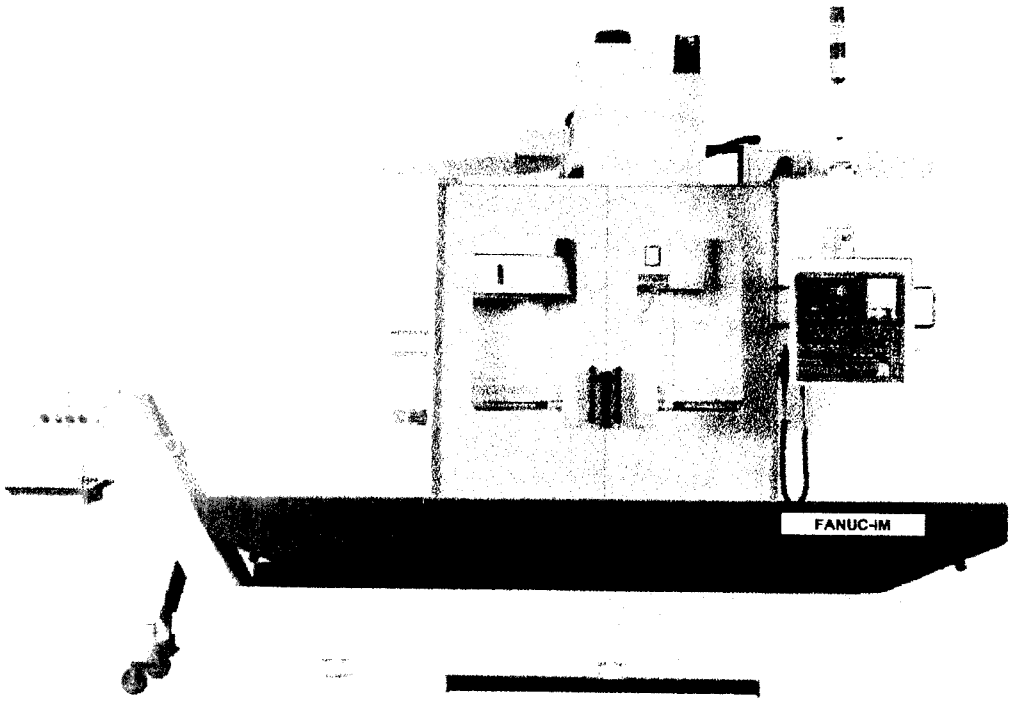
IMN

[redacted]

FANUC-iM CONTROL TRAINING MANUAL



RETECON (Pty) Ltd

100 Plane Road , Spartan Industrial Township , Kempton Park ,1620
e-mail: machines@retecon.co.za

JOHANNESBURG
Gerhard Hetzler
Tel: (011) 394-2810
Fax: (011) 394-2471

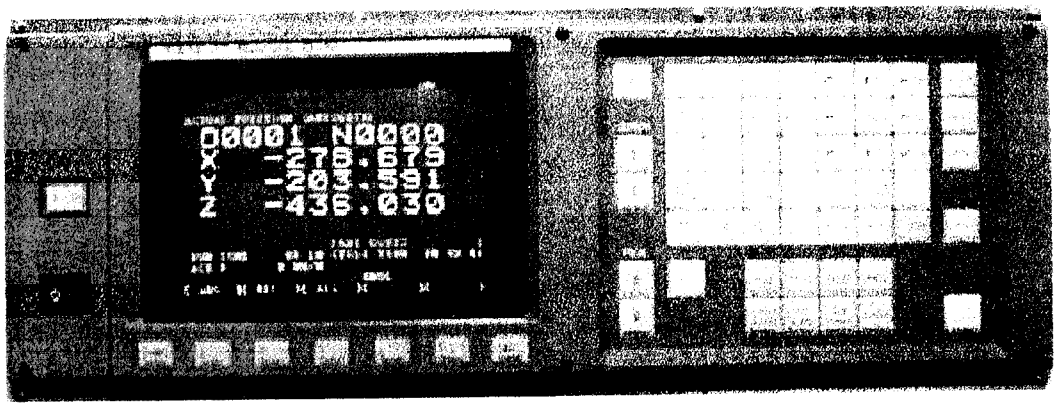
DURBAN
Tel: (031) 701-8149
Fax: (031) 701-0313

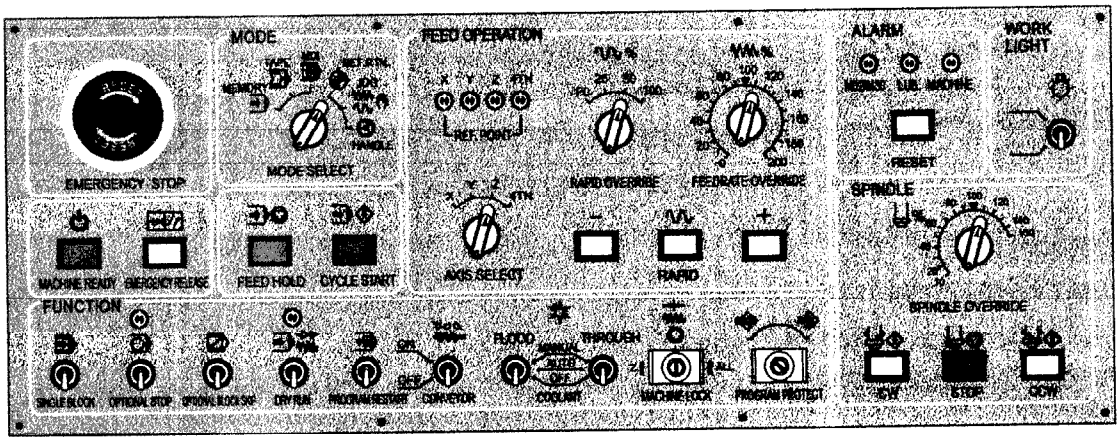
PORT ELIZABETH
Tel: (041) 453-2720
Fax: (041) 453- 6678

CAPE TOWN
John Whitehead
Tel: (021) 555-2270
Fax: (021) 555-2272

YOUR MACHINE TOOL SUPPLIER
WELCOMES ALL PARTICIPANTS
TO THE FANUC- iM COURSE

CONTROL LAYOUT





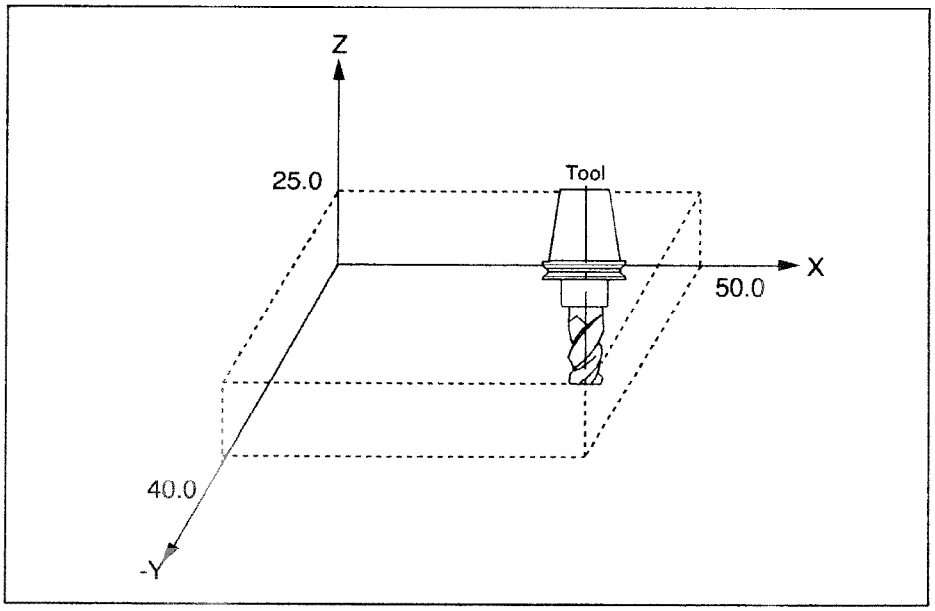
AXIS COORDINATES

By teaching the CNC a desired tool position, the tool can be moved to the position. Such a tool position is represented by coordinates in a coordinate system. Coordinates are specified using program axes.

When three program axes, the X–axis, Y–axis, and Z–aixs, are used, coordinates are specified as follows :

X_Y_Z_

This command is referred to as a dimension word.



[Tool position Specified by X50.0 Y-40.0 Z25.0]

Coordinates are specified in one of following three coordinate systems :

(1) Machine coordinate system

(2) Workpiece coordinate system

(3) Local coordinate system

The number of the axes of a coordinate system varies from one machine to another. So, in this manual, a dimension word is represented as IP_.

G - CODE LIST

A number following address G determines the meaning of the command for the concerned block.

G codes are divided into the following two types.

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

(Example)

G01 and G00 are modal G codes in group 01.

```
G01 X_ ;┐
    Z_ ;│  G01 is effective in this range.
    X_ ;┘
G00 Z_ ;
```

Notes)

1. G codes marked ★ are initial G codes when turning power on. For G20 and G21, the G code before turning power off remains. G00 / G01 / G17 / G18 / G19 and G90/G91 can be selected by parameter (No. 3402) setting.
2. G codes in the 00 group are one–shot G codes. However, G10 is once specified, it is valid until being cancelled by G11.
3. If a G code not listed on the table of G codes in inputted, or optional G code not specified in the system is commanded, an alarm (No.010)is displayed.
4. A number of G codes can be specified in the same block. When more than one G code of the same group is specified, the G code specified later is effective.
5. If any G code of group 01 is specified in a canned cycle mode, the canned cycle is automatically cancelled and the G80 condition is entered. However, a G code of group 01 is not affected by any of the canned cycle G codes.
6. A G code is displayed from each group.

[G code list]

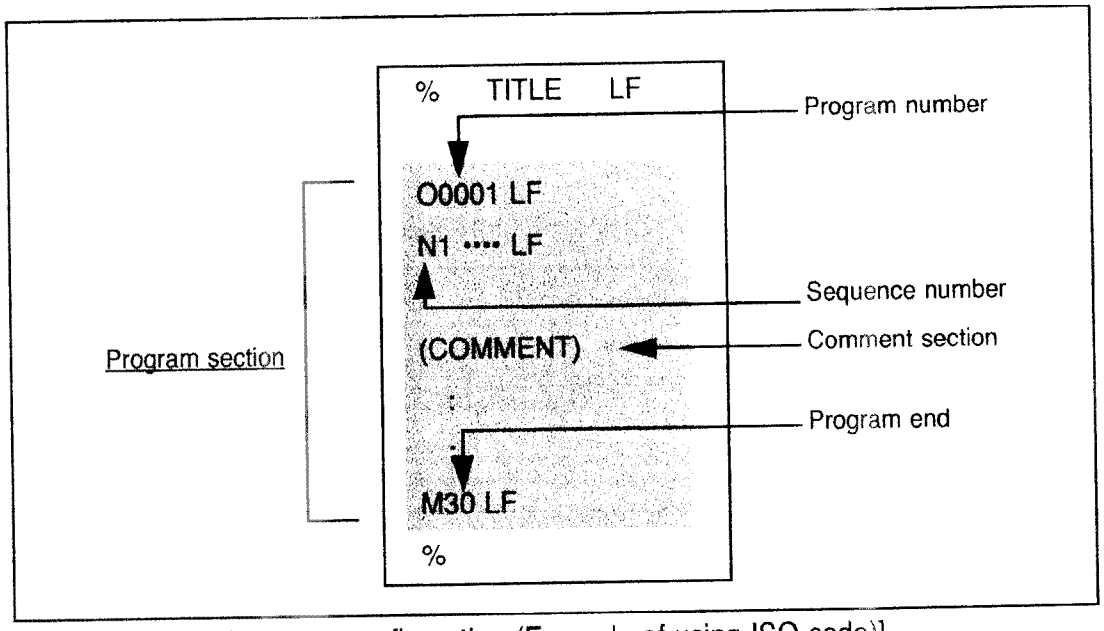
G code	Group	Function	
★ G00	01	Positioning	
G01		Linear interpolation	
G02		Circular interpolation/Helical interpolation CW	
G03		Circular interpolation/Helical interpolation CCW	
G04	00	Dwell, Exact stop	
G05		High speed cycle machining	
G07.1 (G107)		Cylindrical interpolation	
G09		Exact stop	
G10		Data setting	
G11		Data setting mode cancel	
G12.1 (G112)	25	Polar coordinate interpolation mode	
★ G13.1 (G113)		Polar coordinate interpolation cancel mode	
★ G15	17	Polar coordinate command cancel	
G16		Polar coordinates command	
★ G17	02	XpYp plane selection	Xp : X axis or its parallel axis
G18		ZpXp plane selection	Yp : Y axis or its parallel axis
G19		YpZp plane selection	Xp : Z axis or its parallel axis
G20	06	Input in inch	
G21		Input in mm	
★ G22	04	Stored stroke check function on	
G23		Stored stroke check function off	
G27	00	Reference position return check	
G28		Return to reference position	
G29		Return from reference position	
G30		2nd, 3rd and 4th reference position return	
G30.1		Floating reference point return	
G31		Skip function	
G33	01	Thread cutting	

[G code list]

G code	Group	Function
G37	00	Automatic tool length measurment
G39		Corner offset circular interpolation
★ G40	07	Cutter compensation cancel
G41		Cutter compensation left
G42		Cutter compensation right
★ G40.1 (G150)	19	Normal direction control cancel mode
G41.1 (G151)		Normal direction control left side on
G42.1 (G152)		Normal direction control right side on
G43	08	Tool length compensation + direction
G44		Tool length compensation – direction
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
★ G50.1	18	Programmable mirror image cancel
G51.1		Programmable mirror image
G52	00	Local coordinate system setting
G53		Machine coordinate system selection
★ G54	14	Workpiece coordinate system 1 selection
G54.1		Additional workpiece coordinate system selection
G55		Workpiece coordinate system 2 selection
G56		Workpiece coordinate system 3 selection
G57		Workpiece coordinate system 4 selection
G58		Workpiece coordinate system 5 selection
G59		Workpiece coordinate system 6 selection
G60	00	Single direction positioning
G61	15	Exact stop mode
G62		Automatic corner override
G63		Tapping mode
★ G64		Cutting mode

[G code list]

G code	Group	Function
G65	00	Macro call
G66	12	Macro modal call
★ G67		Macro modal call cancel
G68	16	Coordinate rotation
★ G69		Coordinate rotation cancel
G73	09	Peck drilling cycle
G74		Counter tapping cycle
G76	09	Fine boring cycle
★ G80	09	Canned cycle cancel/external operation function cancel
G81		Drilling cycle, spot boring cycle or external operation function
G82		Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle
★ G90	03	Absolute command
G91		Increment command
G92	00	Setting for work coordinate system or clamp at maximum spindle speed
★ G94	05	Feed per minute
G95		Feed per rotation
G96	13	Constant surface speed control
★ G97		Constant surface speed control cancel
★ G98	10	Return to initial point in canned cycle
G99		Return to R point in canned cycle



[Program configuration (Example of using ISO code)]

■ **Program number**

A program number consisting of address O followed by a four–digit number is assigned to each program at the beginning registered in memory to identify the program.

In ISO code, the colon (:) can be used instead of O.

When no program number is specified at the start of a program, the sequence number (N....)at the start of the program is regarded as its program number. If a five–digit sequence number is used, the lower four digits are registered as a program number. If the lower four digits are all 0, the program number registered immediately before added to 1 is registered as a program number. Note, however, that N0 cannot be used for a program number.

If there is no program number or sequence number at the start of a program, a program number must be specified using the CRT/MDI panel when the program is stored in memory.

Note)

Program numbers 8000 to 9999 may be used by machine tool builders, and the user may not be able to use these numbers.

■ **Block configuration (word and address)**

A block consists of one or more words. A word consists of an address followed by a number some digits long. (The plus sign(+) or minus sign(–) may be prefixed to a number.)

Word = Address + number (Example : X-1000)

For an address, one of the letters (A to Z) is used ; an address defines the meaning of a number that follows the address. Table 13.2(b) indicates the usable addresses and their meanings.

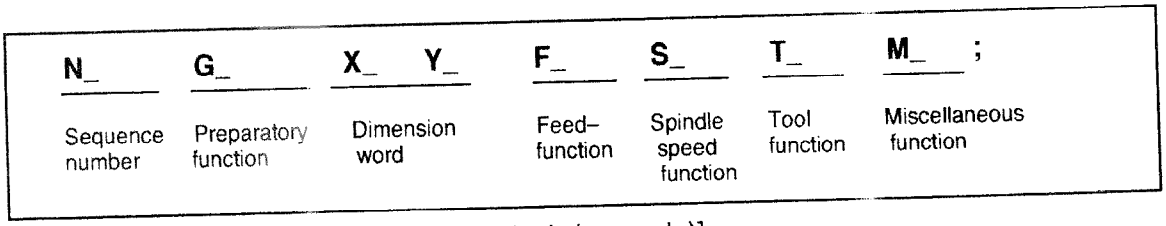
The same address may have different meanings, depending on the preparatory function specification.

[Major functions and addresses]

Function	Address	Meaning
Program number	O(1)	Program number
Sequence number	N	Sequence number
Preparatory function	G	Specifies a motion mode (linear, arc, etc.)
Dimension word	X, Y, Z, U, V, W, A, B, C	Coordinate axis move command
	I, J, K	Coordinate of the arc center
	R	Arc radius
Feed function	F	Rate of feed per minute, Rate of feed per revolution
Spindle speed function	S	Spindle speed
Tool function	T	Tool number
Auxiliary function	M	On/off control on the machine tool
	B	Table indexing, etc.
Offset number	D, H	Offset number
Dwell	P, X	Dwell time
Program number designation	P	Subprogram number
Number of repetitions	P	Number of subprogram repetitions
Parameter	P, Q	Canned cycle parameter

Note)

1. In ISO code, the colon(:) can also be used as the address of a program number.



[1 block (example)]

■ **Major addresses and ranges of command values**

Major addresses and the ranges of values specified for the addresses are shown below. Note that these figures represent limits on the CNC side, which are totally different from limits on the machine tool side. For example, the CNC allows a tool to traverse up to about 100m(in millimeter input) along the X axis.

However, an actual stroke along the X axis may be limited to 2m for a specific machine tool. Similarly, the CNC may be able to control a cutting federate of up to 240m/min, but the machine tool may not allow more than 3m/min. When developing a program, the user should carefully read the manuals of the machine tool as well as this manual to be familiar with the restrictions on programming.

[Major addresses and ranges of command values]

Function		Address	Input in mm	Input in inch
Program number		O(1)	1-9999	1-9999
sequence number		N	1-99999	1-99999
Preparatory function		G	0-99	0-99
Dimension word	Increment system IS-B	X, Y, Z, U, V, W, A, B, C, I, J, K, R	±99999.999mm	±9999.9999inch
	Increment system IS-C		±99999.999mm	±999.99999inch
feed per minute	Increment system IS-B	F	1-240000mm/min	0.01-9600.00 inch/mm
	Increment system IS-C		1-100000mm/min	0.01-4000.00 inch/mm
feed per revolution		F	0.001-500.00mm/rev	0.0001-9.9999 inch/rev
Spindle speed function		S	0-20000	0-20000
Tool function		T	0-99999999	0-99999999
Auxiliary function		M	0-99999999	0-99999999
		B	0-99999999	0-99999999
Offset number		H, D	0-400	0-400
Dwell	Increment system IS-B	X, P	0-99999.999s	0-99999.999s
	Increment system IS-C		0-9999.9999s	0-9999.9999s
Designation of a program number		P	1-9999	1-9999
Number of repetitions		P	1-9999	1-9999

Note)

1. In ISO code, the colon(:) can also be used as the address of a program number.

POSITIONING (G00)

The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental command at a rapid traverse rate.

In the absolute command, coordinate value of the end point is programmed.

In the incremental command the distance the tool moves is programmed.

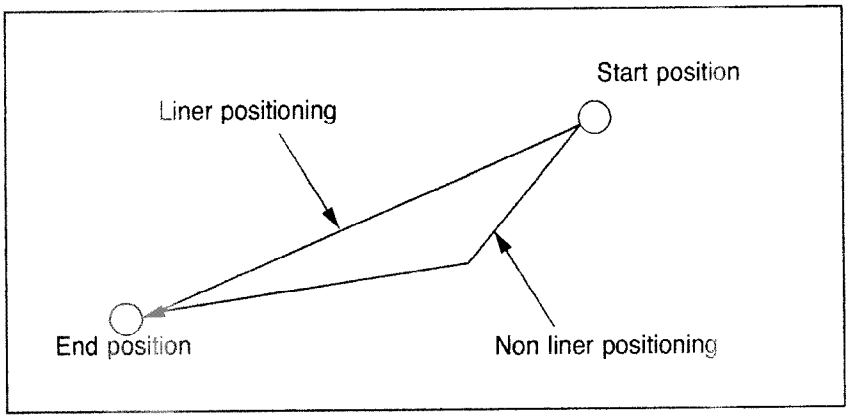
[Format]

G00IP_ ;

IP_ : For an absolute command, the coordinates of an end position, and for an incremental command, the distance the tool moves.

[Explanations]

Tool path generally does not become a straight line.



The rapid traverse rate in the G00 command is set to the parameter No.11420 for each axis independently by the machine tool builder. In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution proceeds to the next block after confirming the in–position.

"In–position" means that the feed motor is within the specified range.

