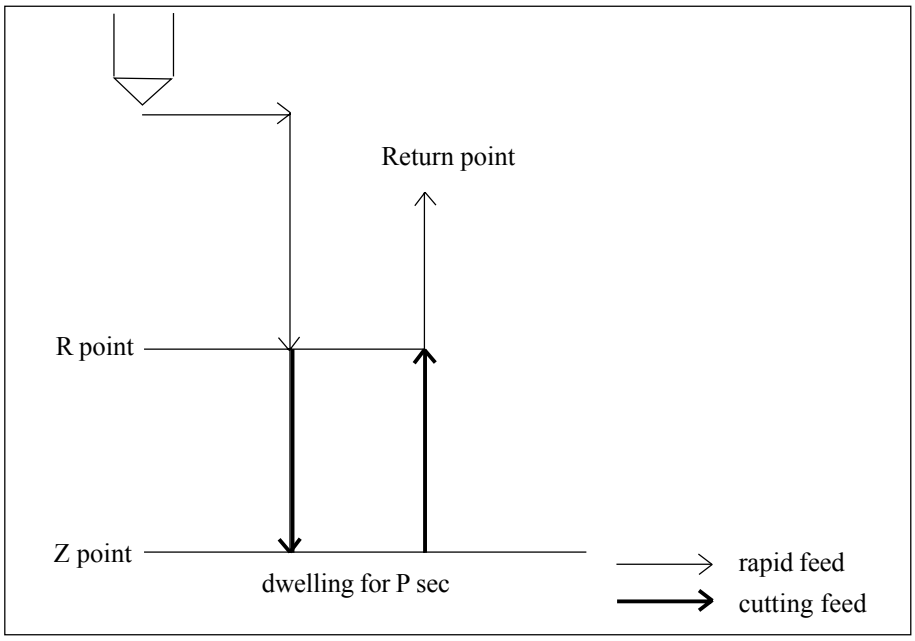
If the address S exceeds the max, spindle speed in tapping, an alarm is generated.

5.4.9 Boring cycle (G85, G89)

Command format

G85
G89

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



High speed cycle

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

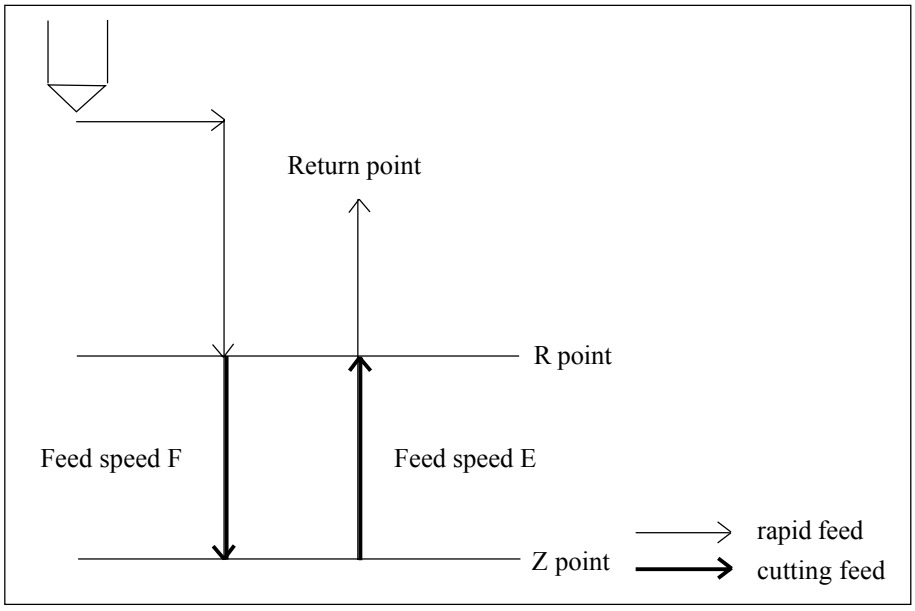
Command format

G85
G89

X _ Y _ Z _ R _ F _ E _
;

F : Feed speed from point "R" to point "Z"

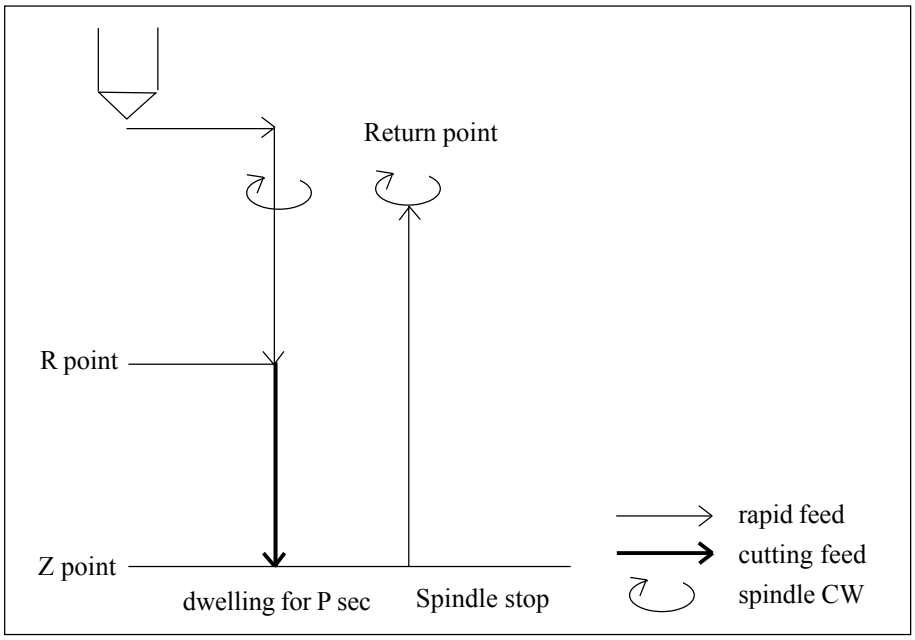
E : Feed speed from point "Z" to point "R"



5.4.10 Boring cycle (G86)

Command format

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



5

High speed cycle

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

Command format

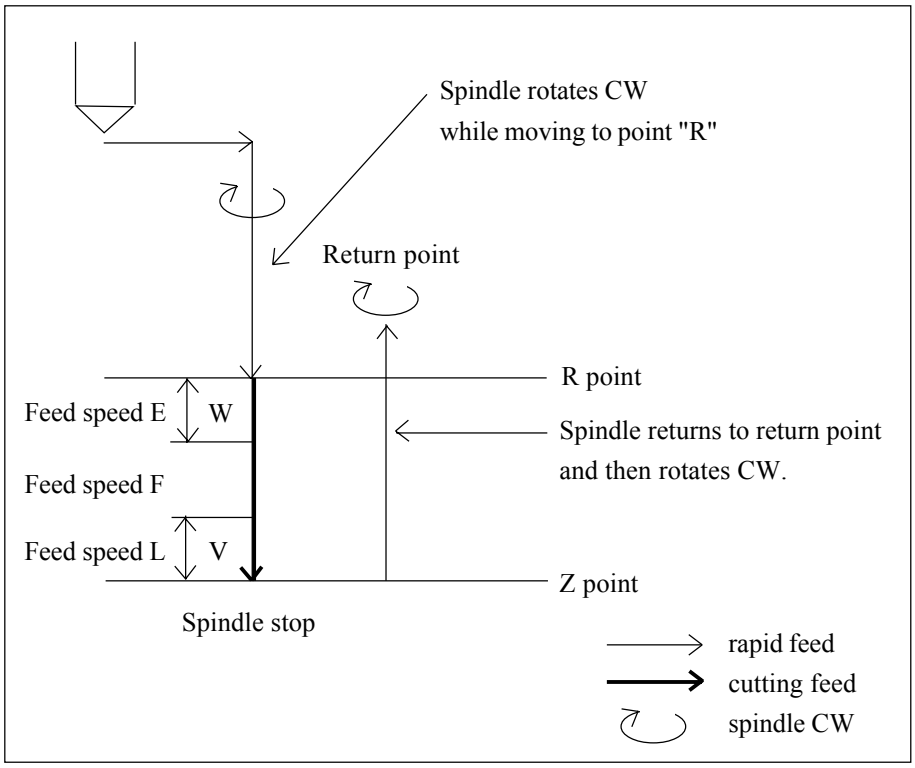
G86 X _ Y _ Z _ R _ W _ V _ F _ E _ L _ ;

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)

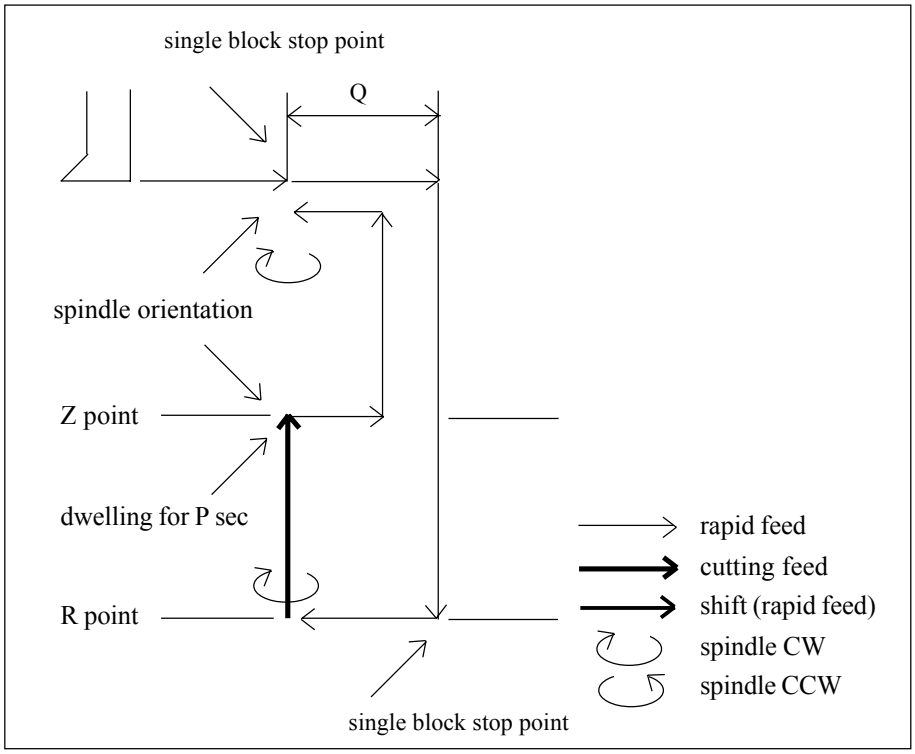
L : Feed speed from point "Z" to point specified by "V"



5.4.11 Back boring cycle (G87)

Command format

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



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.

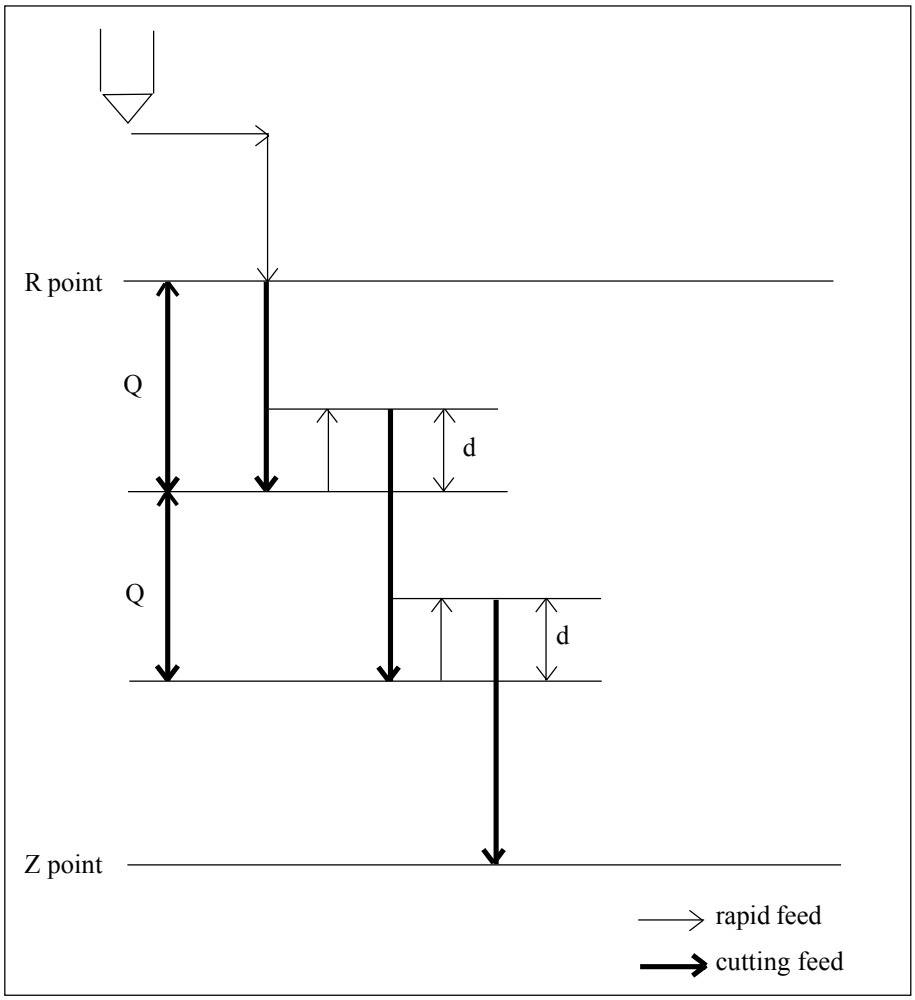
The shift direction can be selected only from the above four. Therefore, the tool should be mounted so that the tool nose faces in one of the specified direction when the spindle orientation executes.

G99 (R point level return) is unused.

5.4.12 High speed peck drilling cycle (G173)

Command format

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



5.4.13 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.

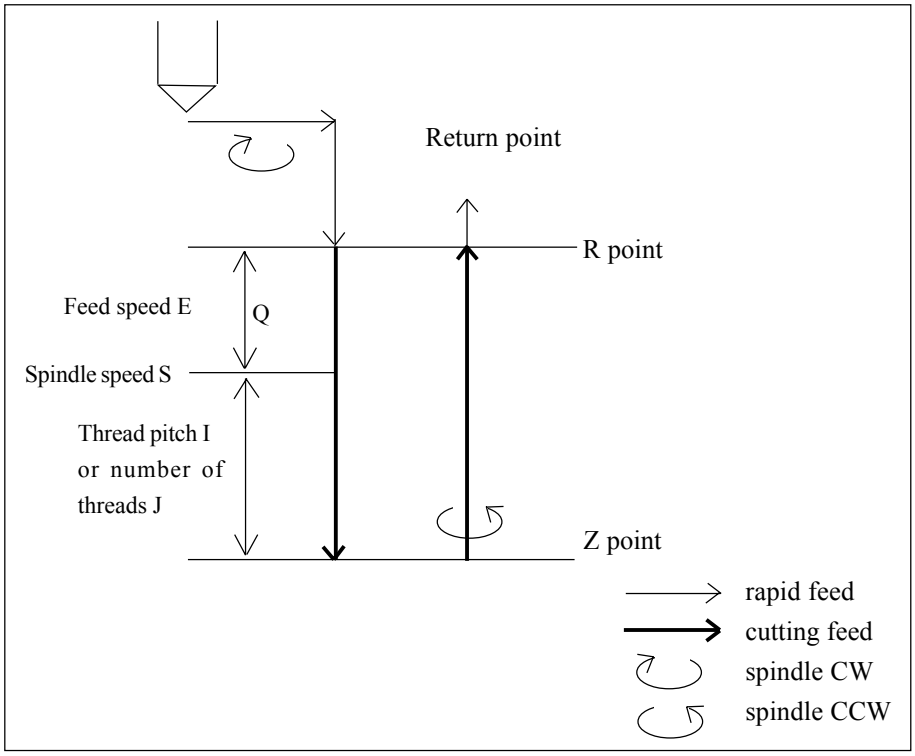
E : Feeding speed in zone "Q"

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.



5.4.14 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.

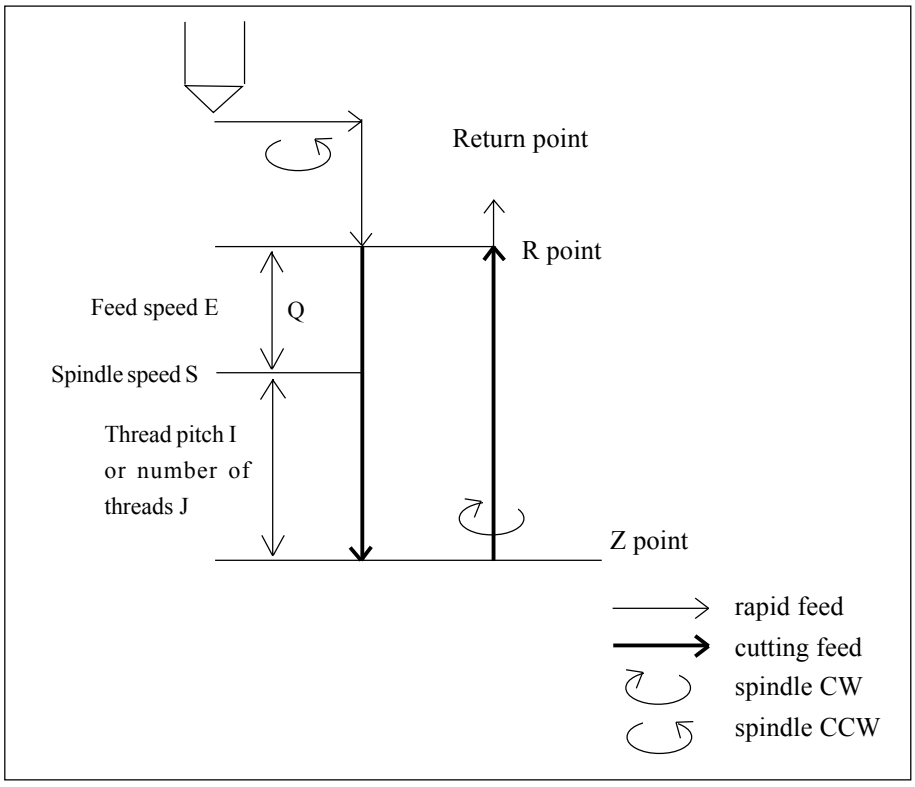
E : Feeding speed in zone "Q"

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.



5.4.15 Double drilling cycle (G181,G182)

Command format

```
G181
     X _ Y _ Z _ R _ I _ J _ W _ V _ F _ E _ L _ ;
G182
```

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

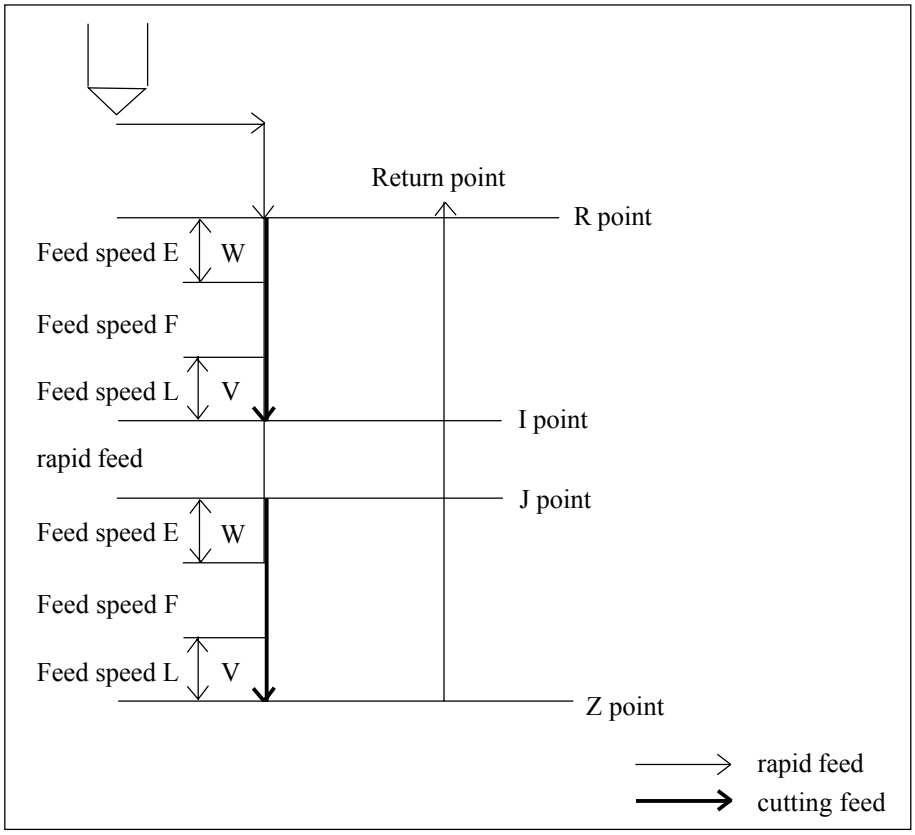
J : Double cutting feed start point (follow G90/G91)
Distance from point "I" when incremental mode is specified

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".

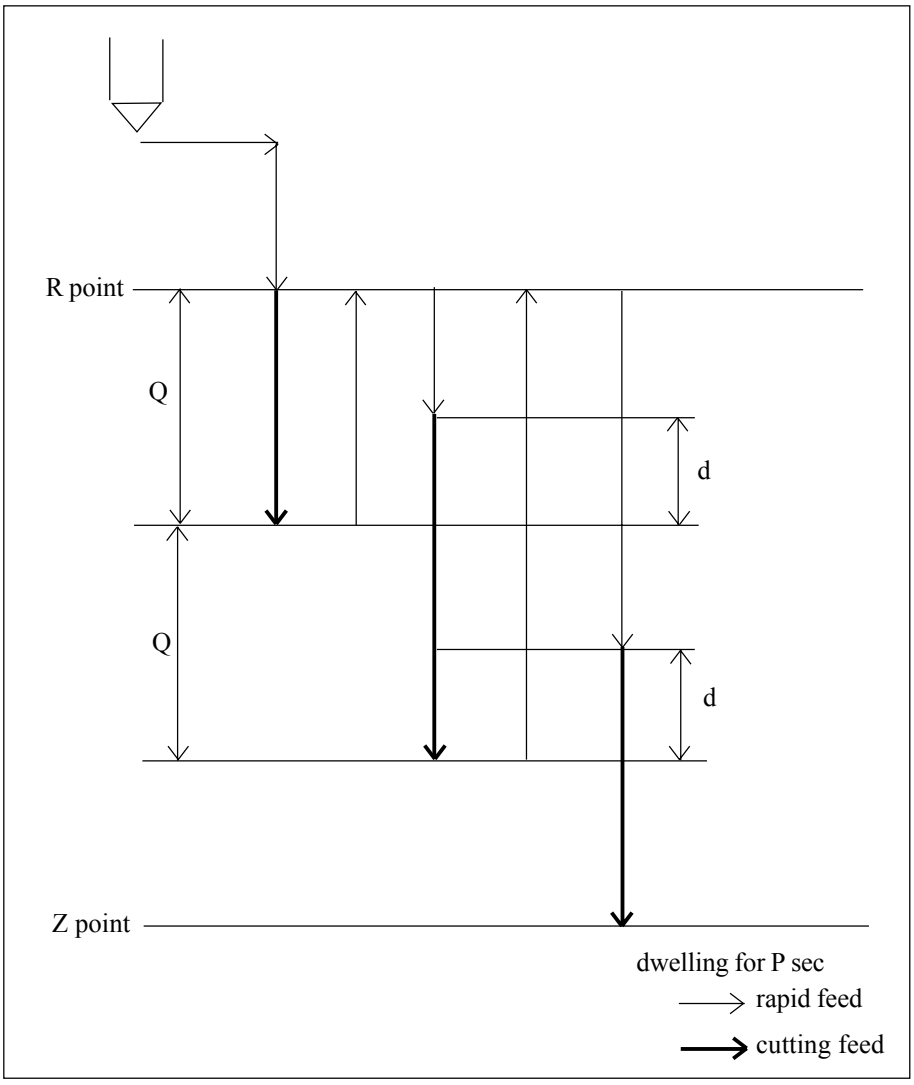


5.4.16 Peck drilling cycle (G183)

Command format

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

This is cycle where return operation is removed from G83.



5.4.17 Double boring cycle (G185,G189)

Command format

G185
G189

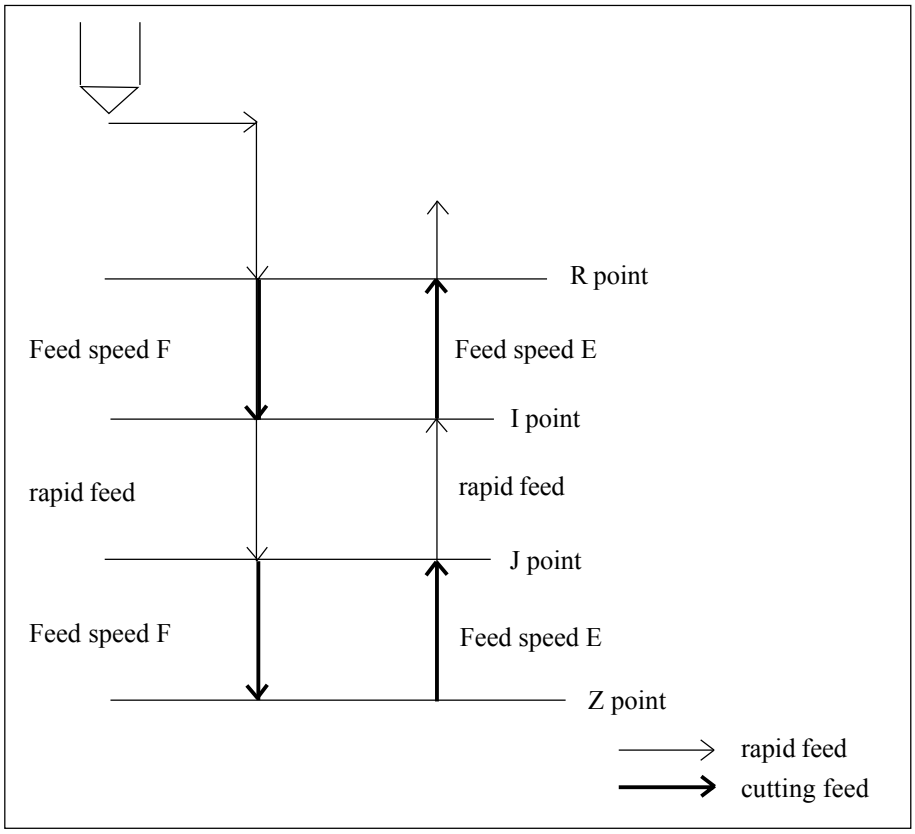
X _ Y _ Z _ R _ I _ J _ F _ E _ ;

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"



5

5.4.18 Double boring cycle (G186)

Command format

G186 X _ Y _ Z _ R _ I _ J _ W _ V _ F _ E _ L _ ;

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

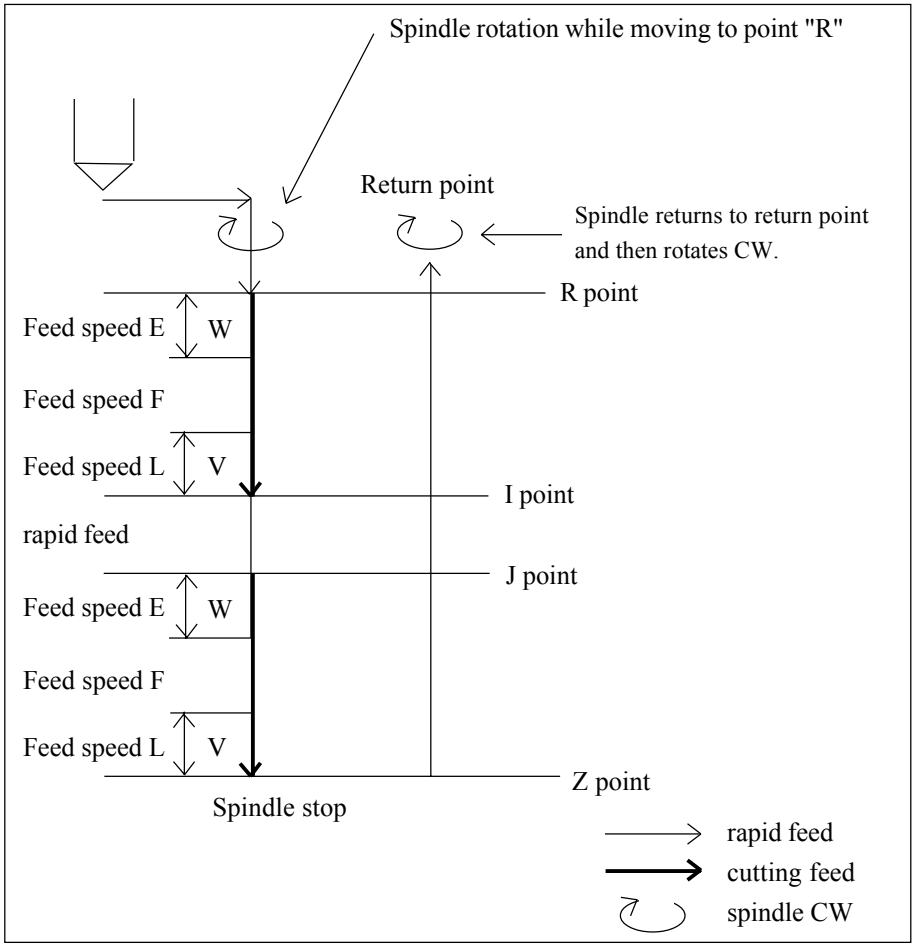
J : Double cutting feed start point (follow G90/G91)
Distance from point "I" when incremental mode is specified

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"



5.4.19 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.

