A-LNC Milling Machine Series

Programming manual

2016/05 Version:V01.02.000(4408230009)

Leading Numerical Controller

Advantech-LNC Technology Co., Ltd.

Table of Contents

1 G code function list

G code	Functional description	Group
G00	Rapid positioning	01
G01	Linear cutting	01
G02，G03	Clockwise, counterclockwise arc cutting	01
G04	Pause	00
G09	Correct stop	00
G10	Data input setting	00
G15	Cancel polar coordinate command	17
G16	Polar coordinate command	17
G17	XY plane selection	02
G18	ZX plane selection	02
G19	YZ plane selection	02
G21	Metric command	06
G27	Reference point reset inspection	00
G28	Reset the first reference point	00
G29	Reset from the first reference point	00
G30	Automatic reset of the second, third, and fourth reference points	00
G31	Single block terminated by the skip signal	00
G40	Cancel tool nose radius compensation	07
G41	Tool radius compensation to the left	07
G42	Tool radius compensation to the right	07
G43	Tool length compensation along the positive direction	08
G44	Tool length compensation along the negative direction	08
G49	Cancel tool length compensation	08
G50	Cancel zoom command	11
G50.1	Cancel mirror image command	22
G51	Zoom command	11
G51.1	Mirror image command	22
G52	Interval coordinate system settings	00

G code	Functional description	Group
G53	Rapid positioning of machine coordinate system	00
G54 ~ G59	Selection of machining coordinate systems	14
G61	Correct stop mode	15
G64	General cutting mode	15
G65	Macro single call	12
G66	Macro modal call	12
G67	Cancel macro modal call	12
G68	Coordinate rotation command	16
G69	Cancel coordinate rotation command	16
G73	High speed peck drilling cycle	09
G74	Left-hand thread tapping cycle	09
G76	Precision boring cycle	09
G80	Cancel fixed cycle cutting mode (canned cycle)	09
G81	Drilling cycle	09
G82	Drilling cycle	09
G83	Peck drilling cycle	09
G84	Right-hand thread tapping cycle	09
G85	Reaming cycle	09
G86	Boring cycle	09
G87	Back boring cutting	09
G88	Boring cycle	09
G89	Reaming cycle	09
G90	Absolute command	03
G91	Incremental command	03
G92	Coordinate value setting	00
G94	Setting the feed rate per minute	05
G95	Setting the feed rate per revolution	05
G98	Reset to reference point	10
G99	Reset to R point	10

2 General M code function table

M code	Function		Remark
M00	Program pause	Program stop	
M01	Optional program pause	Optional stop	
M02	Program termination	End of program	
M30	Program terminates and returns to the beginning	Program rewind	
M98	Subprogram call	Calling of subprogram	
M99	Returning to main program from a subprogram	End of subprogram	

3 Command syntax

G00 rapid positioning

Command format:

G00 <axis name><target position>;

Argument description:

Axis name : This is for assigning the name of axial direction to be moved, which can be any combination of X, Y, Z, A, B, C, or U, V, W. However, it must match the current axis name (axis name is set up by parameters 70464 to 70495).

Target position : Coordinate value of the target point should be determined by the status of G90 or G91 as either an absolute value or incremental value.

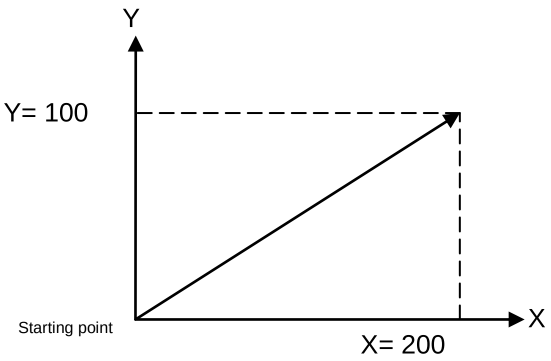
Operation description:

The function of the G00 command is to command the tool to be quickly positioned at the coordinates of the command point.

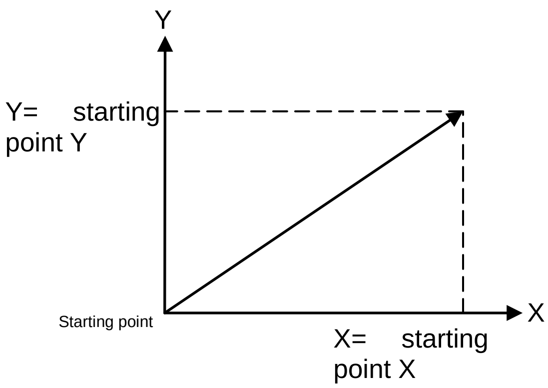
When G00 is being used, traversal speed cannot be determined by the F__ format. Instead, it is determined by the set values in parameters 60286 to 60317. The rapid feed rate adjustment knob can be used to adjust its percentage rate (F0, 25%, 50%, 100%).

Program example:

G90 G00 X200. Y100.;(Absolute)



G91 G00 X200. Y100.;(Incremental)



G01 linear cutting

Command format:

G01 <axis name><target position> F___;

Argument description:

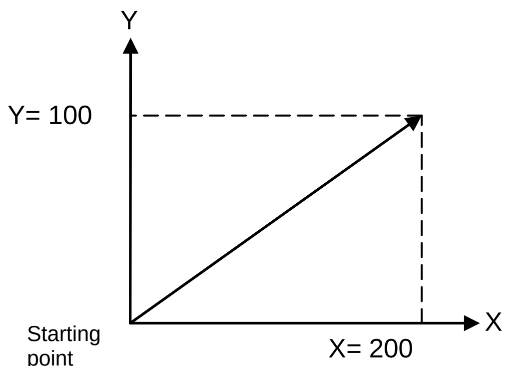
Axis name	:	This is for assigning the name of axial direction for the cutting to be performed, which can be any combination of X, Y, Z, A, B, C, or U, V, W. However, it must match the current axis name (axis name is set up by parameters 70464 to 70495).
Target position	:	Coordinate value of the target point should be determined by the status of G90 or G91 as either an absolute value or incremental value.
F__	:	If the feed rate (with unit mm/min or inch/min) is not assigned, the default value should be obtained from parameter 50066.

Operation description:

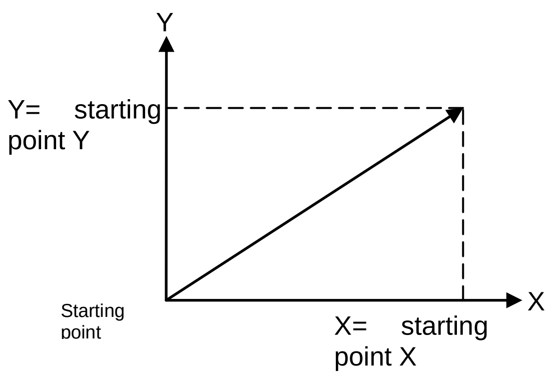
The function of G01 command is to command the tool to traverse from its current position to the next command position for linear cutting based on the preset feed rate F. During G01 cutting, the actual feed rate can be adjusted at any time via the continuous feed rate adjustment knob (0% to 150%). The maximum cutting feed rate can be set by parameter 60172. When the F value assigned by machining program surpasses the value set for this parameter, the actual cutting speed will be the value set for parameter 60172.

Plot legends:

G90 G01 X200. Y100. F200.;(Absolute)



G91 G01 X200. Y100. F200.;(Incremental)



G02, G03 clockwise and counterclockwise arc cutting

Command format:

$$G17\begin{bmatrix}G02\\G03\end{bmatrix}X\_\_Y\_\_\begin{bmatrix}R\_\_\\I\_\_J\_\_\end{bmatrix}F\_\_;$$

$$G18\begin{bmatrix}G02\\G03\end{bmatrix}X\_\_Z\_\_\begin{bmatrix}R\_\_\\I\_\_K\_\_\end{bmatrix}F\_\_;$$

$$G19\begin{bmatrix}G02\\G03\end{bmatrix}Y\_\_Z\_\_\begin{bmatrix}R\_\_\\J\_\_K\_\_\end{bmatrix}F\_\_;$$

Argument description:

X__、Y__、Z__	:	Coordinate value of the target point should be determined by the status of G90 or G91 as either an absolute value or incremental value.
I__	:	The starting point of the distance to the center of the circle along the X axis is determined by the starting point of the center's incremental value.
J__	:	The starting point of the distance to the center of the circle along the Y axis is determined by the starting point of the center's incremental value.
K__	:	The starting point of the distance to the center of the circle along the Z axis is determined by the starting point of the center's incremental value.
F__	:	Feed rate (mm/min or inch/min).
R__	:	Arc radius.

Operation description:

G02: Clockwise (CW) arc cutting.

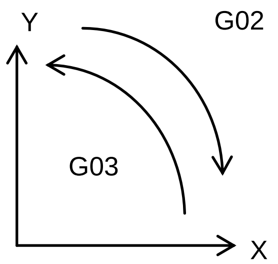
G03: Counterclockwise (CCW) arc cutting.

G02 and G03 are arc cutting command. Because workpieces are three dimensional, arc cutting directions on different planes are shown in the figure below. The default plane at machine startup can be set by parameter 50042.

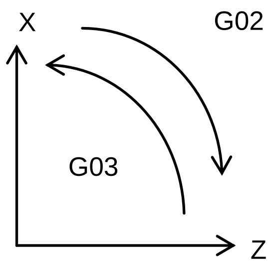
In machining commands, I, J, and K can be directly replaced by R, which is the arc radius. When R, I, and J have all been issued a value, the system will use R.

As for the G02 and G03 commands, the system will examine whether the distance from the arc's starting point to the center of the circle is identical to the distance from the arc's end point to the center of the circle (it must be equal to the arc radius). When the error between these two values is greater than the set value of parameter 50048, it will trigger system alert 【510204 - coordinates of the end point are not on the arc when using G02/G03】.

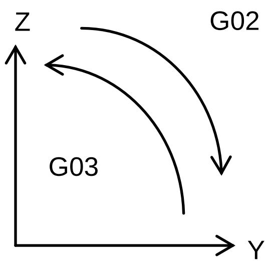
Plot legends:



G17 X－Y plane arc



G18 Z－X plane arc



G19 Y－Z plane arc

G04 pause

Command format:

G04 X___;

G04 P___;

Argument description:

X__ : Set pause time in terms of seconds within the range of 0.001 to 99999.999.

P__ : Set pause time in terms of milliseconds without using a decimal point, setting range is from 1 to 99999999.

Operation description:

The pause function, the pause period should be set after G04. When the time is up, execution will automatically resume with the next single block.

Program example:

G04 X100.;-- Pause time is 100 seconds

G04 P100; -- Pause time is 0.1 second

G04; --Equivalent to actual stop (G09)

G09 correct positioning

Command format:

$$\mathrm{G09}\begin{bmatrix} \mathrm{G01}\_\_ \\ \mathrm{G02}\_\_ \\ \mathrm{G03}\_\_ \end{bmatrix};$$

Argument description:

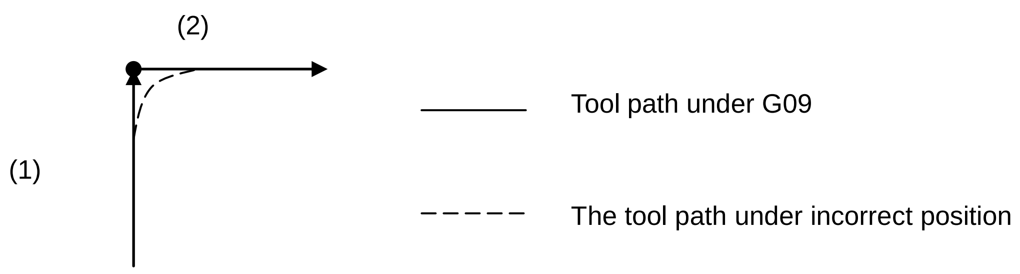
G09 is a command used in coordination with the actual stop of cutting. For every positioning command executed by the system under G09 status, it must be confirmed that the positioning condition is in compliance with the setting before the next single block can be executed. Therefore, if cutting positioning is taking place between single blocks during operation, there will be discontinuity because speed is sacrificed for precision requirements of the positioning point. This method can be used to obtain better shape precision, and the degree of positioning precision can be set by parameters 56064 to 56095. The G09 function can only be applied within the single block which G09 belongs to, it will then return to its previous status.