This range is determined by the machine tool builder by setting to parameter No.1827.

[Restrictions]

The rapid traverse rate cannot be specified in the address F.

G01

LINEAR INTERPOLATION (G01)

Tools can move along a line

[Format]

G01 IP_ F_;

IP_ : For an absolute command, the coordinates of an end position, and for an incremental command, the distance the tool moves.

F_ : Speed of tool feed (Feedrate)

[Explanations]

A tools move along a line to the specified position at the feedrate specified in F.

The feedrate specified in F is effective until a new value is specified need not be specified for each block.

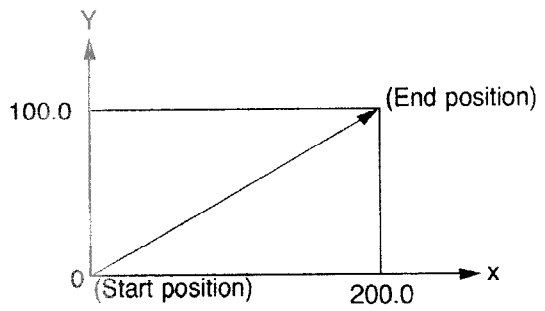
The feedrate commanded by the F code is measured along the tool path.

If the F code is not commanded, the feedrate is regarded as zero.

[Examples]

■ **Linear interpolation**

G01 X200.0 Y100.0 F200.0 ;



G02 / G03

CIRCULAR INTERPOLATION (G02, G03)

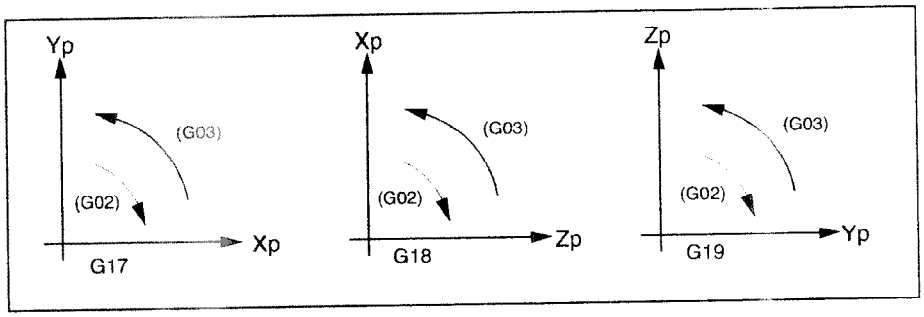
The command below will move a tool along a circular arc.

Command	Description
G	Specification of arc on XpYp plane
G	Specification of arc on ZpXp plane
G19	Specification of arc on YpZp plane
G02	Circular interpolation Clockwise direction (CW)
G03	Circular interpolation Counterclockwise direction (CCW)
Xp_	Command values of X axis or its parallel axis (set by parameter No. 1022)
Yp_	Command values of Y axis or its parallel axis (set by parameter No. 1022)
Zp_	Command values of Z axis or its parallel axis (set by parameter No. 1022)
I_	Xp axis distance from the start point to the center of an arc with sign
J_	Yp axis distance from the start point to the center of an arc with sign
K_	Zp axis distance from the start point to the center of an arc with sign
R_	Arc radius with sign fixed to radius designation
F_	Feedrate along the arc

[Explanations]

■ **Direction of the circular interpolation**

"Clockwise"(G02) and "Counterclockwise"(G03) on the XpYp plane (ZpXp plane or YpZp plane) are defined when the XpYp plane is viewed in the positive–to–negative direction of the Zp axis (Yp axis or Xp axis, respectively) in the Cartesian coordinate system. See the figure below.



■ **Distance moved on an arc**

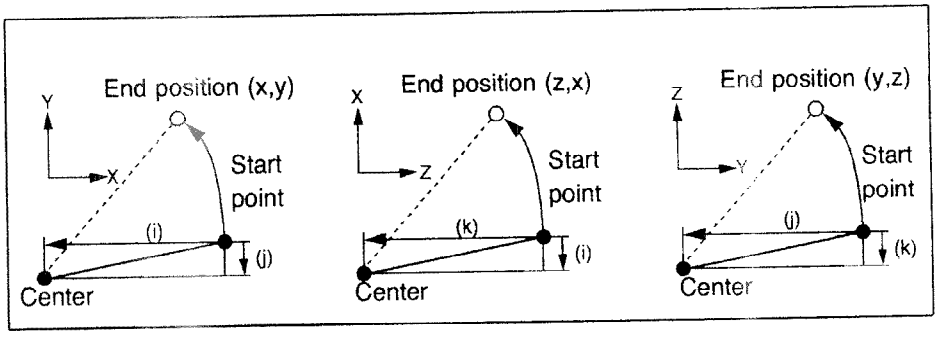
The end point of an arc is specified by address Xp, Yp or Zp, and is expressed as an absolute or incremental value according to G90 or G91.

For the incremental value, the distance of the end point which is viewed from the start point of the arc is specified.

■ **Distance from the start point to the center of arc**

The arc center is specified by addresses I, J, and K for the Xp, Yp and Zp axes, respectively. The numerical value following I, J, or K, however, is a vector component in which the arc center is seen from the start point, and is always specified as an incremental value irrespective of G90 and G91, as shown below.

I, J and K must be signed according to the direction.

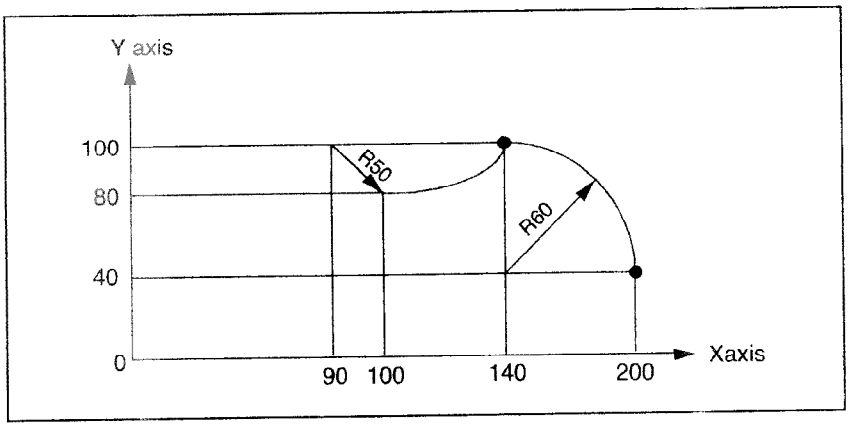


I0, J0, and K0 can be omitted. When Xp, Yp and Zp are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360° arc(circle) is specified.

G02I ; Command for a circle

If the difference between the radius at the start point and that at the end point exceeds the value in a parameter (No.3410), an alarm (No.024) occurs.

[Examples 1]

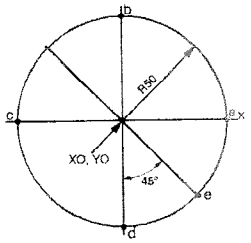


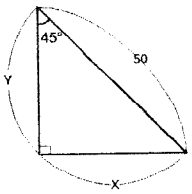
The above tool path can be programmed as follows ;

(1) In absolute programming
G92 X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0 R60.0 F300.;
G02 X120.0 Y60.0 R50.0 ;
or
G92 X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0 I-60.0 F300. ;
G02 X120.0 Y60.0 I-50.0 ;

(2) In incremental programming
G91 G03 X-60.0 Y60.0 R60.0 F300. ;
G02 X-20.0 Y-40.0 R50.0 ;
or
G91 G03 X-60.0 Y60.0 I-60.0 F300. ;
G02 X-20.0 Y-40.0 I-50.0 ;

[Examples 2]





$$\text{SIN } 45 = \frac{X}{50}$$

$$X = \text{SIN}45° \times 50$$

$$= 35.355$$

CW → / CCW	
a CCW → b	G03 X0 Y50. I-50. J0
a CCW → d	G03 X0 Y-50. I-50. J0
b CW → e	G02 X35.355 Y-35.355 I0 J-50.
d CW → b	G02 X0 Y50. I0 J50.
e CCW → c	G03 X-50. Y0 I-35.355 J35.355
c CW → d	G02 X0 Y-50. I50. J0
a CCW → a	G03 X50. Y0 I-50. J0

DWELL (G04)

[Format]

Dwell	G04 X_ ; or G04 P_ ; X_ ; Specify a time (decimal point permitted) P_ ; Specify a time (decimal point not permitted)

[Explanations]

By specifying a dwell, the execution of the next block is delayed by the specified time. In addition, a dwell can be specified to make an exact check in the cutting mode (G62 mode).
When neither P nor X is specified, exact stop is performed.

[Command value range of the dwell time (Command by X)]

Increment system	Command value range	Dwell time unit
IS-B	0.001~99999.999	sec or rev
IS-C	0.0001~9999.9999	

[Command value range of the dwell time (Command by P)]

Increment system	Command value range	Dwell time unit
IS-B	1~99999999	0.001 sec or rev
IS-C	1~99999999	0.0001 sec or rev

PLANE SELECTION

[Explanations]

[Plane selected by G code]

G code	Selected plane	Xp	Yp	Zp
G17	Xp Yp plane	X–axis or an axis parallel to it	Y–axis or an axis parallel to it	Z–axis or an axis parallel to it
G18	Zp Xp plane			
G19	Yp Zp plane			

Xp, Yp, Zp are determined by the axis address appeared in the block in which g17, G18 or G19 is commanded.

When an axis address is omitted in G17, G18 or G19 block, it is assumed that the addresses of basic three axes are omitted.

Parameter No.1022 is used to specify that an optional axis be parallel to the each axis of the X, Y–, and Z–axes as the basic three axes.

The plane is unchanged in the block in which G17, G18 or G19 is not commanded.

When the power is turned on or the CNC is reset, G17 (XY plane), G18 (ZX plane), or G19(YZ plane) is selected by bits 1(G18) and 2(G19) of parameter 3402.

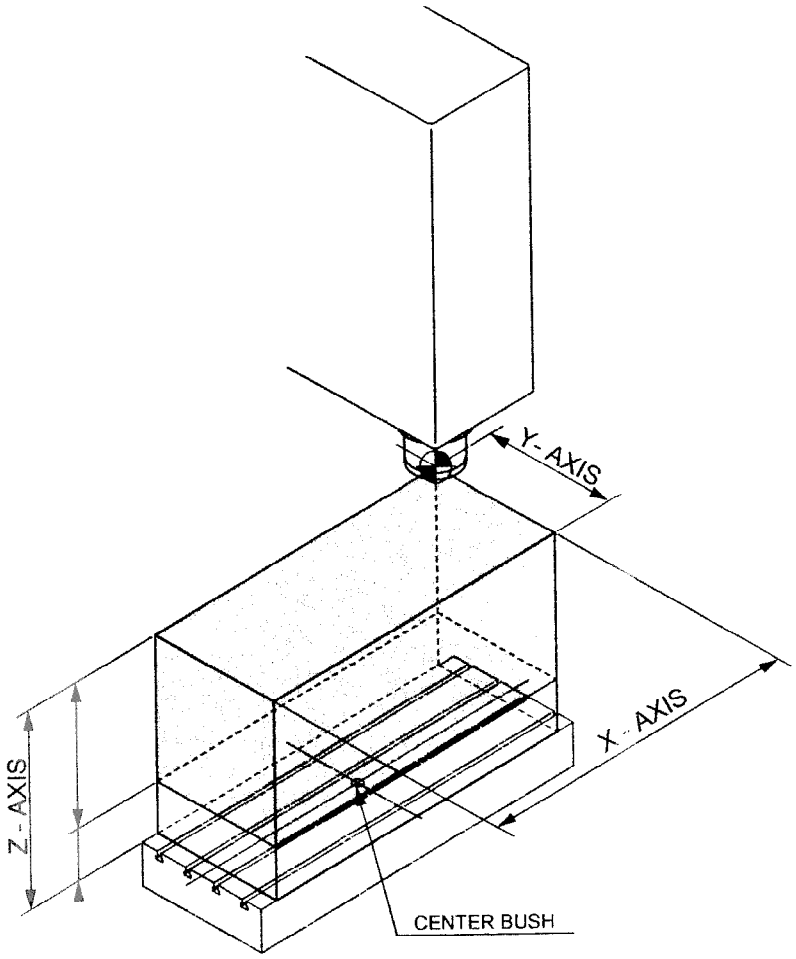
The movement instruction is irrelevant to the plane selection.

[Examples]

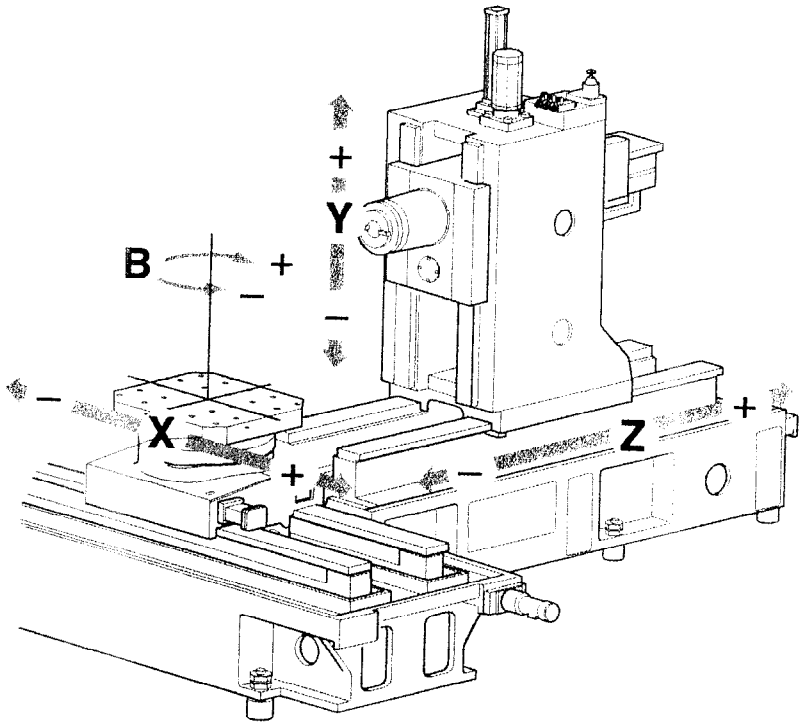
Plane selection when the X–axis is parallel with the U–axis.

G17X_Y_ ;	XY plane,
G17U_Y_ ;	UY plane
G18X_Z_ ;	ZX plane
X_Y_ ;	Plane is unchanged (ZX plane)
G17 ;	XY plane
G18 ;	ZX plane
G17 U_ ;	UY plane
G18 Y_ ;	ZX plane, Y axis moves regardless without any relation to the plane.

VERTICAL AXIS UNIT



HORIZONTAL [illegible]



X axis : Pallet table cross

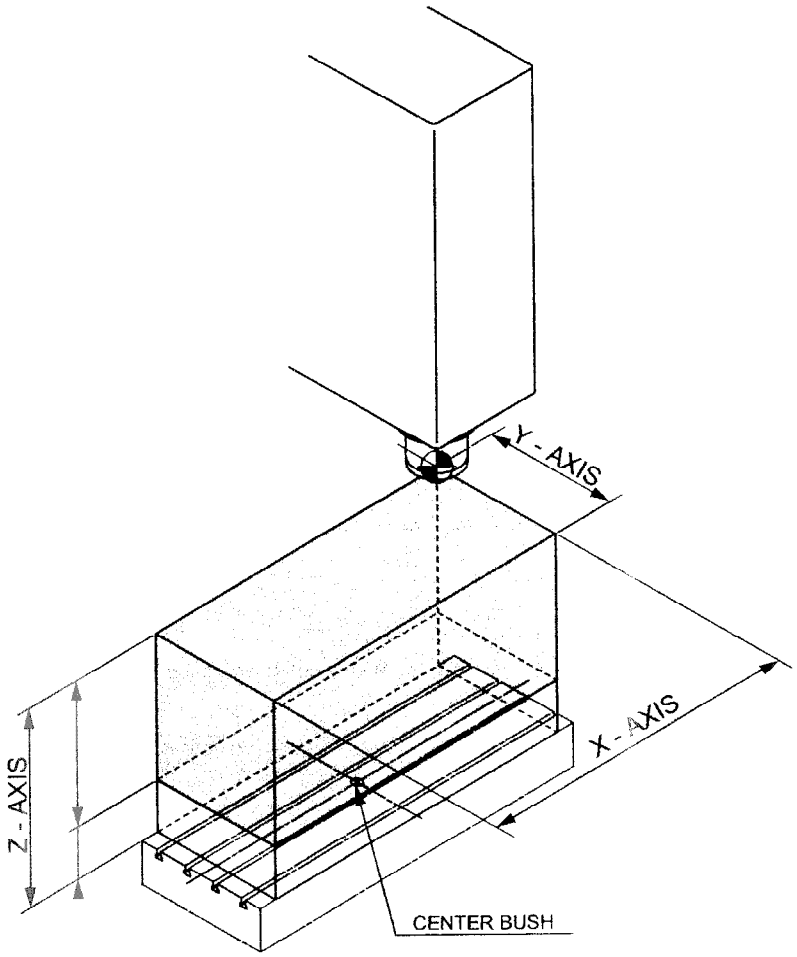
Y axis : Spindle head vertical

Z axis : Column longitudinal

B axis : Table rotation

The figure below shows the machine reference point on the basis of a workpiece.

PROGRAM : G9[illegible] G28 X0. Y0. Z0. ;



WORK PIECE ZERO POINTS

(1) Choosing from six workpiece coordinate systems set using the CRT/MDI panel

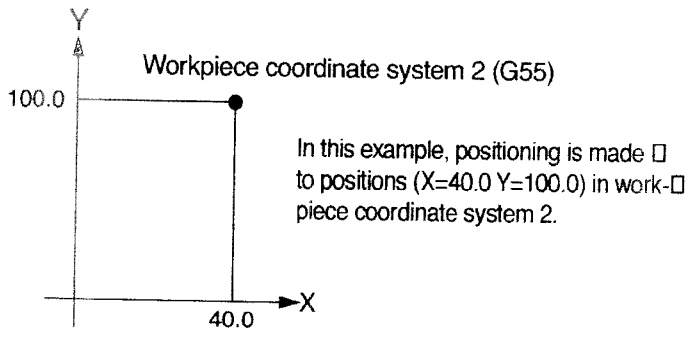
By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.

G54 ..... Workpiece coordinate system 1
G55 ..... Workpiece coordinate system 2
G56 ..... Workpiece coordinate system 3
G57 ..... Workpiece coordinate system 4
G58 ..... Workpiece coordinate system 5
G59 ..... Workpiece coordinate system 6

Workpiece coordinate system 1 to 6 are established after reference position return after the power is turned on. When the power is turned on, G54 coordinate system is selected.

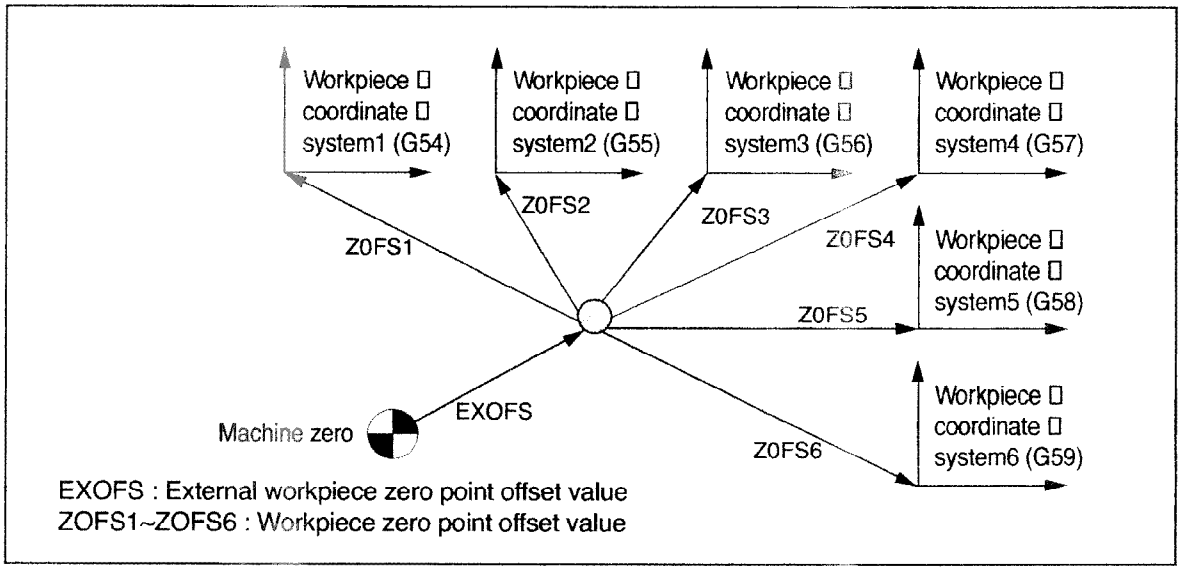
[Examples]

G90 G55 G00 X40.0 Y100.0 ;



The six workpiece coordinate systems specified with G54 to G59 can be changed by changing an external workpiece zero point offset value or workpiece zero point offset value.
Three methods are available to change an external workpiece zero point offset value or workpiece zero point offset value.