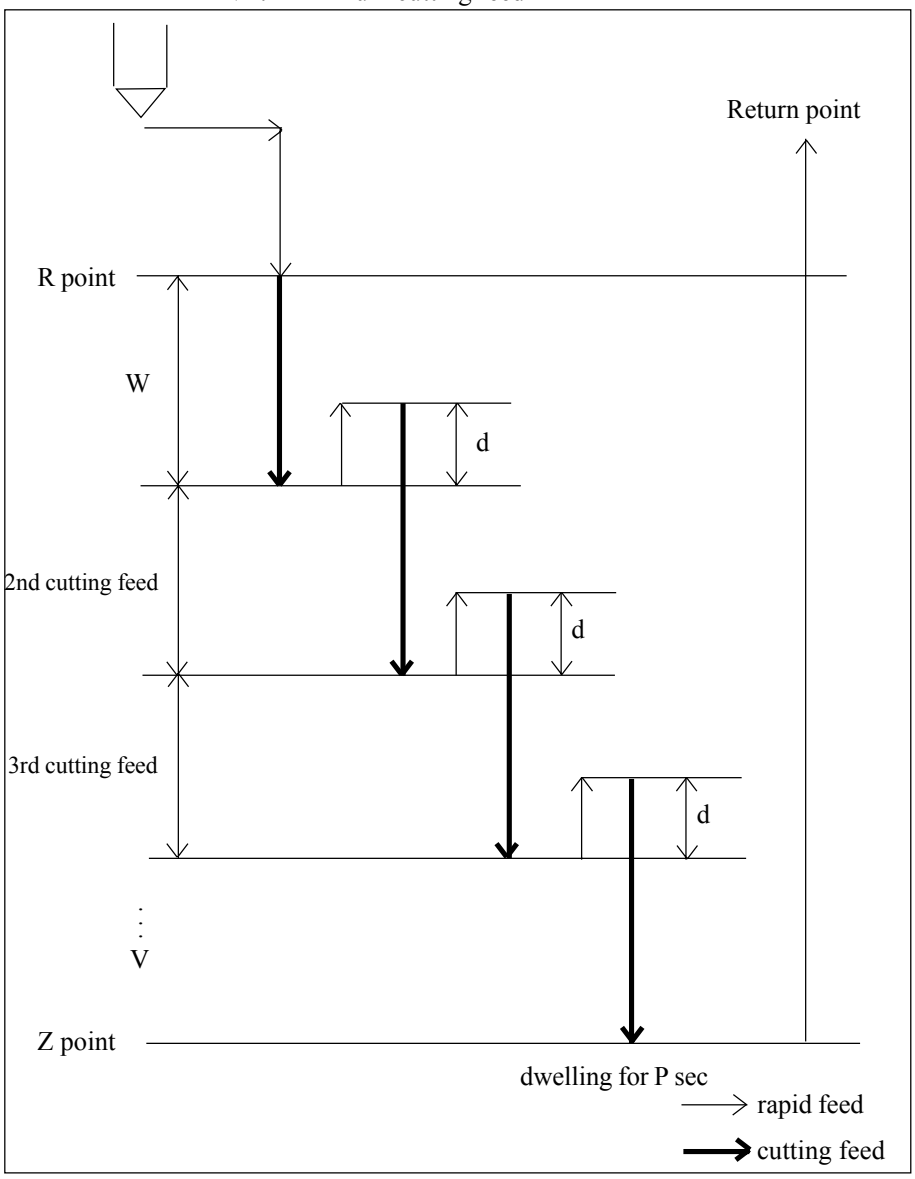
(1) High-speed peck drilling cycle (G73) (Reducing step)

Command format

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

W : 1st cutting feed

V : Minimum cutting feed



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.

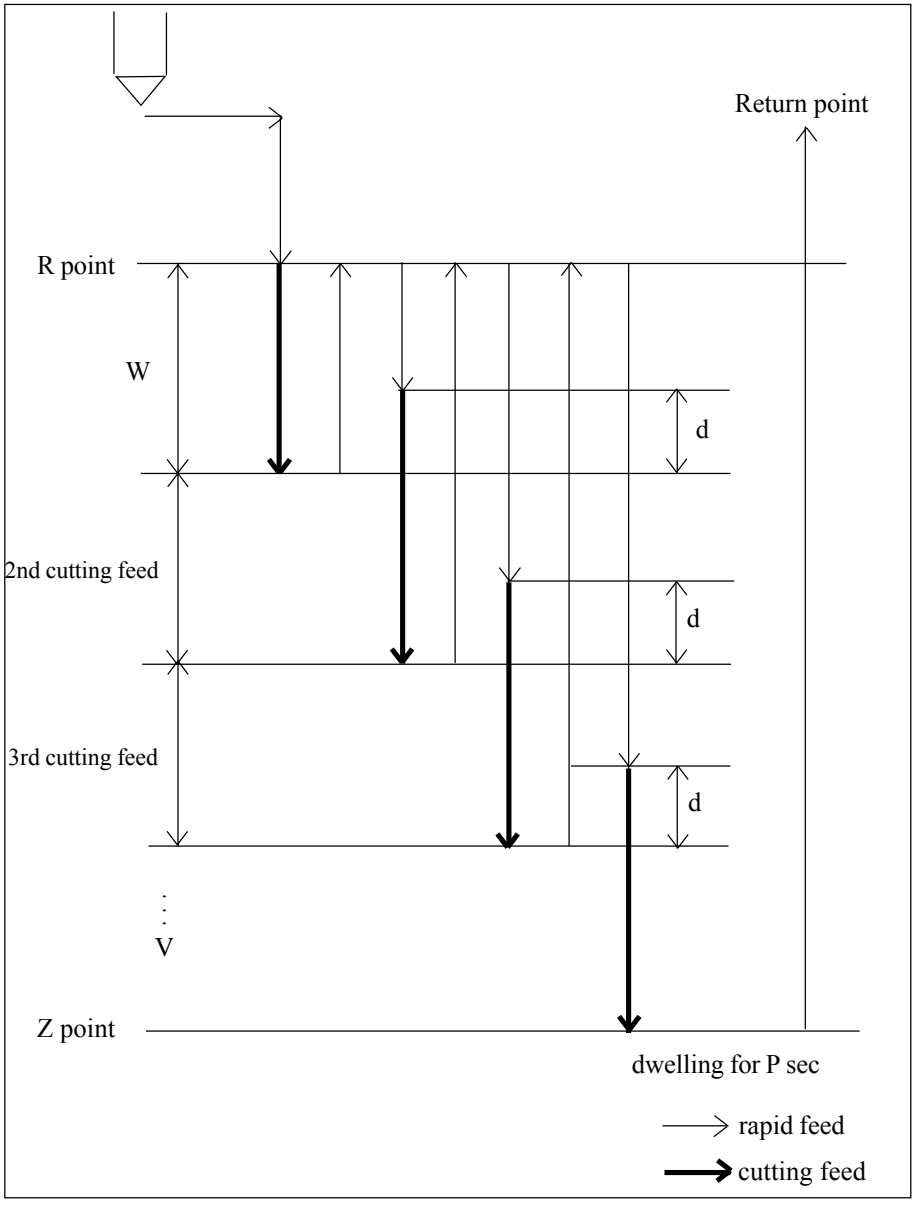
(2) Peck drilling cycle (G83) (Reducing step)

Command format

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

W : 1st cutting feed

V : Minimum cutting feed



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.

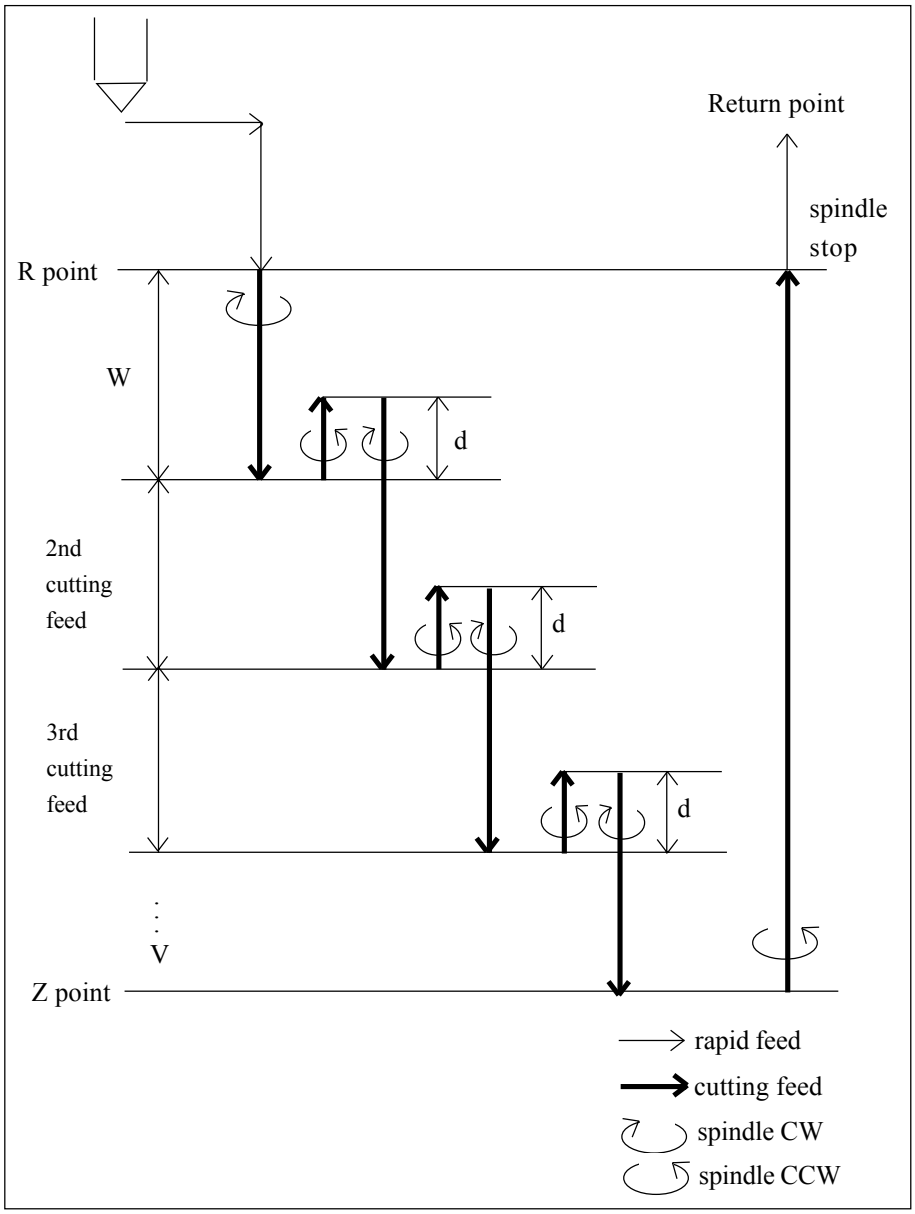
(3) Tapping cycle (synchro mode)(G77) (Reducing step)

Command format

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

W : 1st cutting feed

V : Minimum cutting feed



- 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.

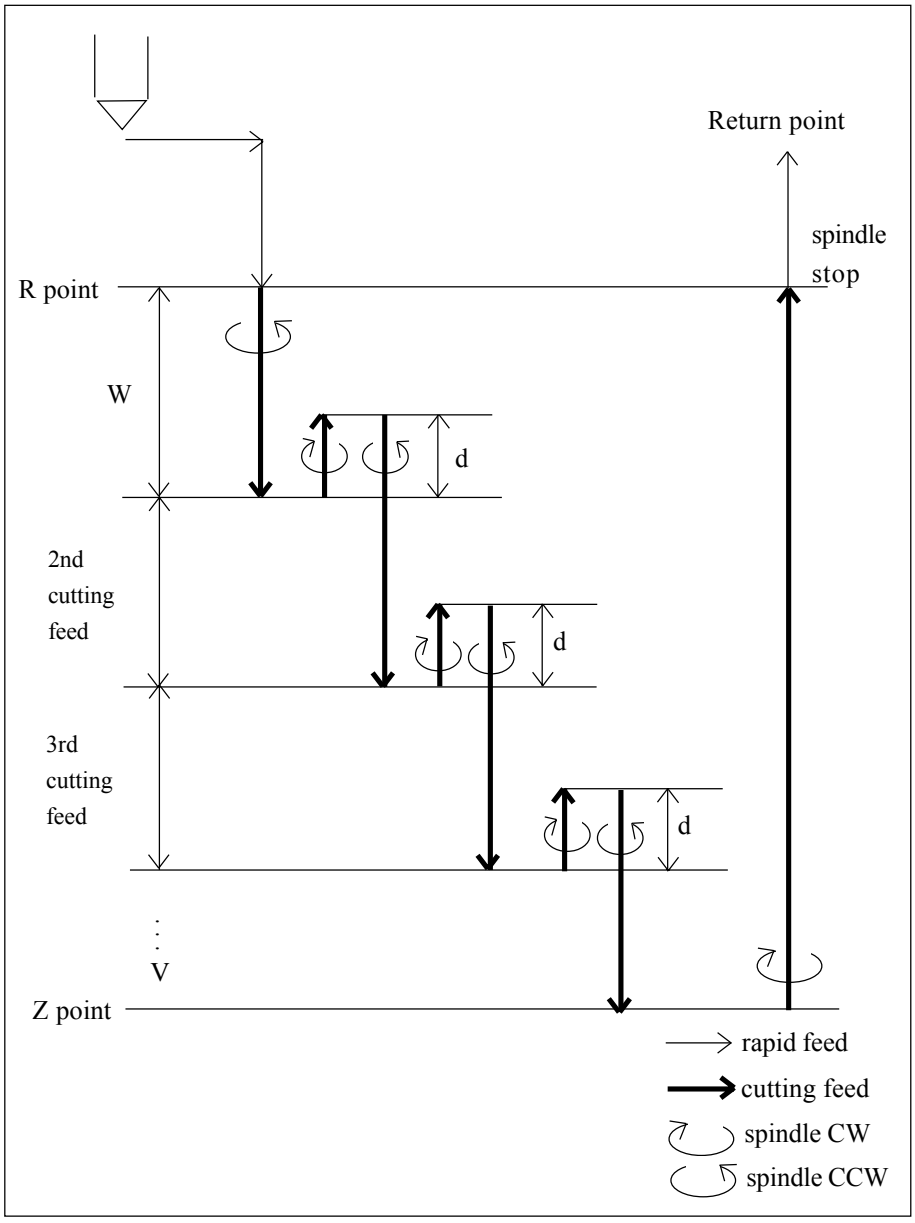
(4) Reverse tapping cycle (synchro mode)(G78) (Reducing step)

Command format

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

W : 1st cutting feed

V : Minimum cutting feed



· 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.

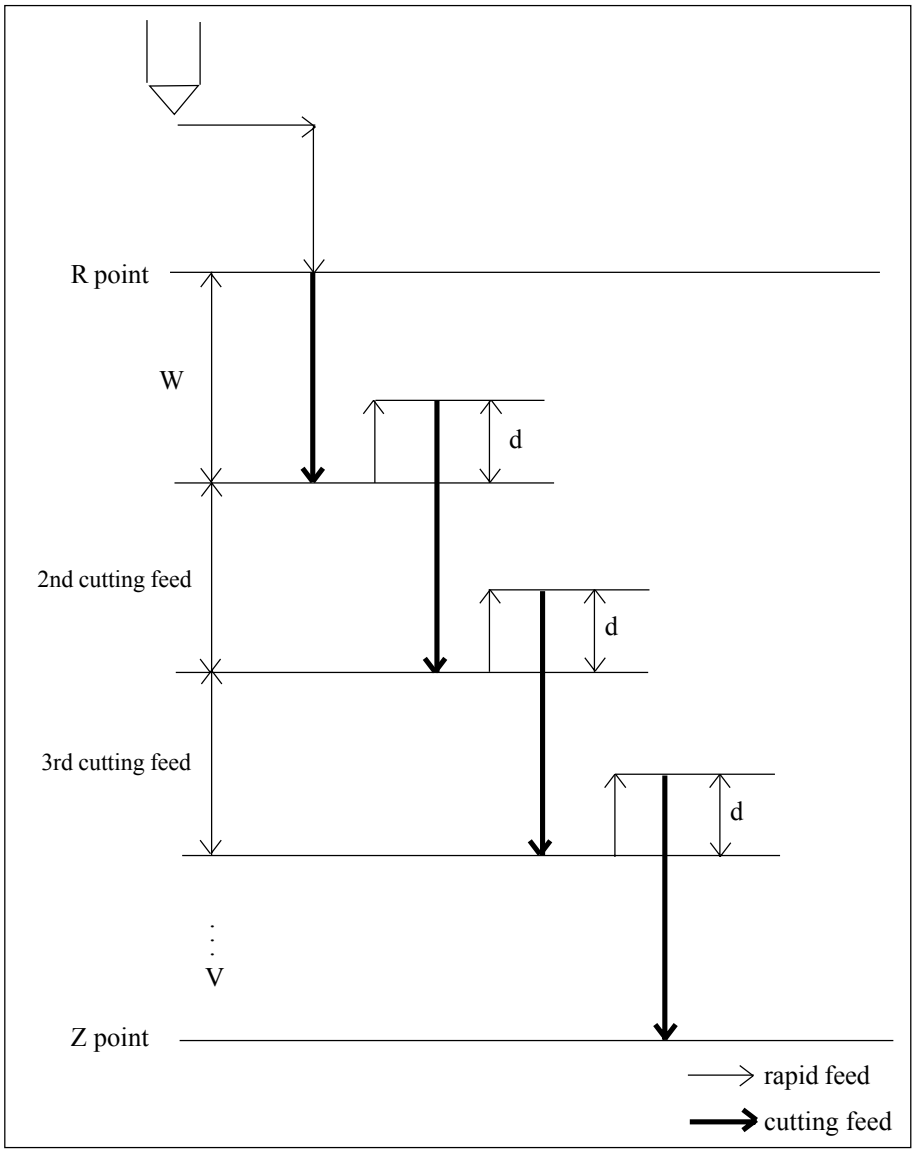
(5) High-speed peck drilling cycle (G173) (Reducing step)

Command format

G173 X _ Y _ Z _ R _ W _ V _ F _ ;

W : 1st cutting feed

V : Minimum cutting feed



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.

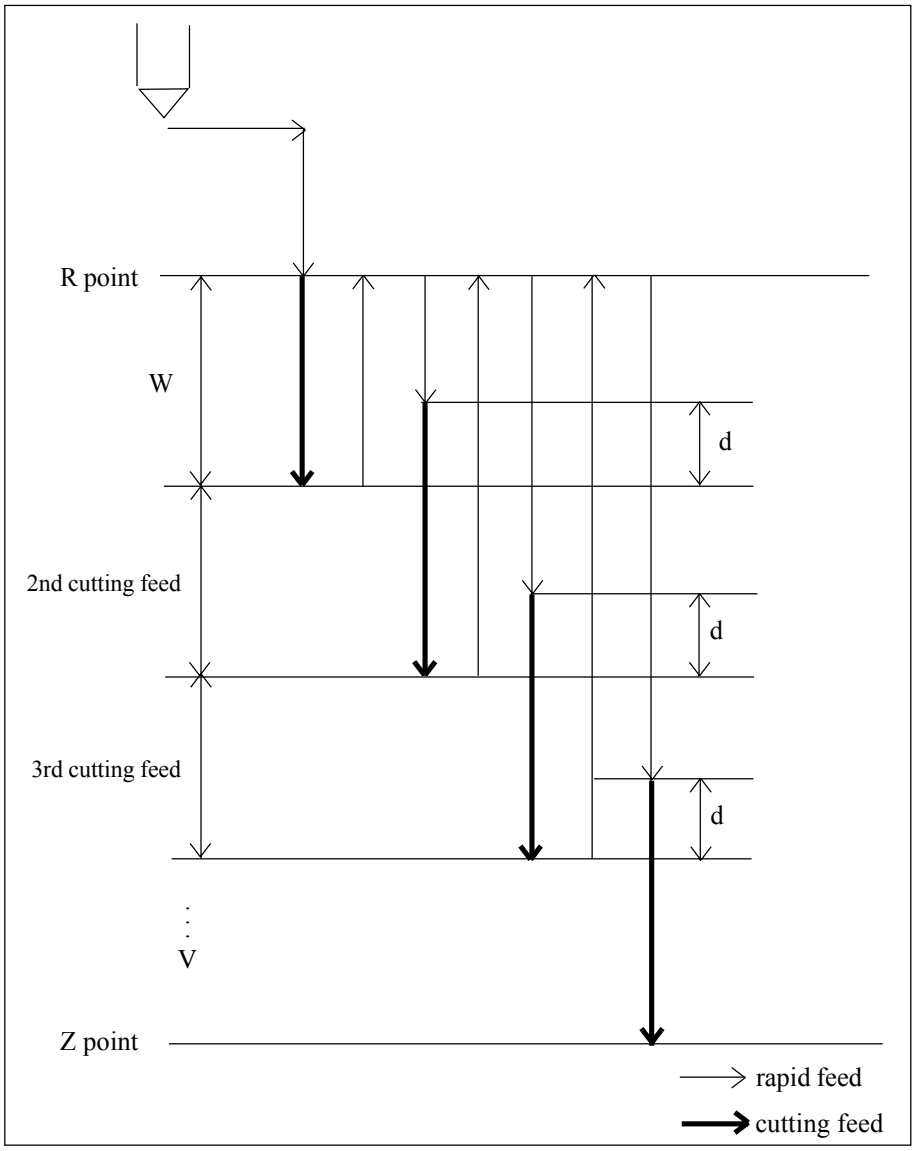
(6) Peck drilling cycle (G183) (Reducing step)

Command format

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

W : 1st cutting feed

V : Minimum cutting feed



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.

(Note 4) When the V command is not given, or 0 is given, V=0.001 (metric) or V=0.0001(inch) is used.

5.4.20 Canned cycle cancel (G80)

The canned cycle (G73, G74, G76 to G78, G81 to 87, G89, G173, G177, G178, G181 to G183, 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.

5.4.21 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 spindleshould 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.

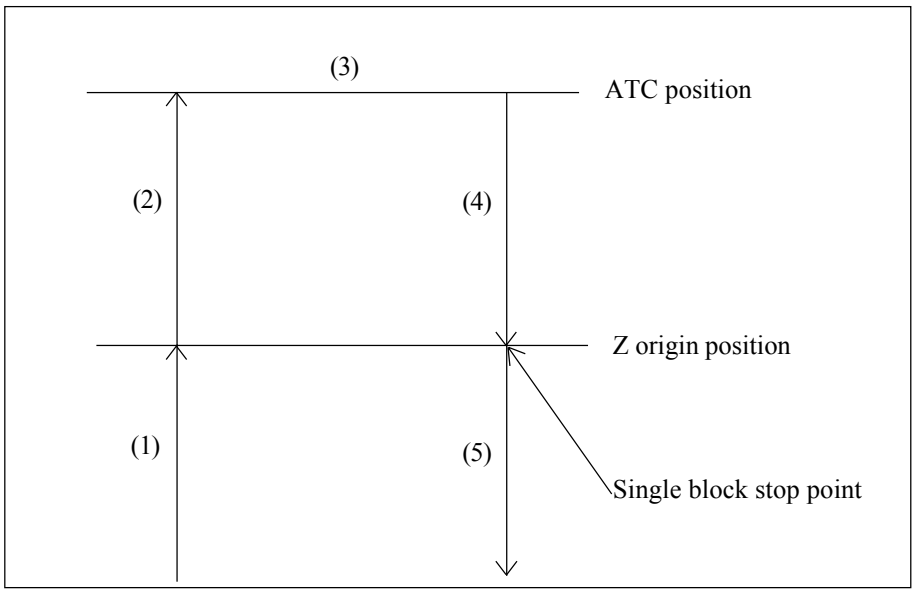
(5) M200, M201, and M120 cannot be commanded in the same block as that of the canned cycle.

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

(1) When TC-31A or TC-22A.

Command format

G100 T _ X _ Y _ Z _ R _ A _ B _ L _ ;



T _ _	:	Tool number.
T1 _ _	:	Pot number.
T9	:	Group number.
X, Y, A, B	:	Target value when moving X, Y, A, and B axes at in rapid feed simultaneously with tool change motion.
Z	:	Target value of movement 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 (3) after ATC. (Next tool preparation)

Operations

(1) Tool moves to the Z-axis point "R" while performing 0-degree spindle orientation.

(2) Tool moves to theZ-axis ATC origin while X,Y,A and B axes move to commanded value.
When "T" is commanded, the tool moves to Z-axis ATC origin and magazine swivels.

(3) Arm swivels to change tool.
Tool change motion differs depending on setting on **[MAGAZINE TOOL]** screen.
Refer to the "Tool Change Motion" for detailes.

(4) Tool moves to the Z-axis point "R".
Operation, if specified by spindle command, occurs simultaneously.

(5) Tool moves to Z-axis commanded position.

5

Caution

- **When performing cycle operation, tool moves in cutting mode between operations (1)and (2), operations (4) and (5).**
- **When [RESET] key or [STOP SWITCH] key is pressed between operations (2) and (4) , machine stops after motion (4) is completed. For operation (3) (X, Y, A, and B axes movement), machine stops immediately.**
- **Machine does not stop if single block occurs between operations (1) and (4).**
- **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 (2) 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 (4) .
- **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 (4) 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.
  When point "R" command position (4) is lower than Z axis command position (5), tool moves to Z-axis commanded position, and operation (5) 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.
Operation (9) is performed for the TC-31A only. However, it may also be performed for the TC-22A when "next tool preparation" is performed.

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 (not performed for TC-31A)

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 (not performed for TC-31A)

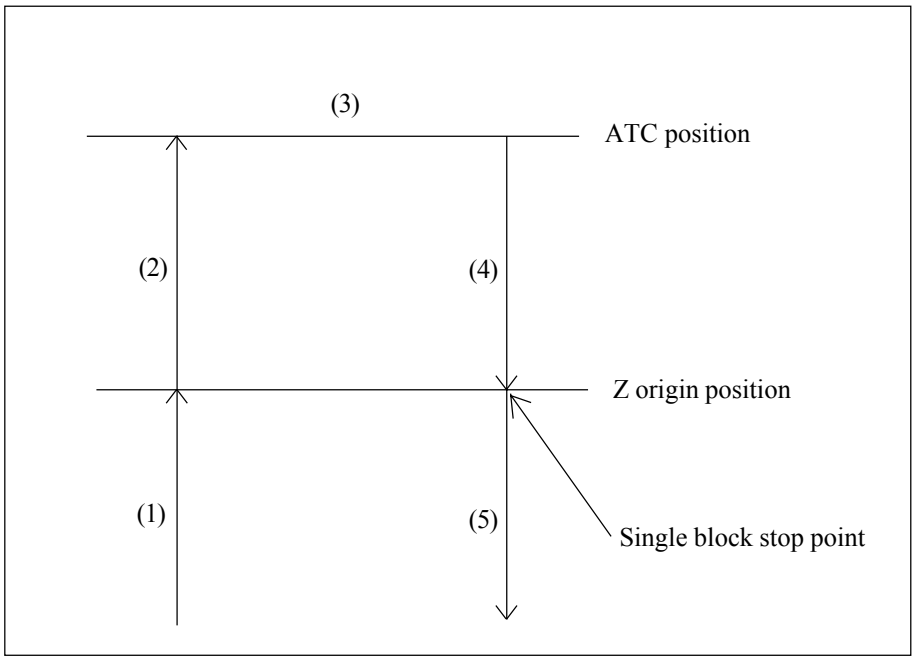
Next tool preparation ends when the above sequence is completed.
The sequence starts from (2) when the pot is already at the upper limit before next tool preparation commences.

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.

(2) When TC-S2A, TC-R2A or TC-S2B.

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



T _ _	:	Tool number.
T1 _ _	:	Magazine number.
T9 _ _	:	Group number.
X, Y, A, B	:	Target value when moving X, Y, A, and B axes at in rapid feed simultaneously with tool change motion.
Z	:	Target value of movement(5) in rapid feed. Command the distance from the Z origin when the incremental mode is selected.
L	:	Specify the tool number, magazine number, and group number after “L”. The number specified by “L” becomes the T model afterG100.

Operations

(1) Tool moves to the Z-axis point and spindle orientation are executed at the same time.
(2) Movement to the ATC origin position.
When "X, Y, A or B" is commanded in the same block, the motions are done at the same time.
(3) The magazine indexes the tool as commanded by the address "T".
(4) Movement to the Z-axis origin position.
(5) Movement to the commanded position when Z-axis is commanded.
When the spindle command (M3 group) is given, the motion is done at the same time.

· When performing cycle operation, tool moves in cutting mode between operations (1) and (2)and operations (4) and (5).

