PROGRAMMING MANUAL
FOR
VM40II, VG, VK, HG TYPE
MACHINING CENTER
SEIKI-SEICOS MIII

Edition 1 10-1992

MORI SEIKI
THE MACHINE TOOL COMPANY

C O N T E N T S

INTRODUCTION

Thank you for your selection and introduction of our Machining Center.

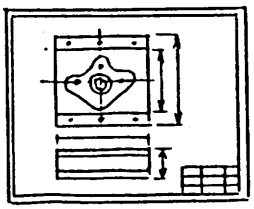
This manual describes the programming of Machining Center with SEIKI-SEICOS MⅢ. In order to use this machining center effectively, it is necessary to understand and program the features and functions of machine.

Explanation is made for necessary items on programming, such as the programming words and methods.

Accordingly, make programming upon full understanding of the contents.

As to the details of parameters etc., refer to the Instruction Manual for "SEIKI-SEICOS MⅢ".

1-1 FLOW CHART FOR MACHINING WORK BY MACHINING CENTER



Machining diagram

Function of using machine:
- stroke, machining faculty, accuracy, ATC faculty, work-limit

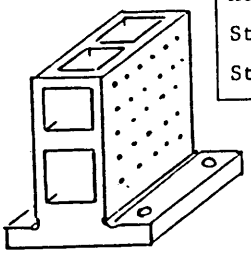
Selection of machining position

Mounting method

Tool, cutting tool

Program

Mounting device
Standard face
Stopper



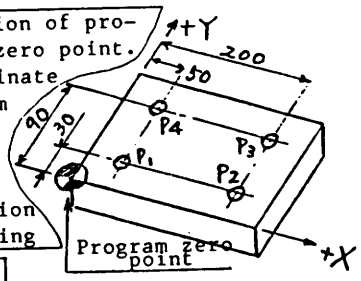
Selection of tool, Tool, cutting tool

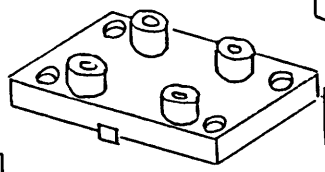
Preparation of tooling list

Tooling list					
Condition			H/A	S	F
1 Phase		T01	H01	500	300
2 Drill		T02	H02	1000	100
3 Boring		T03			

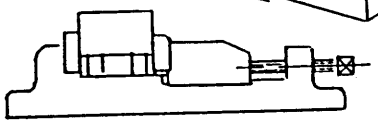
Decision of machinig order
Decision of cutting condition

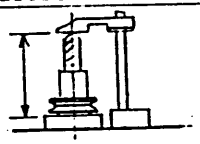
Decision of program zero point.
Coordinate system and cutter path calculation and drawing





Preparation of tool, cutting tool
Measurement of tool length, diameter

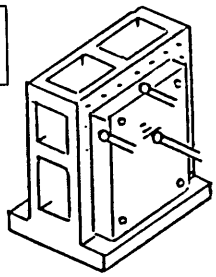




Programming

```
Ō1234
G40G80G98
G91G30Z0
G30X0Y0
N001
T01M06
G54G90G00X0
```

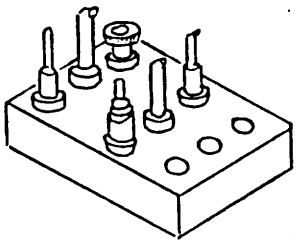
Centering for Work-coordinate setting

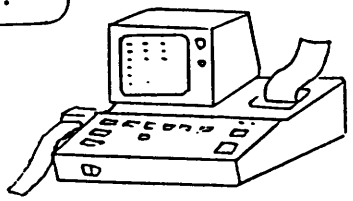


Input of work coordinate system

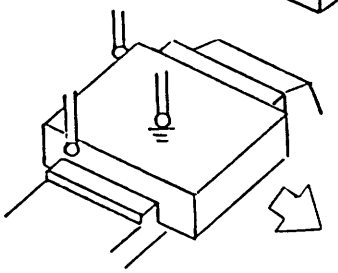
Input of tool compensation

TOOL OFFSET
01.....
02.....
03.....
04.....



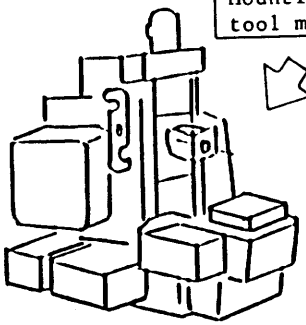


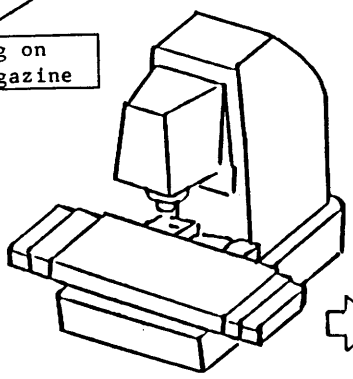
Tape punch

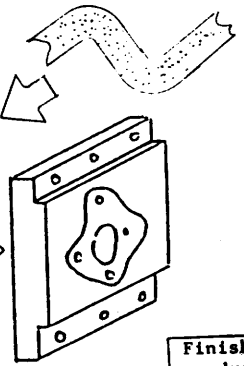


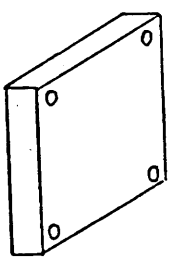
Mounting on tool magazine

Tape









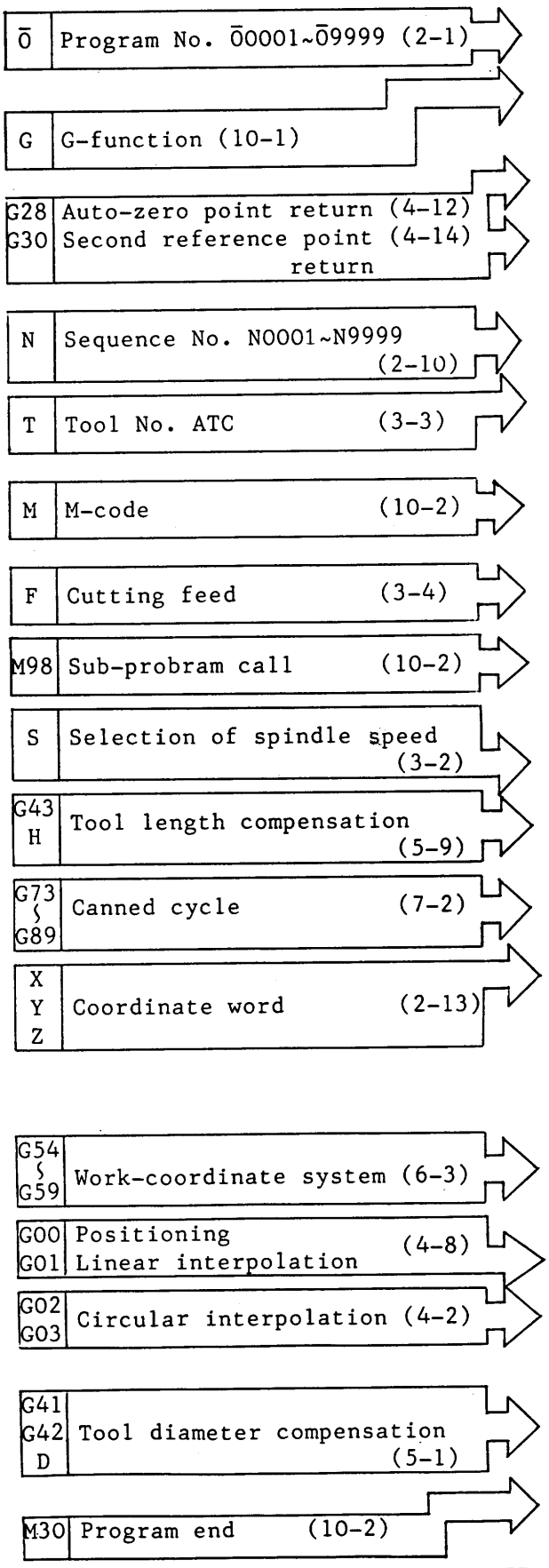
Material

Single block/ Continuous machining

Finished product

1 - 2 Programming type

"Programming" means the preparation of process sheet while looking at the diagram. Programming examples are as follows. In programming, the sequence to write, various symbols and numerals are decided.



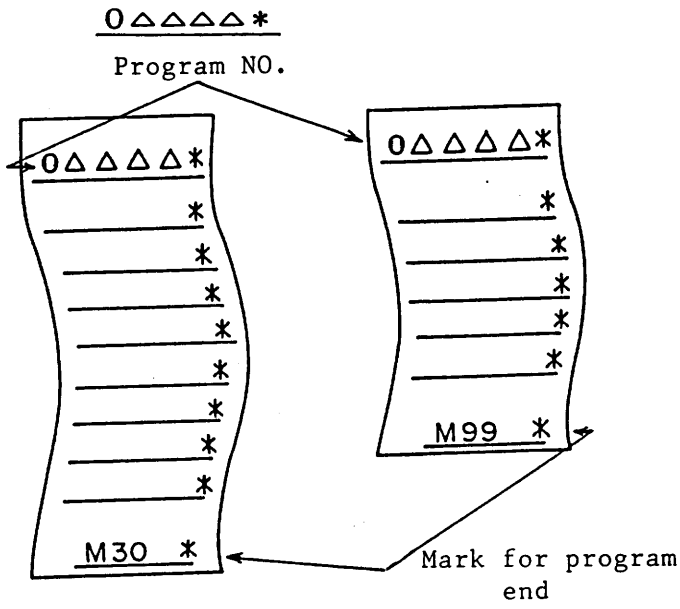

```
Ō 1986 (TEST CUT PROGRAM)

G17G40G80G98
G91G28Z0
G28X0Y0
*
G91G30X0Y0Z0M19
*
N001 (100 MM FACE)
T □ M06
G54G90G00X □ Y □ S □ T □
G43Z □ H □
M03
G01Z □ F □
X □ F □
:
M98 P98
*
N002 (10.2 MM DRILL)
G54G90G00X □ Y □ S □ T □
G43 Z □ H □
M03
G98 G81 R □ Z □ F □
X □ Y □
:
M98P98
*
N005 (20MM END MILL)
T □ M06
G54G90G00 X □ Y □ S □ T □
G43Z □ H □
M03
G00Z □
G01 Z □ F □
G02 X □ Y □ R □
G01 X □ Y □
:
G41 X □ Y □ D □
:
M98 P 98
M30
```

2. Programming language

2-1 Program No.

Be sure to attach max. 4-digit numeral in following "O" of alphabet on the head of the program.

*) The program without Program No. cannot be registered in NC-system (memory).

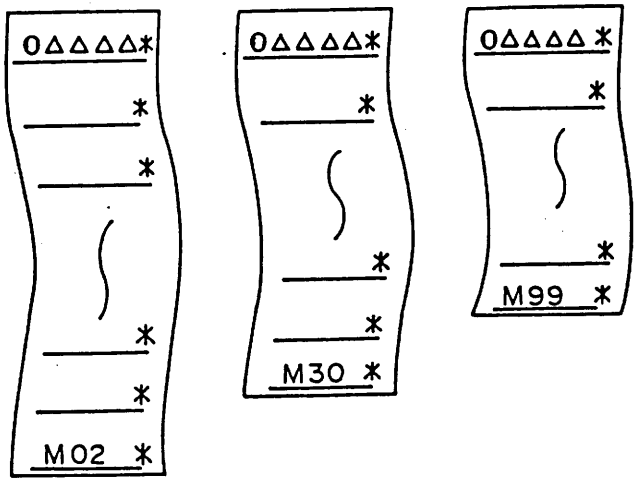


2-2 Program

One program is certainly starts with Program No., and it ends with either one of M02, M30, M99.

M02, M30 mean the end of main program.

M99 is the end of program.
There 2 types of programs, the one is (1) main program and the other is (2) sub-program.



(Note) 1. Be sure to attach Program No. at the head of one program.

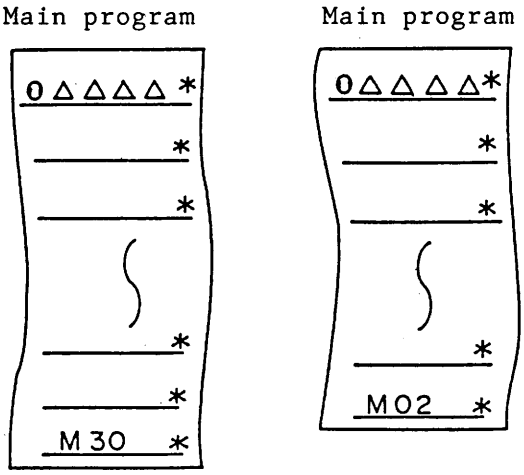
2. Program No. is also certainly necessary for the head of sub-program.

3. Use No. 0001 ~ 9999.
No. 8000 ~ 9999 may be used for special program.

2. Programming language

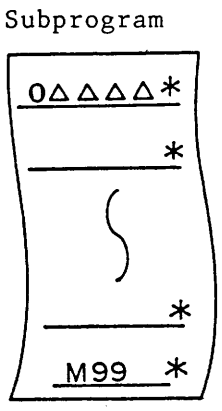
2-3 Main program

Main program means that there is Program No. on the head and M30 or M02 Program at the end.



2-4 Subprogram

Subprogram means that there is Program No. at the head, and that there is M99 Program certainly at the end.



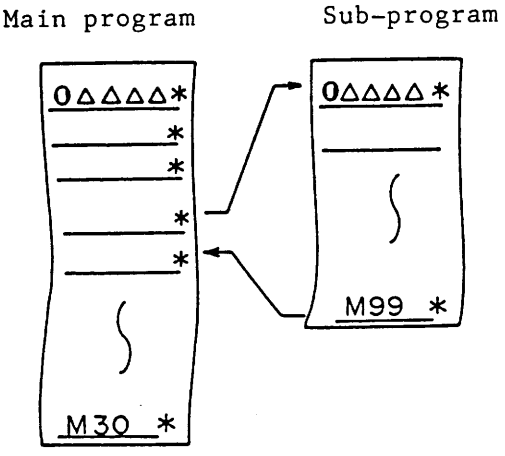
2-5 Composition of program

Program is subdivided into

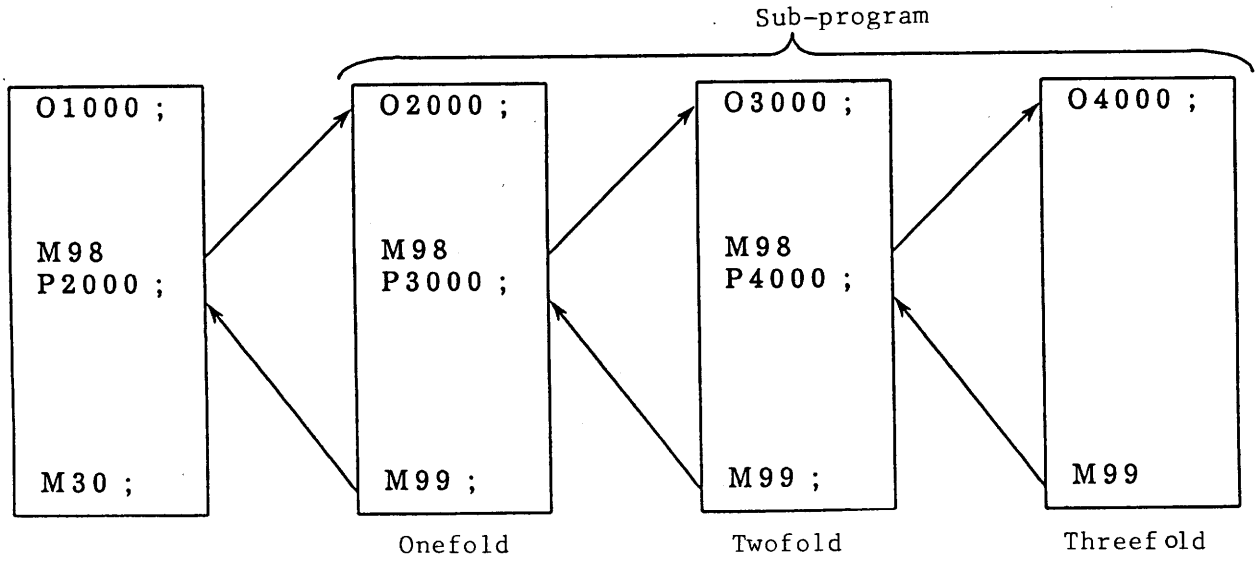
① Main program

② Sub-program.

The NC generally moves in accordance with the indication of the main programs. When a command, "Obey the indication of the subprogram" is contained in the main program, the NC moves in accordance with the indication of the subprogram therafter. This is named SUB-PROGRAM CALL. When a command, "Return to the indication of the main program" is contained in the subprogram, the NC moves in accordance with the indication of the main program. This is named RETURN FROM SUB-PROGRAM.



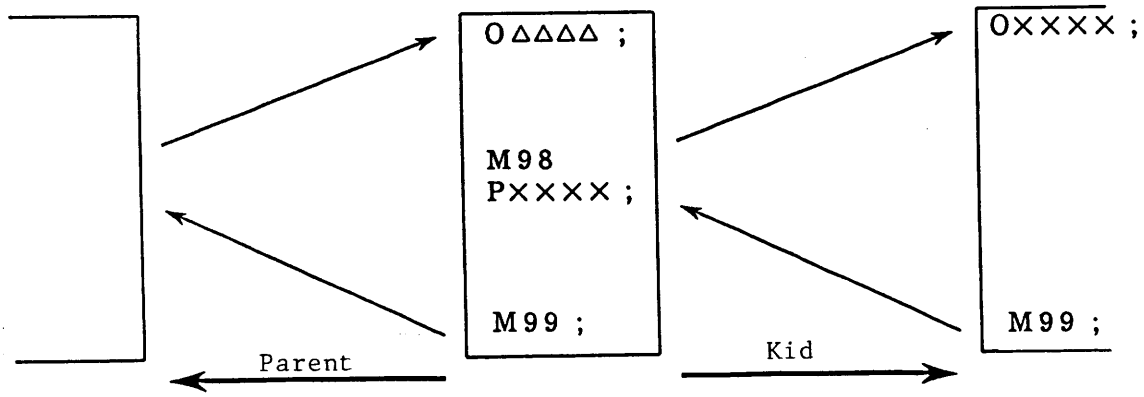
It is possible to call other sub-program further among the sub-programs.



When counting a sub-program called from the main program as onefold of sub-program call, sub-program can be called up to max. eightfold.

When some program calls other program, that program to be called is named kid program.

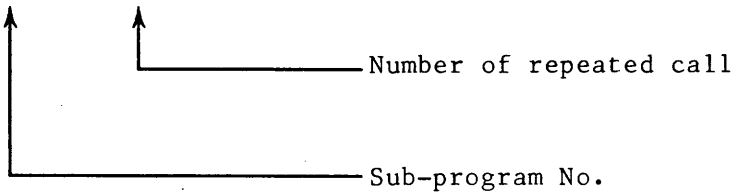
When some program is called from other program, that calling program is named parent program.



2-5-1 Sub-program call

The method calling a sub-program is as follows.

M 9 8 P ····· L ····· ;



The sub-program call of the program No. designated by P is executed by L times.

When M98 P_ L_ is commanded in the same block as the travel command, the sub-program is called after the travel is finished.

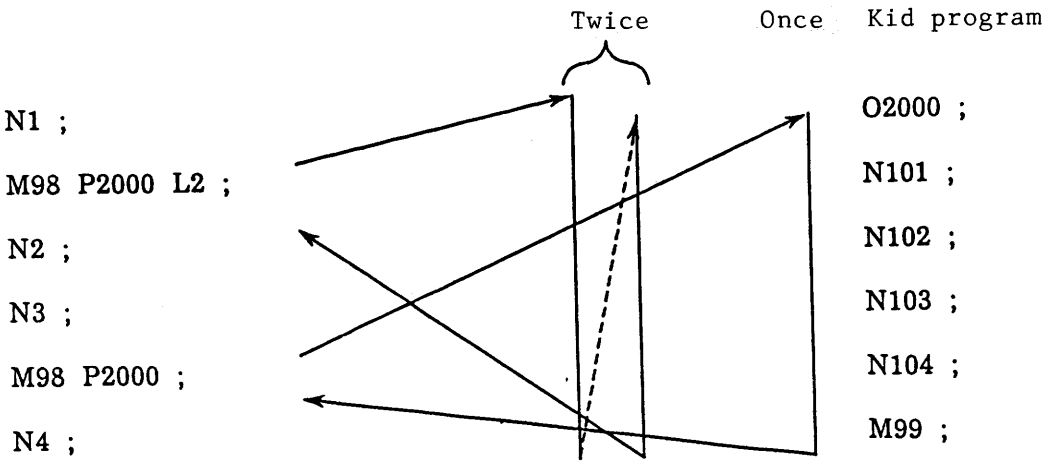
2-5-2 Return from the sub-program

The return from the sub-program is performed as follows

M99 ;

When M99 is commanded, it returns to the block next to the command of the parent program call.

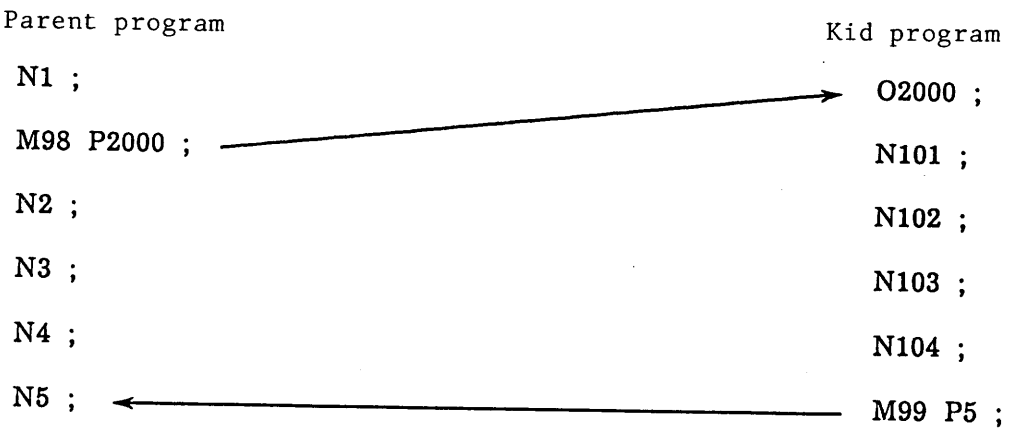
(Example) Procedures executing the sub-program



2-5-3 Special using method

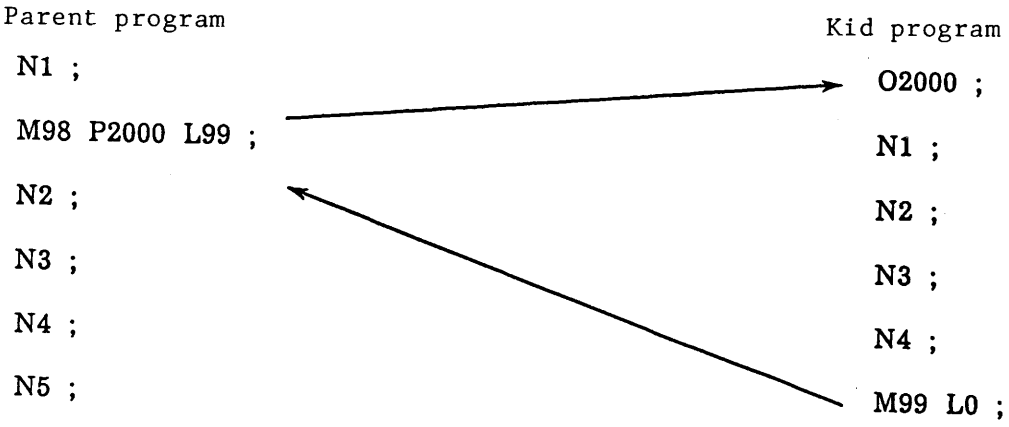
a) M99 Pα ;

When returning from the sub-program to the parent program, it returns to the sequence No. designated by P.



b) M99 Lβ ;

The L value of number of the sub-program call is shifted forcedly to the β time.

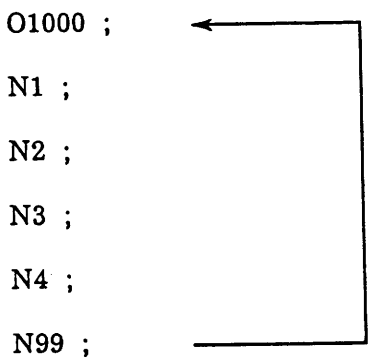


c) M99 of the main program

When M99 in the main program is executed, it returns to the top of the main program and the program is executed repeatedly from the top.

In this case, it doesn't return to the top when it becomes M99Pα, but returns to the position of the sequence No. designated by P.

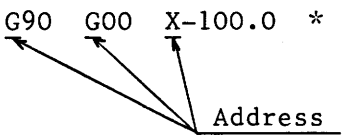
Main program



2. Programming language

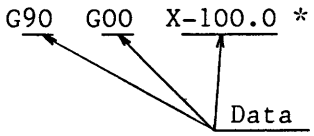
2-6 Address

"Alphabet" is particularly called "address".



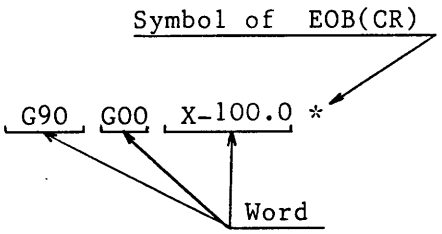
2-7 Data

Numerals (including decimals, symbols) following the address (alphabet) are called data.



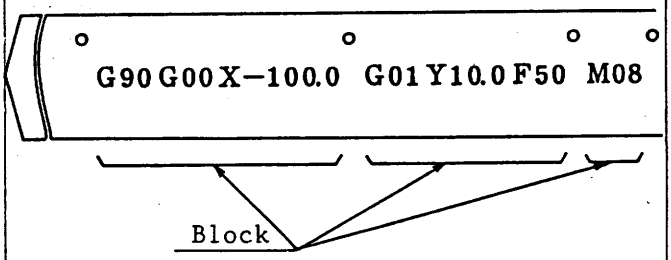
2-8 Word

Address + data is called "word".



2-9 Block

Block means one line on the program, and the interval between EOB (CR) and EOB on tape.



EOB : It means end-of block (END OF BLOCK).

CR : It means (CARRIAGE RETURN) or return of carriage in puncher word (starting of new line). In connection with this operation, punch is made at the deletion on tape.
EOB and CR are synonym.

2-10 Sequence No.

Initial part of block can be attached with number by numerals within 8 digits following the address No. It is called "Sequence No." Sequence No. is not related with machining.

N00000001 ~ N99999999

Example
```
N1   G__X__Y__S__ *
N2   Z__M__ *
     G__Z__R__F__ *
N3   X___ *
  :
N10 *
     G__M__*
     M___ *
```

2-11 How to prepare sequence No.

For setting the actual Sequence No., it is convenient to put it at a well punctuating position for the program, for instance, tool is changed to be a new tool

Also, the following way is convenient for taking the number.

In case of VM and VK

N ⌊___⌋

↑ Operations for every tool

When machined by 1st tool : N1
When machined by 2nd tool : N2
When machined by 3rd tool ; N3

In case of HG

N ⌊___⌋ ⌊___⌋

↑ Operations for every tool

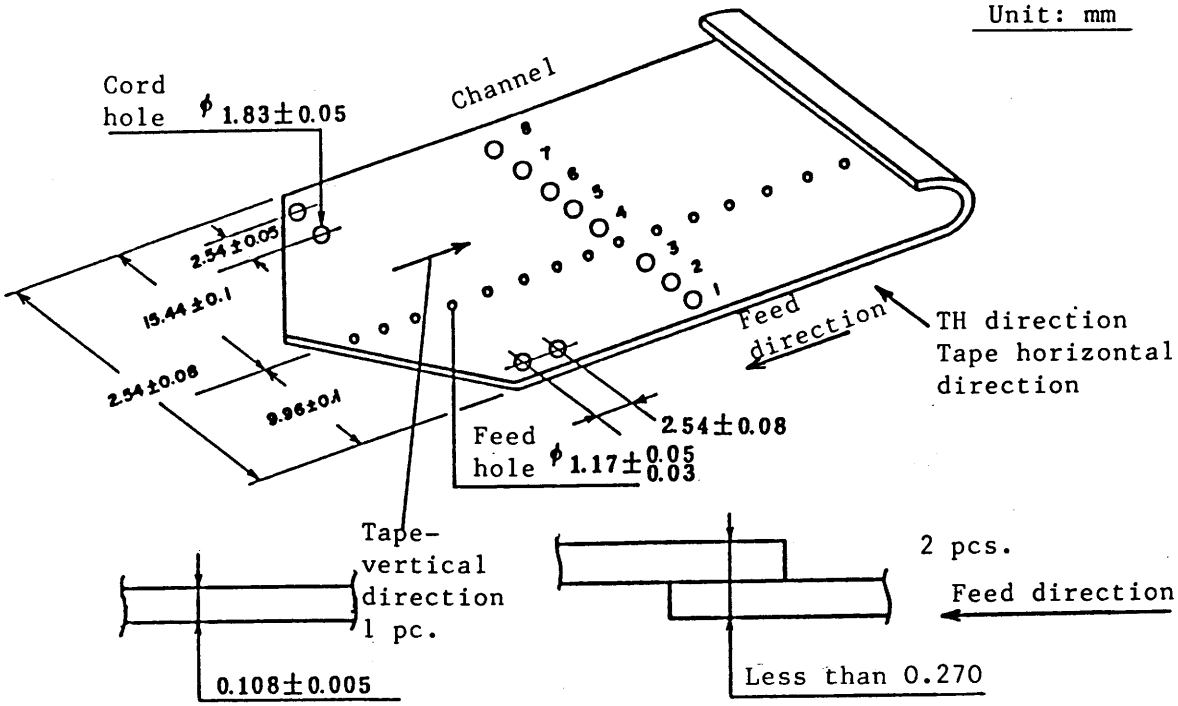
↑ Identification of pallets

When machined by 1st tool of 1st pallet, N101 is attached.
When machined by 2nd tool of 1st pallet, N102
When machined by 3rd tool of 1st pallet, N103

When machined by 1st tool of 2nd pallet, N201
When machined by 2nd tool of 2nd pallet, N202
When machined by 3rd tool of 2nd pallet, N203

2-12 Tape dimension specification

Dimension specification is based on EIARS227-A.



2-13 Tape code

Character (word) is a minimum information of command given on tape, it can be variously combined to make into readable word by the system.

Character as minimum information can be expressed by the existence of 8 holes on tape.

Character using NC-machine:

a) Numeral: 0, 1, 2, 3, 4, 5, 6, 7, 8, 9

b) English character: A, B, C, D, ... X, Y, Z

c) Special symbol: +, −, / (slash) CR(EOB), ER, SP (space) Del (delete), etc.

2-14 Tape format

Format of command tape is as follows:

N8 G3X (Y, Z) ± 5.3 I (J, K) ± 5.3 B3 F4 H2 T4 S5 M4 *
(D2)

N8	Sequence No. in 8 digits
G3	Preparatory function of 3 digits
X (Y,Z) ± 4.3	4 digits over decimal point, 3 digits less than decimal point in the positive/negative values of axial commands X, Y, Z.
I (J,K) ±4.3	4 digits over decimal point, 3 digits less than decimal point in the positive/negative values of axial commands I, J, K.
B3	B means 3 digits of positive value alone.
F4	4 digits feed function.
H2 or D2	2 digits Offset No.
T4	4 digits' tool function code.
S5	5 digits' spindle function code.
M4	4 digits' miscellaneous function code.
*	It shows end of block.

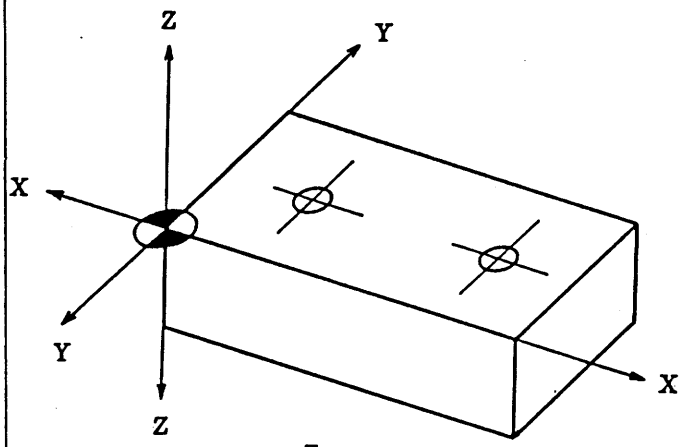
2-15 Address and meaning

Address usable with NC and its meaning are as follows:
Same address may be used for different meanings by the indication of preparatory function (G-function).
Pay attention to the fact that the indicated value range is different by the specifications of machine.

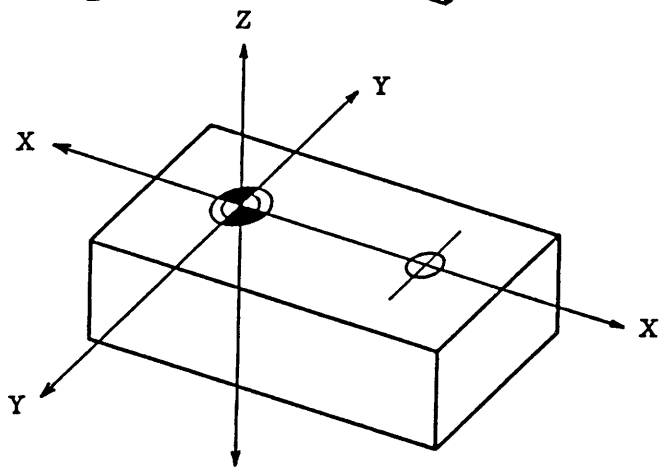
Function	Address	Meaning	Range of indication value	Remarks
Program No.	:(ISO)/O(EIA)	Program No.	1~99999999	
Sequence No.	N	Sequence No.	1~99999999	
Preparatory function	G	Indication of operation mode	0~999	See the attached list
Coordinate language	X.Y.Z	Moving command of coordinate axis	±99999.999 mm	
	A.B.C	Moving command of additional axis	±99999.999°	
	R	Circular radius R-point of canned cycle	±99999.999 mm	
	I.J.K	Circular Center coordinate	±99999.999 mm	
Feed rate	F	Indication of feed rate	1 ~ 5000mm/min	
Spindle function	S	Indication of spindle speed	Based on machine specification	S5 digits
Tool function	T	Indication of Tool No.	0~9999	T4 digits
Miscellaneous function	M	Indication of ON/OFF on machine side	0~9999	M4 digits See the attached list
Second miscellaneous function	B	Table indexing	0~359°	0-point return of B-axis B = 0
Offset No.	H.D	Indication of Offset No.	0~32	max option 208 sets
Dwell	P.X	Indication of dwell time	±99999.999 sec.	
List of Program No.	P	Indication of sub-program No.	1~99999999	
Repeating frequency	L	Repeating frequency of sub-program NO.	1~9999	Octuple calling frequency is possible
Shift amount	Q	Cut-in amount of canned cycle Shift amount	8388.607	

Note) Decimal-point input possible addresses X.Y.Z.I.J.K.R

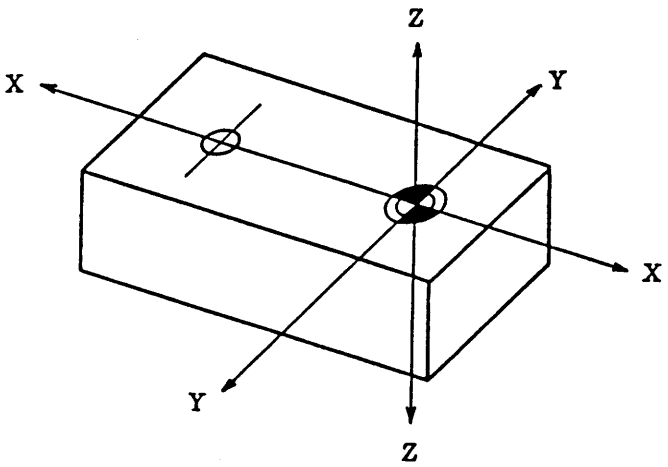
2-16 Program zero point and coordinate system

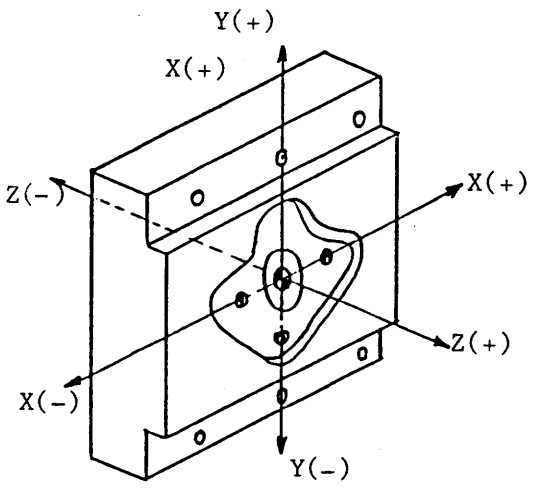


In case of program, be sure to decide the program-zero point (O-point) firstly.
Program zero point is decided by programmer by looking at the machining diagram.



Coordinate: Numeral to decide the position of diagram with the standard of 3 straight lines crossing mutually perpendicularly for firm deciding the position of optional point in the space.

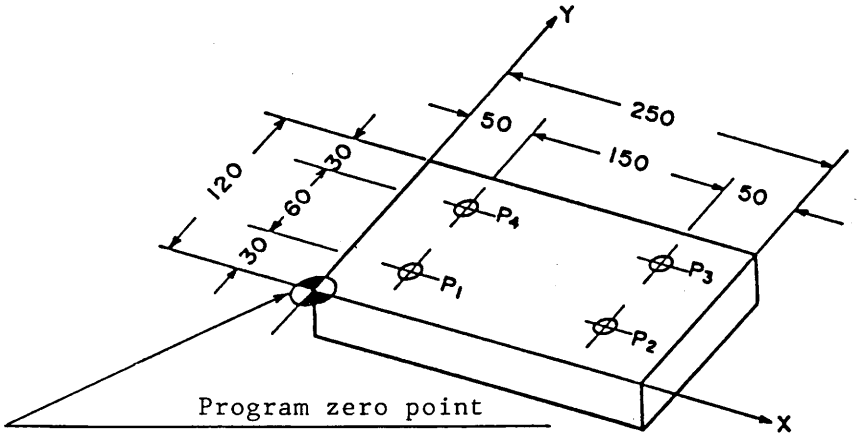


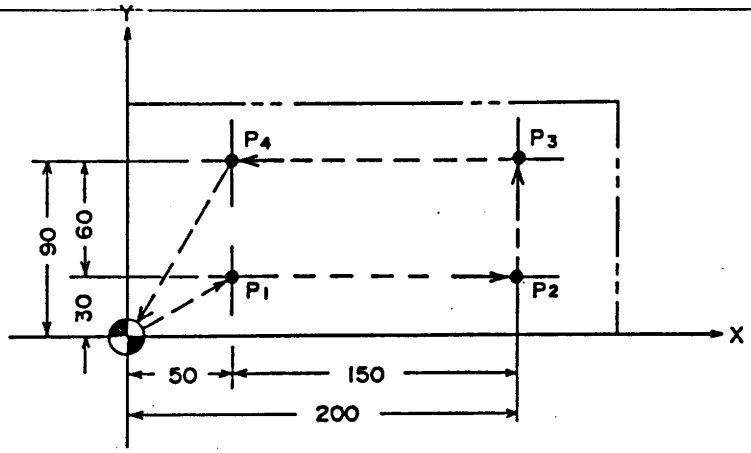


2-17 Absolute command (absolute coordinate value)

This is done by G90.

In program, there are 2 commands for axial (X,Y,Z) movement, and one of them is absolute (absolute coordinate value) command. Command is made at the position (absolute coordinate value) from program zero point . There is one zero point.



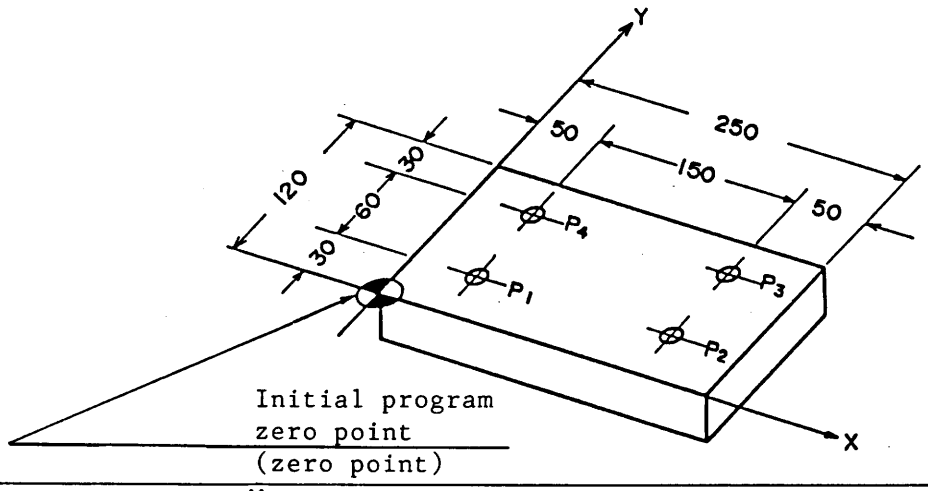


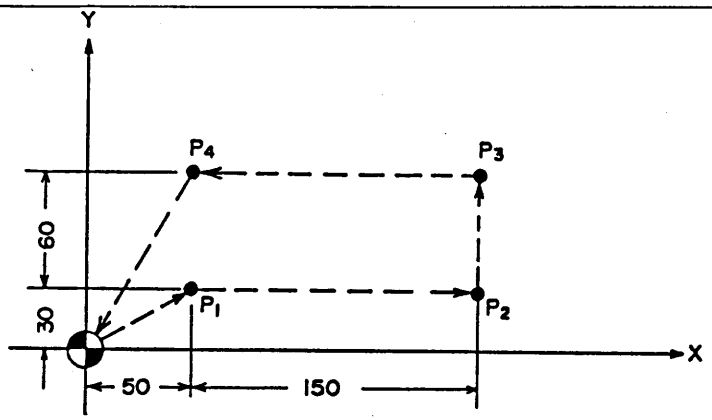
N1 G90	X50.0	Y30.0	G90 hereafter-moving command is absolute command. Moving to P1.
N2	X200.0	Y30.0	Moving from P1 to P2.
N3	X200.0	Y90.0	Moving from P2 to P3.
N4	X50.0	Y90.0	Moving from P3 to P4.
N5	XO	YO	Movement from P4 to program zero point.

2-18 Incremental command (incremental value)

It is done by G91.

In program, there are 2 commands of axial (X, Y, Z) movement.
One of them is the incremental (value) command.
Now, the place where there is the spindle is program-zero point.
Accordingly, zero point moves with the axial movement.

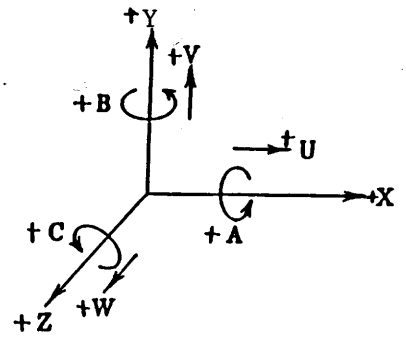




N1	G91 X50.0	Y30.0	G91 hereafter-moving command is incremental (value) command. Moving to P1.
N2	X150.0	Y0	With P1 as zero point moving to P2.
N3	X0	Y60.0	With P2 as zero point moving to P3.
N4	X-150.0	Y0	With P3 as zero point, moving to P4.
N5	X-50.0	Y-90.0	With P4 as zero point moving to the start point.

2-19 Right hand perpendicularly crossing coordinate system.

Standard axis	X, Y, Z
Swivel axis	A, B, C
Auxiliary axis	U, V, W



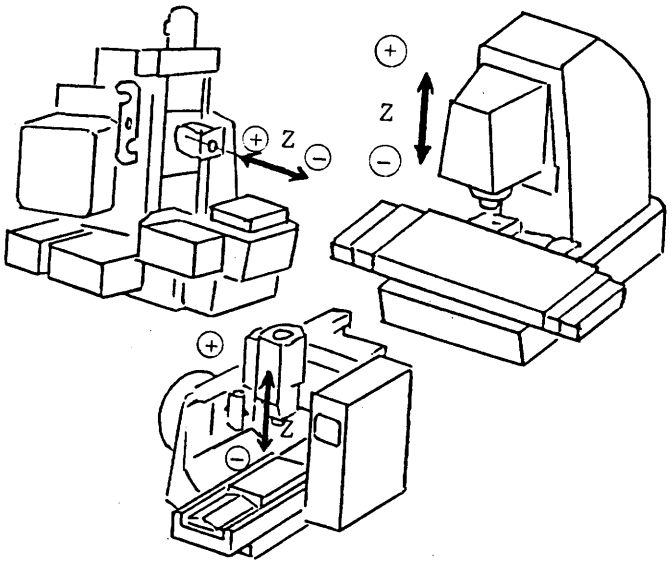
2-20 Z-axis

Vertical movement of spindle unit is expressed as Z-axis.

DIRECTION

Z-minus: movement approximating to the article.

Z-plus : movement getting away from the article.



2-21 X-axis

Lateral movement of table is expressed as X-axis.

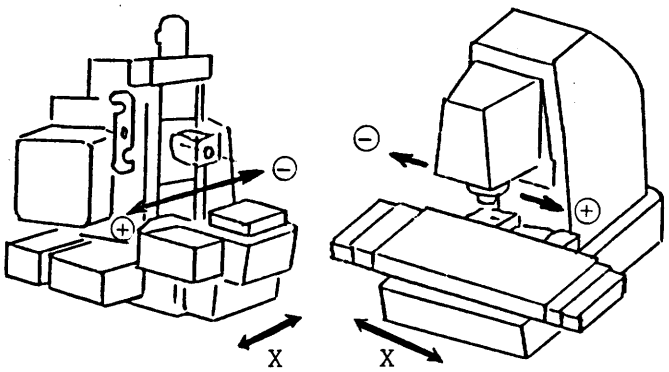
Axial movement is mainly considered with the center of spindle (tool).

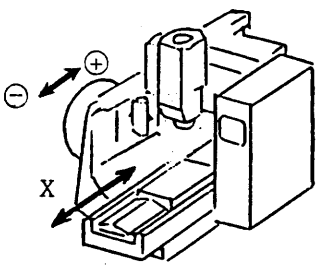
DIRECTION

X-plus : Right-ward
(table: leftward)

X-minus: Leftward
(table: rightward)

* Movement should be considered with spindle mainly, and should not be sticked to the movement of table.





2-22 Y-axis

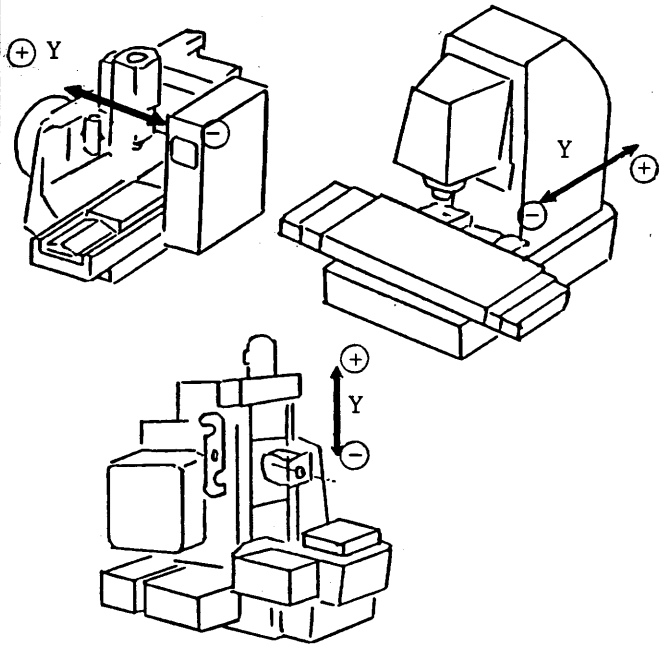
Longitudinal movement of spindle (tool), namely, column-longitudinal movement is expressed by Y-axis.

With VM, movement is made on cross rail in the longitudinal direction of spindle unit.

DIRECTION

Y-plus : direction getting away from operator side.

Y-minus : direction towards the operator (in front).



2-23 Z-axis zero point

Program is generally made by deciding the standard face at the machining diagram as zero, namely,

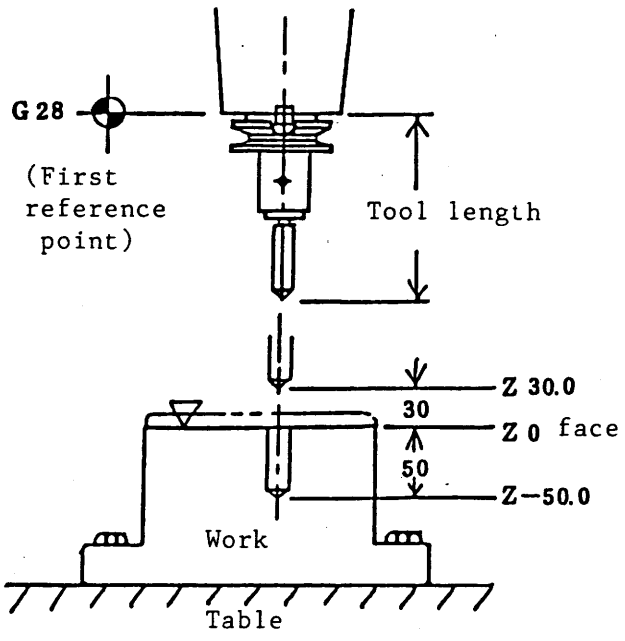
① machining surface with face-milling,

② surface regarded as standard by the description of dimension even without machining.

With ①, ② decided as Z0 (zero), program is performed.

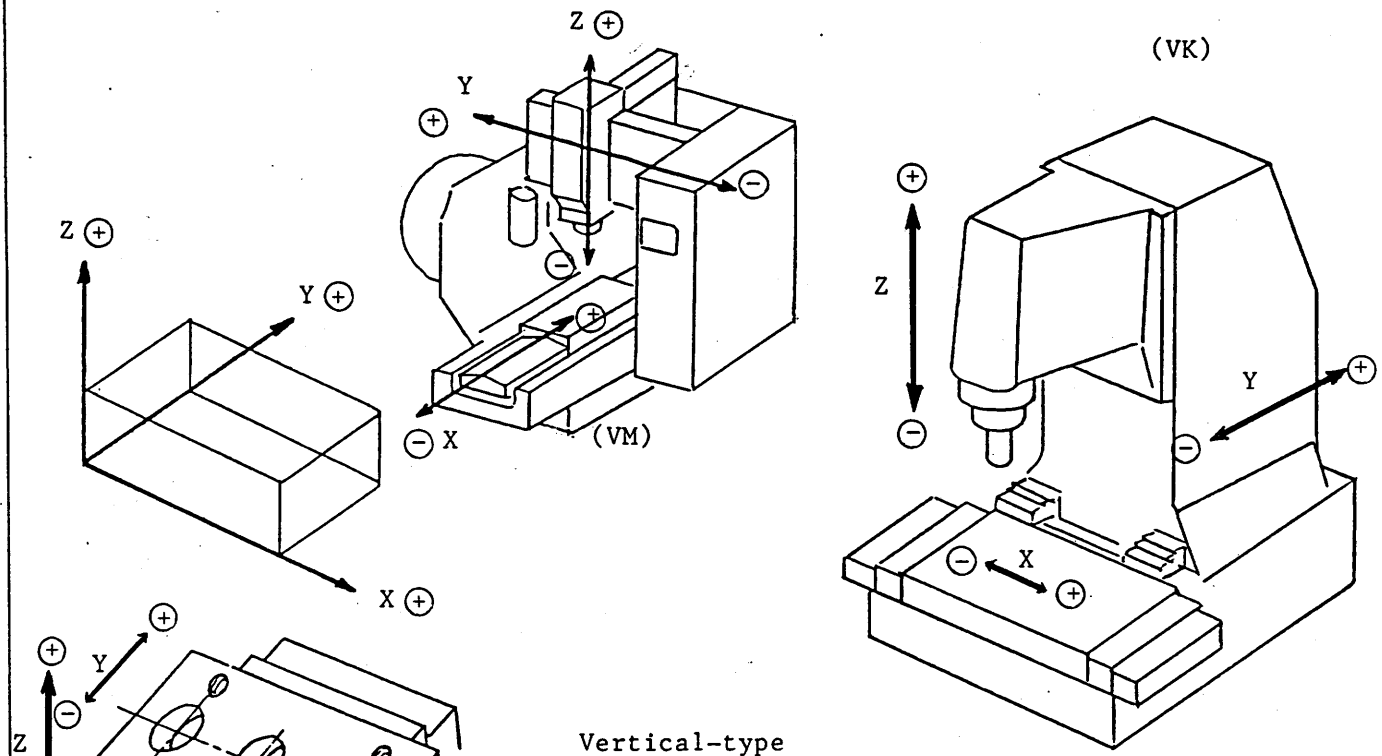
Z-value on the program is always tool-nose position.

Tool-length compensation will be mentioned later.



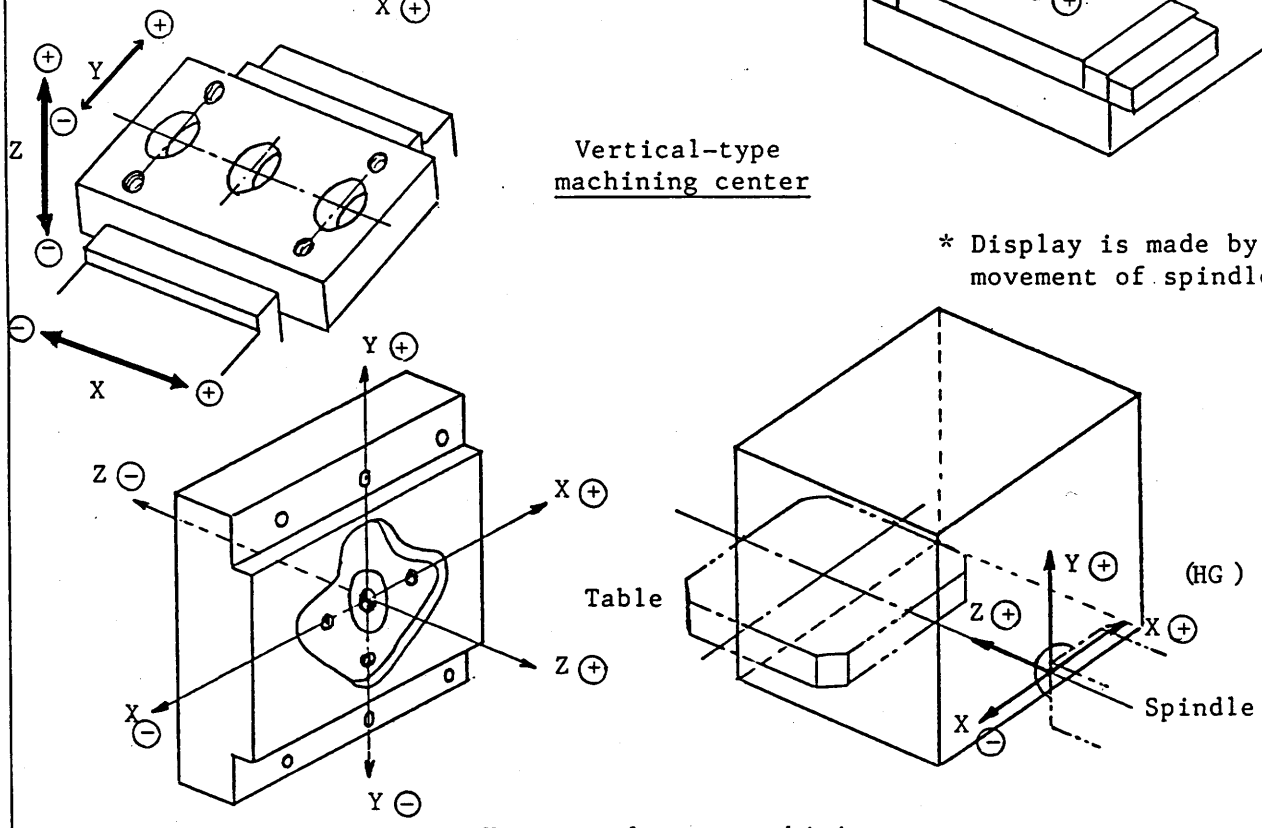
*) Instruction of Z0 (zero) to machine side is made with one touch by the use of W-setter.

2-24 X, Y, Z standard coordinate and actual work



Vertical-type machining center

* Display is made by the movement of spindle



Horizontal type machining center

* In programming on the desk, do not mind the movement of table by considering it on the diagram.

3. M, S, T, F, B functions

3-1 Miscellaneous (M-function)

At the time of operation of this machine, command is made with spindle-rotation start, stop, coolant ON, OFF, mirror image, table rotation, ON-OFF control on the side of tool-change, etc. within 4 digits (machine for use with 2 digits usually) following the address-M.

M△△△△*

M00

M01

M02

M99

Note)-1 It is impossible to make command for M-function over 2 pieces on the same block.

* Actual program

Spindle rotation ON : M03

Coolant ON : M08

Spindle rotation stop : M05

Coolant stop : M09

G90G54G00X100.0Y150.0S1000*

G43Z50.0H01*

M03*

M08*

Z0 *

G01 X300.0 F250*

M05*

M09*

For the list of M-functions, refer to P.10-7.

3-2 Commond method of pallet change (For VK with APC)

1) Home position at the pallet change

① X-axis 3rd reference point position. (G91G30P3X0)

② A pallet is mounted on the table and clamped.

③ The slider forwards.

④ The pallet has been completed to turn either clockwise or counter-clockwise.

⑤ The APC door is closed. (The notch for manual open/closed is inserted.)

⑥ The position of the Z-axis G28.

2) Program example

G91G28Z0 ———— Z-axis machine reference point

G28Y0 ———— Y-axis machine reference point

G30P3X0 ———— X-axis 3rd reference point

M60 ———— Pallet change

Command method of pallet change (M-function)

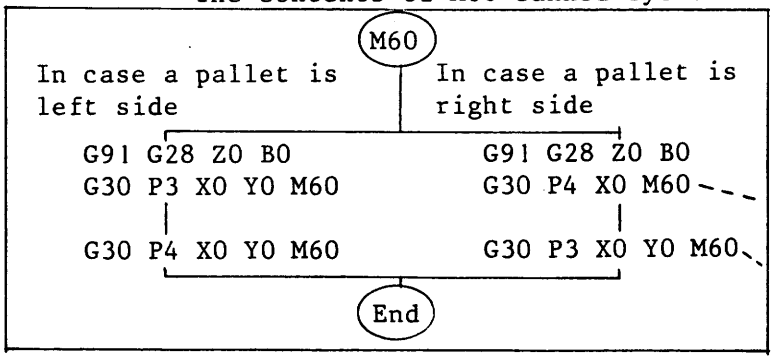
(Case of HG type)

1. There are three kinds of APC programs as follows:
   1) M60 cycle.....Changing operation is performed without distinguishing between the left pallets and the right.
   2) M61 cycle.....A pallet on machine is carried out left side and A right side pallet is carried in.
   3) M62 cycle.....A pallet on machine is carried out right side and A left side pallet is carried in.