(1) Inputting from the CRT/MDI panel.
(2) Programming by G10 or G92
(3) Changing an external workpiece zero point offset value (refer to machine tool builder's manual)



[Changing an external workpiece zero point offset value or workpiece zero point offset value]

[Format]

■ **Changing by G10**

G10 L2 Pp IP_ ;
p=0 : External workpiece zero point offset value
p=1 to 6 : Workpiece zero point offset value correspond to workpiece coordinate system 1 to 6
IP : Workpiece zero point offset value of each axis

G90 / G91

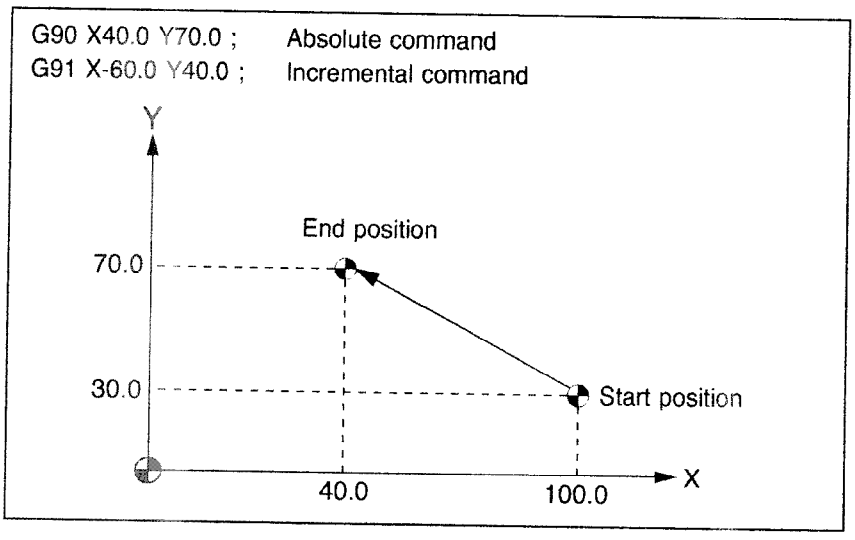
ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91)

There are two ways to command travels of the tool ; the absolute command, and the incremental command. In the absolute command, coordinate value of the end position is programmed ; in the incremental command, move distance of the position ifself is programmed. G90 and G91 are used to command absolute or incremental command, respectively.

[Format]

Absolute command	G90 IP_ ;
Incremental command	G91 IP_ ;

[Examples]



M CODES

M-CODE	FUNCTION	REMARK
M 00	PROGRAM STOP (UNCONDITIONAL STOP)	
M 01	OPTIONAL BLOCK STOP	
M 02	PROGRAM END	
M 03	SPINDLE CLOCKWISE ROTATION	
M 04	SPINDLE COUNTER CLOCKWISE ROTATION	
M 05	SPINDEL ROTATION STOP	
M 06	AUTOMATIC TOOL CHANGE	
M 07	COOLANT 2 ON(THRU-THE COOLANT)	
M 08	COOLANT 1 ON(FLOOD COOLANT)	
M 09	COOLANT 1, 2, 3 OFF	
M 10	TABLE(B-AXIS) CLAMP	
M 11	TABLE(B-AXIS) UNCLAMP	
M 13	COOLANT 4 ON(INTERMITTENT COOLANT)	
M 19	SPINDLE ORIENTATION	
M 30	PROGRAM END & REWIND RESET	
M 33		
M 34	TOOL DATA COMPARE OPEN	
M 35	TOOL DATA COMPARE CLOSE	
M48	OVERRIDE 100% CLAMP	
M 49	OVERRIDE 100% CLAMP CANCEL	
M 60	AUTO. PALLET CHANGE	
M 61	PALLET 1 LOAD	
M 62	PALLET 2 LOAD	
M 68	SPINDLE TOOL CLAMP	
M 69	SPINDLE TOOL UNCLAMP	
M 80	MIRROR IMAGE CANCEL	
M 81	X-AXIS MIRROR IMAGE	
M 82	Y-AXIS MIRROR IMAGE	
M 84	AXIS MOVING VALID WHEN SPDL. STOP	
M 85	AXIS MOVING INVALID WHEN SPDL. STOP	
M 98	CALLING OF SUB PROGRAM	
M 99	CALLING END OF SUB PROGRAM	

MISCELLANEOUS FUNCTION(M CODE)

When a figute is specified following address M, a code signal and a stroke signel are transmitted. These signals are used for ON/OFF control of a machine function. (Usually, only one M code can be specified in on block. In some cases, up to three M codes can be specified for some types of machine tool)

Which M code corresponds to which machine function is determined by the machine. The machine processes all operation specified by M codes except those specified by M98, M99, M198 or called subprogram(parameter No. 6071~6079) or called custom macro(parameter No. 6080~6089). Refer to the machine's instruction manual for details.

M00, M01, M02, M30(stop code)

(1) M00(Program stop)

This code is used for stopping of the machine for manual operation or inspection.

If a axis moving command is in the same block with M00 code, the machine is stopped after the axis moving is executed.

For restart the operation, the cycle start button must be pressed.

The spindle rotation command(M03, M04) and coolant ON command(M07, M08, M13, M50 and M51) are canceled by M00 code. After program stops, the spindle rotation or coolant ON can be commanded by necessary.

(2) M01(Optional stop)

Similarly to M00, automatic operation is stopped offer a block containing M01 is executed. This code is only effective when the optional stop switch on the machine operator's panel has been pressed.

(3) M02(End of program)

This indicates the end of program. This code is used at a program end that the tape rewinding is not needed.

The CNC unit is reset by this code.

The spindle rotation and coolant ON are canceled.

M02 code must be specified in a single block.

If a axis moving command is in the same block with M02 code, The end of program is executed after the axis moving is executed.

(4) M30(Tape end)

This code is used at a end of block that a tape rewinding is needed.

The tape rewinving is added on M02 funcdtion.

The tape rewing is started by M30, and stopped by %(Jis code) or EOR(EIA code).

Bit 5 of parameter 3404(M02) or bit 4 parameter 3404(M30) can be used to disable M02, M30 from returning control to the start of the program.

M03, M04, M05, M19(Spindle rotating and stop)

(1) M03, M04 spindle rotating

M03 ····· Clockwise spindle

M04 ····· Counter clockwise spindle

M03 and M04 function is to rotate spindle. If a axis moving command is in the same block with these code. The axis moving is executed after the spindle rotates.

If the spindle is rotating this rotation command is ignored.

(2) M05(Spindle stop)

When the spindle is rotating, M05 stops the spindle.

If a axis moving command is in the same block with M05, the spindle stops after the axis moving is executed.

(3) M19(Spindle orientation stop)

The spindle is stopped at a fired angle by M19.

For automatic tool change, the spindle orientation stop has to be executed.

This spindle orientation stop can be used for back-boring maching.

The M19 can be commanded during the spindle ratating.

M08, M13, M07, M09(Coolant ON/OFF command)

M08 ····· Flood coolant ON

M07 ····· Through the spindle coolant ON

M09 ····· Coolant OFF

(1) M08(Flood coolant ON)

Coolant is applied from the nozzle mounted in front of spindle head.

The coolant is stopper by M09. The coolant stops when M06, M00, M01, M02 or M30 is sent without M09.

(2) M07(Through the spindle coolant On)

The through the spindle coolant can be applied by optional specification.

M48, M49(Override ignore/cancel)

M49 makes the feed rate override select switch on the machine control panel invalid.

M48 ····· M49 cancel

M49 ····· Overrice ignore

Ecen if the feed rate override select switch is set to other than 100%, override is fixed at 100%.

M48 makes M49 function invalid.

M98, M99(Calling end of subprogram)

(1) M98(Calling of subprogram)

This code is used to call a subprogram.

(2) M99(End of subprogram)

This code indicates the end of a subprogram.

This code is executed, control returns to the main program.

CANNED CYCLES

Canned cycles make it easier for the programmer to create programs.
With a canned cycle, a frequently–used machining operation can be specified in a single block with a G function ; without canned cycles, normally more than one block is required. In addition, the use of canned cycles can shorten the program to save memory.

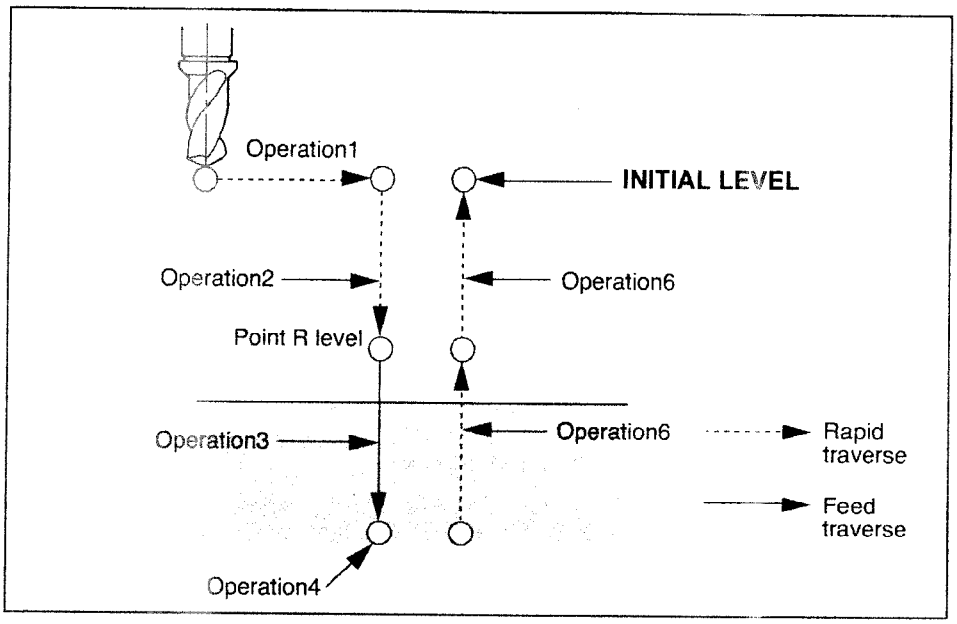
[Canned cycles]

G code	Drilling (–Z direction)	Operation at the botton of a hole	Retraction (+Z direction)	Application
G73	Intermittent feed	–	Rapid traverse	High–speed peck drilling cycle
G74	Feed	Dwell □M Spindle CW	Feed	Left–hand tapping cycle
G76	Feed	Oriented spindle stop	Rapid traverse	Fine boring cycle
G80	–	–	–	Cancel
G81	Feed	–	Rapid traverse	Drilling cycle, spot drilling cycle
G82	Feed	Dwell	Rapid traverse	Drilling cycle, counter boring cycle
G83	Intermittent feed	–	Rapid traverse	Peck drilling cycle
G84	Feed	Dwell□M Spindle CCW	Feed	Tapping cycle
G85	Feed	–	Feed	Boring cycle
G86	Feed	Spindle stop	Rapid traverse	Boring cycle
G87	Feed	Spindle CW	Rapid traverse	Back boring cycle
G88	Feed	Dwell □M Spindle stop	Manual	Boring cycle
G89	Feed	Dwell	Feed	Boring cycle

[Explanations]

A canned cycle consists of a sequence of six operations

Operation 1 ..... Positioning of axes X and Y
(including also another axis)
Operation 2 ..... Rapid traverse up to point R level
Operation 3 ..... Hole machining
Operation 4 ..... Operation at the bottom of a hole
Operation 5 ..... Retraction to point R level
Operation 6 ..... Rapid traverse up to the initial point



[Canned cycle operation sequence]

■ **Positioning plane**

The positioning plane is determined by plane selection code G17, G18, or G19.

The positioning axis is an axis other than the drilling axis.

■ **Drilling axis**

Although canned cycles include tapping and boring cycles as well as drilling cycles, in this chapter, only the tern drilling will be used to refer to operations implemented with canned cycles.

The drilling axis is a basic axis (X, Y, or Z) not used to define the positioning plane, or any axis parallel to that basic axis.

The axis (basic axis or parallel axis) used as the drilling axis is determined according to the axis address for the drilling axis specified in the same block as G codes G73 to G89.

If no axis address is specified for the drilling axis, the basic axis is assumed to be the drilling axis.

[Positioning plane and drilling axis]

G code	Positioning plane	Drilling axis
G17	Xp–Yp plane	Zp
G18	Zp–Xp plane	Yp
G19	Yp–Zp plane	Xp

Xp : X axis or an axis parallel to the X axis.

Yp : Y axis or an axis parallel to the Y axis.

Zp : Z axis or an axis parallel to the Z axis.

[Examples]

Assume that the U, V and W axes be parallel to the X, Y and Z axes respectively. This condition is specified by parameter No.1022.

G17 G81 ..... Z _ _ : The Z axis is used for drilling.
G17 G81 ..... W _ _ : The W axis is used for drilling.
G18 G81 ..... Y _ _ : The Y axis is used for drilling.
G19 G81 ..... X _ _ : The X axis is used for drilling.
G19 G81 ..... U – – : The U axis is used for drilling.

G17 to G19 may be specified in a block in which any of G73 to G89 is not specified.

Notes 1)

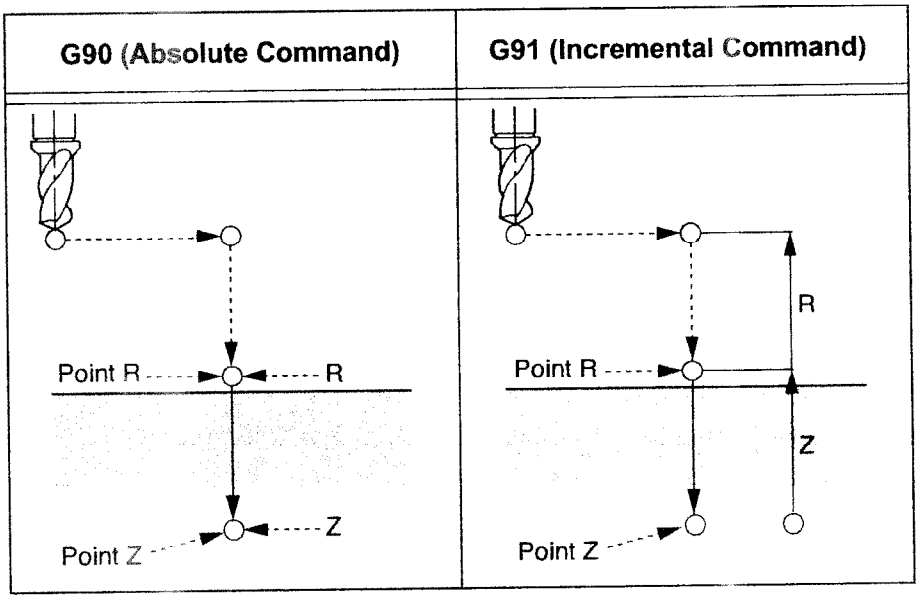
A parameter FXY (No.6200 #0) can be set to the Z axis always used as the drilling axis. When FXY=0, the Z axis is always the drilling axis.

Notes 2)

Switch the drilling axis after canceling a canned cycle.

■ **Travel distance along the drilling axis G90/G91**

The travel distance along the drilling axis varies for G90 and G91 as follows :



■ **Drilling mode**

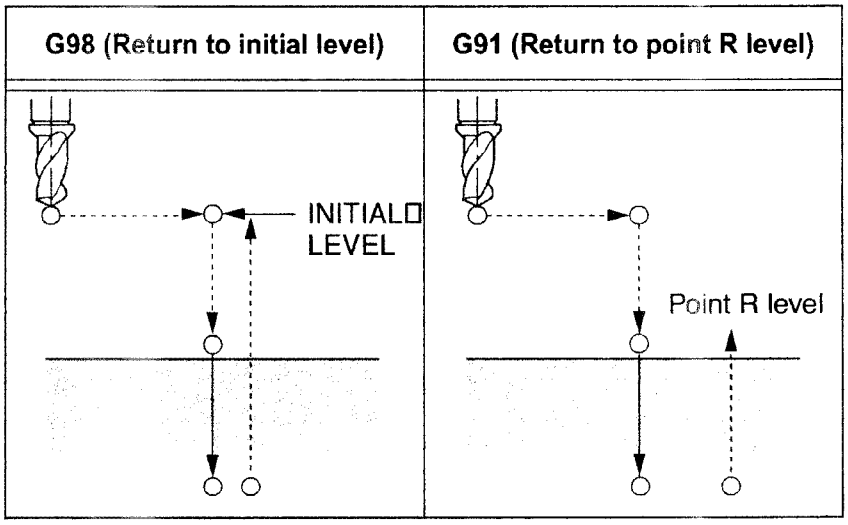
G73, G74, G76, and G81 to G89 are modal G codes and remain in effect until canceled. When in effect, the current state is the drilling mode.

Once drilling data is specified in the drilling mode, the data is retained until modified or canceled. Specify all necessary drilling data at the beginning of canned cycles ; when canned cycles are being performed, specify data modifications only.

■ **Return point level G98/G99**

When the tool reaches the bottom of a hole, the tool may be returned to point R or to the initial level. These operations are specified with G98 and G99. The following illustrates how the tool moves when G98 or G99 is specified. Generally, G99 is used for the first drilling operation and G98 is used for the last drilling operation.

The initial level does not change even when drilling is performed in the G99 mode.



■ **Repeat**

To repeat drilling for equally–spaced holes, specify the number of repeats in K_.

K is effective only within the block where it is specified.

Specify the first hole position in incremental mode (G91).

If it is specified in a absolute mode (G90), drilling is repeated at the same position.

If K0 is specified, drilling data is stored, but drilling is not performed.

■ **Cancel**

To cancel a canned cycle, use G80 or a group 01 G code.

Group 01 G codes

G00 : Positioning (rapid traverse)

G01 : Linear interpolation

G02 : Circular interpolation or helical interpolation (CW)

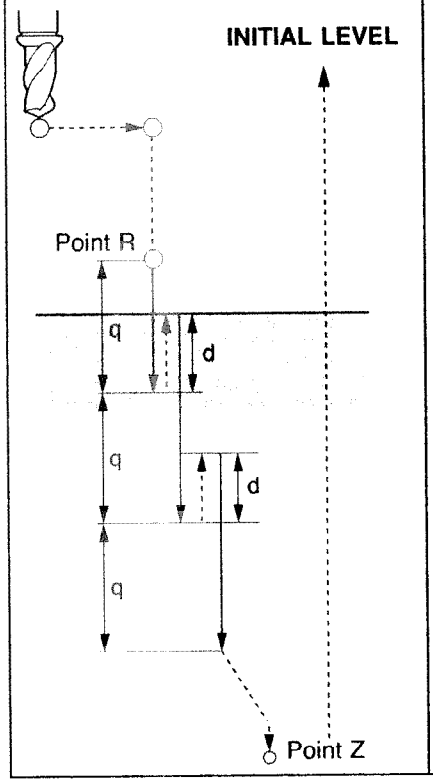
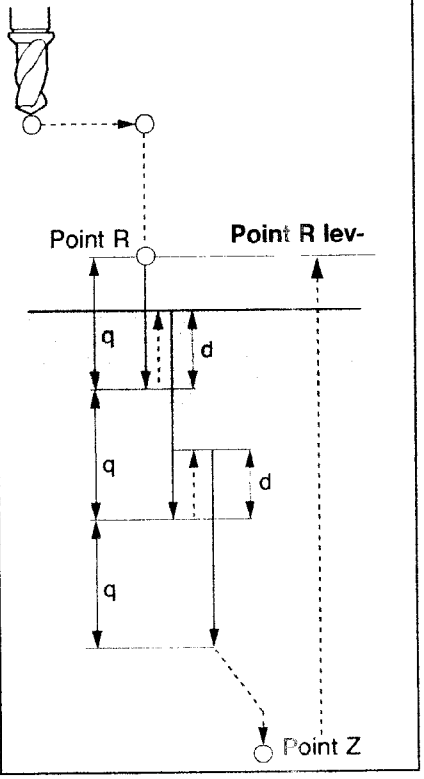
G03 : Circular interpolation or helical interpolation (CCW)

G73 (HIGH SPEED PECK DRILLING)

[Format]

G73 X_ Y_ Z_ R_ Q_ F_ K_ ;

X_Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
Q_	: Depth of cut for each cutting feed
F_	: Cutting feedrate
K_	: Number of repeats

073 (G98)	073 (G99)
	

(Symbols in figures)

----▶	Positioning (rapid traverse G00)
——▶	Cutting feed (linear interpolation G01)
~~▶	Manual feed
OSS	Oriented spindle stop(The spindle stops at a fixed rotation position)
⇨	Shift (rapid traverse G00)
P	Dwell

[Explanations]

The high–speed peck drilling cycle performs intermittent feeding along the Z–axis. When this cycle is used, chips can be removed from the hole easily, and a smaller value can be set for retraction. This allows, drilling to be performed efficiently. Set the clearance, d, in parameter 5114. The tool is retracted in rapid traverse.

Before specifying G73, rotate the spindle using a miscellaneous function (M code).

When the G73 code and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only ; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

[Restrictions]

- **Axis switching**

  Before the drilling axis can be changed, the canned cycle must be canceled.

- **Drilling**

  In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.

- **Q/R**

  Specify Q and R in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

- **Cancel**

  Do not specify group 01 G code (G00 to G03) and G73 in the same block.