Program example:

G91 G09 G01 Y100. F200.; --(1)

G01 X100.;--(2)

Plot legends:



G10 coordinate system data input settings

Command format 1: Set the reference point's machine coordinates of coordinate systems 00 and G54 to G59

$\begin{bmatrix} \mathrm{G90} \\ \mathrm{G91} \end{bmatrix}$ G10 P 153~159 <axis name><target position>;

Command format 2: Set the reference point's machine coordinates of coordinate systems 00 and G54 to G59

$\begin{bmatrix} \mathrm{G90} \\ \mathrm{G91} \end{bmatrix}$ G10 L2 P 0~6 <axis name><target position>;

Command format 3: Set the reference point's machine coordinate of an extended coordinate system

$\begin{bmatrix} \mathrm{G90} \\ \mathrm{G91} \end{bmatrix}$ G10 L20 P 1~100 <axis name><target position>;

Argument description:

Function 1: Set the reference point's machine coordinates of coordinate systems 00 and G54 to G59.

P__	:	Coordinate system and setting range within 153 to 159 corresponds to coordinate system 00 and G54 to G59.
Axis name	:	This is for assigning the name of axial direction to be set up, which can be any combination of X, Y, Z, A, B, C, or U, V, W. However, it must match the current axis name (axis name is set up by parameters 70464 to 70495).
Target position	:	Machine coordinate values of a target point. The status of G90 or G91 should be used to determine whether it should be an absolute or incremental value.

Function 2: Set the reference point's machine coordinates of coordinate systems 00 and G54 to G59.

L2	:	Notify the system to set coordinate systems 00 and G54 to G59.
P__	:	Coordinate system, setting range within 0 to 6 corresponds to coordinate system 00 and G54 to G59, respectively.
Axis name	:	This is for assigning the name of axial direction to be set up, which can be any combination of X, Y, Z, A, B, C, or U, V, W. However, it must match the current axis name (axis name is set up by parameters 70464 to 70495).
Target position	:	Machine coordinate values of a target point. The status of G90 or G91 should be used to determine whether it should be an absolute or incremental value.

Function 3: Set the reference point's machine coordinate of an extended coordinate system.

L20	:	Notify the system to set extended coordinate system mode.
P__	:	The extended coordinate system and setting range within 1 to 100 corresponds to G54 and P1 to P100, respectively.
Axis name	:	This is for assigning the name of axial direction to be set up, which can be any combination of X, Y, Z, A, B, C, or U, V, W. However, it must match the current axis name (axis name is set up by parameters 70464 to 70495).
Target position	:	Machine coordinate values of a target point. The status of G90 or G91 should be used to determine whether it should be an absolute or incremental value.

Operation description:

The offset of coordinate systems 00 and G54 to G59, as well as the extended coordinate system are generally entered manually through the human machine interface; it can also be set by using the G10 command from the machining program. However, the setting must be completed before accessing coordinate system 00/G54 to G59 or the extended coordinate system, such that the set value can be valid in the machining program afterward. Under G90 absolute mode, the offset amount being set will become the new offset amount of the coordinate system. In G91 incremental mode, the new offset amount of a coordinate system will be the current offset amount plus the offset amount being set.

Program example:

G10P154X50.; ------------Set the X axis reference point of coordinate system G54 at machine coordinate 50.

G10L2P2X30.; ------------Set the X axis reference point of coordinate system G55 at machine coordinate 30.

G10L20P1Y40.;Set the X axis reference point of the first set of extended coordinate system at machine coordinate 40.

Setting G10 tool compensation input data

Command format 1: Set the compensation value for tool shape

$\begin{bmatrix} \text{G90} \\ \text{G91} \end{bmatrix}$ G10 P 1~99 <Axis name><Compensation value> R__;

Command format 2: Set the compensation value for Z axis' tool shape

$\begin{bmatrix} \text{G90} \\ \text{G91} \end{bmatrix}$ G10 L10 P 1~99 R__;

Command format 3: Set the tool wear compensation value for Z axis

$\begin{bmatrix} \text{G90} \\ \text{G91} \end{bmatrix}$ G10 L11 P 1~99 R__;

Command format 4: Set the compensation value for tool radius shape

$\begin{bmatrix} \text{G90} \\ \text{G91} \end{bmatrix}$ G10 L12 P 1~100 R__;

Command format 5: Set the tool wear compensation value for tool radius

$\begin{bmatrix} \text{G90} \\ \text{G91} \end{bmatrix}$ G10 L13 P 1~100 R__;

Argument description:

Function 1: Set the compensation value for tool shape.

P__	:	Tool compensation number, setting range is within 1 to 99.
R__	:	Tool radius compensation value. The status of G90 or G91 should be used to determine whether it should be an absolute or incremental value.
Axis name	:	This is for assigning the name of axial direction to be set up, which can be any combination of X, Y, Z, A, B, C, or U, V, W. However, it must match the current axis name (axis name is set up by parameters 70464 to 70495).
Compensation value	:	Tool length compensation value. The status of G90 or G91 should be used to determine whether it should be an absolute or incremental value.

Function 2: Set the compensation value for Z axis' tool shape.

L10		Notify the system to set Z axis' tool length and shape compensation values.
P__	:	Tool compensation number, setting range is within 1 to 99.
R__	:	Z axis tool length compensation value. The status of G90 or G91 should be used to determine whether it should be an absolute or incremental value.

Function 3: Set Z axis' tool wear compensation value.

L11		Notify the system to set tool wear compensation for the Z axis.
P__	:	Tool compensation number, setting range is within 1 to 99.
R__	:	Z axis tool length compensation value. The status of G90 or G91 should be used to determine whether it should be an absolute or incremental value.

Function 4: Set the compensation value for the exterior shape of tool nose radius.

L12	:	Notify the system to set the compensation value for the tool nose radius shape.
P__	:	Tool compensation number, setting range is within 1 to 99.
R__	:	Tool radius compensation value. The status of G90 or G91 should be used to determine whether it should be an absolute or incremental value.

Function 5: Set the tool wear compensation value for tool radius.

L13	:	Notify the system to set tool wear compensation value for the tool radius.
P__	:	Tool compensation number, setting range is within 1 to 99.
R__	:	Tool radius compensation value. The status of G90 or G91 should be used to determine whether it should be an absolute or incremental value.

Operation description:

The tool compensation value is generally entered manually through the human machine interface; it can also be set by using the G10 command from the machining program. However, the setting must be completed before accessing these tool compensation values such that the set value can be valid in the machining program afterward. Under G90 absolute mode, the compensation amount being set will become the new compensation amount of the tool table. In G91 incremental mode, the new compensation amount of the tool table will be the current compensation value plus the compensation value being set.

Program example:

G10P1R6.Z10.; For tool compensation number 1, the radius compensation value should be set as 6, and the length compensation value should be set as 10.

G10L10P2R20.; -----For tool compensation number 2, the shape compensation value should be set as 20.

G10L11P3R0.01; For tool compensation number 3, the tool wear compensation value should be set as 0.01.

G10L12P4R30.; For tool compensation number 4, the radius shape compensation value should be set as 30.

G10L12P5R0.5; For tool compensation number 5, the radius tool wear compensation value should be set as 0.5.

Cancel G15 polar coordinate command

G16 Polar coordinate command

Command format:

G17 G16 X___ Y___;

G18 G16 Z___ X___;

G19 G16 Y___ Z___;

Argument description:

X__ Y__: On the G17 plane, X__ is for assigning the radius of polar coordinates, and Y__ is for assigning the angle of polar coordinates.

Z__ X__: On the G18 plane, Z__ is for assigning the radius of polar coordinates, and X__ is for assigning the angle of polar coordinates.

Y__ Z__: On the G19 plane, Y__ is for assigning the radius of polar coordinates, and Z__ is for assigning the angle of polar coordinates.

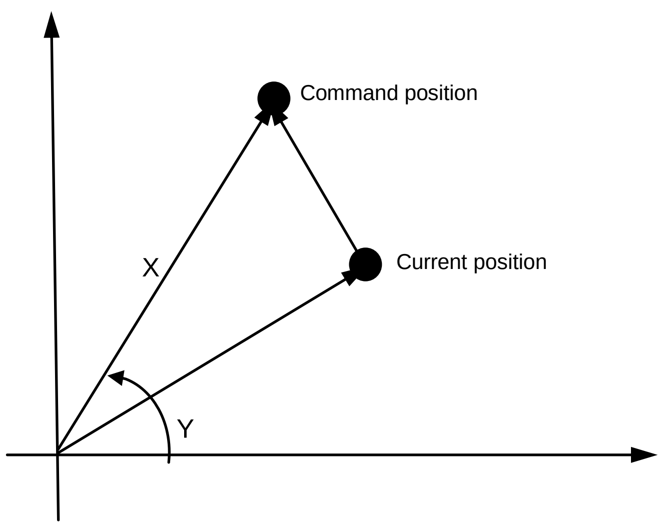
Angles can be executed via incremental or absolute commands.

Operation description:

Via G16, cutting path's target point can be assigned by using polar coordinates as shown in the figure below.

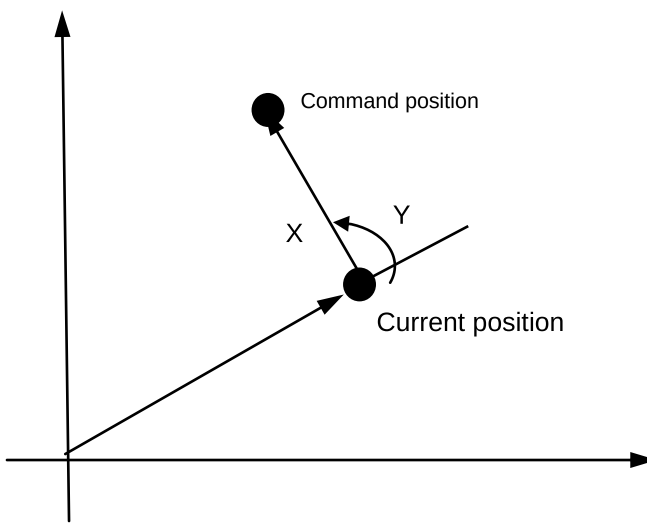
Plot legends:

G17 G90 (absolute)



X: Radius Y: Angle

G17 G91 (incremental)



X: Radius Y: Angle

G17, G18, G19 cutting plane settings

Command format:

```
G17;  (XY plane)
G18;  (ZX plane)
G19;  (YZ plane)
```

Argument description:

When the arc command or tool nose radius compensation command are used, the cutting plane must be set first to ensure there is no confusion during system calculation.

Parameter 50042 can be set to restore the default machining plane after RESET.

G21 Metric system unit conversion

Command format:

G21;

Argument description:

G21 : The metric unit is set based on the unit of "mm" with a minimum value of 0.001 mm.

The use of this command will trigger system alert 【510211 - Metric system/inch system conversion has been reset】.

Following items must be noted during unit conversion:

(1) All coordinate systems must be reset.

(2) Tool compensation value must be reset.

(3) System related parameters must be corrected according to the set unit system.

G27 Reference point reset inspection

Command format:

G27 <axis name><target position>;

Argument description:

Axis name : This is for assigning the name of axial direction for resetting the reference point, which can be any combination of X, Y, Z, A, B, C or U, V, W. However, it must match the current axis name (axis name is set up by parameters 70464 to 70495).

Target position : Coordinate value of the target point should be determined by the status of G90 or G91 as either an absolute value or incremental value.

Operation description:

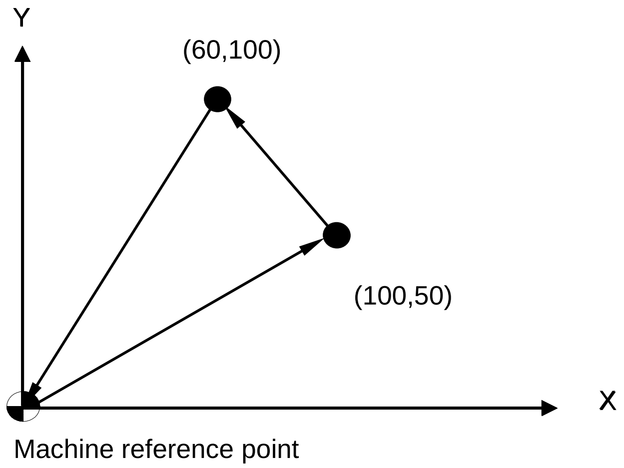
When an operation cycle is completed by the program before reaching the end point or returning to the starting point, G27 can be used to execute reference point reset inspection to ensure the accuracy of the current position. The G27 command will examine whether the current position has reached the machine reference point (the first reference point) after the assigned movement stroke has been executed; if it stays at the reference point after execution, the reference point indicator light will be turned on and the next single block will be executed; if it is not at the reference point, it will trigger system alert 【610009 - the G27 command has failed to return to macro reference point】.