2. Program example

   1) M60 cycle

The contents of M60 canned cycle



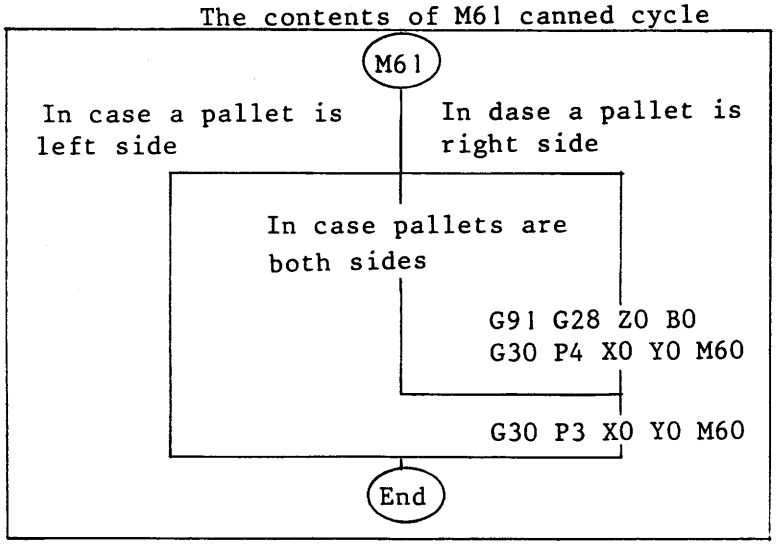

```
O 1234
G91 G28 Z0
G28 X0 Y0
M60
T01 M06
  }
```

A pallet on machine is carried out left side.

A right side pallet is carried in.

   2) M61 and M62 cycle

The contents of M61 canned cycle



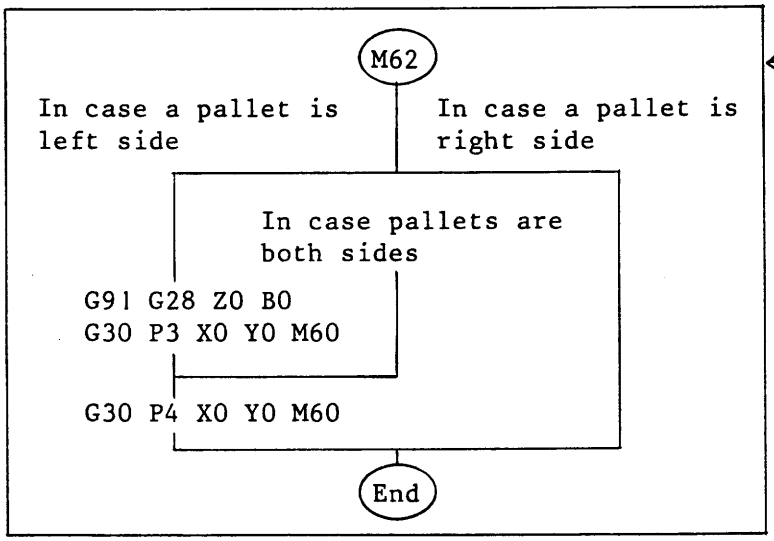

```
O 0001
G91 G28 Z0
G28 X0 Y0
M61...A right side
      pallet is carried
      in.
M98 P1234....Program
      for A work machi-
      ning.
```

3. M, S, T, F, B functions

Command method of pallet change (M-function)

(Case of HG type)

The contents of M62 canned cycle



M62....A left side pallet is carried in.
M98 P2345.....Program for B Work machining.
M30

3-3 Command method of spindle speed (S-function)

(A) Make direct command for spindle speed by 5 digits following the address S.	S △△△△△
(B) Command value	S45 (45 rpm) ~ S4500 (4500 rpm)
Note) Option	S120 ~ S12000
(C) Programming example	
Change to lower feed,500 rpm Spindle rotation	S500; M03;
Change spindle speed to 5000 rpm for rotation.	S5000
Reverse rotation Spindle stop	M04; M05;
Change the speed for rotation.	S500 M03;

Note)-1 S-command makes speed change alone, and spindle does not rotate.
Whereas, in case of S-command during rotation, change the speed for rotation.

Note)-2 With S-command except for max/min spindle speed, upper/lower max/min speed can be set.

Note)-3 By spindle rotation with auto-mode (MDI, Memory, tape), spindle is stopped by manual mode, thereafter, the auto-mode will induce "alarm".
Set to auto-mode by rotating the spindle by manual mode again.

Note)-4 Simultaneous command for axial movement with M03 or M04 will induce rotation with axial movement. Simultaneous command of axial movement with M05 will cause spindle stop after the end of axial movement.

Note)-5 With movement in high/low speed area, gear-shift operation is made automatically.

Note)-6 Changing speed with gear shift in feeding operation is not allowed for fear of tool damage.

3-4 Tool No. call (T-function)

(A) Command is made within 4 digits (2 digits generally) of numerals in following the address-T. After execution, tool is called to the stand-by position, and arm is hold. This code is effective until the next T is commanded.	T △△△△
(B) Program example △ Case in calling the Tool No. 15	T15 *
(C) Case in calling the Tool No. 20 during the positioning operation While making rapid feed to absolute coordinate values X200.0 Y150.0, select Tool No.20, and then, it is hold in arm.	G90 G00 X200.0 Y150.0 T20*
(D) Relation with tool change: At N1, call T01. At N2 with T01 spindle, make automatic change operation for tool. After the end of operation, T01 is kept at the spindle.	N1 T01 * N2 M06 *

Tool No. call (T-function)

(E) Tool No. is same as with Magazine No. Then, T.No. is T01 ~ T30 by Magazine No. (*Magazine No. is the same as with Tool No.)	Tool mounted on Magazine No. is T△△.
(F) At time of tool change, there is stand-by position for simultaneous change of the current and next tools. Case calling Tool No.15 to the stand-by position. Simultaneously with X, Y-axial movement and spindle change, Tool No. 15 is called to the stand-by position.	T15 * (Actual programming example) G54 G90 G00 X100.0 Y0 S100 T15 *
(G) X, Y, Z-axial positions at time of tool change (Second zero point)	(Case of VK) (Case of HG) { G91 G30 Z0 / G30 X0 Y0 } (Case of VMⅢ) { G91 G30 X0 Z0 / G30 Y0 (Swing to the arm-spindle side. then, arm is set to its horizontal position. }

(H) Precaution for tool change

Note 1) There is a limit on tool shape, do not use other tool than limited.

Note 2) In case commanding TxxM06 of spindle-tool again, there is nothing, and advance is made to the next.

Note 3) In case of VM, VG, VK and HG, the axes move automatically to the 2nd reference point by the command of M06 (ATC canned cycle).

3-5 Programming example of tool change (Case of VM, VK and VG)

```
                                          Ō1234 *
M31(Chip conveyor start)..................G17 G40 G80 M31  *
                                          N1  *
Keeping tool-change operation,
T01 spindle...............................T01 M06  *
T02 stand-by..............................T02  *
                                          G54 G90 G00 X0 Y0 S300  *
                                          G43 Z30.0 H01  *
                                          M03  *

                                          ∫     Machining program
                                          N2  *
                                          T02 M06  *
                                          T03  *
                                          G54 G90 G00 X100.0 Y-50.0 S1000  *
                                          G43 Z30.0 H02  *
                                          M03  *

                                          ∫     Machining program
                                          N8  *
                                          T08 M06  *
                                          T01  *
                                          G54 G90 G00 X0 Y0 S800  *
                                          G43 Z30.0 H08  *

                                          ∫     Machining program
                                          G91 G28 Z0  *
                                          G28 X0 Y0  *
                                          M30  *
```

M06(ATC canned cycle)

Case of VK, VG

```
M15 *
G91 G30 Z0  *
G30 X0 Y0 M19  *
(Txx) M06  *
```

Case of VMII

```
M15 *
G30 G91 Z0
G28 X0 M19
(Txx)
G30 X0
M06
G28 X0
‡G28 Z0
```

‡When 1 bit of the parameter No.6203 is 1, it becomes effective.

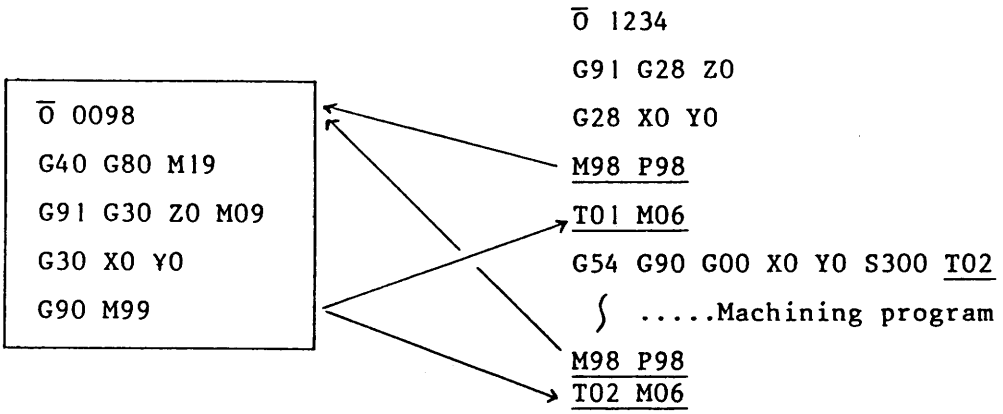
The above operations are performed by M06 command.

3-6 Programming example of tool change (Case of HG)

1. There are three kinds of ATC program of HG as follows:
   1) ATC position return is performed by main or sub-program.
   2) A method of using ATC canned cycle (available by changing the parameter)
   3) A method of performing ATC position return and arm swing opration. (effective to save the ATC time)

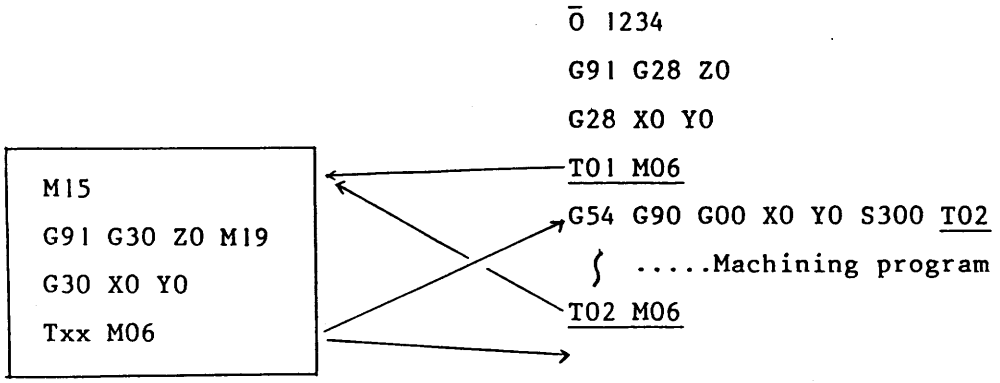
2. Program example
   1) Performing ATC position return by sub-program.

      Registered as sub-program of ATC position return.



   2) A method of using ATC canned cycle.

      Registered as ATC canned cycle in SEICOS.



Programing example of tool change (Case of HG)

3) Performing ATC position return and arm swing operation at a time.

The contests of the operation.

① Axes of X,Y,Z are returned to ATC position.

② M09, M05, M19 are performed.

③ Arm swing operation will be start when Z Axis comes to 40mm before the ATC position.

Note) -1 Operation of 3) will be done when rapid feed override is 100 %.

-2 Take care that tool noses do not interfere with fixture and so on.

```
O 1234
G91 G28 Z0
G28 X0 Y0
G91 G30 X0 Y0 Z0 T01 M06
G54 G90 G00 X0 Y0 S300 T02
(  .....Machining program
G80
G91 G30 X0Y0 Z0 T02 M06
(
```

3. M, S, T, F, B functions

3-7 Command method of feed speed (F-function)

Ⓐ Command the distance between 2 commanded points by linear or circular interpolation, also, command the moving speed by the numerals 1 ~ 5000 following 1 ~ 5000.

Command	Actual speed
F0001	1mm/min (minimum)
F1	1mm/min
F0010	10mm/min
F0100	100mm/min
F5000	5000mm/min (Maximum)

Ⓑ Actual program

Note)-1 Be sure to put F △△△△ on G01, G02, G03.

G01 X150.0 F80 *

G02 X200.0 Y300.0 R50.0 F400 *

G03 X250.0 Y50.0 R100.0 F300 *

Ⓒ With G00 (rapid feed), F △△△△ is unnecessary with G00 (rapid feed).

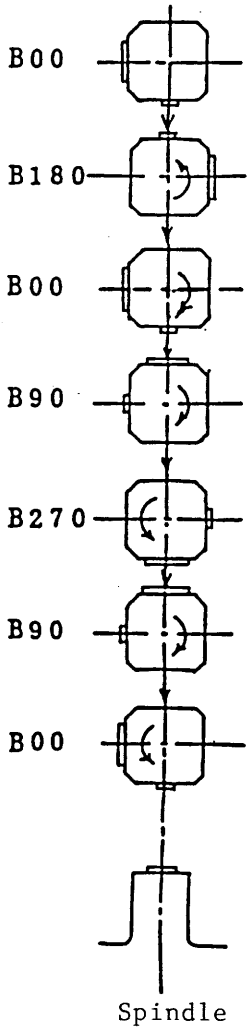
Note)-1 F-unit is the moving amount (mm) per minute, namely, mm/min.

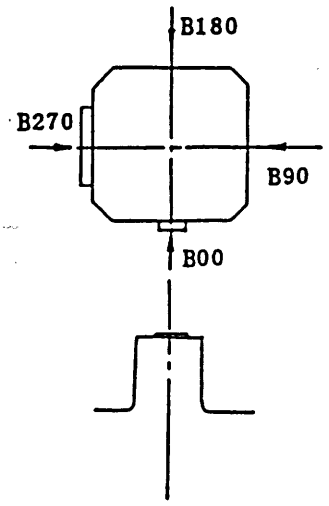
Note)-2 It is possible to omit "0" of the upper digit by numeral. It is called "reading zero".

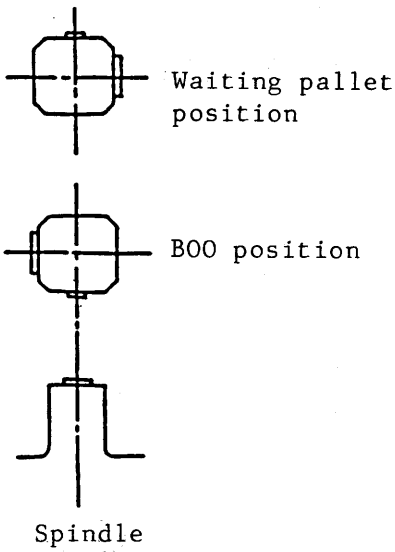
3-8 Table-indexing command method (B-function) HG series

Command the table rotation by address-B and 3-digit numeral.

By machine specification, minimum indexing angle is 1°.
With absolute command, B00~B359 (1°) is the standard.
Rotation is made short-circuit direction (the left diagram shows the case of 180°).







4. G-function (preparatory function)

4-1 G △ △ △	
It shows the meaning of program-command by the numeral of 3 digits (usually 2 digits) following Address G. That is, it is a preparatory function concerning the movement of spindle (tool).	G00 Positioning (rapid feed) G01 Linear interpolation (cutting feed) G02 Circular interpolation CW G03 Circular interpolation CCW ↓
Refer to the list of G-functions. (P. 10 - 1)	G00 ~ G99 G501 G511

Note)-1 G-code with the mark " ◥ " shows G-code state after resetting or power-input. As for G01, G17, G22, G54, G64, G90, G98, it is set after power input, and it shows the state before reset even after being reset.
As for G40, G49, G80, it is set after reset or at time inputting the power supply. For G20 and G21, it shows the previous state before pressing the reset button or before power-set off.

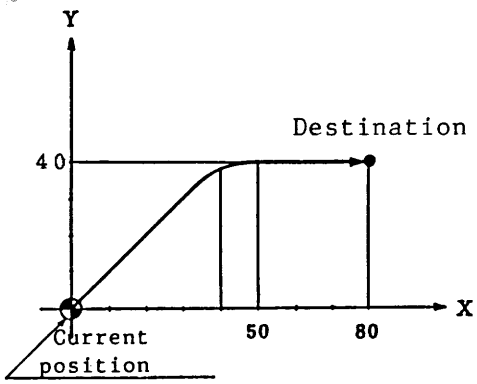
Note)-2 G-code in 00-group shows G-code without modal, and only designated block alone is effective.

Note)-3 Command of G-code without being listed on G-code list will show the alarm display.
(041 Program error)
Also, command of G-code without corresponding option will show the alarm display (PS010).

Note)-4 G-code in different group can be commanded to the same block in any quantity.
In commanding over 2G-codes belonging to the same group in the later commanded G-code is effective.

4-2 G00 (Positioning)

It is called rapid feed or rapid traverse, and rapid feed is made from the present position to the next destination (X, Y, Z).



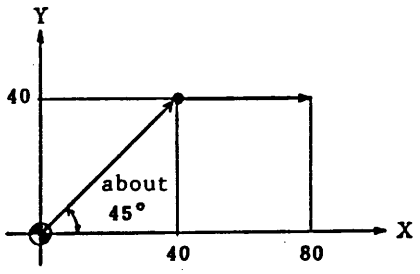
How to write the program

G00 X80.0 Y40.0 *

Note)-1 The route at moving time is not necessarily limited to the straight line.

Note)-2 Rapid traverse rate is different by the type of machine.

It is 13m/min ~ 15m/min.



Note 3) Movement may be different by the following.

Absolute command G90
Incremental command G91

Absolute command

G90 G00 X40.0 Y40.0 *
X80.0 Y40.0 *

Incremental command

G91 G00 X40.0 Y40.0 *
X40.0 Y0 *

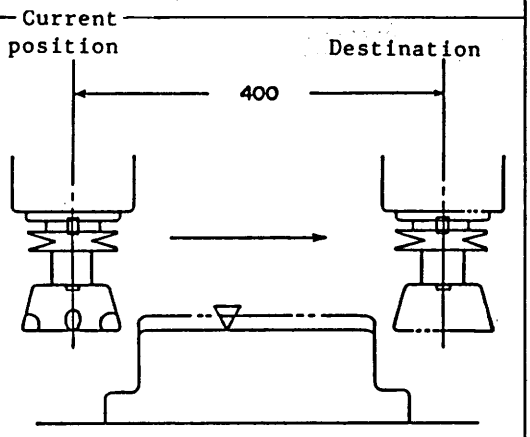
Note)-4 After commanding once, it is possible to omit by efficacy (called modal).

4. G-function (preparatory function)

4-3 G01 (Linear interpolation)

It called linear cutting or cutting feed, and linear movement is made from the current position to the next destination.
Feed rata (feed function) F is necessary.

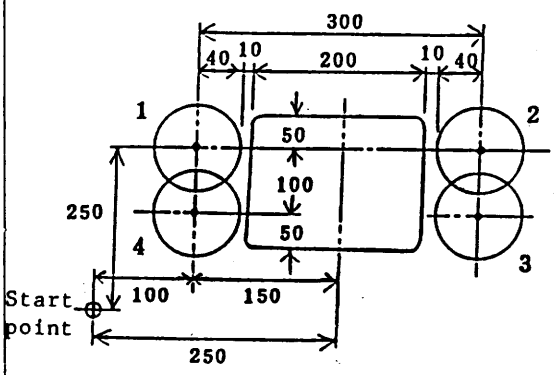
How to write program



G91 G01 X400.0 F200 *

△ F is a moving amount (mm/min) per minute

Example



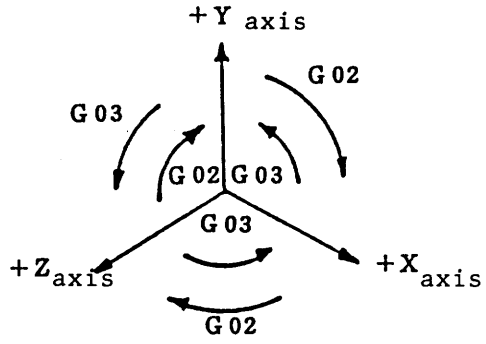
Put clear arrow mark.

Start point	→ 1	Rapid traverse rate positioning	G91 G00 X100.0 Y250.0 *
1	→ 2	Movement by cutting speed	G01 X300.0 F300 *
2	→ 3	Rapid traverse	G00Y-100.0 *
3	→ 4	Movement by cutting speed	G01X-300.0 F300 *
4	→ Starting point	Return to the start point by rapid traverse rate.	G00X-100.0Y-150.0 *

4-4 G02, G03 (circular interpolation)

Rotary direction

It is called circular cutting, and it moves to the direction in the feed rate F△△△△ along the circle (arc) towards the commanded point. Circular radius is commanded with "R".



How to write program

X___ } Coordinate value of end point (destination)
Y___ }

R___ Circular radius

F___ Feed rate per minute

G02 X__ Y__ R__ F__ *

G03 X__ Y__ R__ F__ *

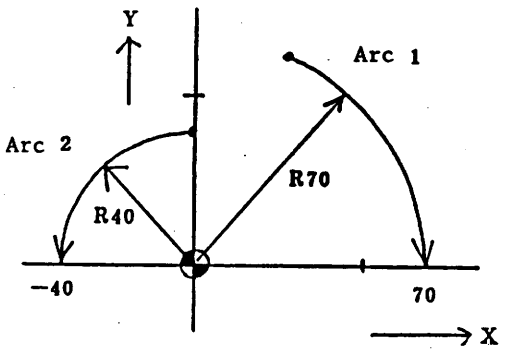
Actual program

Case of arc 1

G02 X70.0 Y0 R70.0 F150 *

Case of arc 2

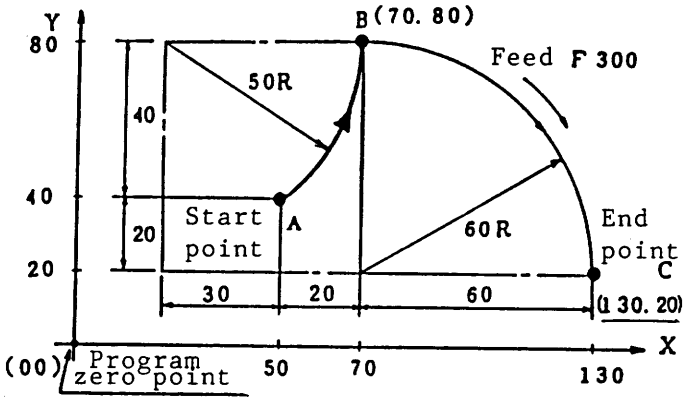
G03X-40.0 Y0 R40.0 F150 *



o With R-indication, when the arc is over 180°, command it with R-_____.
Whereas, full circle (circle of 360°) cannot be commanded with R-indication.
For full circle, command is made by the use of I, J, K. Without using R even by the ordinary circular command, it is possible to command by using I, J, K.
In the use of I, J, K, command is made with the direction and distance in view of the circular center from start point.
(I corresponds to X-direction, J to Y-direction, K to Z-direction)

4-5 G02, G03 (program example)

Program example



How to write absolute command program by radius R indication

A → B: G90 G03 X70.0 Y80.0 R50.0 F300 *

B → C: G90 G02 X130.0 Y20.0 R60.0 F300 *

How to write incremental command program by radius R indication

A → B: G91 G03 X20.0 Y40.0 R50.0 F300 *

B → C: G91 G02 X60.0 Y - 60.0 R60.0 F300 *

How to write absolute command program by the use of I, J, K

A → B: G90 G03 X70.0 Y80.0 I-30.0 J40.0 F300 *

B → C: G90 G02 X130.0 Y20.0 I0 J-60.0 F300 *

How to write incremental command program by the use of I, J, K

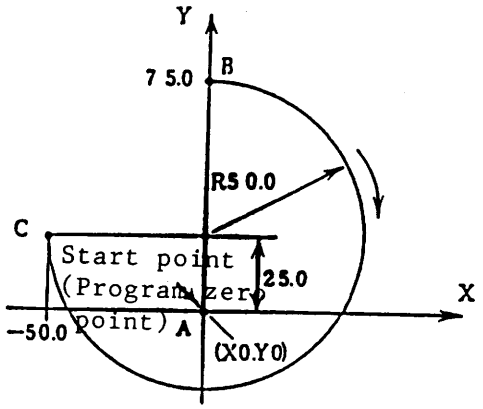
A → B: G91 G03 X20.0 Y40.0 I-30.0 J40.0 F300 *

B → C: G91 G02 X60.0 Y-60.0 I0 J-60.0 F300 *

G90, G91, F__ are modals, then, those after the second digit can be omitted.

4-6 G02, G03 (program example)

Example of circular program exceeding 180°



How to write absolute command program by radius R indication

A → B G90 G00 Y75.0 *

B → C G02 X-50.0 Y25.0 R-50.0 F 300 *

How to write absolute command program by the use of I, J, K

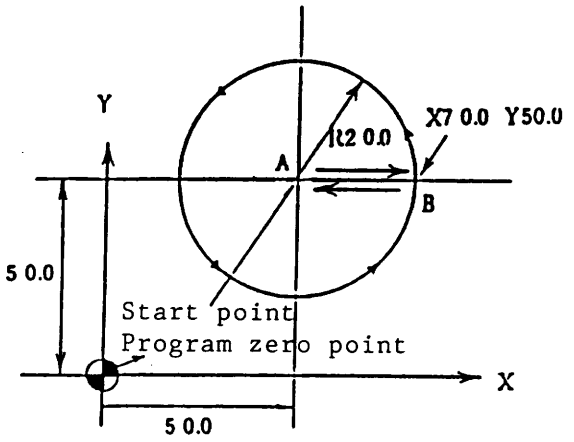
A → B G90 G00 Y75.0 *

B → C G02X-50.0 Y25.0 I0J-50.0 F300 *

4-7 G02, G03 (program example)

Full circular program example

* In case of full circle, R is not used.



With absolute command

A → B	G90 G00 X70.0 Y50.0 *
B ↻ B	G30 (X70.0)(Y50.0) I-20.0 F100 *
B → A	G00 X50.0 (Y50.0) *

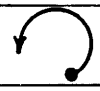
With incremental command

A → B	G91 G00 X20.0 *
B ↻ B	G03 (X0)(Y0) I-20.0 F100 *
B → A	G00 X-20.0 *

Note)-1 ( ) word can be omitted.

Note)-2 As explained on the above X-Y plane, the same can be said on Y-Z plane. Whereas, command G18 in case of X-Z plane, and command G19 in case of Y-Z plane.
Refer to G17, G18, G19

4-8 Summary on G00, G01, G02, G03

①		G00 X_____ Y_____ (Z_____) *
②		G01 X_____ Y_____ (Z_____) F_____ *
③		G02 X_____ Y_____ (Z_____) R_____ F_____ *
④		G03 X_____ Y_____ (Z_____) R_____ F_____ *

①	Case positioning with rapid traverse toward destination	G00 X100.0 Y200.0 * G00 Z50.0 *
②	Moving case with linear interpolation toward destination	G01 X100.0 Y200.0 F250.0 * G01 Z-20.0 F100 *
③	Moving case with radius R, with circular interpolation toward dostination (clockwise turn CW)	G02 X100.0 Y200.0 R75.0 F250 *
④	Same as the above ③ (Counter clockwise C.C.W)	G03 X100.0 Y200.0 R75.0 F250.0 *

(Note) G00, G01, G02, G03 are G-codes of the same group.
Once they are commanded, it is effective until the other G-code of the same group is commanded.

4-9 G04 (dwell)

It is used for command of stopping time during auto-operation.

It stops for only the indicated time.

In addition to address-P, X can be indicated.

G04 P △△△△△△△△ *

or

G04 X △△△△.△△△ *

Program example

```
G04 P2500 *2.5 sec. dwell
G04 P500 *0.5 sec. dwell
G04 X2.5 *2.5 sec. dwell
G04 P2.5 *2.5 sec. dwell
```

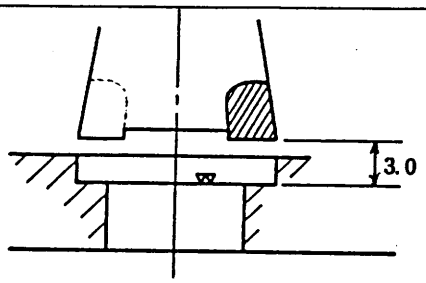
At time of address-P and X, decimal point can be input.

Max. command time

99999.999 sec.

Example of actual program

△ Case of rotation of finishing spindle at spot-facing final end with speed 60rpm:




```
G91 G01 Z-3.0 F6 *
G04 P2000 *      2 sec. dwell
G00 Z3.0 *
```

*) In 2 rotations of tool, cutting-in of Z-axis is not made, thus plane-degree of surface can be obtained.

How to obtain the dwell

Calculating formula

$$P = K.\frac{60}{N}$$

P: Dwell(Second)
K: Rotating speed for dwell
N: Rotating speed per minute

4-10 Exact stop (G09)

When G09 command is commanded in the same block as travel command, the feed is decelerated to stop when one block is finished, and after checking that the machine position is located within the range designated by a command position, the program moves to the next block.

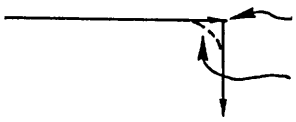
(1) Command from

G09 ... .;

(2) Program example

N1 G09 G91 G01 X100. F500;

N2 G01 Y-50.;



When G09 is commanded, an edge occurs on the corner.

When G09 is not commanded, a round occurs on the coener.

(3) Related parameter

No.1827 In-positopn width of each axis

4-11 G17, G18, G19 (plane indication)

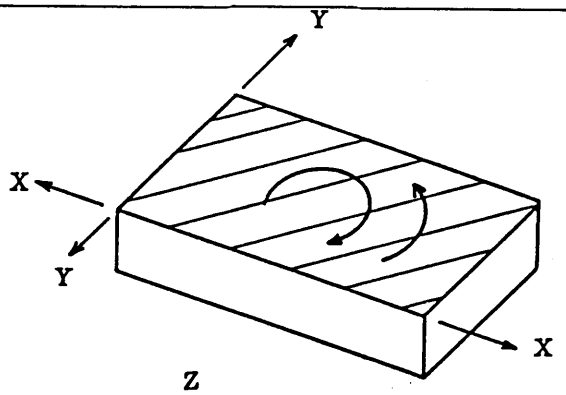
Plane indication

In performing the next ① or ② , it is necessary to make plane indication previously.

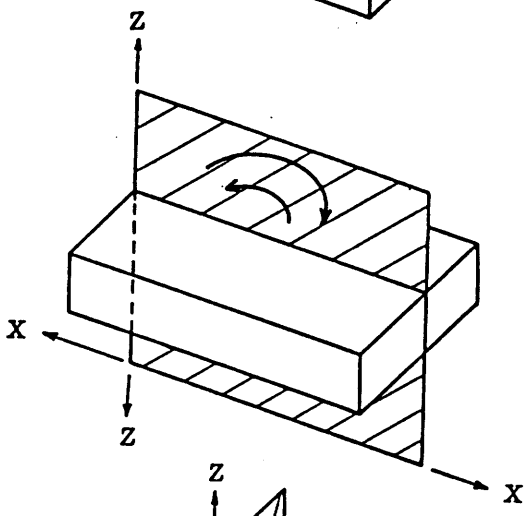
① Circular interpolation
G02, G03

② Tool diameter compensation
G41, G42

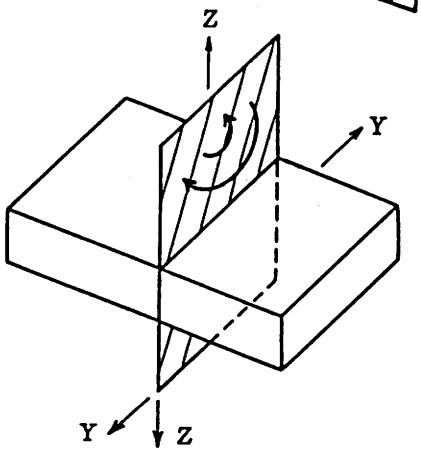
G17 (X-Y plane)



G18 (Z-X plane)



G19 (Y-Z plane)



Note)-1 G17 is selected at the time of power-input.

Note)-2 Usually, it is used with X-Y plane.

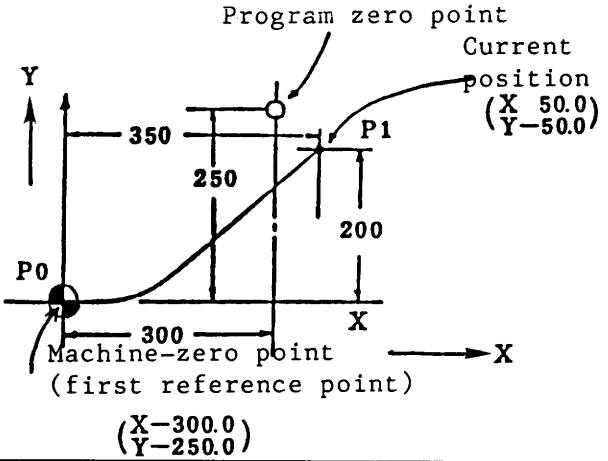
4-12 G27 (zero-point return check)

It is also called "reference-point return check".
When the end point (positioning position) is matched to the machine-zero point (first reference point), zero-point return lamp lights.

When different, no lighting is made, then, alarm (122 G27 ZRN FAULT) occurs.

G27 X___Y___Z___*

(Case of simultaneous 3 axes)



* Case of incremental command :

Distance from current position to machine-zero point

G27 X - 350.0 Y - 200.0 *

* Case of absolute command :

Coordinate value of machine-zero point

G27 X - 300.0 Y - 250.0 *

Note) G27 is hardly used.

4-13 G28 (auto-zero return)

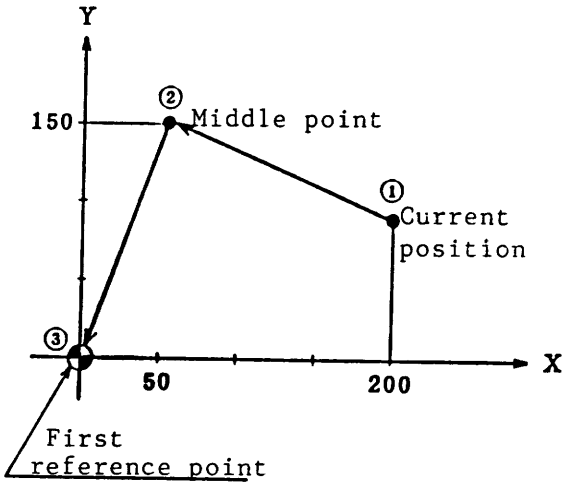
It is also called first-reference point. Zero-point return lamp lights by positioning (return) to zero point of machine-body proper.

Program command

G28 X___Y___Z___ *

Note)-1 Here X___Y___Z___ is called mid-point.

(Case of simultaneous 3 axes)



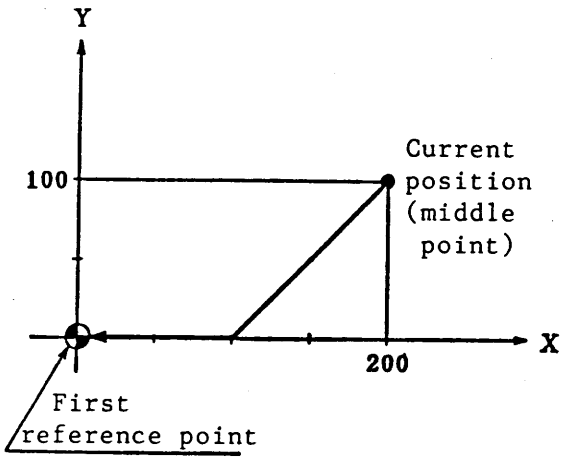
G90 G28 X50.0 Y150.0 *

Actual movement

① Current position

② Middle point — Moved with rapid traverse (X50.0 Y150.0)

③ First reference point — o Move with rapid feed; o Zero-point return lamp lights.



Note)-1 Pay full attention to the movement with G90 and G91.

Note)-2 Mid-point can be taken at any point.

Note)-3 Execution with single block stops once at mid-point. By pressing the re-start button, zero-point return is made.

Note)-4 Right diagram shows the movement of actual machine in case taking the current position and mid-point as the same position.

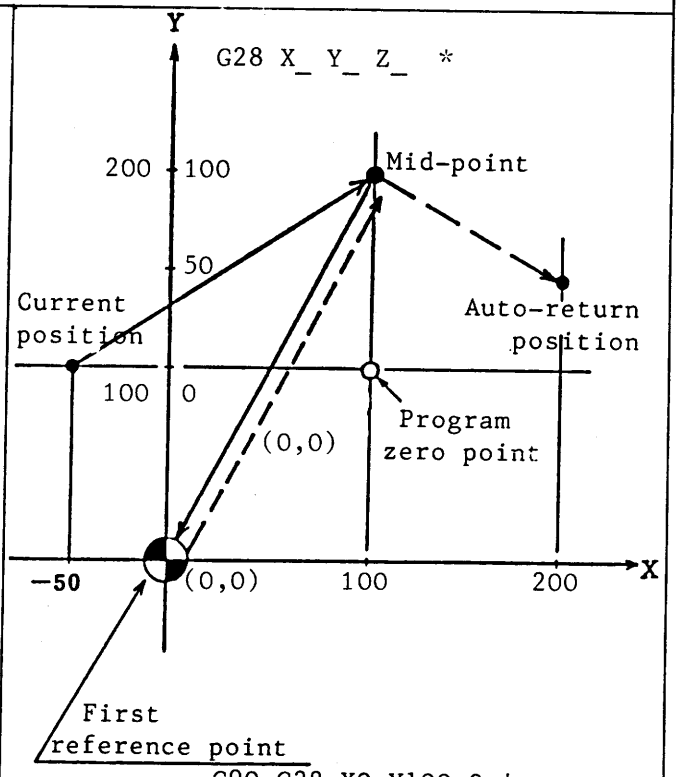
◎ G91 G28 X0 Y0 *

△ G90 G28 X200.0 Y100.0 * } Same meaning

4-14 G29 (auto-return from zero point)

It is called auto-return from reference point. Positioning can be made for the command-position (namely, X___Y___Z___ in the same block of G29) through the mid point (mid-point already command by G28).

* Command is made just after G28 in general.



Case of absolute command

G90 G28 X0 Y100.0 *

G29 X100.0 Y50.0 *

Case of incremental command

G91 G28 X150.0 Y100.0 *

G29 X200.0 Y150.0 *

Note)-1 G29 is hardly used.

4-15 G30 (2nd, 3rd, 4th reference point return)

It is also called second zero point return (3rd, 4th). Commanded axis by G30-command is positioned through the commanded point to the 2nd (3rd, 4th) reference point.

It is positioned by reference point.

```
G30 P2 X___Y___Z___*
G30 P3 X___Y___Z___*
G30 P4 X___Y___Z___*
```

Note)-1 In omitting P", 2nd zero point can be selected.

```
G30 X___Y___Z___*
```

Program example

Z-axial 2nd zero point return

```
G91 G30 Z0   *
```

↓

2nd-zero point return of X, Y 2 axes

```
G91 G30 X0 Y0 *
```

↓

Tool No. No.5 is kept at the spindle

```
T05 M06      *
M01          *
```

Note)-2 For G91, 2nd block is omitted.

Example using X-axial 3rd, 4th zero point for pallet-change position.

```
G30 P3 X0    *
G30 P4 X0    *
```

Note)-3 Previously set 2nd, 3rd and 4th reference point positions as parameters.

Note)-4 This command is used when auto-tool change (ATC) position is different from reference point.

Note)-5 Before commanding G30, reference-point return should be made by G28 or manual reference point return once at least after power input.

Note)-6 Similar to G28 command, when executing this command as a rule, cancel the tool-diameter compensation, tool-position offset and tool-length compensation.

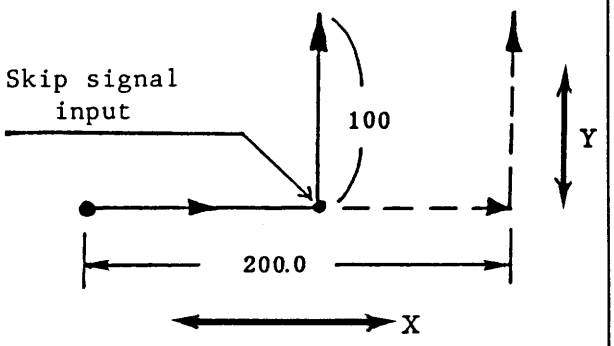
4-16 G31 (skip function)

By the input of skip signal from the outside into command of X, Y, Z following G31, this command-remaining is intercepted, and next block is executed.

Similar to G01, up to the destination, linear interpolation is made.

Actual movement ————————

Movement without input of skip signal – – – – – –



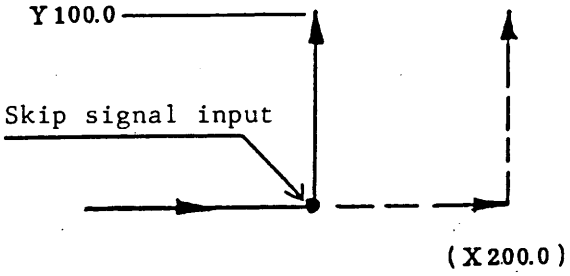
N1 G31 G91 X200.0 F150 *

N2 Y100.0 *

Movement of absolute command

Actual movement ————————

Movement without input of skip signal – – – – – – –



N1 G31 G90 X200.0 F150 *

N2 Y100.0 *

Note)-1 Generally, it is used for auto-centering and tool-length measurement.

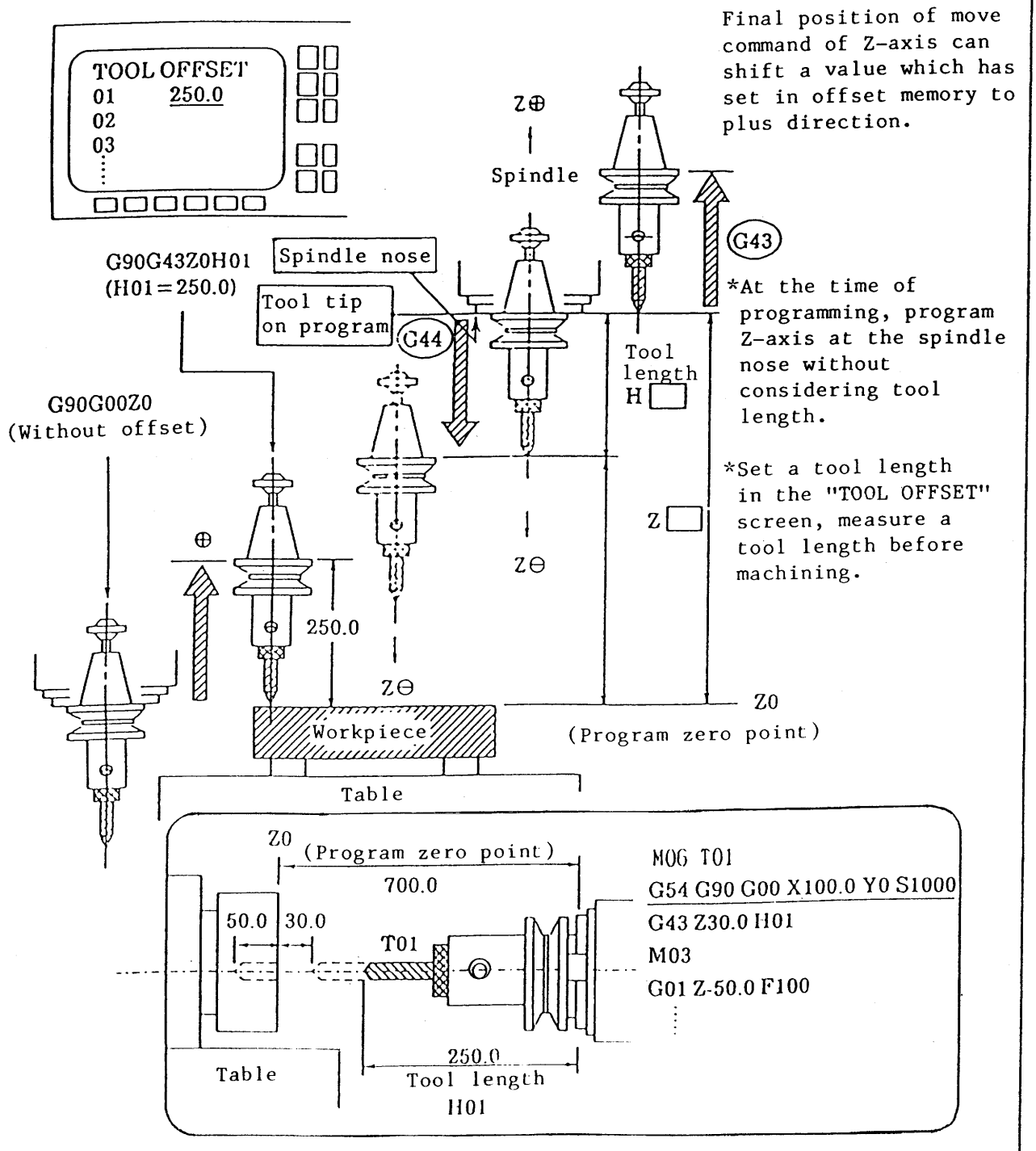
Note)-2 Under G40 state, command G31.

5. G-function (length compensation, diameter compensation, position compensation)

5-1 Philosophy of tool length compensation (G43, G44, G49)

Ⓐ Command : G90 (G91) G43 (G44) Z□ H□ *

- G43 (+ Offset)
- G44 (- Offset)
- Offset Number



Final position of move command of Z-axis can shift a value which has set in offset memory to plus direction.

*At the time of programming, program Z-axis at the spindle nose without considering tool length.

*Set a tool length in the "TOOL OFFSET" screen, measure a tool length before machining.

5. G-function (length compensation, diameter compensation, position compensation)

5-2 Tool length compensation (G43, G44, G49)

By this command, add or subtract a compensating amount, which is designated in H code, on the final position of any one axis.

A G code

G43 : Tool length offset (+) direction (Final position + compensating amount by H code)
G44 : Tool length offset (-) direction (Final position - compensating amount by H code)
G49 : Cancel tool length offset

B Form of command

$\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$ α__H__; α : any one axis

By this command, add by G43 or subtract by G44 a compensating amount, which is designated in H code, on the move command of axis on and after.

$\left\{\begin{matrix}G49\\H00\end{matrix}\right\}$;

By this command, cancel tool length offset.
When excecuting this block, there is no axis movement that an amount of cancel which is corresponding tool length compensation.
On and after the next block at the time of axis movement of absolute command execute cancelling the compensation.

Alteration of offset No. (H__)

In case of alteration of offset No. (H__) in the middle program, offset becomes tool compensation amount designated by altered offset No.

Program example

	(Final position)	(Tool length compensation)
G54 G90 G00 X0 Y0 ;		
G43 Z0 H01 ;	Z-axis : 200.	+ 200. compensation
G01 Z-30. F500 ;	Z-axis : 170.	
Z-100. ;	Z-axis : 100.	
G44 G00 Z0 H02 ;	Z-axis : -150.	- 150. compensation
G01 Z-30. F500 ;	Z-axis : -180.	
Z-100 ;	Z-axis : -250.	
G49 ;	Z-axis : -250.	Cancel
G00 Z0 ;	Z-axis : 0.	Axis movement of cancellation : -150

Note) H01 : 200.
H02 : 150.

5. G-function (length compensation, diameter compensation, position compensation)

Tool length offset can be fixed on the Z axis by parameter setting.

$\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$ Z__H__ ; or $\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$ H__ ;

By this command, add by G43 or subtract by G44 a compensating amount, which is designated in H code, on the move command of axis on and after.

C Cancel of tool length offset

$\left\{\begin{matrix}G49\\H00\end{matrix}\right\}$;

By this command, cancel the tool length offset.

When executing this block, there is no axis movement that an amount of cancel which is corresponding tool length compensation.

On and after the next block, at the time of axis movement of absolute command execute cancelling the compensation.

Program example (Example of incorrect program)

```
G54 G90 G00 X0 Y0 ;
G43 Z30.0 H01 ;
 (
G49 ; ..... Cancel of tool length offset
G00 Z3.0 ; ..... Z-axis movement with cancellation of offset
 )                amount ⇨ Collision
```

D Precautions

(a) Alarm will occur by the following command in the block of G43, G44 or H.

G04, G53, G92, G52

G28, G30

(b) In case of commanding H individually, alarm will occur if in the condition with tool length offset on any two axes.

(c) Tool length offset can be used maximum two axes one axis each. Alarm will occur if applying offset on three axes or more.

G43 Z__H__ ; Apply tool length offset on Z-axis.

G43 W__H__ ; Apply tool length offset on W-axis.

(d) In case of applying tool length offset on any one axis, alarm will occur if tool length offset can not be determined in G43 or G44 block.

(e) When altering compensating amount, it becomes effective from the block with G43, G44 or H.

(f) Reset condition can be set any one of G43, G44 or G49 by parameter setting.

(g) In case of making a reset condition as G43 or G44 by parameter setting, correct offset condition is given when commanding and performing any one of G43, G44 or H.

(h) The following notice is required, about vector of tool length offset when pushing the reset button.

(i) In case of parameter No.6000 #5=0 (Clear a vector of tool length offset by reset.), clear a vector of tool length offset when pressing reset button.
Therefore, establishment of tool length offset is required by commanding any one of G43, G44, H even if reset condition is G43 or G44.

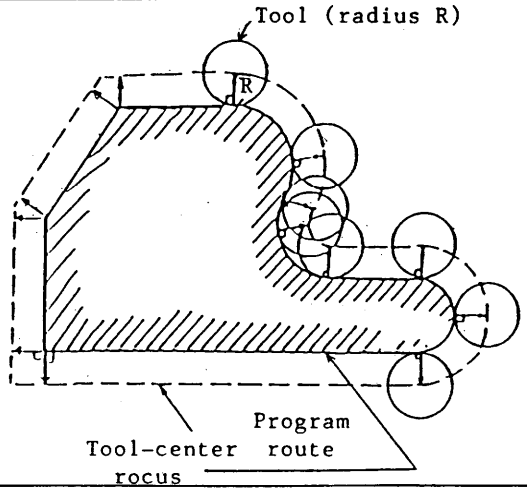
(ii) In case of parameter No.6000 #5=1 (Hold a vector of tool length offset by reset.), hold a vector of tool length offset when pressing reset button.
Therefore, a vector of tool length offset is established if reset condition is G43 or G44.

5. G-function (length compensation, diameter compensation, position compensation)

5-3 Tool diameter compensation G41, G42, G40

Ⓐ **Purpose**

Generally, in machining the surrounding inside), it is possible to obtain the intended shape by offset of tool-radius (R) alone to the outside (inside).



Ⓑ **Program pattern**

① G17 G41 G00 X_ Y_ (D△△)

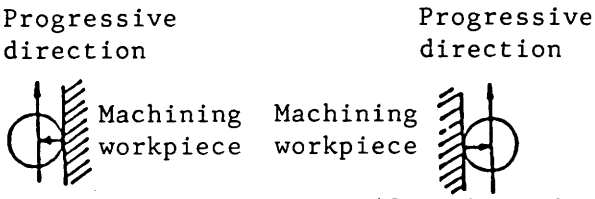
② G17 G42 G01 X_ Y_ F100 (D △△)

Note)-1 Be sure to indicate the plane (G17, G18, G19) and make D△△ compensation by G01,G00 modes.
When it is the same offset No. as the spindle tool No., D △ △ can be omitted.

Ⓒ **Offset** (to progressive direction)

Left side ⇨ G41
right side ⇨ G42

With negative (-) compensation amount, movement is made for changing G41 and G42.



Ⓓ **Offset amount (D-code)**

The tool offset can be given by not the same offset No. as the spindle tool No., but by the offset amount designated by D code through D command.

2 digits following the address-D (standard 32 pieces)

D01 ~ D32

Offset cancel G40
D00

D00 is plural offset cancel by D code.

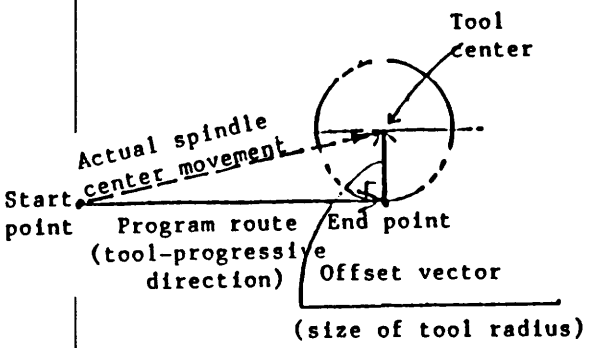
Offset input amount

0 ~ ±9999.999mm

5-4 Tool diameter compensation G41, G42, G40

Ⓐ Offset vector

This size equals to offset amount indicated by D-code at right angle against the progressive direction of tool, and it faces the tool center from the workpiece.



Note)-1 Make execution with G00, G01.
No execution is made with G02, G03.

Case of tool diameter φ30
D01 15.000 Offset amount

* Good example

o G41G01 X__Y__(D__)F__ *
o G42G01 X__Y__(D__)F__ *

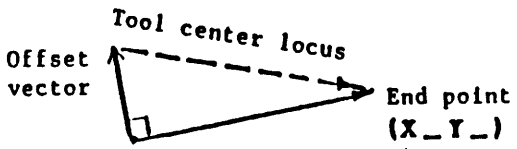
* No good example

o G41G02 X__Y__I__J__(D__)F__ *
o G42G03 X__Y__R__(D__)F__ *

Ⓑ Case of G40 offset cancel

*) Offset vector at the start point becomes zero at end point.

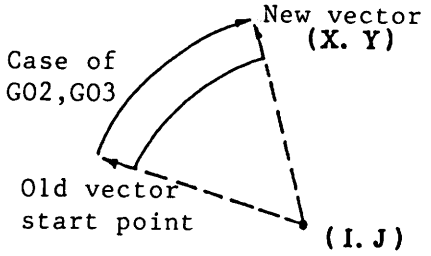
Note)-2 Make execution with G00, G01.
No execution is made with G02, G03.



G00 G40 X___Y___*

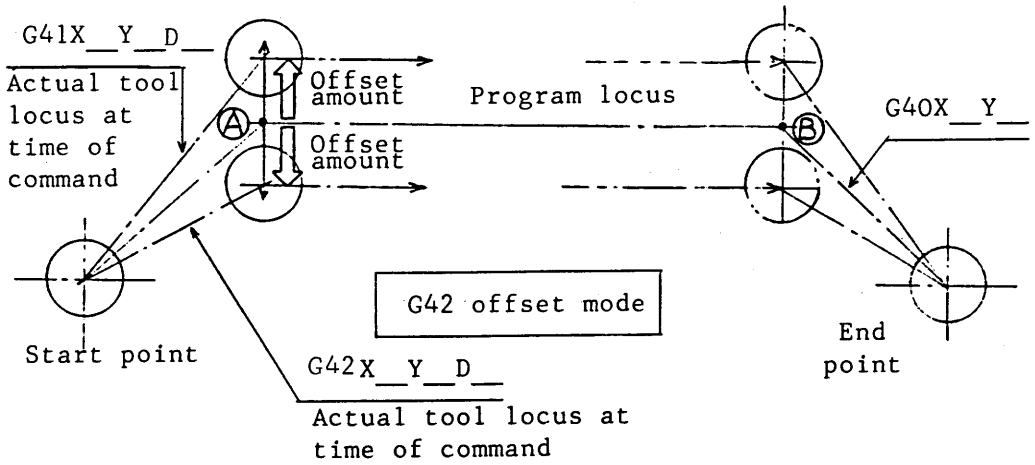
Ⓒ Case of circular compensation

* In case of G02, G03, (I, J) commands the circular center.



5-5 Summary of tool diameter compensation

Start-up block | G41 offset mode | Cancel block



Offset direction to the outside by tool-radius (diameter compensation amount) takes a right angle against the start-point Ⓐ of the next block to the start block.

With offset to the outside in the amount of tool-radius (diameter-compensation amount), machining is made, and it stops to the perpendicular direction to the end point Ⓑ of the block in front of cancel block.

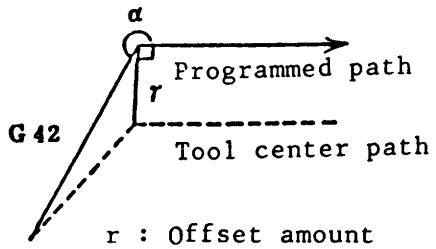
5-6 G41, G42 (start-up)

Ⓐ Start-up

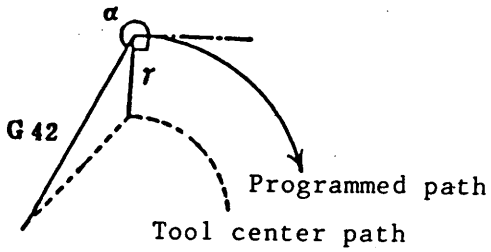
This is a movement to change from cancel mode (G40) to offset mode (G41, G42).

Ⓑ Case turning around the inside ($180^\circ \leqq \alpha$)

Linear → Linear

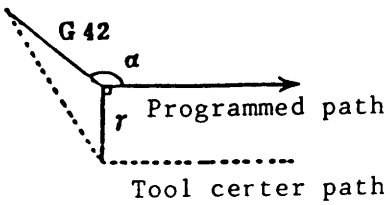


Ⓒ Linear → Circular

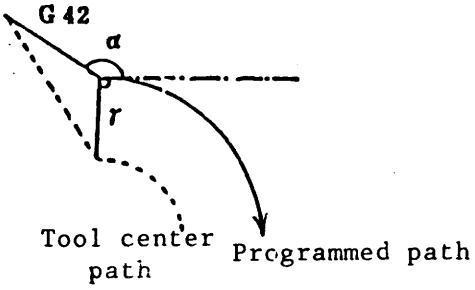


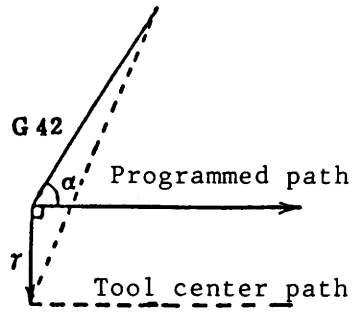
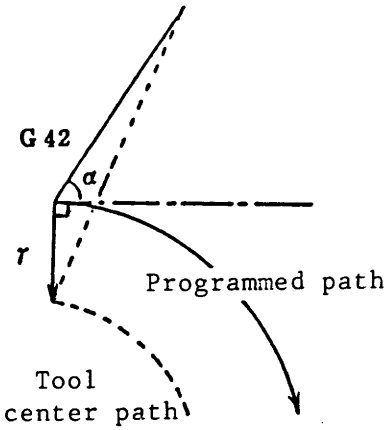
Ⓓ Case turning around the outside obtusely ($90^\circ \leqq \alpha < 180^\circ$)

Linear → Linear



Ⓔ Linear → Circular



5. G-function (length compensation, diameter compensation, position compensation)	
Start-up Ⓕ Case turning around the outside acutely ($\alpha < 90^{\circ}$) Linear → Linear	
Ⓖ Linear → Circular	

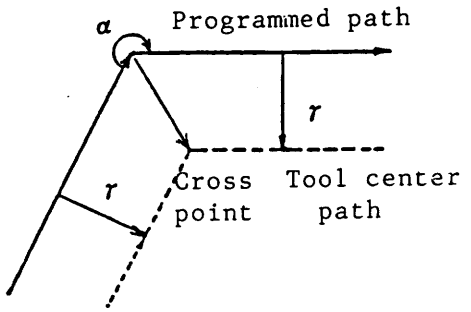
5-7 G41, G42 (offset mode)

Ⓐ Offset mode

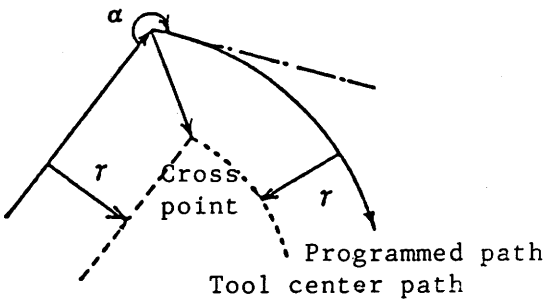
During offset mode, offset can be made not only for linear compensation and circular compensation, but also for positional command.