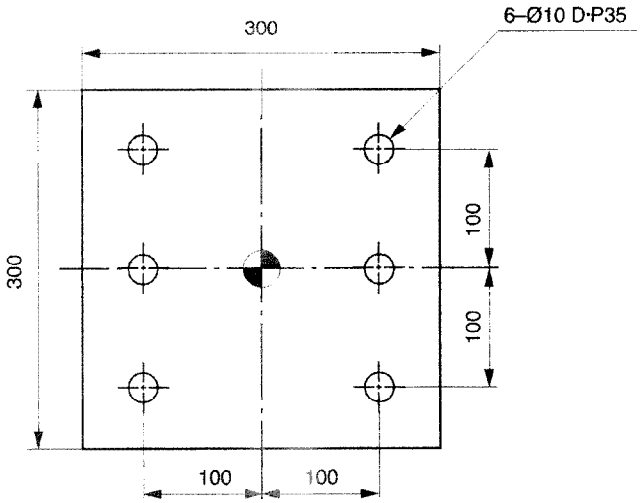
  If they are specified together, G73 is canceled.

- **Tool offset**

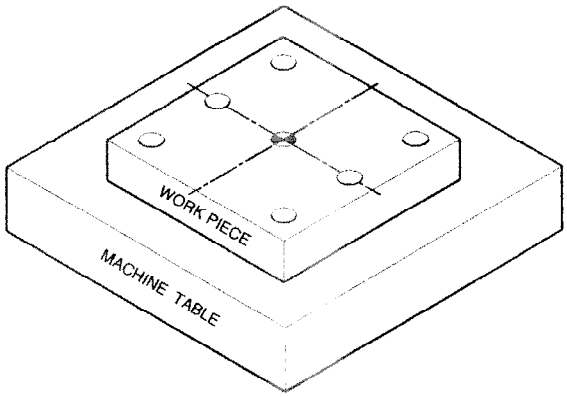
  In the canned cycle mode, tool offsets are ignored.

[Examples]

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G73 X300. Y-250. Z-150. R-100. Q15. F120 ;	
	Position, drill hole 1, then return to point R.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.




```
G90 G54 X-100. Y100 S1200 M03□
G43 Z50. H01 M08□
G98 G73 Z-40. R3. Q10. F240□
Y0□
Y-100□
X100□
Y0□
Y100□
G80 M09□
G91 G28 Z0 M05□
 •□
 •□
 •□
 •□
M30□
□
```

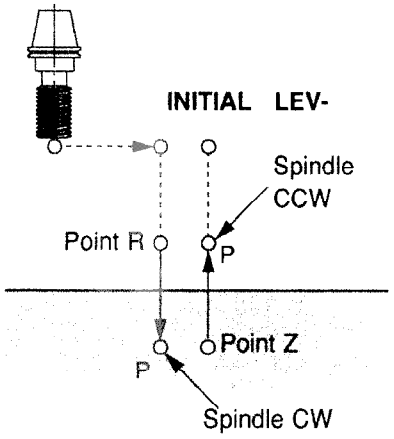
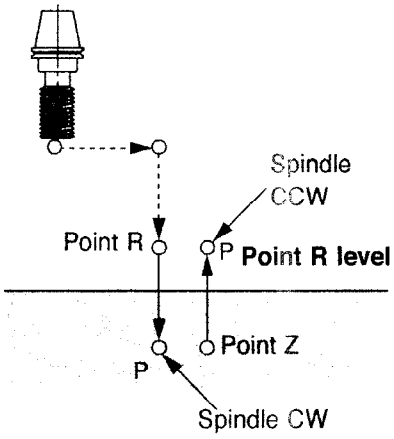


G74 (LEFT HAND TAPPING)

[Format]

G74 X_ Y_ Z_ R_ F_ K_ ;

X_Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
F_	: Cutting feedrate
K_	: Number of repeats

G74 (G98)	G74 (G99)
	

[Explanations]

Tapping is performed by turning the spindle counterclockwise. When the bottom of the hole has been reached, the spindle is rotated clockwise for retraction. This creates a reverse thread.

Feedrate overrides are ignored suring left–handed tapping. A feed hold does not stop the machine until the return operation is completed.

Before specifying G74, use a miscellaneous function (M code) to rotate the spindle counterclockwise.

When the G74 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation. When K is used to specity the number of repeats, the M code is executed for the first hole only ; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

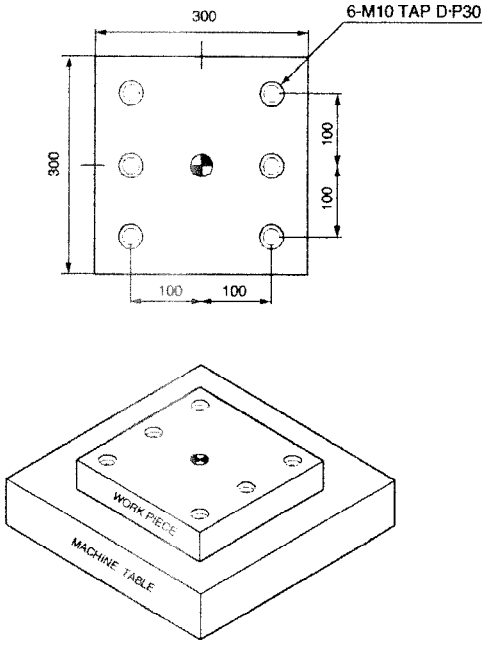
[Restrictions]

- **Axis switching**
  Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling**
  In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **R**
  Specify R in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.
- **Cancel**
  Do not specify group 01 G code (G00 to G03) and G74 in the same block.
  If they are specified together, G74 is canceled.
- **Tool offset**
  In the canned cycle mode, tool offsets are ignored.

[Examples]

M4 S100 ;	Cause the spindle to start rotating.
G90 G99 G74 X300. Y-250. Z-150. R-120. F120. ;	
	Position, drill hole 1, then return to point R.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

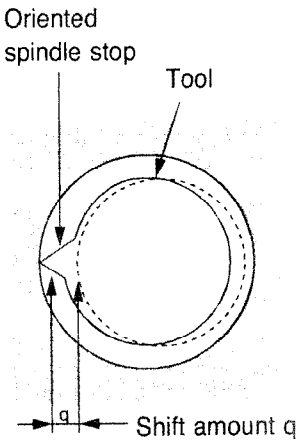
O




```
G90 G54 X-100. Y100 S300 M04
G43 Z50. H01 M08
G98 G74 Z-35. R3. F450
Y0
Y-100.
X100.
Y0
Y100.
G80 M09
G91 G28 Z0 M05
 .
 .
 .
 .
M30
```

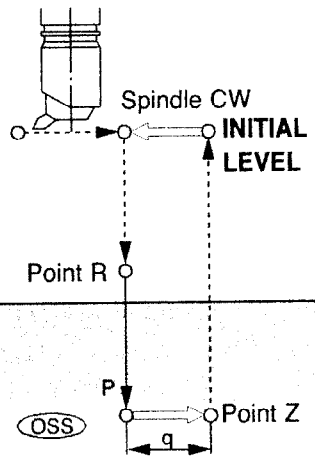
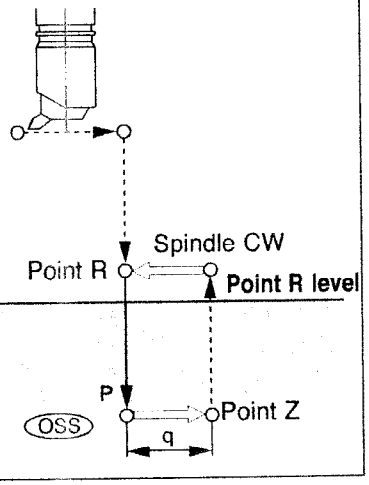
G76 (FINE BORING)

[Format]



G76 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;

X_Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
Q_	: Shift amount at the bottom of a hole
P_	: Dwell time at the bottom of a hole
F_	: Cutting feedrate
K_	: Number of repeats

G76 (G98)	G76 (G99)
	

Notes)

Q(shift at the bottom of a hole) is a modal value retained within canned cycles. It must be specified carefully because it is also used as the depth of cut for G73 and G83.

[Explanations]

When the bottom of the hole has been reached, the spindle is stopped at the fixed rotation position, and the tool is moved in the direction opposite to the tool tip and retracted. This ensures that the machined surface is not damaged and enables precise and efficient boring to be performed.

Before specifying G76, use a miscellaneous function (M code) to rotate the spindle.

When the G76 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only ; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

[Restrictions]

- **Axis switching**

  Before the drilling axis can be changed, the canned cycle must be canceled.

- **Drilling**

  In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.

- **Q/R**

  Be sure to specify a positive value in Q. If Q is specified with a negative value, the sign is ignored. Set the direction of shift in bits 4 (RD1) and 5(RD2) of parameter 5101. Specify Q and R in a block that performs boring. If they are specified in a block that does not perform boring, they are not stored as modal data.

- **Cancel**

  Do not specify group 01 G code (G00 to G03) and G76 in the same block.

  If they are specified together, G76 is canceled.

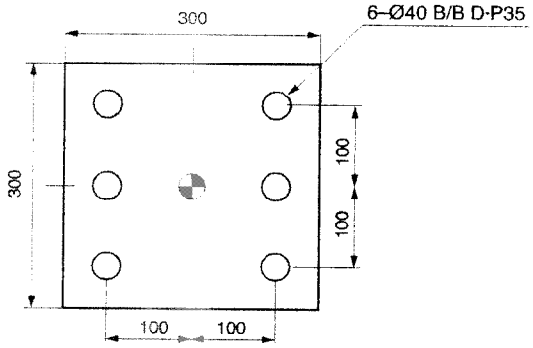
- **Tool offset**

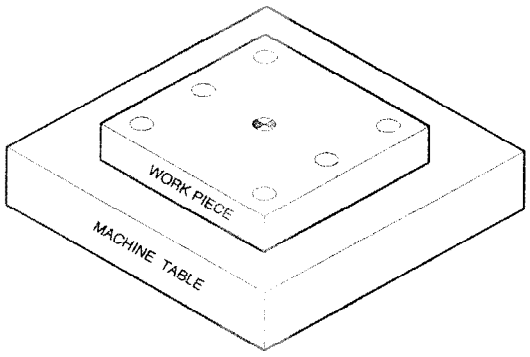
  In the canned cycle mode, tool offsets are ignored.

[Examples]

M3 S500 ;	Cause the spindle to start rotating.
G90 G99 G76 X300. Y-250.	Position, bore hole 1, then return to point R.
Z-150. R-120. Q5.	Orient at the bottom of the hole, then shift by 5mm.
P1000 F120. ;	Stop at the bottom of the hole for 1 s.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

o





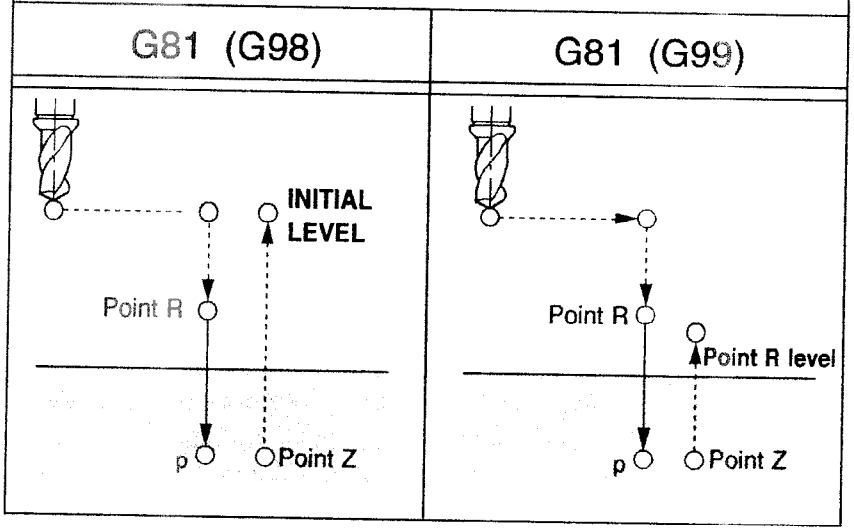

```
G90  G54  X–100. Y100   S800 M03
G43  Z50. H01 M08
G98  G76  Z–35. R3. Q5. F120
Y0
Y–100
X100
Y0
Y100.
G80 M09
G91  G28  Z0  M05
 .
 .
 .
 .
M30
```

G81 (DRILLING, SPOT FACE)

[Format]

G81 X_ Y_ Z_ R_ F_ K_ ;

X_Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
F_	: Cutting feedrate
K_	: Number of repeats



[Explanations]

After positioning along the X–and Y–axes, rapid traverse is performed to point R.

Drilling is performed from point R to point Z.

The tool is then retracted in rapid traverse.

Before specifying G81, use a miscellaneous function (M code) to rotate th spindle.

When the G81 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is performed for the first hole only ; for the second and subsequent holes, the M code is not executed.

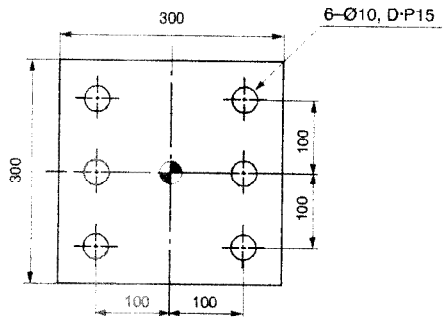
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioninf to point R.

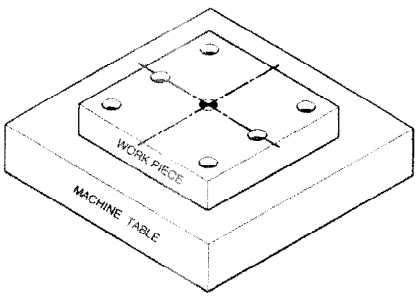
[Restrictions]

- **Axis switching**
  Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling**
  In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **R**
  Specify R in blocks that perform dilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.
- **Cancel**
  Do not specify group 01 G code (G00 to G03) and G81 in the same block.
  If they are specified together, G81 is canceled.
- **Tool offset**
  In the canned cycle mode, tool offsets are ignored.

Examples]

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G81 X300. Y-250. Z-150. R-100. F120. ;	Position, drill hole 1, then return to point R.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.





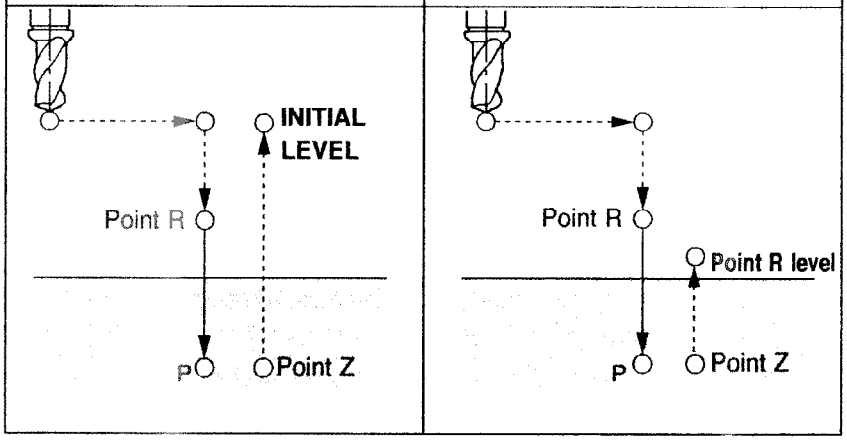

```
G90 G54 X-100. Y100 S1200 M03
G43 Z50. H01 M08
G98 G81 Z-15. R3. F240
Y0
Y-100.
X100.
Y0
Y100.
G80 M09
G91 G28 Z0 M05
 .
 .
 .
 .
M30
```

G82 (DRILLING WITH DWELL)

[Format]

G82 X_ Y_ Z_ R_ P_ F_ K_ ;

X_Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
P_	: Dwell time at the bottom of a hole
F_	: Cutting feedrate
K_	: Number of repeats



[Explanations]

After positioning along the X- and Y- axes, rapid traverse is performed to point R.

Drilling is then performed from point R to point Z.

When the bottom of the hole has been reached, a dwell is performed. The tool is then retracted in rapid traverse.

Before specicying G82, use a miscellaneous function (M code) to rotate the spindle.

When the G82 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation. When K is used to specify the number of repeats, the M code is executed for the first hole only ; for the second and subsequent holes, the M code is not executed.

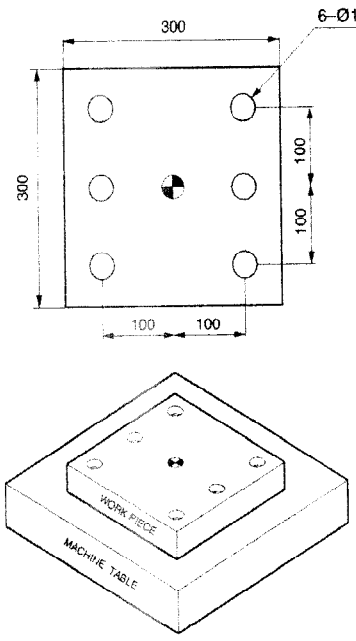
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

[Restrictions]

- **Axis switching**
  Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling**
  In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **R**
  Specify R in blocks that perform dilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.
- **Cancel**
  Do not specify group 01 G code (G00 to G03) and G82 in the same block.
  If they are specified together, G82 is canceled.
- **Tool offset**
  In the canned cycle mode, tool offsets are ignored.

Examples]

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G82 X300. Y-250. Z-150. R-100. P1000 F120. ;	Position, drill hole 1, and dwell for 1 s at the bottom of the hole, then return to point R.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.



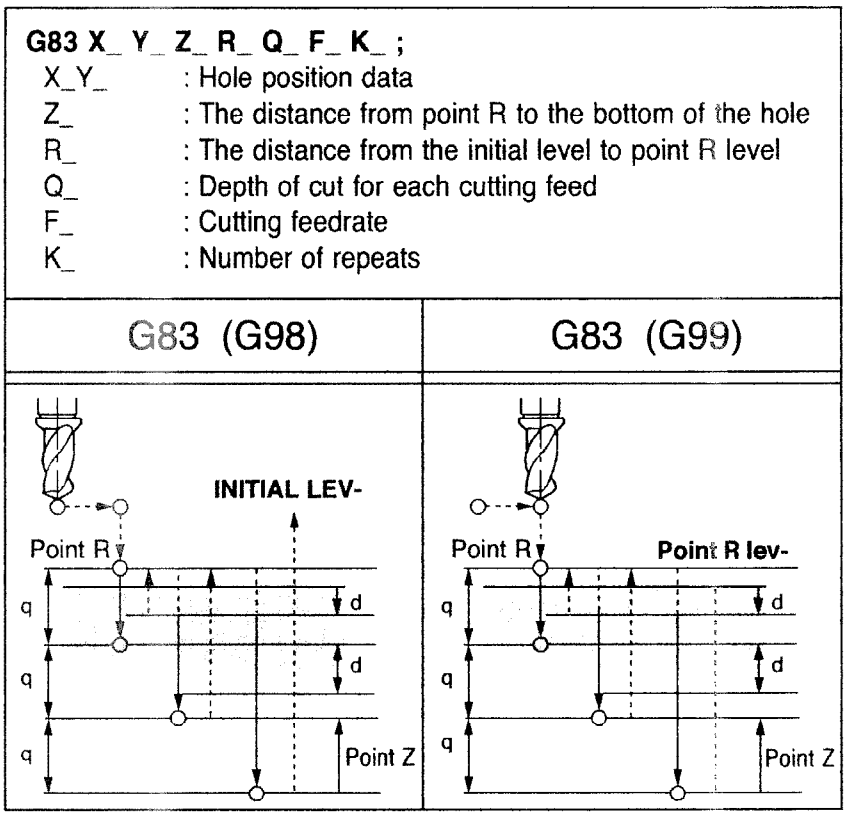

```
G90 G54 X-100. Y100 S700 M03□
G43 Z50. H01 M08□
G98 G82 Z-10. R3. P500. F120□
Y0□
Y-100□
X100□
Y0□
Y100.□
G80 M09□
G91 G28 Z0 M05□
 •□
 •□
 •□
 •□
M30□
□
```

G83 (PECK DRILLING)

[Format]

G83 X_ Y_ Z_ R_ Q_ F_ K_ ;

X_Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
Q_	: Depth of cut for each cutting feed
F_	: Cutting feedrate
K_	: Number of repeats



[Explanations]

Q represents the depth of cut for each cutting feed. It must always be specified as an incremental value.

In the second and subsequent cutting feeds, rapid traverse is performed up to a point just before where the last drilling ended, and cutting feed is performed again.

Be sure to specify a positive value in Q. Negative values are ignored.

Before specifying G83, use a miscellaneous function (M code) to rotate the spindle.

When the G83 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation. When K is used to specify the number of repeats, the M code is executed for the first hole only ; for the second and subsequent holes, the M code is not executed.

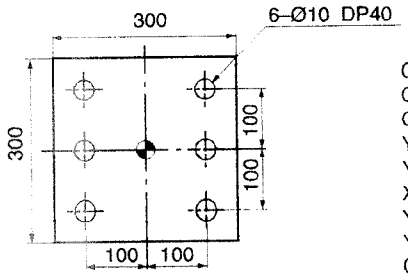
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

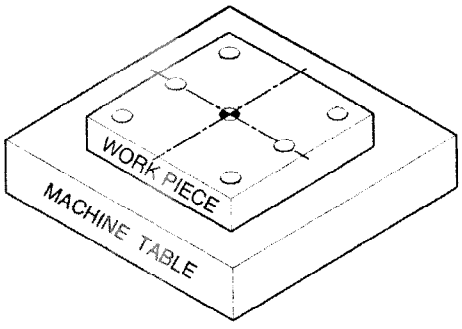
[Restrictions]

- **Axis switching**
  Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling**
  In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **Q./R**
  Specify Q and R in blocks that perform dilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.
- **Cancel**
  Do not specify group 01 G code (G00 to G03) and G83 in the same block.
  If they are specified together, G83 is canceled.
- **Tool offset**
  In the canned cycle mode, tool offsets are ignored.