When argument X___ is assigned, reset and inspection will be executed for the X axis. If it is not assigned, the X axis will not be moved and examined; the same shall be applied to other axes.

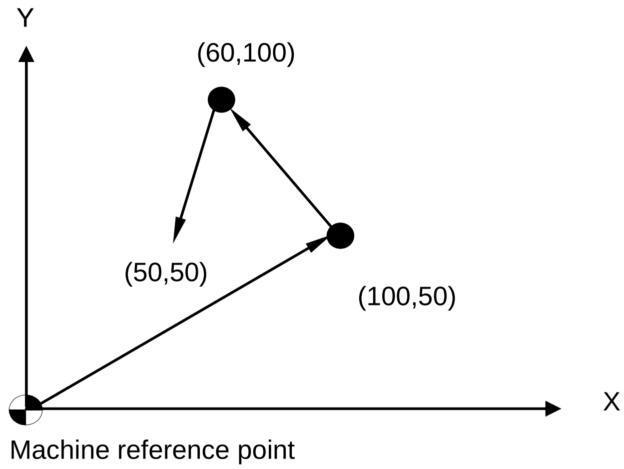
Before executing G27, it is suggested to cancel all tool compensation to avoid misjudgment.

Plot legends:

G90 G00 X100. Y50.;
G00 X60. Y100.;
G91 G27 X-60. Y-100.;(Correct)



G90 G00 X100. Y50.;
G00 X60. Y100.;
G91 G27 X-10. Y-50.;(Alert)



G28 reset the first reference point

Command format:

```
G28 <Axis name><Position of the mid point>;
```

Argument description:

Axis name	:	This is for assigning the name of axial direction for resetting the first reference point, which can be any combination of X, Y, Z, A, B, C, or U, V, W. However, it must match the current axis name (axis name is set up by parameters 70464 to 70495).
Position of the mid point	:	Coordinate value of the mid point should be determined by the status of G90 or G91 as either an absolute value or incremental value.

Operation description:

The system will retain the mid point coordinate assigned by G28 to be used by G29 later.

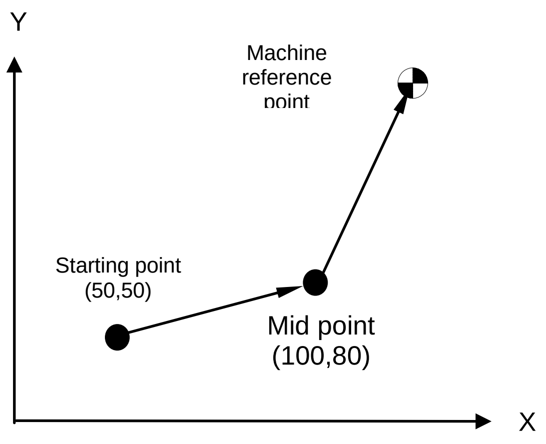
In the machining program, the G28 command can be used to control the tool to travel past the specified mid point before automatically returning to the first reference point (machine reference point). Manual reference point return procedure must be executed before executing G28, or it will trigger system alert【610007 - G28, G30 macro commands did not execute reference point return after machine startup】.

When argument X___ is not assigned, the X axis will not execute the return of the first reference point; the same shall be applied to other axes. However, if there are no axial direction arguments assigned, all axial directions will execute the return to first reference point procedure.

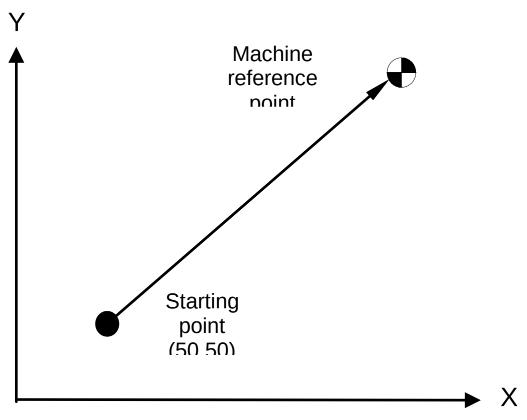
It must be noted that the previously assigned tool length compensation value will automatically be canceled after executing G28.

Plot legends:

G90 G28 X100. Y80.;



G91 G28 X0. Y0.;(No mid point)



G29 return from the first reference point

Command format:

G29 <axis name><target position>;

Argument description:

Axis name : This is for assigning the name of axial direction for resetting the first reference point, which can be any combination of X, Y, Z, A, B, C, or U, V, W. However, it must match the current axis name (axis name is set up by parameters 70464 to 70495).

Target position : Coordinate value of the target point should be determined by the status of G90 or G91 as either an absolute value or incremental value.

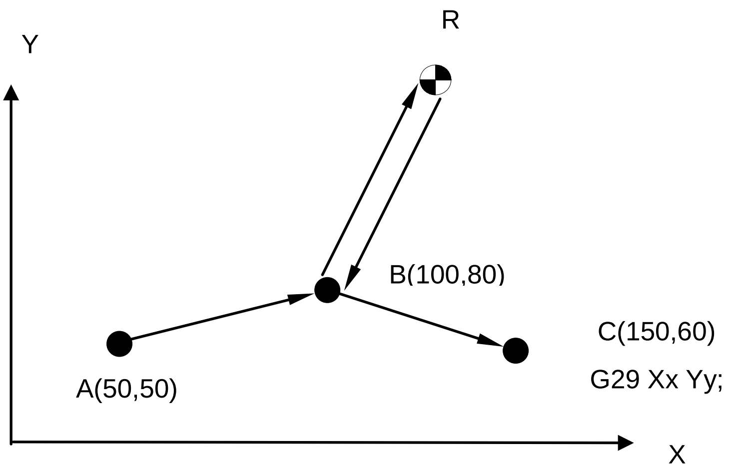
Operation description:

The G29 command is only used after G28. After executing G28, the tool will stop at the position of the first reference point; at this moment, G29 can be used to control the tool traversal to the target position from the first reference point after passing through the mid point assigned by G28.

Plot legends:

G90 G28 X100. Y80.; (A→B→R)

G29 X150. Y60.; (R→B→C)



G30 Return of the second, third, and fourth reference points

Command format:

$$\text{G30 P}\begin{bmatrix}2\\3\\4\end{bmatrix} < \text{Axis name} >< \text{Position of the mid point} >$$

Argument description:

P__	:	It is for assigning a reference point with a setting range between 2 to 4, which corresponds to the second to the fourth reference points, respectively.
Axis name	:	This is for assigning the name of axial direction for resetting the reference point, which can be any combination of X, Y, Z, A, B, C or U, V, W. However, it must match the current axis name (axis name is set up by parameters 70464 to 70495).
Position of the mid point		Coordinate value of the mid point should be determined by the status of G90 or G91 as either an absolute value or incremental value.

Operation description:

This command can be used for the return of the second, third, and fourth reference point. The tool will return to the second, third, and fourth reference point from its current position while passing through the assigned mid point.

The offset of the second reference point and the machine reference point can be set by parameters 166032 to 166063; the offset of the third reference point and the machine reference point can be set by parameters 166064 to 166095; and the offset of the fourth reference point and the machine reference point can be set by parameters 166096 to 166127.

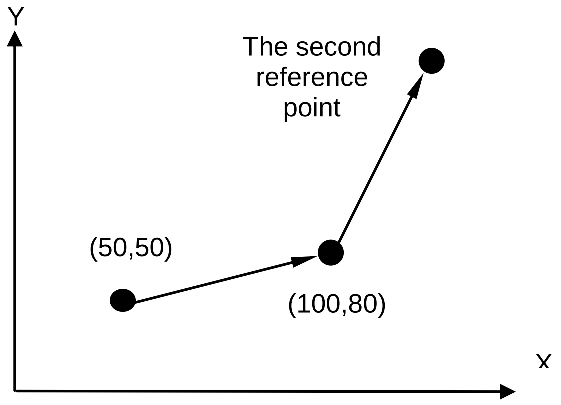
The manual reference point return procedure must be executed before executing G30, or it will trigger system alert 【610007 - G28, G30 macro commands did not execute reference point return after machine startup】.

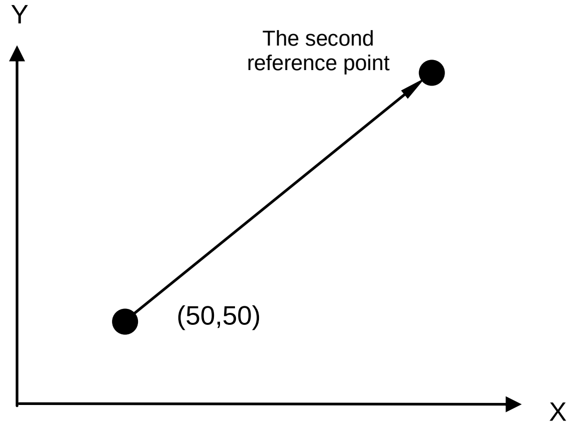
When argument X___ is not assigned, the X axis will not execute the return to the reference point; the same shall be applied to other axes. However, if there are no axial direction arguments assigned, all axial directions will execute the return to reference point procedure.

It must be noted that the previously assigned tool length compensation value will automatically be canceled after executing G30.

Plot legends:

G90 G30 P2 X100. Y80.;





G31 Single block execution stopped by the Skip signal

Command format:

G31 <Axis name><Target position> F___;

Argument description:

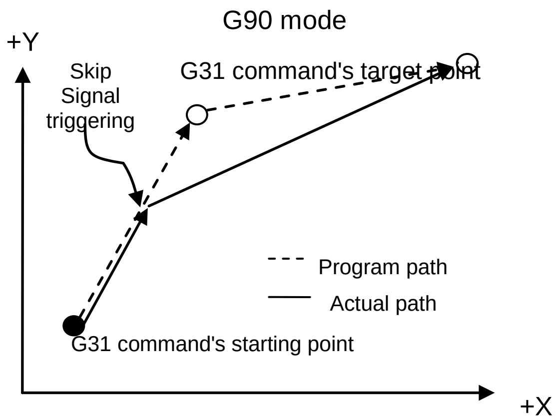
Axis name	:	This is for assigning the name of axial direction to be set up, which can be any combination of X, Y, Z, A, B, C, or U, V, W. However, it must match the current axis name (axis name is set up by parameters 70464 to 70495).
Target position	:	Coordinate value of the target point should be determined by the status of G90 or G91 as either an absolute value or incremental value.
F__	:	Feed rate, the feed rate setting is only valid within this single block. If it is not set, then the set value of parameter 170000 will be used as the feed rate for this single block.

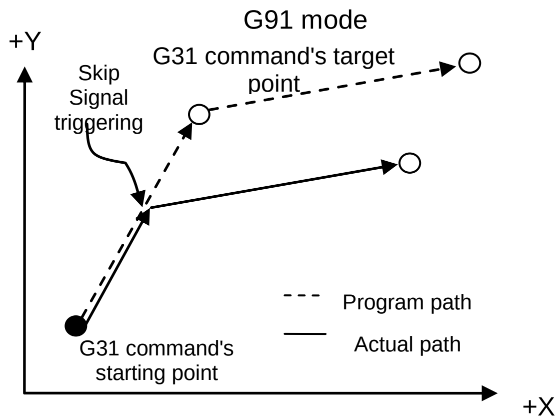
Operation description:

The function of this command is the same as G01; however, if Skip signal is triggered during execution, this single block will end immediately and the next single block will be executed.

The triggering of the G31 Skip signal can be obtained by macro function R_SKIP(0, 1). When G31 Skip signal is triggered, the absolute coordinate can obtain path axis sequence for axis 1 to 32 via the macro function R_SKIP(0, 101~132), and the machine coordinate can obtain path axis sequence for axis 1 to 32 via the macro function R_SKIP(0, 201~232). However, before the G31 Skip signal is triggered, the values obtained by macro function R_SKIP(…) are the target point coordinates of the G31 command. For more information on the macro function R_SKIP(…), please refer to the next page.

Plot legends:





Relevant parameters:

1. Parameter 160000: Macro parameter, G31 default feed rate (1 to 2100000000KLU/MIN).
2. Parameter 160001: Macro parameter, G31 signal triggering method (0: rising edge, 1: lower edge).
3. Parameter 160002: Macro parameter, G31 type of signal source (0: Local input, 1: path PLC input, 2: PLC input of each axis, 3: servo axis).
4. Parameter 160003: Macro parameter, G31 signal source Local input (1 to 2).
5. Parameter 160004: Macro parameter, G31 signal source PLCl input (0 to 4095).
6. Parameters 166159.0 to 31: Macro parameter, when the G31 signal source type for axis 1 to 32 is (PLC input of each axis, servo axis), triggering is activated (0: No, 1: Activated).
7. Parameters 166160 to 166191: Macro parameter, G31 signal source PLC I point for axis 1 to 32 (0 to 4095).