Case turning the inside ($180^{\circ} \leqq \alpha$)

Linear → Linear

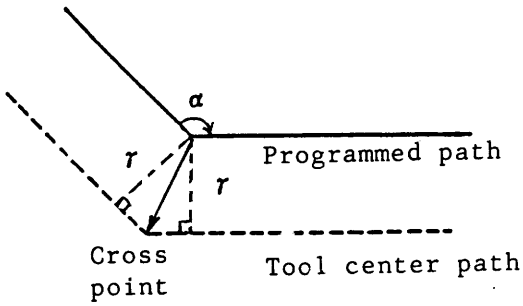


Ⓑ Linear → Circular

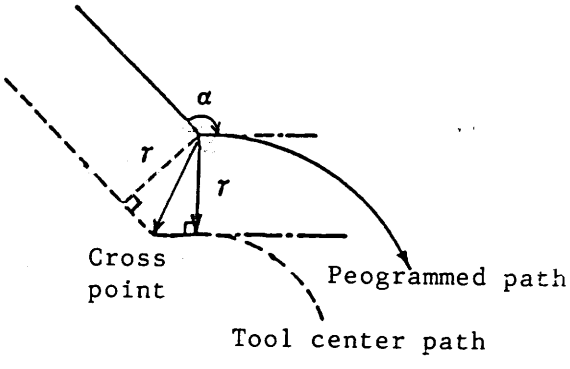


Ⓒ Case turning around the outside obtusely ($90^{\circ} \leqq \alpha < 180^{\circ}$)

Linear → Linear



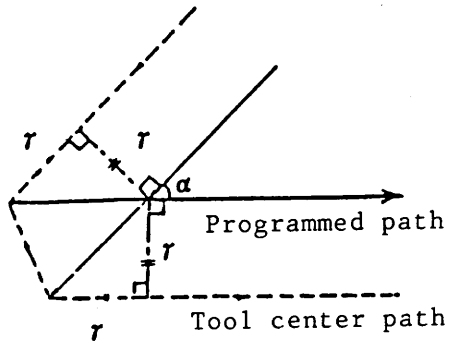
Ⓓ Linear → Circular



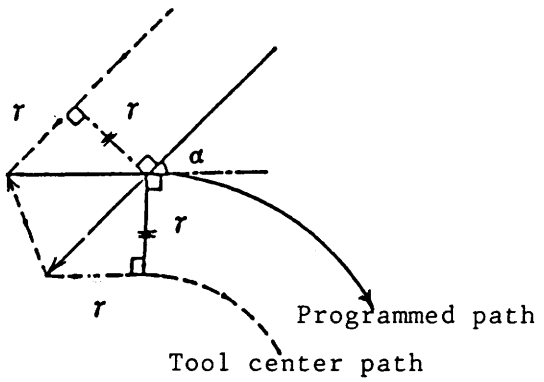
Offset mode

Ⓔ Case turning around the outside acutely ($\alpha < 90^{\circ}$)

Linear → Linear



Ⓕ Linear → Circular



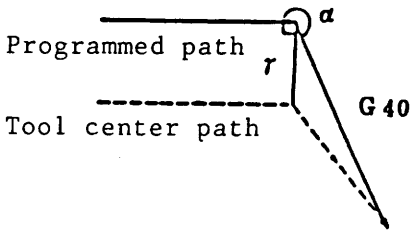
5-8 G40 (cancel)

Ⓐ **Offset cancel**

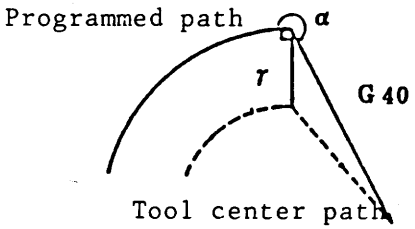
This is a movement to change from offset mode (G41, G42) to cancel mode (G40).

Case turning the inside ($180^\circ \leqq \alpha$)

Linear → Linear

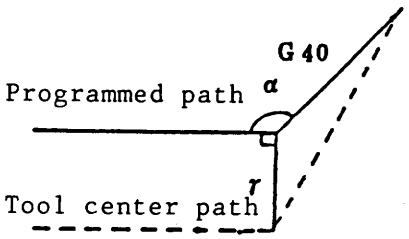


Ⓑ Circular → Linear

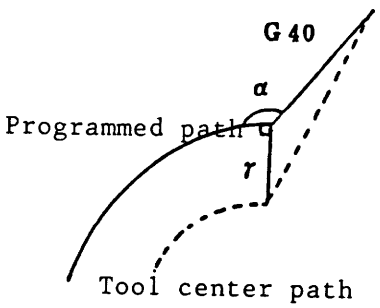


Ⓒ Case turning around the outside obtusely ($90^\circ \leqq \alpha < 180^\circ$)

Linear → Linear



Ⓓ Circular → Linear

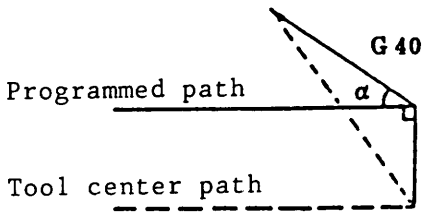


5. G-function (length compensation, diameter compensation, position compensation)

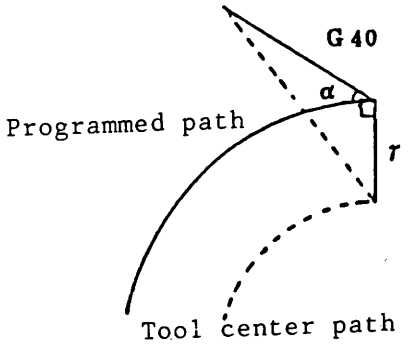
Offset cancel

Ⓔ Case turning around the outside acutely ($\alpha < 90^{\circ}$)

Linear → Linear



Ⓕ Circular → Linear



Precautions

(a) When the offset plain is changed over during tool diameter compensation mode, an alarm occurs.

(b) When no axial command is given for 3 blocks from the block next to G41 and G42 commands, the start-up will be stopped and the start-up will be done from the after axial travel command block.

(c) When a traval command is commanded to the G39 command block, an alarm occurs.

(d) When more than 3 blocks without axial travel block are commanded, during offset mode, the workpiece is left unmachined or machined excessively.

(e) When the following commands are given during offset mode, an alarm occurs.

G31,
G37,
G53,
G73, G74, G76, G81~G89
G45~G48
G302~G305
G322~G333

(f) When the following commands are given during offset mode, an interference check (excessive machining) alarm occurs.

(i) When inner circumferance of a circular arc smaller than the tool radius is machined.

(ii) A groove smaller than the tool radius is machined.

(iii) A step smaller than the tool raduis is machined.

(g) In case of start-up and its cancellation, an alarm occurs when G code in the same group other than G00 or G01 is commanded.

(h) When tool offset is given to a circular arc, a full circle may become a short circular arc or a short circular arc may moves in a full circle in a special case.
In such a case, make a program by dividing the circular arc.

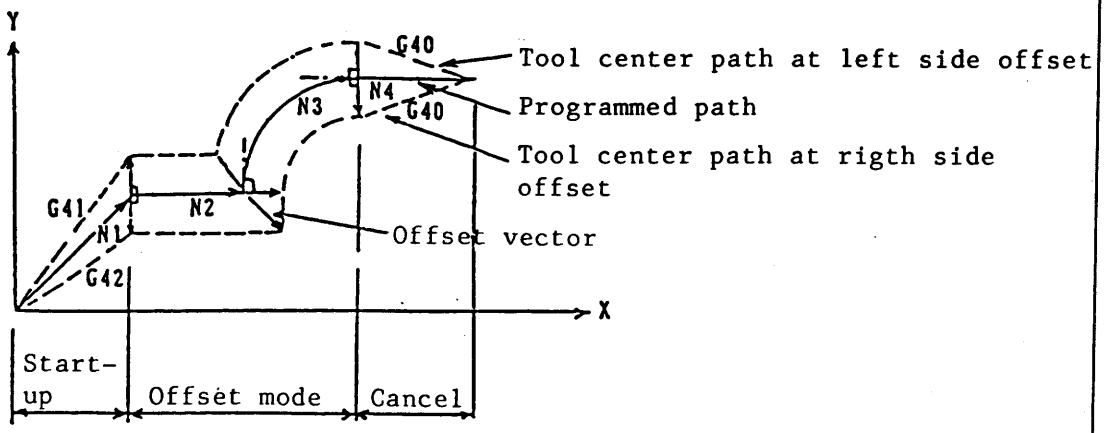
5-9 Example of tool diameter compensation program

(Left side offset) When D10 = 20.

```
G90 G00 X0 Y0.;
N1 G17 G01 G90 G41 X50. Y50. D10 F200;        Start-up
N2 X100.;                                     }
N3 G02 X150. Y100. I50.;                      } Offset mode
N4 G01 G40 X200.;                             Cancel
```

(Right side offset) When D10 = 20.

```
G90 G00 X0 Y0.;
N1 G17 G01 G90 G42 X50. Y50. D10 F200;        Start-up
N2 X100.;                                     }
N3 G02 X150. Y100. I50.;                      } Offset mode
N4 G01 G40 X200.;                             Cancel
```



Start-up : When either G41 or G42 is commanded in the state of cancellation, a tool will move to the position offset by the radius value. The offset direction is on the normal line of the start point of the next bolck.

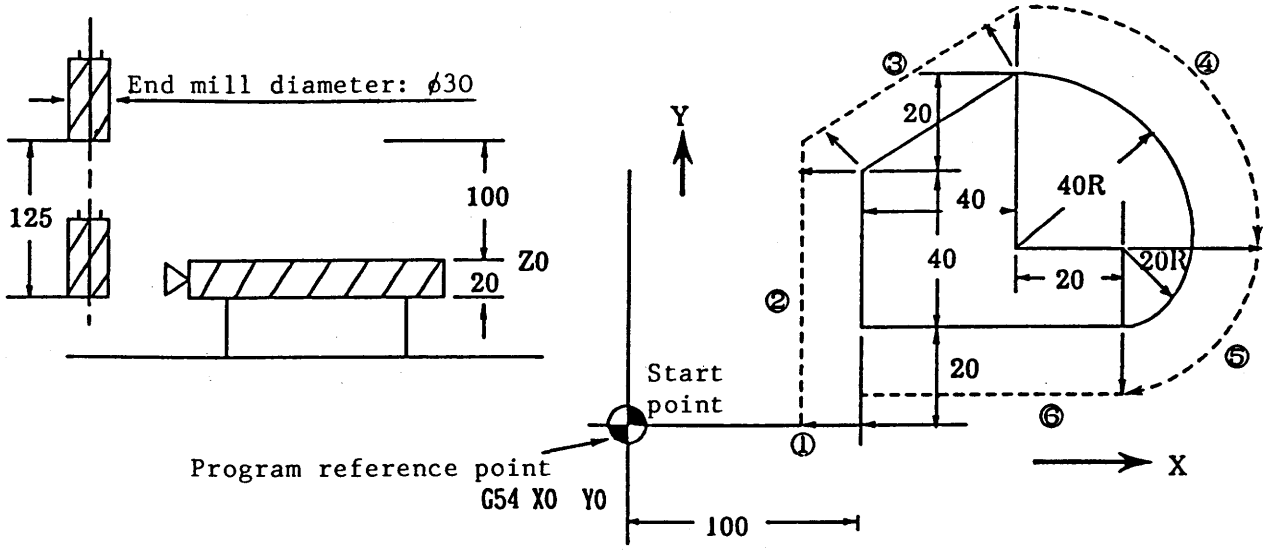
Offset mode: When the tool radius compensation mode is applied by either D41, or G42, the tool pass the path offset since then until either G40 or D00 is commanded.

Cancel : When either G40 or D00 is commanded during the offset mode, the tool redius compensation is cancelled and the tool moves to the terminal point of the program. The offset direction of the previous block becomes the terminal point of the normal line.

5-10 Example of tool diameter compensation program

Ⓐ Tool No.T01 when D21 = 15.000

The radius of end mill becomes the same dimension as that of the offset vector (arrow mark).



Ⓑ ☆Absolute program (absolute value formula)

```
T01 M06
G90 G54 G00 X0 Y0 S300
(G43) Z3.0 (H01)
M03
G90 G17 G00 G41 X100.0 (D01)
G01 Z-25.0 F2000
Y60.0 F120
X140.0 Y80.0
G02 X180.0 Y40.0 R40.0
X160.0 Y20.0 R20.0
G01 X99.0
G00 Z30.0
G40 X0 Y0
M05
M91 G28 Z0
```

← Start-up left side offset

The radius value of end mill has been set to D01.

This block is either G00 or G01.

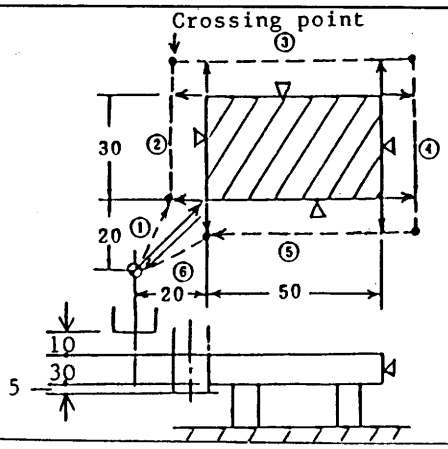
The rest is during offset mode.

← Cancel mode

5-11 Example of tool diameter compensation program

Ⓐ Oblique-line-part surrounding cutting

Under machining state, downward cutting is done.



Ⓑ ① Start-up

Vector generation from zero to D02 for its setting value amount

Cutting to Z-direction

②, ③, ④, ⑤

Corner is linear with cross-point calculation system

Z-axis escape

⑥ To the start point by offset cancel

Incremental

① G91G17G00G41(D02)X20.0Y20.0 *
G01Z-45.0F100 *

② Y30.0F200 *

③ X50.0 *

④ Y-30.0 *

⑤ X-50.0 *

G00Z45.0 *

⑥ G40X20.0Y-20.0 *
M30

Ⓒ Coordinate system setting

Positional relation is set for current spindle (blade, tool) or cutter and program zero point.

Others are same as those of the above ① ~ ⑥.

Absolute

G90G54G00X0Y0
(G43)Z10.0(H02) *

① G90G17G00G41D02X20.0Y20.0*
G01Z-35.0F100 *

② Y50.0F200 *

③ X70.0 *

④ Y20.0 *

⑤ X20.0 *

G00Z10.0 *

⑥ G40X0Y0 *
M30

5-12 Example of tool diameter compensation program

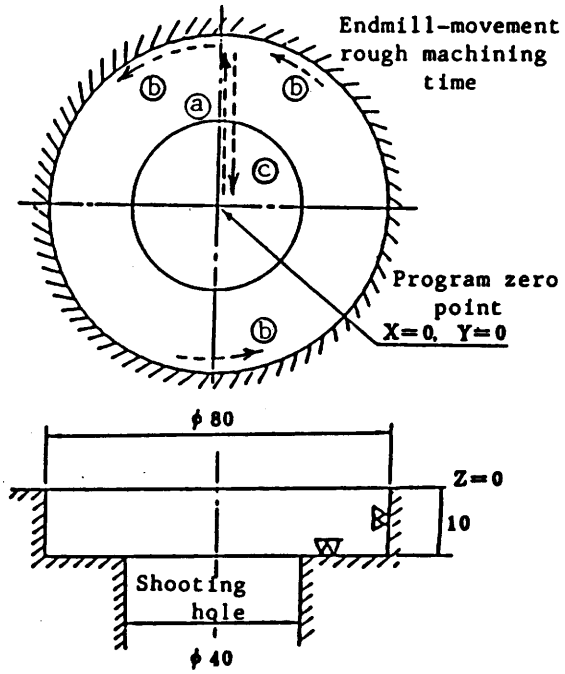
Ⓐ Example of circle-cutting in the use of tool-diameter compensation (G41, G42)

Machining (ϕ80 x depth as 10) shown in right diagram is attempted.

- o Using cutter: ϕ30.0 2-blade endmill
- o Machining method : Rough machining Finishing can be done by one endmill.
- o Movement of endmill

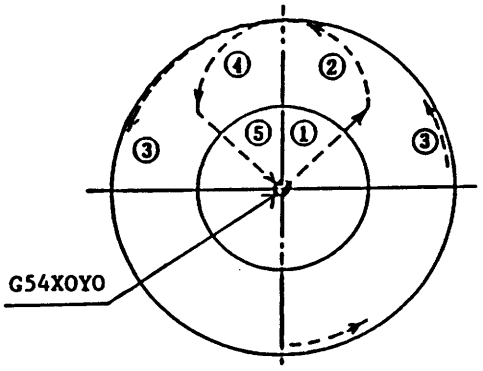
Rough machining

ⓐ → ⓑ → ⓒ



Ⓑ Program at rough machining time

```
T2 M06
G54 G90 G00 X0 Y0 S250
G43 Z50.0 H02
M03
Z-9.8
G01 G41 Y40.0 D21 F60
G03 (X0 Y40.0) J-40.0
G01 G40 Y0 F1000
G00 Z-10.0
```



Note)-1 If 15.2 is set to the dia diameter compensation of D21 and 15.000 is set to D31, the finishing allowance is 0.2mm on one side.

Note)-2 Using G302, G303, similar program can be executed by one block.

Refer to (7-14) G302, G303.

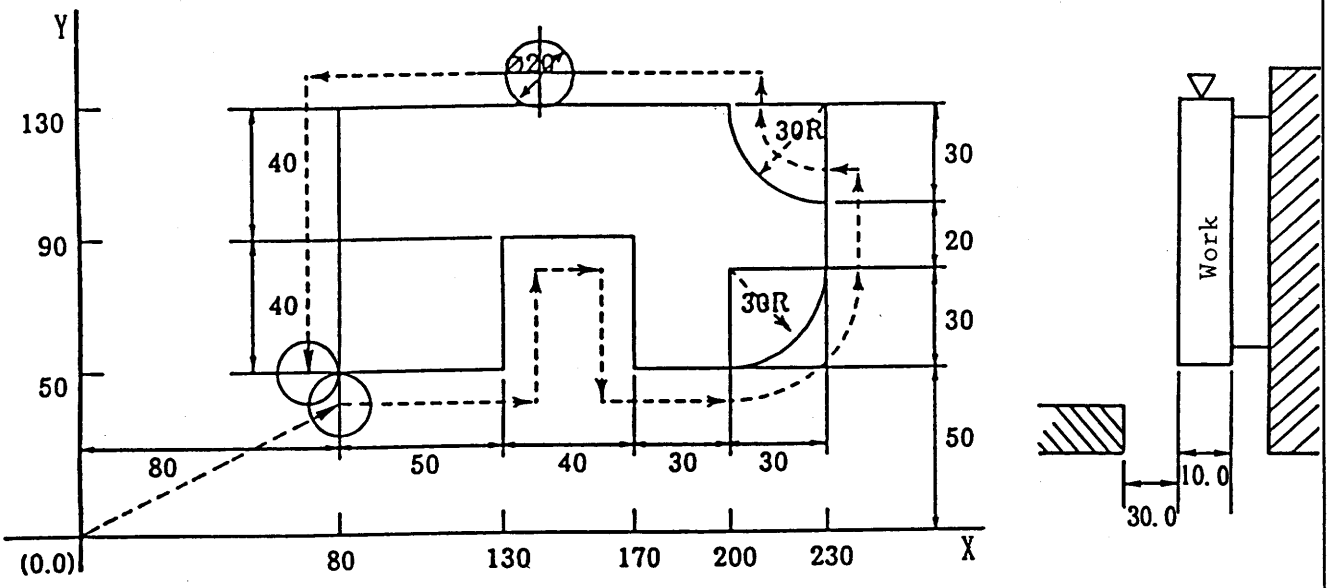
Programming route at finishing time

```
G54G90*
S300*                          ↓
G41G01X20.0Y20.0D31F100*       ①
G03X0Y40.0R20.0F80*            ②
(G03X0Y40.0) J-40.0*           ③
(G03) X-20.0Y20.0R20.0*        ④
G01 G40X0Y0*                   ⑤
```

Finishing maching sequence

5-13 Example of tool diameter compensation program

Tool No.T01 Tool diameter ϕ20 Offset No. D01 (Offset amount 10.0)

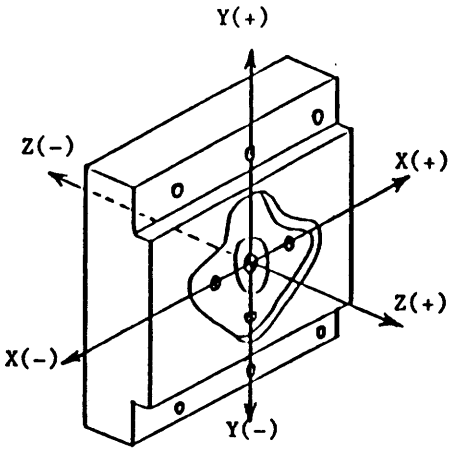


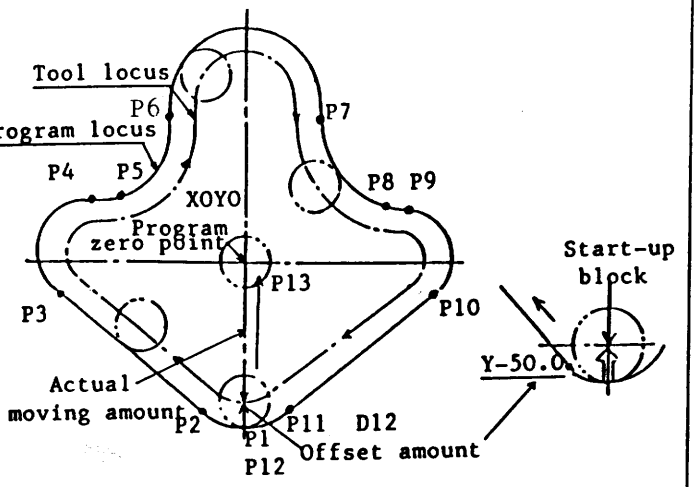
Ⓐ Absolute program

```
T01 M06
G90 G54 G00 X0 Y0 S400
(G43) Z30.0 (H01)
M03
Z3.0
G42 G17 G00 X80.0 Y50.0 (D01)
G01 Z-15.0 F100
X130.0
Y90.0
X170.0
Y50.0
X200.0
G03 X230.0 Y80.0 R30.0
G01 Y100.0
G02 X200.0 Y130.0 R30.0
G01 X80.0
Y50.0
G00 Z30.0
G40 X0 Y0
M05
G91 G28 Z0
M30
```

5-14 Example of tool diameter compensation program

Offset amount D12 = $\frac{\text{Tool diameter}}{2}$






```
                    T12 M06
                    G90 G54 G00 Z0 Y0 S500
                    (G43) X50.0 (H12)
                    M03
                    Z3.0
                    G01 Z-5.0 F50
Start-up block  P1  G01 G42 Y-50.0 F100 D12
                P2  G02 X-12.585 Y-45.544 R20.0
                P3  G01 X-54.439 Y-11.659
                P4  G02 X-45.0 Y15.0 R15.0
                P5  G01 X-41.306
                P6  G03 X-21.867 Y39.708 R20.0
                P7  G02 X21.867 R-22.5
                P8  G03 X41.306 Y15.0 R20.0
                P9  G01 X45.0
                P10 G02 X54.439 Y-11.659 R15.0
                P11 G01 X12.585 Y-45.544
                P12 G02 X0 Y-50.0 R20.0
Cancel block    P13 G01 G40 Y0 F500
                    G00 Z50.0
                    M05
                    G91 G28 Z0
```

5-15 Tool diameter compensation vector keep (G38), Offset vector change and Tool radius compensation corner circular arc (G39)

(1) Tool diameter compensation vector keep (G38)

The offset vector of the previous block can be kept or its direction can be changed by G38 during the offset mode.

(a) Offset vector keep

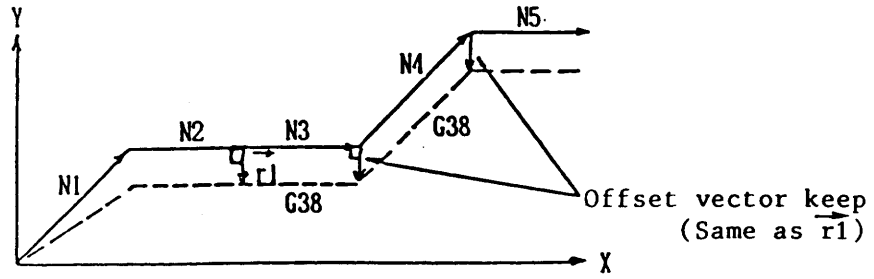
$$\left\{\begin{matrix}G00\\G01\end{matrix}\right\}\ G38\ \underbrace{\alpha\_\beta\_}\ ;$$

Travel axis of offset plain

The offset vector ins't made by this command, but the offset vector of the terminal position of the previous block is kept.

(Program example) However, radius offset amount: 20.0

```
G54 G90 G00 X0 Y0;
N1 G17 G01 G42 X50. Y50. D12 F200;
N2 X100.;
N3 G38 X150.;                    Offset vector keep
N4 G38 X200. Y100.;              Offset vector keep
N5 X250.;
```



(b) Offset vector change

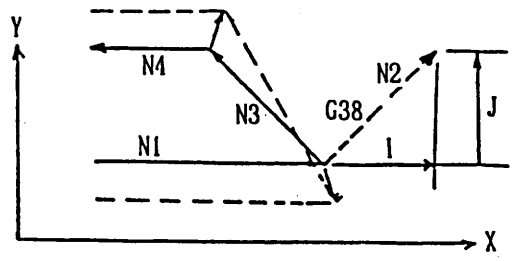
$$\left\{\begin{matrix}G00\\G01\end{matrix}\right\}\ G38\ \underbrace{I\_J\_K\_}\ ;$$

Terminal point vector change

The offset vector of the previous block terminal point can be changed through designating the terminal point vector I, J and K of the offset plane by this command.

(Program example) However, radius offset amount: 20.0

```
G42 G01 D12 F200;
      :
N1 G01 G91 X100.;
N2 G38 I50. J50.;                    Offset vector change
N3 X-50. Y50.;
N4 X-50.;
```



(2) Tool radius compensation corner circular arc (G39)

During offset mode, the corner can be moved in circular arc by G39 command.

G39;

When I, J and K are omitted in the block of G39 command, the terminal point vector of the corner circular arc moves in the corner circular arc that may become perpendicular to the start point of the next block.

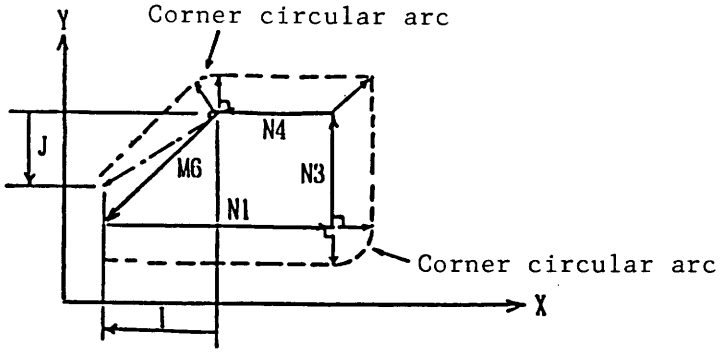
G39 I_J_K_;

Terminal vector change

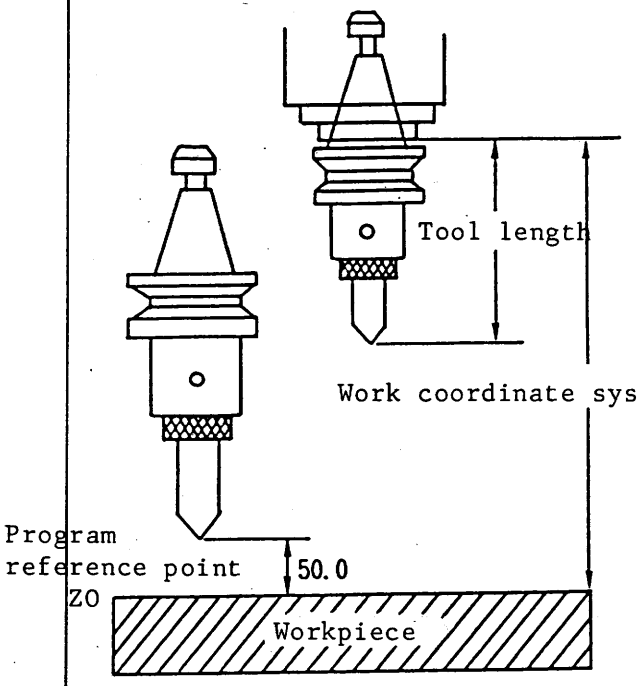
When I, J and K are commanded in the block of G39 command, the terminal point vector of the corner circular arc moves in the corner circular arc that may become perpendicular to the vector designated by I , J and K.

(Program example) However, radius offset amount: 20.0

```
    G42 G01 F200;
          :
          :
N1 G01 G91 X100.;
N2 G39;                                  Corner circular arc
N3 Y50.;
N4 X-50.;
N5 G39 I-50. J-35.;                      Corner circular arc
N6 X-50. Y-50.;
```



5-16 Conception of tool length compensation



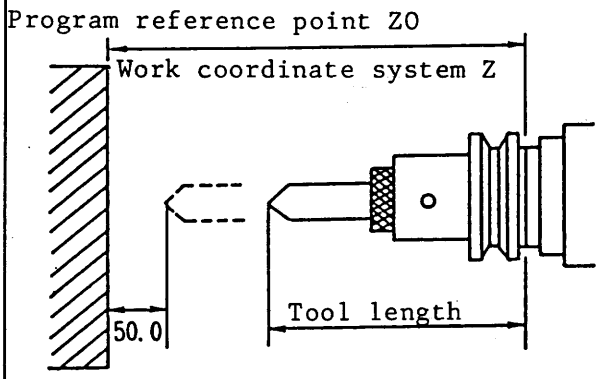
The tool length compensation is engaged by the lower 4 digits tool offset No. of the spindle tool No..

Example) T02 M06

A tool of which tool No. is T02 is mounted on the spindle. The offset amount (length compensation = shape + wear) designated by the tool 02 becomes effective.

Program example

```
G91 G30 Z0
G91 G30 X0 Y0
T02 M06
G90 G54 G00 X100.0 Y0 S1000
Z50.0
M03
G00 Z3.0
  :
```



Tool offset memory

Tool	Tool name	Length compensation		Radius compensation	
		Shape	Wear	Shape	Wear
001		0.000	0.000	0.000	0.000
002		200.000	-0.010	30.000	-0.010
003		0.000	0.000	0.000	0.000
004		0.000	0.000	0.000	0.000
005		0.000	0.000	0.000	0.000
006		0.000	0.000	0.000	0.000

* When making up a program, make up the program of the Z-axis based on the spindle nose without considering the tool length.

* The tool length compensation amount designated by the tool 02 is shape (200.0) + wear (-0.01) = 199.99.

* Measure the tool length before machining and set the tool length on the [TOOL] screen.

* The terminal position of the travel command for the Z-axis can be shifted to the ⊕ side by the value set in the offset memory.

Plural offset when using tool offset by tool No.

When plural H codes are used

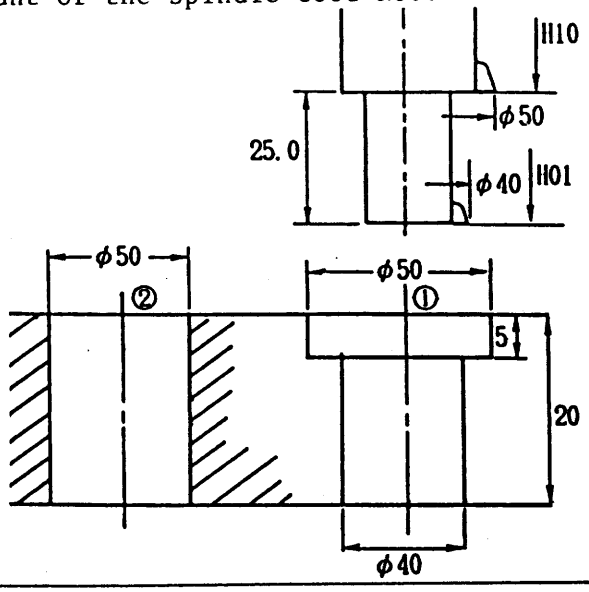
H_ ..... Plural offset by H code is turned ON.

H00 ..... Plural offset by H code is cancelled.

```
T01 M06

①  G90 G54 G00 X0 Y0 S300
   G43 Z50.0 H10
   M03
   G00 Z30.0
   G01 Z-5.0 F30
   G00 Z50.0

   X-100.0
   H00 ←
```

H00 is not for tool length compensation, it means that the tool length compensation returns to the offset amount of the spindle tool No..

```
②  G00 Z3.0
   G01 Z-21.0 F30
   G00 Z50.0
```



5-17 Tool position offset (G45, G46, G47, G48)

Ⓐ G45 ~ G48

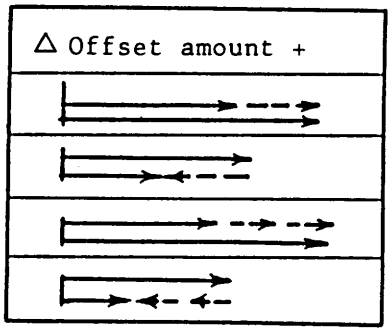
G45 ~ G48 are one shot commands.

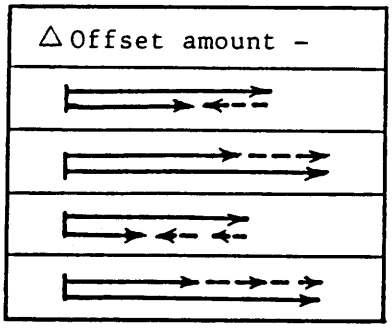
The offset vector of the tool position offset is kept for the travel command after this block.

For axial moving direction

G	Meaning
G45	Elongation
G46	Shrinkage
G47	2-fold elongation
G48	2-fold shrinkage

When the sign of the offset amount is shifted to minus, elongation and reduction are reversed.





{G45, G46, G47, G48} X_Y_D_

When D00 is commanded, the tool position offset is not available.

By D-code, offset No. is indicated.

Ⓑ Indicating method

G45X___D △ △ *

G45X___Y___D △ △ *

G45Z___D △ △ *

Ⓒ

Incremental command

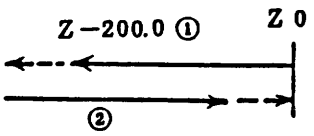
① G45 Z-200.0 D01 *

② G45 Z200.0 D01 *

Absolute command

① G45 Z-200.0 D01 *

② G45 Z0 D01 *



5-18 Tool position offset (G45, G46, G47, G48)

Ⓐ Example of X Y axial program

Upper stage means incremental command

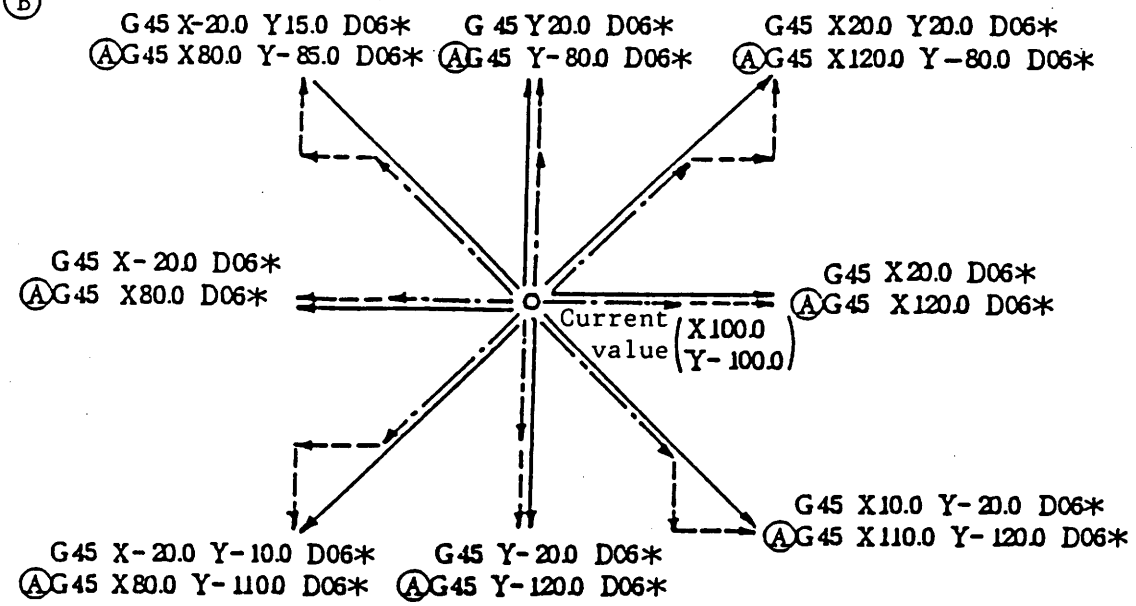
Lower stage means absolute command at time of X100.0Y-100.0.

——- → Program movement

- - - → Compensation amount

——— → Actual movement

Ⓑ



Note)-1 At simultaneous 2-axial time, offset amount is effective on each axis.

Note)-2 G45 ~ G48 are disregarded during the canned cycle mode.

Note)-3 G45 ~ G48 cannot be commanded during the tool radius compensation mode.

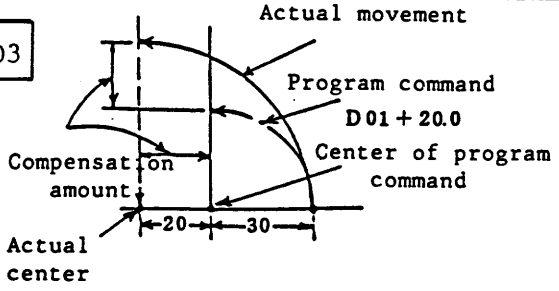
Ⓒ Circular arc interpolation G02, G03

The circular arc is a figure that is right angle to the axis, the tool position offset can be commanded to only a quarter cycle and a one-third cycle.

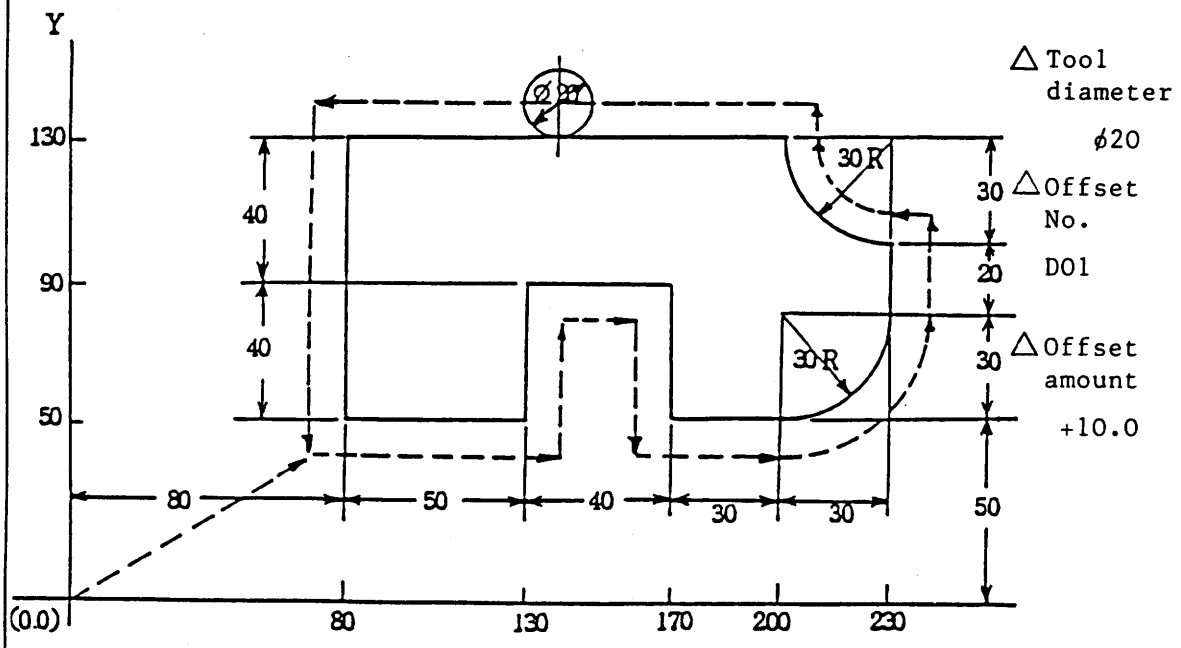
G91 G45 G03 X-30.0 Y30.0 I-30.0 D01*

↓ Equal

G90 G03 X-50.0 Y50.0 I-50.0*



5-19 Example using the positional offset for tool radius



Incremental command

```
N1  G91 G46 G00 X80.0 Y50.0 D01 *
N2  G47 G01 X50.0 F200 *
N3  Y40.0 *
N4  G48 X40.0 *
N5  Y-40.0 *
N6  G45 X30.0 *
N7  G45 G03 X30.0 Y30.0 J30.0 *
N8  G45 G01 Y20.0 *
N9  G46 X0 *
N10 G46 G02 X-30.0 Y30.0 J30.0 *
N11 G45 G01 Y0 *
N12 G47 X-120.0 *
N13 G47 Y-80.0 *
N14 G46 G00 X-80.0 Y-50.0 *
```

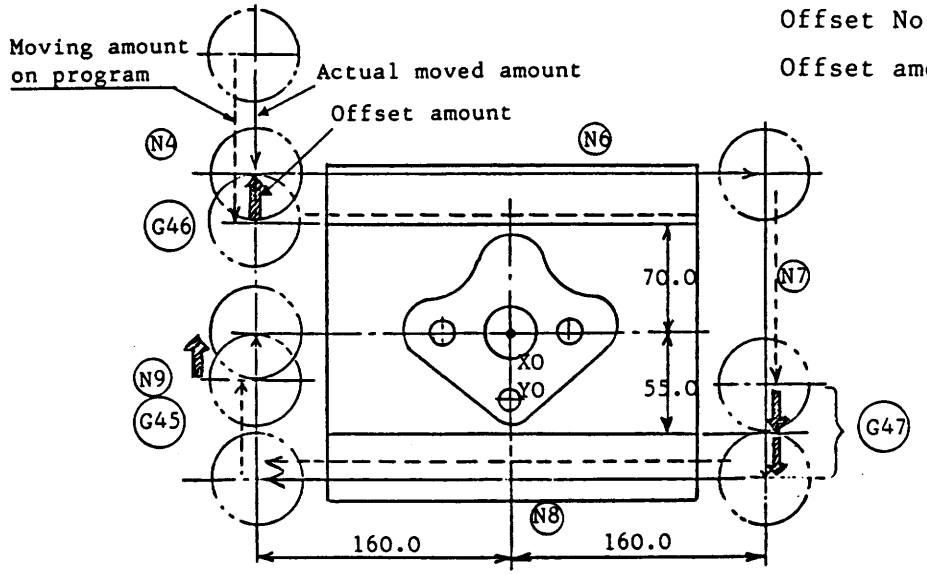
Offset 10.0 is set before auto-operation. Without offset No., D01 setting is effective.

* With G02, G03 command, G45, G46 can be used only in case of $\frac{1}{4}$ and $\frac{3}{4}$ circles.

** With absolute command, offset-amount alone cannot be moved.

5-20 Example using the tool-position offset for milling machining

Tool diameter ϕ75
Offset No. D11
Offset amount +37.5

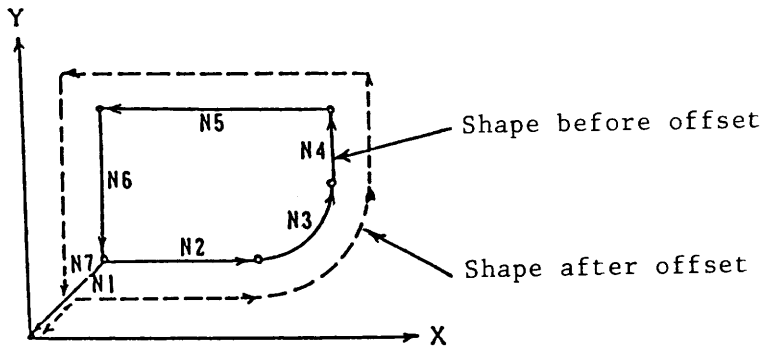


Absolute command

```
N1  G54 G90 G00 X-160.0 S230 *
N2  G43 Z50.0 H01 *
N3  M03 *
N4  G46 Y70.0 D11 *                    1-fold shrinkage
N5  G01 Z0 F2000 *
N6  X160.0 F460 *
N7  G00 G47 Y-55.0 D11 *               2-fold elongation
N8  G01 X-160.0 *
N9  G00 G45 Y0 D11 *
N10 M05 *                              Cancel of offset amount
    :
```

Program example

```
G17 G54 G90 G00 X0 Y0;
G01 G91 F200;
N1 G46 X20. Y20. D01;      Reduces X and Y axes by the offset amount
N2 G45 X40;                Elongates X-axis by the offset amount
N3 G45 G03 X20. Y20. J20.; Elongates X and Y axes by the offset amount
N4 G45 G01 Y20.;           Elongates Y-axis by the offset amount
N5 G47 X60.;               Elongates X-axis by twice as much as the
                           offset amount
N6 G47 Y-40.;              Elongates Y-axis by twice as much as the
                           offset amount
N7 G46 X-20. Y-20.;        Reduces X and Y axes by the offset amount
```



Precautions

(a) H code in place of D code can be used through parameter setting.

(a') G45 ~ G48 are defferent from G41, G42 and G43, and be sure to command D_.

(b) When D00 is commanded, the tool position offset is not available. When D_ is omitted, the tool position offset is not available either.

(c) When the sign of the offset amount designated by D code is shifted to minus, elonation and reduction are reversed.

(d) The tool position offset is treated after the tool raduis compensation.

(e) When G45 ~ G48 are commanded to simultaneous 2 axes travel command, excessive cutting or no cutting portions may occur.

(f) When the tool position offset is engaged during the coordinate rotation mode, correct shape is not obtained.

(g) G45 ~ G48 cannot be commanded during the tool diameter offset mode.

Example) Wrong example

```
G41 G00 X_ Y_
     :
     :
G45 X_ D_
     :
     :
G46 Y_
     :
     :
G40 X_ Y
```

When the tool position offset and the tool redius compensation are combined to use.

```
T01 M06
G90 G54 G00 X_ Y_ S500
Z50.0 ←——— The length compensation of tool No.01 engaged.
M03
G45 X_ D10 ←——— When D_ is commanded by G45 ~ G48, it is not
     )          regarded as D_ of the plural offset.
G46 X_
     )
G41 X_ Y_ ←——— The tool radius compensation is engaged not
     )          by D_, but the radius compensation of the
                spindle tool No.01.
G40 X_ Y_
```

6. G-function (coordinate system)

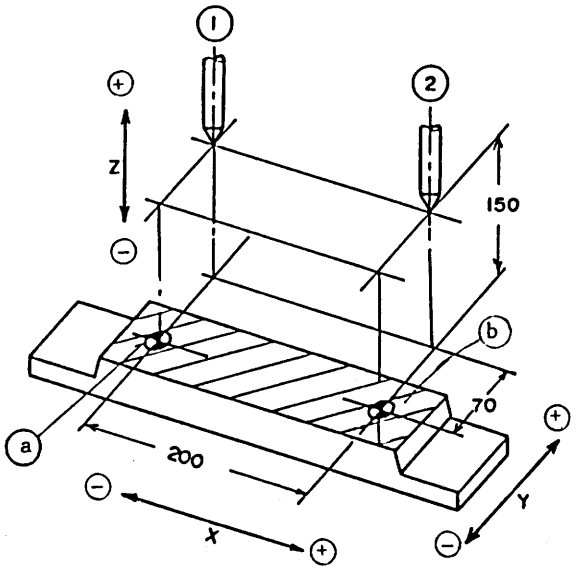
6-1 Setting of coordinate system (G92)

G92

Coordinate system that the current position of tool would be

X△△△△
Y△△△△
Z△△△△
} can be set.

G92 X___Y___Z___*



ⓐ G92X0Y70.0Z150.0 *

Ⓑ G92X200.0Y70.0Z150.0 *

When program zero point is set as ⓐ, tool was located at the current position ①. Coordinate system is this case is as in ⓐ.
Case of ⓑ tool is at ② position.

* That is, command is made for the coordinate value in view of tool-position (spindle position).

Note)-1 G92 can be indicated with no-relation to the absolute/incremental.
In case of absolute, after commanding G92, movement is made in accordance with the coordinate system.

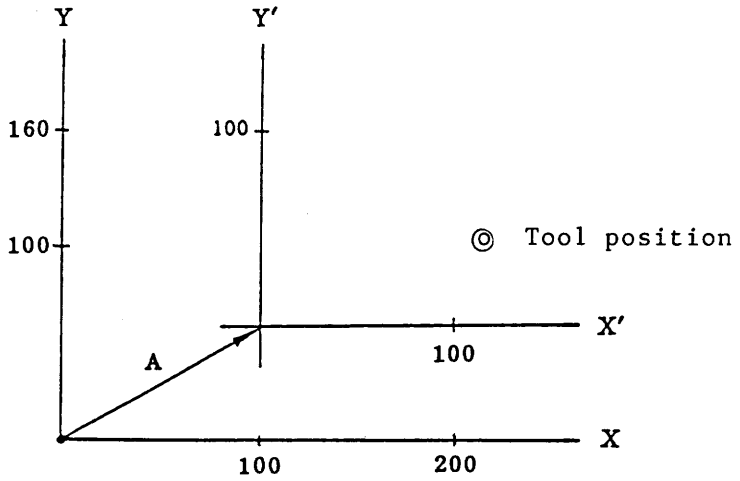
Note)-2 At power-input time, coordinate value of X, Y, Z axes is 0.0.0.
It is necessary to set the coordinate system after manual zero-point return with power input.

Note)-3 At present G92 is hardly used, the work coordinate system is mainly used. (Refer to 6-3.)

6. G-function (coordinate system)

6-2 Caution for the use of (G54 ~ G59) and G92 work-coordinate

Note) In the use of G54 ~ G59, there is no need of setting the coordinate system with G92.
Upon setting the coordinate system with G92, the coordinate system of G54 ~ G59 would move. Especially, do not mix G54 ~ G59 except the case intending to shift G54 ~ G59.



At the state of G54, when tool is at the position (200.160), when commanding G92X100Y100, work-coordinate system with deviation of vector-A amount alone can be formed.

6-3 Work-coordinate system (G54 ~ G59)

G54 ~ G59

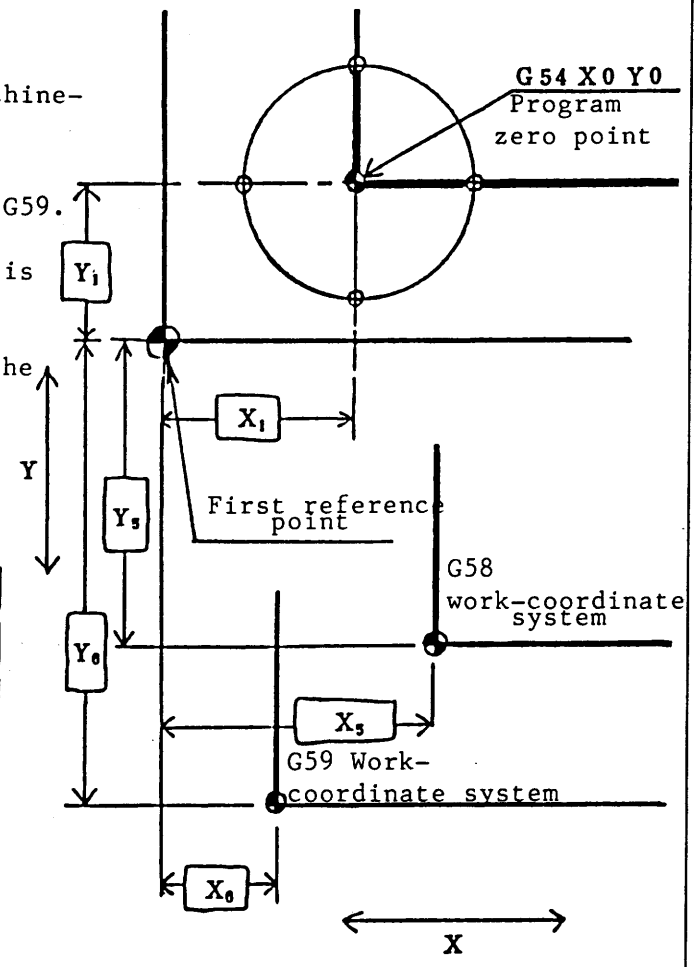
It is possible to set the machine-proper 6 coordinate system.

Selection is made with G54 ~ G59.

Setting of coordinate system is made by the distance from the first reference point up to the program zero point.

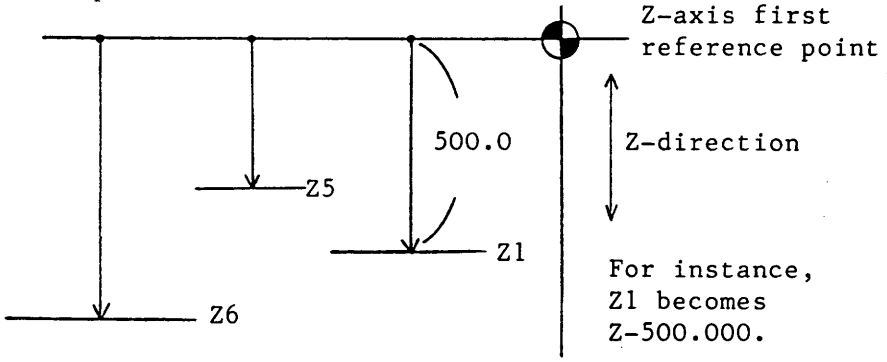
Setting value

Setting value	G54	G58
	X1	X5
	Y1	Y5
	Z1	Z5
G59		
	X6	
	Y6	
	Z6	



Work-coordinate system 1~6 (G54~G59) can be exactly established after zero-point return of machine after power input.

When the power is turned ON, G54 has been selected.



6. G-function (Coordinate system)

6-4 Work-coordinate system G54, G55 and coordinate system setting G92

G54, G55 and G92

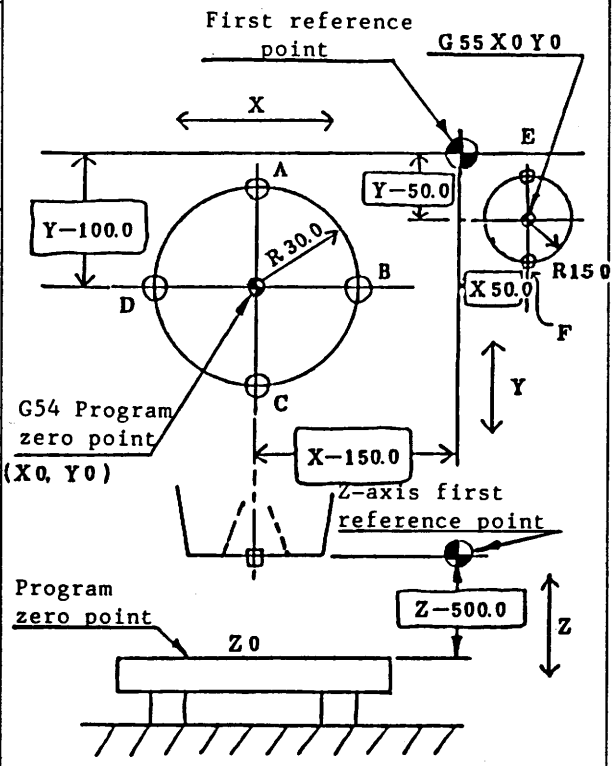
As shown in right diagram, previously set on the screen of offset before auto-operation.

G54	X-150.0	G55 X	50.0
	Y-100.0	Y	-50.0
	Z-500.0	Z	-500.0

△ By commanding G54G90G00X0Y30.0*
Movement is made to work-coordinate system of G54, X0,Y30.0 namely, Ⓐ point.

△ By commanding G55G90G00X0Y15.0*
Movement is made to work-coordinate system of G55 X0Y15.0 namely, Ⓔ point.

△ Comparison in using G92 and G54, G55 is as follows:
Start point in using G92 can be any place for starting G54, G55 on machine-zero point for both and X and Y.
(program of X or Y alone)



Case using G92.

N10 G92X150.0Y100.0 *		
N11 G90G00X0Y30.0 *	Ⓐ point	N51 G54G90G00X0Y30.0 *
N12 X30.0Y0 *	Ⓑ point	N52 X30.0 Y0 *
N13 X0 Y-30.0 *	Ⓒ point	N53 X0 Y-30.0 *
N14 X-30.0Y0 *	Ⓓ point	N54 X-30.0Y0 *
N15 X200.0Y65.0 *	Ⓔ point	N55 G55(G90)(G00)X0Y15.0*
N16 X200.0Y35.0 *	Ⓕ point	N56 X0 Y-15.0 *
N17 G91G28X0Y0 *	First reference point	N57 G91G28X0Y0 *

Case using G54, G55.

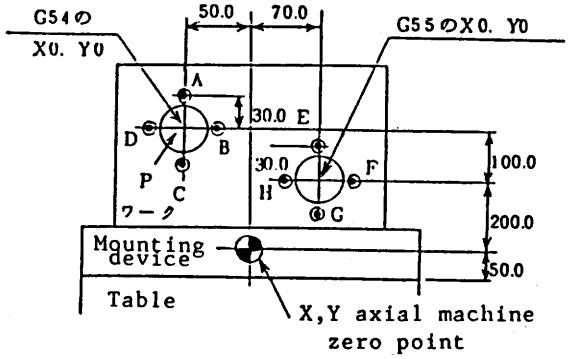
Code within ( ) can be omitted.