Examples]

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G83 X300. Y-250. Z-150. R-100. Q15. F120. ;	Position, drill hole 1, then return to point R.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.





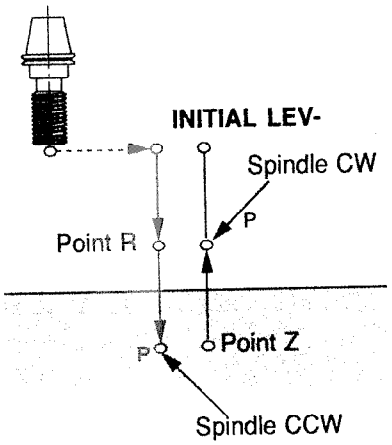
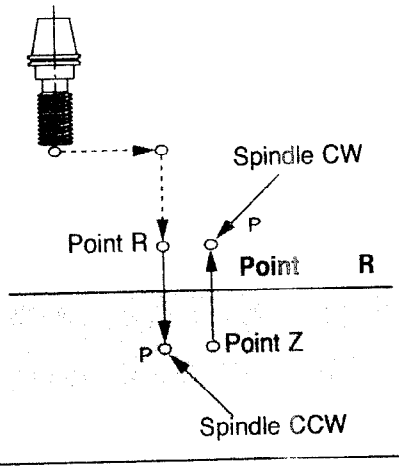

```
G90 G54 X-100. Y100. S1200 M03
G43 Z50. H01 M08
G98 G83 Z-45. R3. Q10. F240
Y0
Y-100
X100
Y0
Y100.
G80 M09
G91 G28 Z0 M05
.
.
.
.
M30
```

G84 (RIGHT HAND TAPPING)

[Format]

G84 X_ Y_ Z_ R_ F_ K_ ;

X_Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
F_	: Cutting feedrate
K_	: Number of repeats

G84 (G98)	G84 (G99)
	

[Explanations]

Tapping is performed by rotating the spindle clockwise. When the bottom of the hole has been reached, the spindle is rotated in the reverse direction for retraction. This operation creates threads.

Feedrate overrides are ignored during tapping. A feed hold does not stop, the machine until the return operation is completed.

Before specifying G84, use a miscellaneous function (M code) to rotate the spindle.

When the G84 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only ; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

[Restrictions]

- **Axis switching**

  Before the drilling axis can be changed, the canned cycle must be canceled.

- **Drilling**

  In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.

- **R**

  Specify R in blocks that perform dilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

- **Cancel**

  Do not specify group 01 G code (G00 to G03) and G84 in the same block.

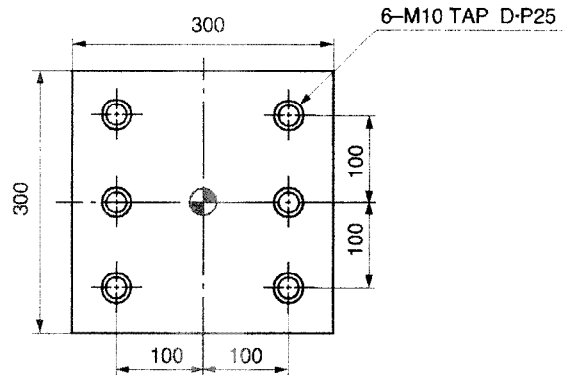
  If they are specified together, G84 is canceled.

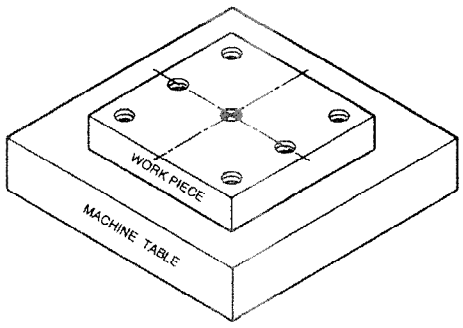
- **Tool offset**

  In the canned cycle mode, tool offsets are ignored.

[Examples]

M3 S100 ;	Cause the spindle to start rotating.
G90 G99 G84 X300. Y-250. Z-150. R-120. P300 F120. ;	
	Position, drill hole 1, then return to point R.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.





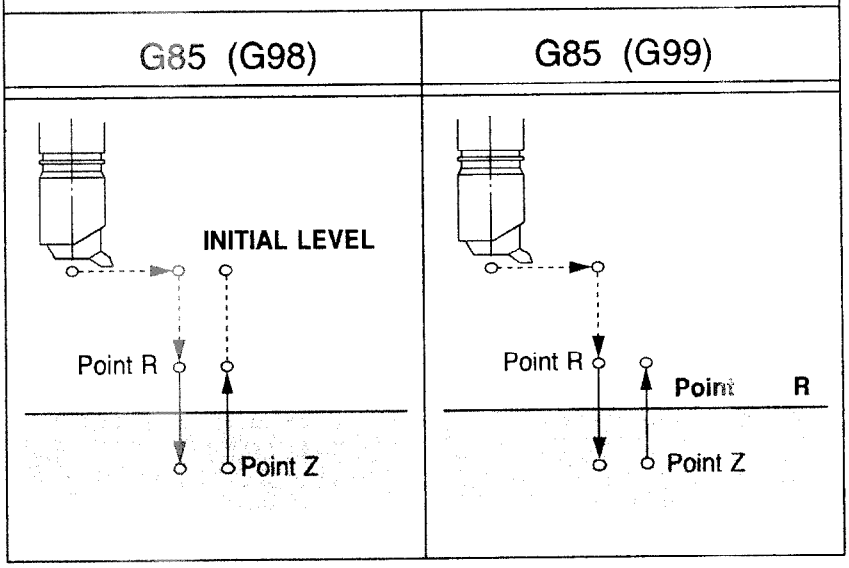
G90 G54 X-100. Y100. S300 M03□
G43 Z50. H01 M08□
G98 G84 Z-30. R3. F450□
Y0□
Y-100.□
X100.□
Y0□
Y100.□
G80 M09□
G91 G28 Z0 M05□
•□
•□
•□
•□
M30□
□

G85 (STANDARD BORING)

[Format]

G85 X_ Y_ Z_ R_ F_ K_ ;

X_Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
F_	: Cutting feedrate
K_	: Number of repeats



[Explanations]

After positioning along the X- and Y- axes, rapid traverse is performed to point R.

Drilling is performed from point R to point Z.

When point Z has been reached, cutting feed is performed to return to point R.

Before specifying G85, use a miscellaneous function (M code) to rotate the spindle.

When the G85 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only ; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

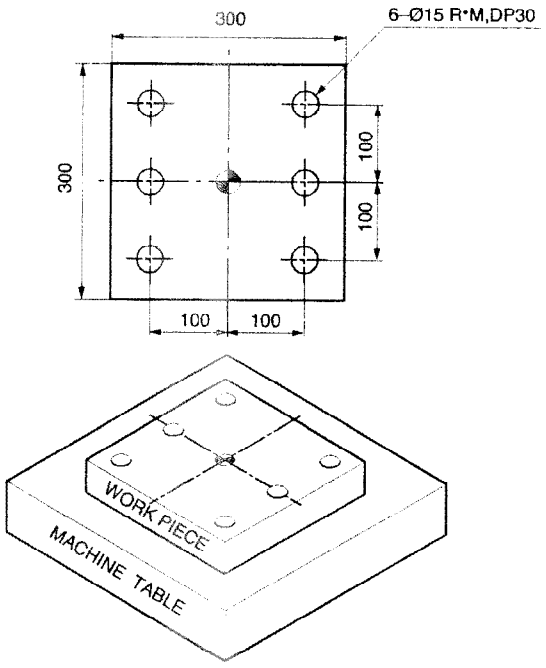
[Restrictions]

- **Axis switching**
  Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling**
  In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **R**
  Specify R in blocks that perform dilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.
- **Cancel**
  Do not specify group 01 G code (G00 to G03) and G85 in the same block.
  If they are specified together, G85 is canceled.
- **Tool offset**
  In the canned cycle mode, tool offsets are ignored.

[Examples]

M3 S100 ;	Cause the spindle to start rotating.
G90 G99 G85 X300. Y-250. Z-150. R-120. F120. ;	
	Position, drill hole 1, then return to point R.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

O



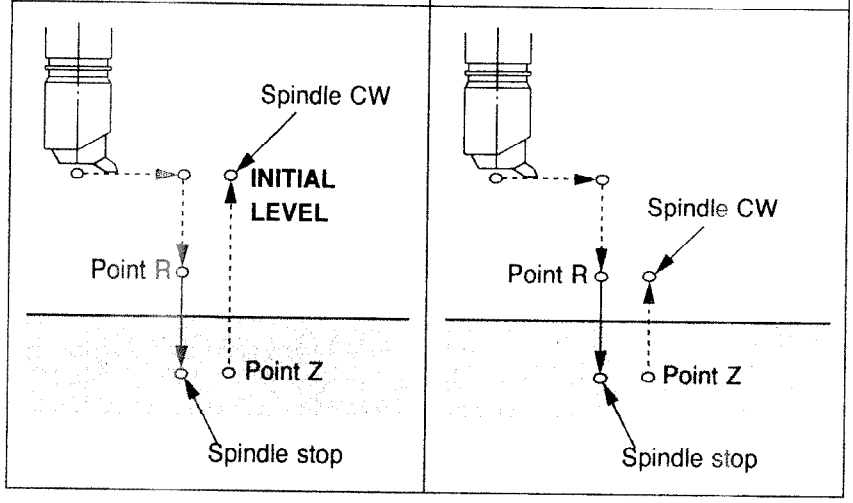

```
G90 G54 X-100. Y100. S200 M03
G43 Z50. H01 M08
G98 G85 Z-30. R3. F120
Y0
Y-100.
X100.
Y0
Y100.
G80 M09
G91 G28 Z0 M05
 .
 .
 .
 .
M30
```

G86 (BORING WITH SPINDLE STOP)

[Format]

G86 X_ Y_ Z_ R_F_ K_ ;

X_Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
F_	: Cutting feedrate
K_	: Number of repeats



[Explanations]

After positioning along the X- and Y- axes, rapid traverse is performed to point R.

Drilling is performed from point R to point Z.

When the spindle is stopped at the bottom of the hole, the tool is retracted in rapid traverse.

Before specifying G86, use a miscellaneous function (M code) to rotate the spindle.

When the G86 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only ; for the second and subsequent holes, the M code is not executed.

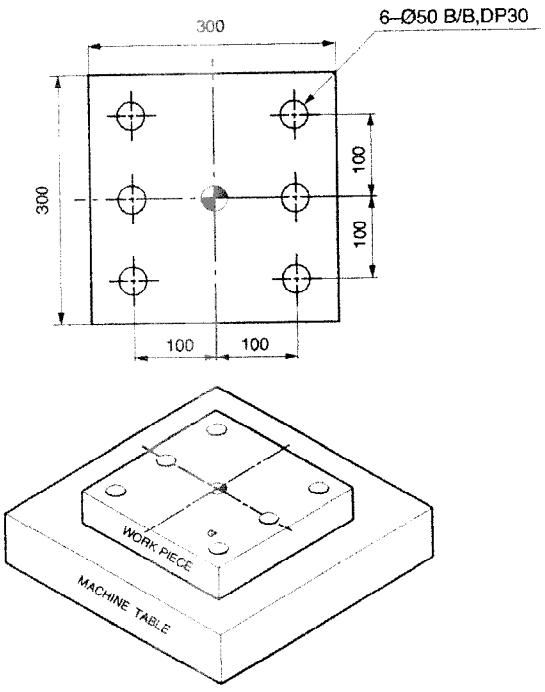
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

[Restrictions]

- **Axis switching**
  Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling**
  In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **R**
  Specify R in blocks that perform dilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.
- **Cancel**
  Do not specify group 01 G code (G00 to G03) and G86 in the same block.
  If they are specified together, G86 is canceled.
- **Tool offset**
  In the canned cycle mode, tool offsets are ignored.

Examples]

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G86 X300. Y-250. Z-150. R-100. F120. ;	Position, drill hole 1, then return to point R.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.




```
G90 G54 X-100. Y100. S1000 M03
G43 Z50. H01 M08
G98 G86 Z-30. R3. F120
Y0
Y-100.
X100.
Y0
Y100.
G80 M09
G91 G28 Z0 M05
 .
 .
 .
 .
M30
```

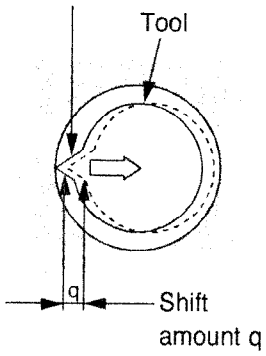
G87 (BACK BORING)

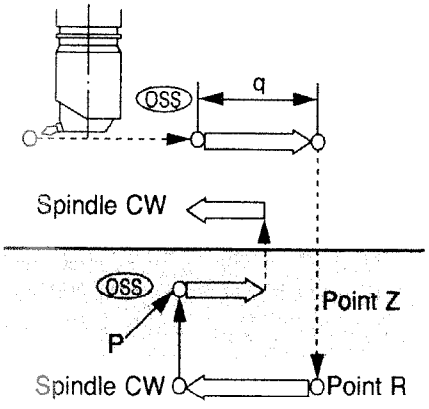
[Format]

G87 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;

X_Y_	: Hole position data
Z_	: The distance from the bottom of the hole to point Z
R_	: The distance from the initial level to point R (the bottom of the hole) level
Q_	: Tool shift amount
P_	: Dwell time
F_	: Cutting feedrate
K_	: Number of repeats

Oriented spindle stop



G87 (G98)	G87 (G99)
	Not used

Notes)

q (shift at the bottom of a hole) is a modal value retained in canned cycles. It must be specified carefully because it is also used as the depth of cut for G73 and G83.

Explanations]

After positioning along the X- and Y- axes, the spindle is stopped at the fixed rotation position. The tool is moved in the direction opposite to the tool tip, positioning (rapid traverse) is performed to the bottom of the hole (point R).

The tool is then shifted in the direction of the tool tip and the spindle is rotated clockwise. Boring is performed in the positive direction along the Z–axis until point Z is reached.

At point Z, the spindle is stopped at the fixed rotation position again, the tool is shifted in the direction opposite to the tool tip, then the tool is returned to the initial level. The tool is then shifted in the direction of the tool tip and the spindle is rotated clockwise to proceed to the next block operation.

Before specifying G87, use a miscellaneous function (M code) to rotate the spindle.

When the G87 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only ; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

[Restrictions]

- **Axis switching**

  Before the drilling axis can be changed, the canned cycle must be canceled.

- **Drilling**

  In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.

- **Q/R**

  Be sure to specify a positive value in Q. If Q is specified with a negative value, the sign is ignored. Set the direction of shift in bits 4 (RDI) and 5 (RD2) of parameter 5101. Specify Q and R in a block that performs boring. If they are specified in a block that does not perform boring, they are not soted as modal data.

- **Cancel**

  Do not specify group 01 G code (G00 to G03) and G87 in the same block.

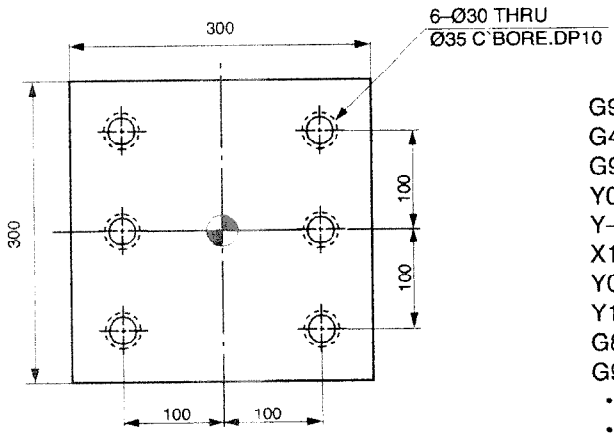
  If they are specified together, G87 is canceled.

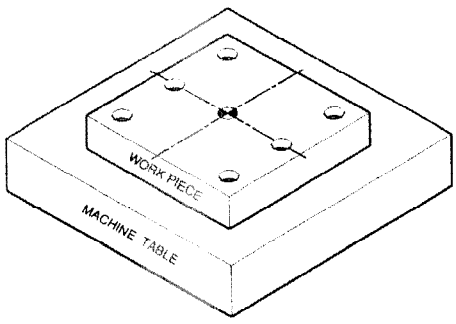
- **Tool offset**

  In the canned cycle mode, tool offsets are ignored.

Examples]

M3 S500 ;	Cause the spindle to start rotating.
G90 G54 X300. Y-250.	Position, bore hole 1.
G98 G87 Z-150. R-120. Q5.	Orient at the initial level, then shift by 5mm.
P1000 F120. ;	Stop at point Z for 1 s.
Y-550. ;	Position, drill hole 2
Y-750. ;	Position, drill hole 3
X1000. ;	Position, drill hole 4
Y-550. ;	Position, drill hole 5
Y-750. ;	Position, drill hole 6
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.





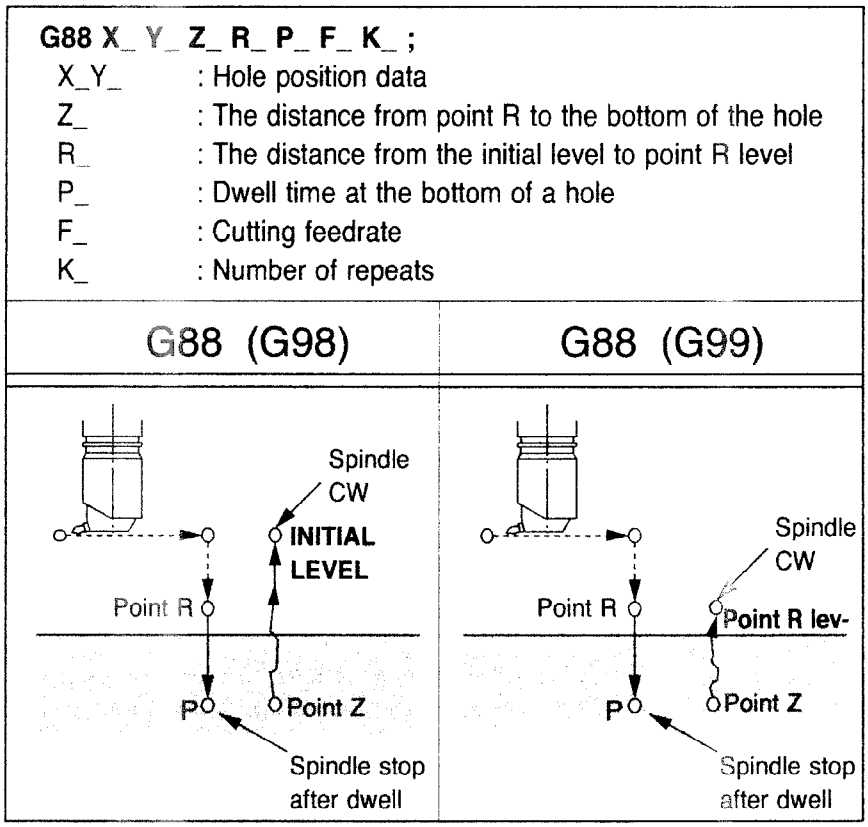

```
G90  G54  X–100. Y100. S1100 M03
G43  Z50. H01 M08
G98  G87  Z–60. R–75. Q3. F60
Y0
Y–100.
X100.
Y0
Y100.
G80 M09
G91  G28  Z0  M05
 .
 .
 .
 .
M30
```

G88 (BORING)

[Format]

G88 X_ Y_ Z_ R_ P_ F_ K_ ;

X_Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
P_	: Dwell time at the bottom of a hole
F_	: Cutting feedrate
K_	: Number of repeats



[Explanations]

After positioning along the X- and Y- axes, rapid traverse is performed to point R. Boring is performed from point R to point Z. When boring is completed, a dwell is performed, then the spindle is stopped. The tool is manually retracted from the bottom of the hole (point Z) to point R.

At point R, the spindle is rotated clockwise, and rapid traverse performed to the initial level.

Before specifying G88, use a miscellaneous function (M code) to rotate the spindle.

When the G88 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only ; for the second and subsequent holes, the M code is not executed.

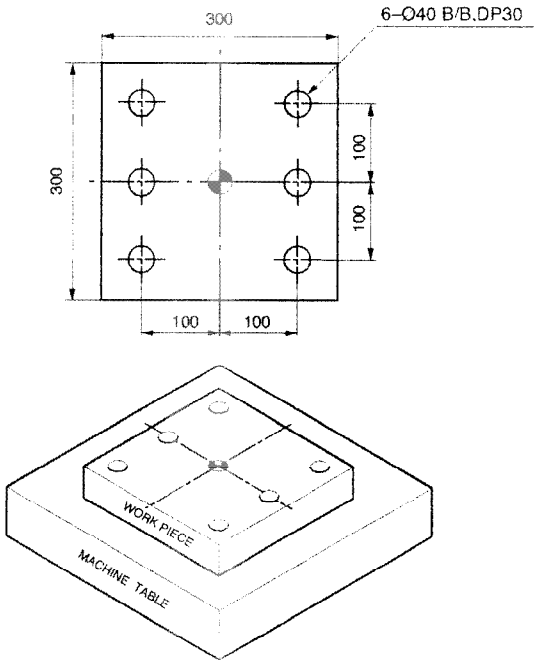
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

[Restrictions]

- **Axis switching**
  Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling**
  In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **R**
  Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.
- **Cancel**
  Do not specify group 01 G code (G00 to G03) and G88 in the same block.
  If they are specified together, G88 is canceled.
- **Tool offset**
  In the canned cycle mode, tool offsets are ignored.