· When **[RESET]** key or **[STOP SWITCH]** key is pressed between operations (2)and (4), machine stops after motion (4) is completed.
For operation (2) (X, Y, A, and B axes movement), machine stops immediately.

Caution

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

· The tool length offset and the tool dia offset are canceled when G100 is commanded.

· 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 (2) commences. Start-up is performed in format 1, regardless of parameters.

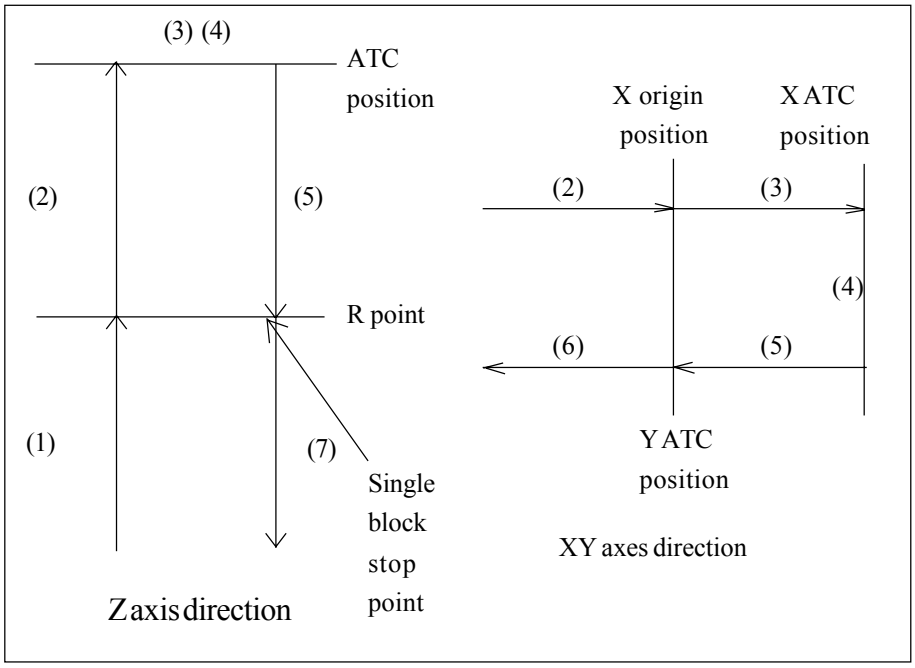
· When tool offset is commanded to block G100, tool offset becomes valid from operation (5).

· 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.

(3) When TC-32A.

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



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.

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]**

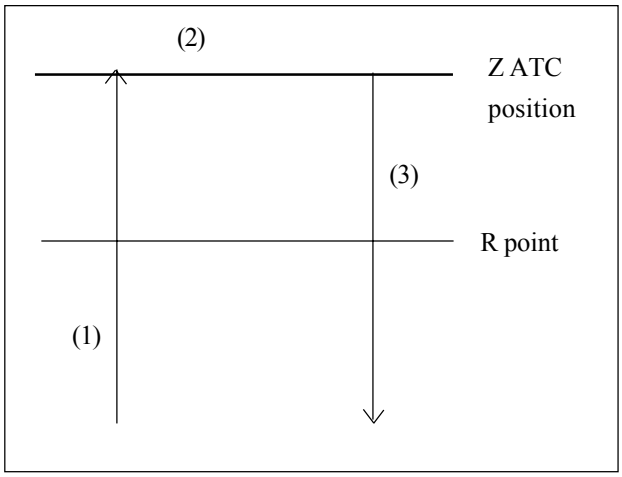
- Both spindle tool and next tool are standard or large
(1) Pot rises
(2) Magazine swivels
(3) Pot lowers
Next tool preparation ends when the above sequence is completed.
The sequence starts from (2) when the pot is already at the upper limit before next tool preparation commences.

- 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 stan dard 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.

(4) When TC-20A.

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



T :T__ :Tool number
:T1__ :Pot number
:T9__ :Group number

X,Y,A,B: Target value when moving X, Y, A, and B axes in rapid feed simultaneously with ATC

Z : Target value of operation (3) in rapid feed

R : Return position before ATC (with tool length compensation applied)
The "Z ATC position" is regarded as "R command value + 50 mm".
When the "R" command is not specified, operation is performed regarding "Z origin - 50 mm" as "R" command value.

L : Specify the tool number, pot (magazine) number, and group number after "L".
The pot with the corresponding tool is indexed by operation (2) after ATC. (Next tool preparation)

Operations

1)The tool moves to Z ATC position (50 mm above "R" point or Z origin) while performing 0-degree spindle orientation.
When the tool specified by "T" is indexed in the magazine, the pot lowers. It starts lowering when the Z-axis has passed the position set for **[POT LOWERING START POS.]** (user parameter).
2)When the tool specified by "T" is not indexed in the magazine, the magazine swivels and then the pot lowers.
When the specified tool is indexed in the magazine and the pot is at the lower limit, the arm swivels and then the pot rises.
X, Y, A, and B axes also move to the specified positions.
See the ATC section for details of ATC sequence.
3)The tool moves to the specified Z-axis position.

5

Precautions

- The pot, magazine, and arm do not stop even when the **[RST]** key or the **[STOP]** button is pressed during operations 1) to 3) above. The X, Y, Z, A, and B axes stop immediately.
- All data other than G100 can be omitted. However, a T code must be used at least once before G100 is used.
- When G100 is used, tool diameter compensation is canceled. Tool length compensation is also canceled in operation 3).
- When the tool diameter compensation (G41 or G42) and XY axes movement commands are used for the G100 block, tool diameter compensation begins in format 1 when X and Y axes movement 2) commences, regardless of the current parameters.
- The tool length compensation command is used for the G100 block, tool length compensation become valid from operation 3).
- The Z-axis moves as described below during operation 1).
  (1) When R point is specified for G100 block
  The Z-axis moves to the position where the tool tip is "R point + 50 mm" in accordance with the tool length compensation.
  The larger tool compensation value before or after change is applied.
  When tool length compensation is not applied, the setting for **[ATC REFERENCE TOOL LENGTH]** (user parameter) is used.
  (2) When R point is not specified for G100 block
  The tool rises to the Z origin, regardless of the tool length.

· When the Z-axis movement and tool length compensation commands are not used for the G100 block, operation 3) is performed in accordance with the setting for **[ATC REFERENCE TOOL LENGTH]** (user parameter).
· An alarm occurs when an M code other than a spindle command is used for the G100 block.
An alarm occurs when the A- or B-axis command is used although the optional A- or B-axis is not installed.
· The target position the pot lowers to is determined by the distance between **[ATC RETURN HEIGHT]** and **[POT LOWERING START POSITION]** (user parameters).

ATC

(1) Magazine swivels
(2) Pot lowers
(3) Arm swivels
(4) Pot rises
ATC ends when the above sequence is completed.
Operation (3) (arm swivels) is performed at high speed.
When the tool breakage detection command is specified for tools in the magazine, detection is performed after operation (4) (pot rises).

Next tool preparation

Next tool preparation is performed after the pot has risen in the ATC sequence.
When ATC is not performed, only next tool preparation is performed.
When the next tool is already indexed, next tool preparation is not performed.
1) Magazine swivels
Next tool preparation ends when the above sequence is completed.

(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 on 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

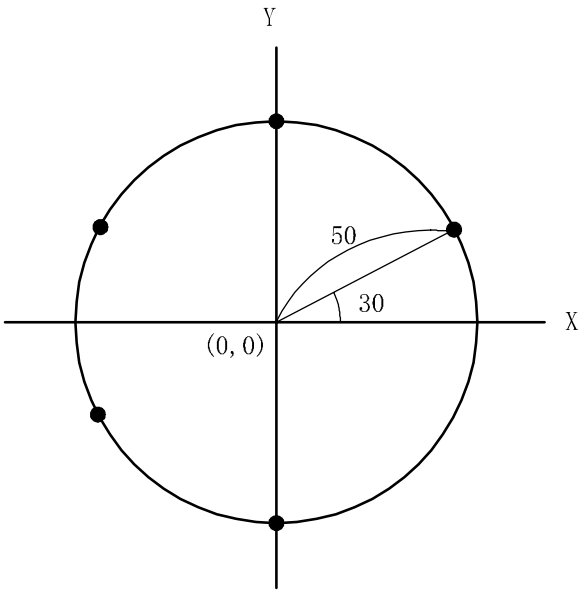
While setting the commanded coordinate value as a circular center, the coordinate value equally dividing a circular arc from a certain point is calculated.

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 ;



manual1:c:ai4.1j:6章図：6004-1.AI

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

6.3.2 Linear (angle)

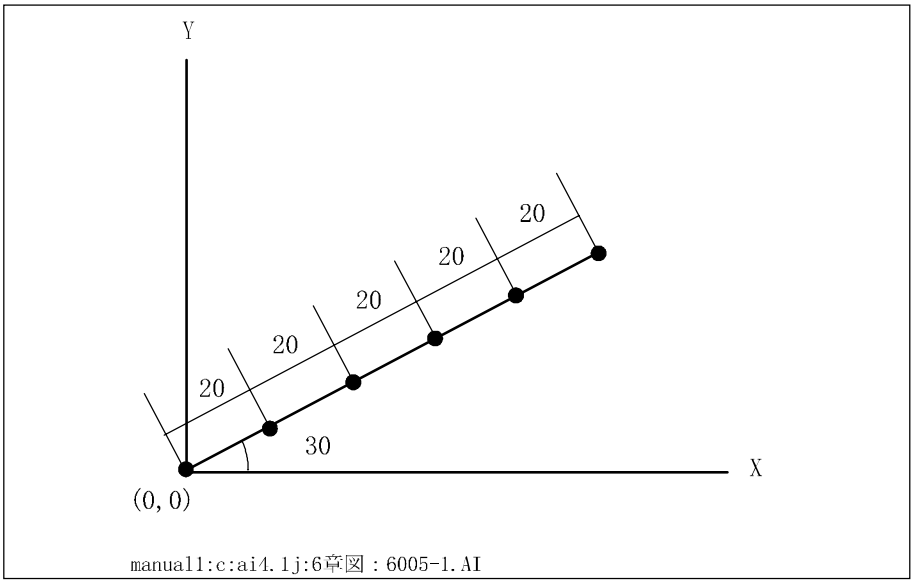
With the reference point at the commanded coordinate, the coordinate values along the linear line at the angle（θ°） formed by the X axis are calculated.

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)

(Ex.) G37 X0 Y0 I20 J30 K6 ;



(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)

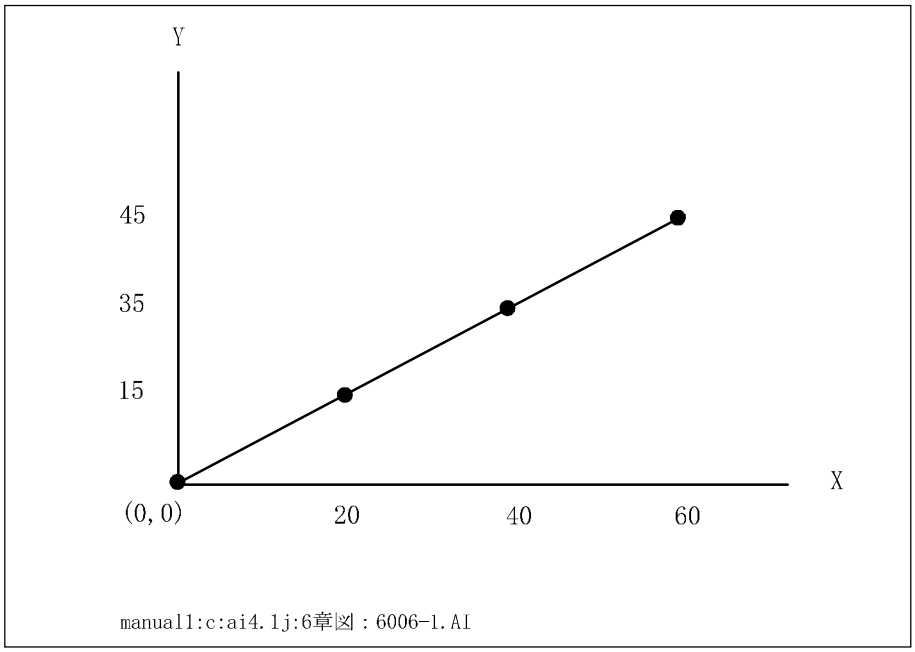
With the reference point at the commanded coordinate, the coordinate values added in the X and Y directions respectively are calculated.

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 ;



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

6.3.4 Grid

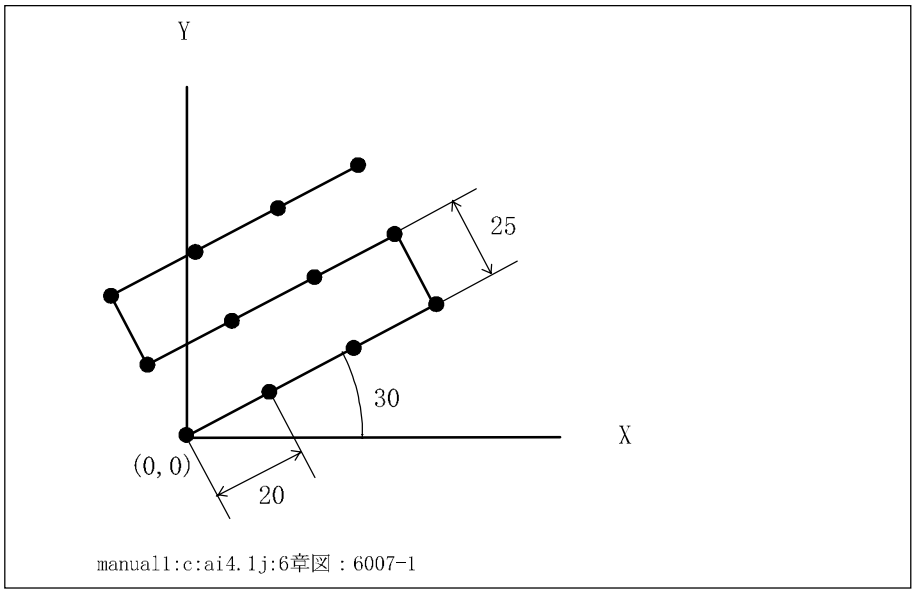
With the reference point at the commanded coordinate, the coordinate values of the grid composed of points arranged at an even interval and parallel to both the X-axis and the vertical axis are calculated.
By specifying an angle to the X axis, the total form can be inclined.

Command format

G39 X_ Y_ I_ J_ K_ P_ Q_ ;

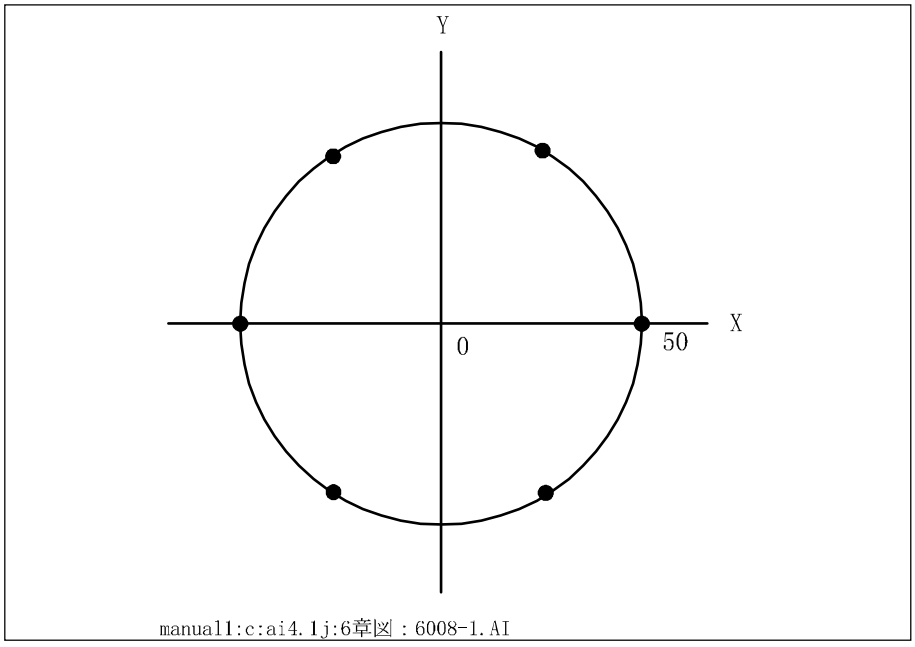
X,Y : Coordinate of reference point
I : Interval on the X axis
J : Interval on the Y axis
K : No. of machining point in the X direction (Max. 999)
P : No. of machining point in the Y direction (Max. 999)
Q : Angle intersecting the X axis

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



(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



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

7.1 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)
Ｎ５１ＩＦ[＃１００ ＬＴ １０]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.

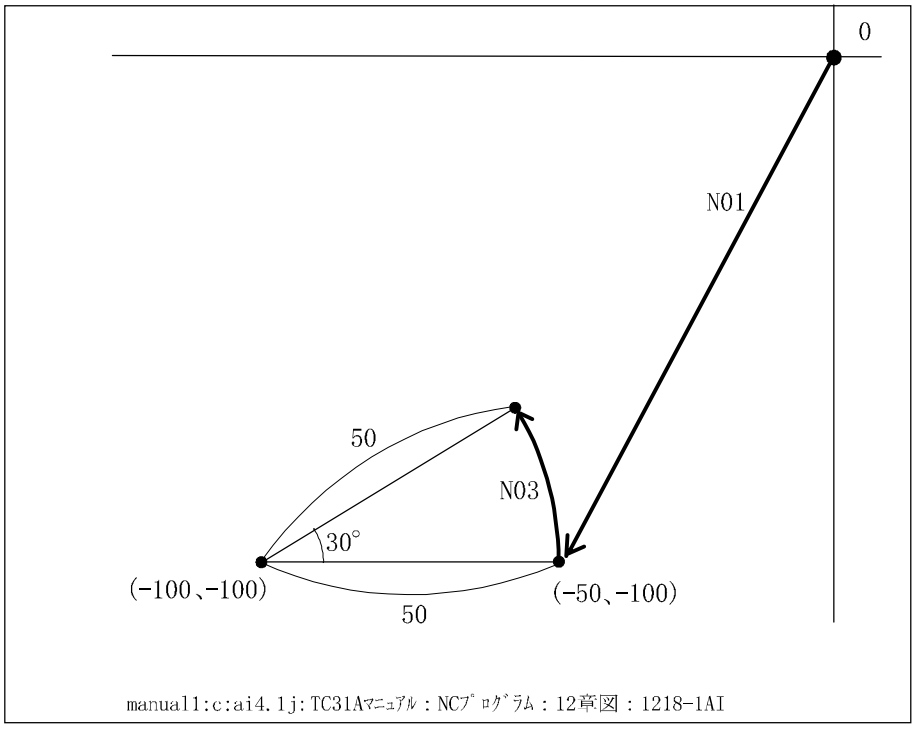
X: CENTER X Y: CENTER Y R: RADIUS Z: CUT POS Z
W: STOP BEFORE WORK U: CUT START ANGLE
V: CUT END ANGLE F: FEEDRATE

Main program

```
Ｎ０１Ｇ９０Ｇ５４Ｇ０Ｚ３０．；
Ｎ０２Ｇ６５Ｐ００４２Ｘ－１００．Ｙ－１００．Ｒ５０．Ｚ－３．
Ｗ２．Ｕ０．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;
```



7.2 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

(Note 1) The variable number is designated after "#".
[Example] #100

(Note2) Using brackets [ ] when designating the variable number allows you to use the value stored in the variable or to use a formula.

[Example] #100=#[100+10]

This formula specifies that the value stored in variable #110 is written to #100.

(Note3) When #1 = 9, #9 = 20, and #20 = 30,
#5 = # [# [#1]] is equal to #5 = 30.

(Note4) A variable can be used instead of designating a numerical value.

[Example] #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.

(Ex.) 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.

(Note5) Variable cannot be quoted in adress N.

[Example] 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 EQ #0 → Not established
#1 NE 0 → Established	#1 NE 0 → Not established
#1 GE #0 → Established	#1 GE #0 → Established
#1 GT 0 → Not 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.

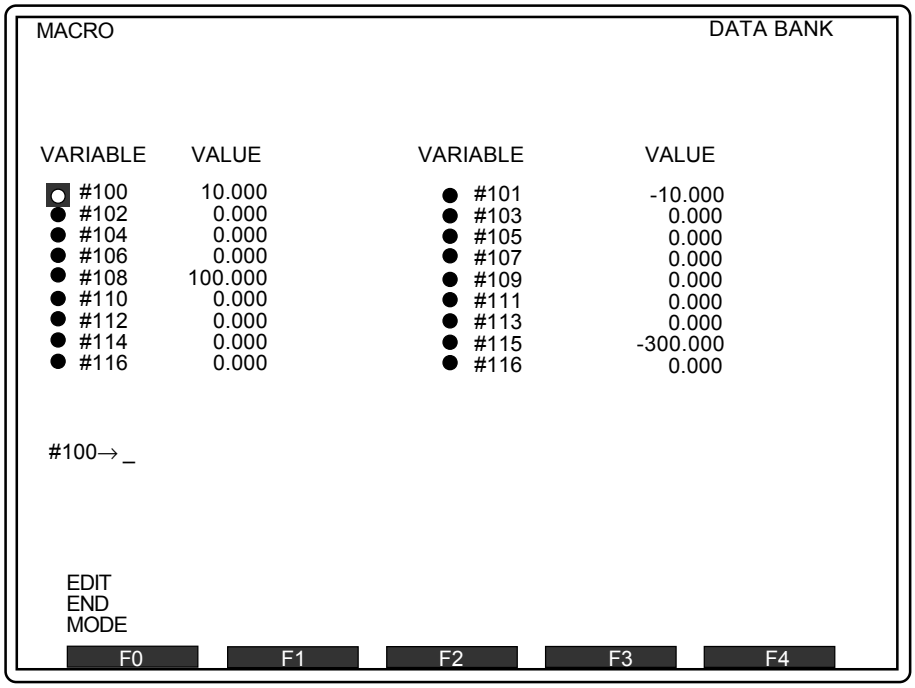
Variable No.	Variable type	Function
#0	Constantly NULL	This variable is constantly kept NULL; thus no value can be entered.
#1 to #26	Local variable	These variables can be used for each macro level. When power is turned off, these variables are cleared to be blank. Substitute the variable for these range bellow. -1.0 by 10^{99} to -1.0 by 10^{-99}, 0, 1.0 by 10^{-99} to 1.0 by 10^{99}.
#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 10^{99} to -1.0 by 10^{-99}, 0, 1.0 by 10^{-99} to1.0 by 10^{99}. 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 occured when substitute to the over range of numbers.

7

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~#599) can be referred to or changed.



Number display
Variable for #100 to #199, display the number, only variable 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 Macro (system variable)

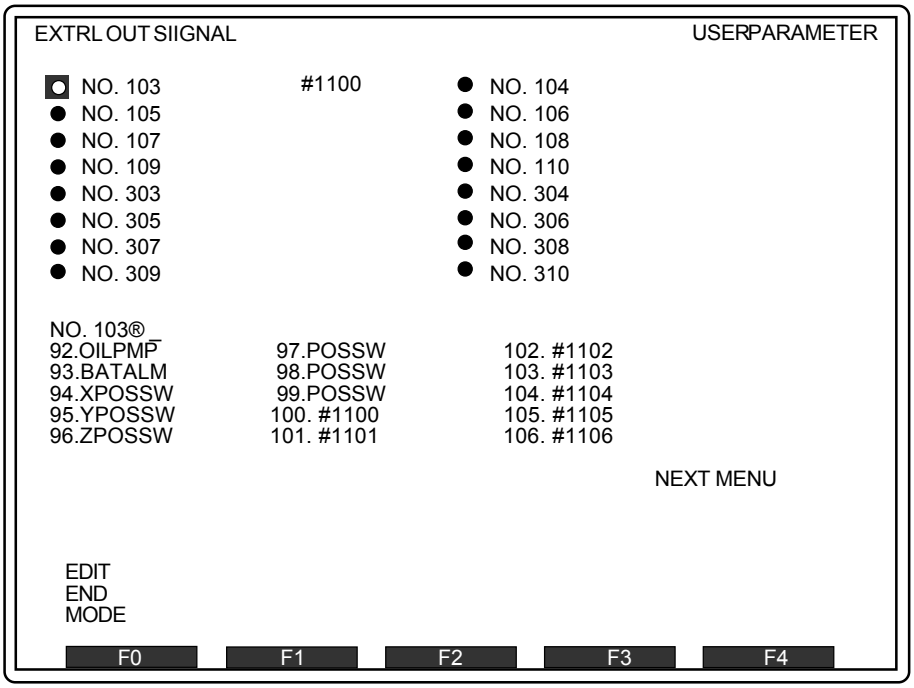
1. Interface input/output signal

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	(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 Ø COMP	#13001 ~ #13099	R/W
FINE Ø 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.

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).

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. Modal information

The called modal information can be read.

Variable 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~G89G177~G189
#4010	G98,G99
#4012	G66,G67
#4014	G54~G59
#4015	G61,G64
#4016	G68,G69,G168
#4107	D code
#4109	F code
#4111	H code
#4113	M code
#4114	Sequence number
#4115	Program number
#4119	S code
#4120	T code

#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.

9. Current position

No.	Contents	Coordinate system	Tool offset	Read while traveling
#5001~ #5006	End point coordinate	Workpiece coordinate system	Included	Possible
#5021~ #5026	Current position	Machine coordinate system	Included	Not possible
#5041~ #5046	Current position	Workpiece coordinate system	Included	Not possible
#5061~ #5066	Skip coordinate	Workpiece coordinate system	Included	Possible
#5081~ #5086	Tool length offset			Not possible
#5101~ #5106	Servo deviation			Not possible

"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;
```

7

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.

Be sure to use brackets [ ] when performing calculation.

[Example] G1Y[#100+0.1]

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 "0".

Note 4

Calculation object	Result of 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

0602.doc

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)

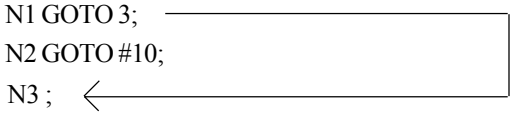
The program is branched unconditionally to sequence number "n" (n: 1 ~ 9999).

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]



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:

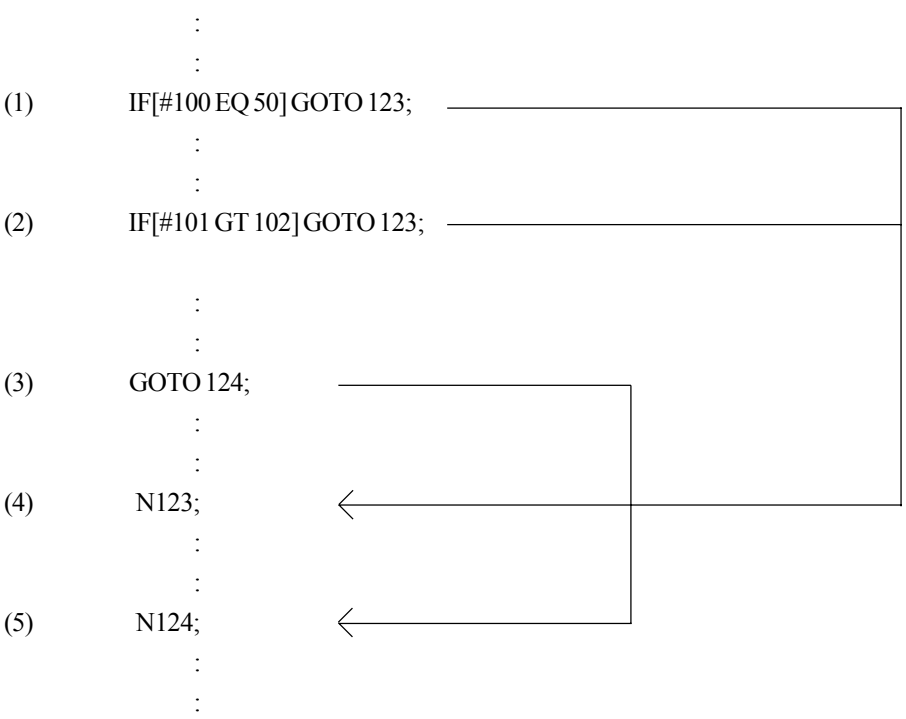
Types of condition formula

#i EQ #j	#i is EQ #j
#i NE #j	#i 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]



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.

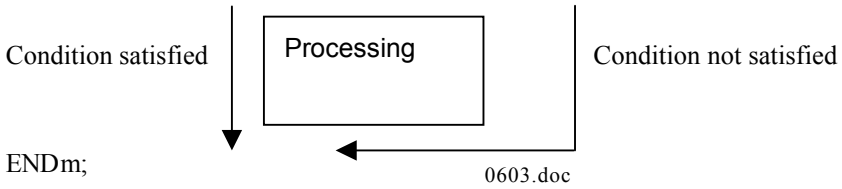
7.4.3 WHILE statement (repetition)

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.

[Example1] WHILE statement]
WHILE [condition formula] DO m; (m = 1 ~ 4)

(Note 1) 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;
  :
```

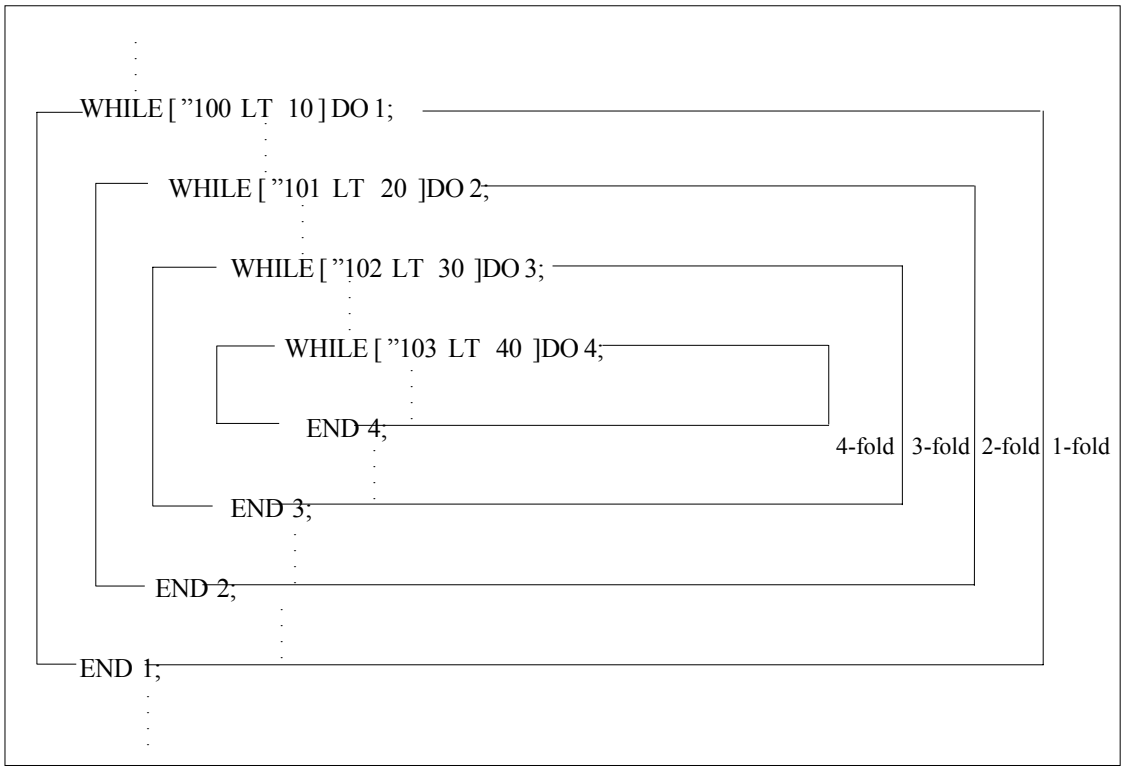
Identifier “m” can be used multiple times as long as the above condition is met.

```
  :
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;
    :
```

Note 3 Depth of nesting for DO is up to 4-fold in the WHILE statement.