6-5 Example using work-coordinate system

G54	X	0	G56	X	700
	Y	0		Y	2000
	Z	0		Z	4000
G55	X	−500	G57	X	900
	Y	3000		Y	1500
	Z	4000		Z	3000

By commanding G54 G90 G00 X0 Y30.0 *, tool moves to X0. Y30.0 (A-point) of work-coordinate system of G54.

By commanding G55 G90 G00 X0 Y30.0 *, tool moves to X0. Y30.0 (E-point) of work-coordinate system of G55.



In case of the above diagram G92 is compared with G54 ~ G59 in each using time.

Start point at time using G92 shall be machine-zero point for both X and Y, while any start point can be taken in case of G54 ~ G59.

It shall be the program of X/Y movement alone.

Case using G92 (program-zero point: P-point)		Case using G54, G55
		Code in ( ) can be omitted.
G92 X50.0 Y-300.0 *		
G90 G00 X0 Y30.0 *	A-point	G54 G90 G00 X0 Y30.0 *
X30.0 Y0 *	B-point	X30.0 Y0 *
X0 Y-30.0 *	C-point	X0 Y-30.0 *
X-30.0 Y0 *	D-point	X-30.0Y0 *
X120.0 Y-70.0 *	E-point	G55(G90)(G00)X0Y30.0 *
X150.0 Y-100.0*	F-point	X30.0Y0 *
X120.0 Y-130.0*	G-point	X0 Y-30.0 *
X90.0 Y-100.0*	H-point	X-30.0Y0 *

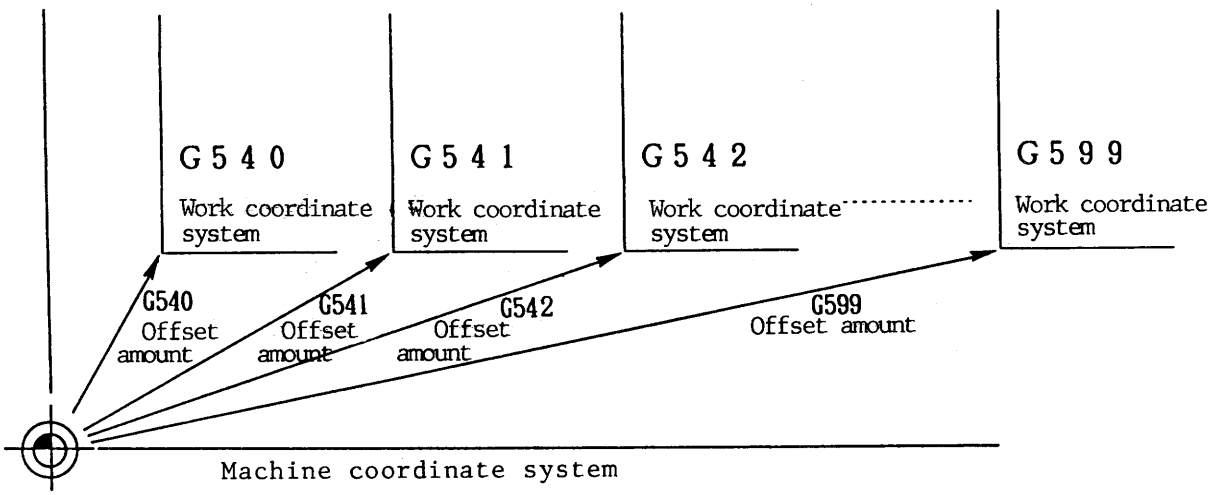
6-6 Addition of work coordinate system pair number (G540 ~ G599) Option

60 pcs. of proper coordinate system can be set by commanding G540 ~ G599.

Before commanding G540 ~ G599, set the offset amount (the position of the machine coordinate system at the time when a tool nose is located on the reference point of the work coordinate system.) form the machine reference to the reference point of each work coordinate system.

(1) Command from

G540;	G540 Work coordinate system selection
G541;	G541 Work coordinate system selection
G542;	G542 Work coordinate system selection
⋮	⋮
G599;	G599 Work coordinate system selection



(2) Program example

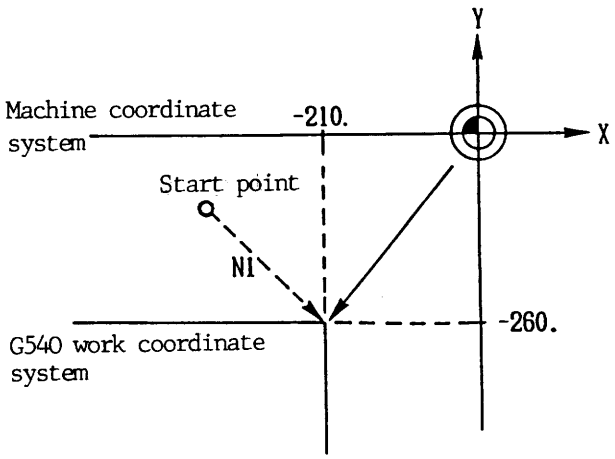
N1 G540 G90 G00 X0 Y0;

However, G540 offset amount

X-210.

Y-260.

When this command is executed, the tool is positioned to the work coordinate system (0.0), and the position of the coordinate system becomes (-210., -260.).



(3) Precautions

(a) G540 ~ G599 and G54 ~ G59 are the same group of G code.

(b) When G540 ~ G590 are commanded after the coordinate system is newly set by G92, the offset amount of the reference point of the machine coordinate system and the offset amount of the work coordinate system of G540 ~ G599 are relatively changed by the new setting amount.

(c) When the work coordinate systems of G540 ~ G599 are established after G92 commanded, it is necessary to either perform the manual reference point return or give G921 (work coordinate system preset) command.

(d) Set the offset amount of G540 ~ G599 on the screen of the work coordinate.

Work coordinate system preset (G921)

When the first manual reference oint return is performed after the power supply for the NC unit is turned ON, the machine coordinate system is set and next the work coordinate system is set.

When the manual reference point return is performed in the state of the reset, the work coordinate system is set.

The work coordinate system is shifted from the machine coordinate system by the following commands or operations.

In this case, to reset the work coordinate system by making the machine coordinate system a reference is named work coordinate system preset.

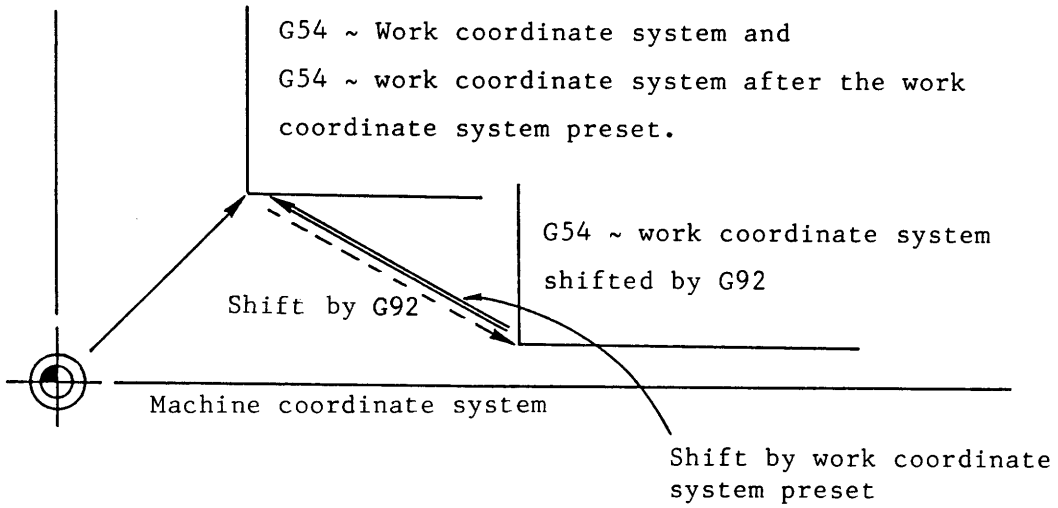
(a) When manual intervention is made at the manual absolute signal OFF.

(b) When an axial travel command is given at the machine lock.

(c) When a tool moves by the handle interruption, and simultaneous manual and automatic operations.

(d) When the work coordinate system is shifted by G92 command.

(e) When the work coordinate system moves to the origin by the MDI operation.



(1) Command form/operation

(a) The case by G code

G921 XO YO ZO ..... ;

Preset axis of the work coordinate system

(b) Case by operation

The manual reference point return is performed at the state of reset (OP signal OFF).

(2) Precautions

(a) In the case when it is made by G921, tool radius compensation, tool length compensation and tool position offset are cancelled.

(b) The work coordinate system can be preset immediately before executing the first block changed over from the state of reset to the state of automatic operation by parameter setting.

6. G-function (coordinate system)

6-7 Selection of machine-coordinate system (G53)

G53

Upon making the first reference point return, the display at (MACHINE) coordinate system shall all be zero as follows:

X0
Y0
Z0

```
G90G53X0Y0Z0 *
```

Program example

```
G90G53IP____*
G90G53G00X100.0Y-100.0 *
```

CRT-screen after execution, namely, with (MACHINE) coordinate system, tool moves to the command position.

```
MACHINE
 X 100.000
 Y-100.000
 Z
```

Note)-1 G53 is one-shot G-code, and it is effective for the commanded block alone.

Note)-2 G90 is effective in the absolute mode, and G91 becomes the alarm No.146(G53 ERR.) in the incremental mode.

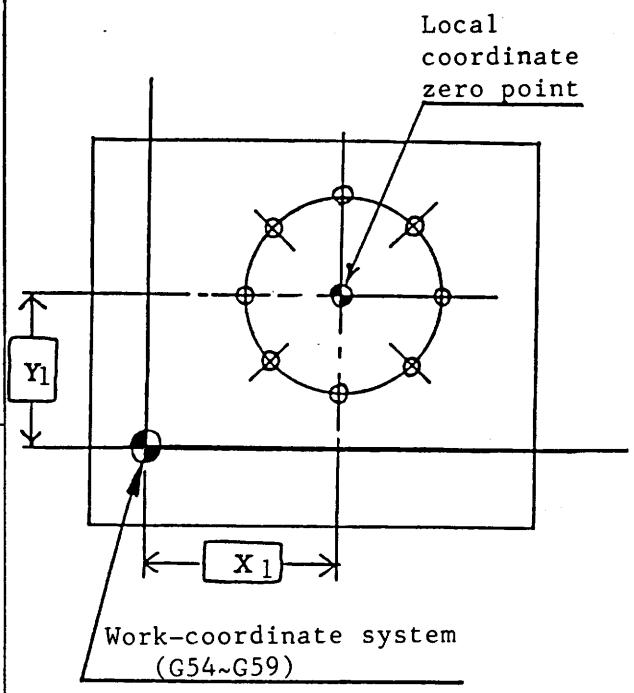
Note)-3 IP means coordinate value. X___Y___Z___ would be added with 3-axes, and A-axis or B-axis would be added with 4 axes.

6-8 Local coordinate system (G52)

G52

In programming with work-coordinate system, another coordinate system may be prepared in the work-coordinate system for easier programming.

It is called "local coordinate system".



G52 X1__ Y1__ *

(Example G52X□ Y□ *)

How to prepare local coordinate system

With the right command, another (child) local (G54~G59) coordinate system can be set to (G54~G59).

Zero-point of each local coordinate system becomes X□, Y□ of X1 Y1-position with each work-coordinate system.

Note)-1 By manual reference point return, zero-point is matched in between work-coordinate system and local coordinate system of returned axis to reference point.
That is, it is the same as that commanding G52α0; (α : returned axis to reference point).

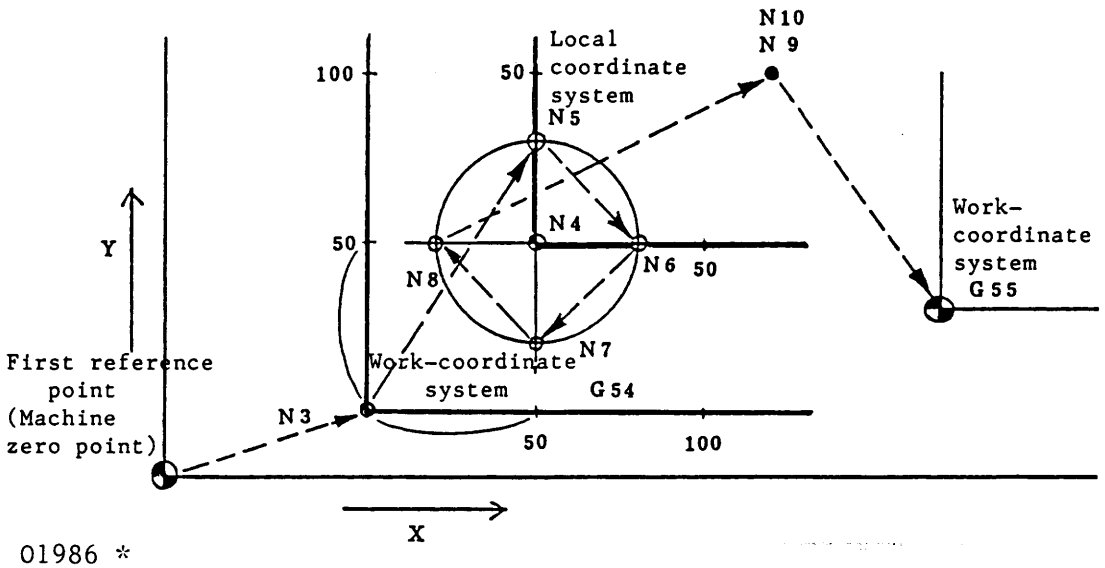
Note)-2 Even by setting the local coordinate system, there is no change in work coordinate system and machine-coordinate system.

Note)-3 By reset, local coordinate system is cancelled.

Note)-4 In case without commanding the total-axial coordinate value for setting the work-coordinate system by G92 command, cancel is made for the local coordinate system of the axis without command of the coordinate value.

Note)-5 At tool-diameter compensation, temporary offset cancel is made with G52.

6-9 G52 program example




```
01986 *

N1   G91G28Z0  *
N2   G28X0Y0   *
N3   G54G90G00X0Y0 *
N4   G52X50.0 Y50.0 *  Local
                       setting
N5   G54X0     Y30.0 *
N6      X30.0 Y0     *
N7      X0     Y-30.0*
N8      X-30.0Y0     *
N9   G54X100.0Y100.0*
N10  G52X0     Y0     * Cancel
N11  G90G55X0Y0       *
N12  M01              *
      S
     M30              *
```

N	Screen MACHINE display		Screen ABS display	
	X	Y	X	Y
1				
2	0.0	0.0	−300.0	175.0
3	300.0	−175.0	0.0	0.0
4	300.0	−175.0	−50.0	−50.0
5	350.0	−95.0	0.0	30.0
6	380.0	−125.0	30.0	0.0
7	350.0	−155.0	0.0	−30.0
8	320.0	−125.0	−30.0	0.0
9	450.0	−25.0	100.0	100.0
10	450.0	−25.0	150.0	150.0
11	500.0	−150.0	0.0	0.0

* Refer to the position display of CRT-screen.

* Machine-display Y-coordinate shows minus-display.

6-10 Programmable data input (G10)

(1) Setting of an offset amount for the work coordinate system

(a) Command form

G10 L2 P_X_Y_Z_...R_; Setting of G54 ~ G59

However, P0 : Designation of external work reference point offset

P1 ~ P6 : Designation corresponding to the work coordinate system, G54 ~ G59

X,Y,Z,... : Work reference point offset amount for each axis

R : Rotating angle

G10 L20 P_X_Y_Z_...R_; Settong of G540 ~ G599

However, P0 : Designation of external work reference point offset

P10 ~ P60 : Designation corresponding to the work coordinate system, G540 ~ G599

X,Y,Z,... : Work reference point offset amount for each axis

R : Rotating angle

G10 L21 P_X_Y_Z_...R_; Setting of common reference point shift amount

However, P0 ~ P5 : No. of common reference point shift amount

X,Y,X,... : Common reference point shift amount for each axis

R : Length of attachment (effective only for P5)

(b) Precautions

(i) The commands such as the below mentioned can be given.

G10 L2 P_X_Y_Z_...R_; Setting of G54 ~ G59

However, P54 ~ P59 : Designation corresponding to the work coordinate system, G54 ~ G59

G10 L20 P_X_Y_Z_...R_; Setting of G540 ~ G599

However, P540 ~ P599 : Designation corresponding to the work coordinate system, G540 ~ G599

(ii) Do not put a decimal point down.

(2) Setting of tool offset amount

Tool offset amounts can be set by program commands.

(a) Command form

G10 L10 P_R_; Setting of tool length form offset

G10 L11 P_R_; Setting of tool length wear offset amount

G10 L12 P_R_; Setting of tool radius form offset amount

G10 L13 P_R_; Setting of tool radius wear offset amount

However, P: Offset No.

R: Offset amount

(b) Precautions

(i) The classification of absolute/incremental for the offset amount commanded is in conformity with G90/G91.

(ii) When G10 L1 P_R_; is commanded, a tool length form offset amount is set.

(iii) Do not put a decomal point down.

6-11 G10 programmable data input

Ⓐ Change of work-coordinate system

By the following commands, each coordinate system can be rewritten into each work-coordinate system.

G10L2P1X__Y__Z__

Example)

```
G90
G10L2P1X100.0Y-100.0Z-300.0
G10L2P2X50.0Y-100.0Z-400.0
N1
G54G90G00X0 Y0 S500
G43 Z50.0 H01
M13
)
(
N2
G55G90G00X0 Y0 S1000
)
(
M30
```

P = 1~6: Indication corresponding to the work-coordinate system 1~6.

P1 = G54

〃 〃

P6 = G59

G54 X100.0	G55 X50.0
Y-100.0	Y-100.0
Z-300.0	Z-400.0

By G10-reading with the left program, the above change is obtained.

Note) Change with G90 (absolute), Adding calculation with G91 (incremental)

6-12 One directional positioning (G60) Option

The final positioning shall be always made from one direction commanded.

By using this function, high accuracy positioning can be obtained.

(1) Command form

(a) Incase of one shot G code

G60 X_Y_Z_...; Effective only for G60 command block.

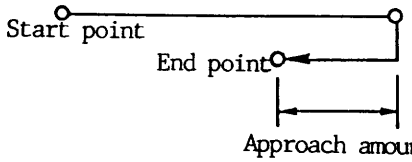
(b) In case of modal G code

G60 X_Y_Z_...;
X_Y_Z_...; } One directional positioning
:
G00; When a command other than G60 is given by G code in the 01 group, G60 is cancelled.

(2) Program example

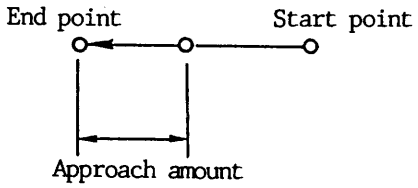
(a) When moved to the + direction

G60 G91 X100.;



(b) When moved to the - direction.

G60 G91 X-100



(3) Final positioning direction PARAMETER NO.6407

Approach amount > 0 : Positioning direction is plus direction.

Approach amount < 0 : Positioning direction is minus direction.

Approach amount = 0 : One directional positioning is not made.

(4) Precautions

(a) Classification of one shot/modal of G60 is set a parameter.

(b) During canned cycle, hole positioning is made by G60. However, one directional positioning is not effective for the shift amounts for G76 and G87.

(c) During mirror image, the mirror image is not effective for the approach amount of one directional positioning.

7. G-function (canned cycle)

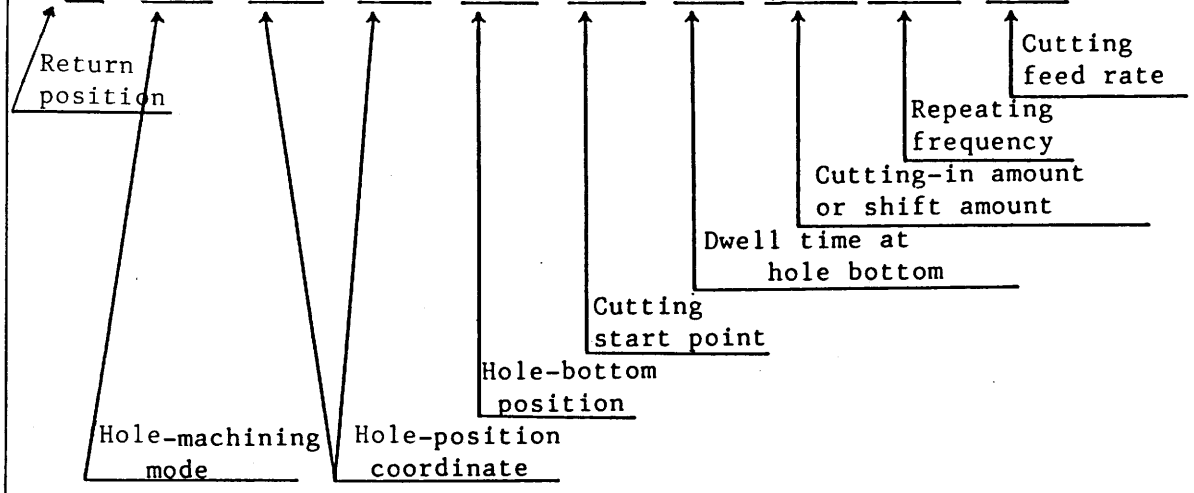
7-1 Canned cycle (G73 ~ G89)

Ⓐ Special operations are requested for machinings such as drilling, boring, spot facing, tapping. It extends to the several blocks. Special movement was enabled by command of 1 block.

G73, G74, G76
G80
G81, G82, G83, G84, G85
G86, G87, G88, G89

Ⓑ Command method

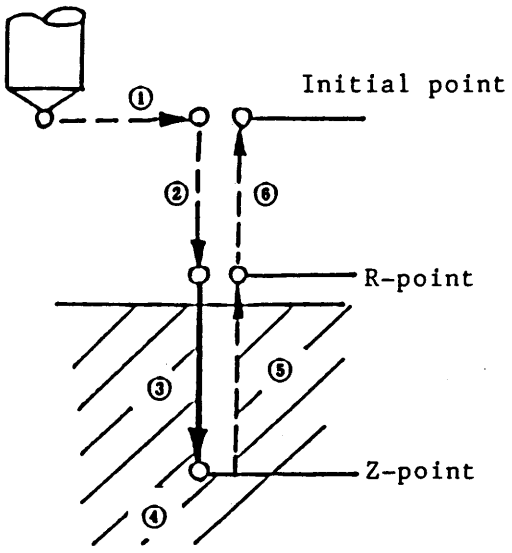
G__ G△△ X____ Y____ Z____ R____ P____ Q____ L____ F____ *



Ⓒ Movement

With canned cycle, movement of ①~⑥ can be executed as one cycle.

① Positioning to the hole position
② Rapid traverse to R-point
③ Hole-machining to Z-point
④ Operation at hole bottom
⑤ Return up to R-point
⑥ Rapid traverse up to the initial point



7-2 Canned cycle

Ⓐ List of canned cycle

Hole-machining mode	Boring operation (-Z direction)	Operation at hole-botton position	Escape operation (+Z direction)	Application
G73	Intermittient feed	—	Rapid feed	High-speed deep hold boring cycle
G74	Cutting feed	Spindle normal turn	Cutting feed	Reverse tapping
G76	Cutting feed	Oriented spindle stop	Rapid feed	Fine boring
G80	—	—	—	Cancel
G81	Cutting feed	—	Rapid feed	Drill, spot-drilling
G82	Cutting feed	Dwell	Rapid feed	Drill, counter boring
G83	Intermittient feed	—	Rapid feed	Deep hole boring cycle
G84	Cutting feed	Reverse turn of spindle	Cutting feed	Tapping
G85	Cutting feed	—	Cutting feed	Boring
G86	Cutting feed	Spindle stop	Rapid feed	Boring
G87	Cutting feed	Spindle stop	Manual/ Rapid feed	Boring, Back boring
G88	Cutting feed	Dwell→Spindle stop	Manual/ Rapid feed	Boring
G89	Cutting feed	Dwell	Cutting feed	Boring

Ⓑ Canned cycle mode

Canned cycle operation can be specified by 3 modes.

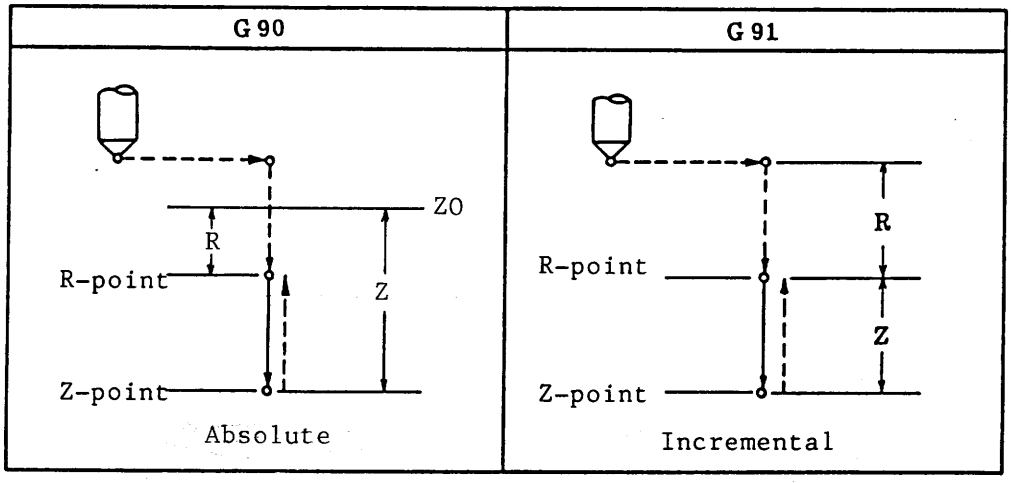
① Data type { G90 Absolute command; G91 Incremental ommand }

② Return level { G98 Initial level return; G99 R-point return }

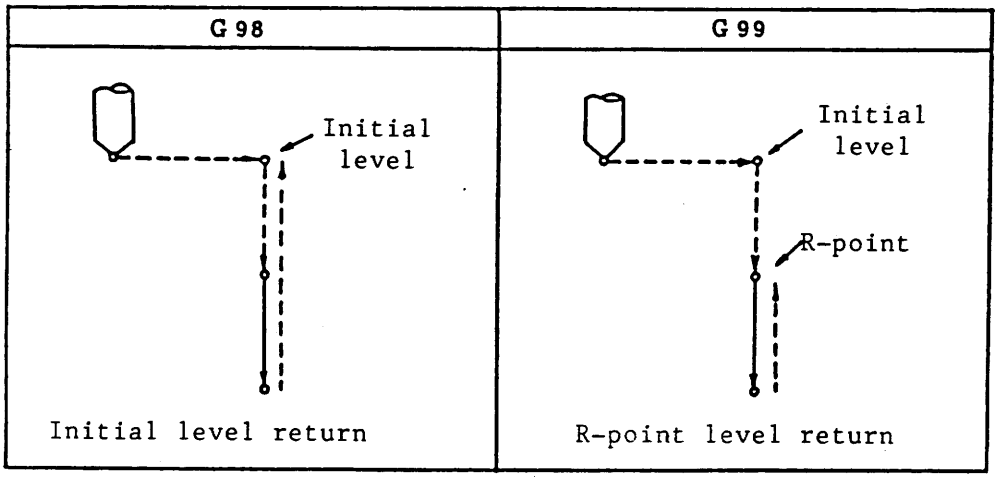
③ Hold-machining mode { G73, G74, G76, G80, G81, ⋮, G89 } See the list.

7-3 Canned cycle (data type, return level)

Ⓐ Data type



Ⓑ Return level (initial level and R-point level)

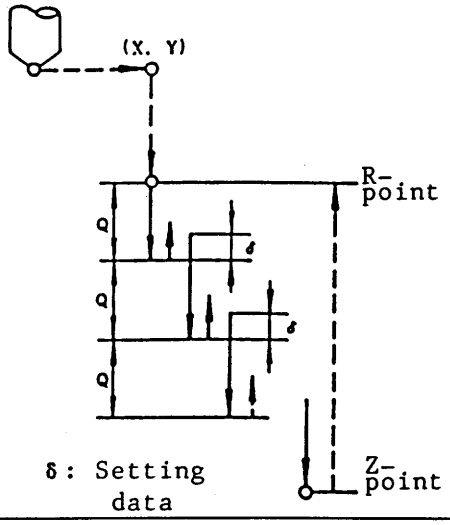
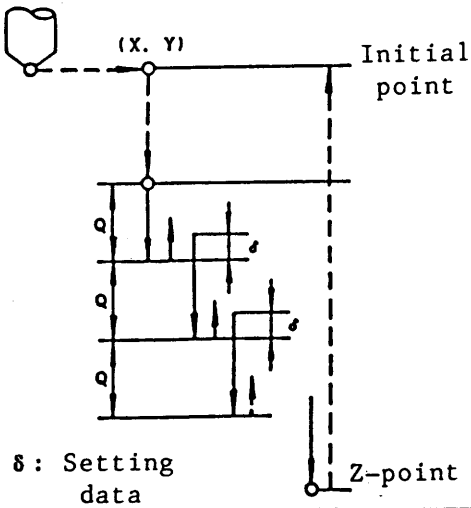
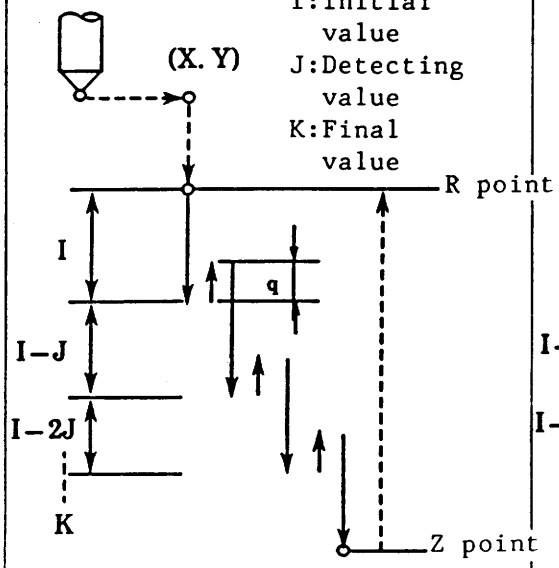
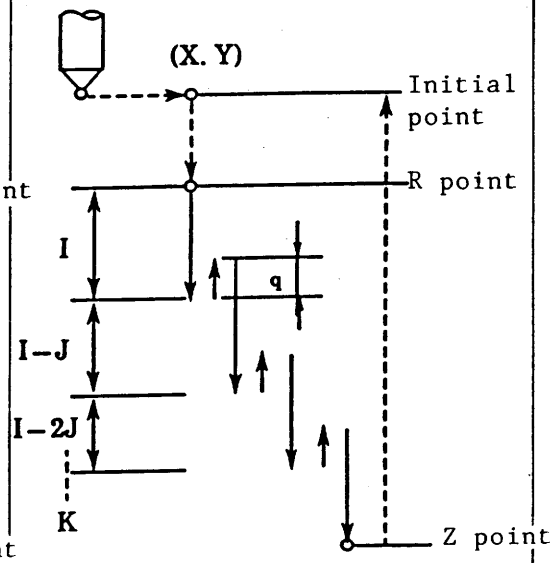


Initial point means Z-axial position at time of canned cycle start.

R-point means cutting-feed starting point (Z-direction) at canned cycle.

7-4 Canned cycle (G73, G74)

Detail of movement

	Case of G99 (R-point return)	Case of G98 (initial point return)
G73 (Canned pitch) High-speed deep hold drill	G73 X.....Y.....Z.....R.....Q.....L.....F.....;  δ : Setting data	 δ : Setting data
G73 (Variable pitch)	G73 X.....Y.....Z.....R.....I.....J.....K.....L.....F.....;  q : Parameter No.6221	

- In G73, it is possible to command a cut-in for variable pitch by using the addresses I, J and K instead of the address Q.

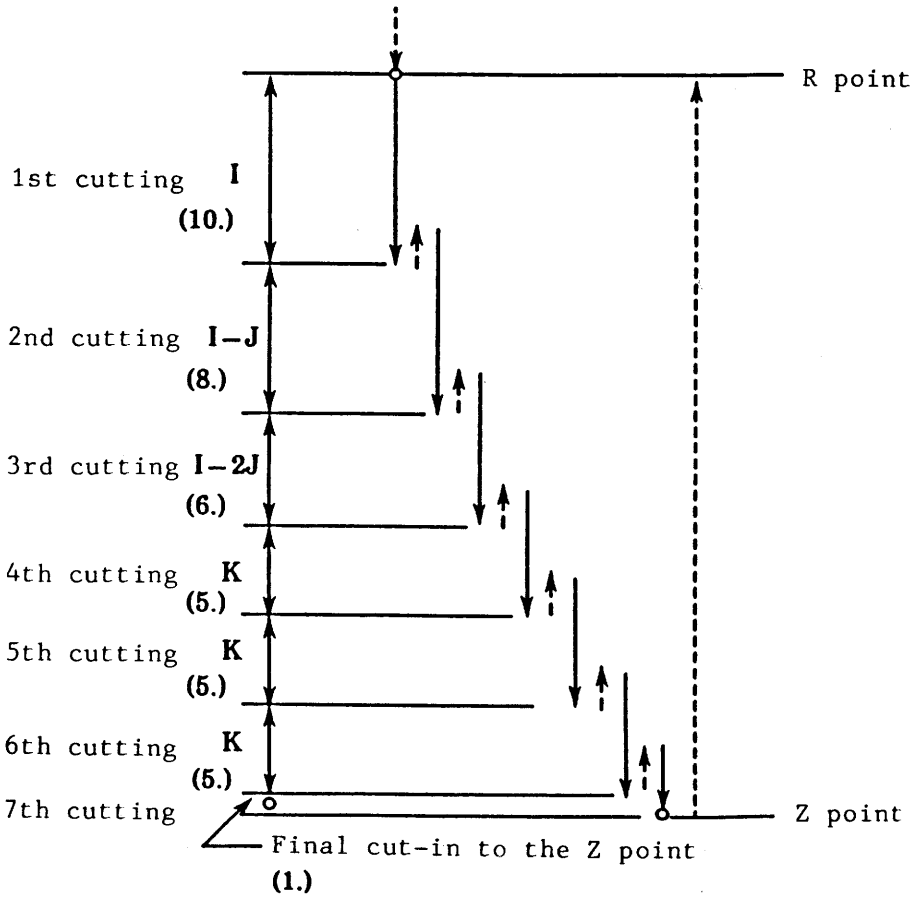
  I : Initial value of cut-in amount
  J : Detecting value after 2nd cut
  K : Final value for cut-in amount

  } Command without sign.

(Example)

G99 G91 G73 X_ Y_ R-10. Z-40. I10. J2. K5. F___ ;

Cut-in amount		
1st cutting	10. mm	← I10.
2nd cutting	8.	← (I10. -J2.)
3rd cutting	6.	← (I10. -2 x J2.)
4th cutting	5.	← K5.
5th cutting	5.	
6th cutting	5.	
7th cutting	1.	← Final cut-in to the Z point



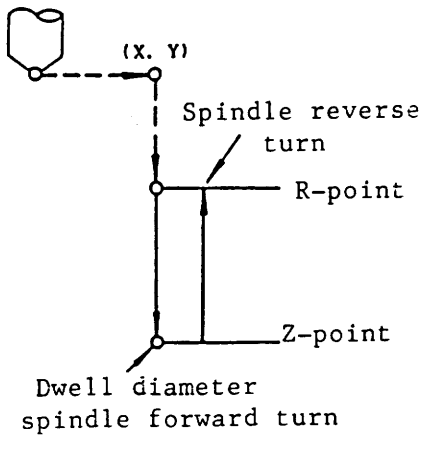
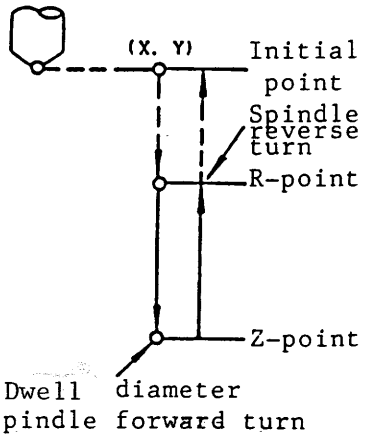
Precautions of the variable pitch commands

Note-1) Q, I, J and K are modal during canned cycle.

Note-2) It is possible to command a variable pitch by using Q instead of the address I.

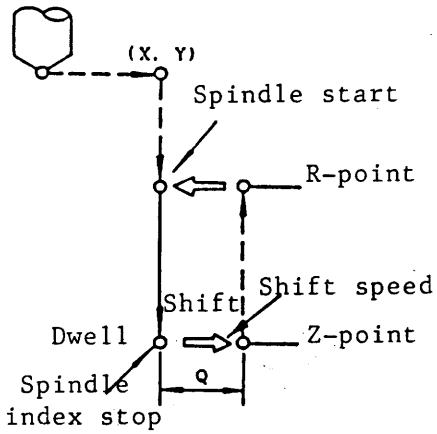
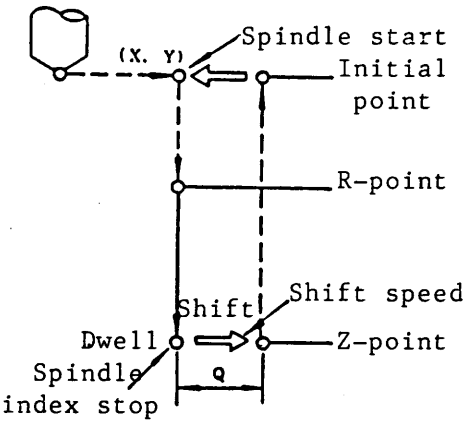
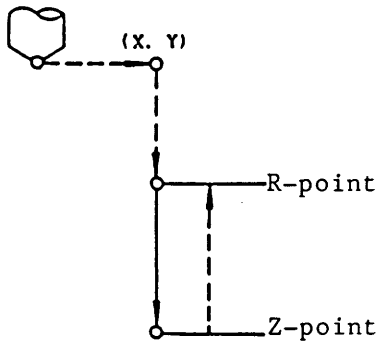
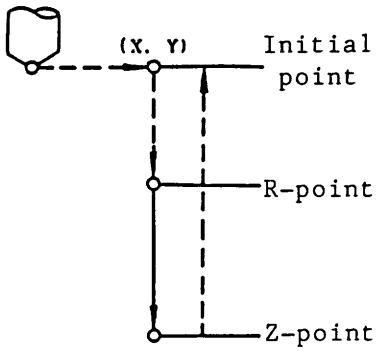
Note-3) Since Q is modal, before commanding variable pitches by I, J and K, command Q0 when Q command was given previously.

Note-4) If Q, I, J and K are simultaneously commanded, the variable pitch cut-in regarding Q as the initial value of cut-in amount is proceeded.

G74 Reverse tap	G74 X.....Y.....Z.....R.....P.....L.....F..... ;	
		

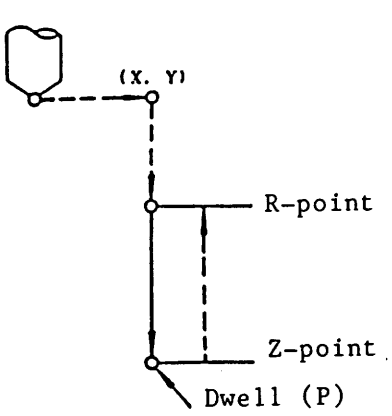
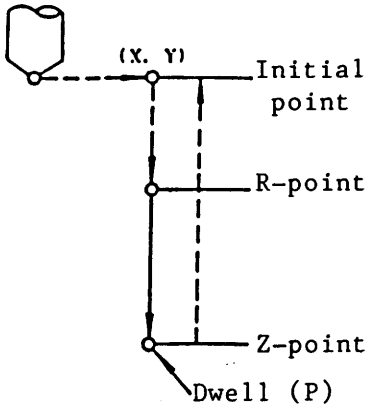
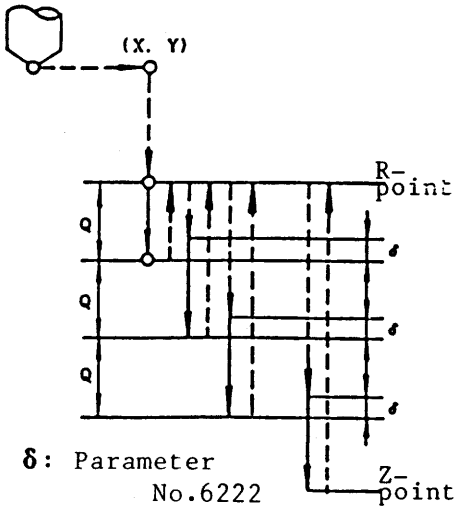
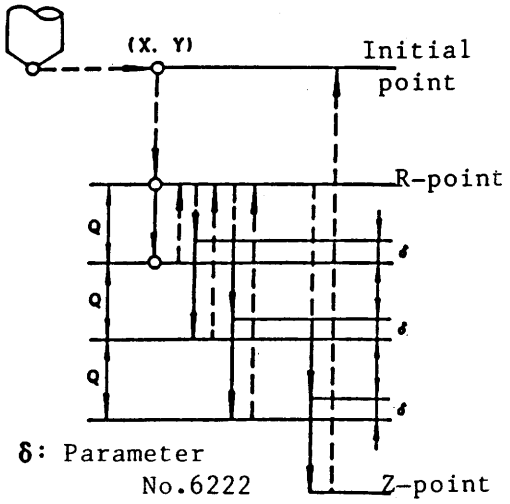
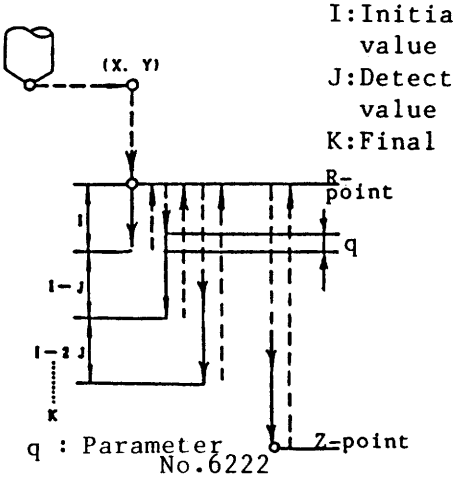
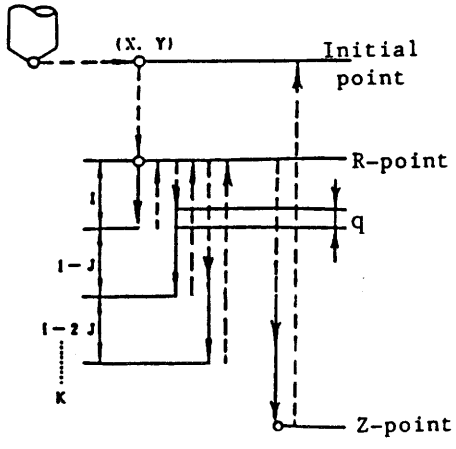
7-5 Canned cycle (G76, G80, G81)

Detail of movement

	Case of G99 (R-point return)	Case of G98 (initial point return)
G76 Boring	G76 X..... Y..... Z..... R..... P..... Q..... L.....F....;  Note) The direction releasing the tool after the spindle positioning has been set by the parameter 6212.	
G81 Drill	G81 X.....Y.....Z.....R.....L.....F..... ;  	
G80 Cancel	G80	G80

7-6 Canned cycle (G82, G83)

Detail of movement

	Case of G99 (R-point return)	Case of G98 (initial point return)
G82 Spot facing	G82 X.....Y.....Z.....R.....P.....L.....F..... ; 	
G83 (Canned pitch) Deep hole drill	G83 X.....Y.....Z.....R.....Q.....L.....F..... ; 	
G83 (Variable pitch)	G83 X.....Y.....Z.....R.....I.....J.....K.....L.....F..... ; I:Initial value J:Detecting value K:Final value 	

- In G83, it is possible to command a cut-in for variable pitch by using the addresses I, J and K instead of the address Q.

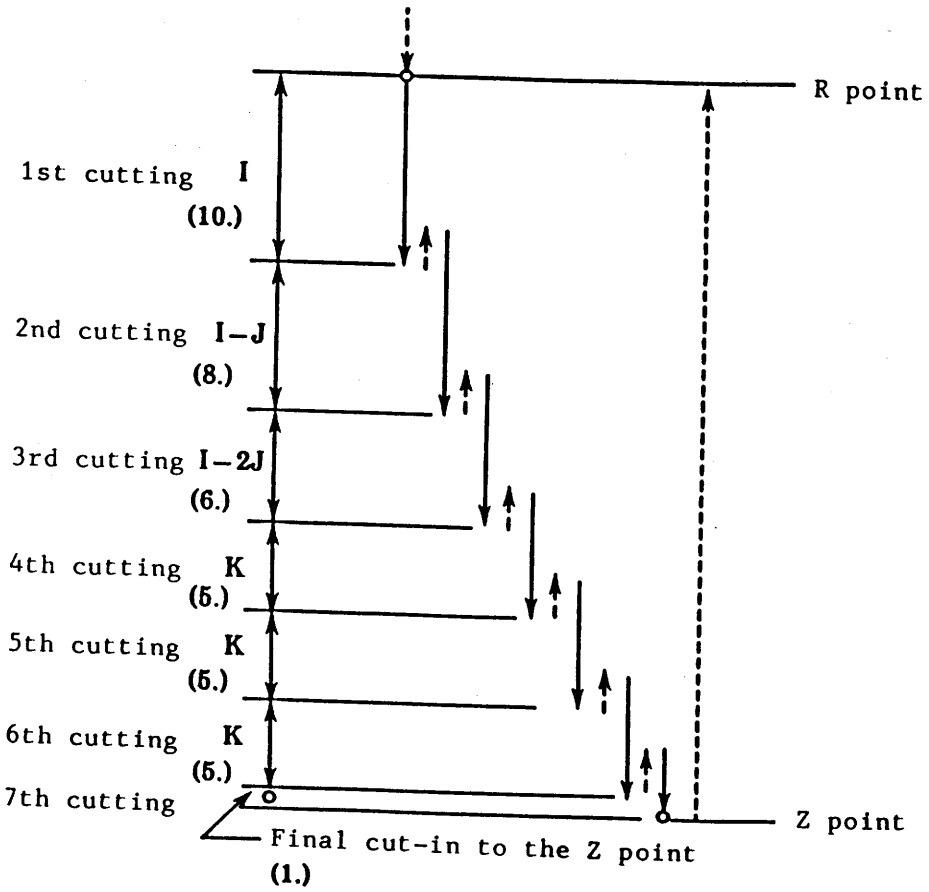
  I : Initial value of cut-in amount
  J : Substract value after 2nd cut
  K : Final value for cut-in amount

  } Command without sign.

(Example)

G99 G91 G83 X_ Y_ R-10. Z-40. I10. J2. K5. F___ ;

Cut-in amount		
1st cutting	10. mm	← I10.
2nd cutting	8.	← (I10. -J2.)
3rd cutting	6.	← (I10. -2 x J2.)
4th cutting	5.	← K5.
5th cutting	5.	
6th cutting	5.	
7th cutting	1.	← Final cut-in to the Z point



Precautions of the variable pitch commands

Note-1) Q, I, J and K are modal during canned cycle.

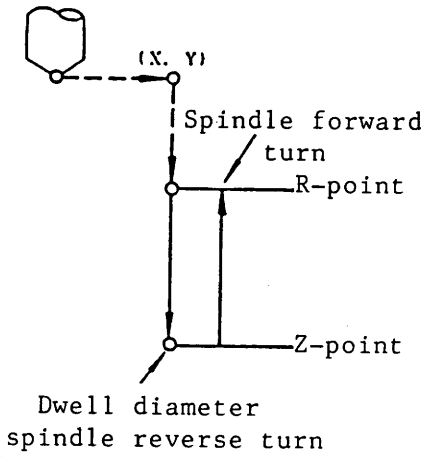
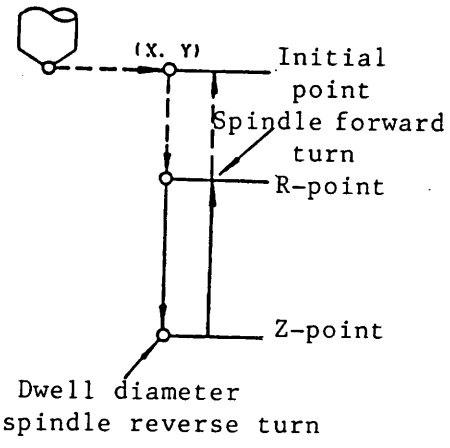
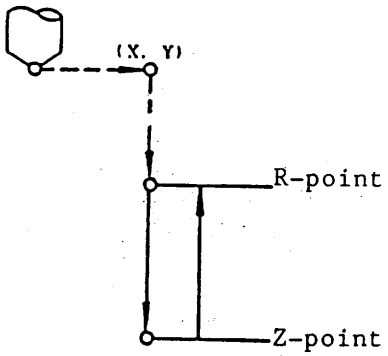
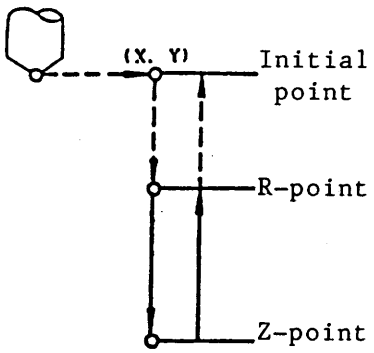
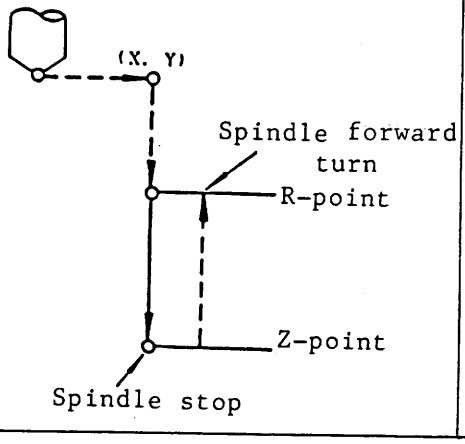
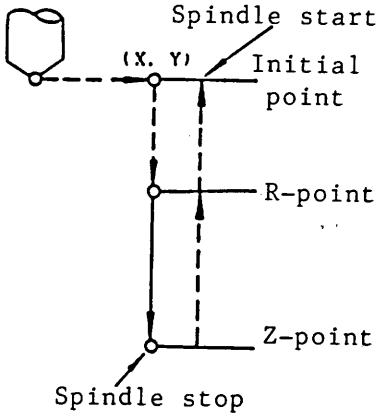
Note-2) It is possible to command a variable pitch by using Q instead of the address I.

Note-3) Since Q is modal, command Q0 when the Q command is given previously.

Note-4) If Q, I, J and K are simultaneously commanded, the variable pitch cut-in regarding Q as the initial value of cut-in amount is proceeded.

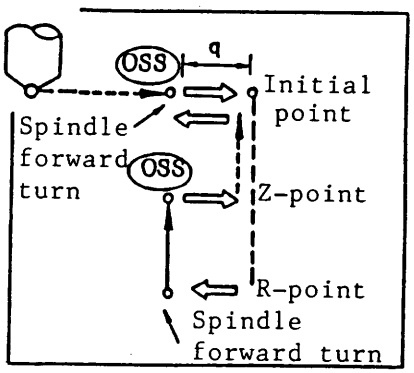
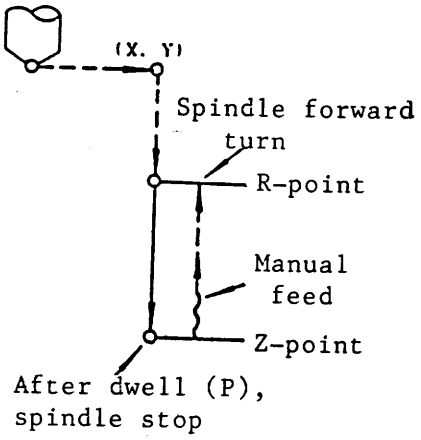
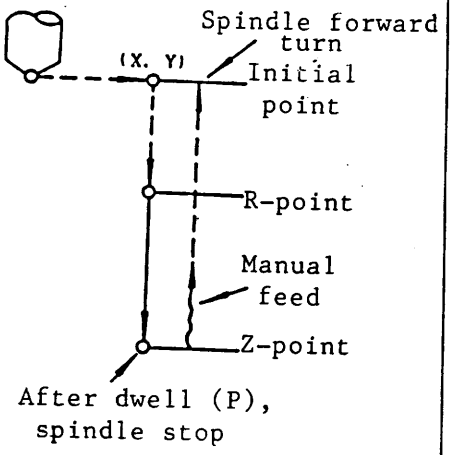
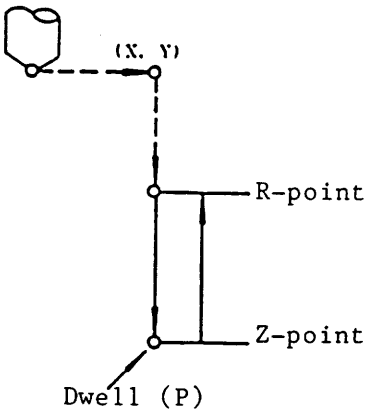
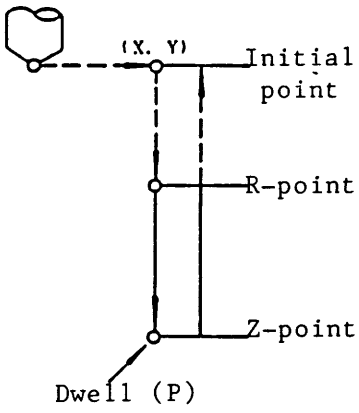
7-7 Canned cycle (G84, G85, G86)

Detail of movement

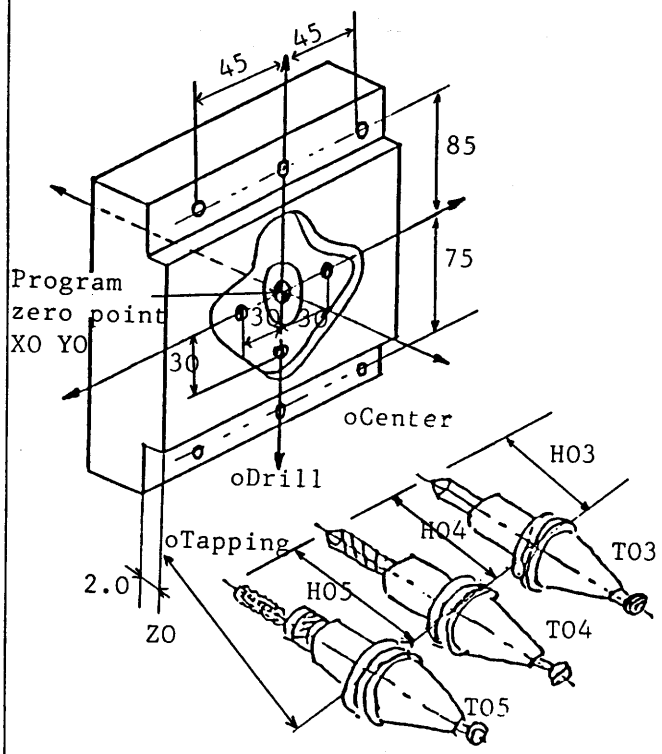
	Case of G99 (R-point return)	Case of G98 (initial point return)
G84 Tapping	G84 X.....Y.....Z.....R.....P.....L.....F..... ; 	
G85 Boring	G85 X.....Y....Z....R.....L.....F..... ; 	
G86 Boring	G86 X.....Y.....Z.....R.....L.....F..... ; 	

7-8 Canned cycle (G87, G88, G89)

Detail of movement

	Case of G99 (R-point return)	Case of G98 (initial point return)
G87 Boring	G87 X.....Y.....Z.....R.....Q.....L.....F..... ; It is not used. Note) The direction releasing the tool after the spindle positioning has been set by the parameter 6212. The tool is shifted to the X + direction for VMII, VK and HG.	
G88 Boring	G88 X.....Y.....Z.....R.....P.....L.....F..... ; 	
G89 Boring	G89 X.....Y....Z.....R.....P.....L.....F..... ; 	

7-9 Example of canned cycle program




```
NOO3 (CENTER TO3 HO3)
TO3MO6
G54G90G00X-45.0Y85.0S800TO4
G43Z50.0HO3
MO3

G99G81R2.0Z-3.9F100
XO
G98X45.0
G99Y-75.0
XO
G98X-45.0
G99X-30.0YO
XOY-30.0
G98X30.0YO
G80MO5
 :
NOO4(10.2 DRILL TO4 HO4)
TO4MO6
G54G90G00X-45.0Y85.0S820TO5
G43Z50.0HO4
MO3
```

① Canned cycle block ----------→ G99G81R2.0Z-20.0F164

② During the canned cycle, if there is hole-machining, indicate the hole position alone. However, make command in combination with the data intended to change by the data of canned cycle (R, Z, F, G99/G98).

```
XO
G98X45.0
G99Y-75.0
XO
 :
G80MO5
 :
 :
 :
```

③ Canned cycle cancel ----------

```
NOO5 (M12 TAP TO5 HO5)
TO5MO6
G54G90G00X-45.0Y85.0S280TO6
G43Z50.0HO5
MO3

G99G84R5.0Z-10.0F490
XO
G98X45.0
 :
 :
 :
```

Calculation method of feed at time of tapping cycle

F △△ = S ○○ rotation x pitch

(Example) M12 x P1.75 tapping
280 rotations

F = 280 x 1.75 = 490 mm/min

* That is, it is F490 .

7-10 Herical cutting G02, G03 (option)

A **Purpose**

By 1-block command, it is effective for spiral oil-groove machining.

Command format

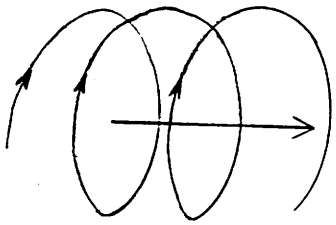
G17 {G02 / G03} X__ Y__ {R__ / I__ J__} Z__ F__ *

G18 {G02 / G03} X__ Z__ {R__ / I__ K__} Y__ F__ *

G19 {G02 / G03} Y__ Z__ {R__ / J__ K__} X__ F__ *

B **Herical interpolation**

The other one axis moving in synchronism with the movement of circular interpolation can be moved with linear interpolation.