Detailed description of macro function R_SKIP(…):

	R_SKIP(PATH,TYPE) Read G31 Skip coordinates R
Description	● PATH =>Path number. Value range 0 to 6. Unit not available. 0: Current path, 1 to 6: Path 1 to Path 6. ● TYPE =>Assigned type. Value range ****. Unit not available. 1: Is the SKIP signal triggered (0: No, 1: Triggered). Unit: not available 101 ~ 132: SKIP absolute coordinates of axis 1 to 32. Unit: mm 201 ~ 232: SKIP machine coordinates of axis 1 to 32. Unit: mm 301 ~ 332: Is the SKIP signal of each axis triggered (0: No, 1: Triggered). Unit: not available
Return	Machining program reads G31 Skip coordinate information. Unit is the same as the previous cell. Error code: -1 => The assigned path number is beyond valid range -2 => The assigned type is beyond valid range -3 => The assigned axis number is beyond valid range
Example	G91 G31 Z-100. F100; #1= R_SKIP(0,103); /* Obtain the absolute coordinates of the third axis during SKIP */ G4 X20.; /* Program is paused for observing whether the content saved in #1 is the absolute coordinate of Z-axis */ M30;

G40, G41, G42 Tool nose radius compensation

Command format:

$$\begin{bmatrix} G17 \\ G18 \\ G19 \end{bmatrix} \begin{bmatrix} G41 \\ G42 \end{bmatrix} \begin{bmatrix} G00 \\ G01 \end{bmatrix} < \text{Axis name} >< \text{Target position} > D\_\_;$$

$$G40 < \text{Axis name} >< \text{Target position} >;$$

Argument description:

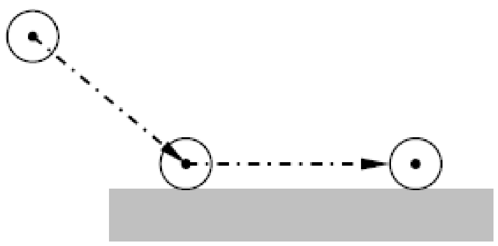
G40	:	Cancel tool nose radius compensation.
G41	:	Compensation for left tool nose radius.
G42	:	Compensation for right tool nose radius.
Axis name	:	This is for assigning the name of axial direction to be set up, which can be any combination of X, Y, Z, A, B, C, or U, V, W. However, it must match the current axis name (axis name is set up by parameters 70464 to 70495).
Target position	:	Coordinate value of the target point should be determined by the status of G90 or G91 as either an absolute value or incremental value.
D__	:	Tool path compensation number, setting range is within 1 to 99.

Selection of offset plane:

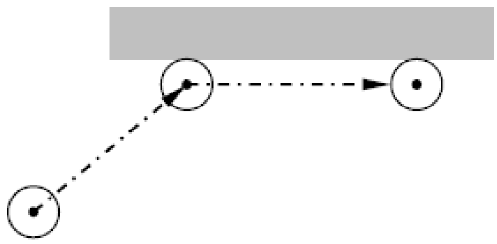
Offset plane	Plane selection command	Axis movement command
X-Y	G17;	X__Y__
Z-X	G18;	Z__X__
Y-Z	G19;	Y__Z__

Plot legends:

G41 : The tool has an offset of an amount of the radius to the left when facing to the direction of tool movement.



G42 : The tool has an offset of an amount of the radius to the right when facing to the direction of the tool movement.



Operation description:

The tool path compensation function can be divided by the sequence process flow into three behavior modes: starting tool compensation, during tool path compensation offset, and canceling tool path compensation.

The starting status of tool path compensation

When tool nose radius compensation commands G41/G42 are assigned and the compensation number's D code of the tool path must be in compliance with the condition of 0 < D ≤ maximum compensation number, the CNC will enter into the starting tool path compensation status. The single block for starting tool path compensation must be a linear command (G00 or G01); an arc command (G02 or G03) must not be used. For the starting tool path compensation mode, the tool path can be divided into Type A and Type B, which can be set by parameter 50060.

The offset status during tool path compensation

During tool path offset compensation, the compensation behavior is completed by rapid positioning (G00), linear cutting (G01), and arc cutting (G02, G03). If there are more than 8 continuous single blocks without any traversal command during the tool path offset compensation process, there will be excessive or insufficient cutting. If the offset plane is replaced during offset compensation, system alert 【510393 - Plane switching is not allowed during compensation calculation】 will be issued and tool movement will stop.

The cancel status of tool path compensation

When tool path compensation is canceled, the compensation vector is 0, and the tool's center path and program path are overlapped. When one of the following conditions is met, the CNC's tool path compensation function will enter the canceling tool path compensation mode.

(1) After the CNC startup power is connected.

(2) Compensation cancel command G40 is assigned.

(3) The value of tool path compensation number D code is assigned as 0.

(4) After program termination, and command M02, M30 are executed.

(5) After pressing the Reset key on the panel.

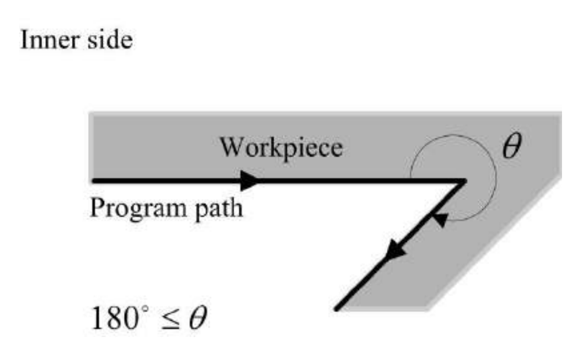
The single block for canceling tool path compensation must be a linear command (G00 or G01), an arc command (G02 or G03) must not be used. For the canceling tool path compensation mode, the tool path can be divided into Type A and Type B, which can be set by parameter 50060.

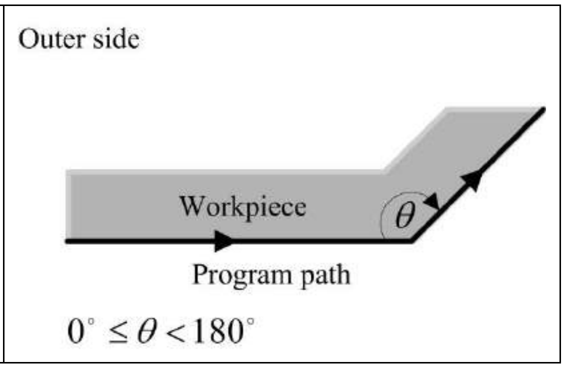
Detailed description of tool path:

The tool path compensation function can be divided into three situations by the tool's travel path, which are tool traversal at the start of compensation, tool traversal under the offset method, and tool traversal when compensation is canceled. These three situations can be further divided into tool traversal along an inner edge, tool traversal along the outer edge of an obtuse angle, and tool traversal along the outer edge of an acute angle. The following section provides detailed information on the tool's travel path at the start of compensation, under the offset method, and while canceling compensation.

Inner edge and outer edge

With the side of the workpiece as a basis, when the angle between the traversal commands of two program blocks is greater than 180 degrees, it is referred to as the "inner edge". When this angle is between 0 to 180 degrees, it is referred to as the "outer edge". The left tool nose radius compensation (G41) angle is defined as the angle generated from the first procedure section to the second procedure section in the counterclockwise direction. The right tool nose radius compensation (G42) angle is designed as the angle generated from the first procedure section to the second procedure section in the clockwise direction. In this manual, the right tool radius compensation (G42) is used as an example.

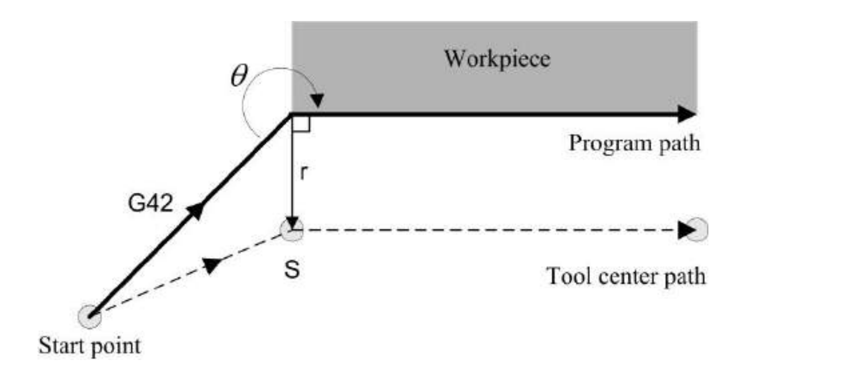


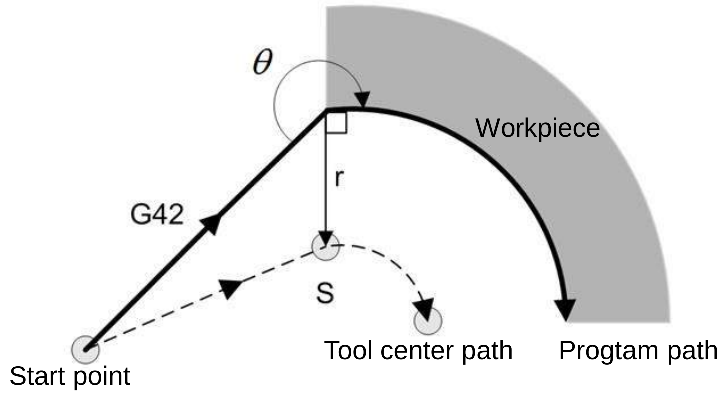


Tool traversal at the start of compensation

The tool traversal path at the start of compensation is as described below

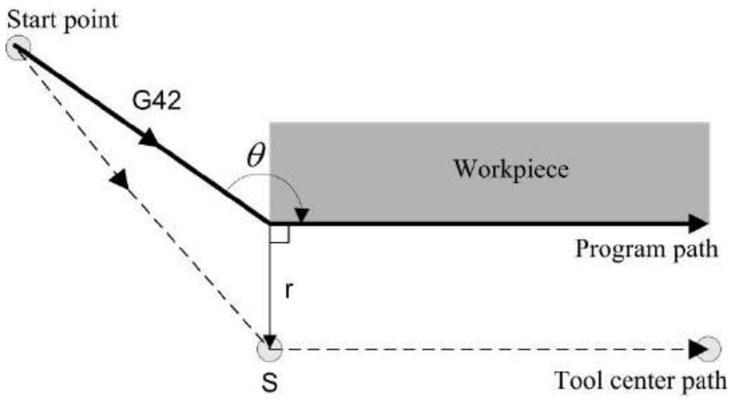
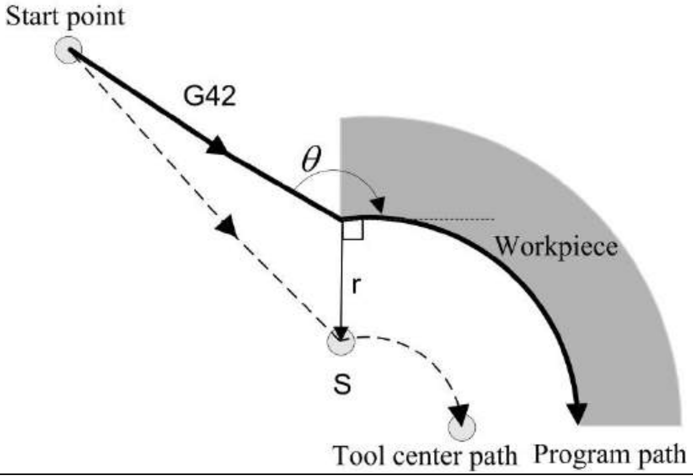
Tool will traverse along the inner edge ($180^\circ \leq \theta$)