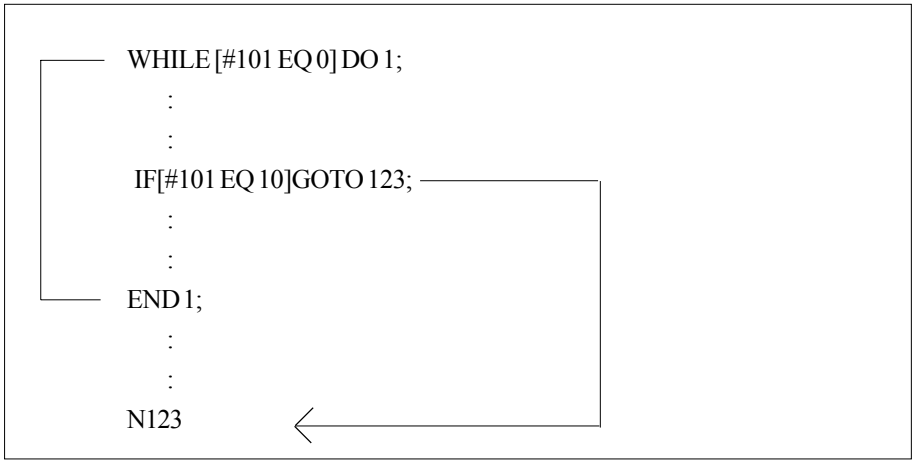
Note 4 IF statement and WHILE statement

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.



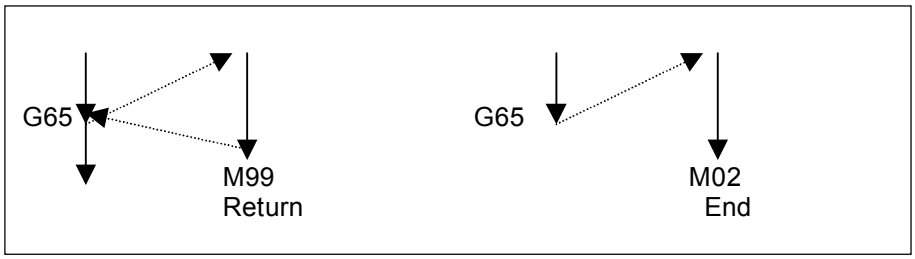
7.5 Call Function

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.

[Example1]
G65 P200;
This format specifies that a program between Nos. 200 and 299 is called once.

[Example2]
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;**

[Example1]
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 *f*
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 Arguments for macro calls

The macro call function enables numerical data to be transferred from the call source to the call destination using arguments.

[Example of argument designation]

Example: **G65 P200L2A50D60;**

In this case, [A50D60] is an argument, meaning A = 50 and D = 60.

(See 7.5.1 for "P" and "L".)

Note 1 "A" and "D" are called an "address".

The macro program called with an argument designated receives the value as a local variable.

[Example]

The values designated for "A" and "D" are copied to local variables #1 and #7 respectively.

Thus, the called macro program obtains the value referring to corresponding variables.

Example of reference: #24 = #7; (variable #24 ← Value of D, 60)

Note 2 Characteristics of local variables differ from those of common variables. See 7.2.4 for details.

Note 3 The command range using an argument is as bellow.

Metric unit system :+/-999999.999

Imperial unit system :+/-99999.999

The table below shows the relationship between addresses with an argument designated and variables in macro.

Argument designation

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

(Note)Arguments can be designated by all address except for G, L, N, O, and P.
Some addresses may not require designation of arguments.
Local variables corresponding to omitted addresses are blank.

7.5.4 Difference between G65 and M98

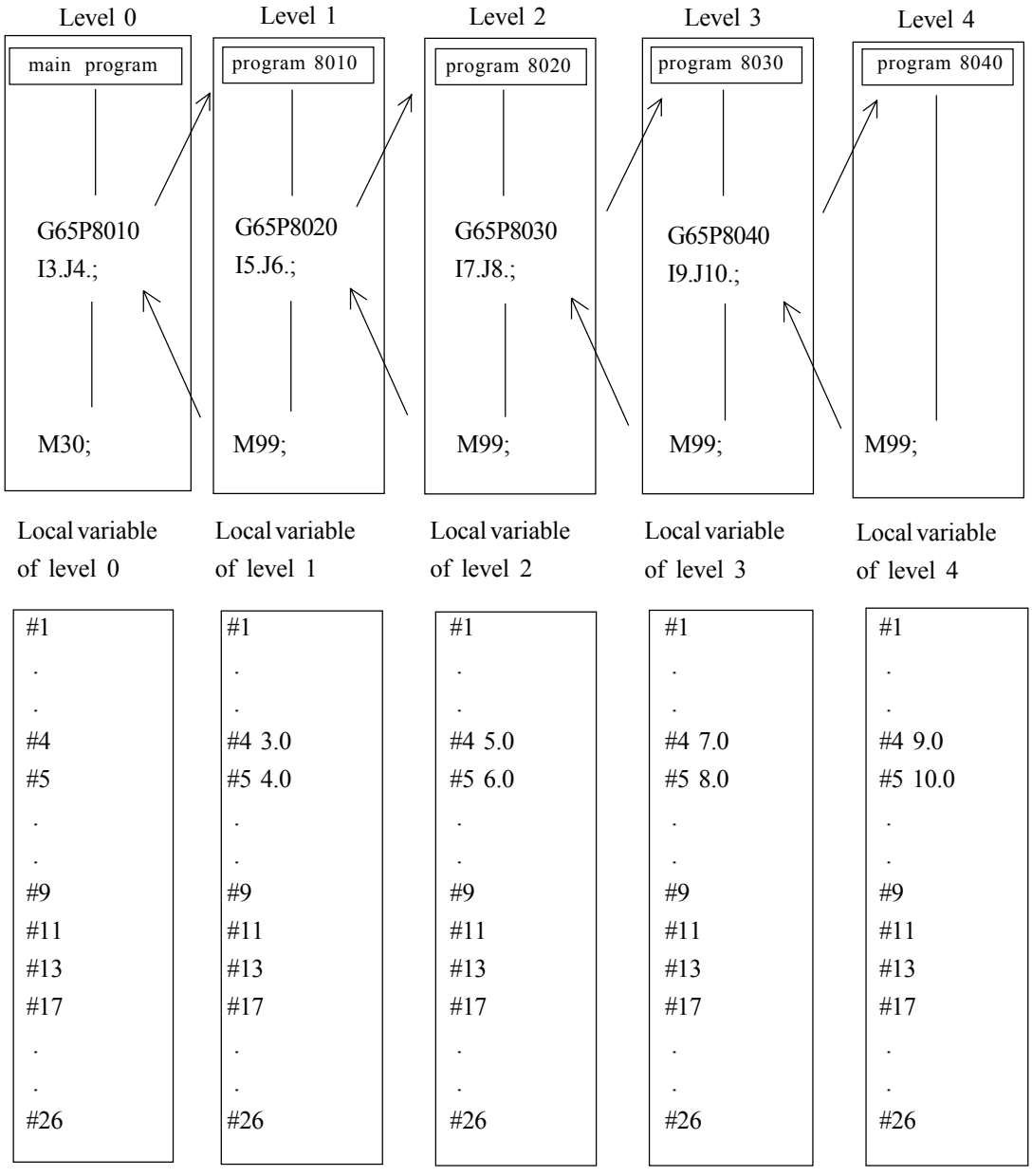
1. Arguments can be designated for G65, but cannot be designated for M98.
2. Local variables are available for G65 depending on the depth of nesting, but are not available for M98.
3. Calling depth of nesting for G65 is up to 4-fold.
   When combined with that for M98, up to 8-fold are possible.

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 valiable 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

CHAPTER 8

Z AXIS MEASUREMENT SYSTEM

8.1 Cautions

We thank you very much for your purchase of Z-axis Measurement System (hereinafter "the product"). The following sections describe the features, parts configuration, precautions on use and compatibility between data bank and machining data of the product and important points in your studying this manual.

Precaution

- M406 signal is used for air blow. This signal cannot be used for other purposes.
- The tool length measurement device is used fastened on the sub-table. This decreases available surface area of the table. In installing workpieces and jigs, exercise due care not to strike the tool length measurement device. Take the same care in removing these.
- Make sure of the Z-axis position. Inaccurate positioning may cause collision with the tool length measurement device, workpieces and jigs.
- After use, remove chips and other debris depositing inside the cover. Particularly be sure to thoroughly remove cutting oil and the like adhering to the contact area.

Feature

This product precisely detects the Z-axis position of the tool tip. This function permits three functions: Z-axis thermal displacement offset, tool breakage detection and automatic tool length measurement. The tool length detector is used for detection. The following are the brief description of each function.

Z-axis thermal displacement offset:

Restricts dimensional displacement due to Z-axis elongation within a given range. Displacement is measured to correct the machine position.

Tool breakage detection:

Detects tool breakage during operation.

Automatic tool length measurement:

Permits the measurement of tool length with the tool remaining set on the spindle. You can enter the automatic tool length measurement mode on the block operation screen for the MDI operation mode.

Parts Configuration

The product includes the following parts. Check for the numbers. If anything is missing, please contact us at the store where you bought the product or a nearby branch of our company.

Instruction manual :	1 copy
Tool length measurement device :	1 unit
Relay plate :	1 pc
M10 bolt :	2 pcs
Flat washer :	2 pcs
M5 nut :	2 pcs
Electric wire :	1 pc
Reference tool :	1 pc

Important Points in Studying the Manual

If you use a single function only, read the description of that function, tool length measurement device installation method and parameters. In setting parameters, choose only those which are required for the function. Alarm codes are listed at the end with alarm causes and remedy therefore provided. Use the list in the event of alarm.

Technical Terms

In this manual alarm messages, parameters and operation indications are written in capitals. In explaining key operation key names are written in capitals as well and enclosed in **[ ]** as shown below.
Example for indicating key operation
"Press **[RST]** key." Since "reset" is the key name, it is enclosed in **[ ]**.

8.2 Setting

There are a number of parameters not included in the Parameter List. The following explain such parameters.

MSMT DEV INSTALL POSITN

This sets the coordinates of the measurement device placed on the sub-table. Begin setting this 1 h or more after turning the machine power off. Heat remaining in the machine may affect operation.

Procedures

1. Install the tool length measurement device. For the method, refer to page.
2. Install the provided reference tool in the tool holder. For the method, refer to page.
3. Install 2 in the magazine.
4. Register the reference tool in the tool menu and on the magazine screen. Check that the tool length value is correctly input.
5. Detaching the tool now on the spindle, attach the reference tool thereon. Press **[Z. RTN]** key.
6. Adjust position in manual operation. Adjust Z-axis in the increment of 0.001 mm.

   X- and Y-axes: Move the reference tool tip to the center of the tool length measurement device contact face.

   Z-axis: Upon completion of X- and Y-axis positioning, adjust Z-axis as follows.

   Z-axis positioning Display the I/O screen (slave). Descend the spindle until its tip contacts the contact face. Further descend the spindle until the numeral on the screen changes from "0" to "1."

   INPUT 4 00000000 00000000
   ↑
   Changes from "0" to "1."

8

7. Press **[POS]** key. Note down the machine coordinates indicated on the screen.
8. Raise the z axis. Set the value so that the tool length of the reference tool is subtracted from the value noted on the parameter.
   With this the setting of measurement device installation position is completed.

THERMAL MEASUREMENT MOTION COND.

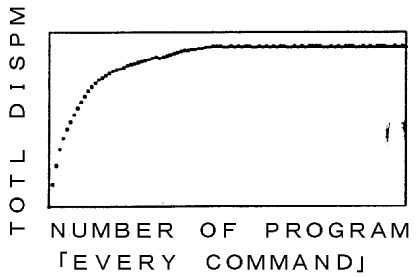
This sets conditions for measurement motion.

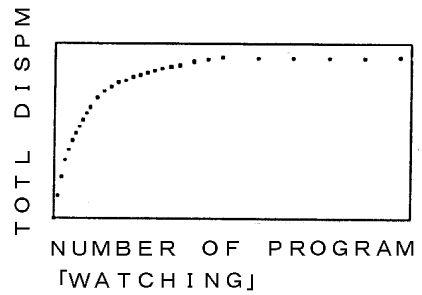
EVERY COMMAND

This specifies a series of measurement motion to be performed each time the command is issued.

WATCHING

With this parameter, measurement motion is not performed each time the command is issued, because frequent measurement is not necessary when elongation due to heat on Z-axis is constant. Based on displacement measured, the subsequent measurement motion is canceled. The premise for WATCHING is that environment of the machine remains unchanged. When room temperature is radically fluctuating and when work replacement time is not constant, WATCHING does not correctly operate. In that case set the thermal measurement motion conditions to EVERY COMMAND.





More about WATCHING

Normal operation of WATCHING requires the setting of the following parameters.

- Z-AXISMACHININGACCURACY
- THERMAL MATCH DISPLACMT AMNT
- NUMBEROF THERMAL WATCHING
- NO. OF THERMAL WATCHING CANCEL
- INITIAL SET NO. OF THERMAL MSMT
- THERMAL WATCHING STOP TIME

In setting these, although the initial set values are used as a rule, values must be changed in some cases depending on the type of works to be machined and work replacement time.

The next page presents examples of such change.

Example changing of set value

Procedures

1. Check a cycle time (work replacement time) of the machine program.
2. Set THERMAL MEASUREMENT MOTION COND. to EVERY COMMAND and begin machining.
3. Upon completion of 280 to 290 or so cycles or 1 h before the end of working hours, stop the machine for a period twice longer than work replacement time.
4. Repeat machining once again (only once).
5. Referring to the thermal measurement data on the PRODUCTION MONITOR screen, set respective parameters as follows.

Z-AXIS MACHING ACCURACY

Set that machining accuracy which must be at least maintained.
The value thus set is used to only check the range of displacement, not to keep the machining accuracy within itself.
(Ex.) Set 0.020 when $\pm 20\ \mu$ is required.

THERMAL WATCH DISPLACMT AMNT

Set about half the Z-axis MACHINING ACCURACY. The value thus set is used to judge displacement. Based on the result of judgment, update the frequency of watching.

THERMAL WATCHING STOP TIME

If displacement after work replacement time is increased twofold (displacement in 4. above) is within THERMAL WATCH DISPLACMT AMNT, set time twice longer than work replacement time. If the value exceeds this amount, set THERMAL MEASUREMENT MOTION COND. to EVERY COMMAND. In this case presume that WATCHING is impossible. The value set herein indicates time to initialize the watching conditions.

THERMAL WATCHING STOP TIME

This sets the number of watching motions at which displacement becomes stable within THERMAL WATCH DISPLACMT AMNT after machining is began. The value thus set indicates the frequency of measurement unconditionally taken after machining is began.

Parameters for watching are valid for works of the same type. When the type of work is changed, change parameters as well.

Caution

Do not turn machine power off until all setting is completed. Turning power off before this causes thermal measurement data on the PRODUCTION MONITOR screen to be lost.

XY POSITION RADIUS SHIFT DIRCTN, THERMAL

Use this depending on the tool used.

THRM OFFSET AMT LIMIT VALUE, RETRACT AMNT AFT MEASUREMENT

Do not change this unless special reasons are justified.

MSMT FEEDRATE B TOOL DIAMTR

For measurement feedrates 1 and 2, A or B applies depending on the tool diameter. This parameter sets the diameter of a tool to which feedrate B applies.

MEASUREMENT FEEDRATE 1 A, MEASUREMENT FEEDRATE 2 A

Set a feedrate not to cause tool breakage. When a tool with a small diameter is caused to contact the tool length measurement device, choose the slower rate. Otherwise the tool may break.

MEASUREMENT FEEDRATE 1 B, MEASUREMENT FEEDRATE 2 B

Set this to 3,000 mm/min or lower. Setting of a larger value than this may break the tool length measurement device.

MEASUREMENT FEEDRATE 2 A, MEASUREMENT FEEDRATE 2 B, THERMAL MEASUREMENT FEEDRATE

Set this to 50 mm/min or lower.

8

8.3 Z-axis Thermal Displacement Offset

This chapter explains thermal displacement offset and the offset method using thermal measurement data.

8.3.1 Offset Capacity

As shown below.

Offset Capacity	Conditions
± 0.01mmmax./day	Aluminummachining. Room temperature fluctuation ±2℃. Machining at the same height as the measuring height. Reference tool is used. Thermal displacement detection is executed immediately beforemachining.

The offset capacity varies with environment in the factory.
Use these conditions only as a guideline.
Offset requires time to complete thermal displacement detection, which is some 12 s or shorter per operation.

8.3.2 For Correct Thermal Displacement Offset

Observe the following instructions not to cause errors in offset.

Measuring tool

Use reference tools. Do not use cutting tools. The following explain reference tools and cutting tools.

Reference tool: Tools used for measurement only are called reference tools in this manual. Although a place on the magazine available for tool installation is occupied, it is advisable to use the reference tool for accurate measurement.

Cutting tool: This indicates a tool used for practical machining. If a cutting tool is used as measuring tool, position detection is executed under the influence of worn cutting edge, tool thermal expansion and chip adhesion. The machine deems the position detected as Z-axis thermal displacement in correcting it. Although measurement is possible with cutting tools, it is advisable to use the reference tool for accurate measurement.

Measurement of measuring tool length

- Use the automatic tool length measurement function.
- To eliminate the effect of thermal displacement during setting, begin measurement 1 h or more after power is turned off.
- For the method for automatic tool length measurement, refer to Chapter 6.
- Do not measure measuring tool length after executing thermal displacement offset.

Programming

- Place the command before machining which requires accuracy.
- If the tool is left installed on the spindle for over 2 or 3 min under heat generating in the spindle, heat conducting via the spindle elongates the tool, consequently deteriorating accuracy. Therefore, write the program such that the reference tool and the cutting tool requiring accuracy are not installed on the spindle.

Measuring height

- Adjust insomuch as possible the measuring height to that at which practical machining is carried out. For the method, refer to page.

Machining environment

- Maintain room temperature as constant as possible.
- Maintain air current in the factory as constant as possible.
- Do not add coolant during machining. Otherwise coolant temperature changes, affecting tool elongation.
- When machining has to be stopped for over 20 or 30 min, cut coolant off. If the coolant motor is kept turning without circulating coolant, coolant temperature abnormally increases.
- Firmly fasten jigs so that works do not chatter or deflect. In jigs with the motor, heat develops in the motor, causing thermal expansion to the jigs. This may deteriorate accuracy in some cases.

8.3.3 Command

This section describes the commanding method and valid conditions for commanding. An example program is presented as well.

Effective conditions

The command is valid after M206 is excuted in the memory operation mode until the next operation is perfoemed or the next command is executed.

- Reset
- Program completion (including program completion during the execution of the schedule program)
- Setting the memory operation program

M206 and M207 cannot be commanded in the MDI mode.

8.3.3.1 Determination of thermal measurement

Command format

```
M206  T__ L__  ;
```

T : Command for the next tool preparation when it is determined that a thermal measurement should be performed based on the motion check. If no command is given, a **[Z MESRMENT COMD ER]** error will occur.

L : Command for the next tool preparation when it is determined that a thermal measurement should NOT be performed based on the motion check.
Whether to execute a thermal measurement is determined, and the number of commands is then updated.

When performing thermal measurement has been determined
Command the next tool preparation to address"T".
M206 is executed for the tool designated to address "T", and the designated sub program is executed using the subsequent M207command.

When not to perform thermal measurement has been determined.
When "L" is present in the address, the next tool preparation is executed.
If "L" is omitted, however, the next tool preparation is not executed.
The subsequent M207 command is ignored.

Note:

. When the Z axis measurement system is not provided, a [NO Z-AX MEASUREMENT] error will occur.
. When M206 is re-commanded between command M206 and M207, a [Z MESRMENT COMD ER] error will occur.
. When the [THERMAL MEASUREMENT FUNCTION] is set to [INVALID], it is determined that the thermal measurement is not performed. [this could mean "should not be performed,* or "has not been performed"]
. When the machine is in the dry run or machine lock mode, the number of commands is not counted, and it is determined that the thermal measurement is not performed.

When the NC is TC-31A, TC-32A, TC-R2A.

When the pallet not selected by the parameter is commanded, it is determined that the thermal measurement is not performed.
When [MEASURING DEVICE] is set to [NO] for the target pallet, a [NO MEASURING DEVICE] error will occur.

8.3.3.2 Thermal measurement operation

Command format

```
M207   P__   ;
```

P: Sub program No.

When the result of the motion check by M206 is EXECUTE, the designated sub program is executed; when NOT EXECUTE, nothing is performed using command M207.

When the tool diameter compensation command is not cancelled, a **[COMMAND INVALID (M)]** error will occur.

Note:

- **When the Z axis measurement system is not provided, a [NO Z-AX MEASUREMENT] error will occur.**
- **When M207 is commanded without M206 being commanded, a [Z MESRMENT COMD ER] error will occur.**

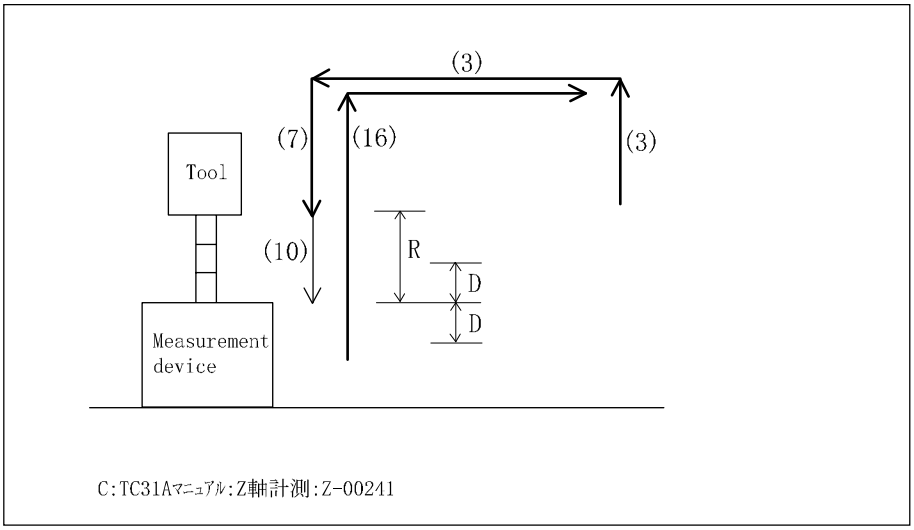
When the NC is TC-31A, TC-32A, TC-R2A.

- **When the pallet not selected by the parameter is commanded, it is determined that the thermal measurement is not performed.**
- **When [MEASURING DEVICE] is set to [NO] for the target pallet, a [NO MEASURING DEVICE] error will occur.**
- **When the pallet is not indexed, a [PALLET POS ERROR] will occur.**

8.3.4 Program examples

This section describes an example for the Z-axis thermal displacement offset subprogram.

(1) M9	Set coolant to OFF.
(2) M406	Air blow ON.
(3) G100 X__Y__	Changes the tool. Positions the XY axes to the thermal measurement position.
(4) M322	Cover open check. Operation is halted until the tool length measurement device cover opens. If the cover does not open after 10 s, alarm *COVER DOESN'T OPEN occurs.
(5) M4 S60	Spindle rotation in CCW direction.
(6) G43 H__	Compensate the tool length.
(7) G0 Z__	Positions the Z axis to the measurement start position.
(8) G4 X__	DWELL.
(9) M321	Detection signal OFF check.
(10) G131 Z__F__	Measurement motion.
(11) G4 X__	DWELL.
(12) G1 Z__F__	Returns the Z axis to the preset retraction value.
(13) M19	Spindle orientation.
(14) M321	Detection signal OFF check.
(15) G131 Z__F__	Measurement motion.
(16) G53 G0 Z__	Returns the Z axis to its home position.
(17) M332	Position compensation.
(18) M407	Air blow OFF.
(19) M08	Set coolant to ON.



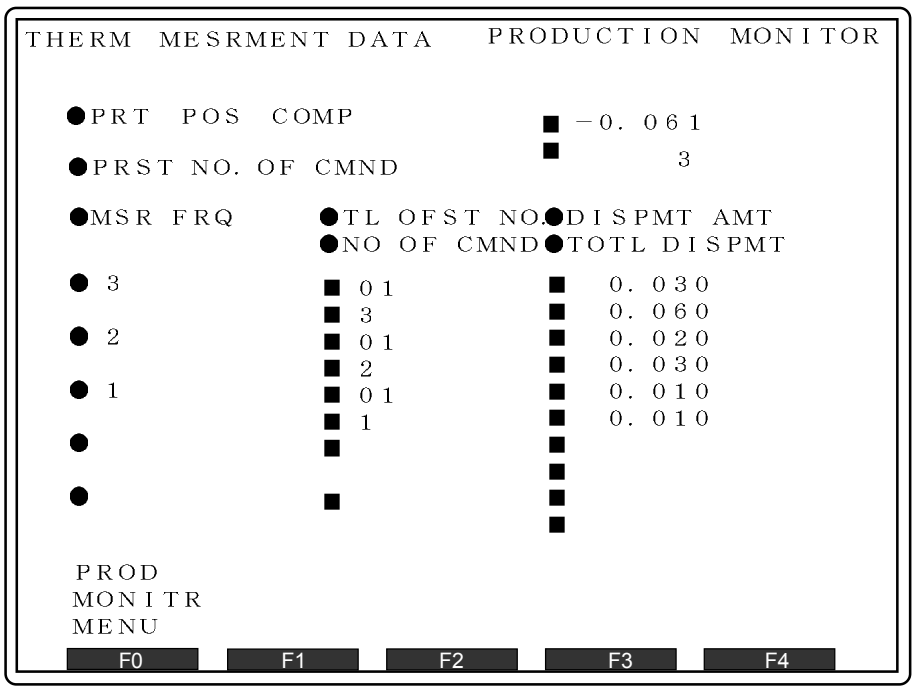
R: Measurement start position

D: Measurement stop range

- When the service life of the tool set for thermal measurement has expired and the tool is replaced, a thermal measurement is performed for the replacement tool. With power turned off, the position offset amount due to thermal displacement is reset.
- When memory operation is started with signal output 406 remaining on, alarm *M406 IS ON occurs.

8.3.5 Thermal Measurement Data Screen

The current state of measurement can be referred to on the PRODUCTION MOINTOR screen. This screen can display the results of a maximum of 300 measurements taken in the past. These data can be browsed for reference only. The screen below presents data of the first thru the third measurements.