C **F-command**

Feed speed along the arc is commanded, thus the speed of linear axis is:

$$F \times \frac{\text{Length of linear axis}}{\text{Length of circular arc}}$$

D **Program**

① G17G03X100.0Y0I0J100.0Z20.0F100 *

or

② G17G03X100.0Y0R100.0Z20.0F100 *

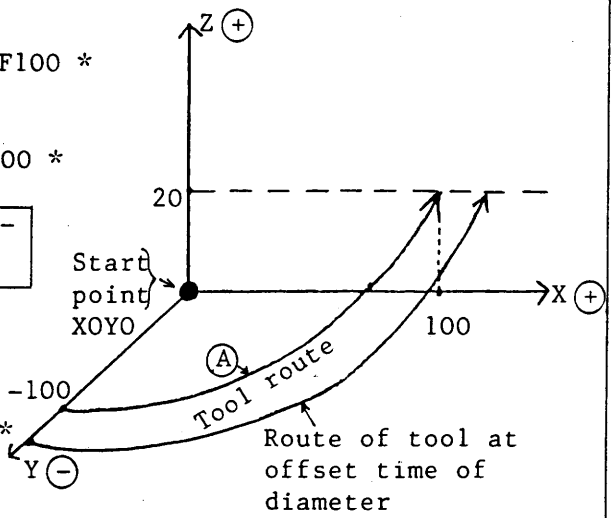
Program in the use of tool-diameter compensation

G90 *

G17G42G01Y-100.0D32F100 *

G03X100.0Y0R100.0Z20.0F100 *

G40G00X0 *



Note 1) Tool-diameter compensation is applied for arc alone.

Note 2) With the block to command helical cutting, it is impossible to command tool-position offset and tool-length compensation.

7-11 Programmable mirror image G511, G501 (option)

A **Purpose**

Program of quadrant-unit can be automatically obtained as mirror-image by G-code.

Command format

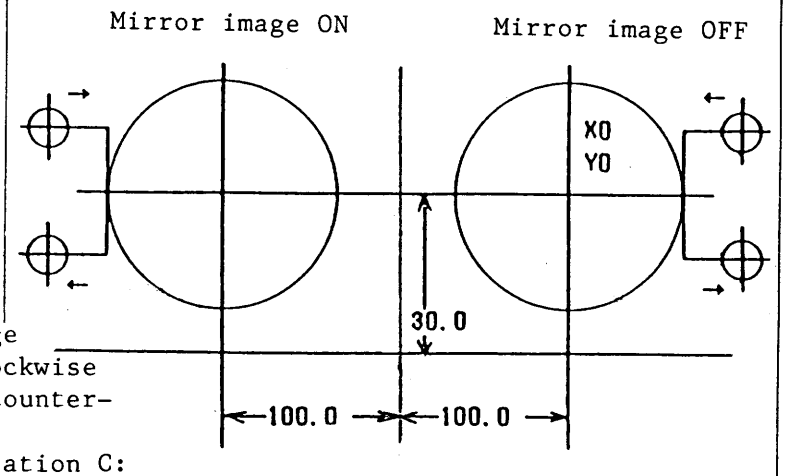
G511 X__Y__Z__ * To set program mirror image

Command value of X, Y, Z sets the mirror to the desired position.

G501 * Cancel

B **Program example**

```
G55G90G00X0Y0    *
*G511X-100.0     *
G00X30.0Y10.0    *
G41G00X16.0D32   *
G01 Y0 F500      *
G02 I-16.0       *
G01 Y-10.0       *
G40 G00 X30.0    *
*G501            *
G91G28X0Y0Z0     *
M30              *
```



In case of 1 axis mirror image

Note 1) Circular command: Clockwise turn is reversed to counter-clockwise turn.

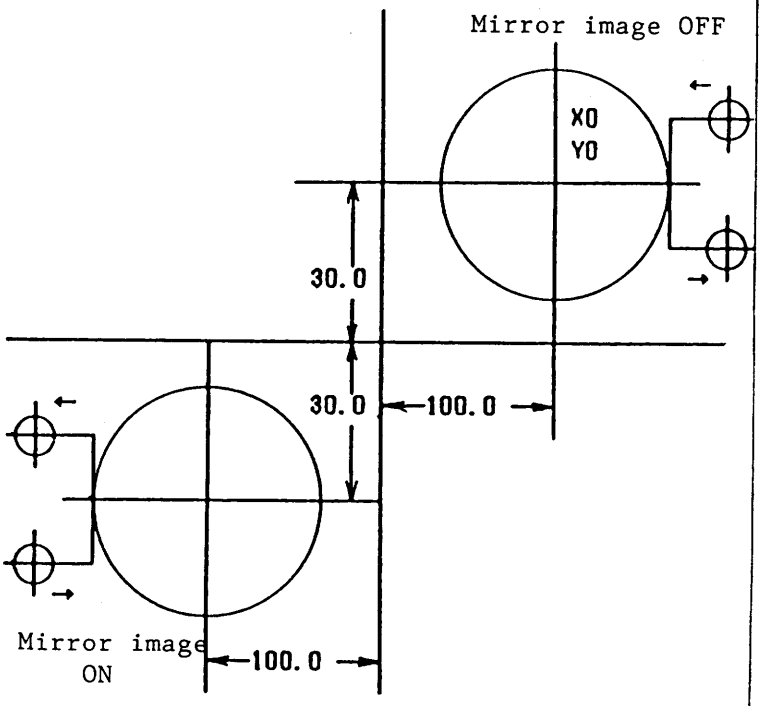
Note 2) Tool diameter compensation C: Right offset is reversed to left offset.

C **Program example**

```
G55G90G00X0Y0      *
* G511X-100.0Y-30.0 *
G00X30.0Y10.0      *
G41G00X16.0D32     *
G01Y0F500          *
G02I-16.0          *
G01Y-10.0          *
G40G00X30.0        *
* G501              *
G91G28X0Y0Z0       *
M30                *
```

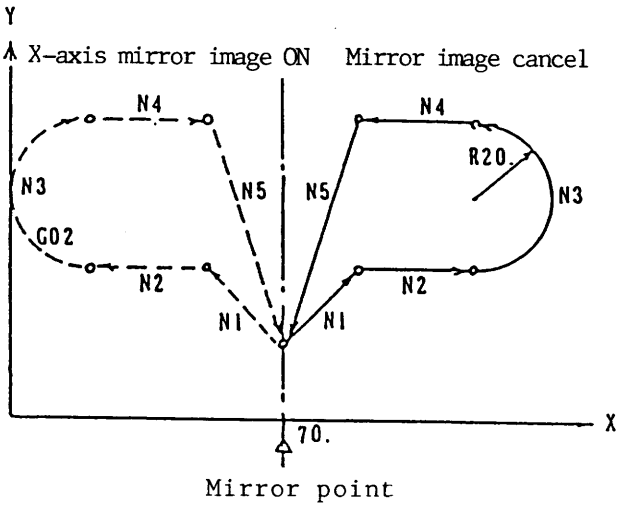
Note 1) When designating 2 axes, the movements become as the right figure.

Note 2) When G501 is commanded, the programmable image is cancelled for all axes.



D Program example

```
G54 G90 G00 X70.0 Y20.0
G511 X70.0
N1 G01 X90.0 Y40.0 F200
N2 X120.0
N3 G03 Y80.0 R20.0
N4 G01 X90.0
N5 X70.0 Y20.0
G501
```



E Precautions

(a) When commanding the coordinate rotation and the mirror image, command as the following order.

```
G511   X_Y_Z_ ;        Programmable mirror image ON
G68 ;                  Coordinate rotation ON
 :
G69 ;                  Coordinate rotation cancel
G501 ;                 Programmable mirror image cancel
```

(b) G511 and G501 shall be commanded in the independent block. When it is commanded by other than the independent, an alarm occurs.

(c) The position display becomes a coordinate value after the program mirror image is applied.

(d) When the programmable image and the setting mirror image are engaged, the setting mirror image is engaged after the programmable mirror image is engaged.

(e) The commands that the programmable mirror image is not engaged are as follows:

(i) Commands such as G28, G30, G53, G27 and G29 etc..
(ii) Shift amount of fine boring (G76) and back boring (G87).

(f) G511 and G501 shall be commanded in the G69 and G50 modes. When they are commanded by G68 and G51, an alarm occurs.

(g) The first travel command after the block that G511 and G501 are commanded shall be commanded in absolute.

(h) When a circular arc is commanded for the first travel command after the block that G511 and G501 are commanded, there are instances where a correct circular arc is not obtained.

7-12 Setting mirror image

The mirror image can be engaged for every axis by the ON/OFF operations on the setting screen or by the external input signal (PC → NC) ON/OFF.

(Note) The program image is engaged through regarding a coordinate value at the time when the mirror image is turned on as the mirror point regardless of the absolute/incremental command of programs.
When the work coordinate system setting, G54~ is commanded, the mirror image is engaged by setting the mirror point as 0.

(1) ON/OFF operations on the setting screen.

(a) Press the [SETTING] key and next press the [C] key.

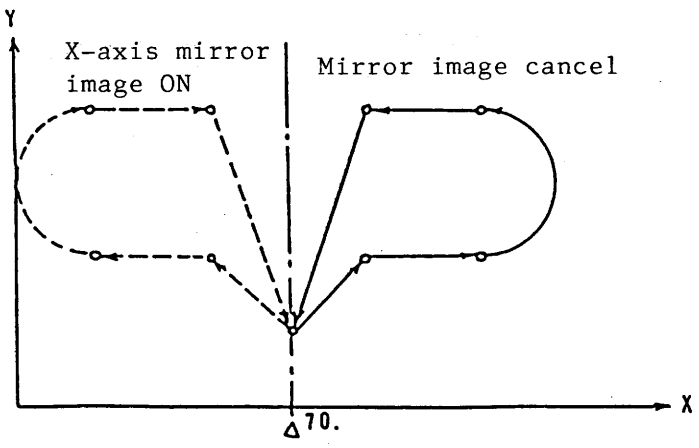
(b) By operating the cursor [↑] [↓] [←] [→] , meet it with effectiveness or ineffectiveness of a designating mirror image axis.

(c) Press the [INPUT] key.

(2) The change-over for the mirror image ON/OFF becomes effective from the next buffering block.

(3) Program example

```
G54 G90 G00 X70. Y20. ;
M42 ;                              X-axis mirror image ON
N1 G01 G90 X90. Y40. F200 ;
N2 X120. ;
N3 G03 Y80. R20. ;
N4 G01 X90. ;
N5 X70. Y20. ;
M37 ;                              Mirror image OFF
```



Mirror point

(4) When the mirror image is engaged only 1 axis of the designated plane.

(a) Circular command (G02, G03) : CW and CCW are reversed.

(b) Tool radius compensation (G41, G42) : The right side offset and the left side offset are reversed.

(5) Precaution

(a) The position display becomes the coordinate value after the setting mirror image is engaged.

(b) When the programmable image and the setting mirror image are engaged, the setting mirror image is engaged after the programmable image is engaged.

(c) The commands that the setting mirror image is not engaged are as follows:

(i) Commands such as G28, G30, G53, G27 and G29 etc..
(ii) Shift amount of fine boring (G76) and back boring (G87).

(d) Arrange so that the setting mirror image ON/OFF position may become the same position.

(e) In the state of reset, the setting mirror image becomes OFF.

(f) When the setting mirror image ON/OFF is the first axial command shall be commanded in absolute.

(g) When the setting mirror image is shifted from OFF to ON, there are instances where a correct circular arc is not obtained when a circular arc is commanded in the first axial command.

7-13 Direct tapping — Special G841, G741

A Purpose

By this synchronizing method of spindle rotation and Z-axis feed, high-speed/high accurate tapping can be done.
Conventional tapper is not needed.

X : } Tapping hole positional
Y : } coordinate
Z : Hole-bottom position
R : Cutting start point
P : Dwell time at R-point
F : Feed speed
L : Repeating times

B Command format

(Same format as that of conventional G84, G74.)

```
S____        * ..... Rotation command

                     Note) The gear shift shall be performed
                           in the previous block.
                                 (M03 command is NG)

G841    X__ Y__ Z__ R__ P__ F__ L__ *
X___ Y___    *                  └---------- Spindle-rotary speed
X___ Y___    *                              x pitch
G80          *                              (e.g. S2000 x P1.0 =
                                             F2000)
                          └-------- Start and stop of spindle
                                    rotation, R-point
```

C Actual program

(Al material M6 x P1.0 tapping case)

```
G54G90G00X50.0Y100.0S2000T05  *
G43Z30.0H04                   *
G98G841Z-20.0R5.0F2000        *
X75.0Y150.0                   *
X100.0Y150.0                  *
G80                           * Cancel(direct and tapping mode)
```

Note)-1 Both G841 tapping cycle and G741 counter tapping cycle are the same as the conventional format.

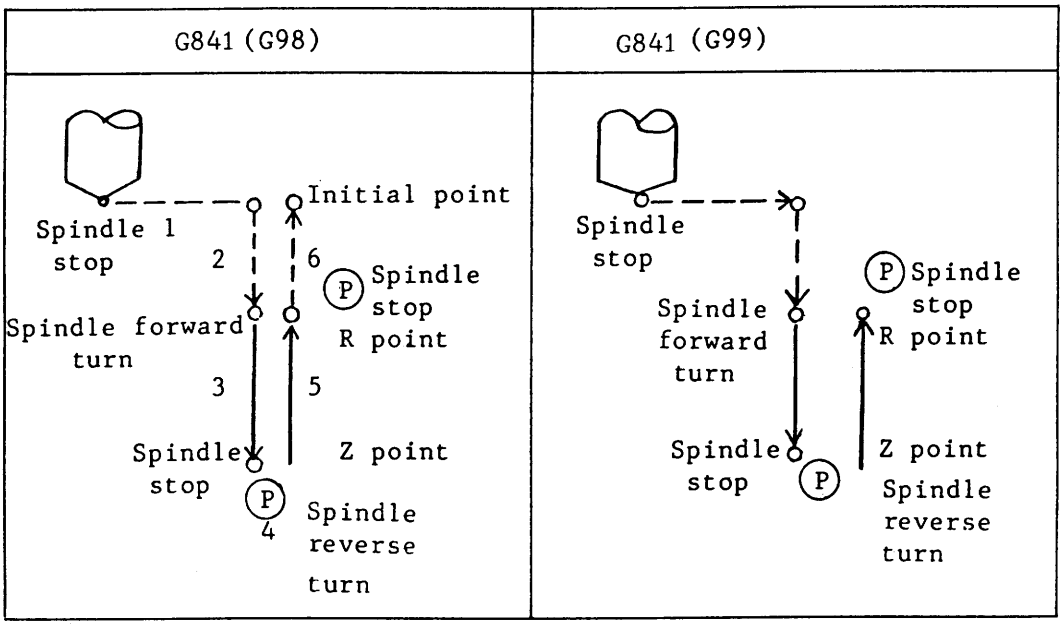
Note)-2 Command within max. rotation speed limit.

Note)-3 When SΔΔΔis performed after G841 and G741 are commanded, the the rotation of SΔΔΔis executed as the high speed range selected before remains.

7-14 Direct tapping Special G841, G741

D Explanation for operation (tapping cycle)

- - - - - - - Rapid traverse
——————— Z-axis (spindle synchro-feed)
(P) Dwell

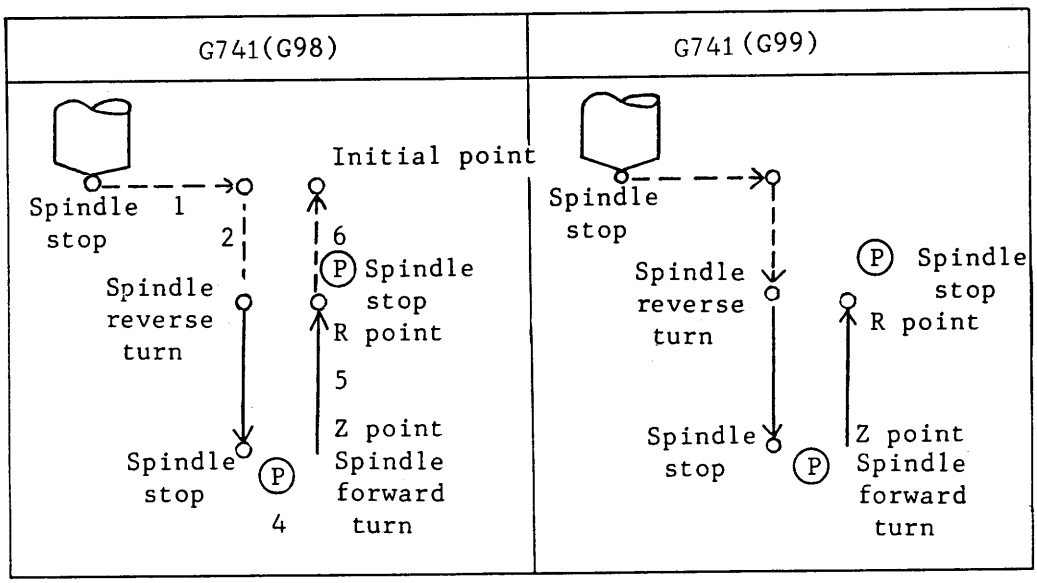


Note 1) During synchro-feeding of spindle/Z-axis, feeding speed and spindle override shall be regarded as 100%.

Note 2) Tapping cycle with single block causes the stopping at end-point of operation 1, 2 and 6.

Note 3) At program STOP in between operations 3~5, STOP - lamp lights, however, it stops after the end of operation 6.

E Explanation for operation (reverse tapping cycle)



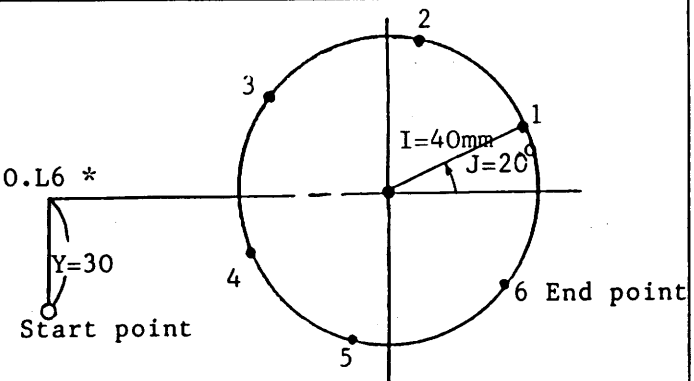
Note 1) The caution is the same as that of G841(tapping cycle).

7-15 Boring pattern cycle G70, G71, G72, G77 (option)

① G70 : Bolt-hole cycle

G70 X__Y__I__J__L__ *

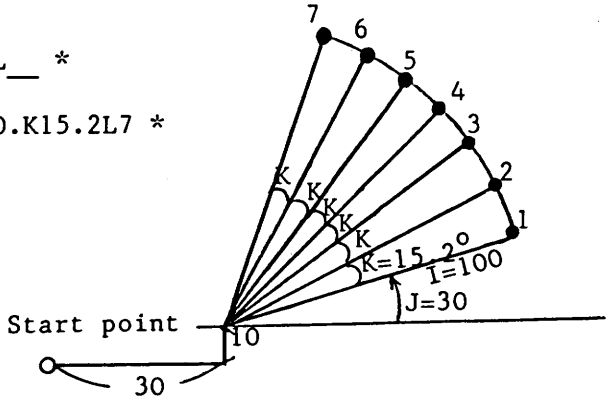
(Example) G70 X90.Y30.I40.J20.L6 *



② G71 : Arc cycle

(Example) G71 X__Y__I__J__K__L__ *

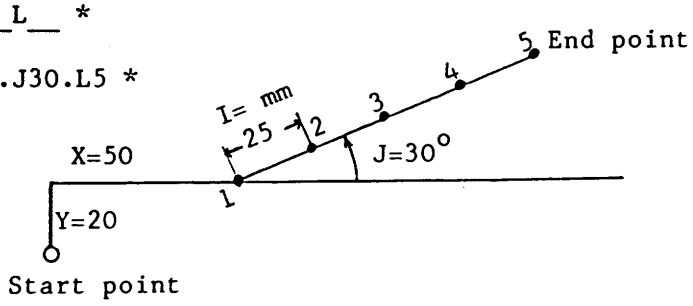
G71 X30.Y10.I100.J30.K15.2L7 *



③ G72 : Line at angle cycle

(Example) G72 X__Y__I__J__L__ *

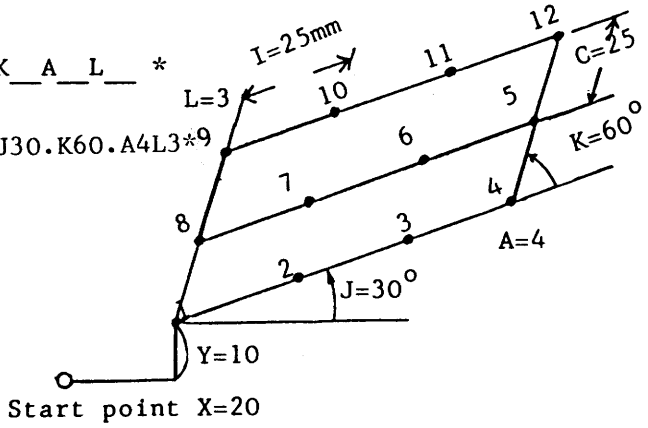
G72 X50.Y20.I25.J30.L5 *



④ G77 : Grid cycle

(Example) G77 X__Y__I__C__J__K__A__L__ *

G77 X20.Y10.I25.C25.J30.K60.A4L3*9



7. G-function (canned cycle)

Note 1) G70, G71, G72, G77 are non-modal G-codes.

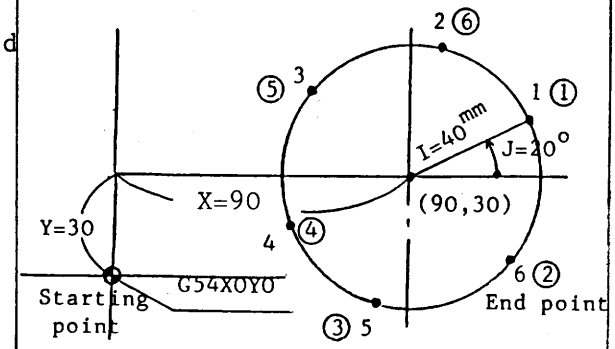
Note 2) Be sure to command G70, G71, G72, G77 by canned-cycle mode.

Note 3) After machining of G70, G71, G72, G77, the next block should be cancelled by G80 exactly.

7-16 Bolt-hole cycle G70 (option)

A **Purpose**

In case of equally distributed drilling on the circumference, this function decides the position by automatic calculation at rectangular coordinate value with the radius and angle.



B **Command format**

G70 X___Y____I___J___L___ *

G70: Bolt-hole cycle

X,Y: Make description based on circle-center coordinate, G90, 91.

I : Radius of circle. It should surely be positive number. It can be indicated by minimum setting.

J : Setting of angle at initially positioned point. 0.001 degree unit CCW turn is positive.

L : Circle-dividing number. Positive is CCW turn. Negative is CW-turn.

C **Actula program**

```
G54G90G00X0Y0S1000T15        * Transfer to the start point
G43Z30.0H14                  * Z-axis approach
M03                          * Spindle forward turn
G98G81Z-30.0R3.0F150L0       * Drill-canned cycle command
                               L0 no-move.
G70X90.0Y30.0I40.0J20.0L6    * Bolt-hole cycle command, CCW-turn
G80M05                       * Canned cycle cancel
M98P98                       * Transfer to sub-program 098
```

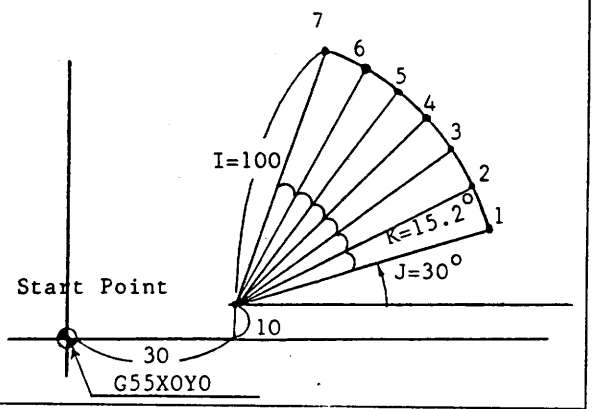
Note 1) Be sure to set L0 for the canned cycle command.

Note 2) When "L" at bolt-hole cycle G70 is minus (-), machining sequence is ① ~ ⑥.

7-17 Arc cycle G71 (option)

A **Purpose**

It is used for machining of drilling-line arranged in equal interval on the arc.



B **Command format**

G71 X___ Y___ I___ J___ K___ L___ *

G71 : Arc cycle

X,Y : Make description based on circle-center coordinate, G90,91.

I : Arc radius. It should surely be positive number. It can be indicated by minimum setting.

J : Setting of angle at initially positioned point. 0.001 degree unit CCW turn is positive.

K : Angle-interval. 0.001 degree unit. Positive (plus) means CCW-turn.

L : Positioning frequency setting. It is surely positive number.

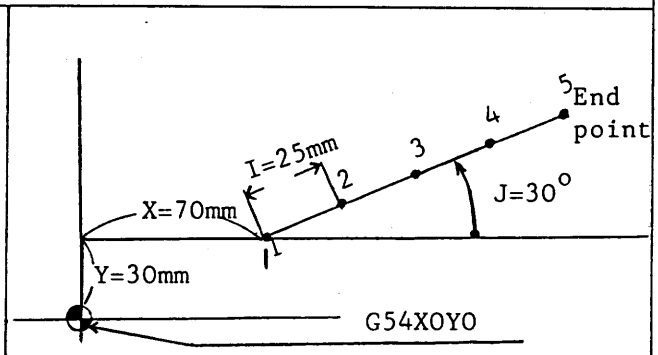
C **Actual program**

```
G55G90G00X0Y0S1500T20      * Transfer to the start point
G43Z30.0H19                * Z-axis approach
M03                        * Spindle forward turn
G98G81Z-25.0R3.0F200L0     * Establishment of canned cycle mode.
                             L0, no movement.
G71X30.0Y10.0I100.0J30.0K15.2L7 * Arc-cycle command, CCW-turn
G80M05                     * Canned cycle cancel
M98P98                     * Transfer to sub-program 098.
```

7-18 Line at angle cycle G72 (option)

A **Purpose**

It is used for machining the arranged holes in equal interval on the declined straight line.



B **Command format**

G72 X___Y___I___J___L___ *

G72 : Line at angle cycle

X,Y : Coordinate of start point (machining start point)

I : Make setting with positive number. Interval-setting. When "I" is negative, make positioning to the symmetric direction on the point with the center of start point. Indication can be made by minimum setting unit.

J : Setting of angle. 0.001 degree unit. CCW-turn is positive (plus).

L : Positioning frequency setting. It is surely positive number.

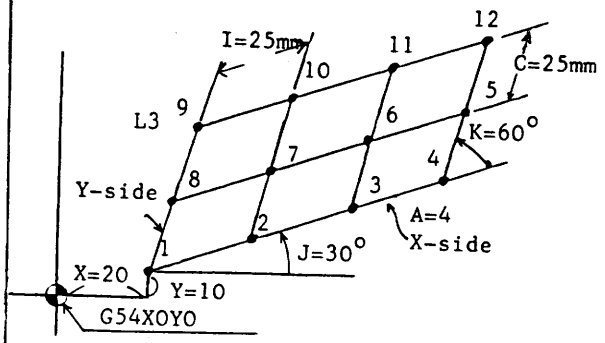
C **Actual program**

```
G54G90G00XOYOS1000T20      * Positioning to work-coordinate XOYO.
G43Z30.0H19                * Z-axis approach
M03                        * Spindle forward turn
G98G81Z-25.0R3.0F200L0     * Establishment of canned cycle mode L0, no movement.
G72X70.0Y30.0I25.0J30.0L5  * Line at angle cycle command No.1 ~ 5 boring.
G80M05                     * Canned cycle cancel
M98P98                     * Transfer to sub-program 098.
```

7-19 Grid cycle G77 (option)

A **Purpose**

It is used for machining of arranged holes in equal interval on lattice.



B **Command format**

G77X___Y___I___C___J___K___A___L___ *

G77 : Grid cycle

X,Y : Coordinate of initial hole position

I : Setting of interval in X-direction

C : Setting of interval in Y-direction

J : Angle formed between X-axe and X-side (0.001 degree unit. CCW-turn is positive.)

K : Angle formed between X-side and Y-side (0.001 degree unit. CCW-turn is positive.)

A : Quantity of holes on X-side (setting with positive number)

L : Quantity of holes on Y-side (setting with positive number)

Whereas, A x L shall not exceed 32767.

C **Actual program**

G54G90G00X0Y0S1000T10	*	Positioning to work-coordinate X0Y0.
G43Z30.0H09	*	Z-axis approach
M03	*	Spindle forward turn
G98G81Z-25.0R3.0F200L0	*	Establishment of canned cycle mode it should be L0.
G77X20.0Y10.0I25.0C25.0J30.0K60.0A4L3	*	Grid cycle command 1~12 boring.
G80M05	*	Canned cycle cancel
M98P98	*	Transfer to sub-program 098.

7-20 True circle cutting (G302 ~ G305)

A series of operation cutting the inner side or outer side of the true circle can be commanded by one block.

(1) G code

G302 : True circle cutting inner side CW (clockwise)

G303 : True circle cutting inner side CCW (counterclockwise)

G304 : True circle cutting outer side CW (clockwise)

G305 : True circle cutting outer side CCW (counterclockwise)

(2) Command form

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

$$\begin{Bmatrix} G302 \\ G303 \end{Bmatrix} I\_ \begin{Bmatrix} R\_ \\ J\_ \end{Bmatrix} U\_Q\_L\_D\_F\_;$$

However, I : Radius of finishing circle

I + is of approach for the plus direction

I - is of approach for the minus direction

R : R command for the high speed feed range

J : J command for the high speed feed range

U : Radius of the final finishing circle for spiral true circle cutting

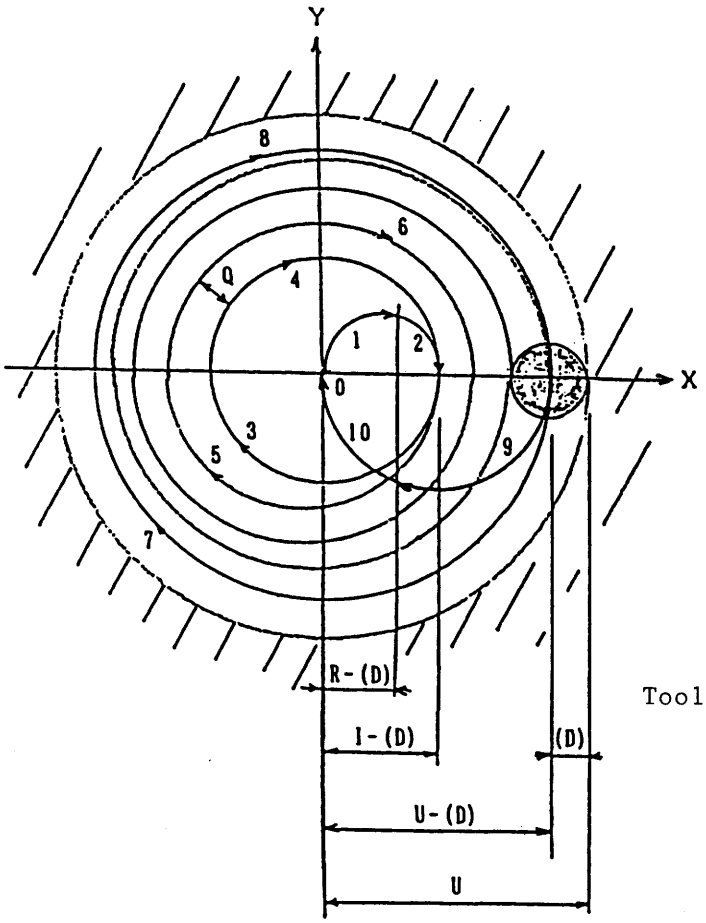
Q : Circular arc increment for spiral true circle cutting

L : Repeated number of times at the true circle section

D : Tool radius compensation No.

F : Cutting feed rate

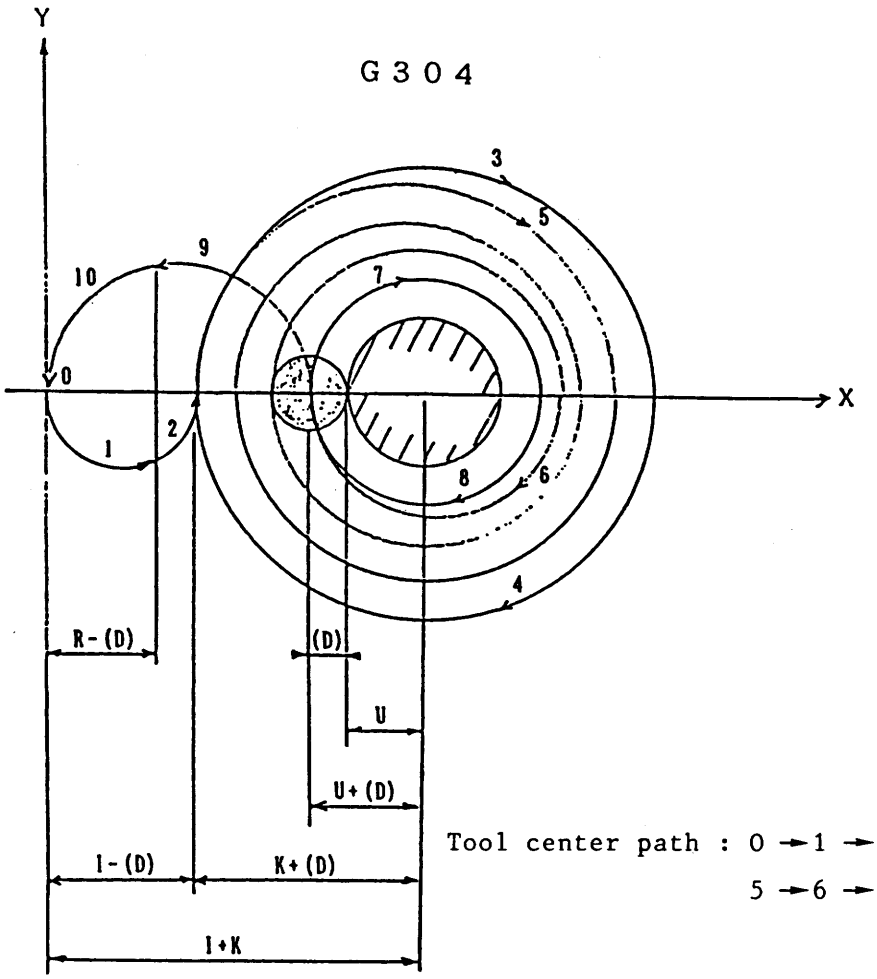
G 3 0 2



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

(b) True circle cutting OD (G304, G305)

$$\left\{\begin{matrix}G304\\G305\end{matrix}\right\} I\_ \left\{\begin{matrix}R\_\\J\_\end{matrix}\right\} K\_U\_Q\_L\_D\_F\_;$$



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

However, I : Diameter of approaching circle

I + is of approach for the plus direction

I - is of approach for the minus direction

R : R command for the high speed feed range

J : J commend for the high speed feed range

K : Radius of the finishing circle

U : Radius of the final finishing circle for spiral true circle cutting

Q : Circular arc increment for spiral true circle cutting

L : Repeated number of times at the true circle section

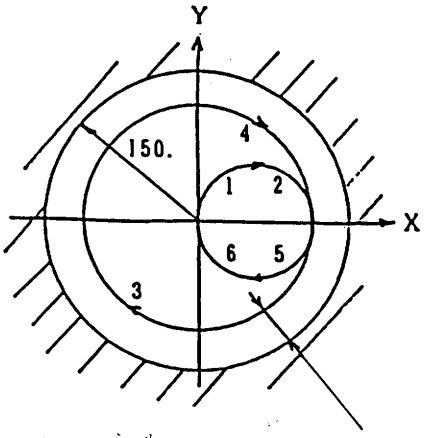
D : Tool radius compensation No.

F : Cutting feed rate

(3) Program example

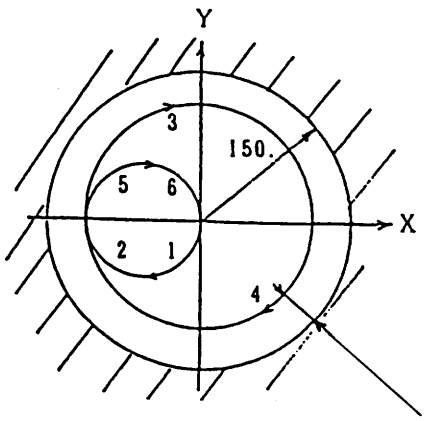
(a) Basic form

G302 I50. D10 F500;



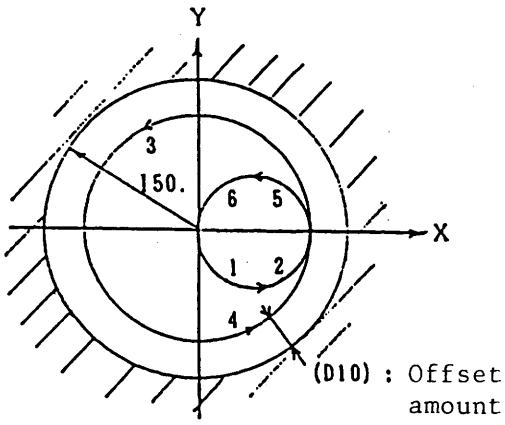
(D10) : Offset amount

G302 I-50. D10 F500;

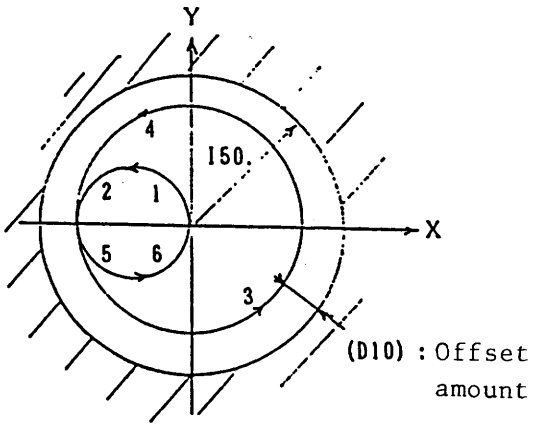


(D10) : Offset amount

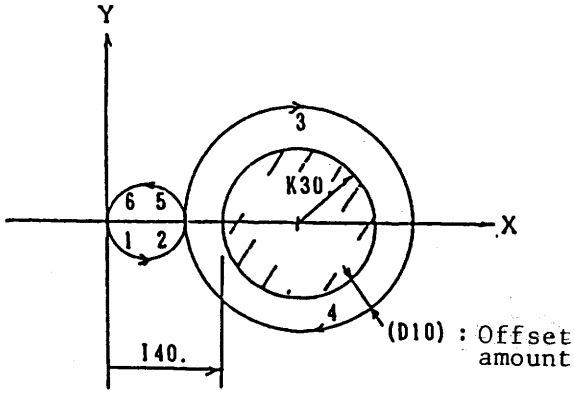
G303 I50. D10 F500;



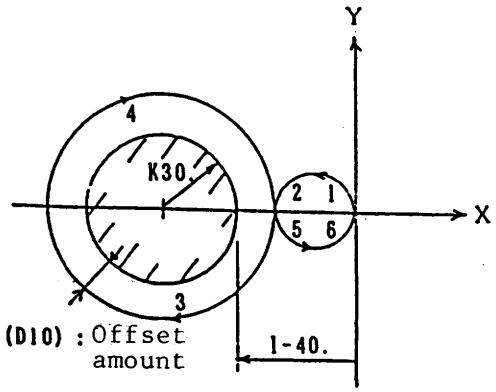
G303 I-50. D10 F500;



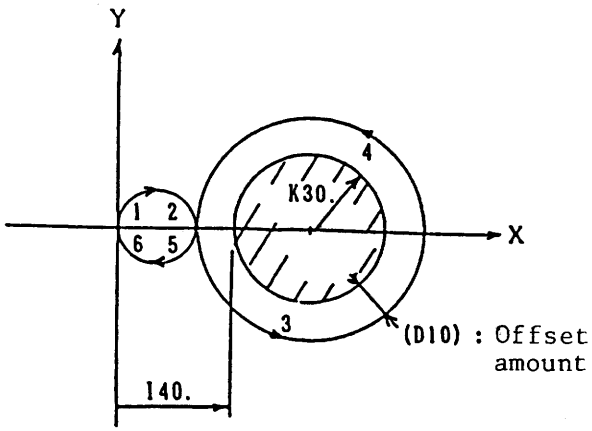
G304 I40. K30. D10 F500;



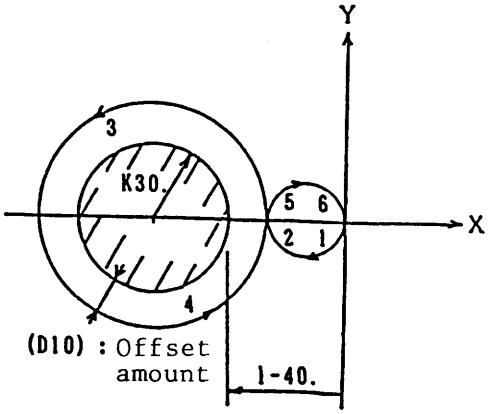
G304 I-40. K30. D10 F500;



G305 I40. K30. D10 F500;

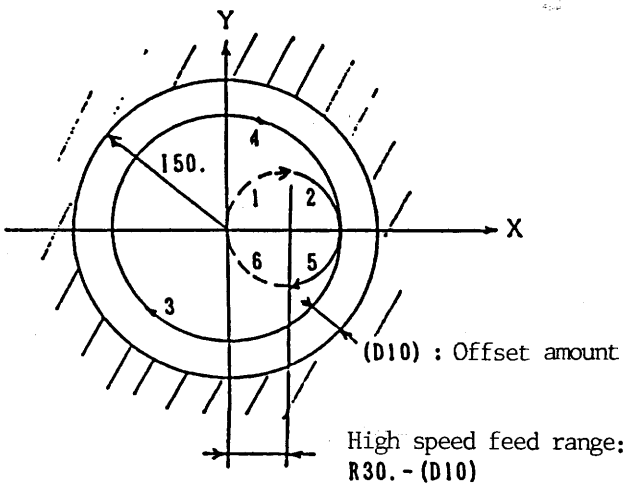


G305 I-40. K30. D10 F500;

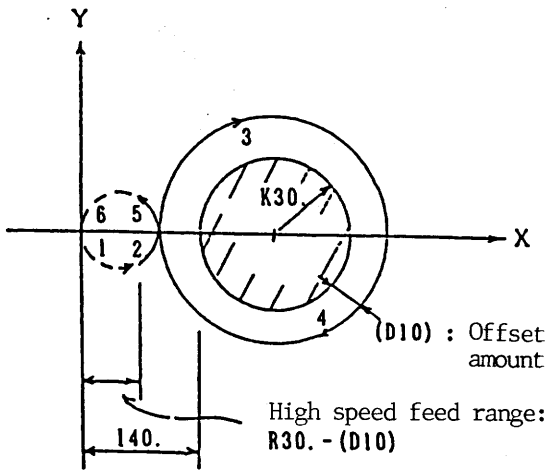


(b) R command for the high speed feed range

G302 I50. R30. D10 F500;

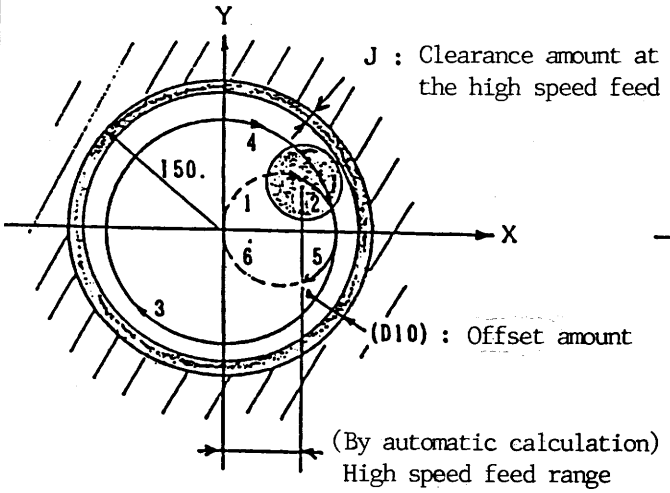


G304 I40. R30. K30. D10 F500;

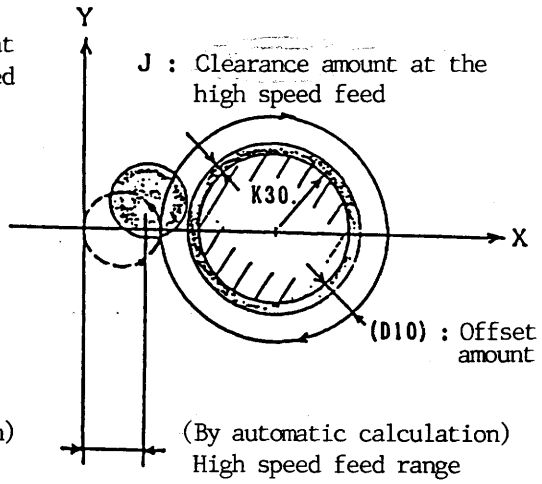


(c) J command for the high speed feed range

G302 I50. J5. D10 F500;

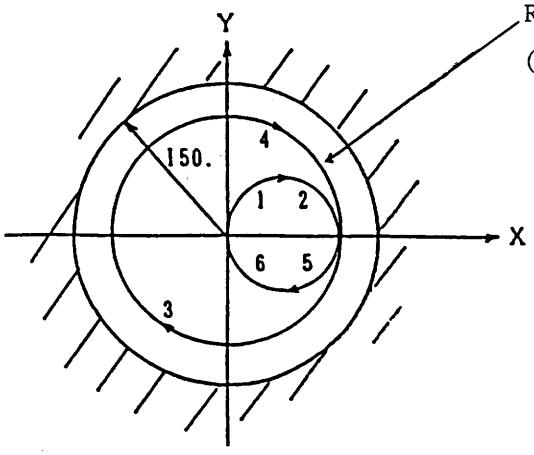


G304 I40. J5. K30. D10 F500;



(d) Repeating designation at the true circle section (L)

G302 I50. L2 D10 F500;

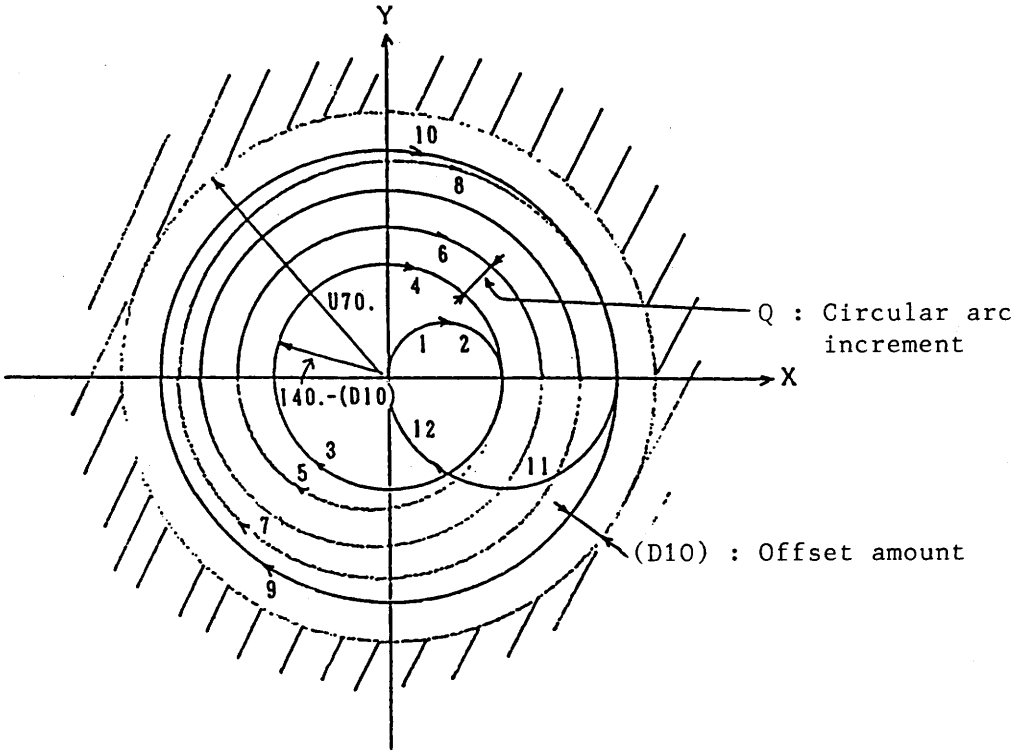


(Note)

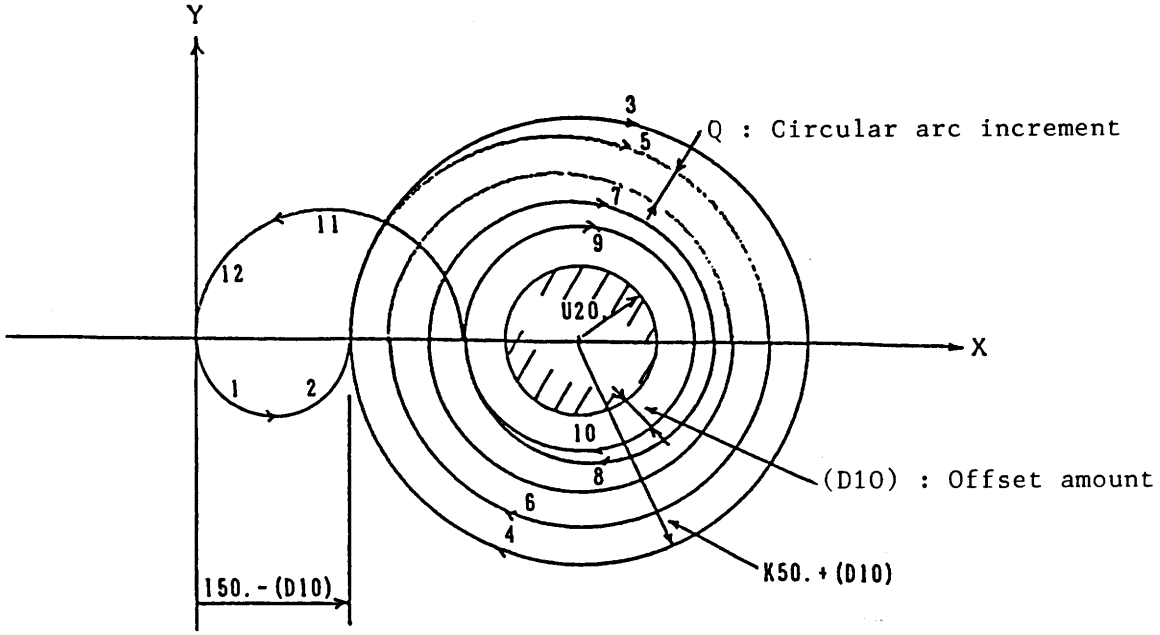
In case of spiral true circle cutting, the final finishing circle shall be repeated.

(e) Spiral true circle cutting designation

G302 I40. U70. Q10. D10 F200;



G304 I50. K50. U20. Q10. D10 F200;



(4) Precaution

(a) Give the G302 ~ G305 commands in the state of tool radius compensation cancel.

(b) The G302 ~ G305 commands are of non-modal G dodes.
Address numerical values other than D and F commanded in the same block are effective only for commanded blocks.

(c) The numerical values of R, J, K, U and Q shall be always commanded by position values.

(d) When R and J are omitted, high speed feed ranges become ineffective.

(e) When U and Q are omitted, spiral true circle cuttings become ineffective.

(f) When D and F are omitted, the D and F already commanded becomes effective.

(g) When L is omitted, the repeated number of a true circle section becomes always 1.

(h) When R and J are commanded in the same block, J has priority.

(i) When the following commands are given, an alarm occurs.

(i) When (radius of finished circle (I) - offset amount) is either 0 or a negative value at G302 and G303.

(ii) When high speed feed ranges are not obtained at the R designation in the high speed feed range.

(iii) When high speed feed ranges are not obtained by automatic calculation at the J designation in the high speed feed range.

(iv) When (radius (U) of final finished circle - offset amount) is either 0 or a negative value in the spiral cutting of G302 and G303 commands.

(v) When (diameter (I) of approach - offset amount) is either 0 or negative value at G302 and G303 commands.

(vi) When the approaching circle doesn't exist through changing an offset amount.

(j) The XY plane can be selected regardless of G17 ~ G19 commands by parameter setting.

(k) When a negative value is set to an offset amount, it has been altered to a positive value.

7-21 Square side frame outer cutting (G322, G323)

A series of operation of square side frame outer cutting can be command in one block.

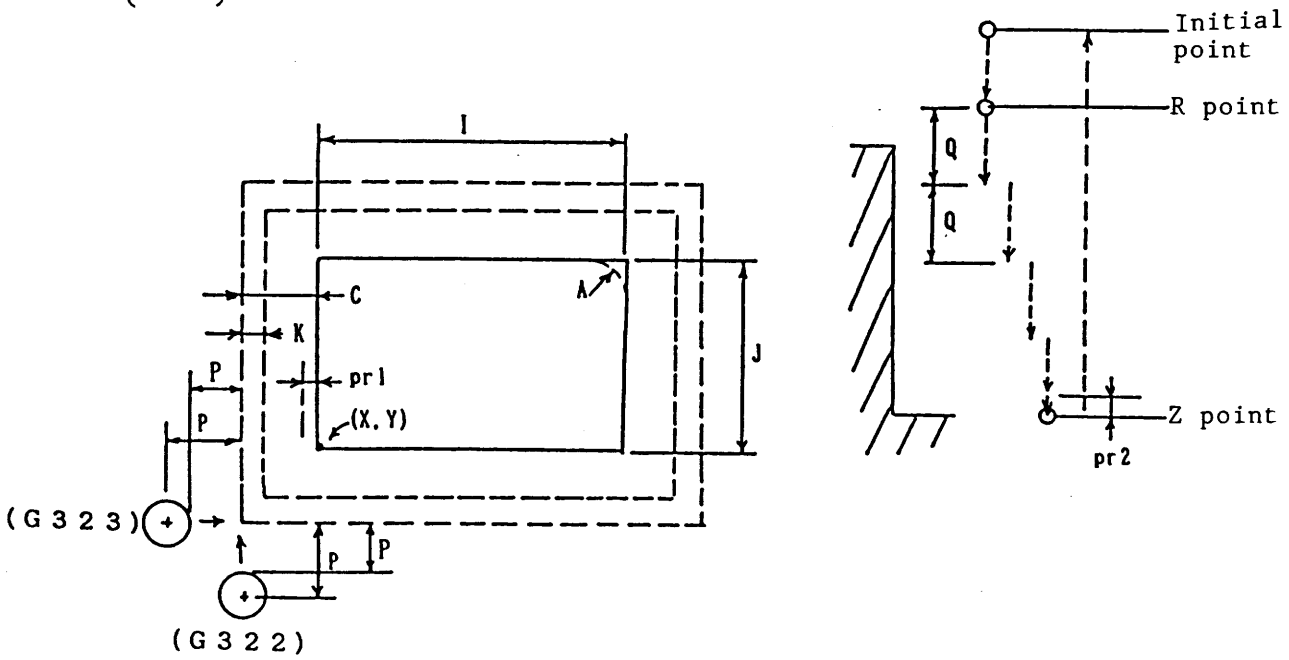
(1) G code

G322 : Square side frame outer cutting CW (clockwise)

G323 : Square side frame outer cutting CCW (counterclockwise)

(2) Command form

$\left\{\begin{matrix}G322\\G323\end{matrix}\right\}$ X_Y_Z_R_Q_I_J_K_P_A_C_D_F;



(X, Y) : Datum point of X and Y axes

Z : Z point

R : R point

Q : Cutting depth per one cutting for Z-axis

I : X-axis length of the quadrangle to be finished

J : Y-axis length of the quadrangle to be finished

K : Cutting width per one cutting for X and Y axes

P : Approaching amount

A : Radius of the corner R

C : Cutting allowance

D : Tool offset No.

F : Cutting feed rate

pr1: Finishing allowance (parameter setting)

pr2: Clearance amount (parameter setting)

(Note 1) When cutting the cutting allowance, an override can be applied to the cutting feed rate by the ratio set by a patameter.

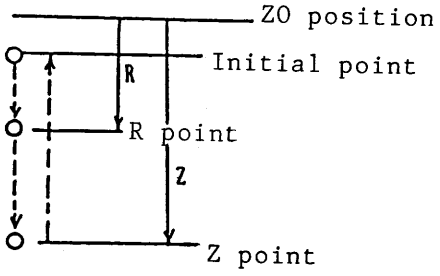
(3) Initial point

This is machining start point for G322 and G323 commands.
When a series of operation is finished, all the X, Y and Z axes return to their atart point.

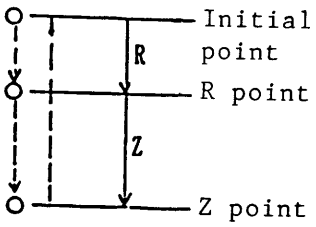
(4) R point and Z point

The R point and the Z point become as follows by G90 and G91 commands.

[G90]

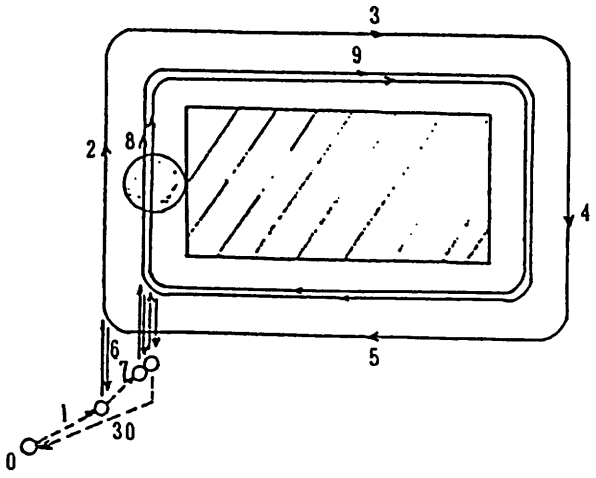


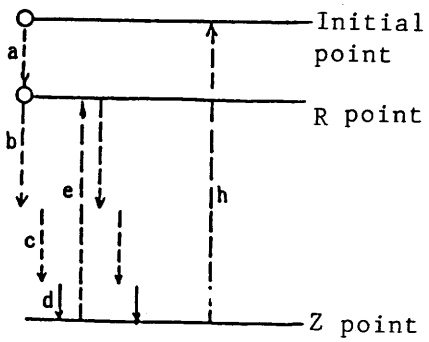
[G91]



(5) Program example

```
G17;
G90 G322 X50. Y-100. Z-50. R-10. Q20. I80.
J40. K8. P30. A2. C15. D10 F200;
```





----→ Rapide traverse

——→ Cutting feed

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

(6) Precautions

(a) Tool radius compensation is applied regardless of the tool radius compensation (G41 and G42) by G322 and G323. Accordingly, command them in the state that the tool radius compensation is cancelled.

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

(c) When A is omitted in the G322 and G323 block, the corner R becomes ineffective.

(d) When the numerical value of K is a negative in the G322 and G323 blocks, finishining becomes ineffective, and when the numerical value of Q is negative, the clearance value becomes ineffective.

(e) All the numerical values such as the address I, J, P, A and C shall be commanded by positive values.
When negative values are commanded, an alarm occurs.

(f) When D and F are omitted, the D and F already commanded become effective.