Examples]

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G88 X300. Y-250. Z-150. R-100. P1000 F120. ;	
	Position, drill hole 1, return to point R
	then stop at the bottom of the hole for 1 s.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.



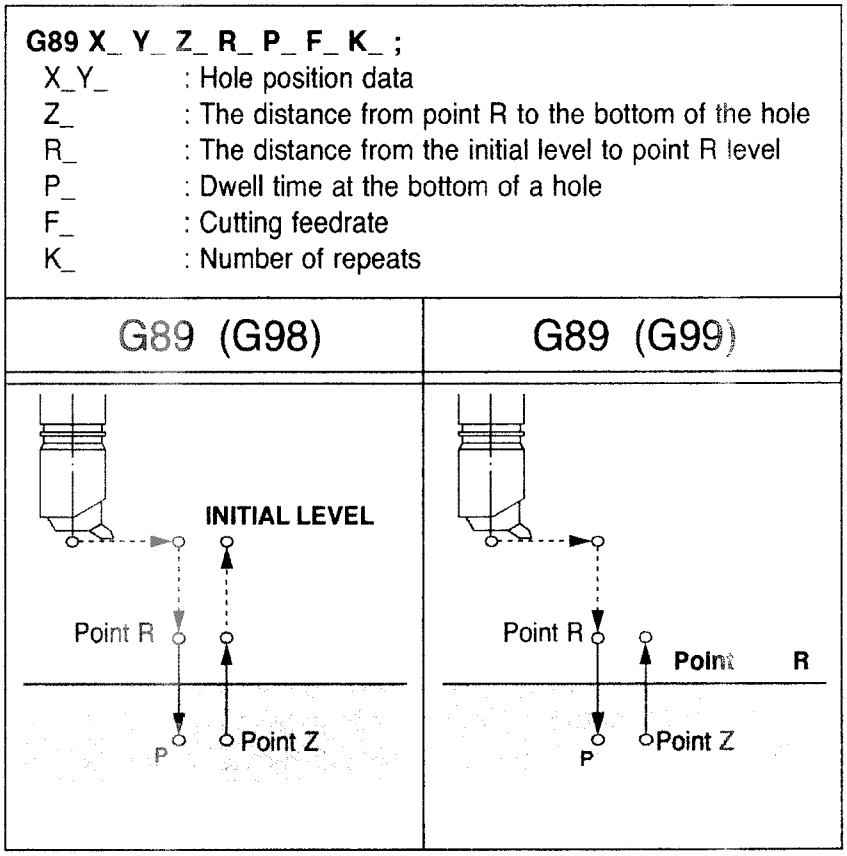

```
G90  G54  X-100. Y100. S1000 M03
G43  Z50. H01 M08
G98  G89  Z-30. R3. P500. F120
Y0
Y-100.
X100.
Y0
Y100.
G80 M09
G91  G28  Z0  M05
 .
 .
 .
 .
M30
```

G89 (BORING)

[Format]

G89 X_ Y_ Z_ R_ P_ F_ K_ ;

X_Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
P_	: Dwell time at the bottom of a hole
F_	: Cutting feedrate
K_	: Number of repeats



[Explanations]

This cycle is almost the same as G85. The difference is that this cycle performs a dwell at the bottom of the hole.

Before specifying G89, use a miscellaneous function (M code) to rotate the spindle.

When the G89 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only ; for the second and subsequent holes, the M code is not executed.

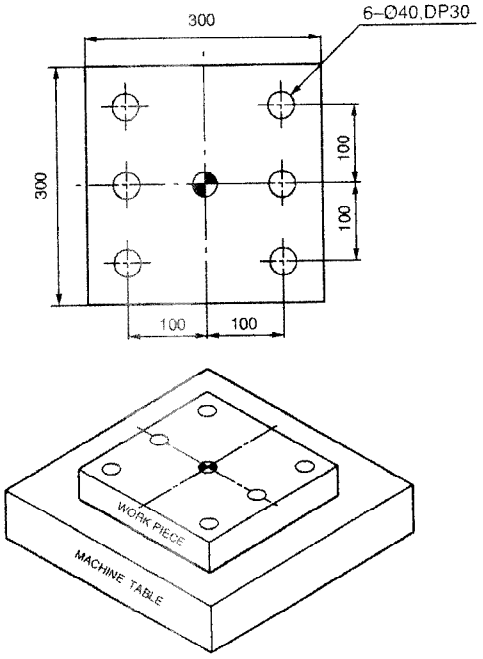
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

[Restrictions]

- **Axis switching**
  Before the drilling axis can be changed, the canned cycle must be canceled.
- **Drilling**
  In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.
- **R**
  Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.
- **Cancel**
  Do not specify group 01 G code (G00 to G03) and G89 in the same block.
  If they are specified together, G89 is canceled.
- **Tool offset**
  In the canned cycle mode, tool offsets are ignored.

Examples]

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G89 X300. Y-250. Z-150. R-100. P1000 F120. ;	Position, drill hole 1, return to point R then stop at the bottom of the hole for 1 s.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.




```
G90 G54 X-100. Y100. S1000 M03
G43 Z50. H01 M08
G98 G88 Z-30. R3. F120
Y0
Y-100.
X100.
Y0
Y100.
G80 M09
G91 G28 Z0 M05
 .
 .
 .
 .
M30
```

G80 : CANCEL CANNED CYCLE

[Format]

```
G80 ;
```

[Explanations]

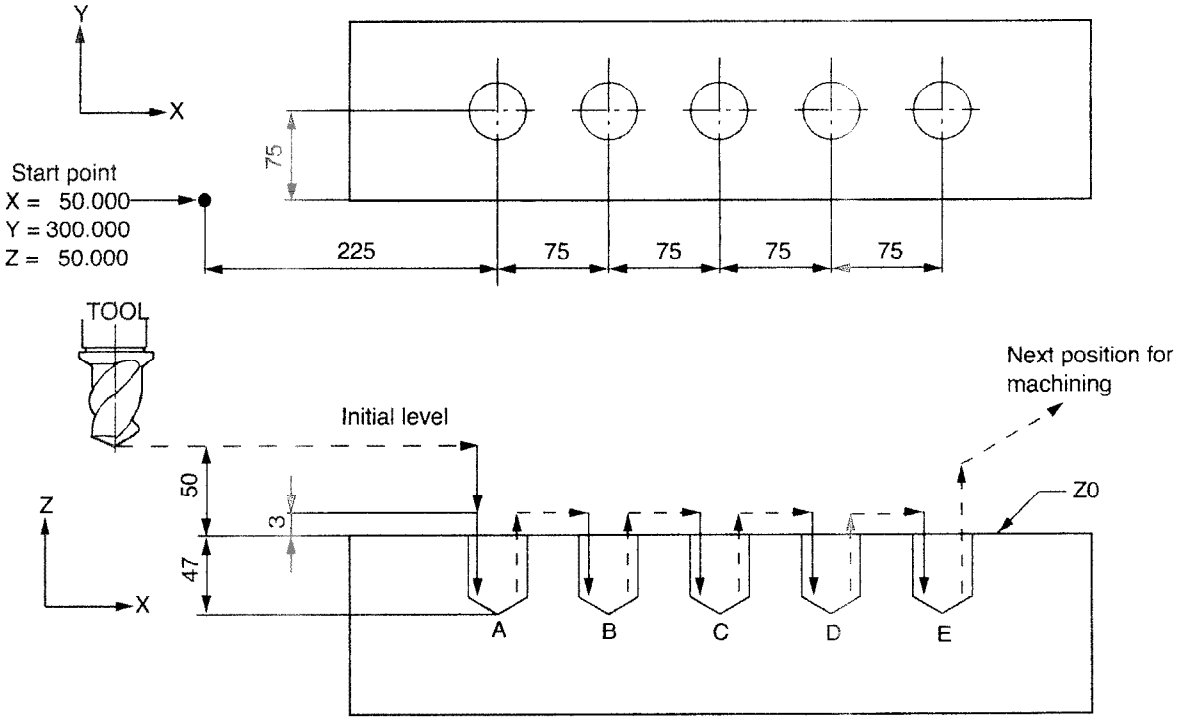
All canned cycles are canceled to perform normal operation. Point R and point Z are cleared.

This means that R=0 and Z=0 in incremental mode.

Other drilling data is also canceled (cleared).

[Example program of number of repeats K]

Unit : mm




```
G90   G54   G00   S1000  M03
G99   G81   X275.  Y375.  Z-47.  R3  F100
G91   X75.   K4
G99   G80   G91   Z100.
```

or

```
G91   G00   S1000  M03
G99   G81   X225.  Y75.   Z-50.  R-47.  F100
            X75.   K4
G00   G80   G90   Z100.
```

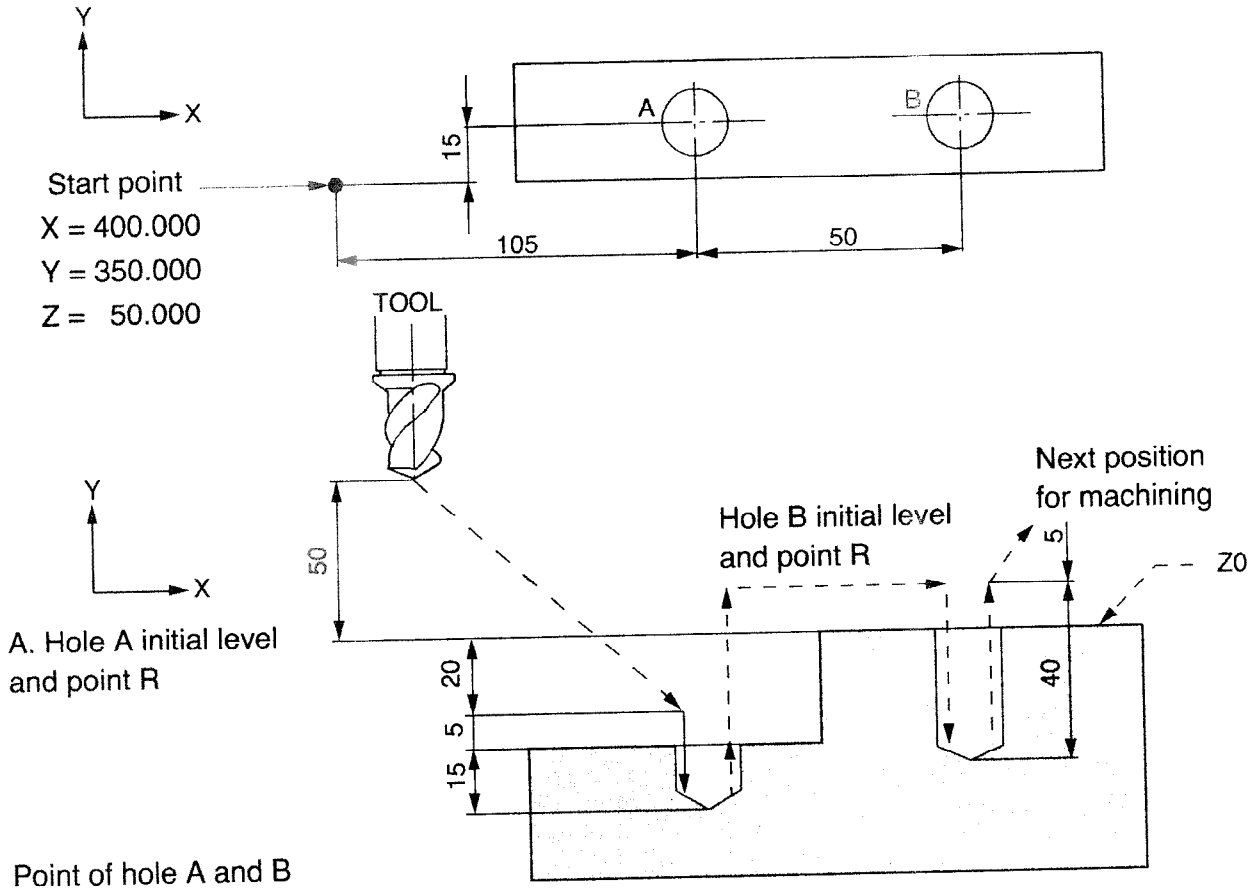
Note) K is effective only in the incremental mode.

If the mode is changed from G90(G91) to G91(G90) during the canned cycle, the previous specified mode is valid in the modified mode.

Number of repeats K should be specified at finished point of A

[Example program of different Z axis position]

Unit : mm



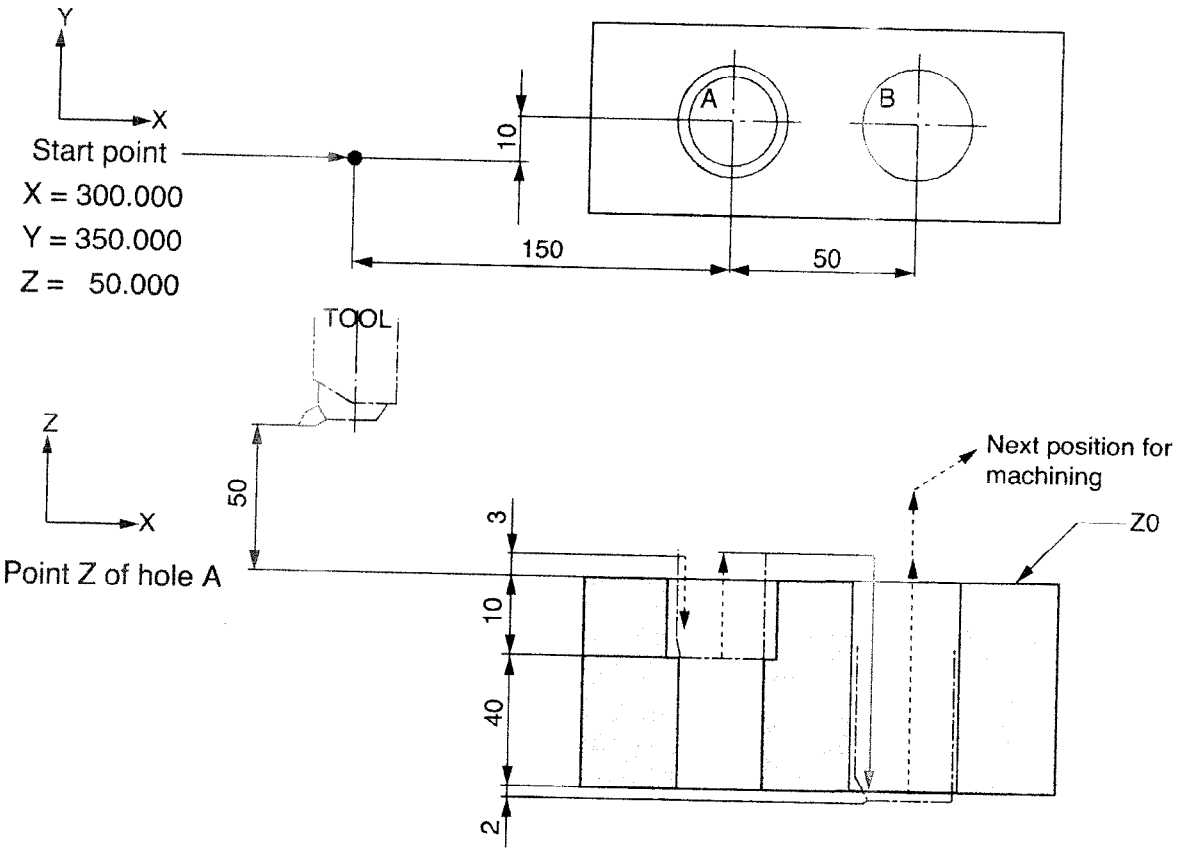

```
G90   G54   X505.  Y365.  Z50.   S960  M03
G99   G81   Z-40.  R-20.  F100
            X555.  R5
G00   G80   Z100.
```

Z axis position is -20.000 after machining hole A.

Normally the X and Y axes are positioned for next machining(hole B) without Z axis moving. But on the above program, because the data of R level for hole B is changed, the Z axis is positioned at next R point for machining hole B after Z axis is moved at changed position.

[Example program of different Z axis DEPTH]

Unit : mm




```
G90   G55   X450.  Y360.  Z50.   S1250  M03
G99   G82   Z-10.  R3.    P1000  F80
            X500.  Z-52.
G00   G80   Z100.
```

After positioning along 3-axes at the initial position for machining hole A, the cannned cycle starts.

Because the X-axis and Y-axis are positioned already, The canned cycle is executed by machining data.

RIGID TAPPING

The tapping cycle (G84) and left–handed tapping cycle (G74) may be performed in standard mode or rigid tapping mode.

In standard mode, the spindle is rotated and stopped along with a movement along the tapping axis using miscellaneous functions M03 (rotating the spindle clockwise), M04 (rotating the spindle counterclockwise), and M05 (stopping the spindle) to perform tapping. In rigid mode, tapping is performed by controlling the spindle motor as if it were a servo motor and by interpolating between the tapping axis and spindle.

When tapping is performed in rigid mode, the spindle rotates one turn every time a certain feed (thread lead) which takes place along the tapping axis. This operation does not vary even during acceleration or deceleration.

Rigid mode eliminates the need to use a floating tap required in the standard tapping mode, thus allowing faster and more precise tapping.

■ **Rigid mode**

Rigid mode can be specified using any of the following methods :

- Specify M29 S ┼R ┼R ┼R ┼R ┼Rbefore a tapping command.
- Specify M29 S ┼R ┼R ┼R ┼R ┼Rin a block which contains a tapping command.
- Specify G84 for rigid tapping (parameter No.5200 #0=1).

■ **Thread lead**

In feed–per–minute mode, the thread lead is obtained from the expression, feedrate X spindle speed. In feed–per–revolution mode, the thread lead equals the feedrate speed.

If a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

[Restrictions]

■ **Axis switching**

Before the drilling axis can be changed, the canned cycle must be canceled. If the drilling axis is changed in rigid mode, alarm (No. 206) is issued.

■ **S command**

If a speed higher than the maximum speed for the gear being used is specified, alarm (No.200) is issued.

■ **F command**

If a value exceeding the upper limit of cutting feedrate is specified, alarm (No.011) is issued.

■ **M29**

Specifying an S command and axis movement between M29 and G84 causes alarm (No.203). Then, speciying M29 in the tapping cycle causes alarm (No.204).

■ **R**

Specify R in a block that performs drilling. If R is specified in a non–drilling block, it is not stored as modal data.

■ **Cancel**

Do not specify a group 01 G code (G00 to G03) and G74 in the same block.

If they are specified together, G74 is canceled.

■ **Tool offset**

In the canned cycle mode, tool offsets are ignored.

■ **Unit of F**

	Metric input	Inch input	Remarks
G94	1mm/min	0.01inch/min	Decimal point programming allowed
G95	0.01mm/rev	0.0001inch/rev	Decimal point programming allowed

[Examples]

Z–axis feedrate 1000mm/min

Spindle speed 1000rpm

Thread lead 1.0mm

<Programming of feed per minute>

G94 ; Specify a feed–per–minute command.

G00 X120.0 Y100.0 ; Positioning

M29 S1000 ; Rigid mode specification

G84 Z-100.0 R-20.0 F1000 ; Rigid tapping

<Programming of feed per revolution>

G95 ; Specify a feed–per–revolution command.

G00 X120.0 Y100.0 ; Positioning

M29 S1000 ; Rigid mode specification

G84 Z-100.0 R-20.0 F1.0 ; Rigid tapping

G41 / G42

CUTTER COMPENSATION LEFT (G41)

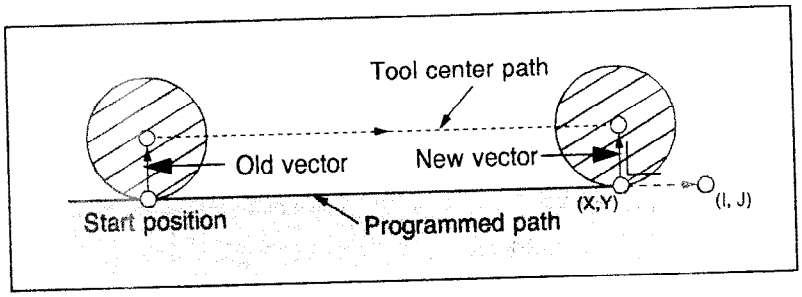
G41 offsets the tool towards the left of the workpiece as you see when you face in the same direction as the movement of the cutting tool.

[Explanations]

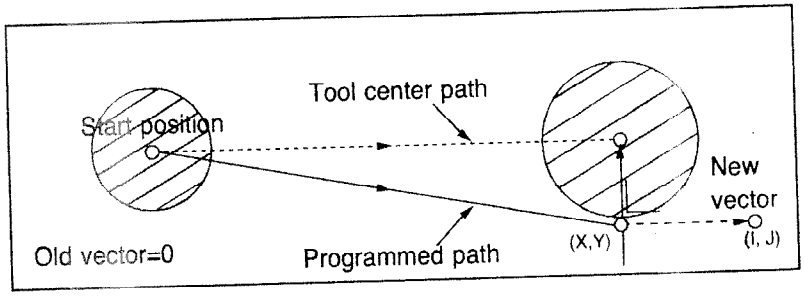
■ **G00 (positioning) or G01 (linear interpolation)**

G41 X_ Y_ I_ J_ H_ ;

Specifies a new vector to be created at right angles with the direction of (I, J) on the end point, and the tool center moves toward the point of the new vector from that of the old vector on the start point. (I, J) is expressed in an incremental value from the end point, and is significant only as a direction, and its amount is arbitrary.