Explanation

PRT POS COMP ..... Indicates how much the offset has been achieved until Z-axis is in the current position.
This screen indicates offset of • |0.061.

PRST NO.OF CMND ..... Represents the number of commands issued by now.
This screen indicates that the command is the third.

MSR FRQ ..... Represents the number of measurements taken.

TL OFST NO. ..... Represents the tool length offset number used for measurement.

DISPMT AMT ..... Represents displacement measured.

TOTL DISPMT ..... Represents total displacement.

* Measurement data are displayed from here.

Caution:

Turning power off causes all data to be lost.

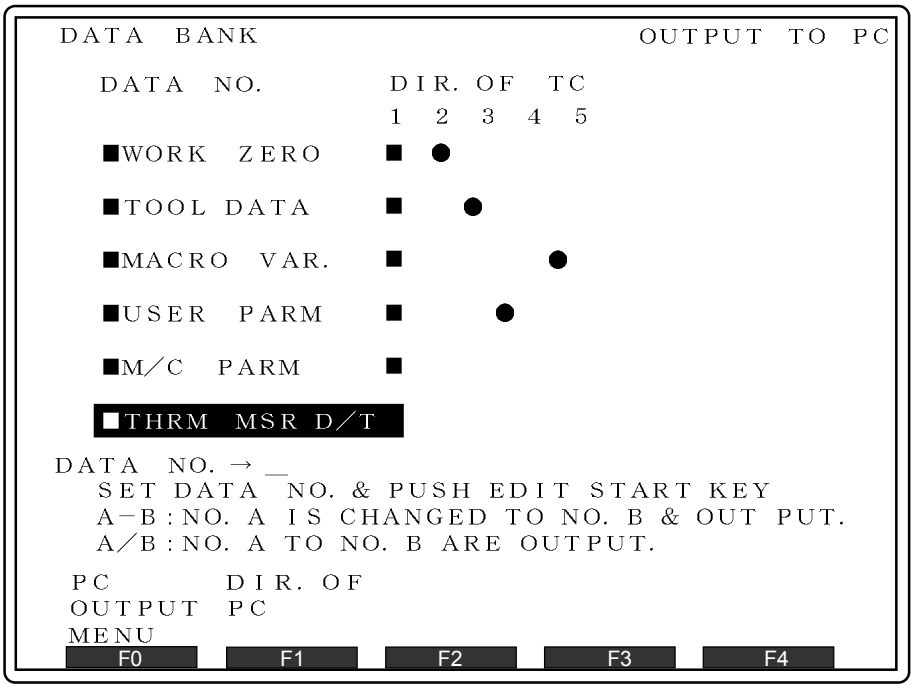
8.3.6 Outputting Thermal Displacement Measurement Data to the Computer

With the aid of the optional external communications equipment, thermal displacement measurement data can be output to the computer.

Data are in the text form.

The use of table calculation software makes possible data analysis.

Data possible to be output:
MSR FRQ
TL OFST NO.
NO OF CMND
DISPMT AMT
TOTL DISPMT



- Move the cursor to **[THRM MSR DT]**, and press the **[1]** and **[E. STA]** keys. The thermal measurement data is output.
- When using the metric system or the imperial system, MSMTNM or MSMTNI is added to the file name respectively.

  Data format.
- A line consists of 35 bytes. Each item is sectioned in a fixed length.

MSR FRQ 6 bytes	TOOL NO. 2 bytes	NO. OF CMND 7 bytes	DISPMT AMT 7 bytes	TOTL DISPMT 7 bytes	Line feed
****** ,	** ,	******* ,	******* ,	*******	(0Dh) (0Ah)

For details of communications, refer to Chapter 11, External Communications Equipment, of the Instruction Manual for Optional Equipment.

8.3.7 Example Parameter Setting and Alarm

This section presents an example of parameter setting. When the value set in the parameter is not as specified, the corresponding alarms occur. Such alarms are explained in this section as well.

Example setting

The following example shows a case where a cycle time in the program is short when periodical measurement is wanted.

ITEM	Setting
THERMAL MEASUREMENT MOTION COND	1(WATCHING)
Z-AXIS MACHINIG ACCURACY	0.999
THERMAL WATCH DISPLACMT AMNT	0.999
NUMBER OF THERMAL WATCHING	1
NO.OF THERMAL WATCHING CANCEL	5
INTIAL SET NO.OF THERMAL MSMT	1

For THERMAL WATCHING STOP TIME, refer to page.

Measurements are taken at the first, second, fourth and eighth commands. After the eighth, measurements are taken every five commands, namely at the 13th, 18th, 23rd and so on.

Alarm occurrence

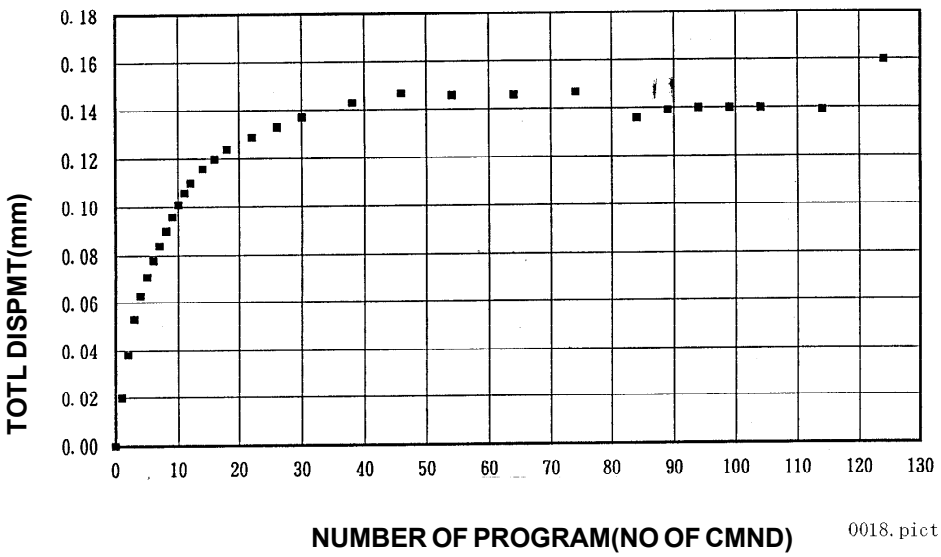
- *TRM DISPMT AMT OVR occurs at the beginning of machining after turning power on.
  The cause is considered unstable displacement at the beginning of machining.
  Increase INITIAL SET NO. OF THERMAL MSMT.
  *TRM DISPMT AMT OVR occurs not only at the beginning of machining after turning power on, but also at other occasions.
- Increase NUMBER OF THERMAL WATCHING and decrease THERMAL WATCH DISPLACMT AMNT.
  With larger NUMBER OF THERMAL WATCHING, measurement begins only after displacement stabilizes better than otherwise. This is because past measurement data are referred to the number of times equal to the number of thermal watching settings.
  THERMAL WATCH DISPLACMT AMNT provides a criterion by which it is judged whether or not measurement can be canceled. With smaller THERMAL WATCH DISPLACMT AMNT, measurement is canceled only after thermal displacement stabilizes better than otherwise.

Should *TRM DISPMT AMT OVR occurs even after these operations, set THERMAL MEASUREMENT MOTION COND. to EVERY COMMAND.

8.3.8 Example Offset

This section presents an example of practical offset using a time-course change graph of Z-axis thermal displacement and thermal measurement data.

Graph of Z-axis thermal displacement



THERMAL MEASUREMENT DATA

The following lists parameters setting values.

THERMALMEASUREMENTMOTIONCMND.....WATCHING
Z-AXIS MACHINIG ACCURACY.....0.020(mm)
THERMALWATCHDISPLACMTAMNT.....0.010(mm)
NUMBEROFTHERMALWATCHING.....2(TIMES)
NO.OFTHERMALWATCHINGCANCEL.....10(TIMES)
INTIALSETNO.OFTHERMALMSMT.....10(TIMES)

MSR FRQ	NO OF CMND	DISPMT AMT mm	TOTL DISPMT-A	
1	1(1)	0.020	0.020	Measurement are taken unconditionally until 10 commands are issued.
2	2(1)	0.018	0.038	
3	3(1)	0.015	0.053	
4	4(1)	0.010	0.063	
5	5(1)	0.008	0.071	
6	6(1)	0.007	0.078	
7	7(1)	0.006	0.084	
8	8(1)	0.006	0.090	
9	9(1)	0.006	0.096	
10	10(1)	0.005	0.101	( )Indicats MSR.FRQ
				(1)
11	11(1)	0.005	0.106	A(11)-A(9) =0.010
12	12(1)	0.004	0.110	A(12)-A(10)=0.009
				(2)
13	14(2)	0.006	0.116	
14	16(2)	0.004	0.120	A(16)-A(12)=0.010
15	18(2)	0.004	0.124	A(18)-A(14)=0.008
				(3)
16	22(4)	0.005	0.129	
17	26(4)	0.004	0.133	A(26)-A(18)=0.009
18	30(4)	0.004	0.137	A(30)-A(22)=0.008
				(4)
19	38(8)	0.006	0.143	
20	46(8)	0.004	0.147	A(46)-A(30)=0.010
21	54(8)	-0.001	0.146	A(54)-A(38)=0.003
				(5)
22	64(10)	0.000	0.146	
23	74(10)	0.001	0.147	
24	84(10)	-0.011★	0.136	
				(6)
25	89(5)	0.003	0.139	
26	94(5)	0.001	0.140	
27	99(5)	0.000	0.140	A(99)-A(89)=0.001
28	104(5)	0.000	0.140	A(104)-A(94)=0.000
				(7)
29	114(10)	-0.001	0.139	
30	124(10)	0.021★	*****	[*TRM DISPMT AMT OVR]

( )Indicates the frequency of watching

(8)

8

Interpretation and explanation of thermal measurement data

The previous page shows the data of the first thru 30th measurements. The data are divided by lateral lines into eight sections (1) thru (8).
The following give explanation by section.

(1) INITIAL SET NO. OF THER•~MAL MSMT is 10. Measurements are taken unconditionally until 10 commands are issued. Watching frequency is initially set to 1. For the meaning of watching frequency, refer to the section describing parameters.

(2) From the 11th measurement onwards, judgment as to whether or not measurement motion should be canceled begins. Since the watching frequency is 1, judgment concerns possibility whether or not the frequency can be 2 (1×2).
The judging method is as follows.
Since watching frequency is wanted to be 2, subtract total displacement A obtained two measurements before from total displacement A at the current measurement. The resulting value is checked if it is within THERMAL WATCH DISPLACMT AMNT.
If it exceeds this amount, that indicates that thermal displacement has not stabilized yet, making judgment that the watching frequency cannot be updated.
If the resulting value of subtraction is within THERMAL WATCH DISPLACMT AMNT, since the NUMBER OF THERMAL WATCHING is 2, subtract total displacement A obtained three measurements before from total displacement A obtained a measurement before.
The resulting value is checked if it is within THERMAL WATCH DISPLACMT AMNT.
If it is within, since the difference between two successive displacements A and another A is within THERMAL WATCH DISPLACMT AMNT, thermal displacement is judged stabler than before, consequently setting watching frequency to 2.
Thus, cancellation is executed once in every two measurements from the subsequent measurement onwards. This process can be summarized as follows.
Judgment is made to see if watching frequency can be made twice larger than the current setting. If judged possible, the watching frequency is updated and cancellation takes place at that frequency. Based on in this judgment is the difference between total displacement A at the current measurement and total displacement A obtained (watching frequency•~2) measurements before, which is checked if it is within THERMAL DISPLACMT AMNT and if it remains so at all measurements taken the number of times equal to the NUMBER OF THERMAL WATCHING.

(3) Since watching frequency is 2, judgment is made to see if the frequency can be made 4 (2×2). At the 18th command specified conditions are met, making it possible to update watching frequency to 4.

(4) Since watching frequency is 4, judgment is made to see if the frequency can be made 8 (4×2). At the 30th command specified conditions are met, making it possible to update watching frequency to 8.

(5) Since watching frequency is 8, judgment is made to see if the frequency can be made 16 (8×2). At the 54th command specified conditions are met, making it possible to update watchingfrequency. However, since NO. OF THERMAL WATCHING CANCEL is 10, the watching frequency is set to 10.

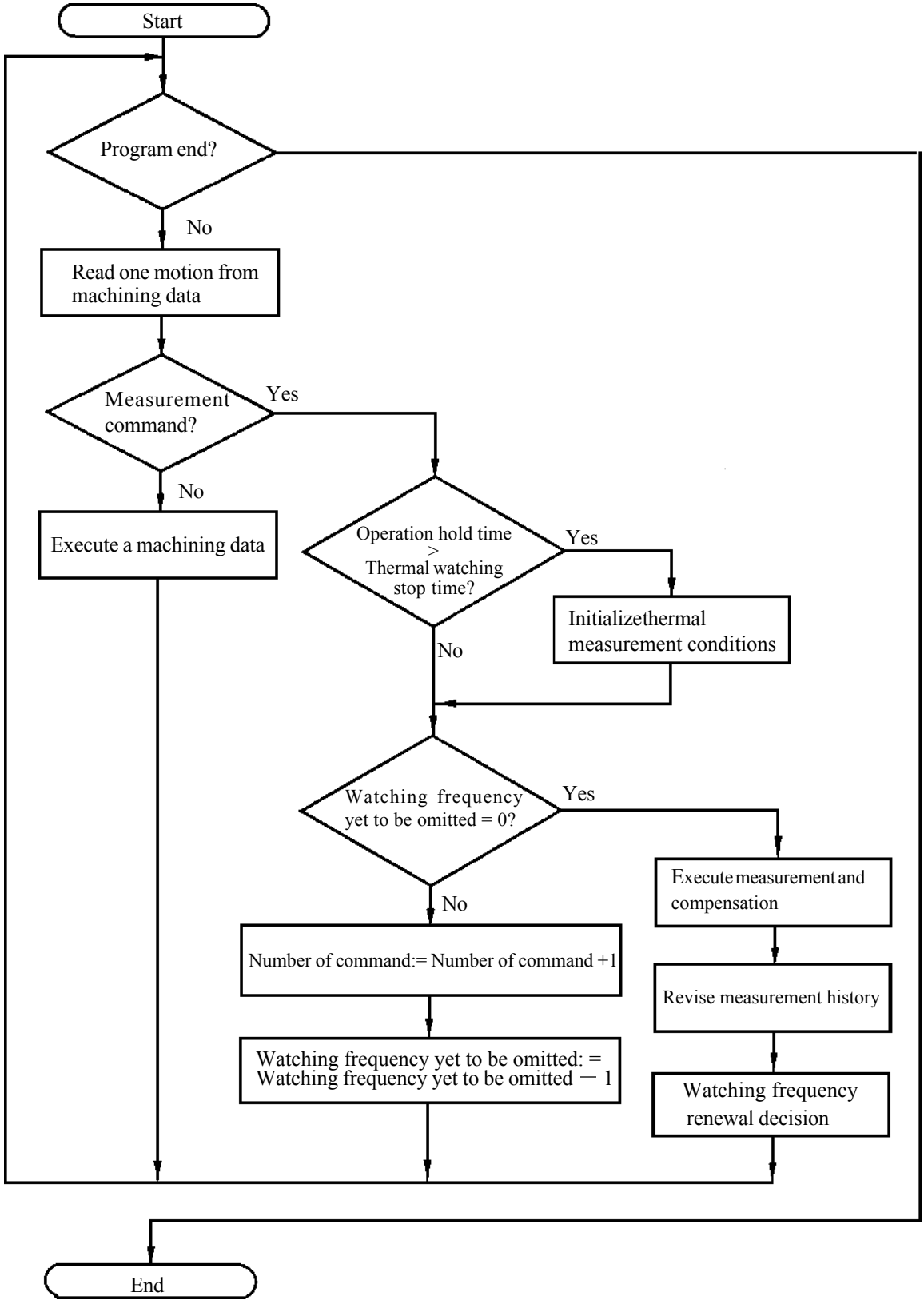
(6) Since watching frequency is 10, judgment is made to see if the frequency can be made 20 (10×2). Since the NO. OF THERMAL WATCHING CANCEL is 10, it is impossible to have 20 for watching frequency. Measured displacement registers -0.011 at the 84th command. This value falls between Z-axis MACHINING ACCURACY and THERMAL WATCH DISPLACMT AMNT, which suggests that thermal displacement begins to lose stability. Hence, the watching frequency should be updated to 5 (10÷2).

(7) Since watching frequency is 5, judgment is made to see if the frequency can be made 10 (5×2). At the 104th command specified conditions are met, making it possible to update watching frequency to 10.

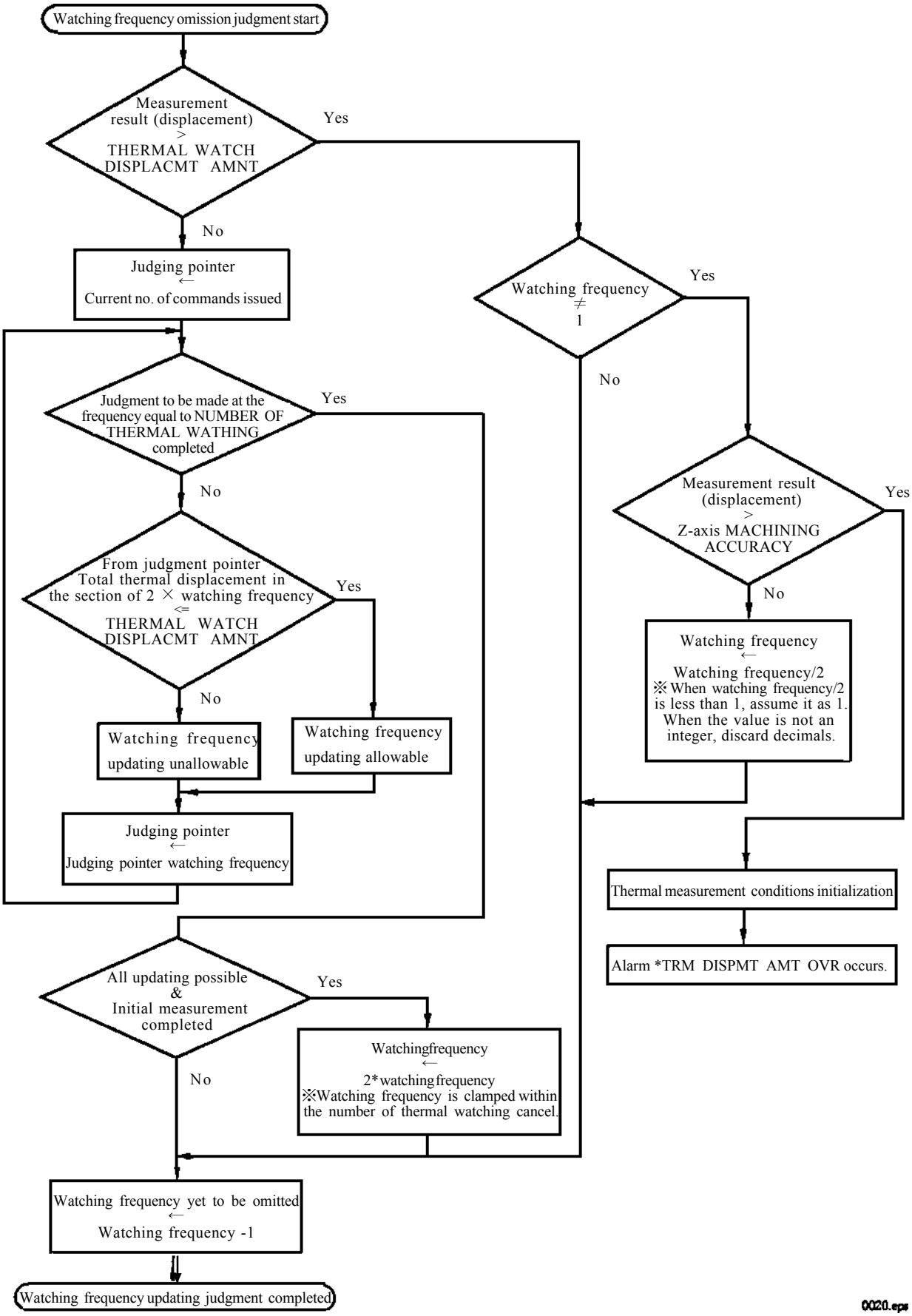
(8) Displacement registers 0.021 at the 124th command. Since this value exceeds the Z-axis machining accuracy, alarm *TRM DISPMT AMT OVR occurs, stop ping the machine.

The flowchart of this method is as shown on the next page.
It is designed to help your understanding.

Flowchart


Start
Program end?
No
Read one motion from machining data
Measurement command?
Yes
No
Execute a machining data
Operation hold time > Thermal watching stop time?
Yes
Initializethermal measurement conditions
No
Watching frequency yet to be omitted = 0?
Yes
Execute measurement and compensation
No
Number of command:= Number of command +1
Revise measurement history
Watching frequency yet to be omitted: = Watching frequency yet to be omitted − 1
Watching frequency renewal decision
End

Flowchart


Watching frequency omission judgment start
Measurement result (displacement) > THERMAL WATCH DISPLACMT AMNT
Yes
No
Judging pointer ← Current no. of commands issued
Watching frequency ≠ 1
Yes
No
Judgment to be made at the frequency equal to NUMBER OF THERMAL WATHING completed
Yes
No
From judgment pointer Total thermal displacement in the section of 2 × watching frequency <= THERMAL WATCH DISPLACMT AMNT
Yes
No
Watching frequency updating unallowable
Watching frequency updating allowable
Judging pointer ← Judging pointer watching frequency
Measurement result (displacement) > Z-axis MACHINING ACCURACY
Yes
No
Watching frequency ← Watching frequency/2
※When watching frequency/2 is less than 1, assume it as 1. When the value is not an integer, discard decimals.
Thermal measurement conditions initialization
Alarm *TRM DISPMT AMT OVR occurs.
All updating possible & Initial measurement completed
Yes
No
Watchingfrequency ← 2*watchingfrequency
※Watching frequency is clamped within the number of thermal watching cancel.
Watching frequency yet to be omitted ← Watching frequency -1
Watching frequency updating judgment completed
0020.eps

8

8.4 Tool Breakage Detection

This chapter describes operation to detect tool breakage. For the method for installing the measurement device, refer to Chapter 2. Relevant parameters are provided in the parameter list in Chapter 3. Choose from the list the ones required for this function and set.

8.4.1 Determination of tool breakage detection

Command format | **M203 P_ ;**

P: Sub program NO.

Note:

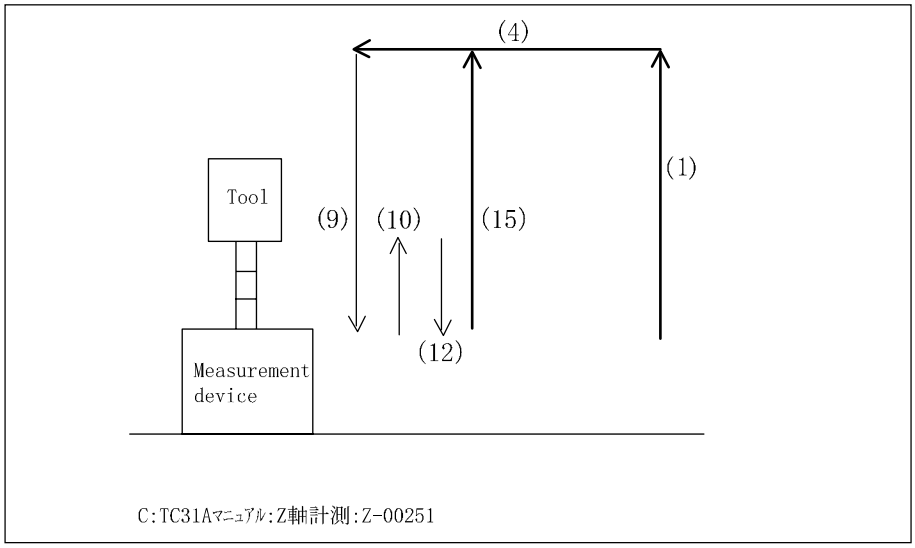
- When the tool diameter compensation command is not cancelled, a [COMMAND INVALID(M)] error will occur.
- When the Z axis is measurement system is not provided, a [NO Z-AX MEASUREMENT] error will occur.
- When the [TOOL BREAKAGE DETECT FUNCTION] is set to [INVALID], it is determined that the thrmal measurement shoud not be performed.
- When the [TOOL BREAKAGE DETECT FUNCTION] is set to [INVALID], the command is ignored.
- When M203 is used in a restarting program, a RESTART ERROR will occur. When the NC is TC-31A , TC-32A , TC-R2A.
- When no measuring device is provided for the target pallet, the command is ignored.
- When the pallet is not indexed, a [PALLET POS ERROR] will occur.

M203 cannot be comanded in the MDI mode.

8.4.2 Program examples

This section describes an example for the Z- axis thermal displacement offset subprogram.

(1) M9	Set coolant to OFF.
(2) G0 Z__	Returns the Z axis to its home position.
(3) M406	Air blow ON.
(4) G0 X__ Y__	Positions the XY axes to the thermal measurement position.
(5) M322	Cover open check.
(6) M19	Spindle rotation
(7) G0 Z__	Positions the Z axis to the measurement start position.
(8) G4 X__	DWELL.
(9) M321	Detection signal OFF check.
(10) G132 Z__ F__	Measurement motion.
(11) G1 Z__ F__	Push in the Z axis direction (to turn the detectin signal ON)
(12) M120	Detection signal ON check.
(13) G0 Z__	Returns the Z axis to its home position.
(14) M407	Air blow OFF.
(15) M8	Set coolant to ON.



R:Measurement start position
D:Measurement stop range

Supplementaryexplanation

If **[RST]** key is pressed after alarm *TOOL BROKEN ERROR occurs, the service life setting of the broken tool is reset to zero (0).

In machines provided with the Z-axis measurement system, when signal output 406 is on, memory operation cannot be started. If memory operation is started nevertheless, alarm *M406 IS ON occurs.

Dry run during operation with broken tools
When dry run is valid, measurement movement and detection signal confirmation are not executed.

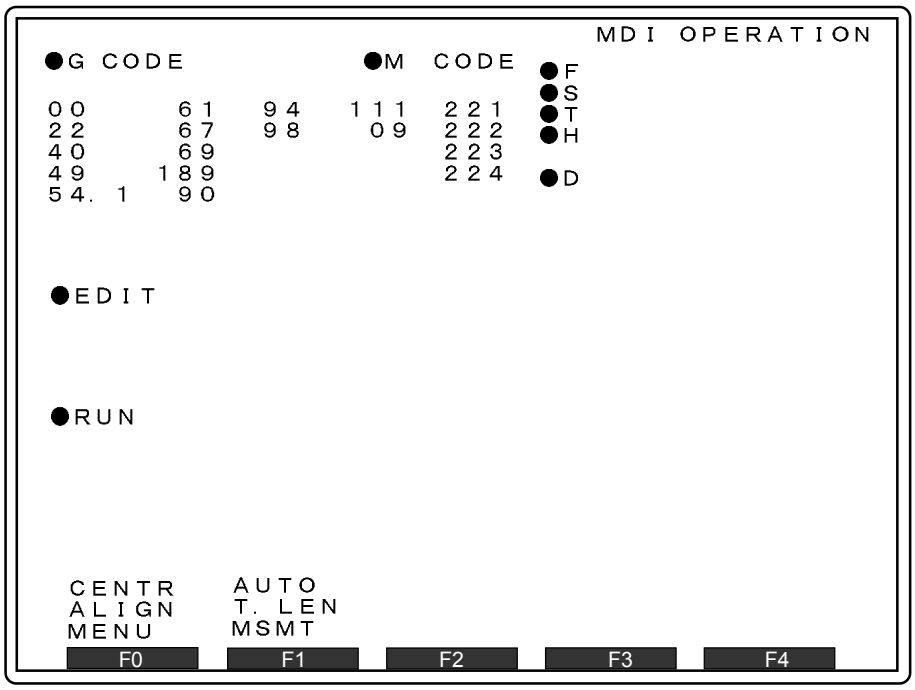
8.5 Automatic Tool Length Measurement

This chapter describes operation to automatically measure tool length.
For the method for installing the measurement device, refer to Chapter 2.
Relevant parameters are provided in the parameter list in Chapter 3.
Choose from the list the ones required for this function and set.