7-22 Coordinate rotation (G68, G69)

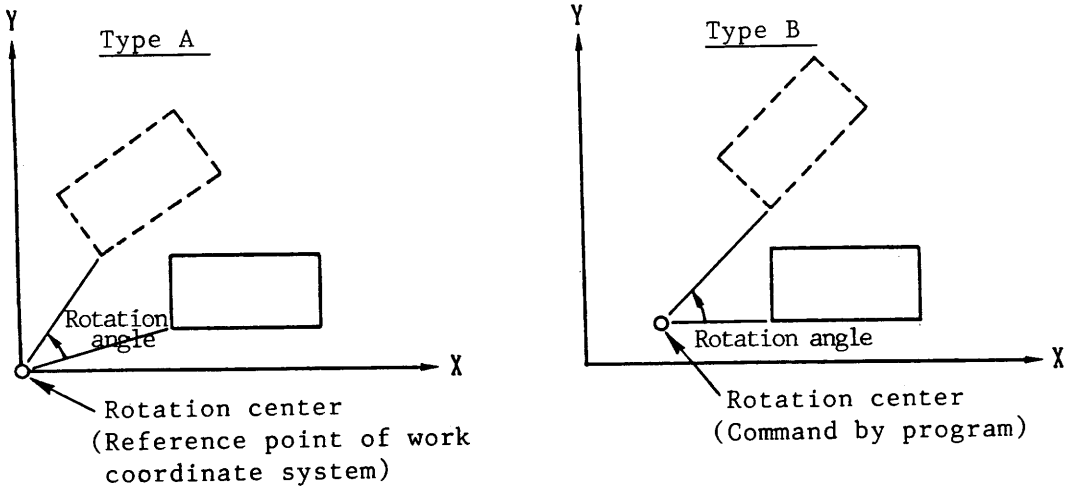
By this command the shape commanded with a machining program can be rotated at the angle designated.

There are 2 sorts of coordinate rotation as follows:

(a) When the rotation center is regarded as the reference point of the work coordinate system. ..... Type A

(b) When the rotation center is commanded by a program. ..... Type B

The coordinate rotation is engaged by the type A after engaged by the type B. Besides, it can be engaged by the type A and the type B independently.



(1) G code

G68 : Coordinate rotation ON

G69 : Coordinate rotation cancel

(2) Command form

G68 α_β_R_; Coordinate rotation type B ON

G68; Coordinate rotation type A ON

(All α_β_R_ are omitted)

G69; Coordinate rotation cancel

However, α, β: Coordinate value of the rotation center

Command 2 axes of the plane designated with G17, G18 and G19 by an absolute value.

R : Rotation angle
The counterclockwise direction is plus
The unit is -360.000 ~ +360.000
or -360.00000 ~ +360.00000

Rotation angle of the type A : R value on the screen of the work coordinate system offset.

(a) When the type A and the type B of the coordinate rotation are used.

```
G68 α_β_R_;          Coordinate ratation type B ON
G68;                 Coordinate rotation type A ON
 :
G69;                 Coordinate rotation cancel
```

(b) When the type A of the coordinate rotation are used.

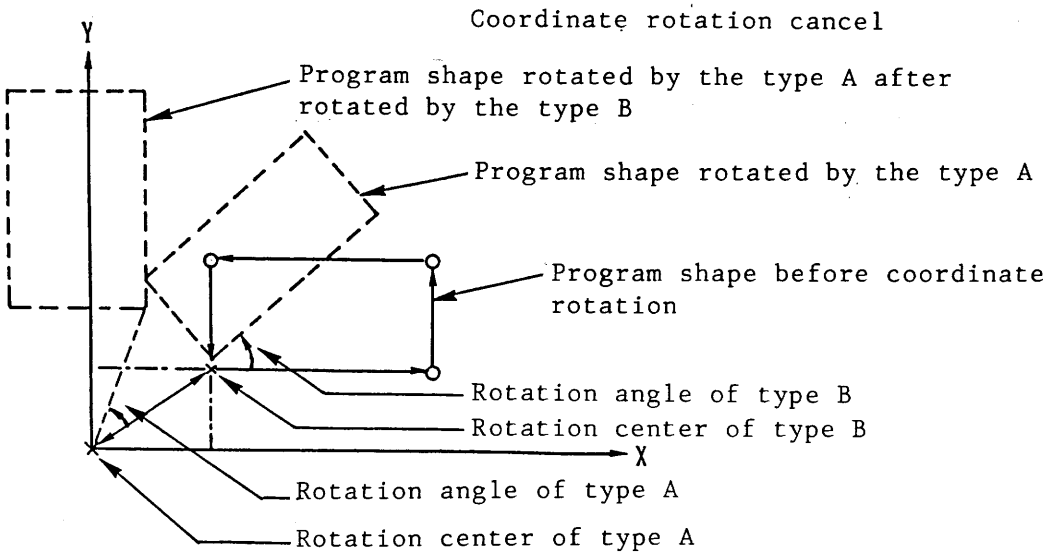
```
G68;                 Coordinate rotation type A ON
 :
G69;                 Coordinate rotation cancel
```

(c) When the type B of the coordinate rotation are used.

```
G68 α_β_R_;          Coordinate rotation type B ON
 :
G69;                 Coordinate rotation cancel
```

(2) Program example

```
G17 G54 G90 G00 X0 Y0;
G68 X30. Y20. R45.;      Coordinate rotation type B ON
G68;                     Coordinate rotation type A ON
N1 G01 G91 X30. Y20. F200;
N2 X60.;
N3 Y30.;
N4 X-60.;
N5 Y-30.;
   G69 X-30. Y20.;
```



(4) When the coordinate rotation is used together with tool radius, scaling and compensation, programmable mirror image etc., command in the order as below.

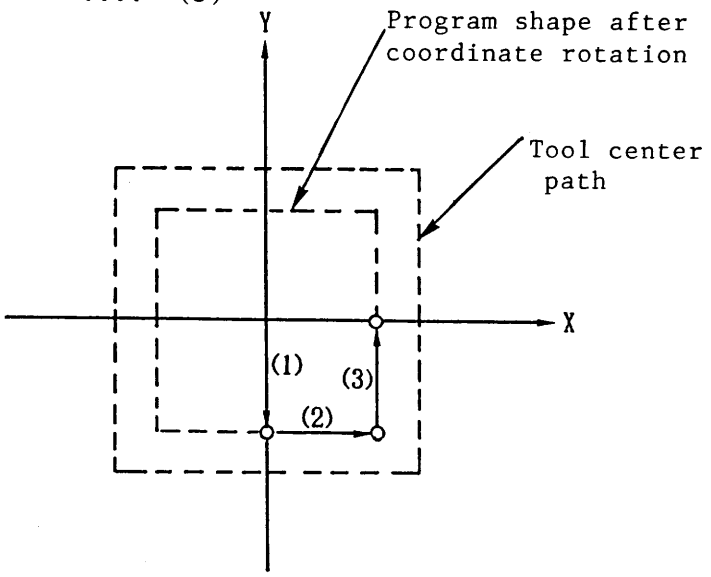
G511...;	Programmable mirror image ON
G51...;	Scaling ON
G68 α_β_R_;	Coordinate rotation type B ON
G68;	Coordinate rotation type A ON
G41...;	Tool radius compensation ON
:	
G40;	Tool radius compensation cancel
G69;	Coordinate rotation cancel
G50;	Scaling cancel
G501...;	Programmable mirror image cancel

(5) When commanding repeatedly

Resister a program as a sub-program by setting a parameter and that program can be called during changing the angle.

```
G17 G54 G90 G00 X0 Y0;
G68 X0 Y0 R0;
M98 P100;
M98 P200 L3;
G00 G90 X0 Y0;
G69;
```

```
O100 G90 G01 G42 X0 Y-10. D10; .... (1)
X10.;                          .... (2)
Y0.;                           .... (3)
G40;
M99;

O200 G91 G68 X0 Y0 R90.;
G90 M98 P100;
M99
```



(6) Precautions

(a) G68 shall be commanded in the independent block.
When it is commanded by other than the independent block, an alarm occurs.

(b) When the plane is changed by commanding the plane selection (G17, G18 and G19) during G68 mode, an alarm occurs.

(c) The first travel command after the block that G68 and G69 were commanded shall be commanded in absolute.

(d) The commands that the coordinate rotation is not engaged are as follows.

(i) The shift amount for fine boring (G76) and back boring (G87).

(ii) G28, G29, G30, G53, G31, G27

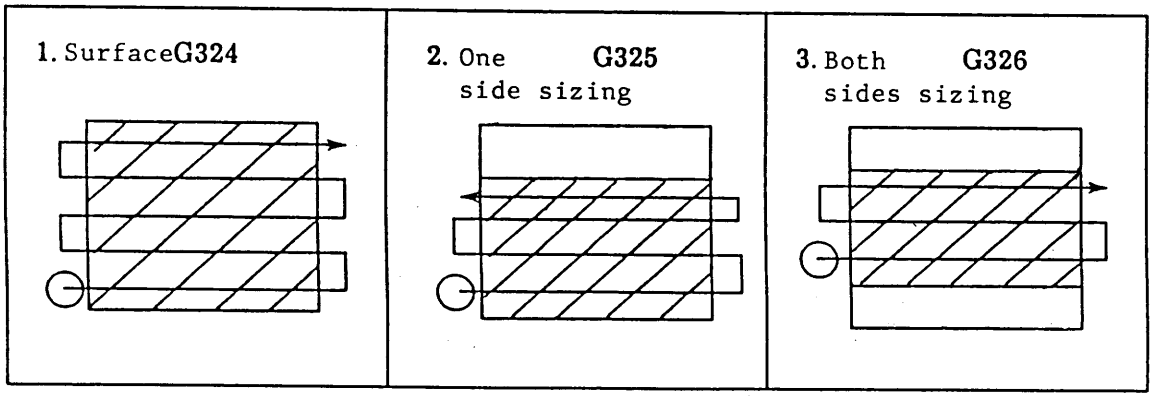
(e) When G92 and G52 are commanded, the rotation center of the coordinate rotation type A becomes (0, 0).

7-23 Surface cutting cycle (G324, G325, G326)

The surface cutting canned cycle consists of 3 kinds of cycles;
Square surface cutting (G324)
Square surface one side sizing (G325)
Square surface both sides sizing (G326)

It is convenient canned cycle to perform the surface cutting and groove cutting by using a face milling cutter or end milling cutter.

In these cycles, assume the coordinate value designated by X and Y as the start point, and assume the range of the length (I) in X axis direction and the length (J) in Y axis direction as the work surface, and a cycle cutting is executed up to the coordinate value of the finishing surface by the cut-in amount (Q) of one surface.



(1) Square surface cutting (G324)

Purpose : This is a function that machines square type plane surfaces in the lamp by unidirectional or bidirectional cutting.

(a) Commanded form

G324 X1_Y2_Z3_R_I_J_K_Q_P_C_D_E_U_F_;

G324 : Square surface cutting

X1,Y2: Start point coordinate value of the plane, enterd based on G90 and G91.

Z3 : 3rd axis coordinate value of the finishing face, entered based on G90 and G91.

R : R point coordinate value in the 3rd direction.

I : Length in the spindle direction Commanded with decimal point

J : Length in the sub-spindle direction "

K : Surface cutting width (The cutting method is designated by the sign ⊕ ⊖ .)

Q : Cutting depth per cutting in the 3rd axis direction (⊖ : No finishing)

P : Approaching amount (Absolute value)

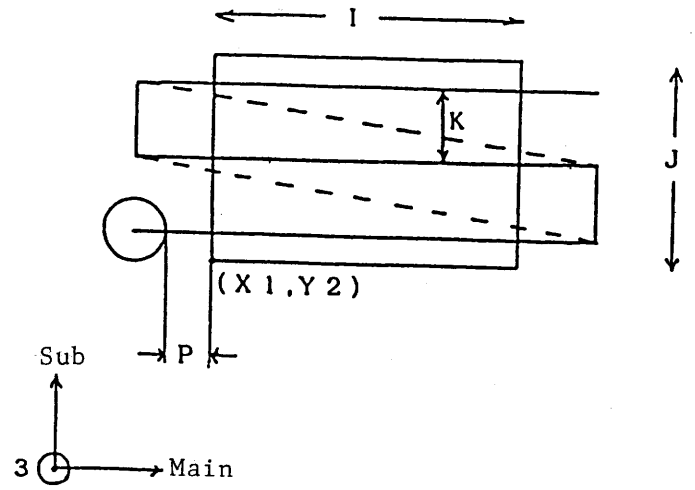
C : Release amount in the 3rd axis direction (Absolute value) Commanded with decimal point

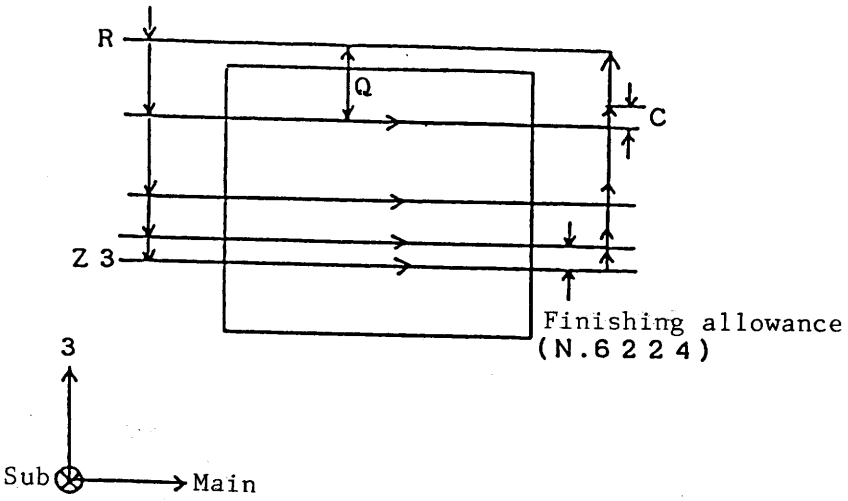
D : Tool radius compensation No. (Radius register)

E : Feed rate for finishing (When omitted, the feed rate is F.) (mm/min)

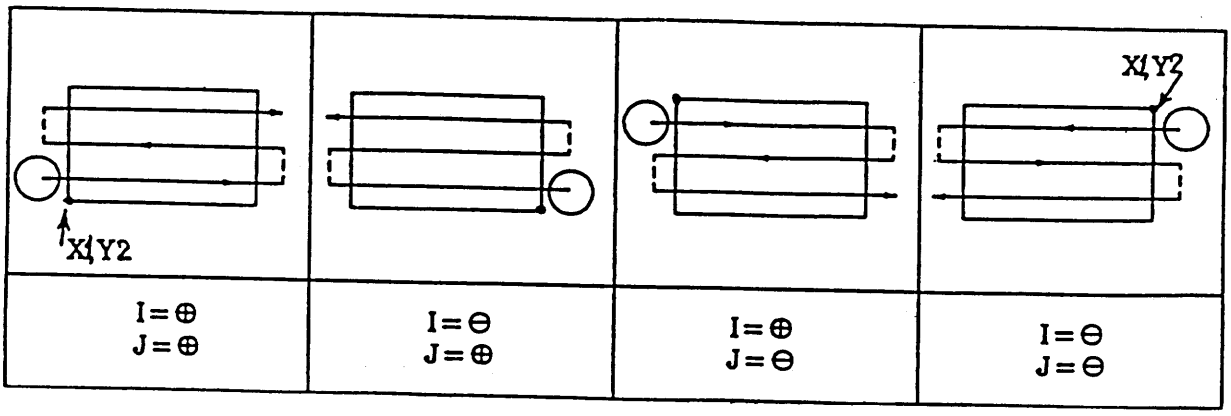
U : S-axis rotation speed for finishing (When omitted, the rotation speed is S.) (r.p.m.)

F : Cutting feed rate (When omitted, F commanded previously) (mm/min)

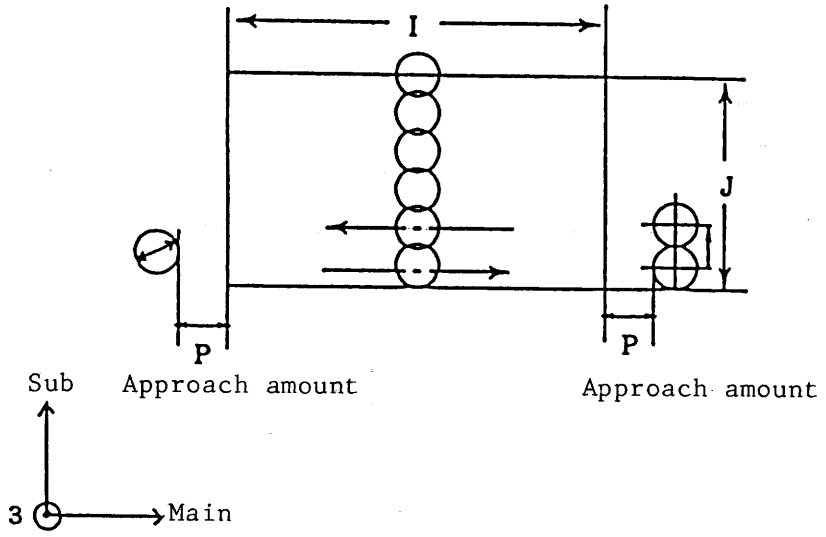


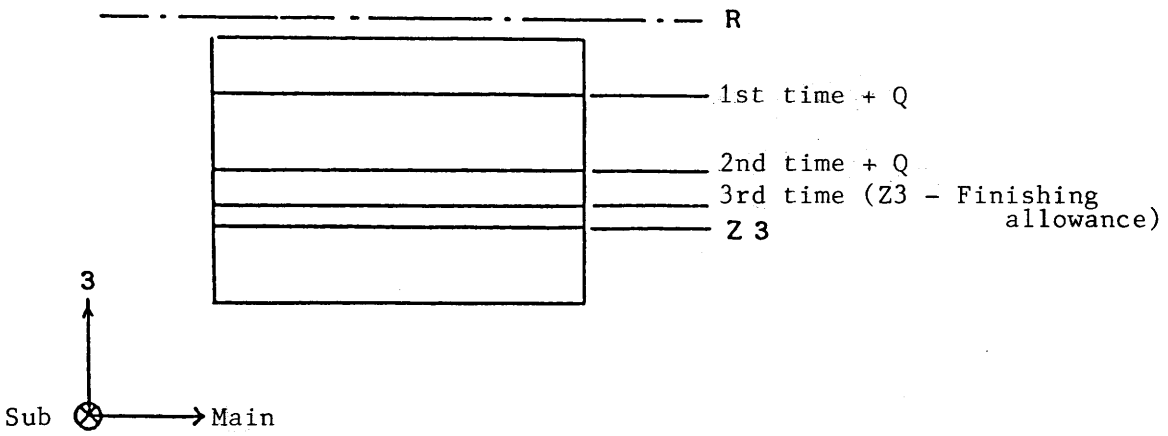


(b) Movements

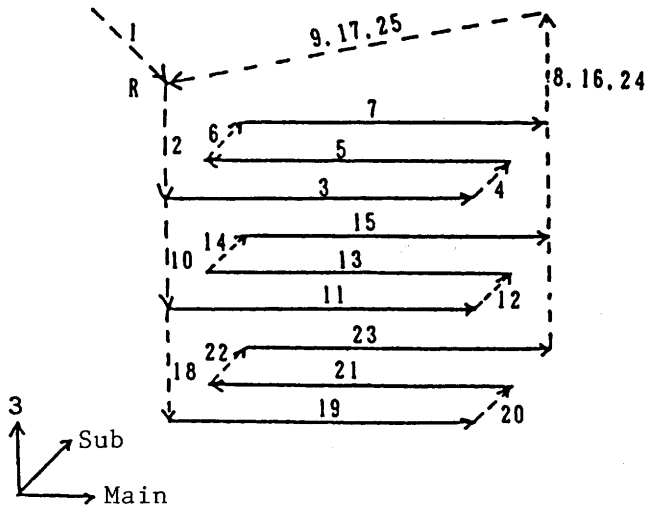


The start point and cutting direction can be changed by altering the sign of J. And when the cutting width: K is set in negative, the cutter center is stepped out to the outside by only the approaching amount.

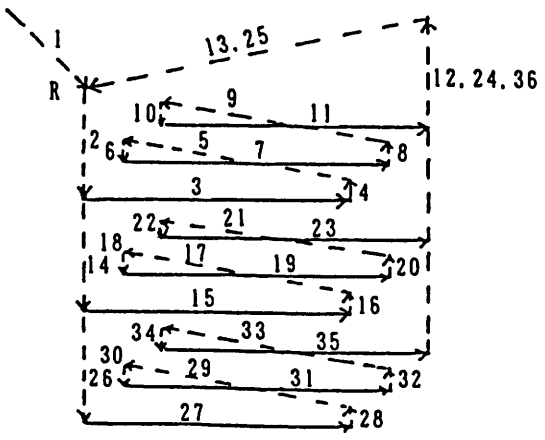




Solid line: Cutting feed
Dotted line: Rapid traverse

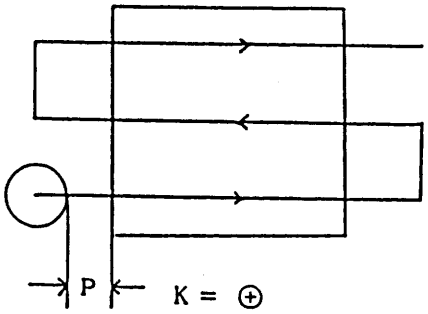


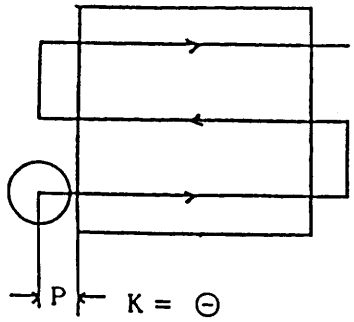
Bidirectional cutting (| Q | ≧ C)



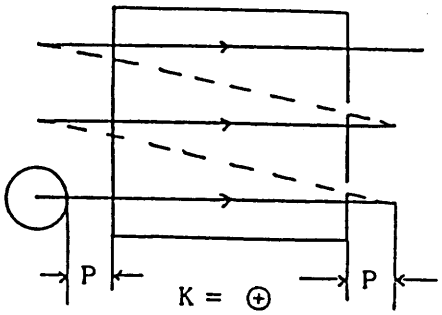
Unidirectional cutting (| Q | < C)

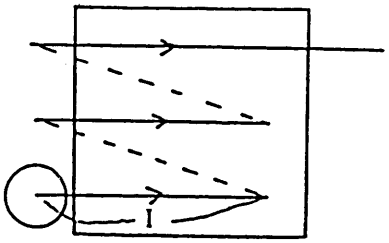
Bidirectional cutting (| Q | ≧ C)





Unidirectional cutting (| Q | < C)





K = ⊖

(2) Square surface one side sizing (G325)

Purpose : Multi-directional cutting is enabled and the last face can be designated as well.

(b) Commanded form

G325 X1_Y2_Z3_R_I_J_K_Q_P_C_D_E_U_F_;

G325 : Square surface one side sizing

X1,Y2: Start point coordinate value of the plane, enterd based on G90 and G91.

Z3 : 3rd axis coordinate value of the finishing face, entered based on G90 and G91.

R : R point coordinate value in the 3rd direction

I : Length in the I-axis direction Commanded with decimal point

J : Length in the J-axis direction "

K : Surface cutting width (The cutting method is designated by the sign ⊕ ⊖ .)

Q : Cutting depth per cutting in the 3rd axis direction (⊖ : No finishing)

P : Approaching amount (Absolute value)

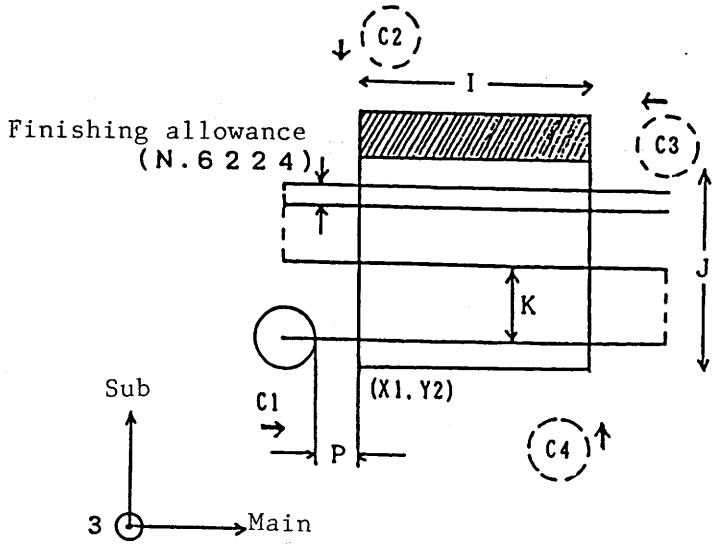
C : Designation of cutting direction ($1 \leq C \leq 4$)

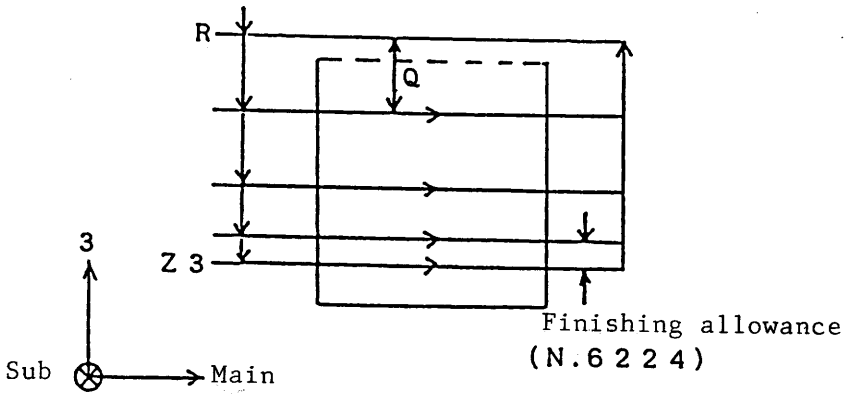
D : Tool radius compensation No. (Radius register)

E : Feed rate for finishing (When omitted, the feed rate is F.) (mm/min)

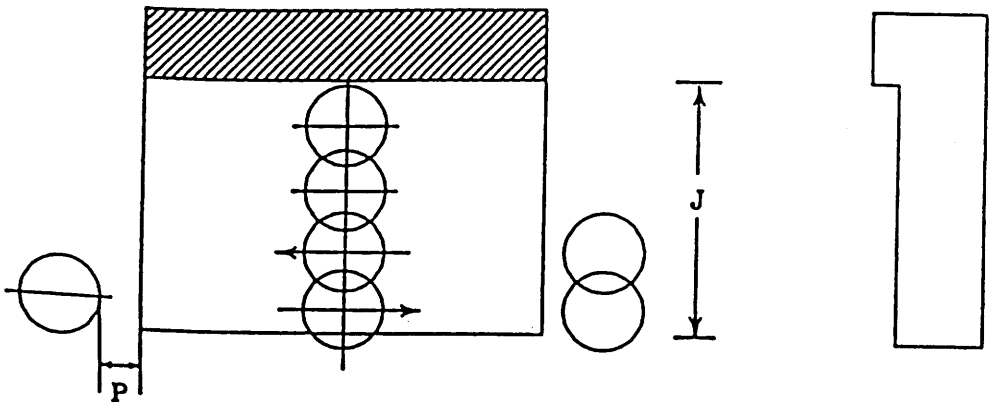
U : S-axis rotation speed for finishing (When omitted, the rotation speed is S.) (r.p.m.)

F : Cutting feed rate (When omitted, F commanded previously) (mm/min)


C2
I
Finishing allowance
(N.6224)
C3
J
K
Sub
C1
(X1, Y2)
P
C4
3
Main

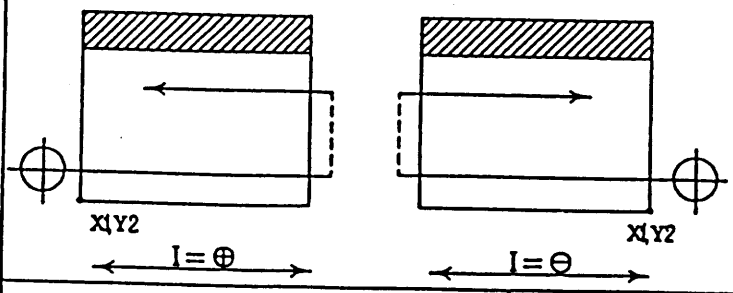

R
Q
3
Z 3
Finishing allowance
(N.6224)
Sub
Main

(b) Movements


J
P
Approach amount

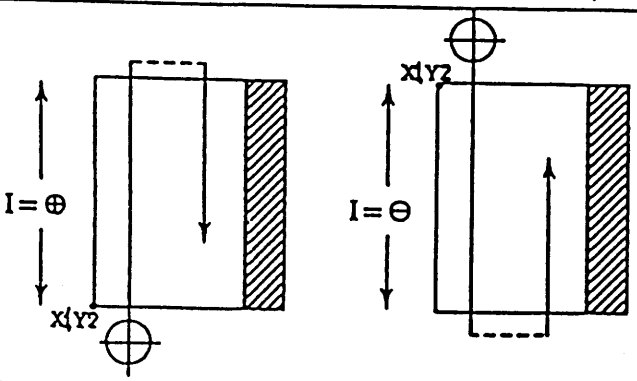
1) X1, Y2 approaching point, Rapid traverse until R point
↓
2) Rapid traverse until the cutting high of 3rd axis
↓
3) Machining in the I command axis direction
↓
4) Machining in the J command axis direction, K= ⊕ : Rapid traverse,
K= ⊖ : Machining
↓
5) I-J plane 3) and 4) shall be repeated until the machining is finished.
↓
6) Rapid traverse return until R point.
↓
7) 1) ~ 6) shall be repeated until 3rd axis coordinate value, and the bottom and the plane are finished in the final step.

C1

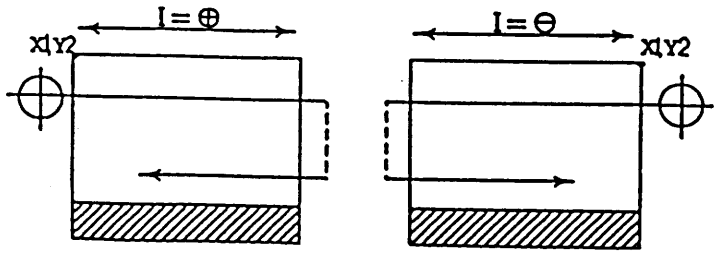


The start point and the cutting direction are designated by the sign C and I.

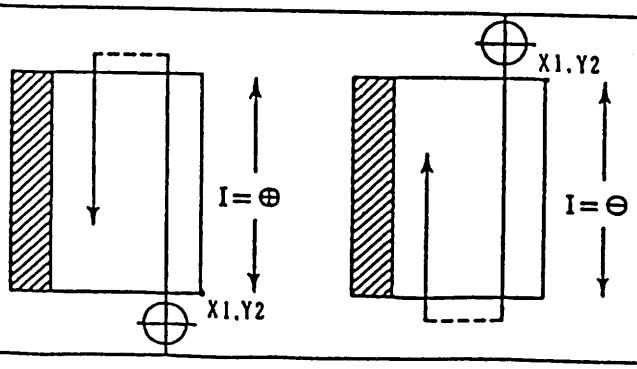
C2



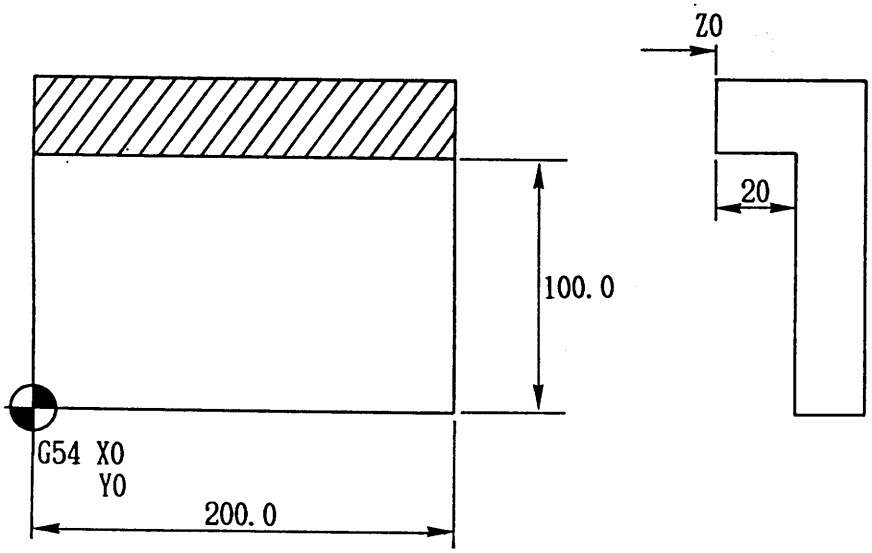
C3



C4



Program example)



Tool diameter: ϕ30
An end mill is used.

```
G90 G54 G00 X0 Y0 S300 ;
(G43) Z50.0 (H01) ;
M03 ;
G325 X0 Y0 Z-20.0 R3.0 I200.0 J100.0 K21.0 C1 Q10.0 D01 F100 ;
M05 ;
```

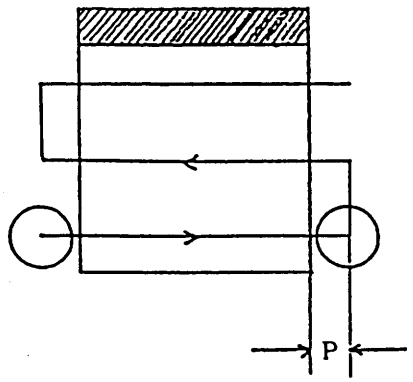
(c) Precautions

(c.1) When cutting depth per cutting (Q) is ⊖ , no finishing is performed. (The plane is included as well.)
And when (|R - Z|) ≦ | Q |, one time cutting is performed.

(c.2) Cut in by cutting depth per cutting (Q) from R point.

(c.3) J : The length in the J direction becomes positive in spite of ⊕ or ⊖ .

(c.4) When K is set to ⊖ , the cutter center goes out to the outside only at approaching.
Cutting in for cutting width is done at cutting feed rate.



(3) Square surface both sides sizing (G326)

Purpose : Used for cutting the inside surface when both side faces exsist.

(a) Command form

G326 X1_Y2_Z3_R_I_J_K_Q_P_C_D_E_U_F_;

G326 : Square surface both sides sizing

X1,Y2: Start point coordinate value of the surface, enterd based on G90 and G91.

Z3 : 3rd axis coordinate value of the finishing face, enterd based on G90 and G91.

R : R point coordinate value in the 3rd direction

I : Length in the I-axis direction

J : Length in the J-axis direction

K : Surface cutting width (The cutting method is designated by the sign ⊕ ⊖ .)

Q : Cutting depth per cutting in the 3rd axis direction (⊖ : No finishing)

P : Approaching amount (Absolute value)

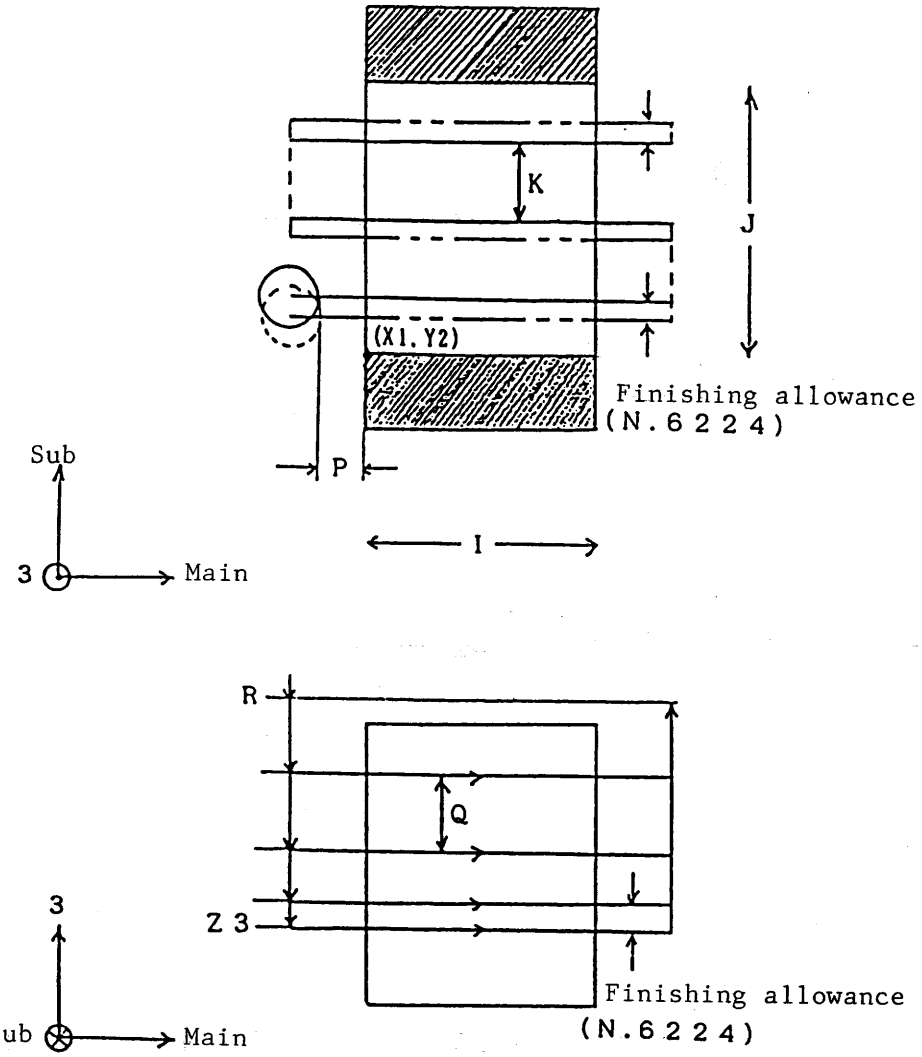
C : Designation of cutting direction (1 = or 2)

D : Tool radius compensation No. (Radius register)

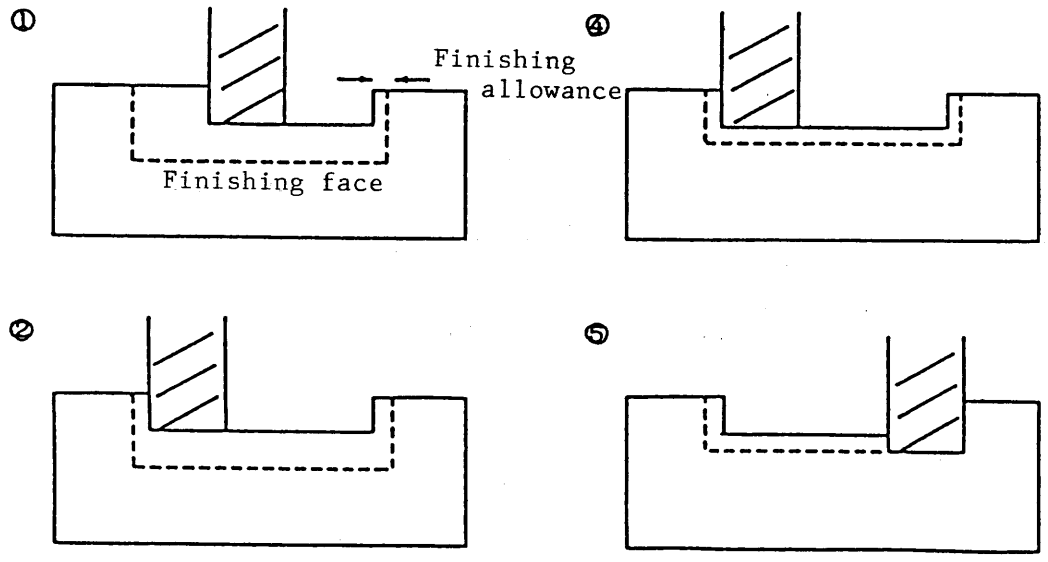
E : Feed rate for finishing (When omitted, the feed rate is F.) (mm/min)

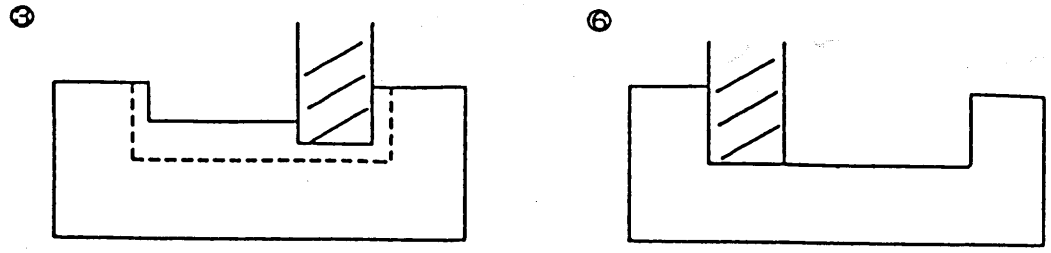
U : S-axis rotation speed for finishing (When omitted, the rotation speed is S.) (r.p.m.)

F : Cutting feed rate (When omitted, F commanded previously) (mm/min)


K
J
(X1, Y2)
Finishing allowance
(N.6224)
P
I
Sub
3
Main
R
Q
Z 3
3
Sub
Main
Finishing allowance
(N.6224)

(b) Movements


①
Finishing
allowance
Finishing face
④
②
⑤



① The start point side shall be cut, leaving finishing allowance of the side face.

↓

② The end point side shall be cut, leaving finishing allowance of the side face.

↓

③ The start point side shall be cut in from the position leaving finishing allowance of the side face.

↓

④ The end point side shall be cut, leaving finishing allowance of the side face.

↓

⑤ The start point side shall be machined for finishing the side face end bottom.

↓

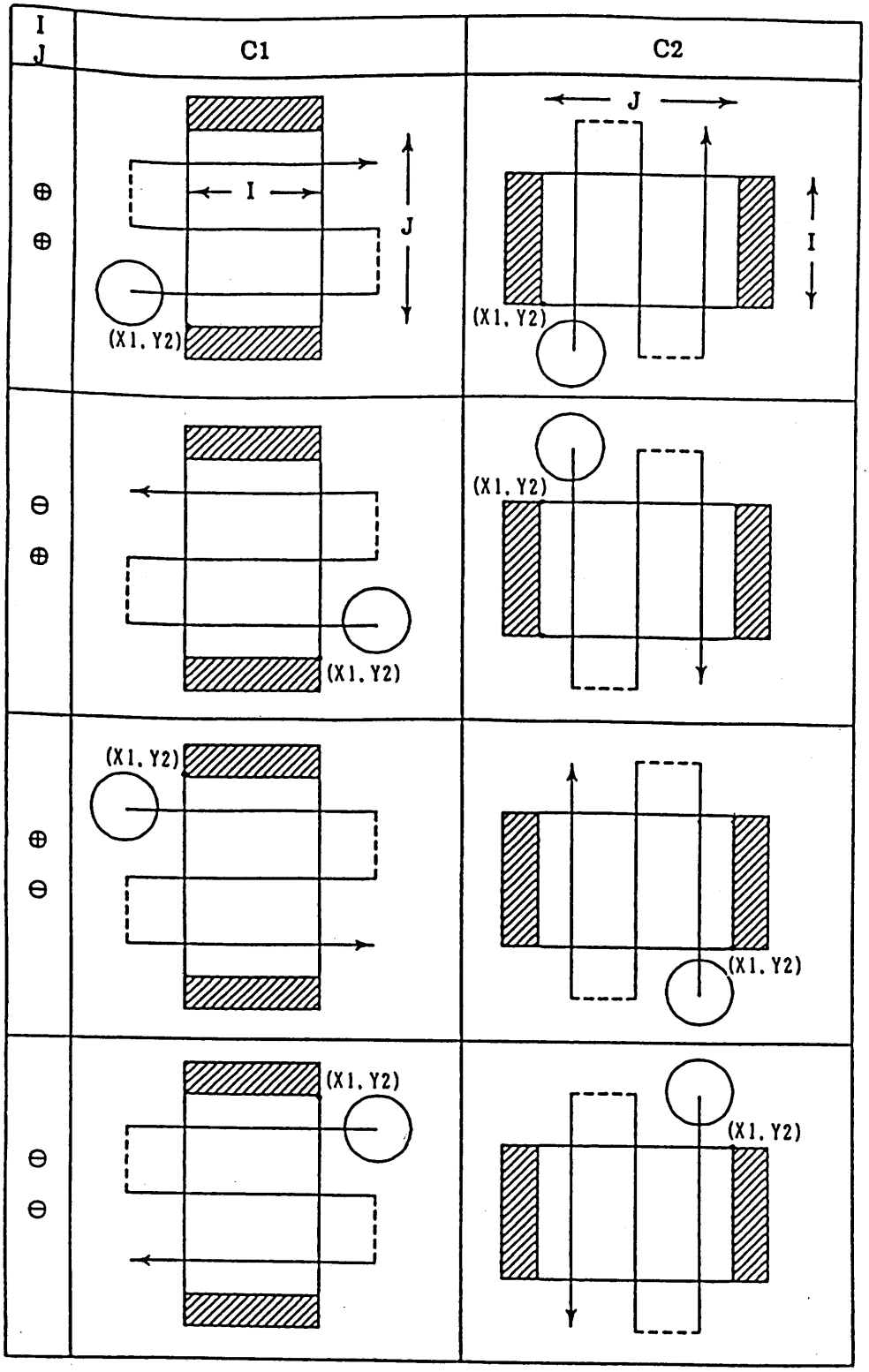
⑥ The end point side shall be machined for finishing the side face and bottom.

(C) Precautions

(c.1) When (|R - Z|) ≦ | Q |, no finishing is done.

(c.2) When the cutting depth per cutting (Q) is set to ⊖, no finishing is done (including the surface.).

(c.3) When the cutting width (K) is ⊖, the cutter center goes out to the outside by the approaching amount.


I
J
C1
C2
⊕
⊕
I
J
(X1, Y2)
J
I
(X1, Y2)
⊖
⊕
(X1, Y2)
(X1, Y2)
⊕
⊖
(X1, Y2)
(X1, Y2)
⊖
⊖
(X1, Y2)
(X1, Y2)

7-24 Pocket cutting (G327 ~ G333)

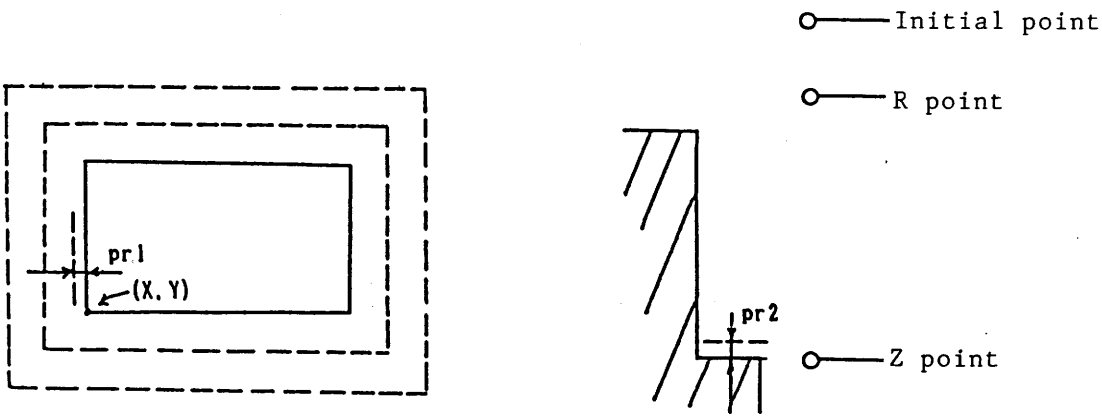
It is possible to command a series of movements for cutting the inner side or outer side of a circle, truck and square in one block.

(1) G code

G327 : Inner circle
G328 : Inner square
G329 : Inner truck
G330 : Outer circle
G331 : Outer square
G332 : Outer truck
G333 : True circle

(2) Command form

G_X_Y_Z_R_Q_I_J_K_P_A_C_U_V_W_E_D_F_;



G_ : Mode
(X, Y) : Reference point of X and Y axes
Z : Z point
R : R point
I, J, K, P, A, C, U, V, W, E : Refer to the description of each function.

I, J, K, P, A and C shall be commanded with decimal point.

D : Tool offset No.
F : Cutting feed rate
pr1 : Finishing allowance (Parameter setting)
pr2 : Clearance amount (Parameter setting)

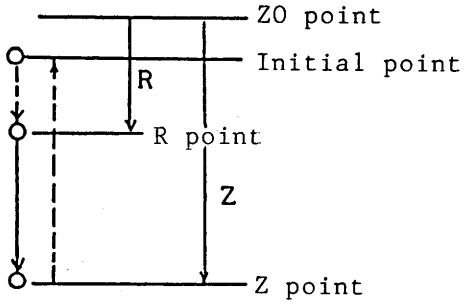
(3) Initial point

The initial point is the machining start point for G327 ~ G333 commands. When a series of movements are finished, all the X, Y and Z axes return to the start point.

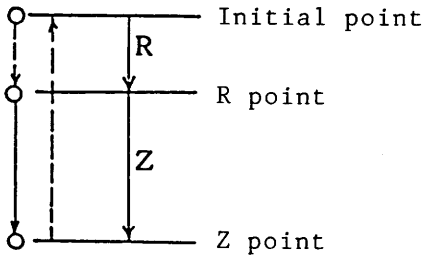
(4) R point and Z point

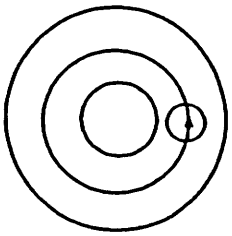
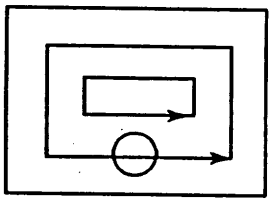
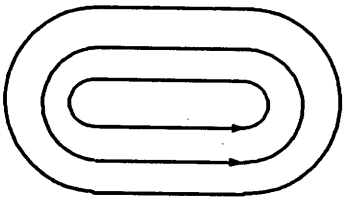
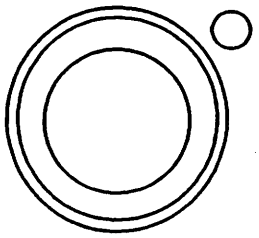
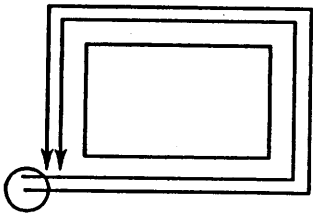
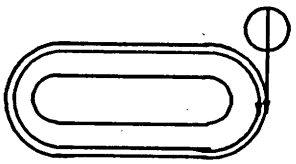
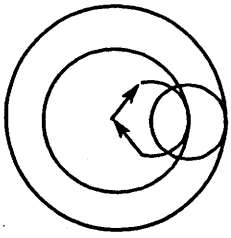
R point and Z point become as follows by G90 and G91 commands.

[G90]



[G91]



1. Inner side of circle **G327** 	2. Inner side of square **G328** 	3. Inner side of truck **G329** 
4. Outer side of circle **G330** 	5. Outer side of square **G331** 	6. Outer side of truck **G332** 
7. True circle **G333** 		

(5) Precautions

(a) Tool radius compensation is engaged regardless of the tool radius compensation (G41 and G42) by G327 ~ G333 commands.
Therefore, command them in the state of tool raduis compensation cancel (G40).

(b) G327 ~ G333 are non modal codes.

(c) When F is omitted, the F already commanded becomes effective.

(Note 1) As to the details of each function, refer to separate description.

(6) Related parameters

No.6201, #0=0		The plane selection of pocket cutting is always in
	1	accordance with G17 ~ G19 commands for the XY plane.
No.6207		Override amount for the F command (1 ~ 100%) when E is omitted in the finish machining.
No.6208		Override amount (1 ~100%) for the feed rate at rouding the corner R when commanding A (the radius of the corner R) by G322, G323 G328 and G333.
No.6224		Finishing allowance
No.6225		Clearance amount

(7) Related alarm

No.137	There is an error in the command of pocket cutting.

1. Circle pocket cutting (G327)

Purpose : This is used for pocket cutting of inner circle by end mill.

(1) Command form

G327 X1_Y2_Z3_R_I_J_K_Q_D_E_U_V_F_;

G327 : Circle pocket

X1,Y2: Circle center coordinate value, entered based on G90 and G91.

Z3 : 3rd axis coordinate value of pocket finishing, entered based on G90 and G91.

R : R point coordinate value in the 3rd axis direction.

I : Radius of cutting circle (the cutting direction is selected by the sign ⊕ ⊖).

J : Radius of the semi-finished hole (the cutting method is designated by the sign ⊕ ⊖).

K : Cutting width (⊖ : No finishing in the side face direction).

Q : Cutting depth per cutting in the 3rd axis direction (⊖ : No finishing in the 3rd axis direction).

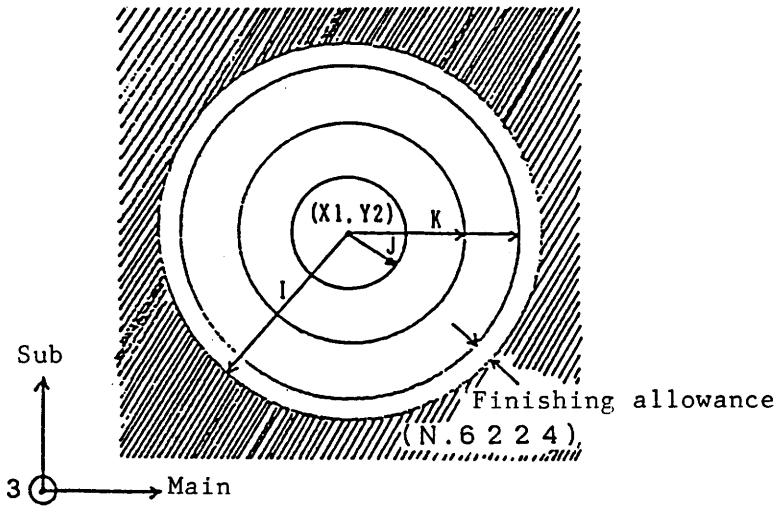
D : Tool radius compensation No. (radius resister).

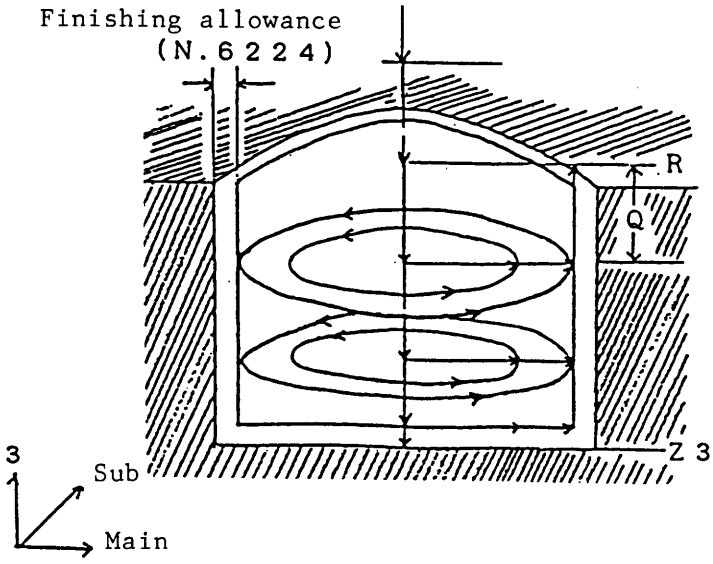
E : Feed rate for finishing (when omitted, the feed rate is F). (mm/min)

U : S-axis rotation speed for finishing (when omitted, the rotation speed is S). (r.p.m)

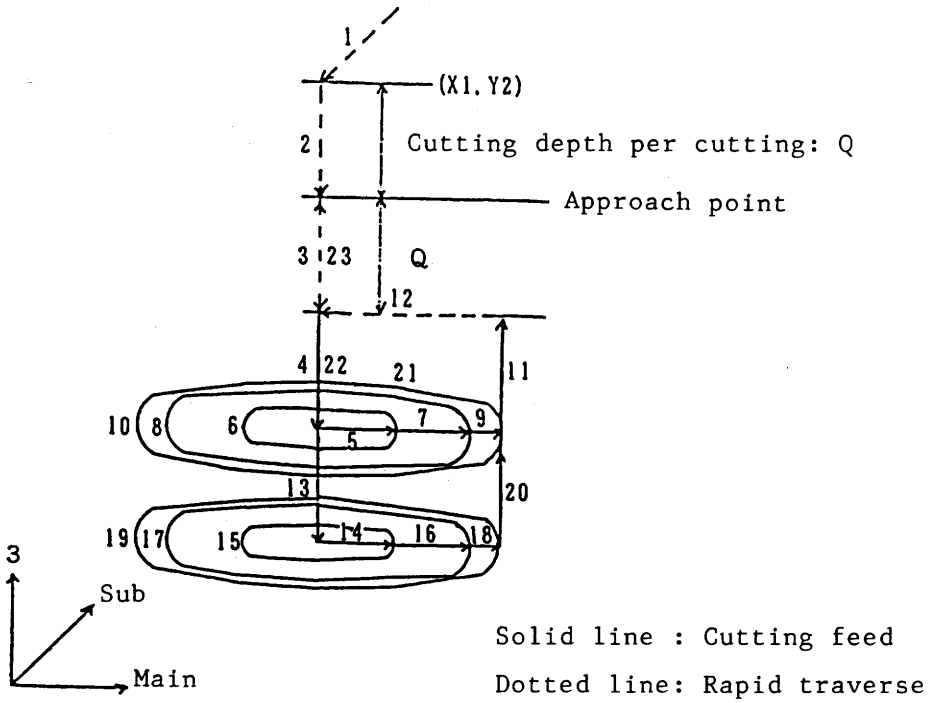
V : 3rd axis cut-in speed (when omitted, the feed rate is F). (mm/min)

F : Cutting feed rate (when omitted, F commanded previously). (mm/min)





(2) Movements



1. Moves to X1, Y2 point in rapid traverse.
2. Moves to the 3rd axis approching point in rapid traverse.
   (R point + cutting depth per cutting in the 3rd axis direction (Q).)
3. Moves to the R point in rapid traverse.
4. Cuts in by the cutting depth per cutting Q by the cutting speed, V or F, in the 3rd axis direction.
5. Cuts by the cutting width (K) considering the semi-finished hole (J) by the cutting speed (F), in the spindle direction.

6. Circular cutting

Circle radius command : I ⊕ (CW)

Circle radius command : I ⊖ (CCW)

7 ~ 10. Item 5 and 6 are repeated.

11. Releases by the cutting depth per cutting Q by the cutting speed, V or F, in the 3rd axis direction.

12. Returns to the spindle center in rapid traverse.

13 ~ 21. Item 4 ~ 12 are repeated.

o Spindle, finish cutting until a designated dimension in the 3rd axis direction.