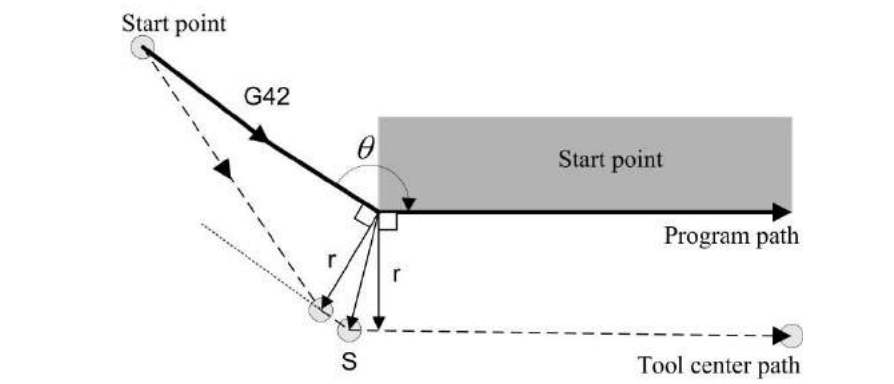
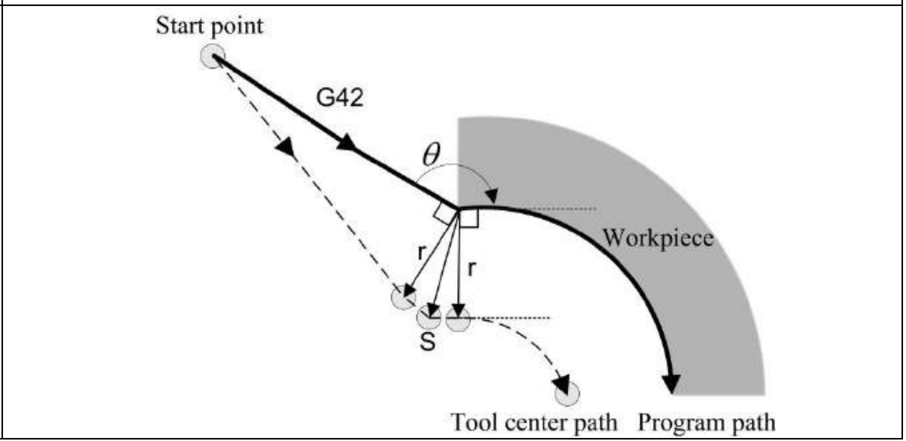
Linear ↓ Linear	

<table>
<tr><td>Linear
↓
Arc</td><td>
</td></tr>
</table>

Tool moving along the outer edge of an obtuse angle ($90^\circ \le \theta < 180^\circ$)

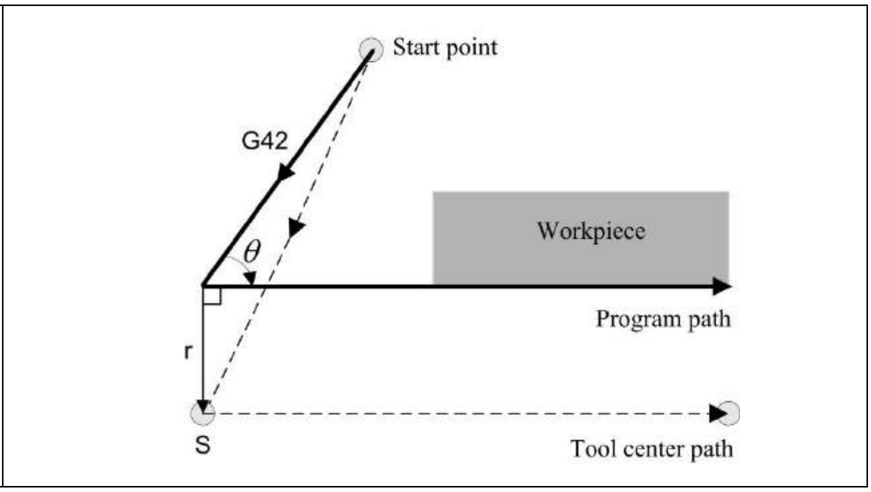
There are two types of tool paths at the start, which are Type A and Type B, and they can be set by parameter 50060.

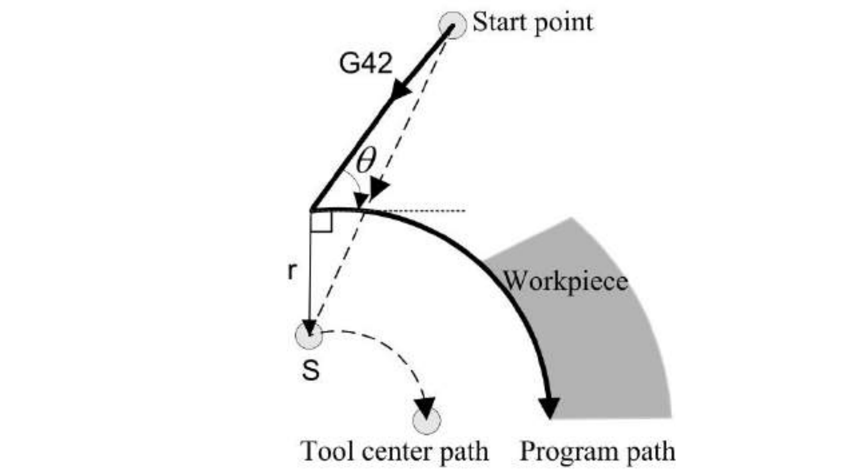
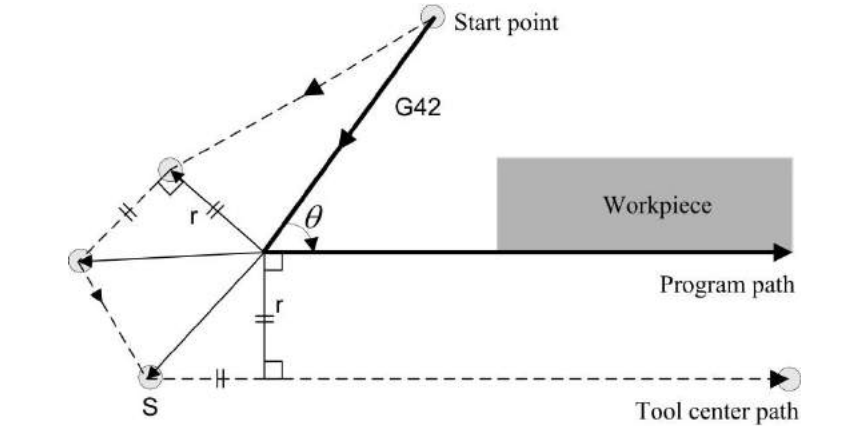
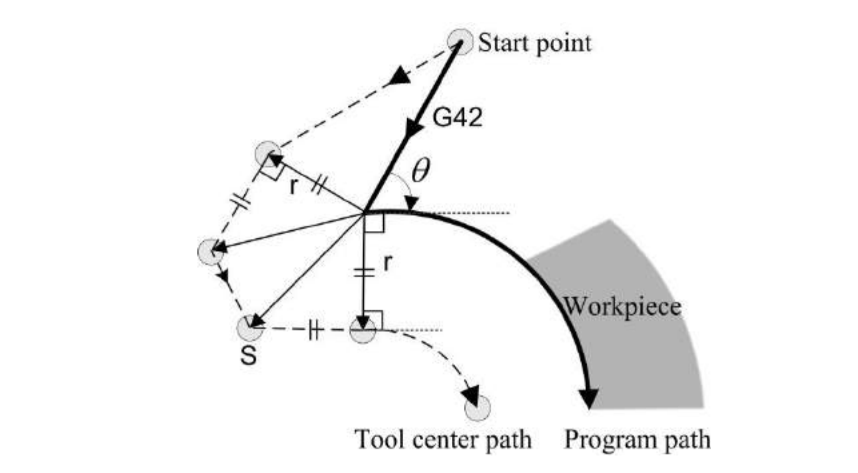
<table>
<tr><td rowspan="2">Type A</td><td>Linear
↓
Linear</td><td>
</td></tr>
<tr><td>Linear
↓
Arc</td><td>
</td></tr>
</table>

<table>
<tr><td rowspan="2">Type B</td><td>Linear
↓
Linear</td><td>
</td></tr>
<tr><td>Linear
↓
Arc</td><td>
</td></tr>
</table>

Tool will traverse around the outside of an acute angle ($\theta < 90^{\circ}$)

There are two types of tool paths at the start, which are Type A and Type B, and they can be set by parameter 50060.

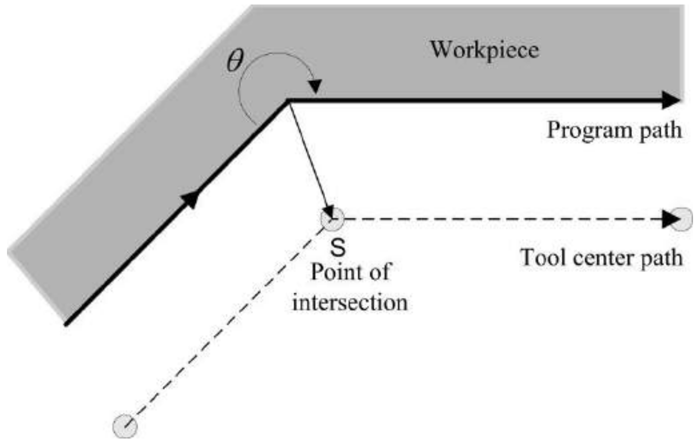
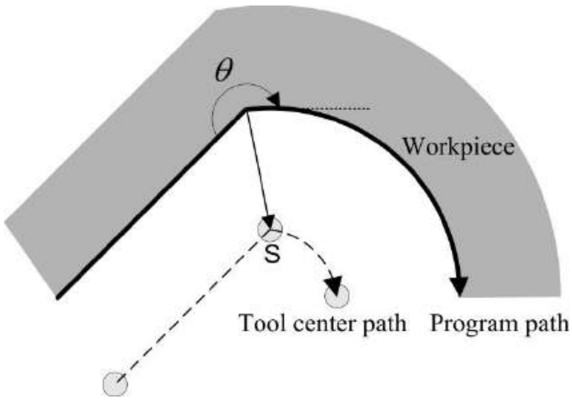
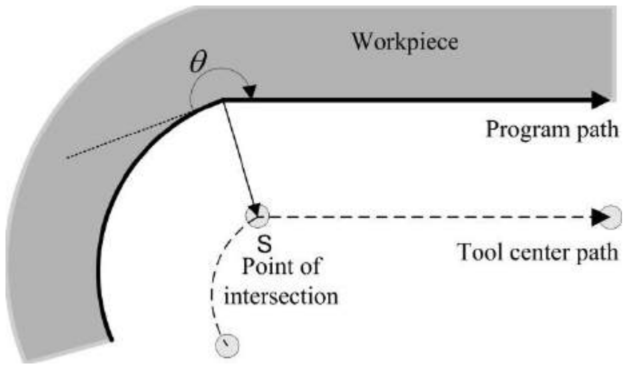
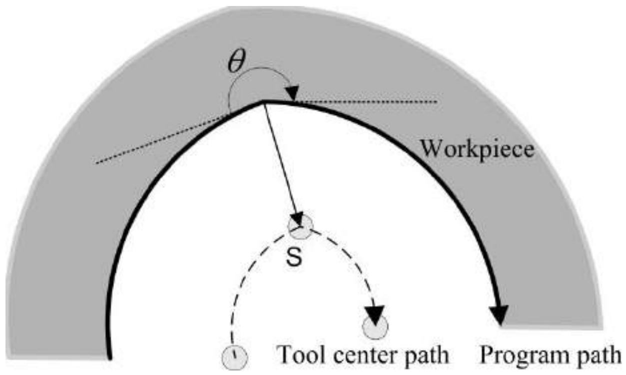
<table>
<tr><td>Type A</td><td>Linear
↓
Linear</td><td>
</td></tr>
</table>