8.5.1 Command

Commandingmethod

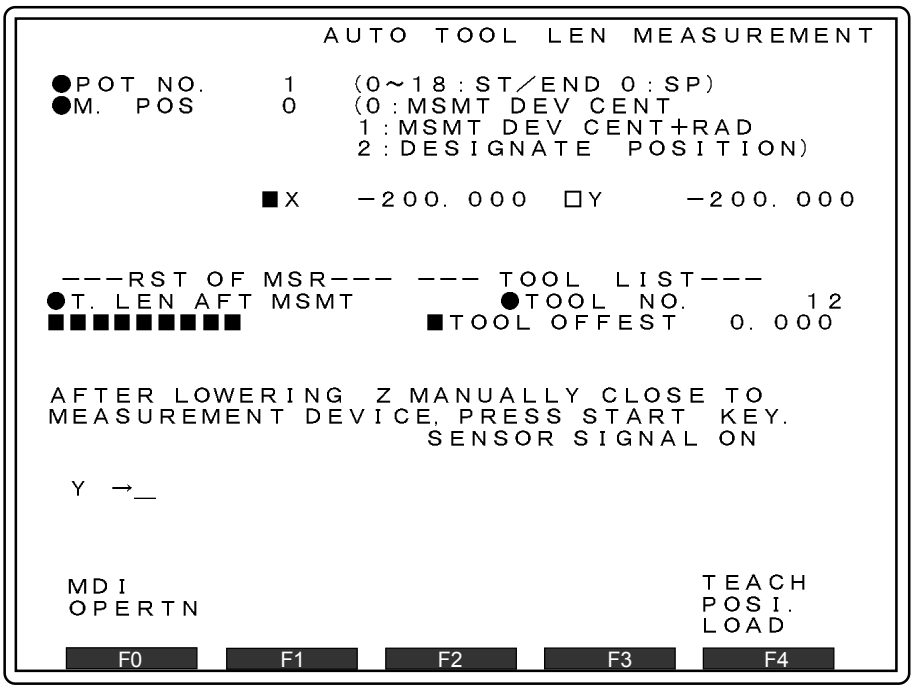
Automatic measurement of tool length: command this on the screen.
First display the MDI operation screen in the MDI operation mode.



Method to display automatic tool length measurement screen

With **[F1]** key pressed on the MDI operation screen, the machine enters the automatic tool length measurement mode, displaying the automatic tool length measurement screen as shown on the next page.

8



Setting method

· Tool

A single tool or more than one tools can be measured. Set the pot number for the tool(s) to be measured.

Ex:

When the setting is "1": The tool in pot 1 is measured.

When the setting is "1/10": Tools in pot from 1 to 10 are measured successively in the order that the pot number increases.

When the setting is "2/1": Tools in pot from 2 to 1 are measured successively in the order that the pot number increases. Since this case begins with "2," measurement ends with "1."

Measurements are taken of those tools only which are registered in magazines. When measuring more than one tools, the measuring position is all the same.

· Measuring position

For "0": XY position is equal to TOOL LENGTH MSM DEV INSTALL POSITN.

For "1": XY position is equal to TOOL LENGTH MSM DEV INSTALL POSITN + tool radius. Set the radius shift direction via XY POSITION RADIUSSHIFTDIRECTN.

For "2": Set the machining XY coordinates position.
When the cursor is on X or Y items, with [F4] key pressed, the current machining coordinates are taken in.

Reference

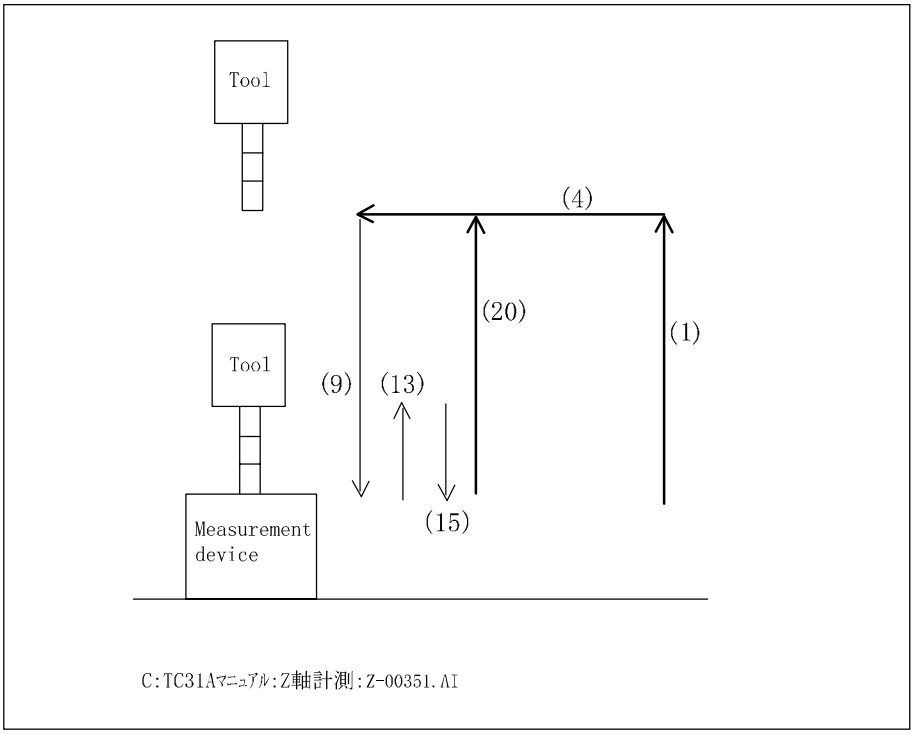
Once these values are set, they remain on the screen even after operation is completed. To clear the screen, press **[DELETE]**.

8.5.2 Operation

This section explains a series of operations for automatic measurement of tool length. The following show operation indications to appear on the screen and the description of the operations indicated.

(1) G0 Z__	Returns the Z axis to its home position.
(2) M19	Spindle orientation.
(3) M6 T__	Changes the tool.
(4) G0 X__ Y__	This allows XY coordinates to be positioned at the comanded point.
(5) M406	Air blow ON. If the cover does not open after 10 s, alarm * COVER DOESN'T OPEN occurs.
(6) M322	Cover open check.
(7) M19	Spindle orientation.
(8) M321	Detection signal OFF check.
(9) G131 Z__ F__	Measurement motion (at MEASUREMENT FEEDRATE1).
(10) G01 Z__ F__	Feed (at MEASUREMENT FEEDRATE1).
(11) M321	Detection signal OFF check.
(12) G131 Z__ F__	Measurement motion(at MEASUREMENT FEEDRATE2).
(13) M311	Measurement display
(14) M312	Set the tool length.
(15) G0 Z__	Returns the Z axis to its home position(Z).
(16) M19	Spindle orientation.
(17) M6 T__	Changes the tool.
(18) G0 X__ Y__	This allows XY coordinates to be positioned at the comanded point. Repeat (6) to (13).
(19) M312	Set the tool length.
(20) G0 Z__	Returns the Z axis to its home position(Z).
(21) M407	Air biow OFF

8



Supplementary explanation

- Use **[SINGL]** key for single operation.
- Manual operation in the MDI operation mode is possible.
- With **[MANU]** pressed while pressing **[MDI]** key, axis movement is possible with **[JOG]** key even if the current mode is set to automatic tool length measurement.
  Axis movement is possible with **[+/-] [RPD] [JOG]** and **[STEP]** keys.
- If **[START]** key is pressed ahead of other keys, alarms may occur. Relationship between circumstances under which the key is pressed and the resulting alarms is as shown below.

Circumstances under which the key is pressed	Resulting alarm
PROTECT switch is ON.	「*PROGRMPROTECT」
Tool menu is being edited.	「TOOLMENUEDITING」
Tool is absent.	「*NO TOOL IN MGZN」
Specified XY position is absent.	「DATA IMCOMPLETE」

- If magazine tools are changed during automatic measurement of tool length, alarm *CHG TOOL DG WK occurs.

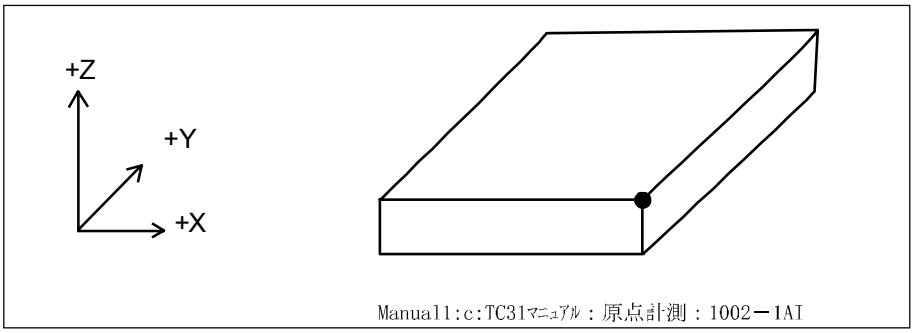
- Single stop in operation (8), SENSOR OFF CONFTN
  When FEEDRATE A for the small diameter is chosen for measurement in operation (9), even if the current operation is cyclic, execute single stop before performing operation (8).
  At this operation the machine becomes ready for accepting manual operation in the key input operation mode, thus displaying message AFTER LOWERING Z MANUALLY CLOSE TO MEASUREMENT DEVICE, PRESS START KEY. Press [-Z] key to move Z-axis to the vicinity of the measurement device and thereafter re-start. This shortens measurement time.

CHAPTER 9

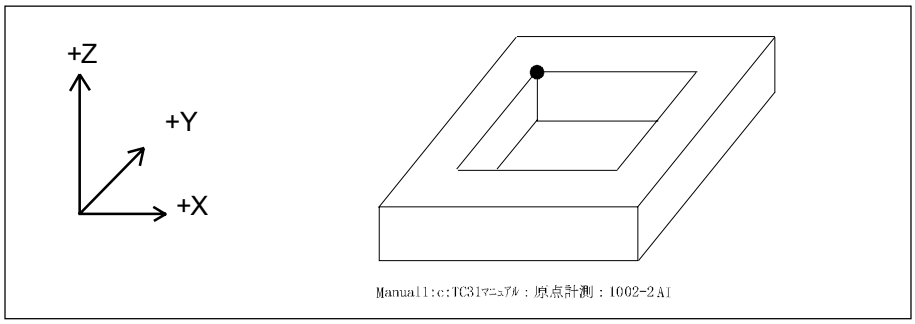
ORIGIN MEASUREMENT

Origin measurement
Origin measuring functions

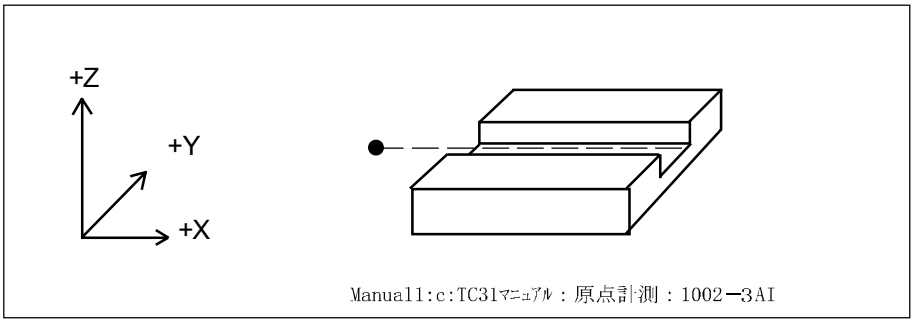
1. G121 -- X and Y coordinates of a corner



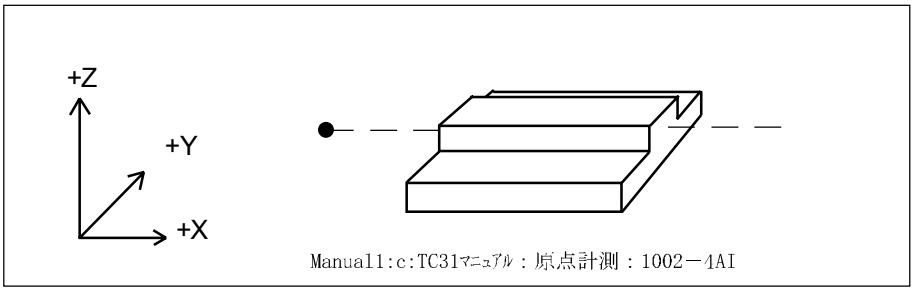
2. G129 -- X and Y coordinates of a groove



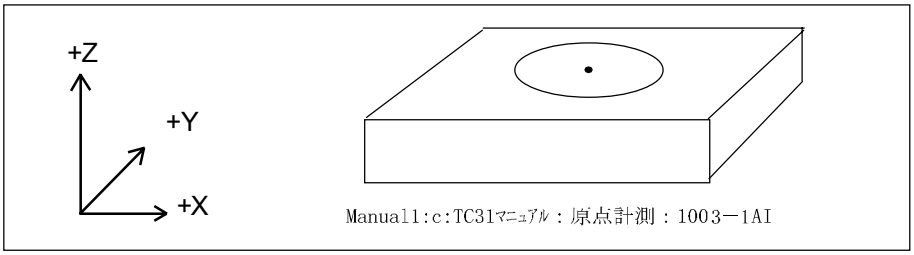
3. G122 -- X and Y coordinates of the axis of parallel groove



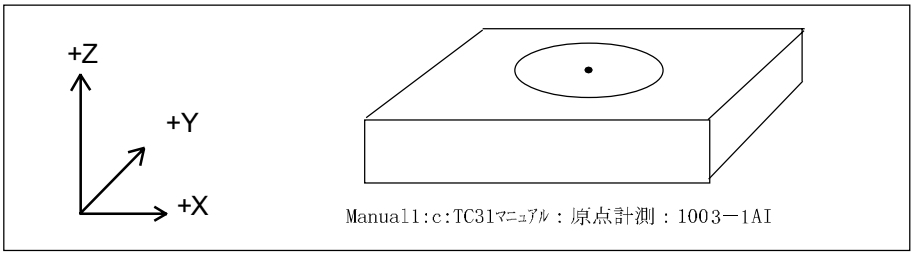
4. G123 -- X and Y coordinates of the axis of parallel bos



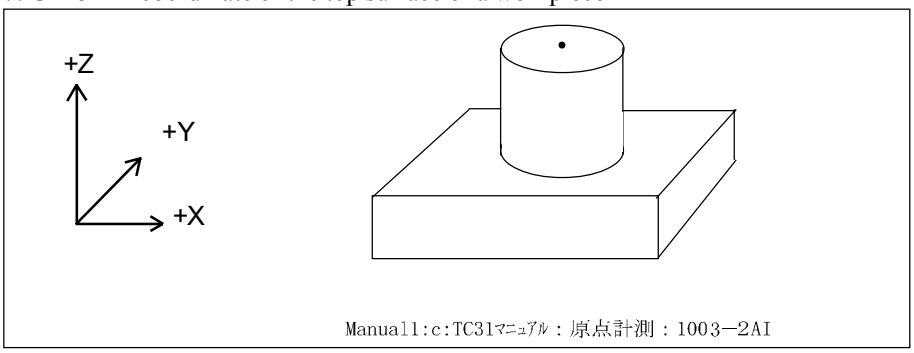
5. G124,G126 -- X and Y coordinates of the center of a hole

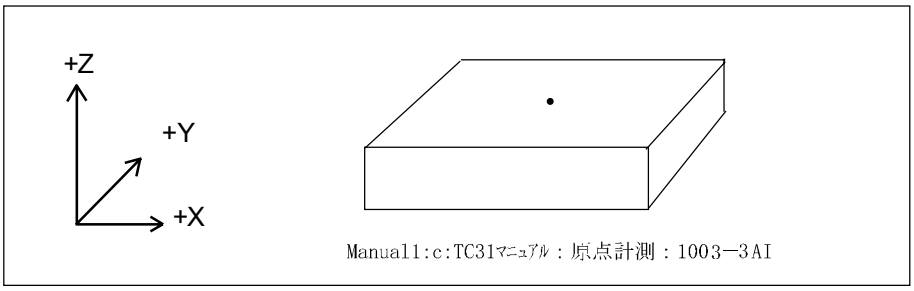


6. G125,G127 -- X and Y coordinates of the center of a boss



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





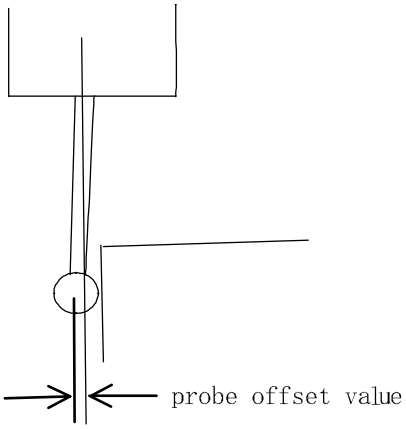
9

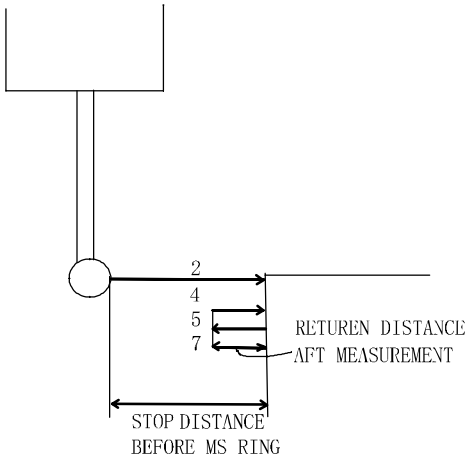
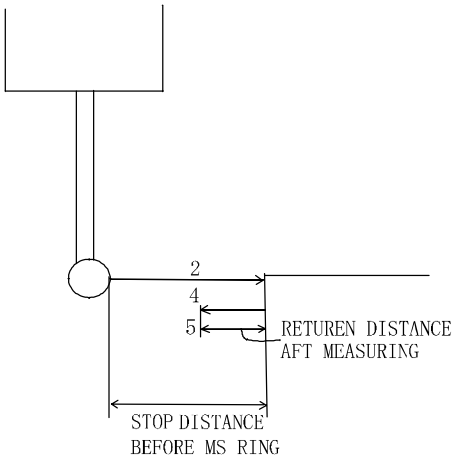
9.1 Before origin measurement

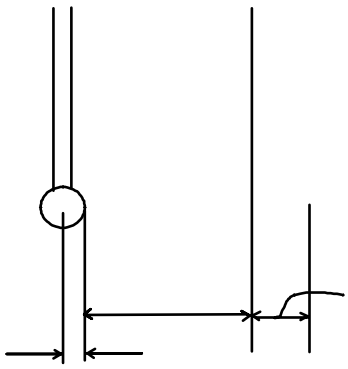
Set the necessary parameters of User Parameter 7 (ZERO MEASUREMENT).
Unless the parameters are set correctly, the probe may be damaged.

9.2 Setting of data on origin measurement

User Parameter 7 (ZERO MEASUREMENT)

Item	Description
PROBEOFFSETVALUE1 PROBEOFFSETVALUE2	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
PROBEOFFSETVALUE3 PROBEOFFSETVALUE4	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. To 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:TYPE1l: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.  Manual1:c:TC31ﾏﾆｭｱﾙ：原点計測：09L05 (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 MEASURING SPEED 2. 3) When the detection signal has turned on, the axis stops traveling. 4) The probe moves forward for the RETURN DISTANCE AFT MEASURNG. 5) It is checked that the detection signal is off.  Manual1:c:TC31ﾏﾆｭｱﾙ：原点計測：09L07

Item	Description
MEASURINGSPEED1	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) $L \geqq \underbrace{((F1 \times td) \div (60 \times 1000))}_{\text{Overtravel amount due to delay in control system}} + \underbrace{((F1 \times t/2) \div (60 \times 1000))}_{\text{Overtravel amount}}$ $F1 \leqq (120000L \div (24 + t)) \div \underbrace{1.2}_{\text{Safety rate}}$ Setting range:1 ～ 5000 mm/min 0.1 ～ 196.8 inch/min
MEASURINGSPEED2	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) $E \geqq (F2 \times td) \div 60 \longrightarrow F2 \leqq 120 \times E$ Setting range:1 ～ 5000 mm/min 0.1 ～ 196.8 inch/min
STOPDISTANCE 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.  Manual:c:TC31ﾏﾆｭｱﾙ：原点計測：1006-1AI Setting range:0.000~99.999mm 0.0000~9.9999inch

Item	Description
MEASURINGTRAVELLMT DISTANCE	Sets the amount of overtravel when the measuring skip has exceeded the estimated value(program command value). Setting range:0.000~99.999mm 0.0000~9.9999inch
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. Settingrange:0.000~99.999mm 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 origin 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:0.000~99.999mm 0.0000~9.9999inch
MEASURNG RETURNDISTANCEAFT	Sets the amount that the measuring probe retracts where it has contacted the non measuring object during automatic measurement. *RETURNDISTANCEAFTMEASURNG=Lb(mm) *SKIP FEED TIME CONSTANT 1=t(msec) *MEASURINGSPEED2=F2(mm/min) $Lb \geqq MAX\ (1.0,\ F2 \times t/6000)$ Setting range:0.000~99.999mm 0.0000~9.9999inch

(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 origin 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 origin measurement with the tool length 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 origin measurement.
If mode selection is attempted, DURING ORIGIN MEASURNG alarm will occur.

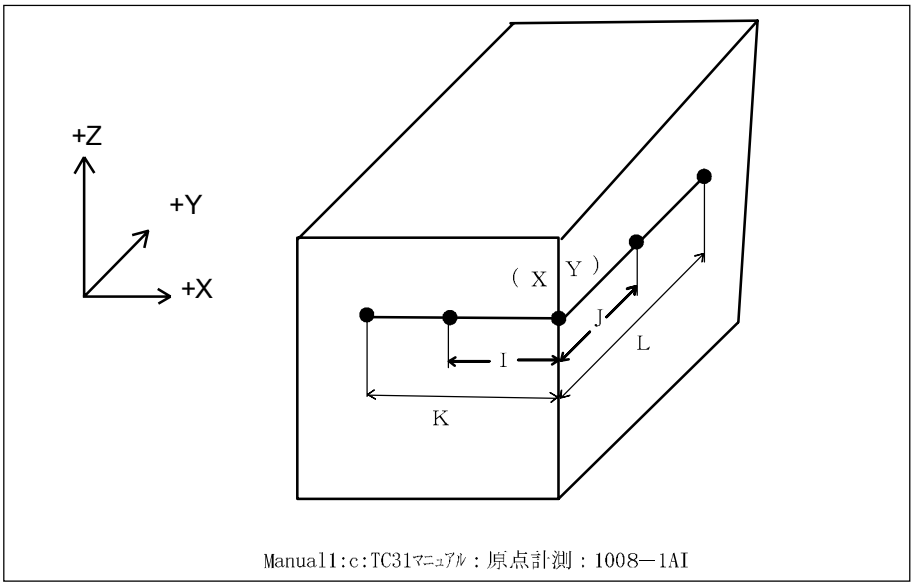
·To prevent the probe from being damaged, press the **[SINGL]** key for the first origin measurement and check the condition every single operation.