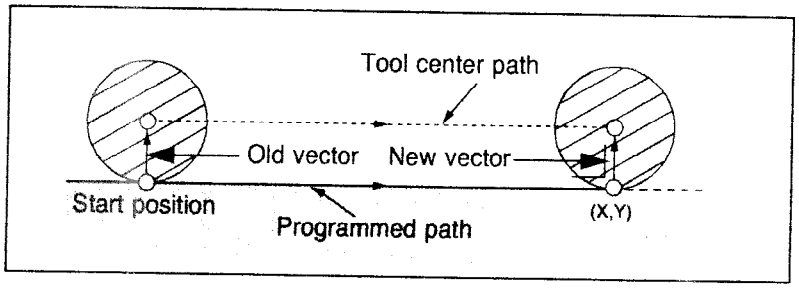
In case the old vector is 0, this command specifies the equipment to enter from the cancel mode into the cutter compensation mode. At this time, the offset number is specified by the H code.



Unless otherwise specified, (I, J(are assumed to be equal to (X, Y). When the following command is specified, a vector perpendicular to a line connecting the start position and position (X, Y) is created.

G41 X_ Y_ ;

If, however, G00 is specified, each axis moves independently at the rapid traverse rate.



■ **G02, G03 (Circular interpolation)**

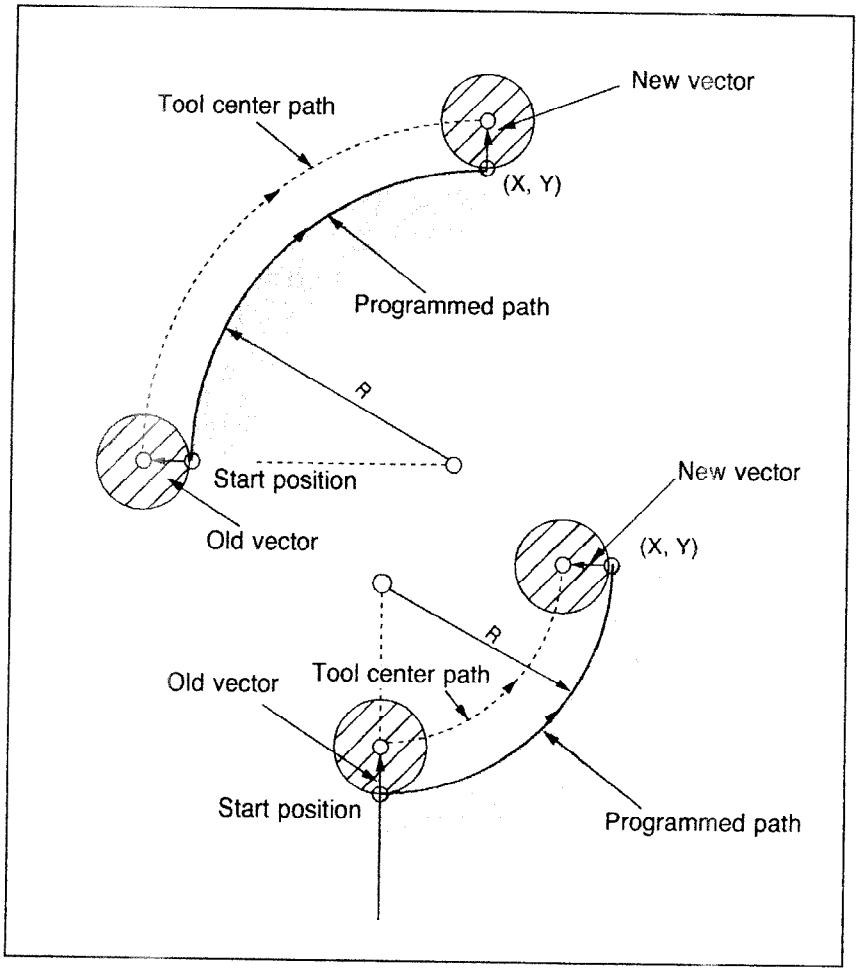
G41... ;

:

G02 (or G03) X_ Y_ R_ ;

Above command specifies a new vector to be created to the left looking toward the direction in which an arc advances on a line connecting the arc center and the arc end point, and the tool center to move along the arc advancing from the point of the old vector on the arc start point toward that of the new vector. This is, however, established on assumption the old vector is created correctly.

The offset vector is created toward the arc center or opposite direction against the arc center.



CUTTER COMPENSATION RIGHT (G42)

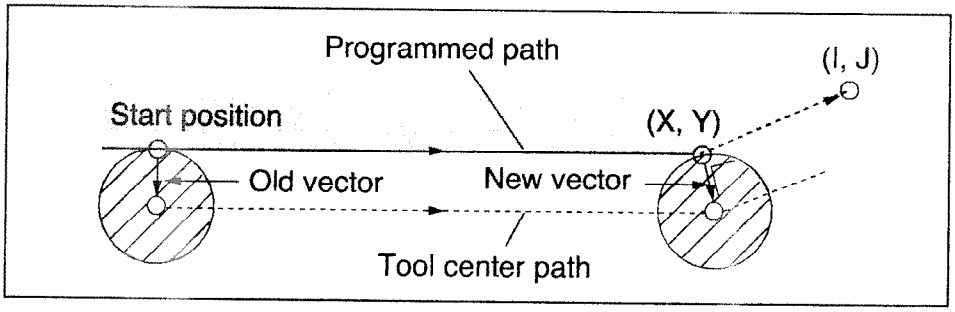
G42, contrary to G41, specifies a tool to be offset to the right of work piece looking toward the direction in which the tool advances.

G42 has the same function as G41, except that the directions of the vectors created by the commands are the opposite.

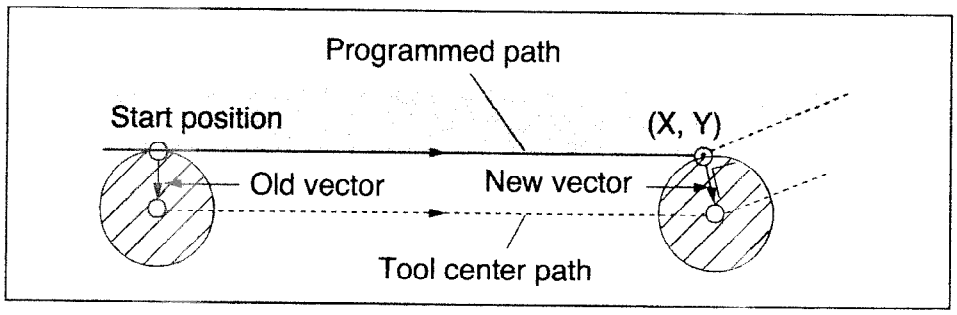
[Explanations]

■ **G00 (positioning) or G01 (linear interpolation)**

G42 X_Y_I_J_H_ ;



G42 X_Y_ ;



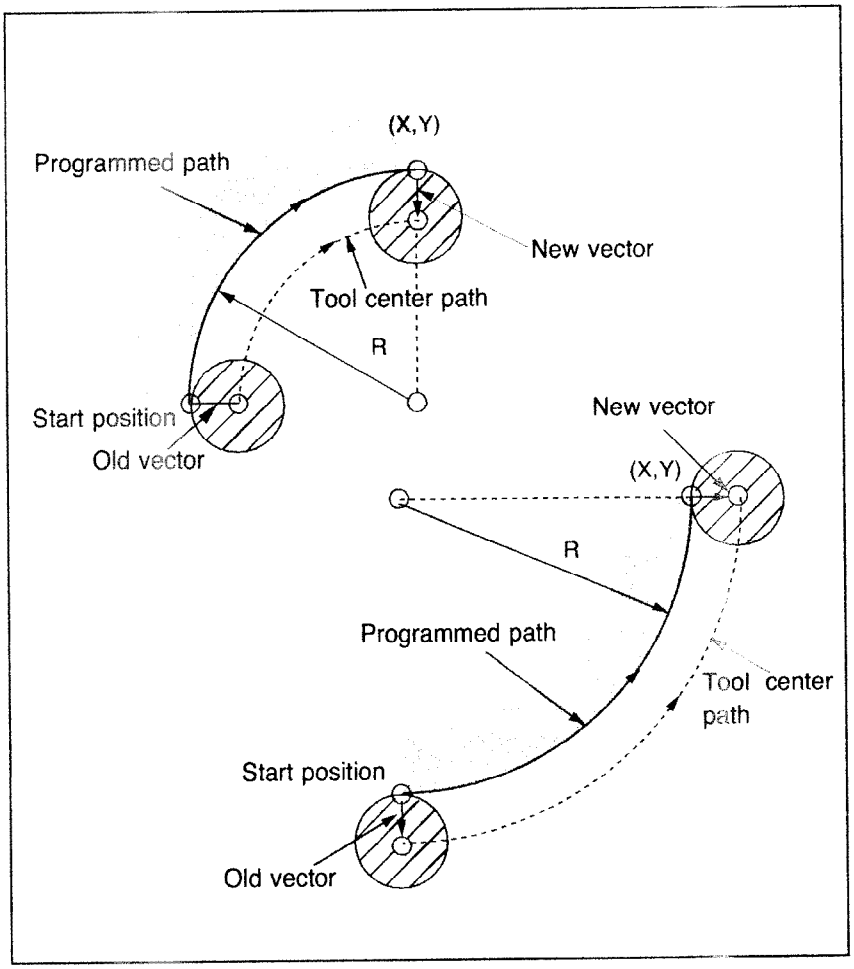
In the case of G00, however, each axis moves independently at the rapid traverse rate.

■ **G02 or G03 (Circular interpolation)**

G42 ... ;

:

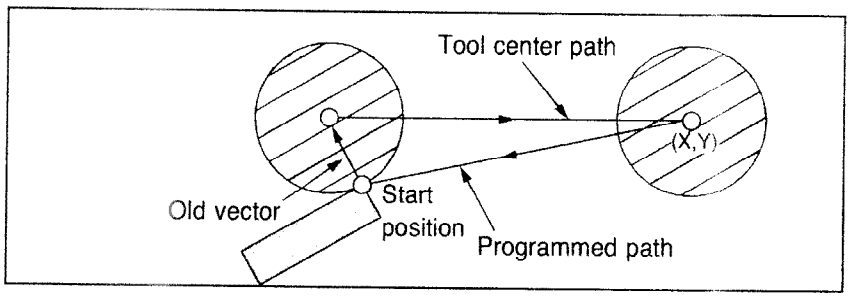
G02 (or G03) X_Y_R_ ;



CUTTER COMPENSATION CANCEL (G40)

When the following command is specified in the G00 or G01 mode, the tool moves from the head of the old vector at the start position to the end position (X, Y). In the G01 mode, the tool moves linearly. In the G00 mode, rapid traverse is carried out along each axis.

G40 X_Y_ ;



This command changes the mode of the equipment from the cutter compensation mode to the cancel mode.

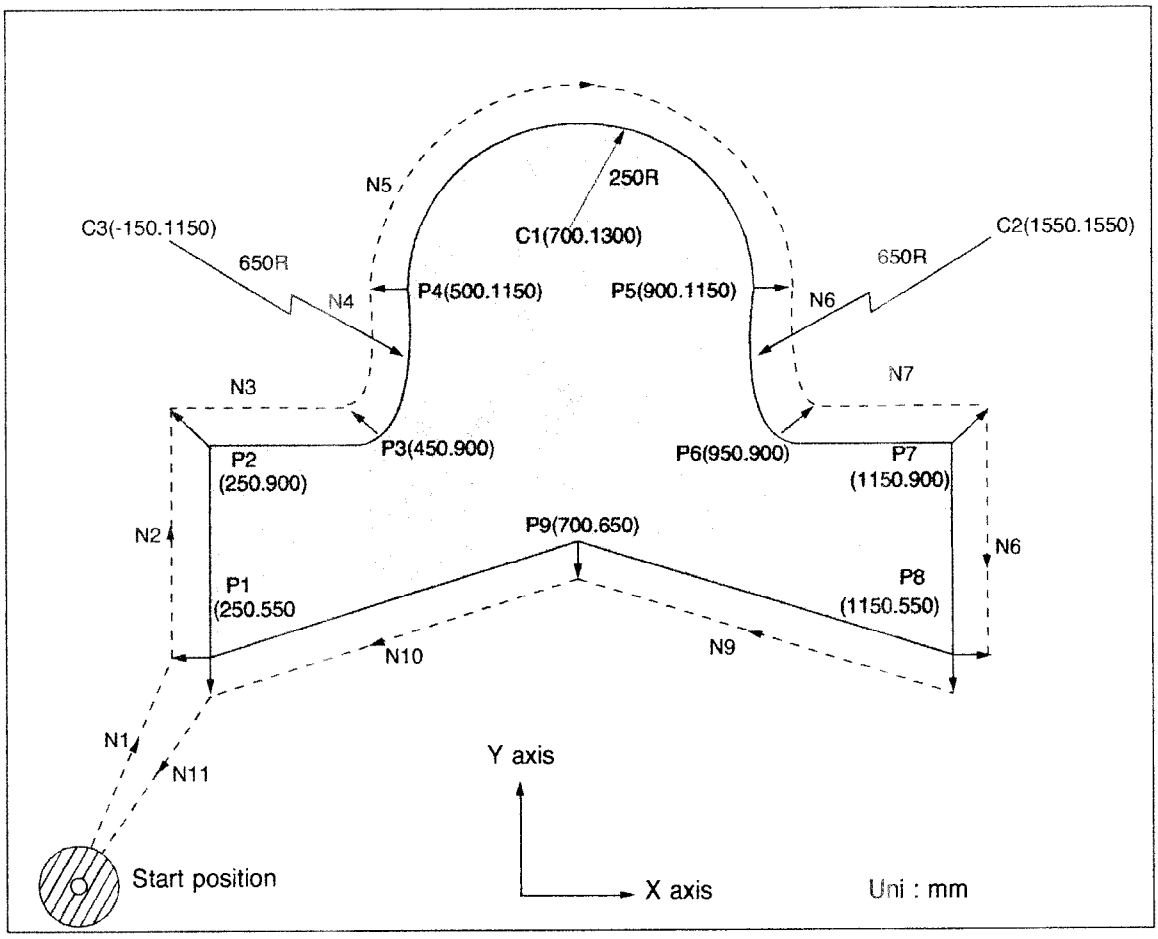
When only G40 ; is specified, and X_Y_ is not specified, the tool moves by the old vector amount in the opposite direction.

[Notes]

Notes)

Cutter compensation cannot be canceled in the circular interpolation (G02, G03) mode.

[Exanple 1]



G54 X0 Y0 Z0 ; ..... Specifies absolute coordinates.

N1 G90 G17 G00 G41 D07 X250.0 Y550.0 ; The tool positioned at the start position (X0, Y0, Z0). Starts cutter compensation (start–up). The tool is shifted to the left of the programmed path by the distance specified in D07.
In other words the tool path is shifted by the radius of the tool (offset mode) because D07 is set to 15 beforehand (the radius of the tool is 15mm).

N2 G01 Y900.0 F150 ; ..... Specifies machining from P1 to P2.

N3 X450.0 ; ..... Specifies machining from P2 to P3.

N4 G03 X500.0 Y1150.0 R650.0 ; ..... Specifies machining from P3 to P4.

N5 G02 X900.0 R-250.0 ; ..... Specifies machining from P4 to P5.

N6 G03 X950.0 Y900.0 R650.0 ; ..... Specifies machining from P5 to P6.

N7 G01 X1150.0 ; ..... Specifies machining from P6 to P7.

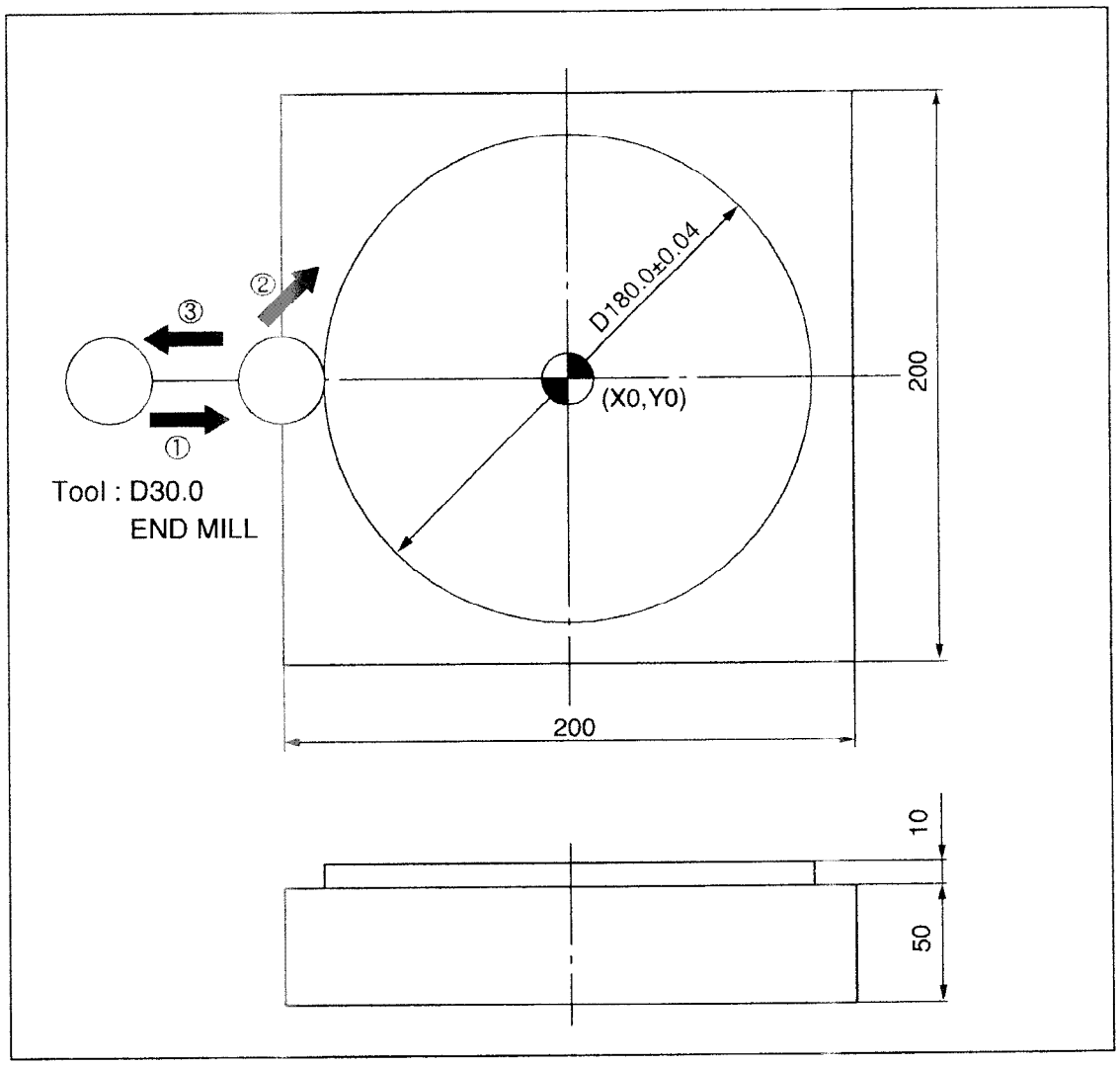
N8 Y550.0 ; ..... Specifies machining from P7 to P8.

N9 X700.0 Y650.0 ; ..... Specifies machining from P8 to P9.

N10 X250.0 Y550.0 ; ..... Specifies machining from P9 to P1.

N11 G00 G40 X0 Y0 ; ..... Cancels the offset mode.
The tool is returned to the start position (X0, Y0, Z0).

[Exanple 2]



N40 G90 G54 X-120. Y0 S600 M3 ;
G43 Z50. H4 ;
G01 Z-10. F2000 ;
G41 X-90. Y0 D25 F100 ; (Input 15.0 to offset No.25)
G02 I90 ;
G00 G40 X-120. Y0 ;
Z100. ;

Because the drawing dimension 9D180.0±0.04) should be kept, If the measured dimension is 181.0, modify the value in the offset No.25 from 15.0 to 145.

G43 TOOL LENGTH COMPENSATION

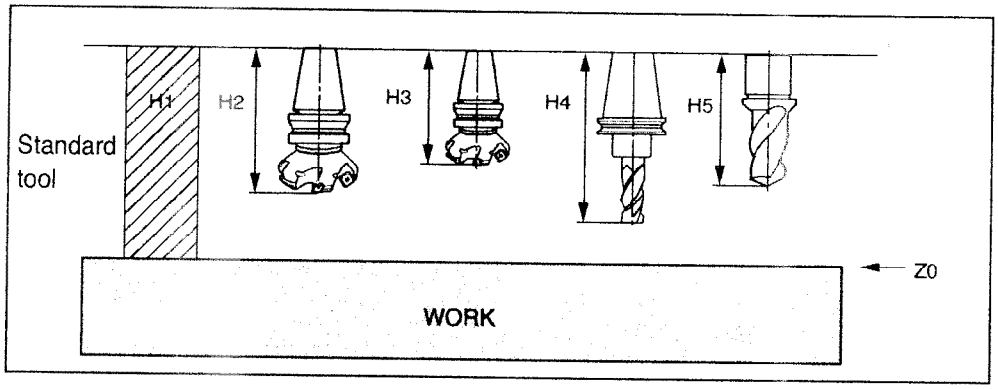
[Explanations]

- **Machining using the end of cutter–Tool length compensation function**

Usually, several tools are used for machining one workpiece. The tools have different tool length. It is very troublesome to change the program in accordance with the tools.