	Linear ↓ Arc	
Type B	Linear ↓ Linear	
	Linear ↓ Arc	

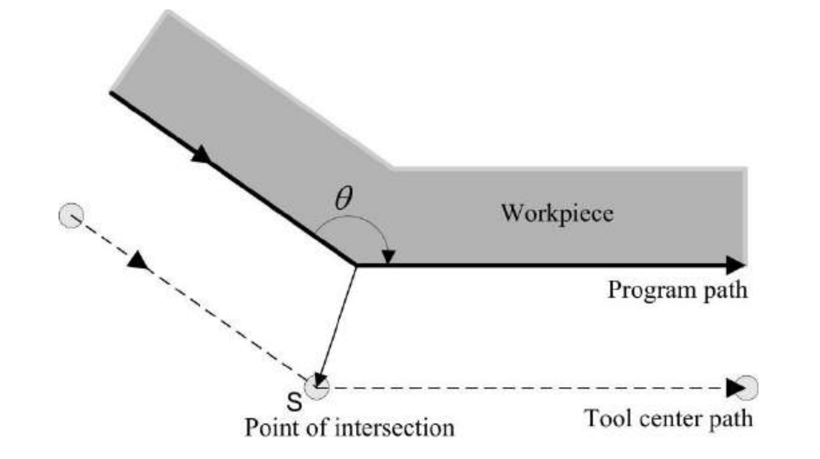
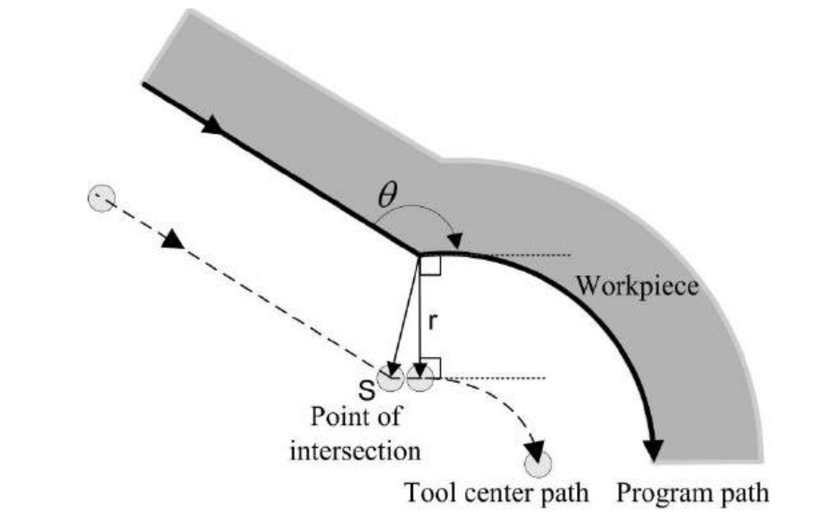
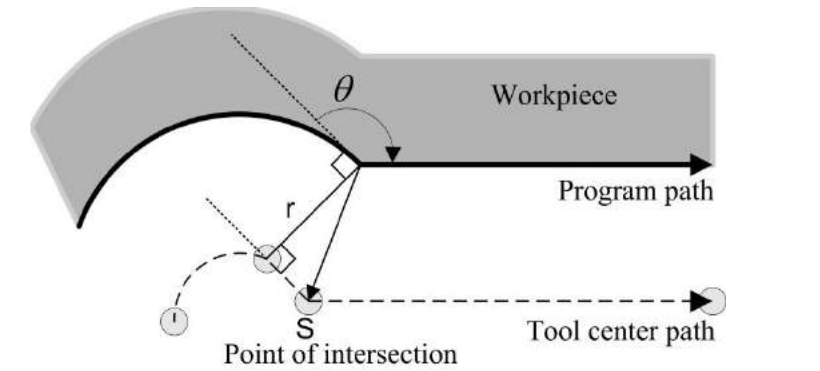
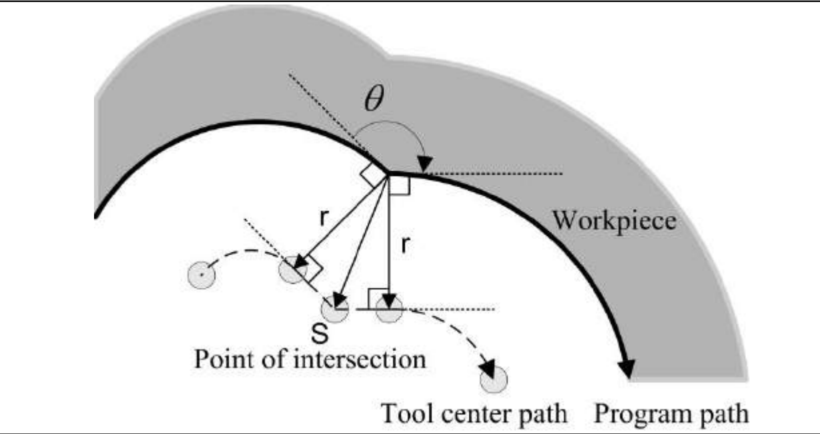
Tool traversal under the offset method

The tool traversal path under offset compensation is as described below

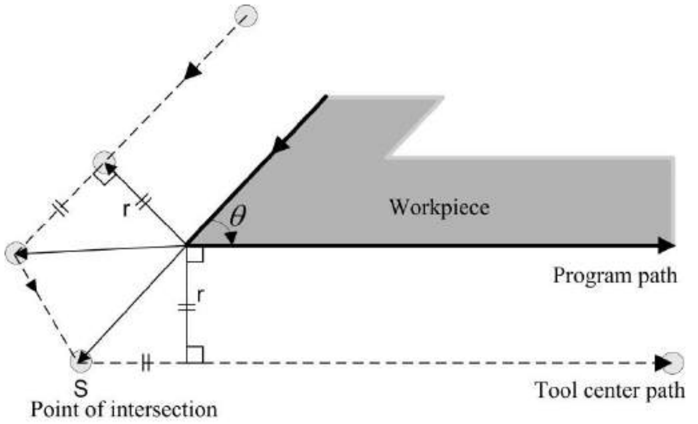
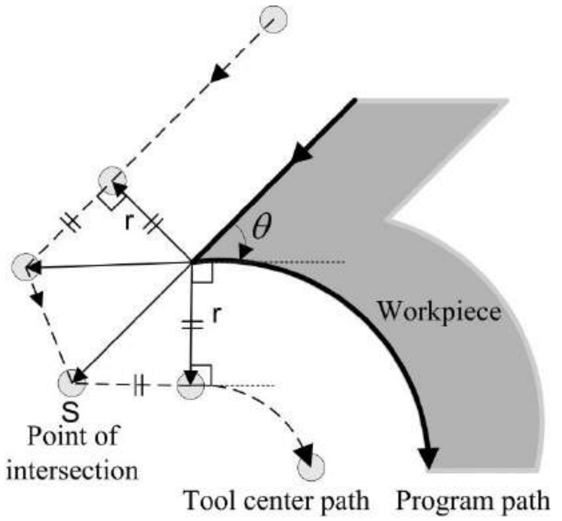
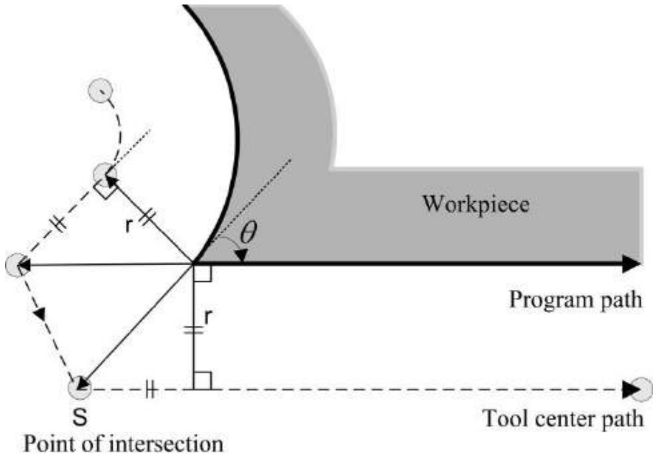
Tool will traverse along the inner edge ($180^\circ \leq \theta$)

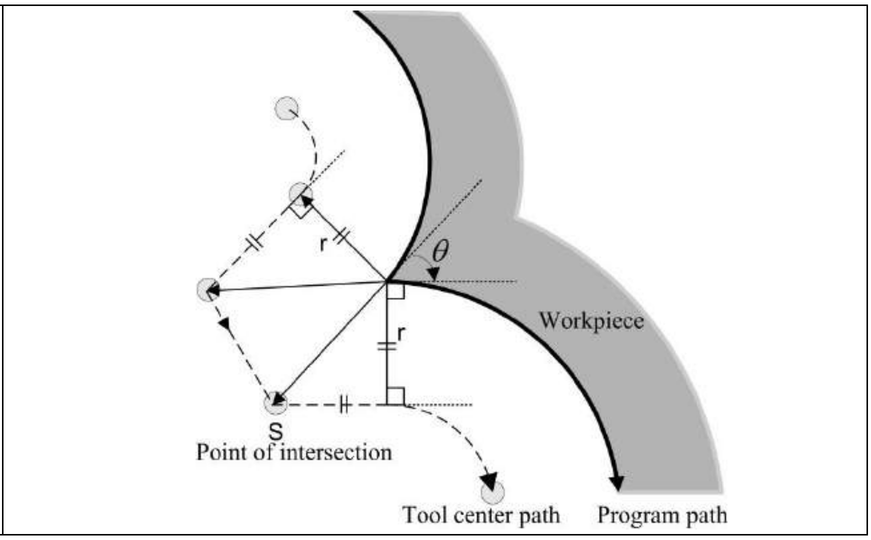
Linear ↓ Linear	
Linear ↓ Arc	
Arc ↓ Linear	
Arc ↓ Arc	

Tool moving along the outer edge of an obtuse angle ($90^\circ \le \theta < 180^\circ$)

Linear ↓ Linear	
Linear ↓ Arc	
Arc ↓ Linear	
Arc ↓ Arc	

Tool will traverse around the outside of an acute angle ($\theta < 90^{\circ}$)

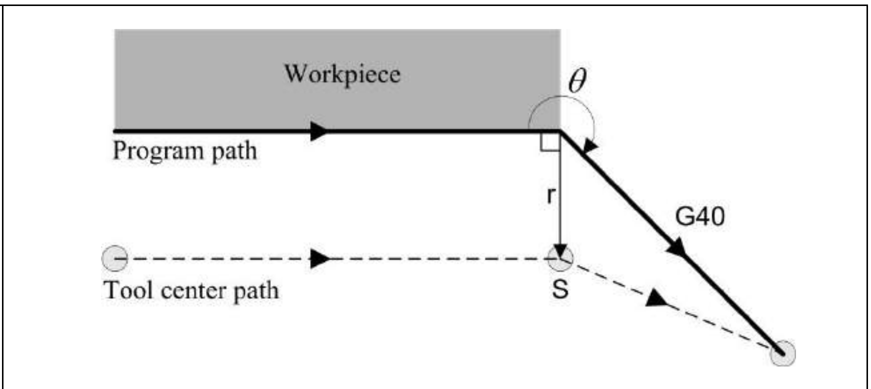
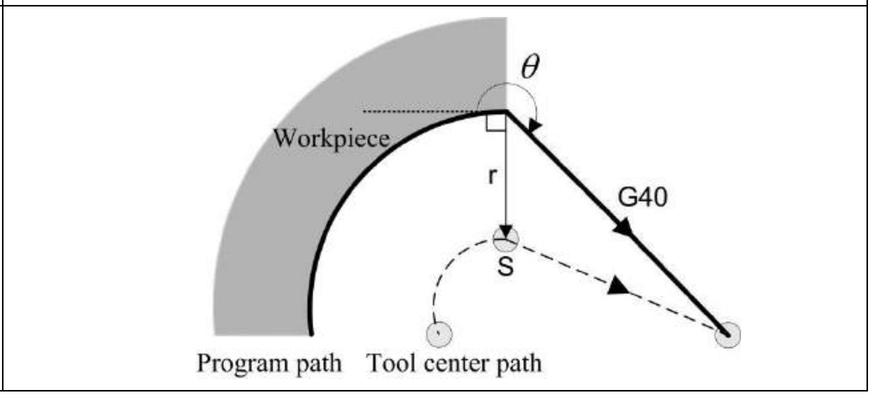
Linear → Linear	
Linear → Arc	
Arc → Linear	