9.3 Operation of origin measurement

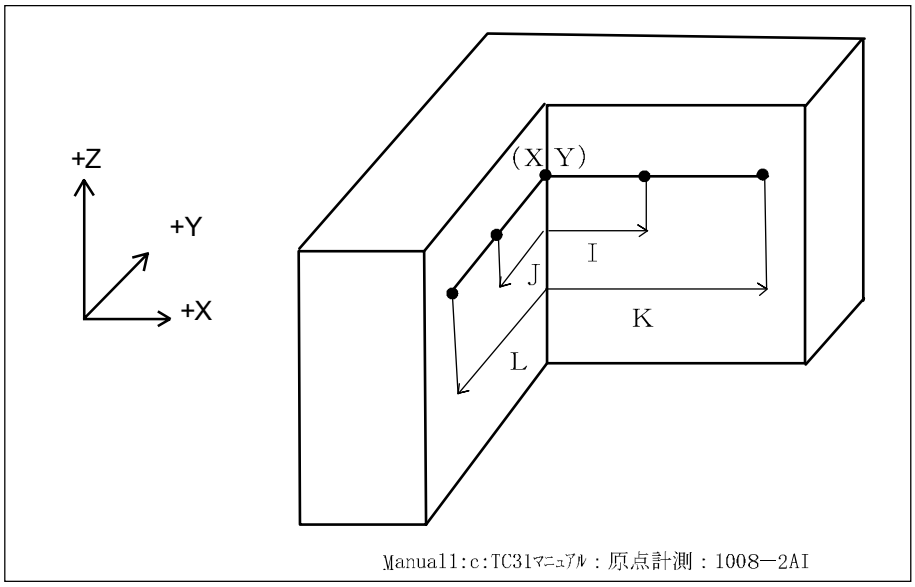
9.3.1 Corner

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_ ;
```



Manual1:c:TC31ﾏﾆｭｱﾙ：原点計測：1008－1AI



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 measure ment point to the other measurement point or when the movement has com pleted.
Q ...Register No. that stores the measured results ("1" when omitted)

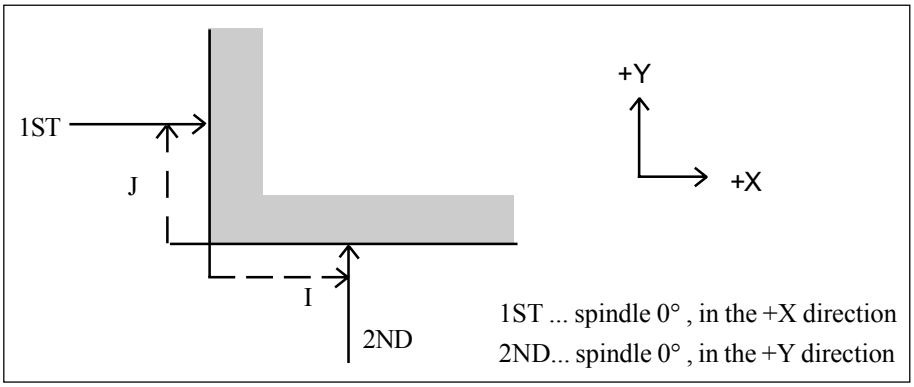
Note 1)
When either “I” or “J” is not commanded , when “I” is zero (o) , or when “J” is zero (0), an alarm occurs.
Note 2)
When "K" or "L" is commanded, the tilt angle (the angle used for coordinate rotation) is calculated should the work piece become tilted. An [ZERO MESR ADRS ERR] error will occur 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

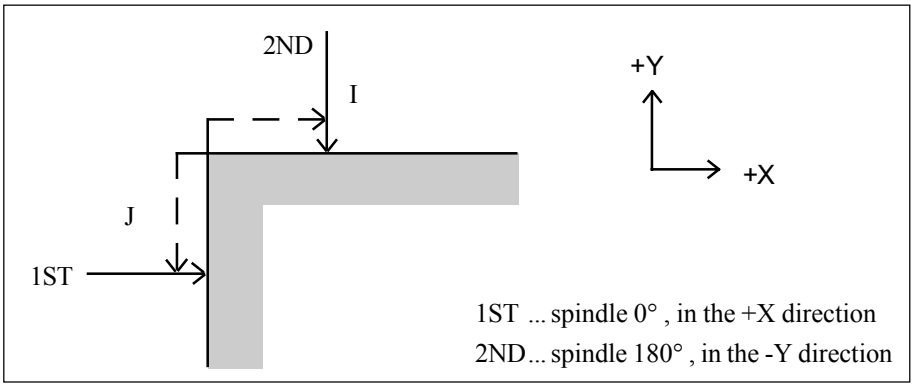
1. The spindle is oriented. The probe moves to the first measurement start point of the X and Y axes.
2. The probe moves to the Z axis measurement height.
3. The first measurement is carried out (Position "J").
4. Measured by position "L" when "L" is commanded
5. The probe moves to the Z axis return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

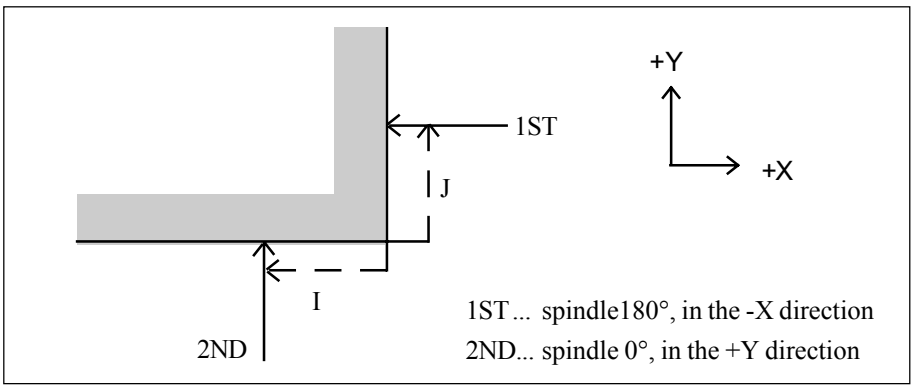
a） I＞0、 J＞0



b） I＞0、 J＜0

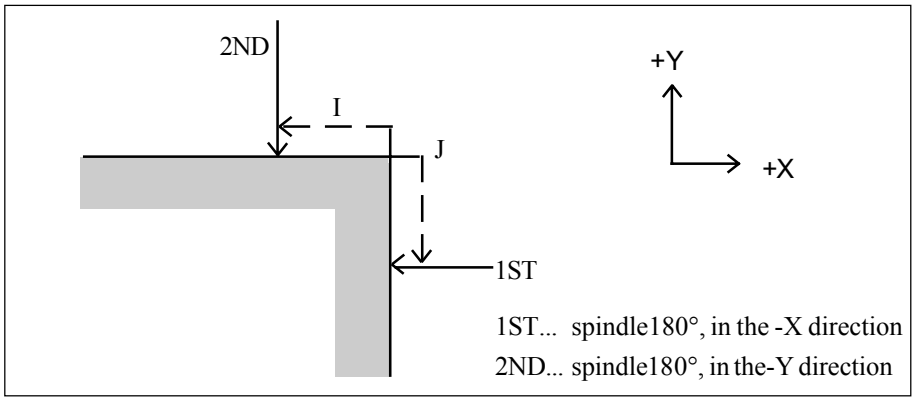


c） I＞0、 J＜0



9

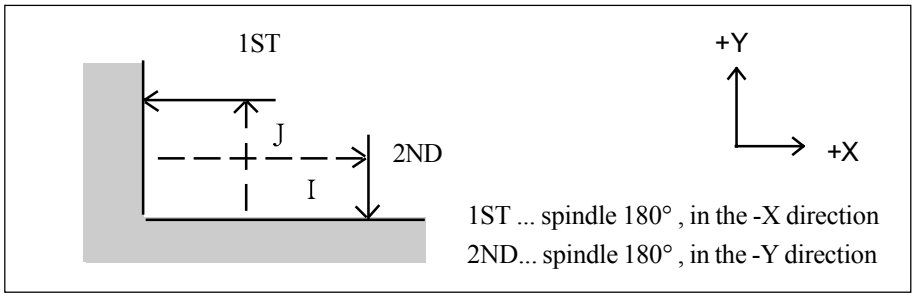
d) I＜0、J＜0



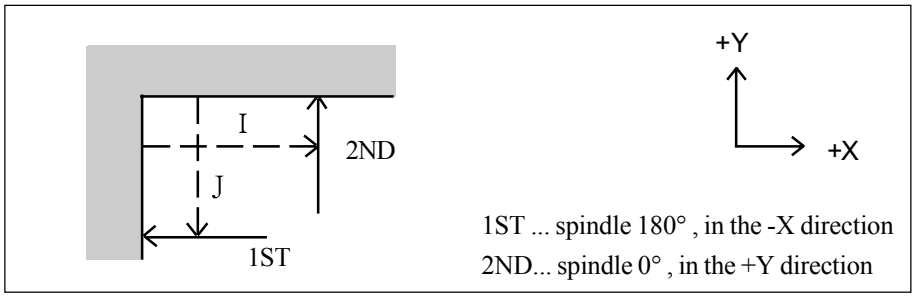
Measurment pattern

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 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.

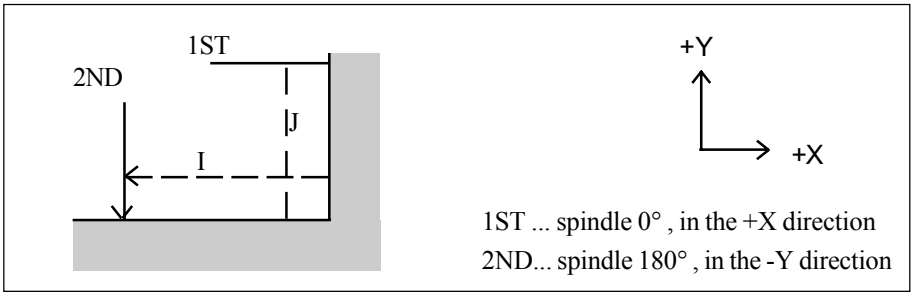
a) I＜0、J＜0



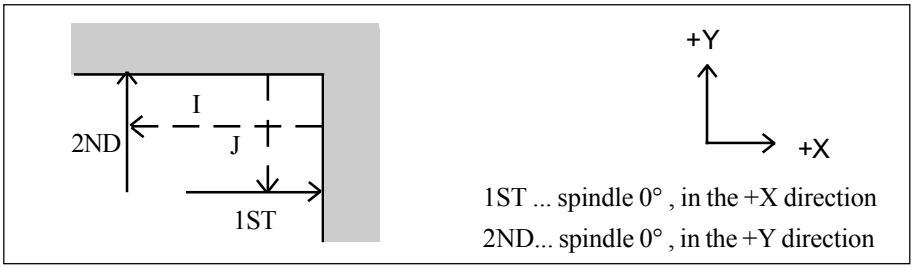
b) I＜0、J＞0



c) I＞0、 J＜0



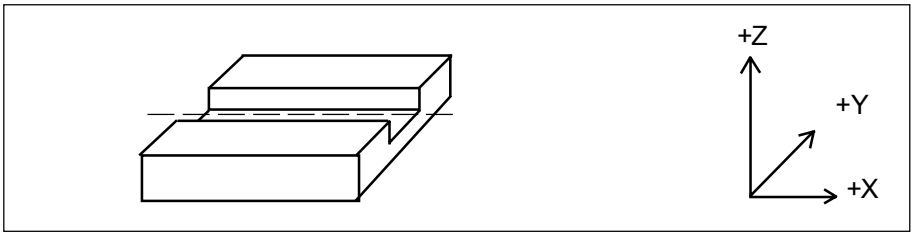
d) I＞0、 J＞0

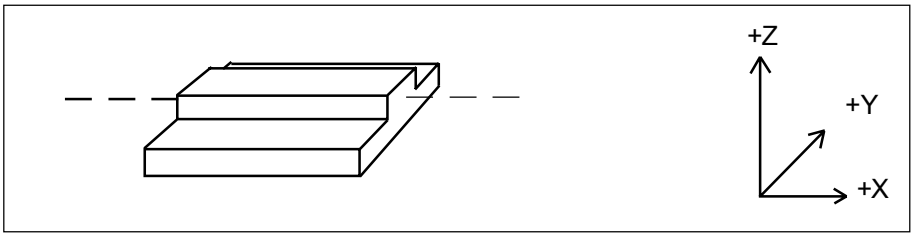


9.3.2 Parallel

Command format

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





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.

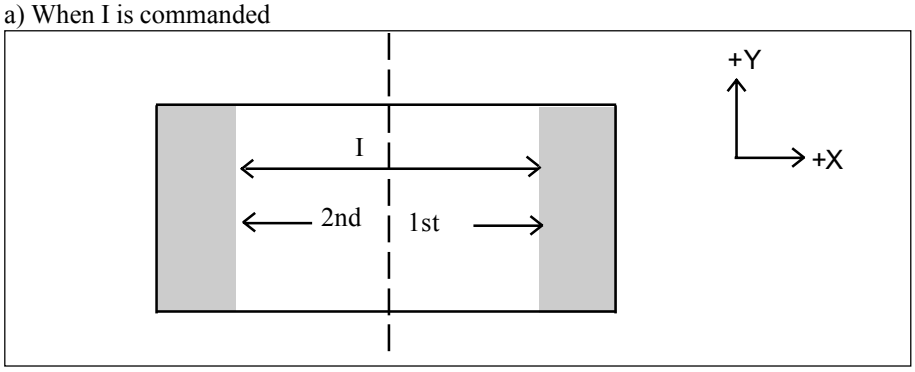
Q ... Register No. that stores the measured results ("1" when omitted)

Groove

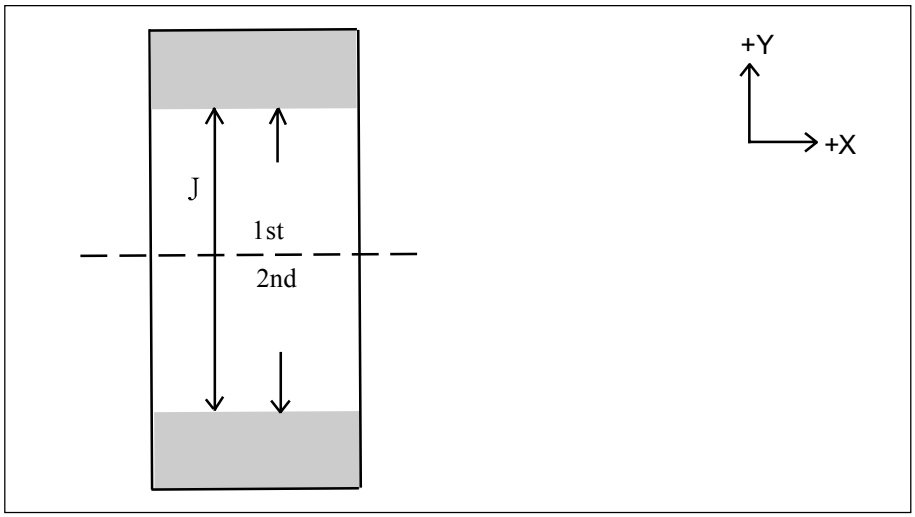
Measurment pattern

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 .
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



b)When J is commanded



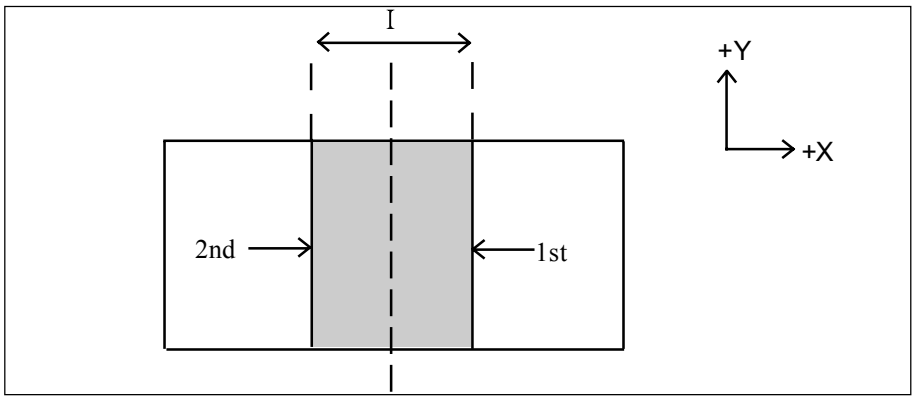
9

Boss

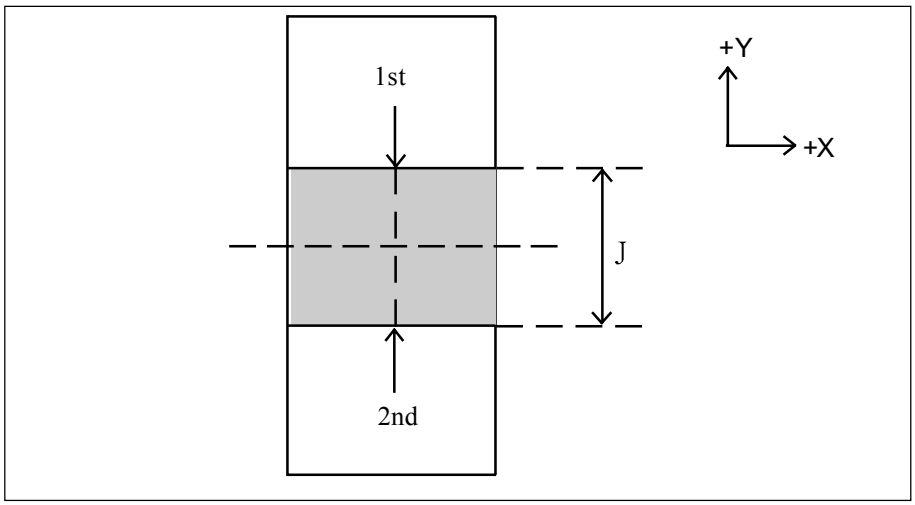
Measurment pattern

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



b) When J is commanded



9

9.3.3 Circle

The circle center is calculated by measuring three points.

Command format

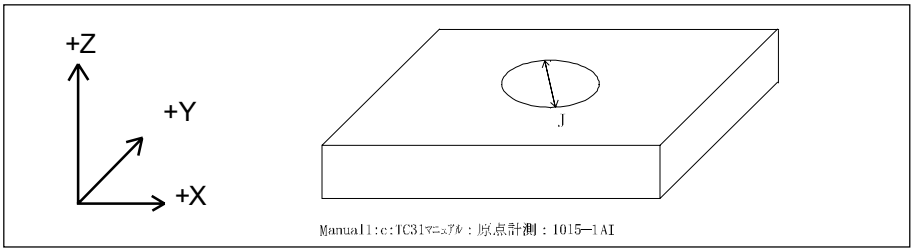
```
hole  G124 X_ Y_ I_ D_Z_ R_Q_
boss  G125 X_ Y_ I_ D_ Z_ R_Q_
```

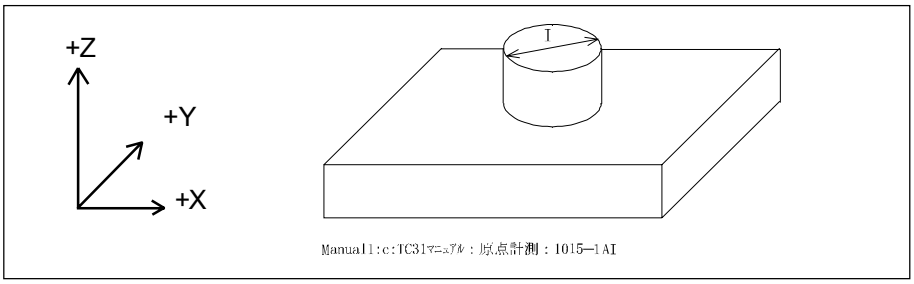
The circle center is calcutaled by measuring four points.

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)



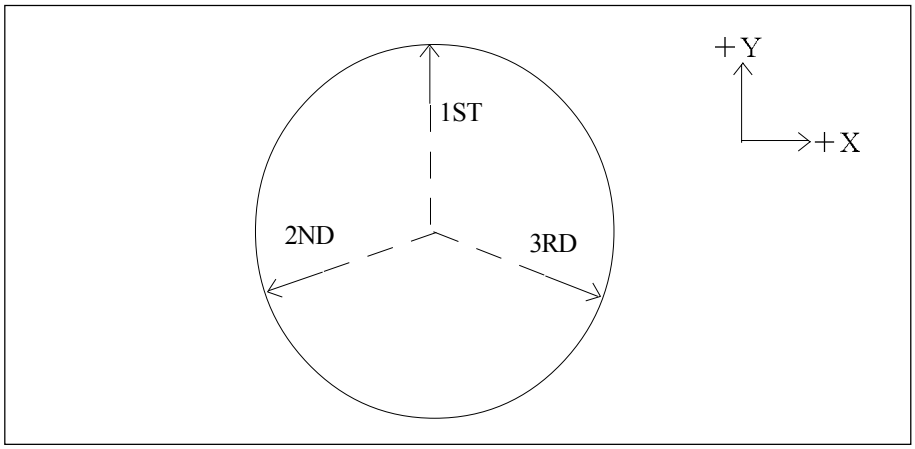


9

Measurment pattern

Hole ... Three-point measurement

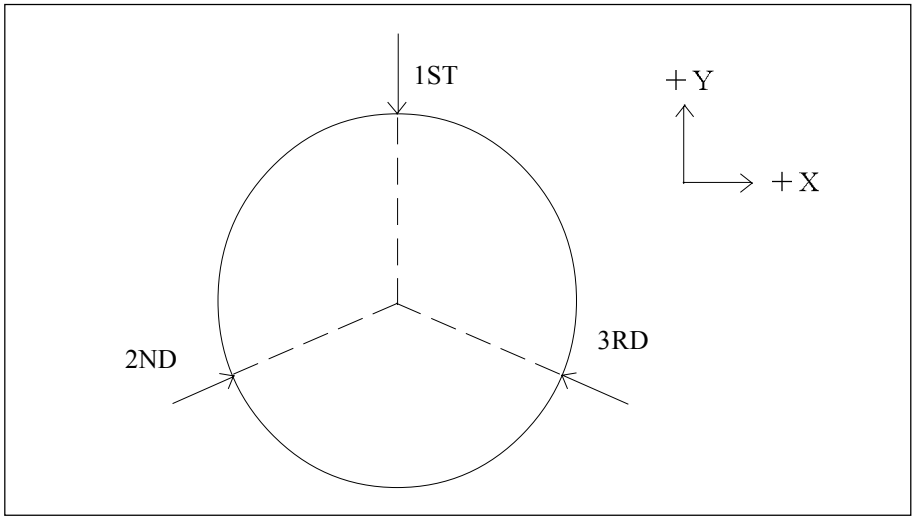
1. Spindle orientation 0•K. The probe moves to the first measurement start point of the X and Y axes.
2. The probe moves to the Z axis measurement height.
3. The first measurement is carried out (in the +Y direction).
4. Spindle orientation 0° . The probe moves to the second measurement start point of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



9

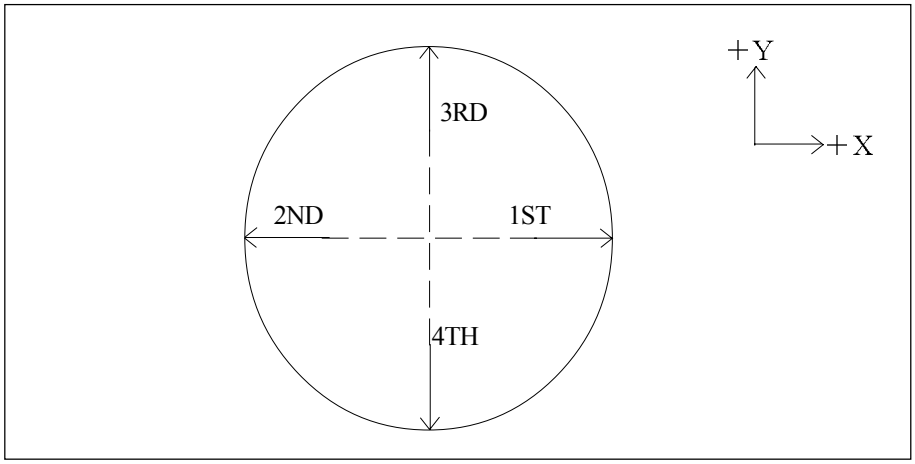
Boss ⋯ Three-point measurement

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.



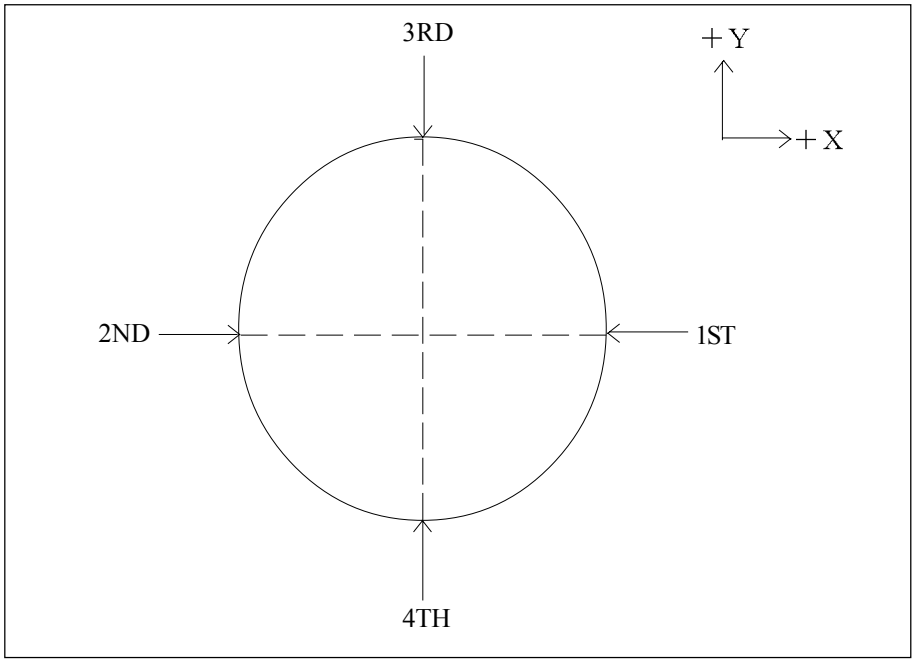
Hole ... Four-point measurement

1. Spindle orientation 0• K. 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• K. 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• K. 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• K. 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.



Boss ... Four-point measurement

1. Spindle orientation 180• K. 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.

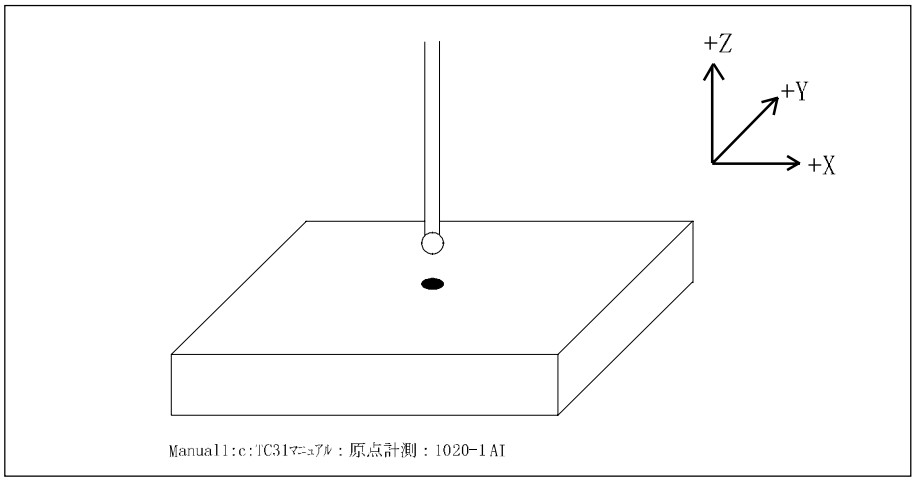


9.3.4 Z LEVEL

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)



Measurment pattern

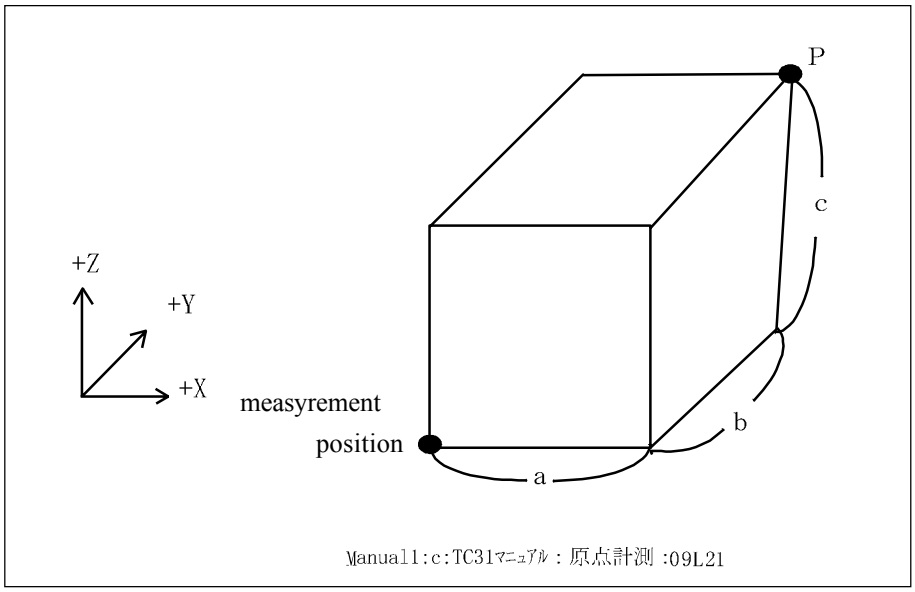
1. Spindle orientation 0° . The probe moves to the first measurement start point of the X and Y axes.
2. The probe moves to the Z axis measurement height.
3. The first measurement is carried out (in the -Z direction).

9.3.5 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).

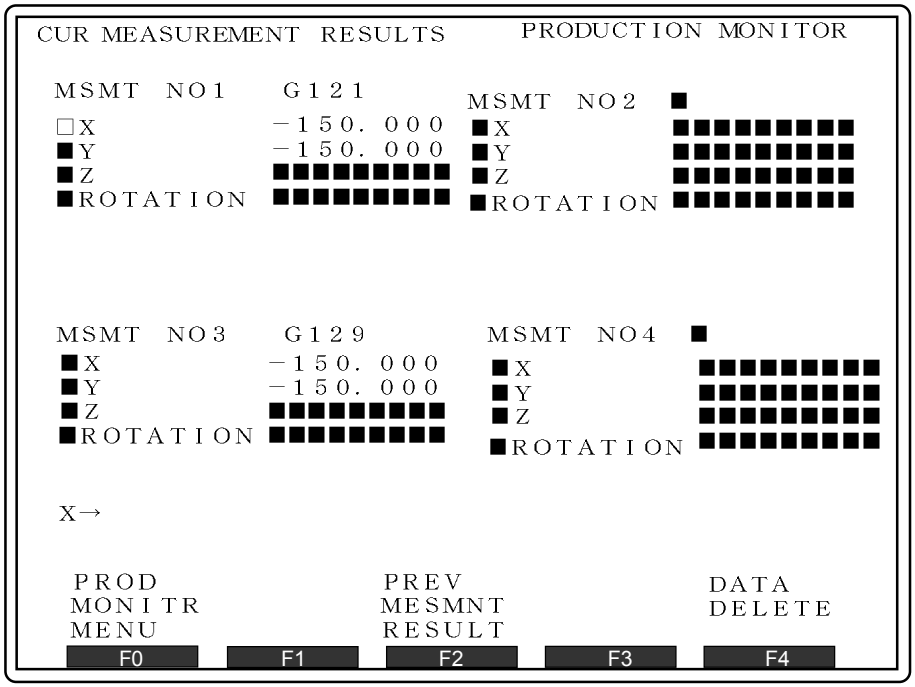


G120 Xa Yb Zc

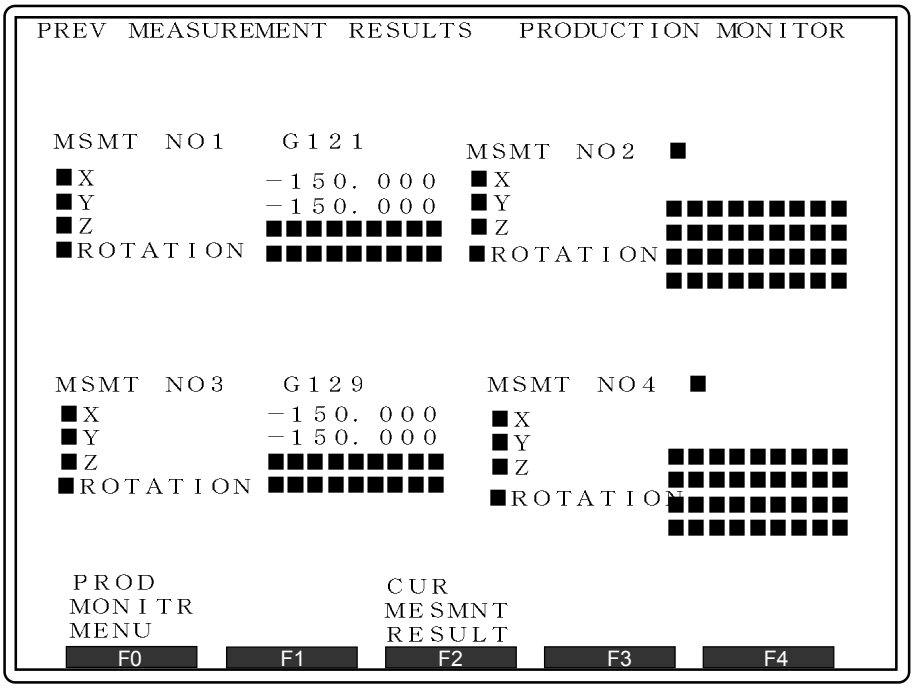
When **[G120 Xa, Yb, and Zc]** is commanded, the probe moves to point P.
When the measurement data does not exist, NO MEASRUING DATA alarm will occur.

9.4 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.



Press the [F2] key and the previous measurement results are displayed.



The measured results are reflected in the working 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

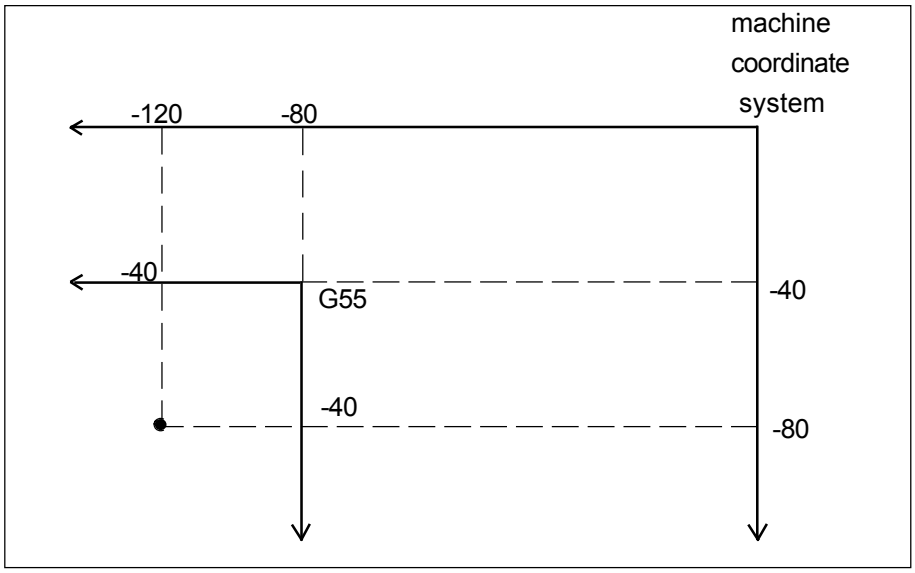
Command format

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.

9.5 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

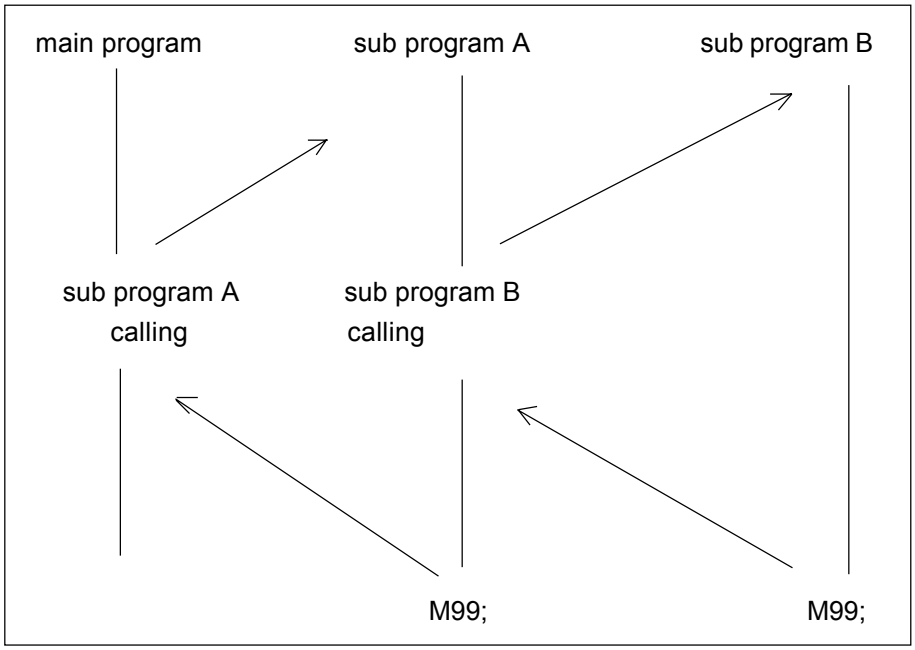
When G121 to G129 are commanded during restarting, RESTART ERROR will occur.

CHAPTER 10

SUBPROGRAM FUNCTION

10. Function of subprogram

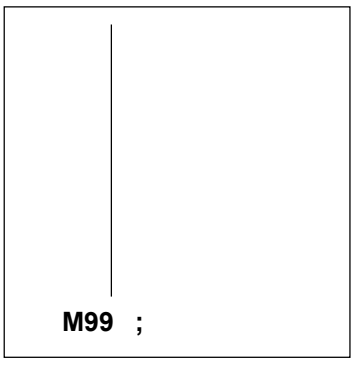
When a program contains fixed sequences or frequently repeated patterns, these sequences or patterns may be entered into the memory as a subprogram.
The subprogram can be called out in the memory operation mode.
A subprogram called by the main program can also call another subprograms.
Max 8 fold nesting is available.



One calling command can call a subprogram repeatedly.

10.1 Making subprogram

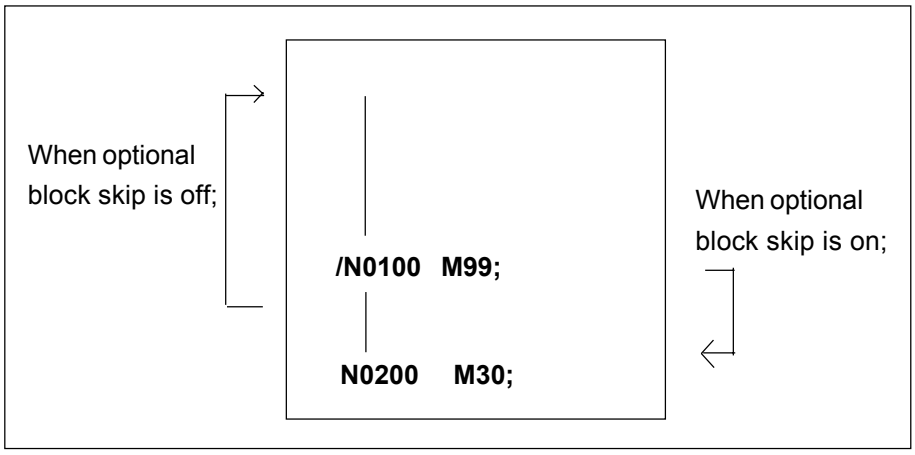
Generally, a subprogram is made by the following format.



(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.



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_ ;
```