Therefore, the length of each tool used should be measured in advance. By setting the difference between the length of the standard tool and the length of each tool in the CNC (data display and setting : see III-11), machining can be performed without altering the program even when the tool is changed.

This function is called tool length compensation.



TOGETHER WITH G43 AN H-CODE NUMBER IS PROGRAMED TO ACTIVATE THE TOOL LENGTH OFFSET. eg. FOR T1 , H1 COULD BE USED, T2, H2 COULD BE USED.

EXAMPLE:
N10 T1 M6
N20 .....
N30 .....
N40 G0 G43 Z100.0 H1

G49 TOOL LENGTH COMPENSATION CANCEL

THIS COMMAND CANCELES THE TOOL LENGTH OFFSET.

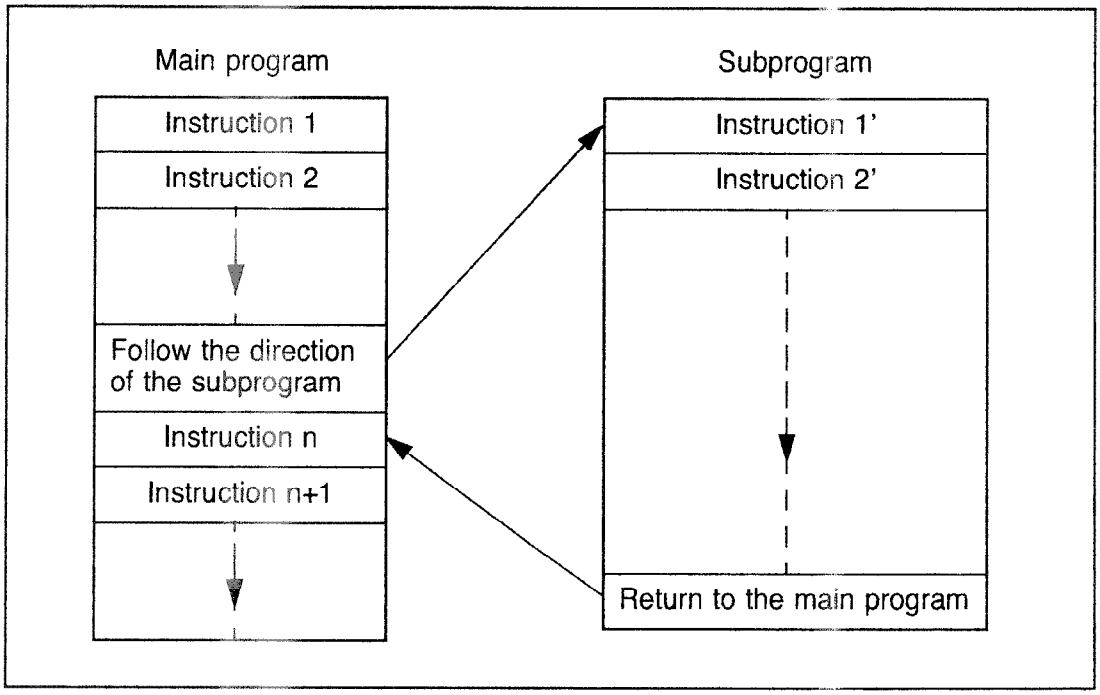
NOTE: ONLY USE THIS COMMAND IF THE TOOL IS WELL CLEAR OF THE WORKPIECE.

[redacted]

[General]

■ **Main program and subprogram**

There are two program types, main program and subprogram. Normally, the CNC operates according to the main program. However, when a command calling a subprogram is encountered in the main program, control is passed to the subprogram. When a command specifying a return to the main program is encountered in a subprogram, control is returned to the main program.



[Main program and Subprogram]

The CNC memory can hold up to 400 main programs and subprograms (63 as standard). A main program can be selected from the stored main programs to operate the machine.

1.2 SUBPROGRAM

If a program contains a fixed sequence or frequently repeated pattern, such a sequence or pattern can be stored as a subprogram in memory to simplify the program.

A subprogram can be called from the main program.

A called subprogram can also call another subprogram.

[Format]

■ **Subprogram configuration**

One subprogram

O□□□□ ;

Subprogram number
(or the colon(:) optionally in the case of ISO)

Program end

M99 need not constitute a separate block as indicated below.

Example) X100.0 Y100.0 M99 ;

■ **Subprogram call**

M98 P○○○○ ○○○○ ;

↑ Number of time the subprogram is called repeatedly

↑ Subprogram number

When no repetition data is specified, the subprogram is called just once.

[Examples]

☆ M98 P51002 ;

This command specifies "Call the subprogram (number 1002) five times in succession." A subprogram call command (M98P_) can be specified in the same block as a move command.

☆ X1000.0 M98 P1200 ;

This example calls the subprogram (number 1200) after an X movement.

☆ Execution sequence of subprogram called from a main program

Main program	**Subprogram**
N0010 O ;	N0010 O ;
N0020 O ;	N0020 O ;
N0030 M98 P21010 ;	N0030 M98 P21010 ;
N0040 O ;	N0040 O ;
N0050 M98 P21010 ;	N0050 M98 P21010 ;
N0060 O ;	N0060 O ;

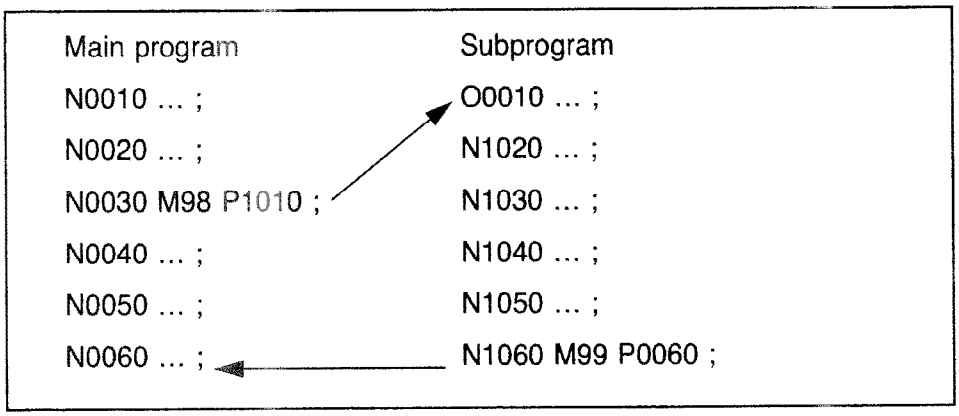
A subprogram can call another subprogram in the same way as a main program calls a subprogram.

[Special Usage]

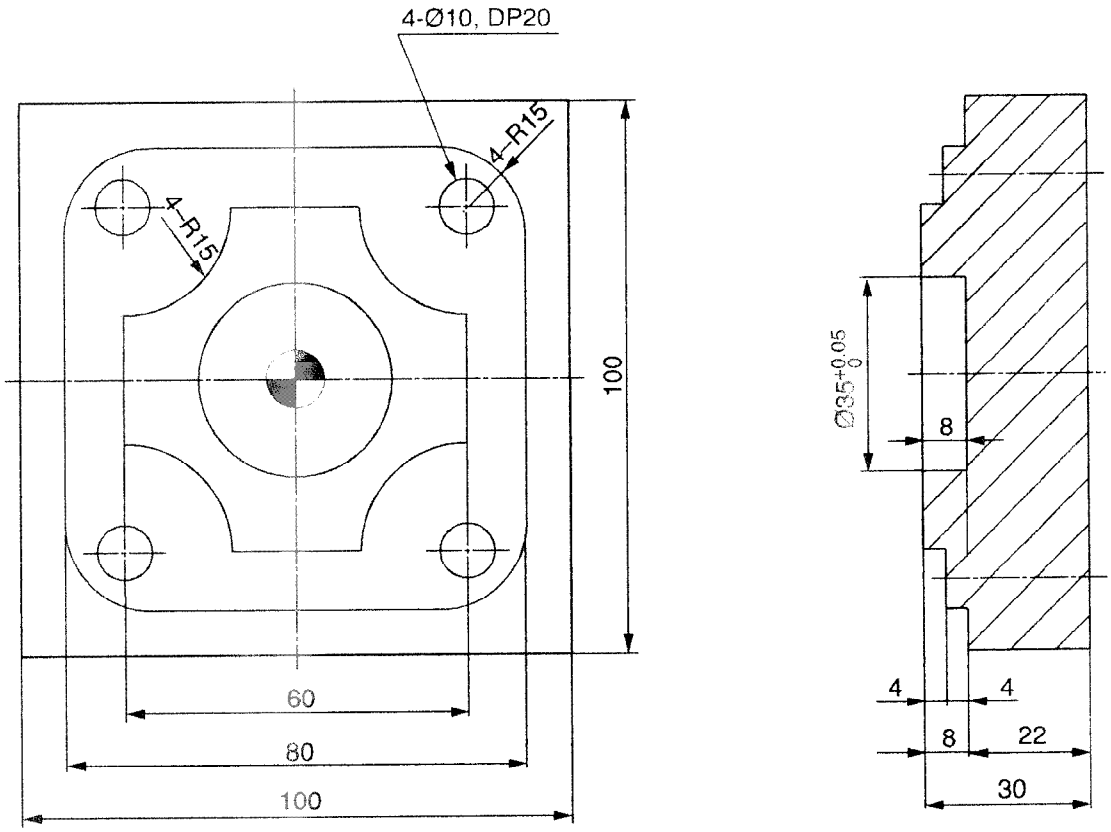
■ **Specifying the sequence number for the return destination in the main program**

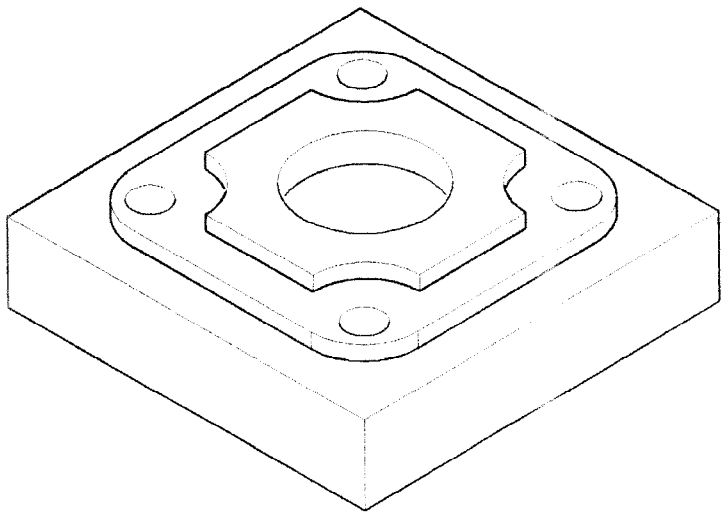
If P is used to specify a sequence number when a subprogram is terminated, control does not return to the block after the calling block, but returns to the block with the sequence number specified by P. Note, however, that P is ignored if the main program is operating in a mode other than memory operation mode.

This method consumes a much longer time than the normal return method to return to the main program.

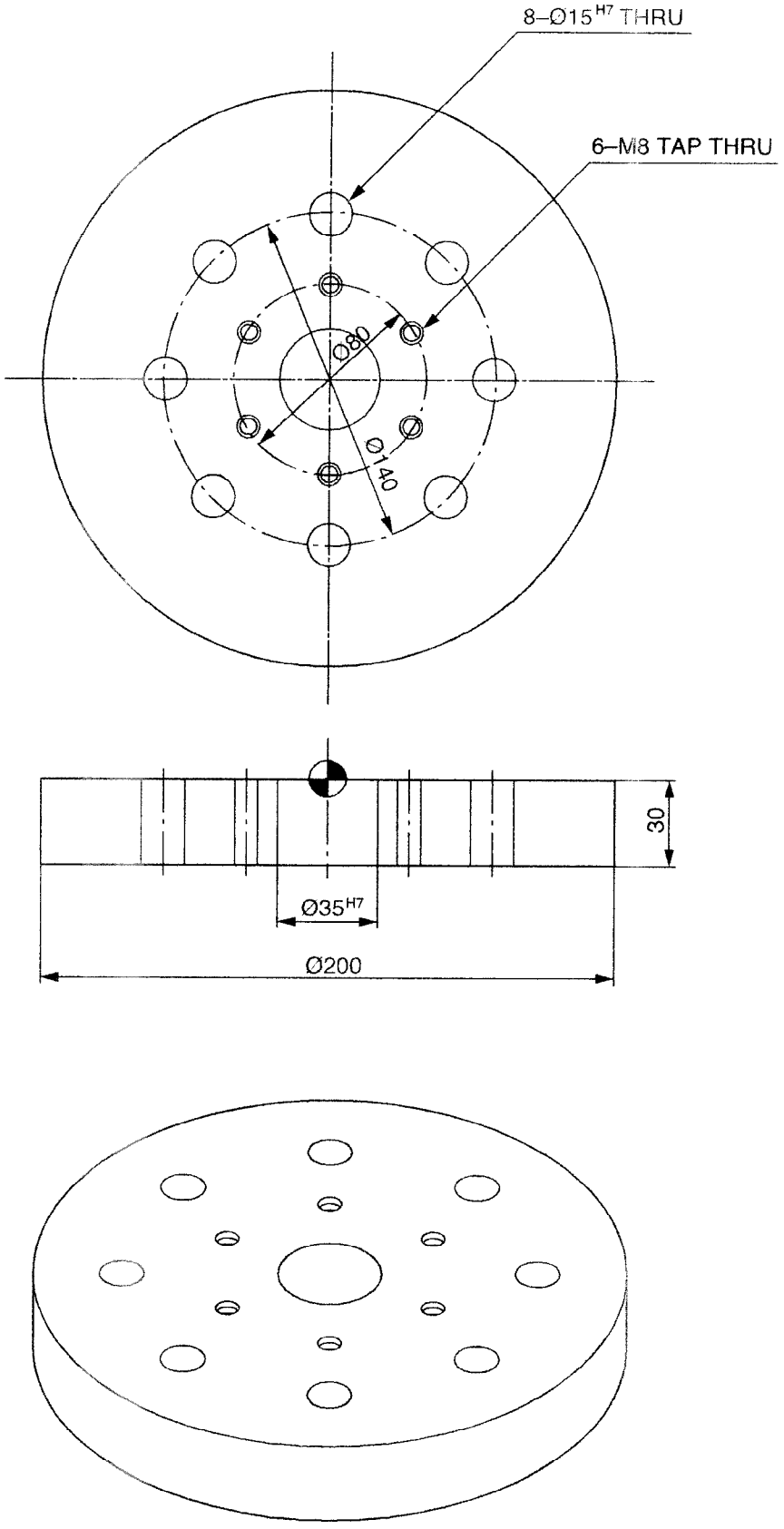


[Examples 1]

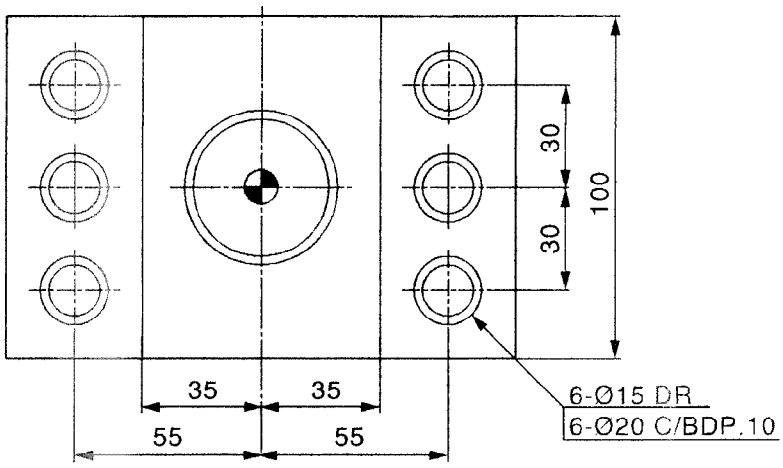


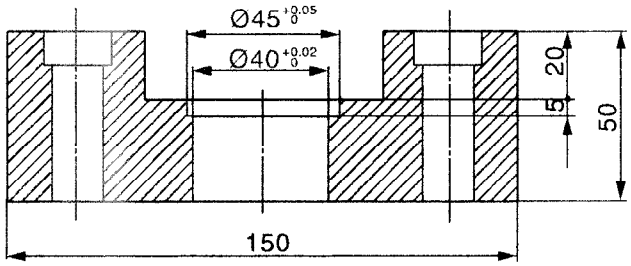


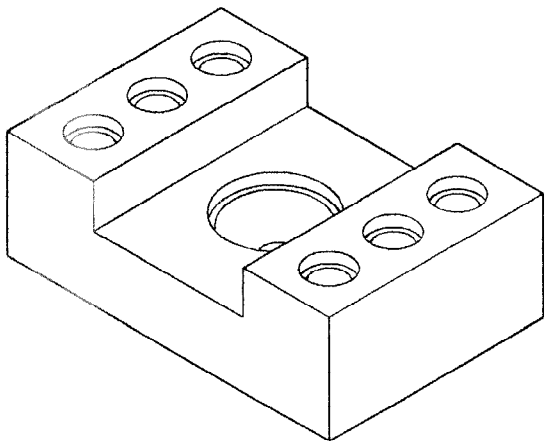
[Examples 2]



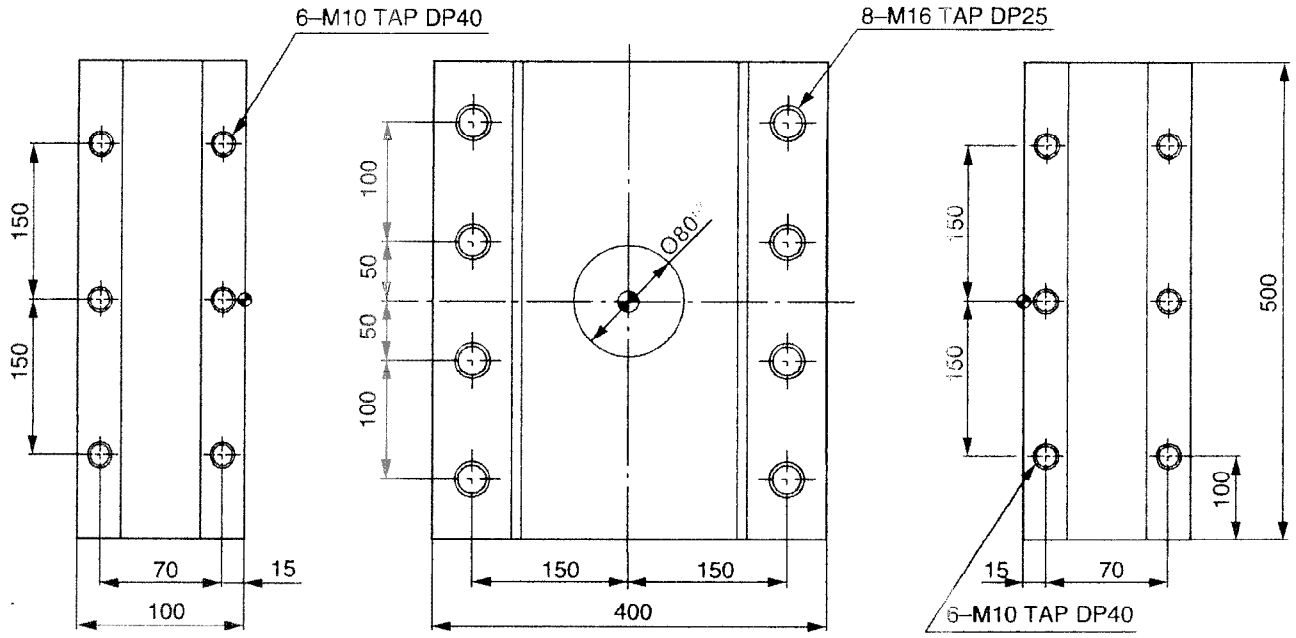
[Examples 3]

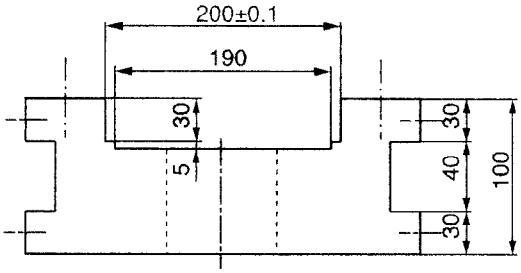

30
30
100
35
35
55
55
6-Ø15 DR
6-Ø20 C/BDP.10


Ø45 +0.05 0
Ø40 +0.02 0
20
5
50
150



[Examples 5]


6–M10 TAP DP40
8–M16 TAP DP25
150
150
70
15
100
100
50
50
100
Ø80
150
150
400
150
150
500
100
15
70
6–M10 TAP DP40


200±0.1
190
30
5
30
40
30
100