Arc ↓ Arc	

Tool traversal when compensation is canceled

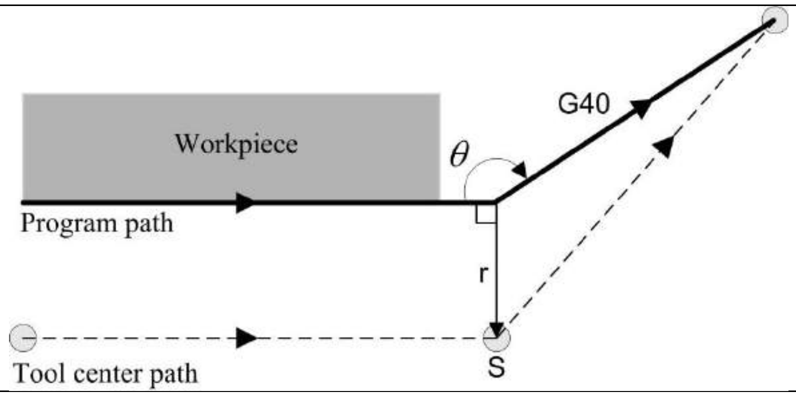
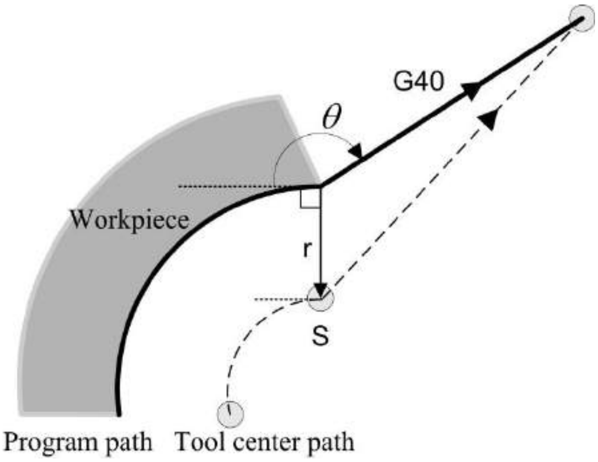
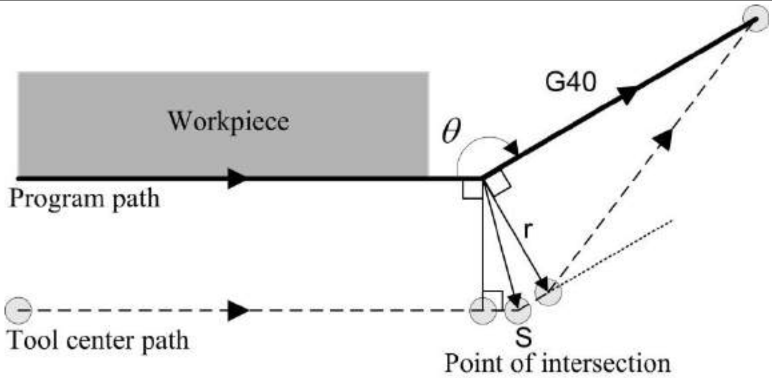
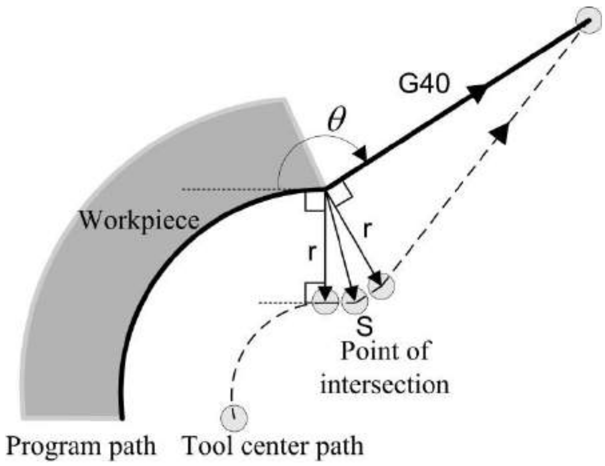
The tool traversal path when compensation is canceled is as described below

Tool will traverse along the inner edge ($180^{\circ} \leq \theta$)

Linear ↓ Linear	
Arc ↓ Linear	

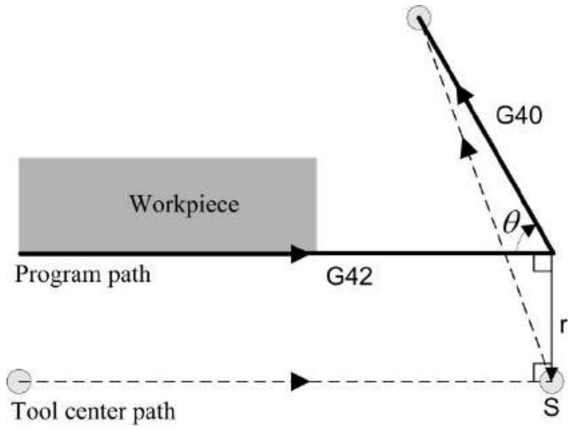
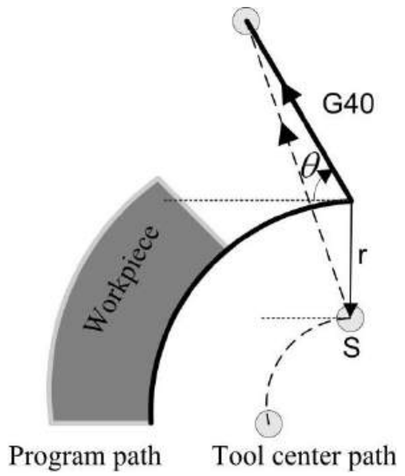
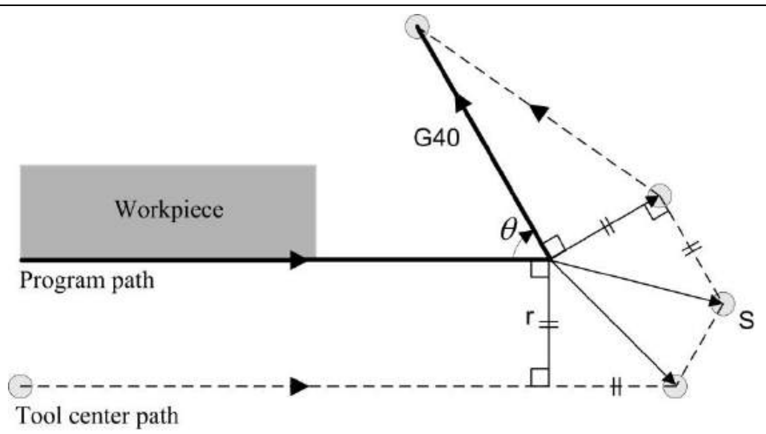
Tool moving along the outer edge of an obtuse angle ($90^{\circ} \leq \theta < 180^{\circ}$)

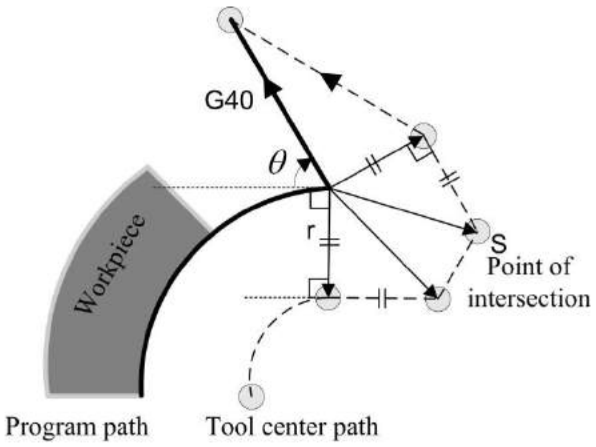
There are two types of tool paths during canceling, which are Type A and Type B, and they can be set by parameter 0131.

<table>
<tr><td rowspan="2">Type A</td><td>Linear
↓
Linear</td><td>
</td></tr>
<tr><td>Arc
↓
Linear</td><td>
</td></tr>
<tr><td rowspan="2">Type B</td><td>Linear
↓
Linear</td><td>
</td></tr>
<tr><td>Arc
↓
Linear</td><td>
</td></tr>
</table>

Tool will traverse around the outside of an acute angle ($\theta < 90^\circ$)

There are two types of tool paths during canceling, which are Type A and Type B, and they can be set by parameter 0131.

<table>
<tr><td rowspan="2">Type A</td><td>Linear
↓
Linear</td><td>
</td></tr>
<tr><td>Arc
↓
Linear</td><td>
</td></tr>
<tr><td>Type B</td><td>Linear
↓
Linear</td><td>
</td></tr>
</table>

	Arc ↓ Linear	

G43, G44, G49 tool length compensation

Command format:

```
G43H_;
G44H_;
G49;
```

Argument description:

- G43 : Positive direction tool length compensation command. If the tool compensation value is positive, the tool axis will traverse in the positive direction.
- G44 : Negative direction tool length compensation command. If the tool compensation value is negative, the tool axis will traverse in the negative direction.
- G49 : Cancel tool length compensation.
- H__ : Tool length compensation number, setting rage is within 1 to 400, and the compensation value of H0 is always 0.

Program example:

H1: 20.0mm、H2: 30.0mm

Program command	Absolute coordinates	Machine coordinates
…		
G00Z0.;	0.	0.
G43H1;	-20.	0.
Z50.;	50.	70.
G43H2;	40.	70.

Z50.;	50.	80.
G49;	80.	80.
…		

H1: 20.0mm、H2: 30.0mm

Program command	Absolute coordinates	Machine coordinates
…		
G00Z0.;	0.	0.
G44H1;	20.	0.
Z50.;	50.	30.
G44H2;	60.	30.
Z50.;	50.	20.
G49;	20.	20.
…		

Note:

1. G53, G28 and G30 under the tool length compensation effect

   When the three commands G53, G28, and G30 are executed within the range of tool length compensation effect, NC will automatically cancel the tool length compensation value and switch to G49 status.

H1: 20.0mm

Program command	Absolute coordinates	Machine coordinates
…		
G00Z0.;	0.	0.
G43H1;	-20.	0.
G00Z50.;	50.	70.
G91G28Z0.;	0.	0.
G00Z50.;	50.	50.
…		

2. M30, M02 under the tool length compensation effect

   When the two program termination commands M30, M02 are executed within the range of tool length compensation effect, NC will automatically cancel the tool length compensation value and switch to G49 status.

3. RESET under the tool length compensation effect

   When NC receives the RESET signal within the range of tool length compensation effect, the tool length compensation value will ne canceled automatically and switch to G49 status.

G50, G51 zoom command

Command format:

$$\text{G51 X\_\_ Y\_\_ Z\_\_}\begin{bmatrix}\text{P\_\_} \\ \text{I\_\_J\_\_K\_\_}\end{bmatrix};$$

G50;

Argument description:

G51	:	Enable proportional zoom.
G50	:	Cancel proportional zoom.
X__ Y__ Z__	:	Coordinate value of the proportional center point.
P__	:	Magnification is a value without decimal point; it has the unit of 0.001X and a setting range of 1 to 99999 (equivalent to 0.001X to 99.999X. The setting of 1000 indicates 1X magnification), and it is the same for every axis.
I__ J__ K__	:	Zoom magnification of each axis.

Operation description:

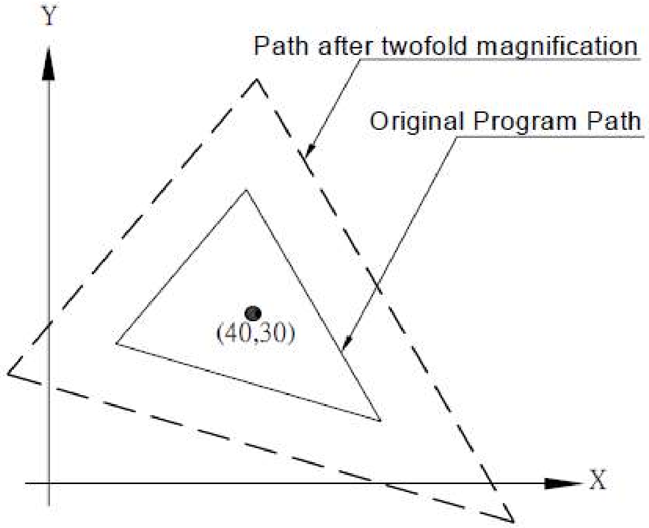
The proportional zoom is based on the use of P___ or I___ J___ K___. When R, I, and J are all written with value, the system will use the value assigned in P as the reference.

When proportional zoom is based on the use of P__, the activation of the proportional zoom function for each axis is determined by whether the proportional center point coordinate of each axis is set. For example, with a 2X magnification along X and Y axis directions, the proportional center point is at the position (40,30):
G51 X40. Y30. P2000;

When the proportional zoom is based on the use of I__J__K__, the activation of the proportional zoom function for each axis is determined by whether the zoom magnification of each axis is set. For example, with a 2X magnification along X and Y axis directions, the proportional center point is at the position (40,30):
G51 X40. Y30. I2. J2.;

Plot legends:

G90 G51 X40. Y30. P2000.



G50.1, G51.1 mirror image command

Command format 1:

```
G51.1 X__ Y__ Z__;
```

Command format 2:

```
G50.1 X__ Y__ Z__;
```

Command format 3:

```
G50.1;
```

Argument description:

Function 1: Enable the mirror image function.

G51.1	:	Enable the mirror image function.
X__ Y__ Z__	:	Assign the absolute coordinate position of the mirror image axis.

Function 2: Cancel the mirror image function for the assigned axial direction.