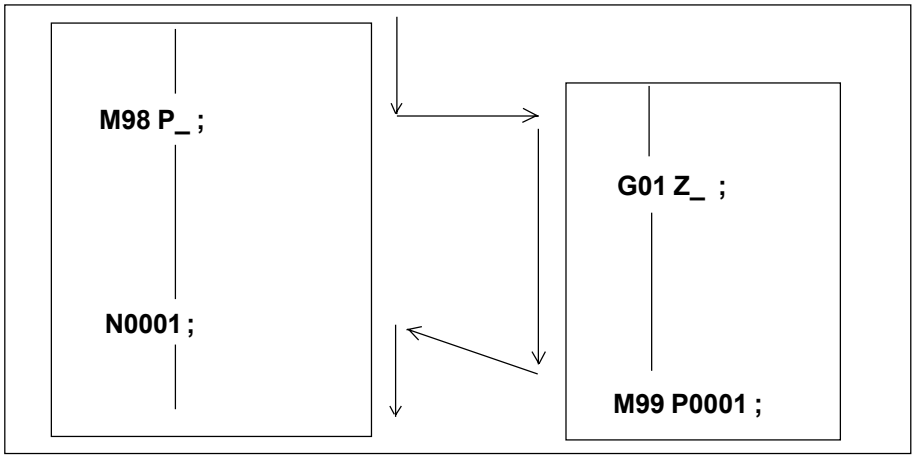
P : Sequence No.

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.



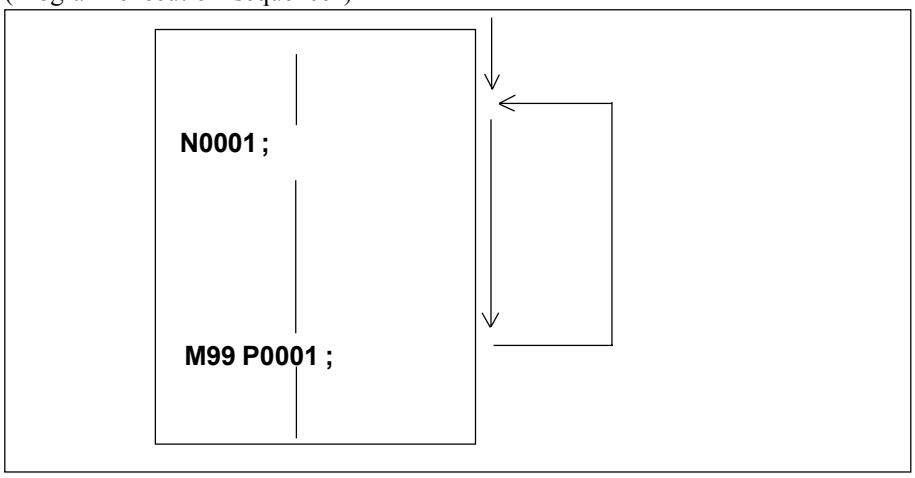
(Note)An alarm will occur unless the number of repeats commanded by M98 is "1."

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)



CHAPTER 11

FEED FUNCTION

11. Feed function

Feedrate is specified by the number following address F.

(1) Command range

Metric system : 1~999999mm/min
1~999999° /min
Inch system : 0.1~99999.9inch/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. (TC-31A,22A,32A,20A)

Three commanded forms are provided as below.

12.2.1 Commanded by tool No.

Command fromat | **T_ _ _;**

Commanded by tool No. after "T" (T1-T99), and the pot with the corresponding tool attached is Indexed. (TC-31A,22A,32A,20A)

12

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) (TC-31A,22A,32A,20A)

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. (TC-31A,22A,32A,20A)

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	Operration order vs. axis feed	Modal/ one-shot
	M00	Program stop	after	one-shot
	M01	Optinal 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	before	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.

List of M code (2)

Group	M code	Content	Operration order vs. axis feed	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	
	M458	Coolant blow On (Coolant blow)	simultaneous	modal
	M459 *	Coolant blow Off (Coolant blow)		
	M490	High pressure coolant On (Coolant blow)	before	modal
	M491			
	M492			
	M493			
	M494	(1)High pressure coolant type High pressure coolant On (Coolant blow) (2)CTS type CTS On		
	M495 *	(1)High pressure coolant type High pressure coolant Off (Coolant blow) (2)CTS type CTS Off	after	
	M496	Back washing cycle On	before	one-shot

12

List of M code (2)

Group	M code	Content	Operration order vs. axis feed	Modal/ one-shot
	M211	Workpiece counter 1set	Simultaneous	Modal
	M221*	Workpiece counter 1cancel		
	M212	Workpiece counter 2set	Simultaneous	Modal
	M222*	Workpiece counter 2cancel		
	M213	Workpiece counter 3set	Simultaneous	Modal
	M223*	Workpiece counter 3cancel		
	M214	Workpiece counter 4set	Simultaneous	Modal
	M224*	Workpiece counter 4cancel		
	M230*	Tool life counter set	Simultaneous	Modal
	M231	Tool life counter cansel		
	M232	Automatic corner deceleration on	Simultaneous	Modal
	M233	Automatic corner deceleration off		
	M800~ M899	Signal output * for sequence controller	Simultaneous	One-shot

The code with * is already set when the power is turned on.

12

List of M code (3)

Group	M code	Content	Operration order vs. axis feed	Modal/ one-shot
	M120	TOUCH signal check	after	one-shot
	M200	Tool breakage detection (with return motion)	after	one-shot
	M201	Tool breakage detection		
	M203	Tool breakage detection judgement	___	one-shot
	M206	Thermal watch condition judgement	___	one-shot
	M207	Thermal measurement motion	___	one-shot
	M320	Measurement device sensor ON confirmation	___	one-shot
	M321	Measurement device sensor OFF confirmation	___	one-shot
	M322	Sensor cover open check	___	one-shot
	M332	Z-Axis position correction	before	one-shot
	M350	Thermal displacement compensation(x)	before	one-shot
	M351	Thermal displacement compensation(Y)	before	one-shot
	M352	Thermal displacement compensation(Z)		
	M353	Thermal displacement compensation (XYZ)		
	M438	Magazine cover open	simultaneous	one-shot
	M439	Magazine cover close		

12

List of M code (3)

Group	M code	Content	Operration order vs. axis feed	Modal/ one-shot
	M241	Tap time constant 10%	simultaneous	modal
	M242	Tap time constant 20%		
	M243	Tap time constant 30%		
	M244	Tap time constant 40%		
	M245	Tap time constant 50%		
	M246	Tap time constant 60%		
	M247	Tap time constant 70%		
	M248	Tap time constant 80%		
	M249	Tap time constant 90%		
	M250*	Tap time constant 100%		
	M234	High-accurate mode ON	simultaneous	modal
	M235*	High-accurate mode OFF		
	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		

The code with * is already set when the power is turned on.

12

List of M code (4)

Group	M code	Content	Operration 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 untill 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 forM464 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		

12

List of M code (4)

Group	M code	Content	Operration order vs. axis feed	Modal/ one-shot
	M480	M480 signal ON	simultaneous	modal
	M481*	M480 signal OFF		
	M482	M482 signal ON		
	M483 *	M482 signalOFF		
	M484	M484 signalON		
	M485 *	M484 signalOFF		
	M486	M486 signalON		
	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

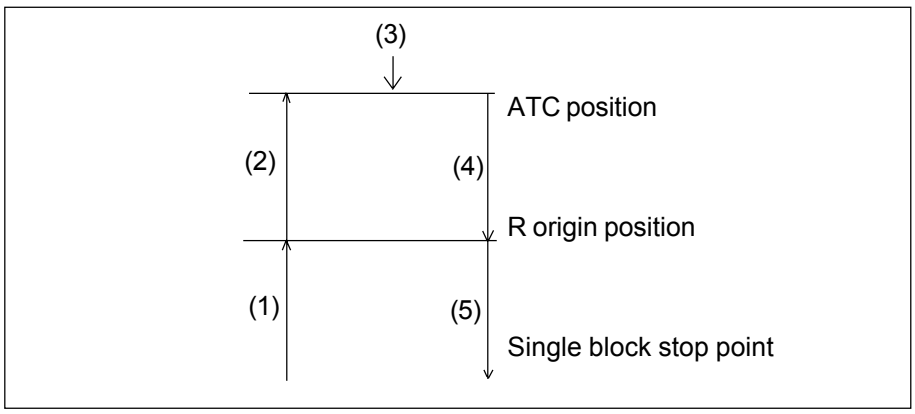
· Using M401 command turns off the chip shower when the time set for [CHIP SHOWER DRAIN TIME] (user parameter - switch 1) has elapsed.

12.3.6 Tool change (M06)

(1) When TC-31A,22A

Command format

```
M06 T _ X _ Y _ Z _ R _ A _ B _ L _;
```



12

T _ _ : Tool number.
T1 : Pot number
T9 : Group number

X, Y A,B Target value when moving the X , Y ,A, and Baxes at in rapid feed simultaneously with tool change motion.

Z : The target value of movement‡D in rapid feed.
R : Return position before tool change motion
(operatedated with tool length compensation applied)
L : Specify the tool number, pot (magazine) number, and group number after “L”. The pot with the corresponding tool attached is indexed by operation (3) after ATC. (Next tool preparation)

Operations
(1)Tool moves to Z axis point "R" while performing O-degree spindle orientation.
(2)Tool moves to Z-axis ATC origin while X,Y,A,and Baxes move to commanded value.
When "T"is commanded tool moves to Z-axis ATCorigin and magazine swivels.
(3)Arm swivels to change tool.
Tool change motion differs depending on setting on **[MAGAZINE TOOL]**screen.
See section on "Tool Change Motion"for details.
(4)Tool moves to Z-axis point"R".
(5)Tool moves to X-axis commanded position.

Caution
When performing cycle operation,tool moves in cutting mode between operations (1) and (2) and operrations (4) and (5).

When **[RESET]**key or **[STOP SWICH]**key is pressed between operrations (2) and (4),machine stops after motion (4) is completed. For operation (3) (X,Y,A and axes movement),machine stops immediately.
Machine does not stop if single block occurs between operations (1) and (4).

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 Yaxes movement 2 commences. Start-up is performed in format 1, regardless of parameters.
· When tool offset is commanded to block M06 ,tool offset becomes valid from operation (4) .
· When M06 commanded is while tool length compensation (G49)is canceled, operation (1),is performed subject to **[ATC REFERRENCE]** selection for user parameter.
· Alarm appears when axis A or B is commanded while optional A and B axis are not in use.
· When point "R” commanded position(4)is lower than Zaxis command position (5), tool moves to Z-axis commanded position , and operation (5) 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 at high speed.
(5)Pot raises.
When Pot is already at upper limit before tool change motion commences, sequences starts from (2).

When changing from large tool to large tool, arm swivels at low speed.
When 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 at low speed when large tool is to be changed, and at high speed when standard tool is to be changed.
Operation (9) is performed for the TC-31A only. However, it may also be performed for the TC-22A when “next tool preparation” is performed.

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 large 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 (not performed for TC-31A)

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

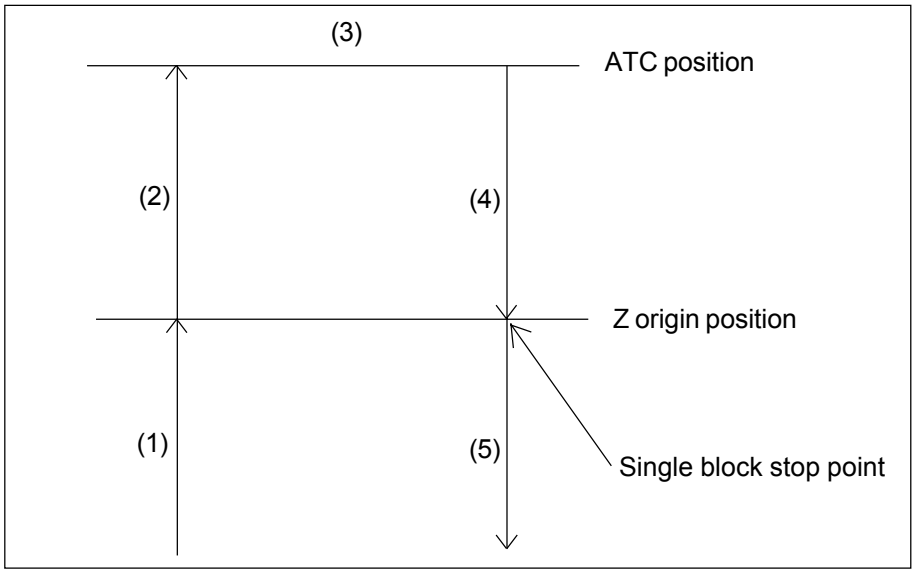
(1) Pot rises
(2) Magazine swivels (empty pot is indexed)
(3) Pot lowers (not performed for TC-31A)

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.

(2) When TC-S2A, TC-R2A or TC-S2B.

Command format **M06 T _ X _ Y _ Z _ A _ B _ L _ ;**



T _ _	:	Tool number.
T1 _ _	:	Magazine number.
T9 _ _	:	Group number.
X, Y, A, B	:	Target value when moving X, Y, A, and B axes at in rapid feed simultaneously with tool change motion.
Z	:	Target value of movement in rapid feed. Command the distance from the Z origin when the incremental mode is selected.
L	:	Specify the tool number, magazine number, and group number after "L". The number specified by "L" becomes the T model after M06.

12

Operations

(1) Tool moves to the Z-axis point and spindle orientation are executed at the same time.

(2) Movement to the ATC origin position.
When "X, Y, A or B" is commanded in the same block, the motions are done at the same time.

(3) The magazine indexes the tool as commanded by the address "T".

(4) Movement to the Z-axis origin position.

(5) Movement to the commanded position when Z-axis is commanded.
When the spindle command (M3 group) is given, the motion is done at the same time.

· When performing cycle operation, tool moves in cutting mode between operations (1) and (2)and operations (4) and (5).

· When **[RESET]** key or **[STOP SWITCH]** key is pressed between operations (2)and (4), machine stops after motion (4) is completed.
For operation (2) (X, Y, A, and B axes movement), machine stops immediately.

Caution

· **All data other than M06 can be omitted. However, code "T" must be commanded once by operator before M06 is commanded.**

· **The tool length offset and the tool dia offset are canceled when M06 is commanded.**
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 (2) commences.
Start-up is performed in format 1, regardless of parameters.

· **When tool offset is commanded to block M06, tool offset becomes valid from operation (5).**

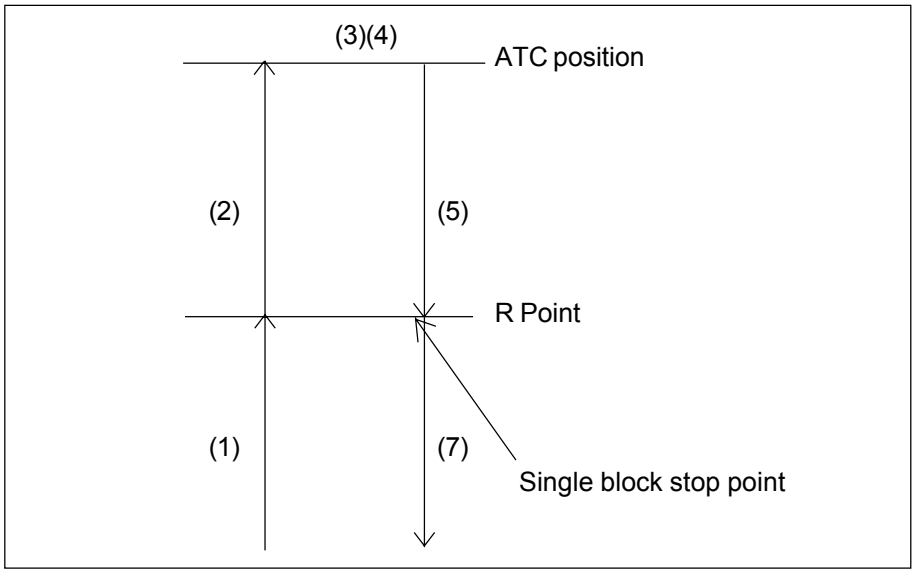
· **Alarm appears when the other code M is commanded to block M06.**

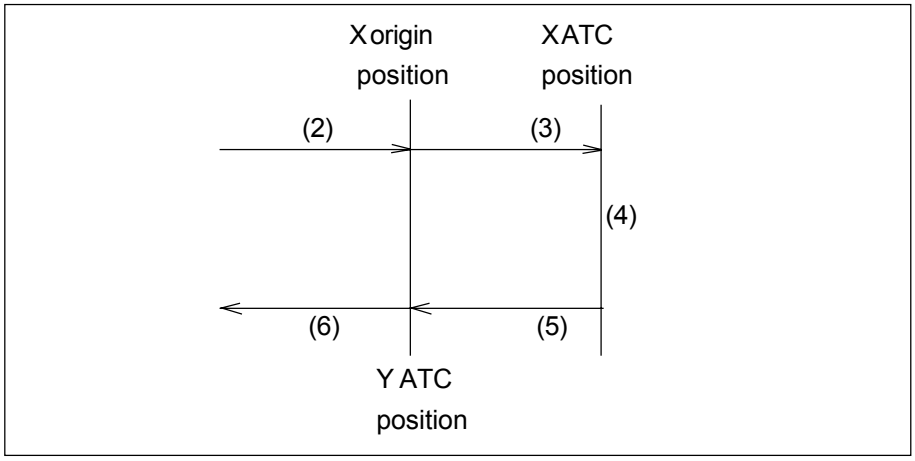
· **Alarm appears when axis A or B is commanded while optional A and B axes are not in use.**

12

(3) When TC-32A.

Command format | **M06 T _ X _ Y _ Z _ R _ A _ B _ L _ ;**





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)

12

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.

12

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.

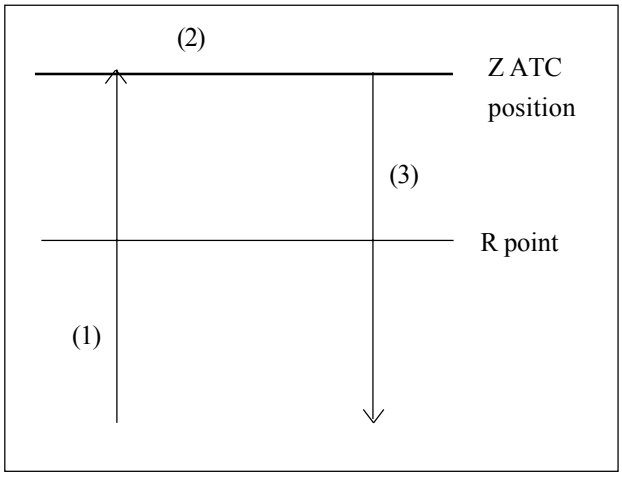
The sequence starts from (2) when the pot is already at the upper limit before next tool preparation commences.

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.

(4) When TC-20A.

Command format | **M06 T _ X _ Y _ Z _ R _ A _ B _ L _ ;**



T :T__ :Tool number
:T1__ :Pot number
:T9__ :Group number

X,Y,A,B: Target value when moving X, Y, A, and B axes in rapid feed simultaneously with ATC

Z : Target value of operation (3) in rapid feed

R : Return position before ATC (with tool length compensation applied)
The "Z ATC position" is regarded as "R command value + 50 mm".
When the "R" command is not specified, operation is performed regarding "Z origin - 50 mm" as "R" command value.

L : Specify the tool number, pot (magazine) number, and group number after "L".
The pot with the corresponding tool is indexed by operation (2) after ATC. (Next tool preparation)

12

Operations

(1) The tool moves to Z ATC position (50 mm above "R" point or Z origin) while performing 0-degree spindle orientation.
When the tool specified by "T" is indexed in the magazine, the pot lowers.
It starts lowering when the Z-axis has passed the position set for **[POT LOWERING START POS.]** (user parameter).

(2) When the tool specified by "T" is not indexed in the magazine, the magazine swivels and then the pot lowers.
When the specified tool is indexed in the magazine and the pot is at the lower limit, the arm swivels and then the pot rises.
X, Y, A, and B axes also move to the specified positions.
See the ATC section for details of ATC sequence.

(3) The tool moves to the specified Z-axis position.

Precautions

· The pot, magazine, and arm do not stop even when the **[RST]** key or the **[STOP]** button is pressed during operations 1) to 3) above. The X, Y, Z, A, and B axes stop immediately.

· All data other than M06 can be omitted. However, a T code must be used at least once before M06 is used.

· When M06 is used, tool diameter compensation is canceled. Tool length compensation is also canceled in operation 3).

· When the tool diameter compensation (G41 or G42) and XY axes movement commands are used for the M06 block, tool diameter compensation begins in format 1 when X and Y axes movement 2) commences, regardless of the current parameters.

· The tool length compensation command is used for the M06 block, tool length compensation become valid from operation 3).

· The Z-axis moves as described below during operation 1).

(1) When R point is specified for M06 block
The Z-axis moves to the position where the tool tip is "R point + 50 mm" in accordance with the tool length compensation.
The larger tool compensation value before or after change is applied.
When tool length compensation is not applied, the setting for **[ATC REFERENCE TOOL LENGTH]** (user parameter) is used.

(2) When R point is not specified for M06 block
The tool rises to the Z origin, regardless of the tool length.

· When the Z-axis movement and tool length compensation commands are not used for the M06 block, operation 3) is performed in accordance with the setting for **[ATC REFERENCE TOOL LENGTH]** (user parameter).

· An alarm occurs when an M code other than a spindle command is used for the M06 block.
An alarm occurs when the A- or B-axis command is used although the optional A- or B-axis is not installed.

· The target position the pot lowers to is determined by the distance between **[ATC RETURN HEIGHT]** and **[POT LOWERING START POSITION]** (user parameters).

ATC

(1) Magazine swivels
(2) Pot lowers
(3) Arm swivels
(4) Pot rises
ATC ends when the above sequence is completed.
Operation (3) (arm swivels) is performed at high speed.
When the tool breakage detection command is specified for tools in the magazine, detection is performed after operation (4) (pot rises).

Next tool preparation

Next tool preparation is performed after the pot has risen in the ATC sequence.
When ATC is not performed, only next tool preparation is performed.
When the next tool is already indexed, next tool preparation is not performed.
1) Magazine swivels
Next tool preparation ends when the above sequence is completed.

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)

·Automatic corner deceleration does not apply during tool compensation.
·This function is valid for commands that perform continuous XY-axes cutting movement.

12.3.10 M120 function

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.

12.3.11 Tool breakage detection (M200 and M201)

Operation procedures are described in Option, Tool Breakage Detection Unit in the Instruction Manual.

12.3.12 Tap time constant selection (M241 to 250)

Z-axis speed time constant while tapping is changed by the M241 to M250 M code command in the range of 10 to 100% of the maximum time constant.

When the M241 command is given, 10% of the maximum time constant is adapted when tapping.

When the M242 command is given, 20% of the maximum time constant is adapted when tapping.

When the M250 command is given, 100% of the maximum time constant is adapted when tapping.

For maximum tap rotation : 6000 rpm.

Spindle rotation speed when tapping(rpm)	0～600	601～1200	1201～1800	1801～2400	2401～3000
Optimum M code	M241 10%	M242 20%	M243 30%	M244 40%	M245 50%

Spindle rotation speed when tapping(rpm)	3001～3600	3601～4200	4201～4800	4801～5400	5401～6000
Optimum M code	M246 60%	M247 70%	M248 80%	M249 90%	M250 100%

For maximum tap rotation :8000 rpm.

Spindle rotation speed when tapping(rpm)	0～800	801～1600	1601～2400	2401～3200	3201～4000
Optimum M code	M241 10%	M242 20%	M243 30%	M244 40%	M245 50%

Spindle rotation speed when tapping(rpm)	4001～4800	4801～5600	5601～6400	6401～7200	7201～8000
Optimum M code	M246 60%	M247 70%	M248 80%	M249 90%	M250 100%

(Note 1)
When a tap time constant command taht will become shorter than the optimum time constant is given, the 'TOO SMALL TIME CONSTANT' alarm is generated and the tool moves using the maximum time constant.
(Note 2)
In tapping high-speed return mode, the tool moves using the maximum time constant. However, tool returns during stepping in the same conditions as for advancing.
(Note 3)
In dry run mode, the tool moves using the maximum time constant.

12.3.13 Pallet related M codes (M410, M411, M430, and M431)

· These functions are sffective for the TC-31A, TC-32A, TC-R2A.
· 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)

· These functions are sffective for the TC-22A, TC-32A, TC-20A, TC-S2B.
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.

(This page is a blank.)

CHAPTER 13

OPTION

13

13.1 Memory capacity expansion

When the memory is expanded, the capacity will be three times larger than it used to be. Then, more programs and data banks can be saved in the main machine.

Memory capacity (standard)	497 block
Memory capacity (after the expansion)	1509 block

(1block =256 byte)

There are same restrictions as those for the standard memory.

1. The maximum file number in total 256
   This number is decided by adding number of programs and data banks.

2. The maximum editable capacity for an one program. 256 block

3. Operational program capacity 512 block

[Note]

All block capacity can not be used as a program area.
(497 block for the standard one, 1509 for the expanded one)

(33 blocks as a maximum are used as a memory maintenance area.)

13.2 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.

13

693308001
0706(1)