22. Returns to the R point in rapid traverse.

23. Returns to the approaching point in rapid traverse to finish in the 3rd axis direction.

(3) Precautions

(a) When the cutting width (K) is ⊖, no finish cutting is done in the spindle and sub-spindle (side face) directions. (Fig. 7.1)

(b) When the radius of the semi-finished hole (J) is ⊖, the tool doesn't return to the X-axis center in the machining cycle. (Fig. 7.2)

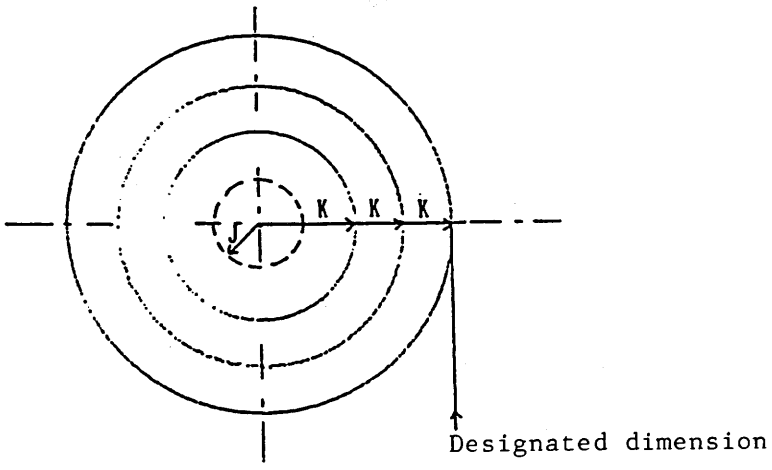


Fig. 7.1 : K-**

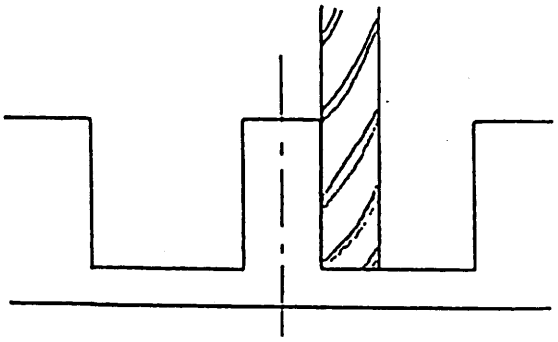


Fig. 7.2 : J-**

(c) When the cutting depth per cutting (Q) in the 3rd axis direction is ⊖, finish machining is not made in the 3rd axis direction.

2. Square pocket cutting (G328)

Purpose : This is used when machining the inside of square workpieces by end mill.
The corner R can be designated as well.

(1) Command form

G328 X1_Y2_Z3_R_I_J_K_Q_C_A_D_E_U_V_F_;

G328 : Square pocket

X1, Y2: Start point coordinate value of the plane, entered based G90 and G91.

Z3 : 3rd axis coordinate value, entered based on G90 and G91.

R : R point coordinate value in the 3rd axis direction.

I : Length of one side in the spindle direction.

J : Length of one side in the sub-spindle direction.

K : Cutting width (⊖ : No finishing in the side face direction).

Q : Cutting depth per cutting in the 3rd axis direction.
(⊖ : No finishing in the 3rd axis direction)

C : Cutting allowance of single side in the side face direction. (Absolute value)

A : Radius when the corner R is designated. (Absolute value)

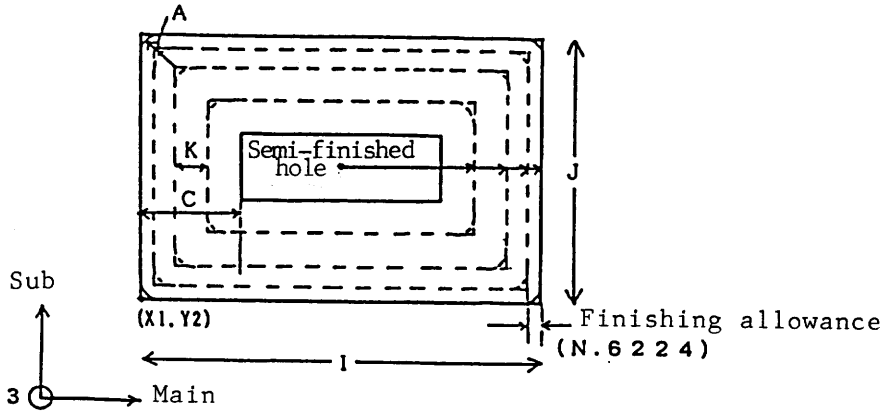
D : Tool radius compensation No.. (Radius resister)

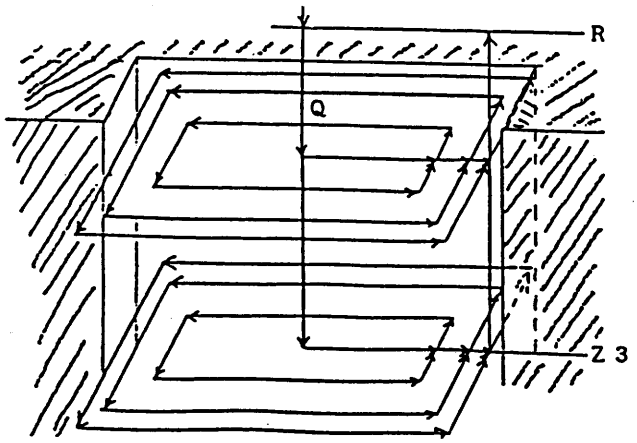
E : Feed rate for finishing. (When omitted, the feed rate is F.) (mm/min)

U : S-axis rotation speed for finishing. (When omitted, the rotation speed is S.) (r.p.m)

V : 3rd axis cut-in feed. (When omitted, the feed rate is F.) (mm/min)

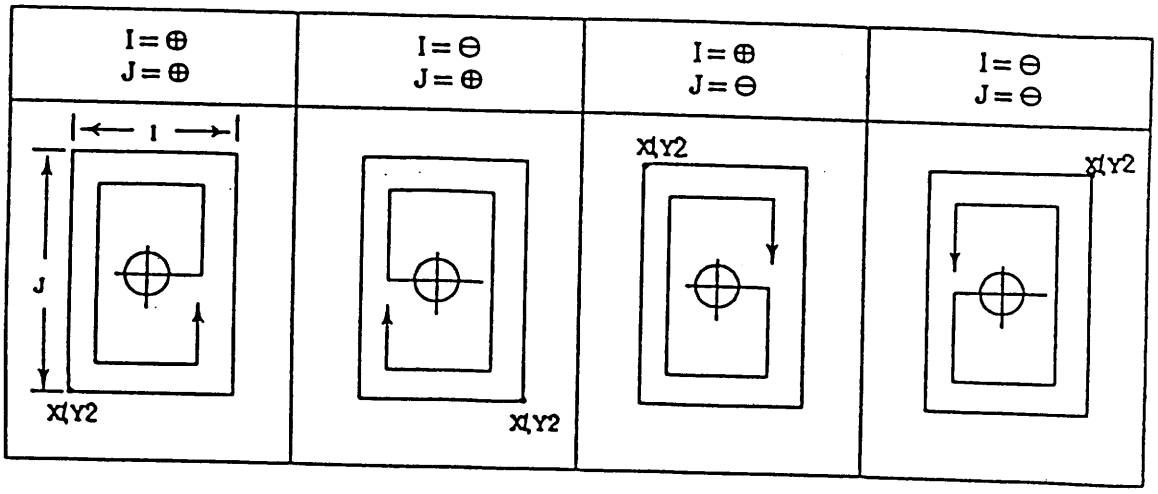
F : Cutting feed rate. (When omitted, F commanded previously.) (mm/min)



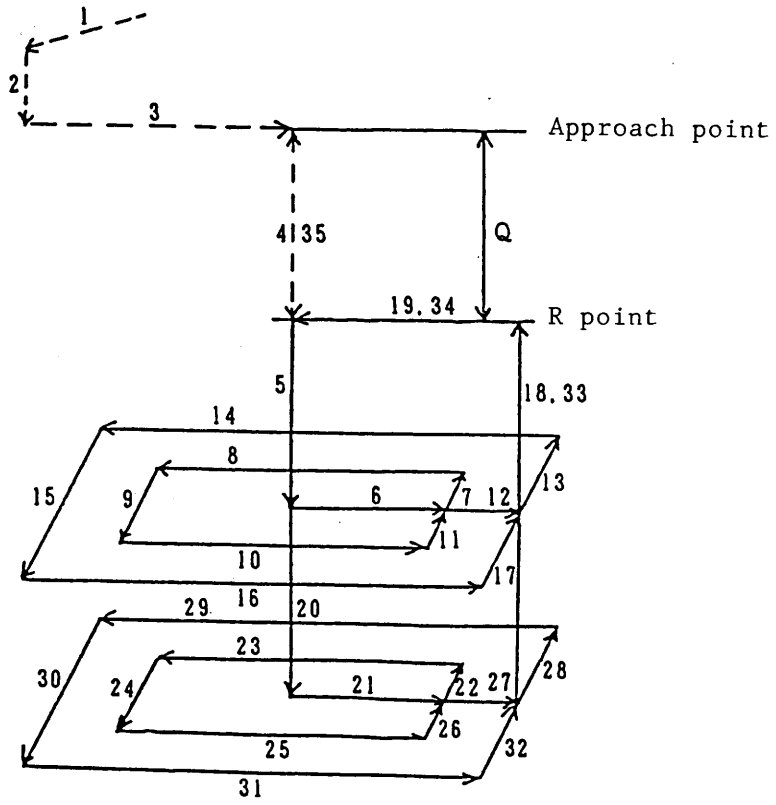


(2) Movements

Cutting pattern



The cutting method can be changed by the sign, I and J.



1. Moves to the X1, Y2 point in rapid traverse.
2. Moves to the 3rd approaching point in rapid traverse.
   (R point + cutting depth per cutting in the 3rd axis direction (Q).)
3. Moves to the work center in the side face direction in rapid traverse.
4. Moves to the R point in rapid traverse.
5. Cuts in by the cutting depth per cutting Q at the cutting speed, V or F, in the 3rd axis direction.
6. Cuts by the cutting width, K, at the cutting speed F, in the spindle direction.
7. Cuts at the cutting speed, nJmax. in the sub-spindle direction.
8. Cuts at the cutting speed, nImin. in the spindle direction.
9. Cuts at the cutting speed, nJmin. in the sub-spindle direction.
10. Cuts at the cutting speed, nImax. in the spindle direction.
11. Cuts until the work center at cutting speed in the sub-spindle direction.

12 ~ 17. Item 6 ~ 11 are repeated.

18. Releases by the cutting depth per cutting (Q) at the cutting speed, V or F, in the 3rd axis direction.

19. Moves to the work center in rapid traverse in the side face direction.

20 ~ 32. Item 5 ~ 17 are repeated.

o When only finishing allowance is left at item 32, the tool returns to the R point and the work is cut on the finishing condition until the dimension designated in the 3rd axis direction. It is cut by a designated dimension in the side face direction too.

33. Returns to the R point at cutting speed in the 3rd direction.

34. Returns to the work center in rapid traverse in the side face direction.

35. Returns to the approaching point in rapid traverse to finish in the 3rd axis direction.

(3) Precautions

(a) When the semi-finished hole is machined, command cutting depth (C) of one side. When no the command (C) is given, it is presumed that no semi-finished hole is machined, and cutting is preformed from the center. (Fig. 7.3)

(b) When the cutting width (K) is ⊖, no finishing is performed in the side face direction.

(c) When the cutting depth per cutting (Q) is ⊖ in the 3rd axis direction, no finishing is performed in the 3rd direction.

(d) In case of corner R designation (radius A), an override is engaged based on the parameter (N6208) on the corner.
And when A is larger than half of one side or J side, no machining is done.

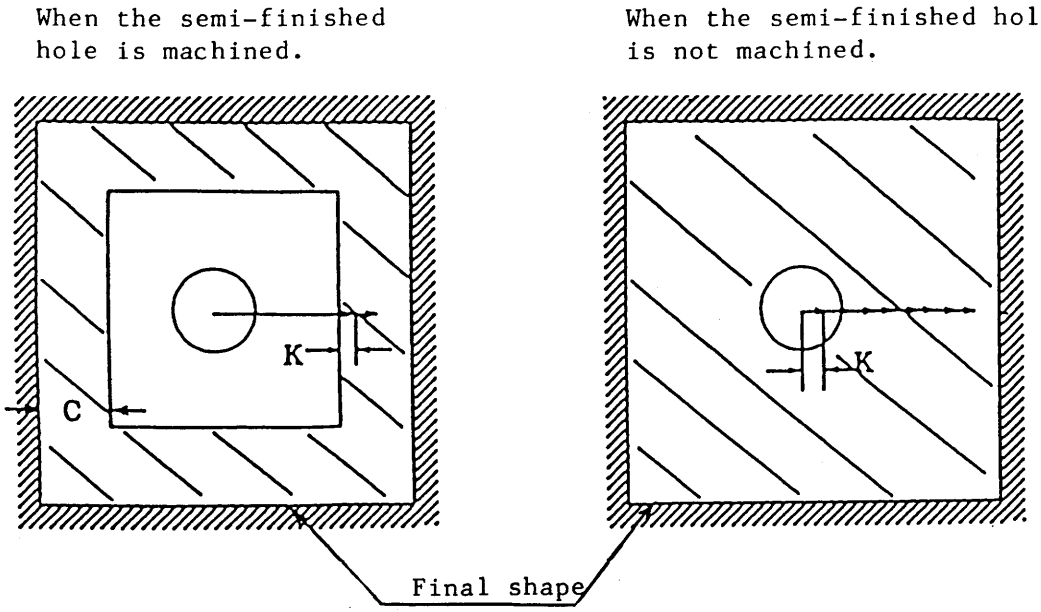


Fig. 7.3

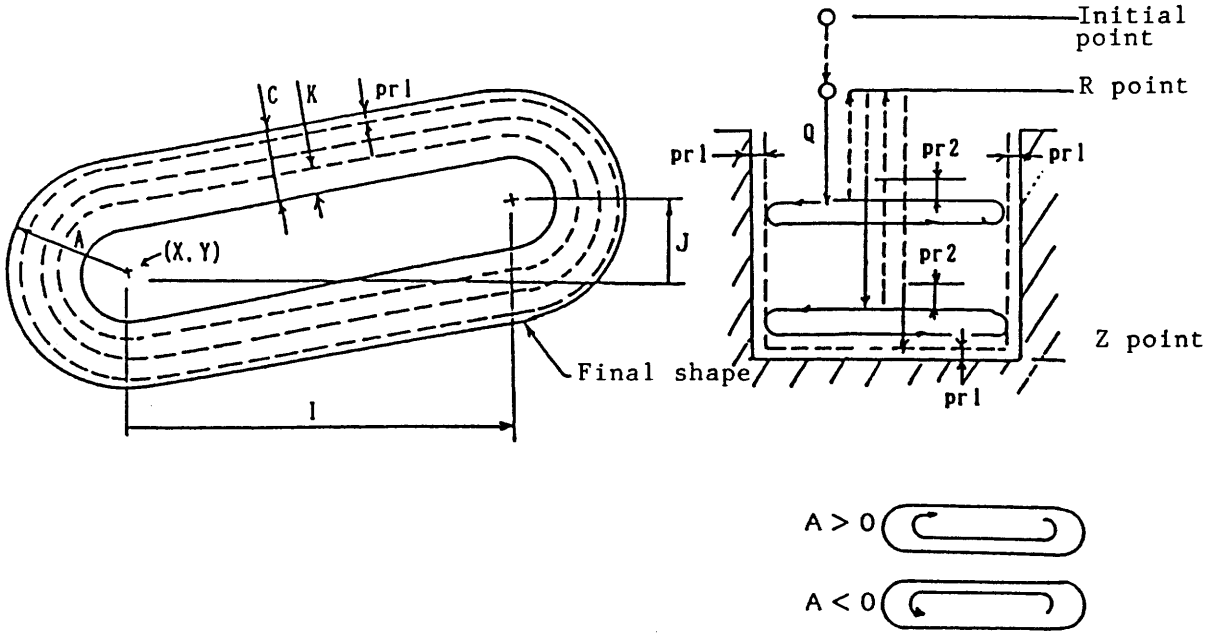
3. Inner truck (G329)

A series of movements cutting the inner periphery of the truck by using an end mill can be commanded in one block.

The below explanation is for G17 (XY plane).

(1) Command form

G329 X_Y_Z_R_I_J_A_C_K_Q_D_V_E_U_F_;



(X, Y) : Reference point of the X and Y axes.

Z : Z point

R : R point

I : Circular arc center constituent in the opposite side (X-axis).

J : Circular arc center constituent in the opposite side (Y-axis).

A : Circular arc radius, commanded with decimal point.

C : Cutting depth, commanded with decimal point.

K : Cutting width per cutting (XY plane), commanded with decimal point.

Q : Cut-in amount per cutting for the Z-axis.

D : Tool offset No..

V : Cut-in speed for the Z-axis (when omitted, the cutting feed rate is F).

E : Feed rate for finishing (when omitted, the cutting feed rate is F).

U : Spindle speed for finishing (when omitted, S already commanded).

F : Cutting feed rate.

pr1 : Finishing allowance (set to the parameter No.6224).

pr2 : Clearance amount (set to the parameter No.6225).

(Note 1) When E is omitted, the feed rate for finishing becomes F × override for finishing (paremeter No.6207).

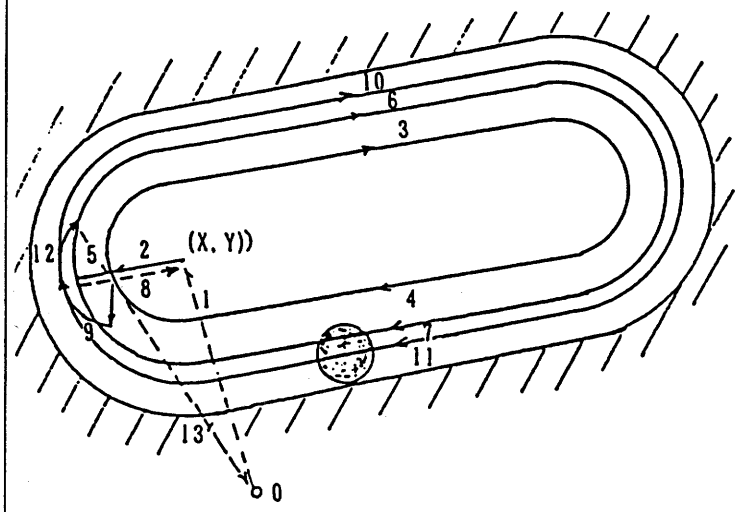
(Note 2) When the numerical value of A is positive, a circular arc becomes CW, and when it is negative, the circular arc becomes CCW.
And when A = 0, an alarm occurs.

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

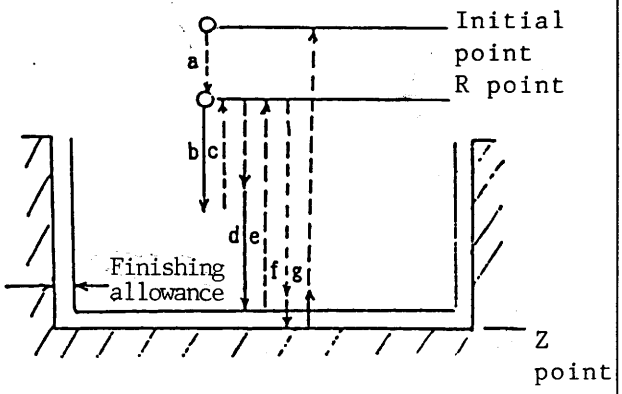
(Note 4) When the numerical value of Q is negative, finishing of the bottom becomes ineffective.

(2) Program example

```
G17;
G90 G329 X50. Y-100. Z-50. R-10. Q20. I50.
J20. A50. C15. K8. D10 F200;
```



(Start point)



However, - - - → Rapid traverse

——→ Cutting rapid traverse

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

Movements : Start point

↓

Moves to the (X, Y) in rapid traverse

↓

Moves to the R point in rapid traverse

↓

Z-axis cut-in

↓

1st inner periphery of truck cutting

↓

2nd inner periphery of truck cutting

↓

Moves to the R point in rapid traverse

↓

Moves to the (X, Y) in rapid traverse

} Rough cotting

↓

Moves to the Z point in rapid traverse and cut in

↓

Finish cutting of the bottom face

↓

Finish cutting of the side face

} Finishing

↓

Returns to the Z-axis initial point (Start point)

↓

Returns to the X and Y axes intial point

(3) Precautions

(a) The numerical values of the address V, E and U shall be commanded without decimal point (in case of Millimetric system).

(b) When tool offset amount (D) > Radius of circular arc (A), an alarm occurs.

(c) When I = 0, J = 0 are commanded, an alarm occurs.

(d) When the address is omitted, the radius of circular arc shall be regarded as cutting allowance.

(e) The values that the inner periphery of truck cutting can be obtained shall be commanded for the numerical values of the addresses such as A, C, K and Q the tool offset amount (D).

4. Outer periphery of circle cutting (G330)

Purpose : This is used for cutting the outer periphery of circle by end mill.

(1) Command form

G330 X1_Y2_Z3_R_I_J_K_Q_P_D_E_U_V_F_;

G330 : Outer periphery of circle

X1, Y2: Coordinate value of the circle center, entered based on G90 and G91.

Z3 : 3rd axis coordinate value, entered based on G90 and G91.

R : R point coordinate value in the 3rd coordinate direction.

I : Radius of the cutting circle (Cutting direction is selected by the sign ⊕ ⊖.)

J : Cutting depth of the cutting circle (Absolute value).

K : Cutting width (⊖ : No finishing in the side face direction).

Q : Cutting depth per cutting in the 3rd axis direction. (⊖ : No finishing in the 3rd axis direction)

P : Approaching amount (Absolute value) Commanded with decimal point.

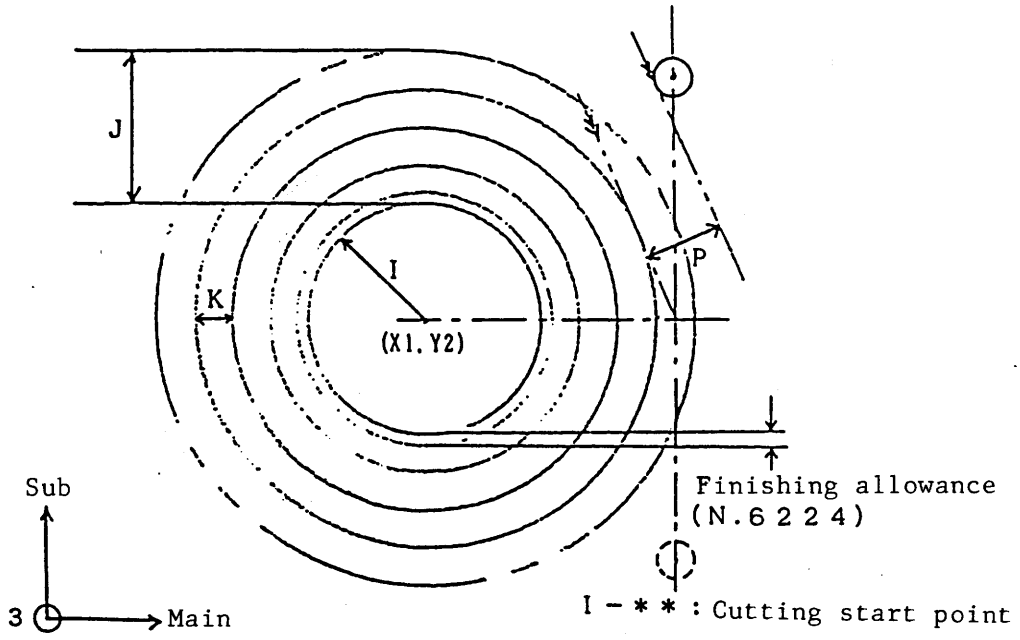
D : Tool radius compensation No. (Radius register).

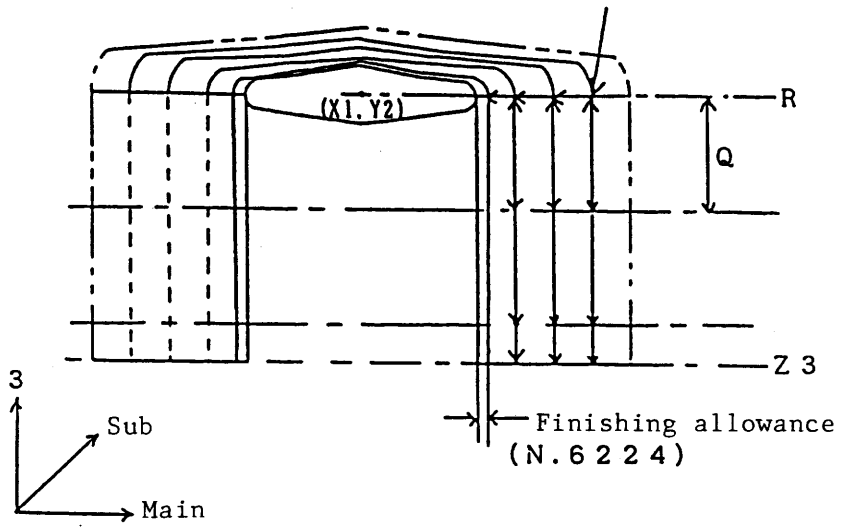
E : Feed rate for finishing (When omitted, the feed rate is F.) (mm/min)

U : S-axis rotation speed for finishing (When omitted, the rotation speed is S.) (r.p.m)

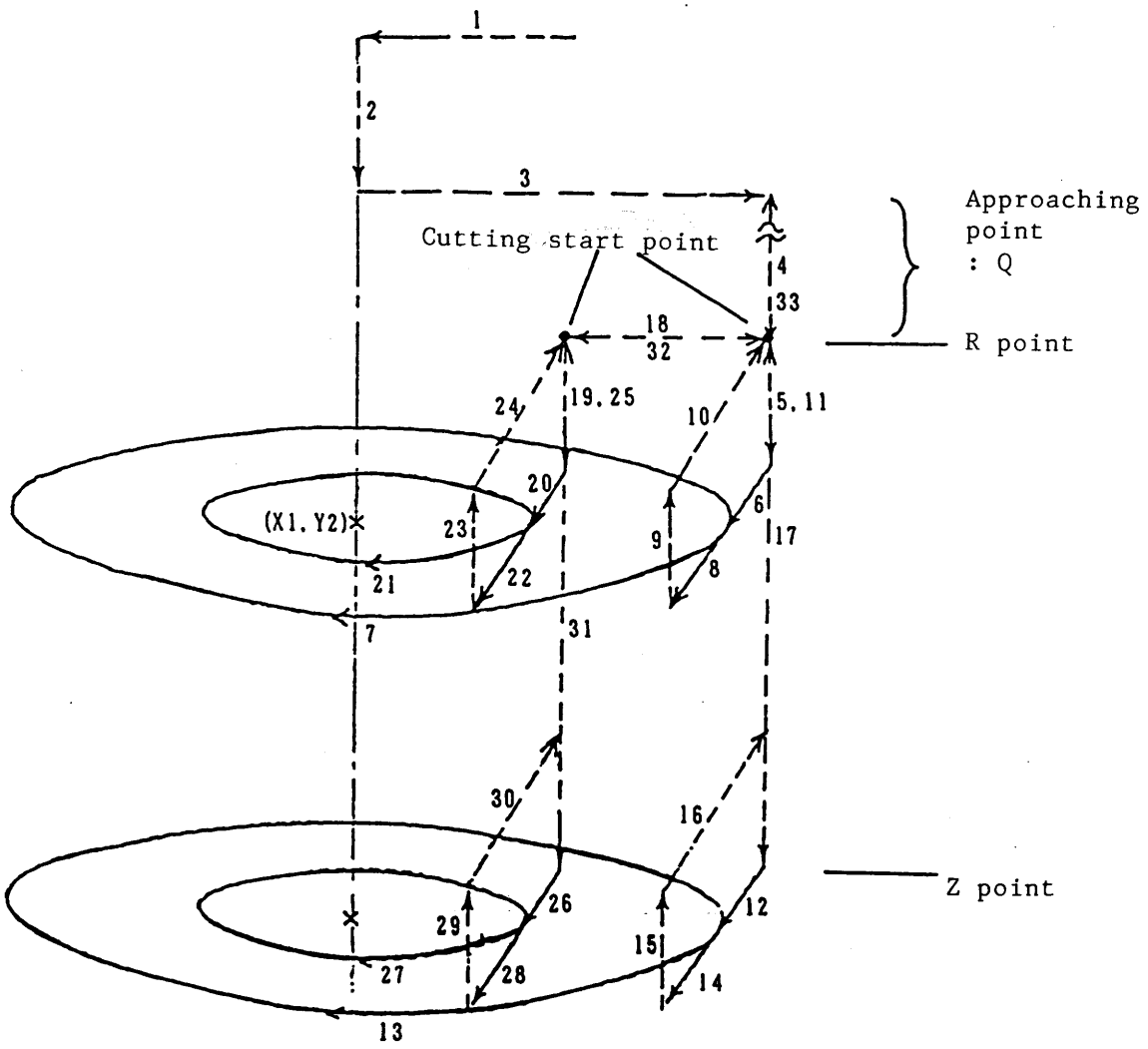
V : 3rd axis cut-in speed (When omitted, the feed tate is F.) (mm/min)

F : Cutting feed rate (When omitted, F commanded previously.) (mm/min)





(2) Movements



5. Square outer periphery cutting (G331)

Purpose : This is used for cutting the outer periphery of circle by using end mill.

The corner R can be designated as well.

(1) Command form

G331 X1_Y2_Z3_R_I_J_K_Q_P_C_A_D_E_U_F_;

G331 : Outer periphery of square.

X1, Y2: Start point coordinate value of the plane, entered based G90 and G91.

Z3 : 3rd axis coordinate value, entered based on G90 and G91.

R : R point coordinate value in the 3rd coordinate direction.

I : Length of one side in the spindle direction. Commanded with decimal point.

J : Length of one side in the sub-spindle direction. Commanded with decimal point.

K : Cutting width (⊖ : No finishing in the side face direction)

Q : Cutting depth per cutting in the 3rd axis direction. (⊖ : No finishing in the 3rd axis direction)

P : Approaching amount (Absolute value) Commanded with decimal point.

C : Cutting allowance of single side in the side face direction. (Absolute value)

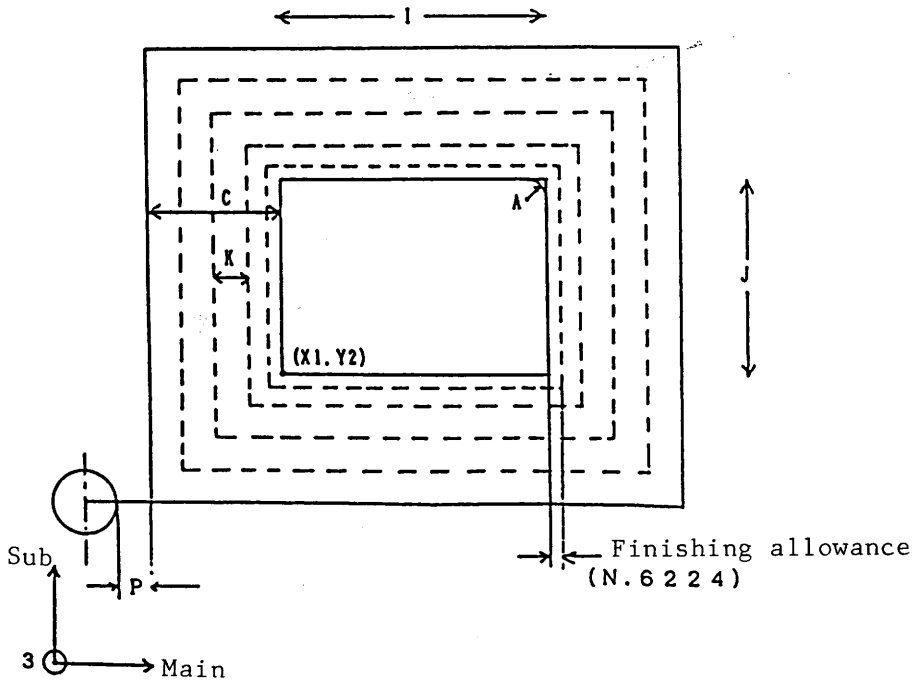
A : Radius when the corner R is designared (Absolute value). Commanded with decimal point.

D : Tool radius compensation No. (Radius resister)

E : Feed rate for finishing (When omitted, the feed rate is F.) (mm/min)

U : S-axis rotation speed for finishing (When omitted, the rotation speed is S.) (r.p.m)

F : Cutting feed rate (When omitted, F commanded previously.) (mm/min)



1. Moves to the X1, Y2 point in rapid traverse.
2. Moves to the 3rd axis approaching point in rapid traverse.
   (R point + cutting depth per cutting in the 3rd axis direction (Q).)
3. Moves to the approaching point considering the cutting width for cutting allowance in rapid traverse.
4. Moves to the R point in rapid traverse.
5. Moves by the cutting depth per cutting (Q) in the 3rd axis direction.
   (In this case, the tool moves to this side by the clearance amount (No.6225) in rapid traverse and moves at cutting feed rate after
6. Cuts to the Y2 point at cutting feed rate in the sub-spindle direction.
7. Circular cutting (Cuter periphery)

   Circle rapid command : I ⊕ (CW)

   Circle rapid command : I ⊖ (CCW)
8. Release by the cutting width (K) at cutting feed rate in the tangential direction of the outer periphery circle.
9. Moves to the cutting start point in the 3rd axis direction in rapid traverse.

10. Moves to the cutting start point in the main sub spindle direction in rapid traverse.

11. ~ 16. Item 5 ~ 10 are repeated.

17. Returns to the R point in rapid traverse.

18. Returns by the cutting width (K) portion in the spindle direction.

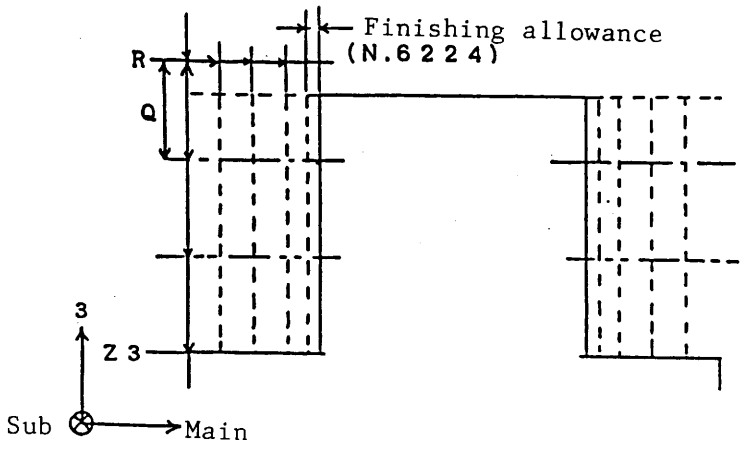
19. ~ 31. Item 5 ~ 17 are repeated.

32. Retuens to the initial cutting start point in rapid traverse in the spindle direction.

33. Returns to the approaching pint in rapid traverse in the 3rd axis direction.

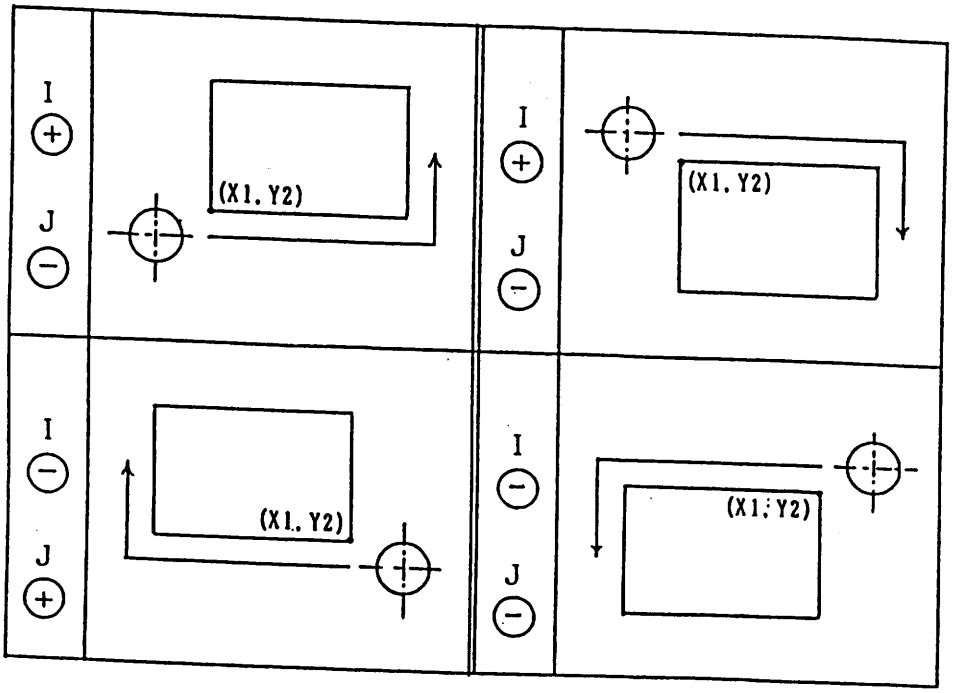
Precautions

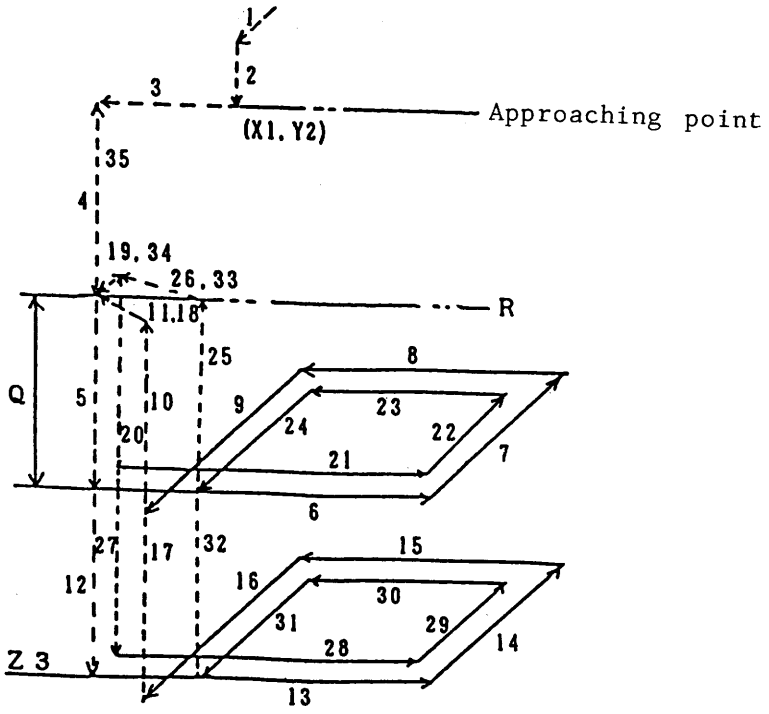
(a) When the cutting width (K) is ⊖, no finish cutting is done in the spindle and sub-spindle (side face) directions.



(2) Movements

Cutting pattern





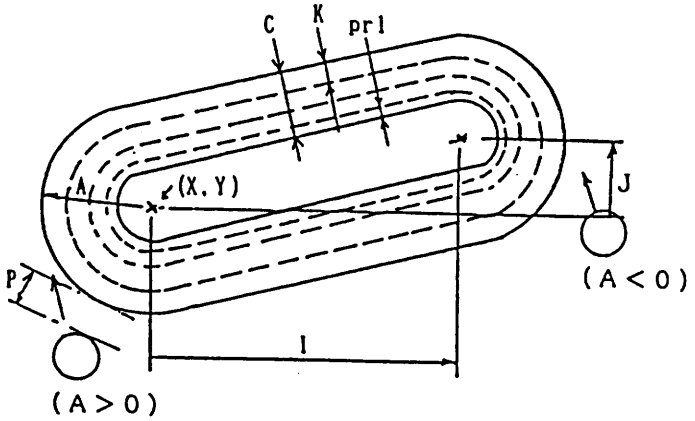
6. Outside truck (G332)

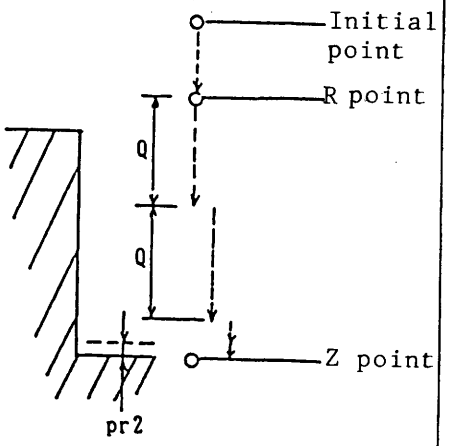
A series of operations cutting the outer periphery of a truck by using end mill can be commanded in one block.

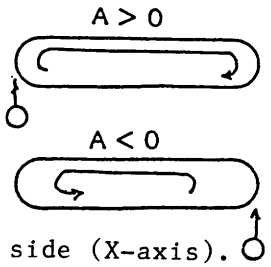
The below explanation is for G17 (XY plane).

(1) Command form

G332 X_Y_Z_R_I_J_A_C_K_Q_P_D_E_U_F_;







(X, Y) : Reference point of X-axis and Y-axis.

Z : Z point

R : R point

I : Circular arc center ingredient of the opposite side (X-axis).

J : Circular arc center ingredient of the opposite side (Y-axis).

A : Radius of circular arc.

C : Cutting allowance.

K : Cutting width per cutting (XY plane).

Q : Cutting depth per cutting for Z-axis.

P : Approaching amount Commanded with decimal point.

D : Tool offset No.

E : Feed rate for finishing (When omitted, cutting feed rate is F.)

U : Spindle speed for finishing (When omitted, S already commanded.)

F : Cutting feed rate.

pr1: Finishing allowance (Set to the parameter No.6224 .)

pr2: Clearance amount (Set to the parameter No.6225.)

1. Moves to the X1, Y2 point in rapid traverse.
↓
2. Moves to the 3rd approching point in rapid traverse.
(R point + Cutting depth per cutting in the 3rd axis direction (Q).)
↓
3. Moves to the approaching point considering the cutting width for cutting allowance in rapid traverse.
↓
4. Moves to the R point in rapid traverse.
↓
5. Moves by the cutting depth percutting (Q) in the 3rd axis direction. (In this case, the tool moves to this side by clearance amount (N6225) in rapid traverse and moves at cutting feed rate after arriving.)
↓
6. Cuts at the cutting speed, nImax. in the spindle direction.
↓
7. Cuts at the cutting speed, nJmax. in the sub-spindle direction.
↓
8. Cuts at the cutting speed, nImin. in the spindle direction.
↓
9. Cuts by one cutting width (K) at the cutting speed, nJmin. in the sub-spindle direction.
↓
10. Release to the R point in rapid traverse in the 3rd axis direction.
↓
11. Returns to the cutting start point.
↓
12. ~ 18. Item 5 ~ 11 are repeated.
↓
19. Moves by the cutting width (K) from the cutting start point in rapid traverse in the sub-spindle direction.
↓
20. ~ 33. Item 5 ~ 18 are repeated.
↓
o When leaving only finishing allowance at the item 33, cut by specified dimensions and under the finishing conditions in theside face direction and in the 3rd axis direction.
↓
34. Returns to the initial cutting start point in the side face direction.
↓
35. Returns to the approaching point in rapid traverse in the 3rd axis direction.

(3) Precautions

(a) When the cutting width (K) is ⊖, finishing is not proceeded in the side face direction.

(b) In case of corner R designation (Radius A) an override is engaged at the corner on the base of the parameter (N6208) it becomes nJmin. in the sub spindle direction. The releasing direction for the remaining K portion is in the spindle direction. When A is large than half of the I sid or the J side, no machining is proceeded.

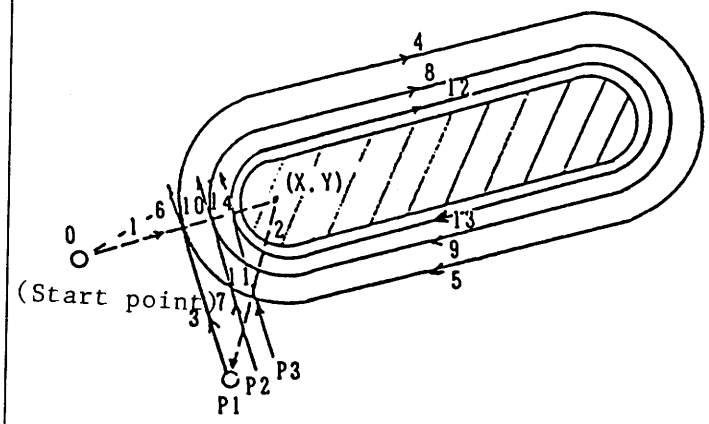
(Note 1) When E is omitted, the feed rate for finishing becomes F × override (parameter No.6207) for finishing.

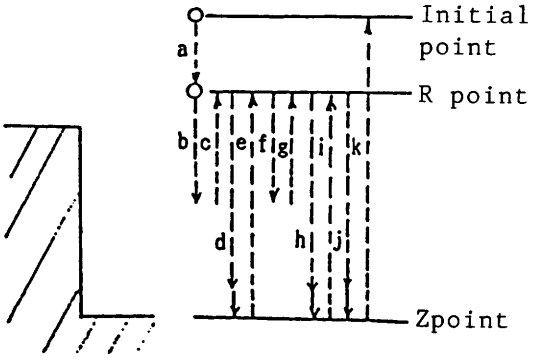
(Note 2) When the numerical value of A is positive, a circular arc becomes CW, and when it is negative, the circular arc becomes CCW. And when A = 0, an alarm occurs.

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

(2) Program example

```
G17;
G90 G332 X50. Y-100. Z-50. R-10. Q20. I50.
J20. A50. C15. K8. P5. D10 F200;
```



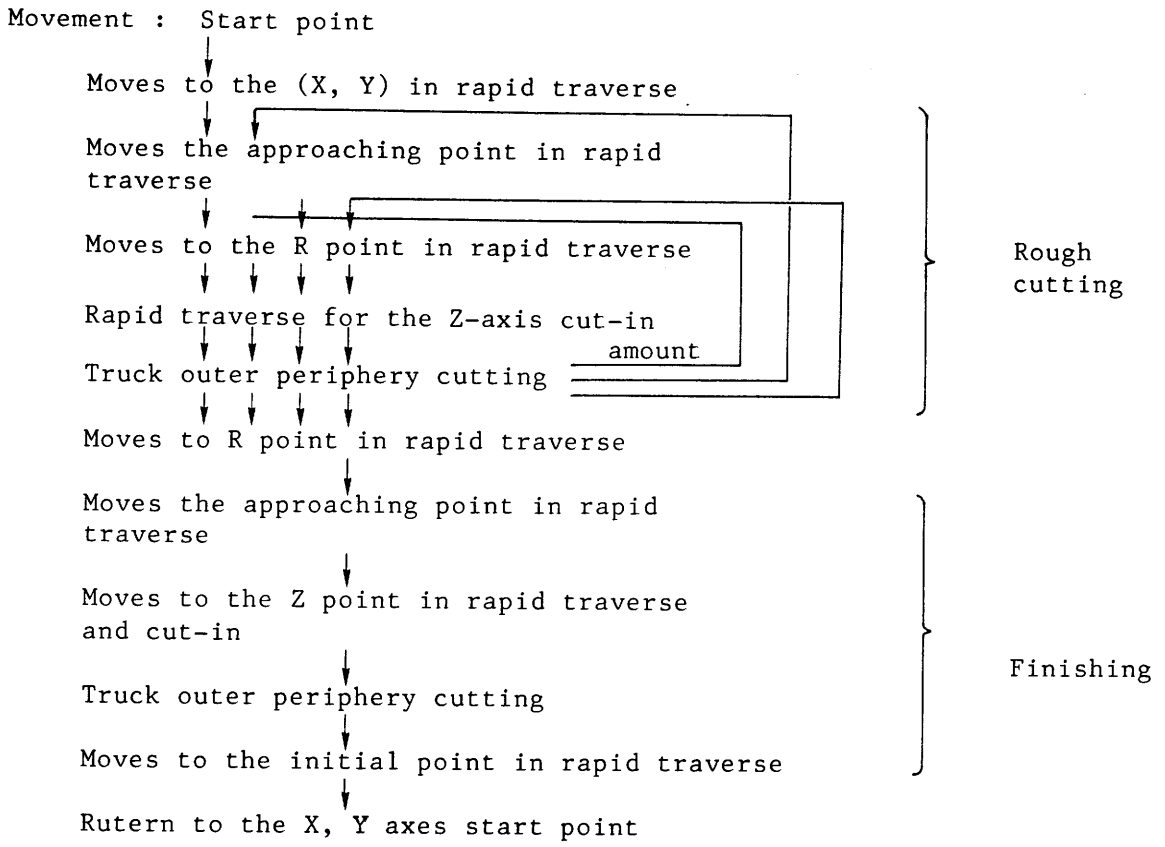


(Approaching point)

However, - - - ► Rapid traverse

——► Cutting rapid

Tool center path : 0 →1 →2 →a →b →3 →4 →5 →6 →c →P1→ d
→3 →4 →5 →6 →e →P2 →f
→7 →8 →9 →10 →g →P2 →h
→7 →8 →9 →10 →i →P3 →j
→11 →12 →13 →14 →k →0



(3) Precautions

(a) The numerical value of the address E and U shall be commanded without decimal point (in case of Millimetric system).

(b) When I = 0, J = 0 are commanded, an alarm occurs.

(c) When P = 0, an alarm occurs.

(d) When the address is omitted, the radius of circular arc shall be regarded as cutting allowance.

(e) The values that the inner periphery of truck cutting can be obtained shall be commanded for the numerical values of the addresses such as A, C, K and Q the tool offset amount (D).

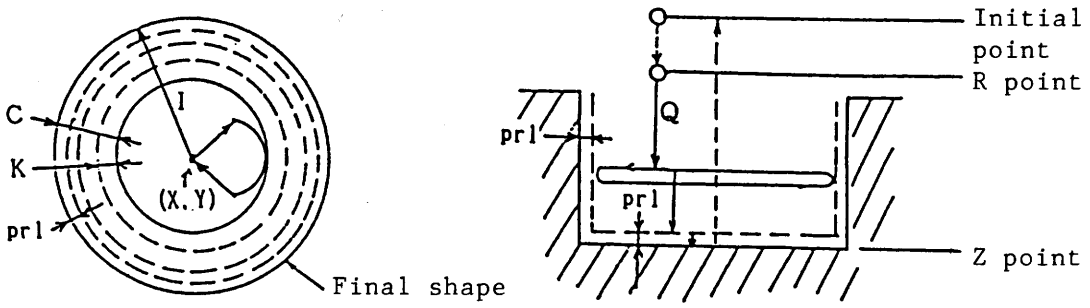
7. True circle (G333)

A series of operations cutting the inner periphery of a true circle by using end mill can be commanded in one block.

The below explanation is for G17 (XY plane).

(1) Command form

G333 X_Y_Z_R_I_Q_C_K_D_U_V_W_E_F_;



(X, Y) : Reference point of X-axis and Y-axis.

Z : Z point

R : R point

I : Circle radius of the final shape.

Q : Cutting depth per cutting of Z-axis.

C : Cutting allowance.

K : Cutting width per cutting (XY plane).

D : Tool offset No.

U : Spindle speed for finishing (When omitted, S already commanded.)

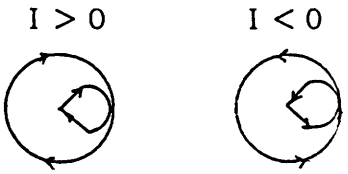
V : Cut-in speed of Z-axis (When omitted, cutting feed rate is F.)

W : Releasing speed of the approcahing circle (When omitted, cutting feed rate is F.)

E : Feed rate for finishing (When omitted, cutting feed rate is F.)

F : Cutting feed rate.

pr1: Cutting allowance (Set to the parameter No.6224.)



(Note 1) When E is omitted, the feed rate for finishing becomes F × override (parameter No.6207) for finishing.

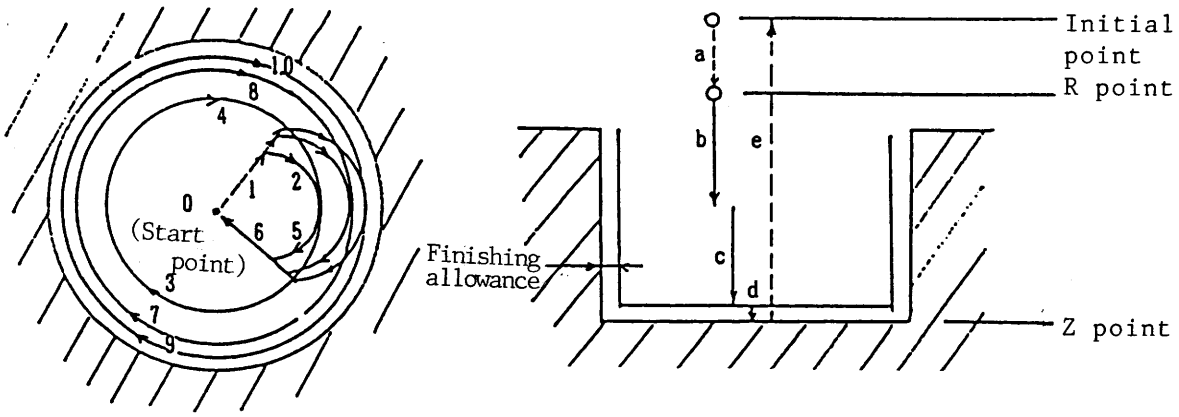
(Note 2) When the numerical value of I is positive, a circular arc becomes CW, and when it is negatuve, the circular arc becomes CCW. And when I = 0, an alarm occurs.

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

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

(2) Program example

```
G17;
G90 G333 X50. Y-100. Z-50. R-10. Q20. I50.
C15 K8. D10 F200;
```



However, - - - → Rapid traverse

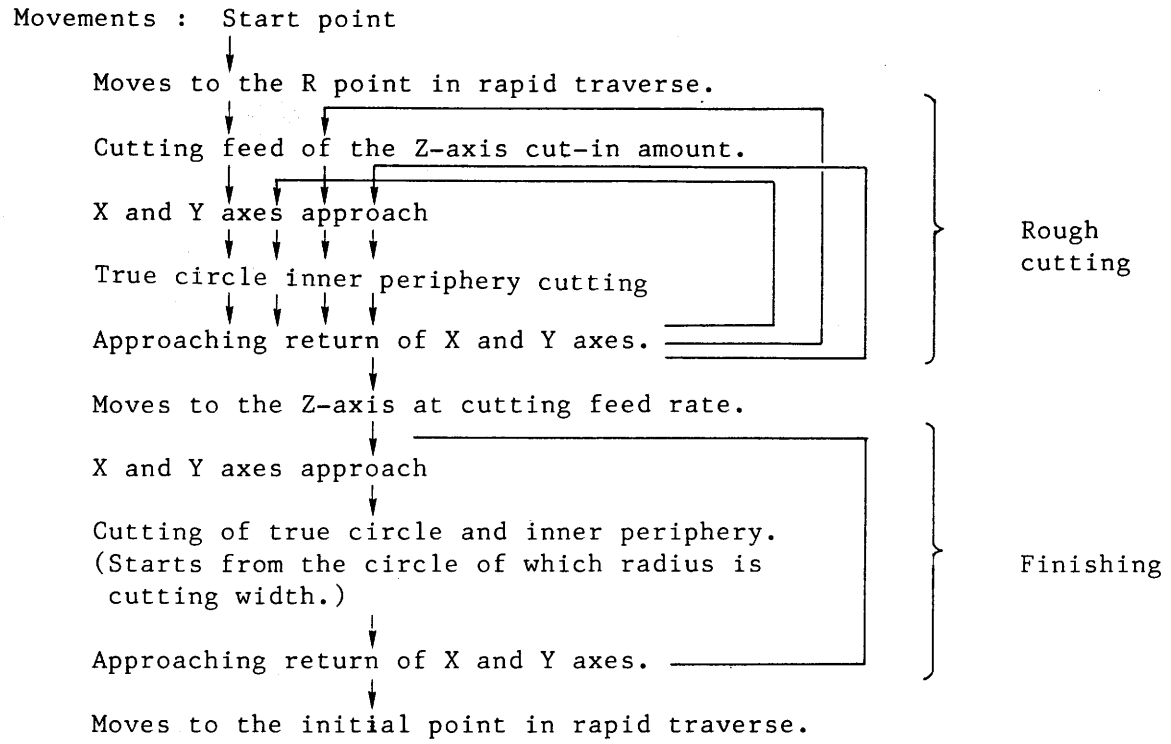
——→ Cutting feed

Tool center path : 0 →a →b →1 →2 →3 →4 →5 →6 →c

→1' →2' →7 →8 →5' →6' →d

→ Bottom face finishing (It starts by the circle of which radius is the cutting width.)

→1' →2' →9 →10 →5' →6' →e

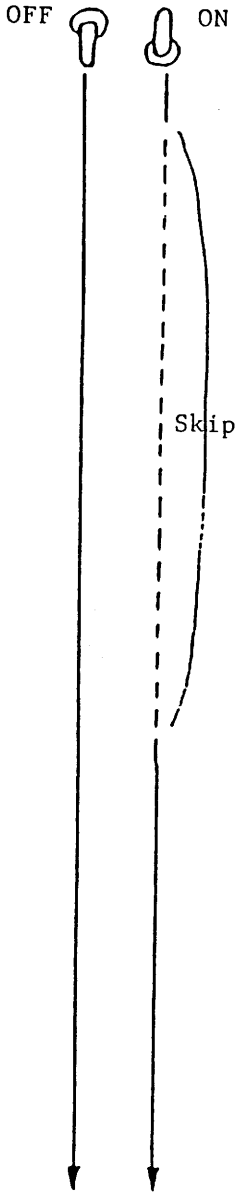


(3) Precaution

(a) The range moving in rapid traverse for approaching becomes automatically cutting feed rate, when rapid traverse is designated in the cutting range by setting cutting depth (K) and offset amount (D).

8. Other functions

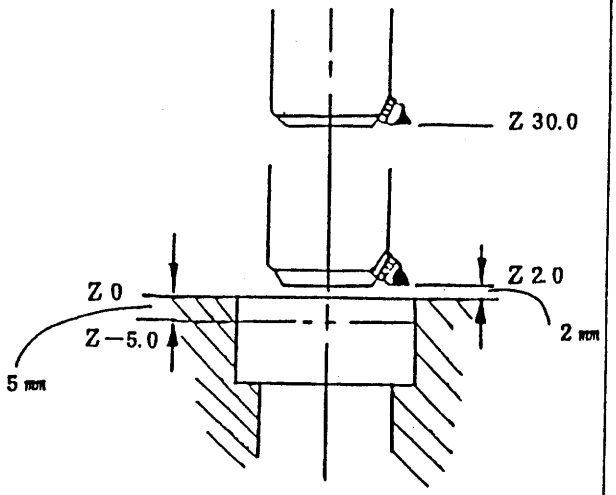
8-1 Optional block skip



This function makes the command ineffective for the block including "/" slash on the program.

By ON/OFF switch, intention is expressed.

```
N101 G54G90X100.0Y150.0S500T03 *
/N102 G43Z30.0H02*          Approach up to Z-axis 30mm
/N103  M08 *                Coolant liquid ON
/N104  M03 *                Spindle turn ON
/N105 G98G86R5.0Z-5.0F50 *  Cutting up to the inlet 5mm by boring
/N106 G80M05 *              Canned cycle cancel spindle stop
/N107 G91G28Z0M09 *         Return to Z-axis first reference point, coolant OFF.
/N108 G49 *                 G43 length compensation cancel
/N109 M00 *                 Hold-diameter measurement, tool-nose blade adjustment
N110 G90G43Z30.0H02  *
N111 M08 *
N112 M03*
N113 G98G86R2.0Z-30.0F50 *
N114  X150.0Y200.0 *
N115 X200.0 *
```



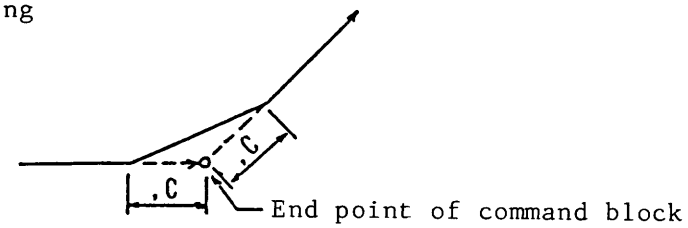
8-2 Arbitrary angle chamfering and corner R (, C, R)

Chamfering or corner R can be inserted by commanding ",C" or ",R" for linear interpolation or circular interpolation.

(1) Command form

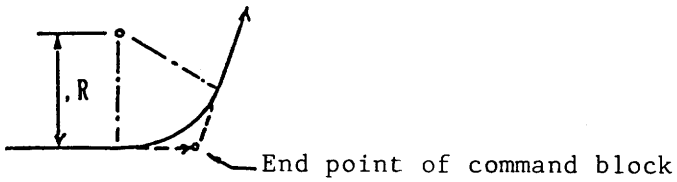
(a) Arbitrary angle chamfering

{ G01 / G02 / G03 } ..., C_;



(b) Arbitrary angle corner R

{ G01 / G02 / G03 } ..., R_;



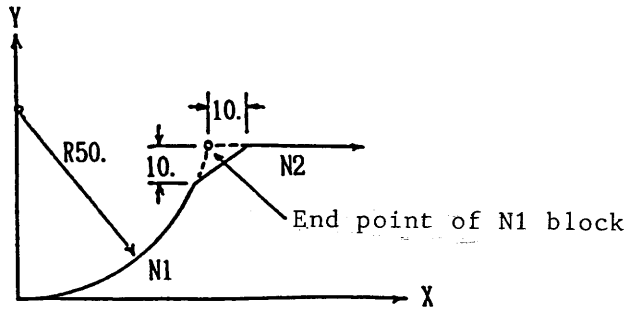
(2) The plane of the arbitrary angle chamfering corner R shall be designated by the plane selections (G17, G18 and G19).

(3) The block next to the block commanding the arbitrary angle chamfering corner R (, C, R) shall be commanded by either linear interpolation or circular interpolation.

(4) Program example

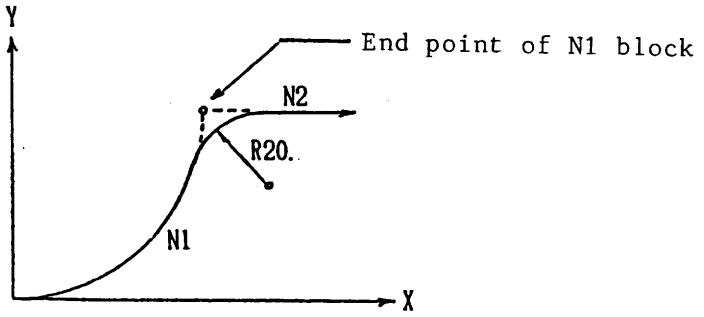
(a) Arbitrary angle chamfering

```
   G17 G54 G90 G00 X0 Y0;
N1 G03 X50. Y50. R50. ,C10. F200;
N2 G01 X90.;
```



(b) Arbitrary angle corner R

```
   G17 G54 G90 G00 X0 Y0;
N1 G03 X50. Y50. R50. ,R20. F200;
N2 G01 X90.;
```



(5) Precautions

(a) When the plane is changed by commanding the plane selections (G17, G18 and G19), an alarm occurs.

(b) The single block stop becomes the end point of the chamfering corner R block newly inserted.

(c) When the following commands are given to the command block of the ",C" or "R" and the next block, an alarm occurs.

G92, G52, G54 ~ G59,
G28, G29, G30, G53,
G31, G45 ~ G48

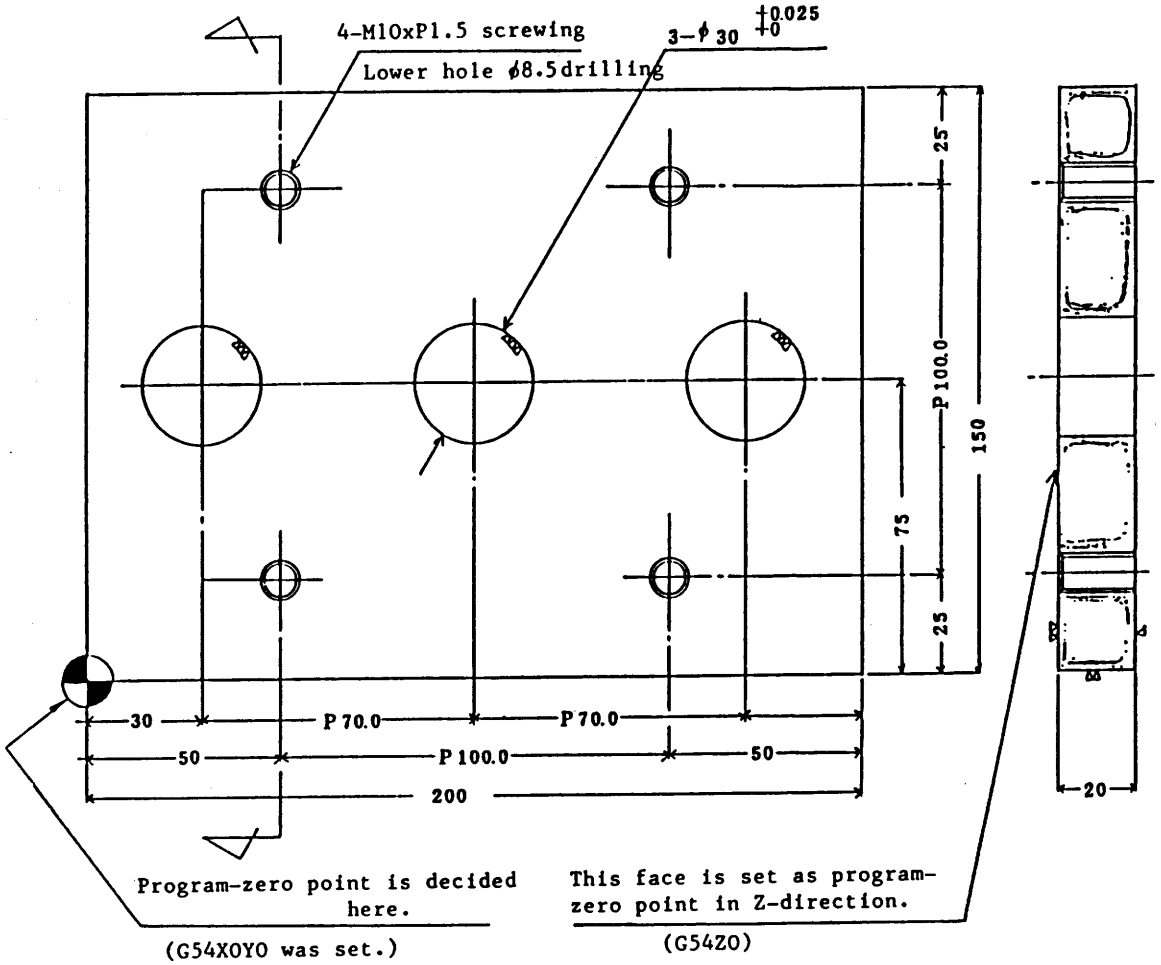
(d) When the angle difference between the command block of ",C" or "R" and the next block is within ±1%, the chamfering corner R becomes ineffective.
In case of circular arc, however, the angle difference shall be obtained from the tangent at the intersecting point.

(e) When no axial travel command in the plane is given the command block of ",C" or "R" and the next block, an alarm occurs for the chamfering corner R.

(f) When the original traveling range is exceeded through inserting either chamfering or corner R, an alarm occurs.

9. Practical example of program

9-1 Machining diagram plate FC30



9. Practical example of program

9-2 Selection of machining position

Premise

ⓐ Bottom face and surrounding 4 faces are already machined by previous process.

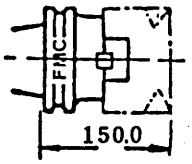
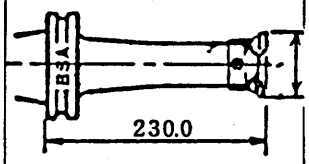
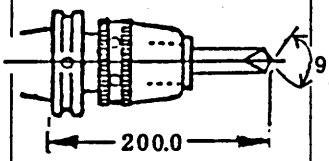
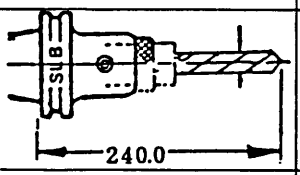
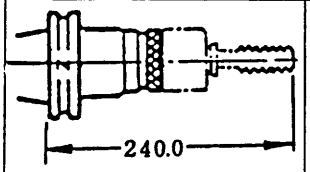
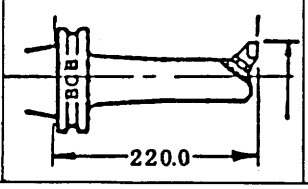
ⓑ ϕ30 boring hole is punched at bottom hole ϕ25.

① Face-cutting

② M10 tapping

③ ϕ30 boring

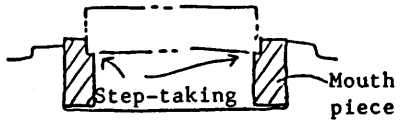
9-3 Setting of selected cutting condition of tool-cutter

Machining sequence	Name dimension	Speed (rpm)	Feed rate	Tool sketch	Compensation No.	Compensation No.
1	ϕ95 Face-cutter	S335 F335	V100m/min 1.0mm/rev		H01	T01
2	ϕ30 Boring (roughing) (2-blades)	S800 F160	V75m/min 0.2mm/rev		H02	T02
3	ϕ18 Center tool	S1200 F120	— 0.1mm/rev		H03	T03
4	ϕ8.5 Drill	S820 F160	V2.2m/min 0.2mm/rev		H04	T04
5	M10 x P1.5 Tapping	S320 F480	V10m/min 1.5mm/rev		H05	T05
6	ϕ30 Boring (finishing)	S1600 F96	V150m/min 0.06mm/rev		H06	T06

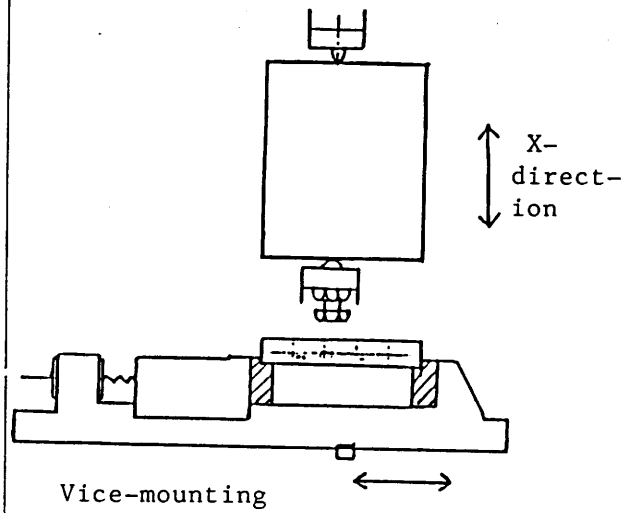
9. Practical example of program

9-4 Mounting method

Clamp with vice-mouth piece by taking a step



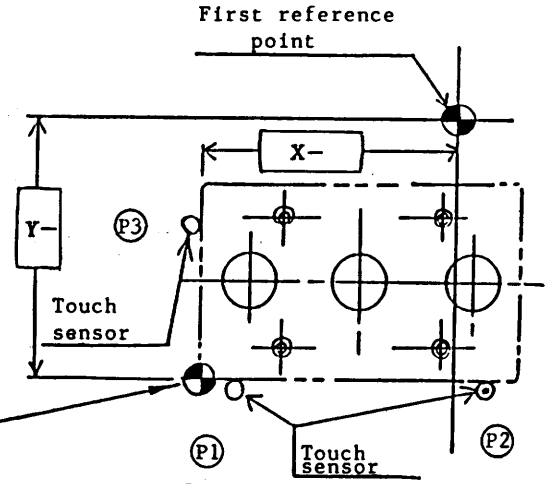
Lateral direction (X-axial direction) should be decided by stopper.



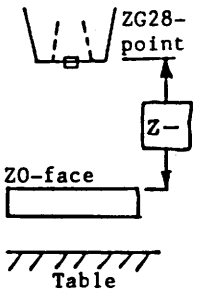
9-5 Relation with work-coordinate system

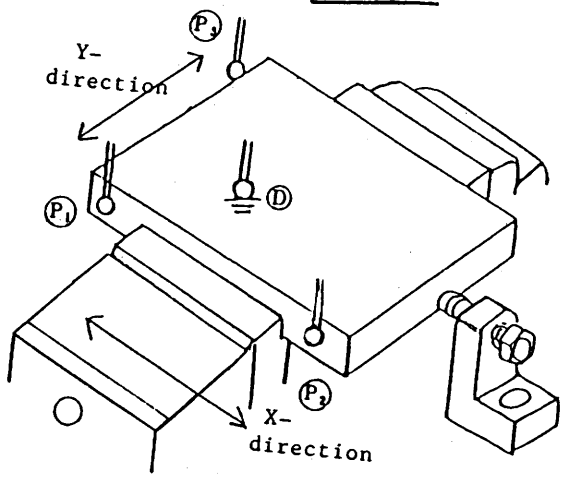
By the specified procedure and work-standard face (manual), make touch sensor contact with Ⓟ1 → Ⓟ2 → Ⓟ3 , thereby, [X-] , [Y-] can be set automatically to the desired work coordinate (G54 ~ G59)

In this case, program-zero point is here.



Make touch sensor contact to 1 point (top face of workpiece), thereby, [Z-] is set at work-coordinate (G54~G59) of CRT-screen. That is, ZO-face is memorized in machine.

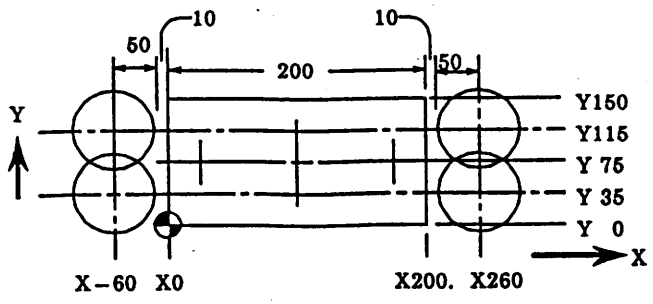




9. Practical example of program

9-6 ϕ95 Face cutter

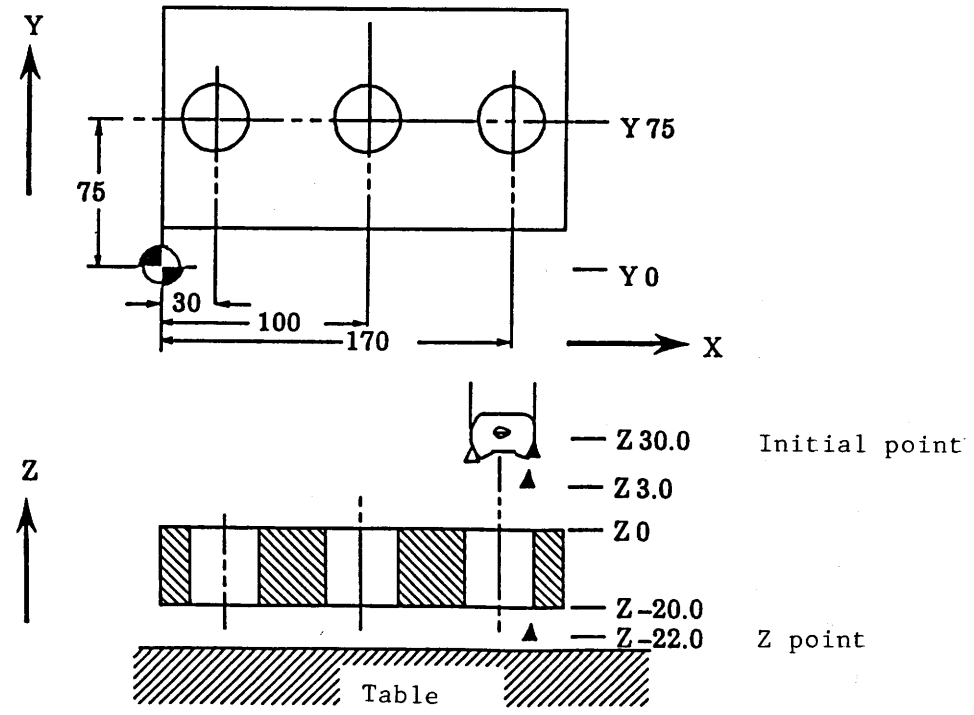
O1968 (MODEL VK, VM FC30) *	Program No.
G91 G28 Z0 M31 *	Z-axis machine zero point return, chip conveyor starts.
G91 G28 X0 Y0 *	Return-lamp lights on X-, Y-axis.
G00 G17 G40 G49 G64 G80 G90 G98 *	Set the inside of NC-head at initial state.
M01 *	Optional stop
N101 (ϕ95 CUTTER T01 H01) *	Sequence No. ( ): memo-writing
T01 M06 *	Tool No. 1 to spindle by tool change
G54 *	Work coordinate G54
G68 *	Coordinate rotation easy setter effective.
G90 G00 X260.0 Y115.0 S320 T02 *	Absolute spindle rotation 320, next tool T02
G43 Z30.0 H01	Tool length compensation No. H01 is effective. Tool nose is positioned to Z30.0.
M03 *	Start of spindle forward turn.
G01 Z0 F3000 *	Tool nose reach to Z0 face (finishing face).
G01 X-60.0 Y115.0 F320 *	Cutting, feeding 320mm/min. to column side of chip
G00 Z30.0 *	Escape of Z-axis up to the height of approach once.
X260.0 Y35.0 *	X_Y_ positioning with rapid traverse
G01 Z0 F3000 *	Reaching Z-axis Z0 face with speed 3000mm/min.
G01 X-60.0 Y35.0 F320 *	Cutting
G69 *	Coordinate rotation cancel
M01 *	Stop with switch ON (optional stop) while passing with OFF.



9. Practical example of program

9-7 φ30 Boring

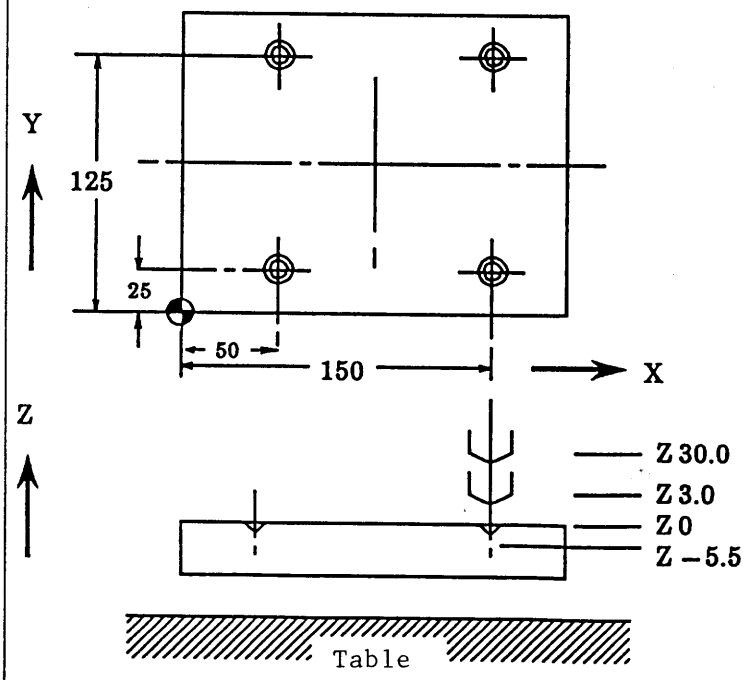
N102 (φ30 BORING T02 H02) *	The inside ( ) of sequence No. means memo.
T02 M06 *	Hold at spindle by T02 tool ATC operation.
G54 *	Work coordinate G54
G68 *	Coordinate rotation effective
G90 G00 X30.0 Y75.0 S800 T03 *	Absolute, X_Y_ Spindle speed selection 800rpm, T03 call
G43 Z30.0 H02	Tool length compensation No.2 plus side offset. Tool nose position : Z30.0
M03 *	Spindle forward turn ON.
G98 G81 R3.0 Z-22.0 F160 *	Boring canned cycle, initial point return, cutting start position 3mm frontward, -22.0 depth
X100.0 Y75.0 *	X_Y_ positioning. Other operations are the same as those of previous block.
X170.0 Y75.0 *	X_Y_ (Same as the above)
G69 *	Coordinate rotation cancel
G80 *	Canned cycle cancel
M01 *	Optional stop



9. Practical example of program

9-8 φ18 Center

N103 (φ18 CENTER T03 H03) *	The inside ( ) of sequence No. means memo.
T03 M06 *	With ATC, T03 ,.. to spindle
G54 *	
G68 *	Coordinate rotation effective
G90 G00 X50.0 Y125.0 S1200 T04 *	During X_Y_ positioning operation, spindle speed is selected. 1200rpm, T04 call
G43 Z30.0 H03	Offset of tool length H03 alone to Z plus side
M03 *	Spindle turn ON
G99 G81 R3.0 Z-5.5 F120 L0 *	With R-point return and drill canned cycle L0, no operation is made by this block.
M98 P100 *	To subprogram of hole position coordinate
G80 *	Canned cycle cancel
M01 *	Optional stop

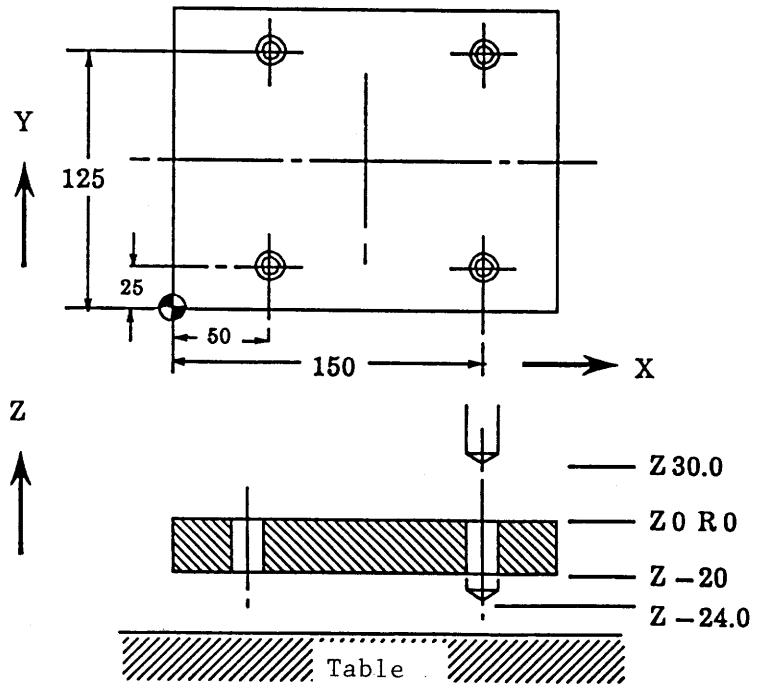


Subprogram of hole position
O100 *
G54 G90 X50.0 Y125.0 *
X50.0 Y25.0 *
X150.0 Y25.0 *
X150.0 Y125.0 *
M99 *

9. Practical example of program

9-9 ϕ8.5 Drill

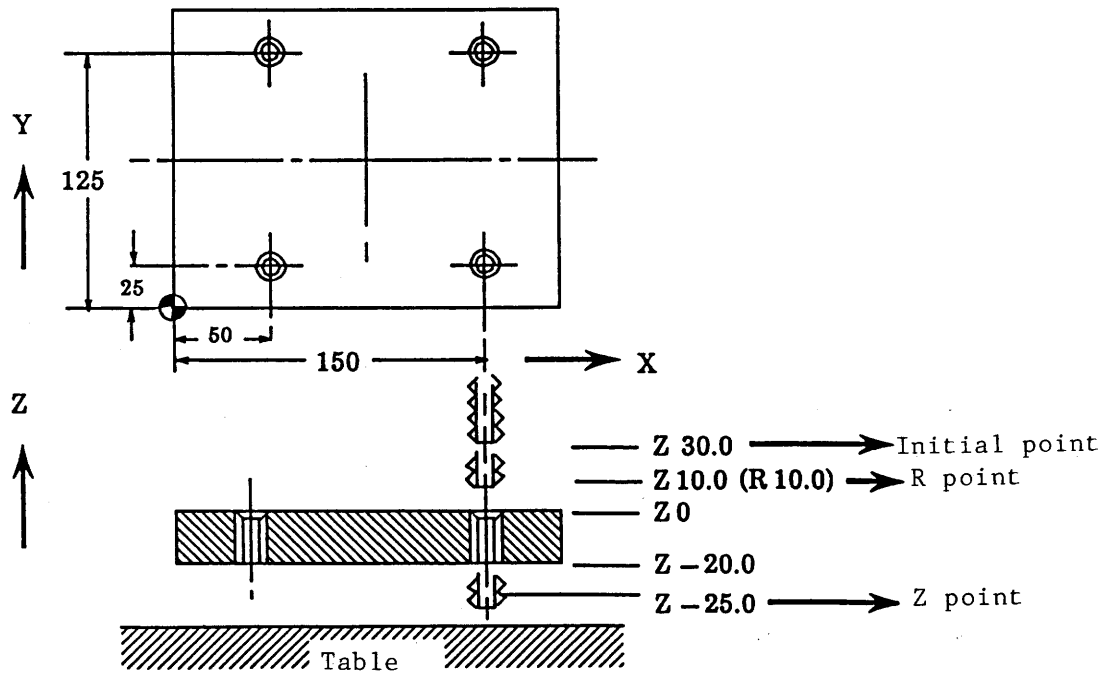
N104 (ϕ8.5 DRILL T04 H04) *	Sequence No. inside ( ) means this process memo.
T04 M06 *	To T04 tool spindle
G54 *	
G68 *	Coordinate rotation effective
G90 G00 X50.0 Y125.0 S820 T05 *	820rpm selection and T05 call during the movement of work coordinate G54.
G43 Z30.0 H04	Offset of H04 value to Z-axis plus side Tool nose to Z30.0 position
M03 *	Spindle turn ON
G99 G81 R0 Z-24.0 F164 L0 *	With R point return and drill canned cycle L0, no operation is made by this block.
M99 P100 *	To subprogram of hole position coordinate
G69 *	Coordinate rotation cancel
G80 *	Canned cycle cancel
M01 *	Optional stop



9. Practical example of program

9-10 M10 Tapping

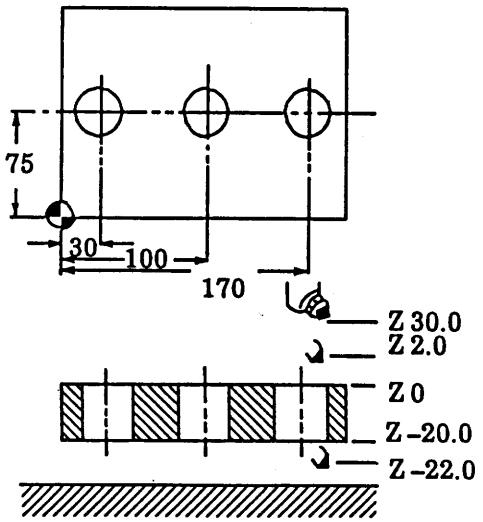
N105 (M10 X P1.5 TAP T05 H05) *	Sequence No. in ( ) means TAP-process memo.
T05 M06 *	Tool No. 05 to the spindle
G54 *	
G68 *	Coordinate rotation effective
G90 G00 X50.0 Y125.0 S320 T06 *	Spindle speed selection 320rpm, next tool T06 call
G43 Z30.0 H05	Z-axis plus side offset, shift of tool length amount entering into H05.
M03 *	Spindle forwarding turn start
G98 G84 R10.0 Z-25.0 F480 L0 *	Initial point return, tap canned cycle hole position to the same subprogram as in lower hole.
M98 P100 *	Initial point return, tap canned cycle hole position to the same subprogram as in lower hole.
G69 *	Coordinate rotation cancel
G80 *	With switch ON, program stops.
M01 *	With switch OFF, passing is made with no relation to M01.



9. Practical example of program

9-11 φ30 Boring finishing

N106 (φ30 BORING F T 06 H06) *	Sequence No. in ( ) means memo.
T06 M06 *	T06 tool is held to the spindle.
G54 *	
G68 *	Coordinate rotation effective
G90 G54 G00 X30.0 Y75.0 S1600 T01 *	Initial tool T01 call
G43 Z30.0 H06	Tool length of length compensating H06
M03 *	Spindle forwarding turn
G99 G76 R2.0 Z-22.0 Q0.5 F96 *	R point return with canned cycle of finishing boring
X100.0 Y75.0 *	The above operation X_Y_ positioning, thereafter
X170.0 Y75.0 *	Same as the above
G69 *	Coordinate rotation cancel (easy setter)
G80 *	Canned cycle cancel
M01 *	Optional stop
G91 G28 Z0 M05 *	Z-axis zero point return, spindle turn stop
G28 X0 Y0 *	X-, Y-axis zero point return
M30 *	End of tape automatic head forwarding



M06 (ATC canned cycle)	
VK	VMII
M15 *	M15
G91 G30 Z0 *	G30 G91 Z0
G30 X0 Y0 M19 *	G28 G91 X0 Y0
(T××) M06 *	M19
	(T××)
	G30 G91 X0
	M06
	G28 X0

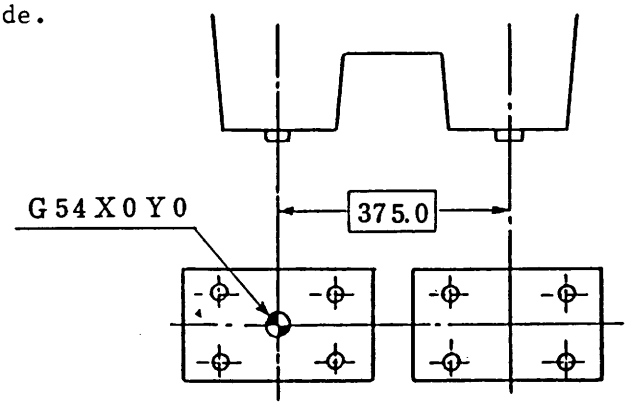
9. Practical example of program

9-12 Program of 2 spindles

① Program is the same as that of single spindle (standard machine).

② Movement of both X-Y and Z-axis (spindle unit) does not change with that of single spindle (standard machine).

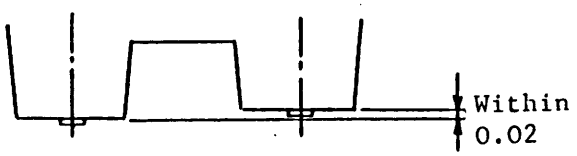
③ Set the work-coordinate system with the standard of spindle of one side.



④ Tool-change program is the same as that of standard machine. (Whereas, it has a premise that the same tool is set to the same address of the left and right magazines.)

⑤ In case of simultaneous machining of the same 2 workpieces, be sure to set { a tool length, b tool diameter } to the same dimensions.

⑥ The difference of Z-direction (gage line) of 2 spindles is within 0.02, then, its difference can be regarded as "0" by ordinary machining.



10. Attached list

10-1 List of G function (preparatory function) (SEICOS - MIII)

Code	Group	Function	Remarks
G00	01	Positioning	
G01		Linear interpolation	
G02		Circular/helical interpolation CW	
G03		Circular/helical interpolation CCW	
G04	00	Dwell	
G05		High speed distribution cutting	
G07		Virtual axis interpolation	
G09		Exact stop	
G10		Data setting	
G11		Data mode setting cancel	
G15	20	Polar coordinate command cancel	
G16		Polar coordinate command	
G17	02	Xp Yp plane However, Xp: X-axis or its parallel axis	
G18		Zp Xp plane Yp: Y-axis or its parallel axis	
G19		Yp Zp plane Zp; Z-axis or its parallel axis	
G20	06	Inch input	
G21		Millimetric input	
G22	04	Stored stroke check ON	
G23		Stored stroke check OFF	
G27	00	Reference point return check	
G28		Reference point return	
G29		Return from reference point	
G30		2nd, 3rd and 4th reference point return	
G31		Skip function	
G33	01	Thread cutting	
G34		Variable lead thread cutting	

Code	Group	Function	Remarks
G37	00	Tool length automatic measurement	
G38		Tool radius compensation vector keep	
G39		Tool radius compensation corner circular arc	
G40	07	Tool radius compensation cancel/3 dimensional tool offset cancel	
G41		Tool radius compensation left/3 dimensional tool offset	
G42		Tool radius compensation right	
G43	08	Tool length compensation "+"	
G44		Tool length compensation "−"	
G45	00	Tool offset increase	
G46		Tool offset decrease	
G47		Tool offset double increase	
G48		Tool offset double decrease	
G49	08	Tool length compensation cancel	
G50	11	Scaling cancel	
G51		Scaling	
G52	00	Local coordinate system setting	
G53		Machine coordinate system selection	
G54	12	Selection of work coordinate system 1	
G55		Selection of work coordinate system 2	
G56		Selection of work coordinate system 3	
G57		Selection of work coordinate system 4	
G58		Selection of work coordinate system 5	
G59		Selection of work coordinate system 6	
G60	00	Unidirectional positioning	1 group by parameter change

Code	Group	Function	
G61	13	Exact stop mode	
G62		Automatic corner override mode	
G63		Tapping mode	
G64		Cutting mode	
G65	00	Macro call	
G66	14	Macro modal call	
G67		Macro modal call cancel	
G68	16	Coordinate rotation	
G69		Coordinate rotation cancel	
G70	00	Bolt hole cycle	
G71		Arc	
G72		Arc	
G73	09	Peg drilling cycle	
G74		Reverse tapping cycle	
G76		Fine boring cycle	
G77	00	Grid cycle	
G80	09	Canned cycle cancel	
G81		Drilling cycle, spot boring	
G82		Drilling cycle, counter boring	
G83		Peg drilling cycle	
G84		Tapping cycle	
G85		Boring cycle	
G86		Boring cycle	
G87		Back boring cycle	
G88		Boring cycle	
G89		Boring cycle	
G90	03	Absolute command	
G91		Incremental command	

Code	Group	Function	Remarks
G92	00	Work coordinate system change/Max. spindle speed setting	
G93	05	Inverse time feed	
G94		Feed per minute	
G95		Feed per revolution	
G96	17	Constant surface speed control	
G97		Constant surface speed control cancel	
G98	10	Canned cycle initial level return	
G99		Canned cycle R point level return	
G113	21	Oscillation mode ON	
G114		Oscillation mode OFF	
G120	22	Polar coordinate interpolation mode cancel	
G121		Polar coordinate interpolation mode	
G130	18	Tool life control OFF	
G131		Tool life control ON	
G203	00	High speed machining resister start	
G204		High speed machining resister end	
G206		Tool release amount setting	
G232	01	Exponential function interpolation CW	
G233		Exponential function interpolation CCW	
G240	24	Machining plane 0 selection (Machining plane selection)	
G241		Machining plane 1 selection	
G242		Machining plane 2 selection	
G243		Machining plane 3 selection	
G244		Machining plane 4 selection	
G245		Machining plane selection (Corresponding to arbitrary angle for horizontal and vertical)	

Code	Group	Function	Remarks
G248	26	Axis change/3 dimensional coordinate change ON	
G249		Axis change/3 dimensional coordinate change cancel	
G251	00	Multiple buffer	
G264	25	Tool nose interference check ON	
G265		Tool nose interference check OFF	
G271	00	Cylindrical interpolation	
G301		Floating reference point return	
G302		True circle cutting ID CW	
G303		True circle cutting ID CCW	
G304		True circle cutting OD CW	
G305		True circle cutting OD CCW	
G322		Square side surface outer cutting CW	
G323		Square side surface outer cutting CCW	
G324		Square surface	
G325		Square surface one side sizing	
G326		Square surface both sides sizing	
G327		Inner circle (pocket cutting)	
G328		Inner side of square (pocket cutting)	
G329		Inner truck (pocket cutting)	
G330		Outer circle (pocket cutting)	
G331		Outer side of square (pocket cutting)	
G332		Outer truck (pocket cutting)	
G333		True circle (pocket cutting)	
G501	15	Programmable mirror image cancel	
G511		Programmable mirror image	

Code	Group	Function	Remarks
G540~ G599	12	Additional work coordinate system selection (60 pairs)	
G611	00	Acceleration/deceleration before interpolation	
G661	14	Macro modal call B	
G741	09	Direct tapping cycle	
G841		Counter direct tapping cycle	
G921	00	Work coordinate system preset	

(Note 1) *1 Disabled to correspond by the reserved G code.

(Note 2) ◤ Mark is attached to every group.
In the state of reset, ◤ mark side is selected.

10-2 List for M function (miscellaneous functions) (VM,VK)

M	Name of function	M	Name of function
00	Program stop	26	Melody hone 1
01	Optional stop	27	Melody hone 2
02	End of program	*28	
03	Spindle forward turn	*29	
04	Spindle reverse turn	30	End of tape
05	Spindle stop	31	Chip conveyor start
06	Tool change A.T.C	*32	
*07	Oil mist start	*33	
08	Flat coolant start	*34	
09	Coolant mist stop	*35	Auto-start ON
*10	Oil mist posture 1	*36	Auto-start OFF
*11	Oil mist posture 2	37	
*12	Work counter	*38	
13	Spindle forward turn & coolant start	*39	
14	Spindle reverse turn & coolant start	*40	Tool nose air blow ON Tool length measuring cover open
15	Stop of M13, M14	*41	Tool nose air blow OFF Tool length measuring cover close
*16	Measuring air blow ON	42	
*17	Measuring air blow OFF	43	
*18	Measuring spindle orientation	44	
19	Spindle orientation	*45	
*20		*46	
		*47	Jet coolant start
22		48	Feed rate override effective
23		49	Feed rate override 100%
24		*50	Oil hole coolant start
25		*51	Tool preparation check

* Mark shows a option.

M	Name of function	M	Name of function
*52	Tool damage detection	76	
*53		77	
*54		*78	Additional axis clamp
*55	M56 cancel	*79	Additional axis unclamp
*56	Tool life feed hold	*80	Tool nose air blow ON
*57		81	
*58		82	
*59	M51 cancel	83	
*60	Pallet change A.P.C	84	
*61	Transfer ready check	85	
62		86	
*63	Pallet position discrimination	*87	
*64		*88	Door open
65		89	Door close
66		90	
67		91	
*68	Additional axis clamp (change with M78 and op)	92	
*69	Additional axis unclamp (change with M79 and op)	93	
*70	M70 output	94	
*71	M71 output Index stand start	95	
*72	M72 output	*96	Macro interrupt ON
*73	M73 output	*97	Macro interrupt OFF
*74	Skip selection OFF	90	Subprogram call
*75	Skip selection ON	99	End of subprogram

List for M-function (miscellaneous functions)(HG)

M	Name of function	M	Name of function
00	Program stop	*26	Melody hone 1
01	Optional stop	27	Melody hone 2
02	End of program	*28	
03	Spindle forward turn	*29	
04	Spindle reverse turn	30	End of tape
05	Spindle stop	31	Chip conveyor start
06	Tool change M64 cancel	*32	
*07	Oil mist start	*33	
08	Flat coolant start	*34	
*09	Coolant mist stop	*35	Auto-start ON (auto-return effective)
*10	Oil mist posture 1	*36	Auto-start OFF (auto-return ineffective)
*11	Oil mist posture 2	*37	Macro interrupt effective
*12	Work counter 1	*38	
13	Spindle forward turn & coolant start	*39	
14	Spindle reverse turn & coolant start	*40	Tool length measuring arm out
15	Spindle & coolant stop	*41	Tool length measuring arm in
*16	Measuring air blow ON	42	
*17	Measuring air blow OFF	43	
*18	Measuring spindle orientation	44	
19	Spindle orientation	*45	
*20		*46	
		*47	Jet coolant start
22		48	Feed rate override effective
23		49	Feed rate override 100%
24		*50	Oil hole coolant on (through)
25		*51	Tool preparation check

* Mark shows a option.

M	Name of function	M	Name of function
*52	Tool damage detection	76	
*53		77	
*54		78	Table clamp
*55	M56 cancel	79	Table unclamp
*56	Tool life feed hold	80	
*57		81	
*58		82	
*59	M51 cancel	83	
60	A.P.C cycle 1	84	
61	A.P.C cycle 2	85	
62	A.P.C cycle 3	86	Measuring NG Tool damage
63		87	A.P.C door right open
*64		88	A.P.C door left open
65		89	A.P.C door close
66	Pallet clamp	90	
67	Pallet unclamp	91	
*68	Outside output M68	92	
*69	Outside output M69	93	
*70	Outside output M70	94	
*71	Outside output M71	95	
*72	Outside output M72	*96	Custom macro interrupt effective
*73	Outside output M73	*97	Custom macro interrupt ineffective
*74	Skip selection OFF	98	Subprogram call
*75	Skip selection ON	99	Main progrm call

List for M-function (miscellaneous function) (VG)

B : Standard
O : Option

M code	Function	Classi-fication
000	Program stop	B
001	Optional stop	B
002	End of program	B
003	Spindle forward turn	B
004	Spindle reverse turn	B
005	Spindle stop	B
006	Tool change (ATC)	B
007	Mist coolabt start	O
008	Flat coolant start	B
009	Coolant mist stop	B
010	Oil mist posture 1	O
011	Oil mist posture 2	O
012	Work counter 1	O
013	Spindle forward turn and coolant start	B
014	Spindle reverse turn and coolant start	B
015	Spindle and coolant stop	B
016	Measuring air blow ON	O
017	Measuring air blow OFF	O
018	Measuring spindle orientation	O
019	Spindle orientation	B
021		
022		
023		
024		
025		

M code	Function	Classi-fication
026	Melody selection 1	B
027	Melody selection 2	B
028		
029		
030	End of tape	B
031	Chip conveyor	B
032		
033		
034		
035	Automatic start ON (auto-return effective)	O
036	Automatic start OFF (auto-return ineffective)	O
037		
038		
039		
040	Tool nose air blow ON/ Tool length measuring cover OPEN	O
041	Tool nose air blow OFF/ Tool length measuring cover CLOSED	O
042		
043		
044		
045	Spare tool EXIST	O
046	Spare tool NON	O
047	Jet coolant start	O
048	Feed rate override effective	B

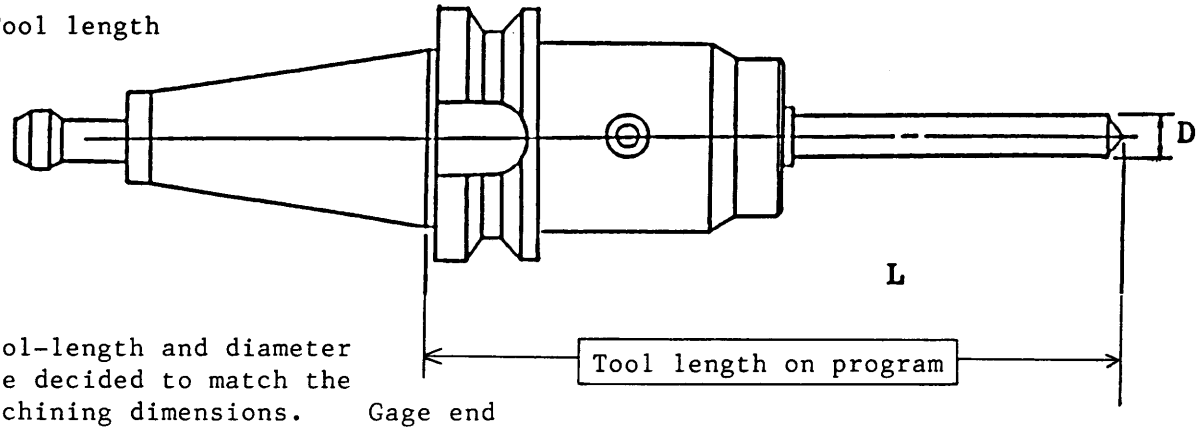
M code	Function	Classi-fication
049	Feed rate override 100%	B
050	Oil hole coolant ON (through coolant)	0
051	Tool preparation check	0
052	Tool damage detection	0
053		
054		
055	M56 cancel	0
056	Tool life feed hold	0
057		
058		
059	M51 cancel	0
060	Pallet change (APC) door OPEN	0
061	APC pallet carry-out	0
062	APC pallet carry-in	0
063		
064	APC carrier right travel	0
065	APC carrier left travel	0
066	Pallet clamp	0
067	Pallet unclamp	0
068	Additional axis clamp (change with M78 and OP)	0
069	Additional axis unclamp (change with M79 and OP)	0
070	M70 output	0
071	M71 output, Index stand Index stand start	0
072	M72 output	0
073	M73 output	0
074	Skip selection OFF	0

M code	Function	Classi-fication
075	Skip selection ON	0
076		
077		
078	Additional axis clamp	0
079	Additional axis unclamp	0
080		
081		
082		
083		
084		
085		
086	Measuring NG Tool damage	0
087		
088	Door OPEN	0
089	Door CLOSED	0
090		
091		
092		
093		
094		
095		
096	Custom macro interrupt effective	0
097	Custom macro interrupt ineffective	0
098	Subprogram call	B
099	Main program call	B

When M code is changed due to special specifications etc., refer to the electric diagram.

10-3 Related items to the tool-set

Tool length



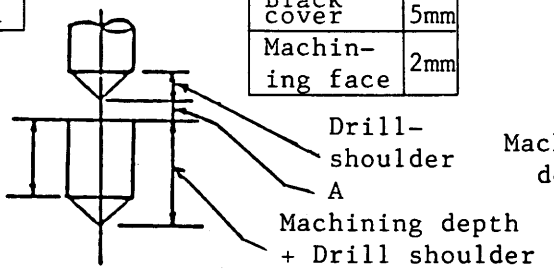
Tool-length and diameter are decided to match the machining dimensions.

Gage end
(matched to spindle end face)

Margin of tool nose and work face (A)

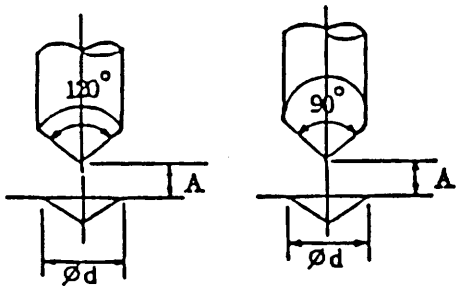
Drill

A	
Black cover	5mm
Machin-ing face	2mm

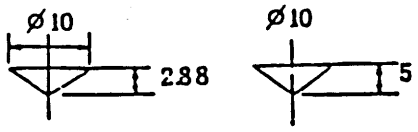


Centering drill

A	
Black cover	5mm
Machin-ing face	2mm



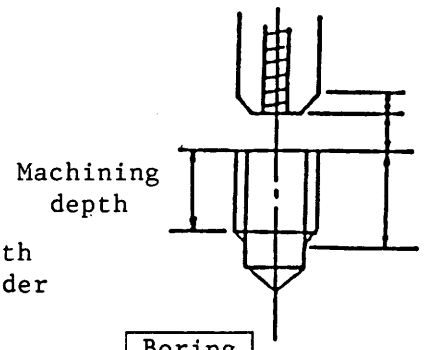
Chamfering size and cutting-in



at time of 120° at time of 90°

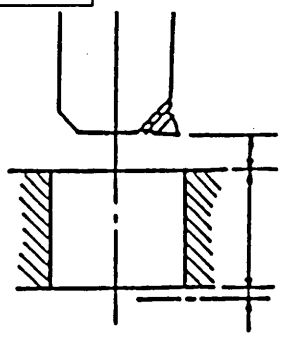
Tap

A	
Black cover	10mm
Machin-ing face	5mm



Boring

A	
Black cover	5mm
Machin-ing face	2mm



10-4 How to obtain the cutting condition

Spindle speed (rpm)

$$N = \frac{V}{31.4 \times D} \times 1000$$

N : Spindle speed (rpm)
V : Cutting speed (m/min)
D : Tool diameter (mm)

Example	3" Front milling	ϕ8 drill	M8 tap
	V = 120m/min	V = 18m/min	V = 9m/min
	D = ϕ76mm 6 blades	D = ϕ8	D = 8
	$N = \frac{120}{3.14 \times 76} \times 1000$	$N = \frac{18}{3.14 \times 8} \times 1000$	$N = \frac{9}{3.14 \times 8} \times 1000$
	= 502	= 716	= 358
	S500 500rpm	S710 710 rpm	S355 355rpm

Cutting feed rate

F = N x fr
F = N x P (tapping case pitch)
F = N x fz x t (case of front milling) End mill

F : Cutting speed (mm/min)
N : Spindle speed (rpm)
fr: Feed amount per one rotation (mm/rev)
fz: Feed amount per one blade
t : Number of blade

Example	3" Front milling	ϕ8 drill	M8 tap
	N = 500 rpm	N = 710 rpm	N = 355rpm
	fz = 0.12mm/1 blade	fr = 0.18mm/rev	p = 1.25
	t = 6 blades		
	F = 500 x 0.12 x 6	F = 710 x 0.18	F = 355 x 1.25
	= 360	= 128	= 444
	F360	F128	F444

10-5 List for standard cutting conditions

Increase/decrease can be made by mounting state, material to be cut, and tool length.

Tool name				Cast iron		Steel material		Aluminium	
				Cutting speed m/min	Feed mm/rev mm/min	Cutting speed m/min	Feed mm/rev mm/min	Cutting speed m/min	Feed mm/rev mm/min
5′face cutter		S	R	90	450	95	400	250	700
			F	120	320	120	250	320	600
ø100shell cutter		S	R	80	280	80	300	160	430
			F	90	190	80	190	160	300
Boring	ø30～55	S	R	60～75	0.15	60～75	0.12	90～115	0.1～0.15
			F	70～85	0.1	75～90	0.08	115～140	0.08
	ø60～100	S	R	70～80	0.15～0.25	80～90	0.15	130～150	0.12～0.2
			F	90～110	0.1～0.12	90～105	0.1	160～190	0.1
	ø100～200	S	R	70～80	0.25	75～80	0.2～0.25	160～195	0.2
			F	100～110	0.1～0.12	105～110	0.1	200～240	0.12
Drill	ø5～10	H		20	0.2	25	0.1～0.2	30～45	0.1～0.2
	ø10～20	H		25	0.3	25	0.2～0.25	50～55	0.2～0.25
		S		40～50	0.3				
	ø20～50	H		25	0.35	20～23	0.25	50～60	0.25
		S		50	0.3				
Tap		H		10～14		10～12		12～17	
Leamer		H		10～12	0.3	10～12	0.25～0.3	15～20	0.25～0.35
		S		11～16	0.3	11～16	0.25～0.3	15～20	0.25～0.35
Endmill		H		25～29	0.1～0.25	25～29	0.1～0.25	30～60	0.1～0.3
		S		35～50	0.1～0.25	30～50	0.1～0.25	50～80	0.15～0.3

(Note) R = Roughing, F = Finishing, S = Super-hard steel, H = High speed steel

10-6 List for tape code

ISO code										EIA code										Meaning
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1	
0			○	○		o				0			○			o				Numeral 0
1	○		○	○		o			○	1						o			○	〃 1
2	○		○	○		o		○		2						o		○		〃 2
3			○	○		o		○	○	3				○		o		○	○	〃 3
4	○		○	○		o	○			4						o	○			〃 4
5			○	○		o	○		○	5				○		o	○		○	〃 5
6			○	○		o	○	○		6				○		o	○	○		〃 6
7	○		○	○		o	○	○	○	7						o	○	○	○	〃 7
8	○		○	○	○	o				8					○	o				〃 8
9			○	○	○	o			○	9				○	○	o			○	〃 9
A		○				o			○	a		○	○			o			○	Address A
B		○				o		○		b		○	○			o		○		〃 B
C	○	○				o		○	○	c		○	○	○		o		○	○	〃 C
D		○				o	○			d		○	○			o	○			〃 D
E	○	○				o	○		○	e		○	○	○		o	○		○	? 〃 E
F	○	○				o	○	○		f		○	○	○		o	○	○		〃 F
G		○				o	○	○	○	g		○	○			o	○	○	○	〃 G
H		○			○	o				h		○	○		○	o				〃 H
I	○	○			○	o			○	i		○	○	○	○	o			○	〃 I
J	○	○			○	o		○		j		○		○		o			○	〃 J
K		○			○	o		○	○	k		○		○		o		○		〃 K
L	○	○			○	o	○			l		○				o		○	○	〃 L
M		○			○	o	○		○	m		○		○		o	○			〃 M
N		○			○	o	○	○		n		○				o	○		○	〃 N
O	○	○			○	o	○	○	○	o		○				o	○	○		〃 O
P		○		○		o				p		○		○		o	○	○	○	〃 P
Q	○	○		○		o			○	q		○		○	○	o				〃 Q
R	○	○		○		o		○		r		○			○	o			○	〃 R
S		○		○		o		○	○	s			○	○		o		○		〃 S
T	○	○		○		o	○			t			○			o		○	○	〃 T
U		○		○		o	○		○	u			○	○		o	○			〃 U
V		○		○		o	○	○		v			○			o	○		○	〃 V
W	○	○		○		o	○	○	○	w			○			o	○	○		〃 W
X	○	○		○	○	o				x			○	○		o	○	○	○	〃 X
Y		○		○	○	o			○	y			○	○	○	o				〃 Y
Z		○		○	○	o		○		z			○		○	o			○	〃 Z
DEL	○	○	○	○	○	o	○	○	○	Del		○	○	○	○	o	○	○	○	* Delete (erasal of erroneous punched hole)
NUL						o				Blank						o				* Unusable at the interval of significant information in case of non-punched hole EIcode
BS	○				○	o				BS			○		○	o		○		* Back space
HT					○	o			○	Tab			○	○	○	o	○	○		* Tabulator
LF or NL					○	o		○		CR or EOB	○					o				End of block
CR	○				○	o	○		○											* Carriage return
SP	○		○			o				SP				○		o				* Space
%	○		○			o	○		○	ER					○	o		○	○	Absolute rewinding stop

ISO code										EIA code											Meaning
	8	7	6	5	4		3	2	1		8	7	6	5	4		3	2	1		
(			○		○	o				(2-4-5)				○	○	o		○			Control out (note-part start)
)	○		○		○	o			○	(2-4-7)		○			○	o		○			Control in (note-part end)
+			○		○	o		○	○	+		○	○	○		o				*	Positive symbol
−			○		○	o	○		○	−		○				o					Negative
:			○	○	○	o		○													Colon
/	○		○		○	o	○	○	○	/			○	○		o			○		Optional block skip
.			○		○	o	○	○		.		○	○		○	o		○	○		Period (decimal)
#	○		○			o		○	○											*	Sharp
$			○			o	○													*	Doller symbol
&	○		○			o	○	○		&					○	o	○	○		*	Ampersand
▼			○			o	○	○	○											*	Apostrophy
*	○		○		○	o		○												*	Asterisk
,	○		○		○	o	○			,			○	○	○	o		○	○	*	Comma
;	○		○	○	○	o		○	○											*	Semicolon
<			○	○	○	o	○													*	Left angle bracket
=	○		○	○	○	o	○		○											*	Equal symbol
>	○		○	○	○	o	○	○												*	Right angle bracket
?			○	○	○	o	○	○	○											*	Question mark
@	○	○				o														*	Commercial at mark
"			○					○												*	Quatation mark
[	○	○		○	○	o		○	○											*	Left big parenthesis
]	○	○		○	○	o	○		○											*	Right big parenthesis

(Note 1) * marked code can be read in tape memory at only time when there is notation part.
It is neglected at the other significant information.

(Note 2) ?-mark code can be read in tape memory at only time when there is notation part.
It becomes alarm at the other significant information.

(Note 3) With custom-macro option, further next code is used at the significant information interval.
at time of ISO + [ ] # * = and E
at time of EIA + [ ] & , £ parameter-set code and E

(Note 4) Code unlisted in this table with correct parity is always neglected.

(Note 5) Code of incorrect parity becomes TH-alarm. Whereas, it is neglected at notation part, and it does not induce TH-alarm.

(Note 6) Whole punched hole in case of EIA-code is handled particularly, and it does not induce parity alarm, and is neglected.

10-7 If Alarm is Issued.

Confirm by alarm list of the maintenance section of SEICOS III/A Instruction Manual.

SEIKI 日立精機株式会社

本社・営業部	☎ 0471-84-1111	千葉県我孫子市我孫子1	〒270-11
東京営業所	☎ 03-213-6411	東京都千代田区丸ノ内2-4-1 丸ビル5階	〒100
山形出張所	☎ 0236-24-3271	山形市薬師町1-9-2	〒990
八王子出張所	☎ 0426-46-3161	東京都八王子市大和田町5-29-5 カスタムビル2階	〒192
長野出張所	☎ 0268-27-8241	長野県上田市材木町1-8-9 信栄ビル2階	〒386
北関東営業所	☎ 0486-44-5510	埼玉県大宮市北袋1-103-1 中村ビル5階	〒330
神奈川営業所	☎ 0462-63-7788	神奈川県大和市中央4-3-8 富士住宅ビル3階	〒242
静岡営業所	☎ 0542-51-1737	静岡市日出町1-2 静岡住友ビル9階	〒420
名古屋営業所	☎ 052-775-2251	名古屋市名東区上社2-168 ステイションビル1階	〒465
三河出張所	☎ 0566-22-4320	愛知県刈谷市東刈谷町3-9-3 大脇ビル2階	〒448
北陸出張所	☎ 0762-23-0633	金沢市北安江1-1-1 第二坂口ビル(株)トミタ内	〒920
大阪営業所	☎ 06-380-0011	大阪府吹田市江坂町1-11-8 日本生命江坂ビル1階	〒564
京都出張所	☎ 075-822-1050	京都市中京区西ノ京下合町37 レトロビル2階	〒604
明石出張所	☎ 078-928-2855	兵庫県明石市和坂12-2 山崎ビル1階	〒673
岡山事務所	☎ 0862-44-2610	岡山市今保148 Kビル1階	〒701-01
広島営業所	☎ 082-294-3811	広島市西区上天満町3-19 第二横山ビル1階	〒733
九州営業所	☎ 092-474-3815	福岡市博多区博多駅東2-6-28 サンライフ第5ビル5階	〒812