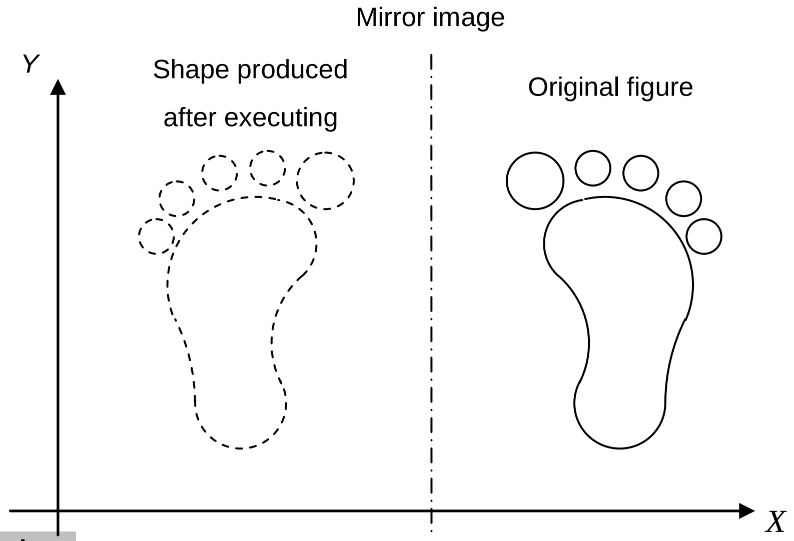
G50.1	:	Cancel the mirror image function.
X__ Y__ Z__	:	Cancel the assignment of mirror image axis, where the argument value is assigned as arbitrary value without any meaning.

Function 3: Cancel the mirror image function for all three axial directions.

G50.1	:	Cancel the mirror image function.

Operation description:

For cutting symmetric shapes, users only need to select the machining program along one direction, and then the symmetrical shape along the other direction can be machined via the mirror image function. As shown in the figure below: After cutting of the right sole is completed, the mirror image function can be used cut the left sole with the same machining program.



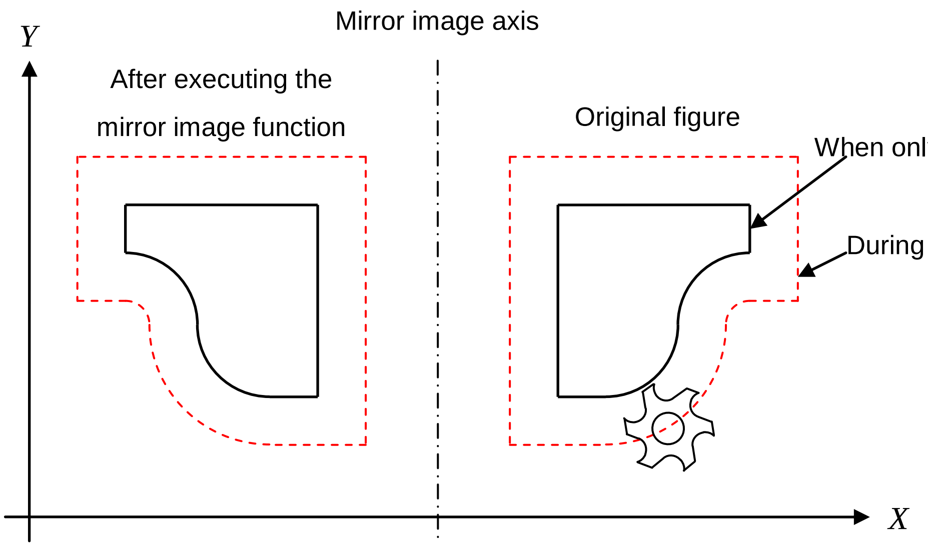
Characteristic description:

The characteristics of the mirror image function are described below:

1. The absolute coordinate position of a mirror image axis can be assigned by the mirror image function in accordance with the incremental/absolute command.
2. The mirror image axis assigned by the mirror image function command can only take effect in coordination with plane selection. For example: On the G17 plane, only X and Y axes' mirror image axis are valid, while the Z axis' mirror image axis is not valid.
3. When the mirror image function is activated, the tool nose radius compensation remains valid as shown in the legends of special item 3.
4. When the mirror image function is activated, and G28 and G30 are being executed, the operating mirror image reaching the mid point remains valid, but it becomes invalid during the process of returning to reference point via the mid point as shown in the plot legends of special item 4.
5. The position of mirror image axis will vary along with the offset of coordinate system and tool length offset as shown in the figure legends of special item 5.
6. G53 command mirror image is invalid.
7. When M30 is encountered, RESET will cancel the mirror image function. When M99 and M02 are encountered, the mirror image function will not be canceled.

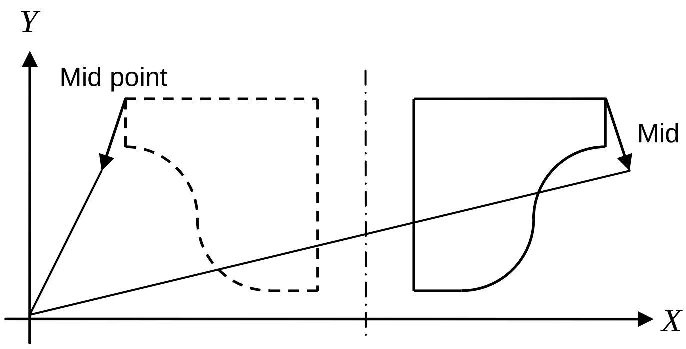
Plot legends for special item 3:

A diagram demonstrating simultaneous execution of the mirror image function and tool path compensation



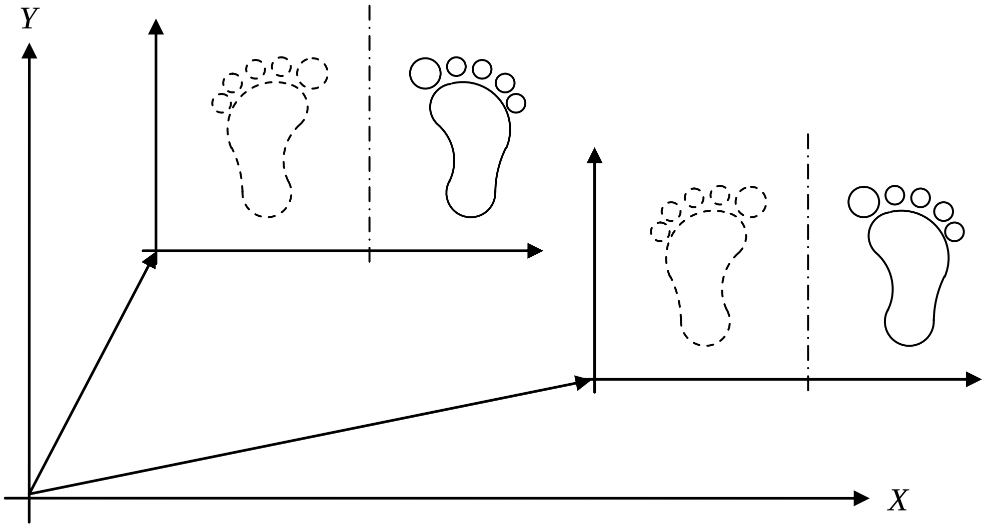
Plot legends for special item 4:

When the mirror image function is executing G28 and G30, the section between the mid point and the reference point of the operating mirror images is invalid



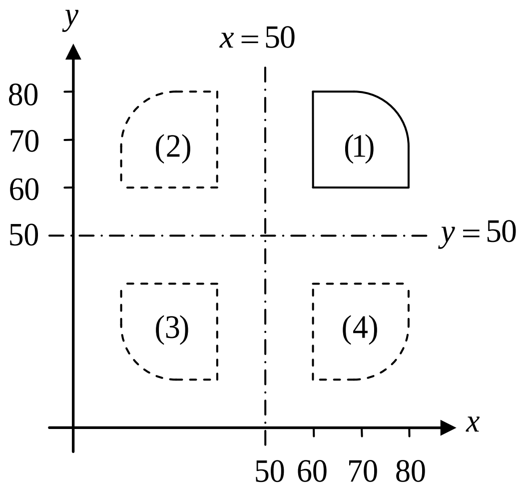
Plot legends for special item 5:

The position of the mirror image axis will vary along with coordinate offset



Program example:

A program example of the mirror image command is shown in the figure below:



Main program	
G90;	
M98 P0001;	Cutting tile 1
G51.1 X50.;	X=50 is assigned as the mirror image axis
M98 P0001;	Cutting tile 2
G51.1 Y50.;	Y=50 is assigned as the mirror image axis, while X=50 remains as the mirror image axis
M98 P0001;	Cutting tile 3
G50.1 X0.;	Cancel the assignment of X axis as the mirror image axis, while Y=50 remains as the mirror image axis
M98 P0001;	Cutting tile 4
G50.1 Y0.;	Cancel the assignment of Y axis as the mirror image axis
M30	

Subprogram O0001	
G00 G90 X60. Y60.	
G01 X80. F100;	
G01 Y70.;	
G03 X70. Y80. R10.;	
G01 X60.;	
G01 Y60.;	
M99	

G52 Interval coordinate system settings

Command format:

G52 <Axis name><Interval coordinate system reference point>;

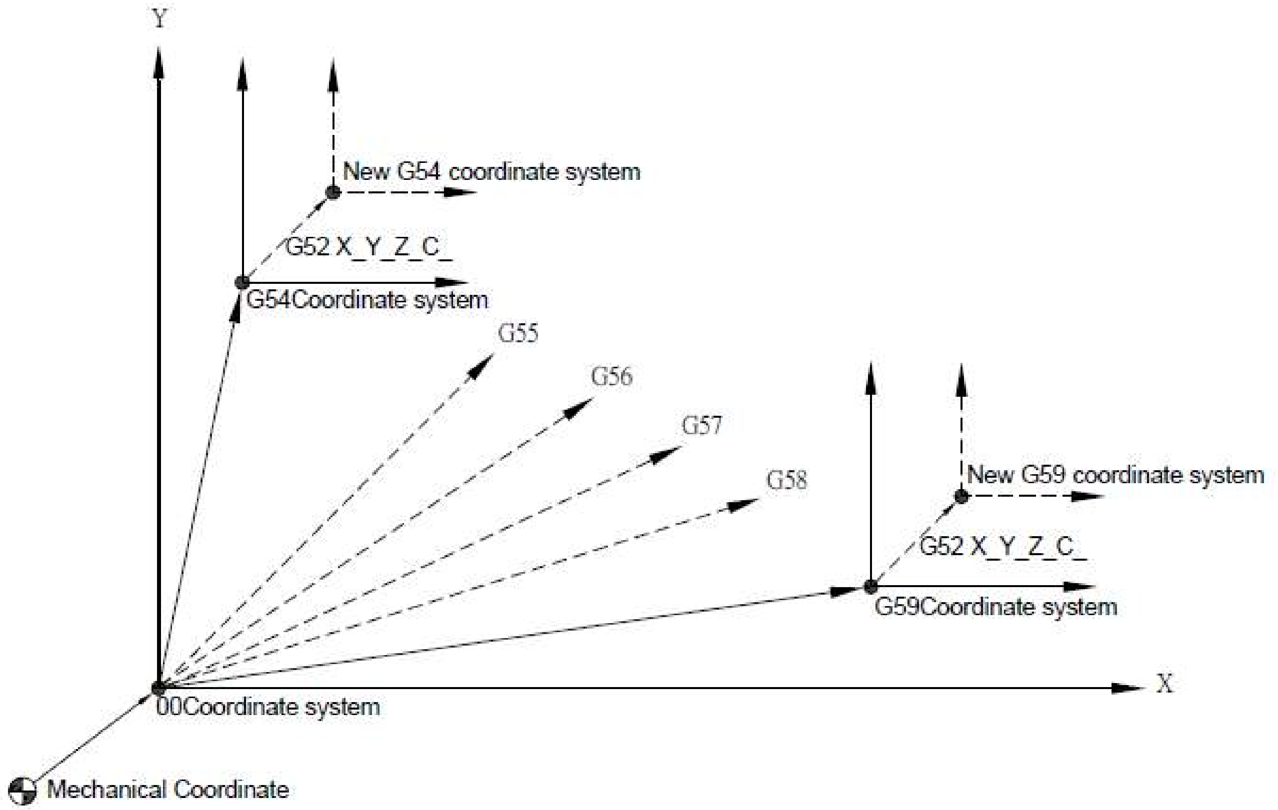
Argument description:

Axis name : It is for setting the interval coordinate system's reference point of the machining coordinate system on a given axis (G54 to G59), which can be any combination of X, Y, Z, A, B, C or U, V, W. However, it must match the current axis name (axis name is set up by parameters 70464 to 70495).

Operation description:

The G52 command can be used to set another interval coordinate system in all machining coordinate systems (G54 to G59), which can sometimes facilitate program editing. Once the setting of G52 is completed, the traversal command under absolute mode (G90) is the interval coordinate system set up for G52.

Plot legends:



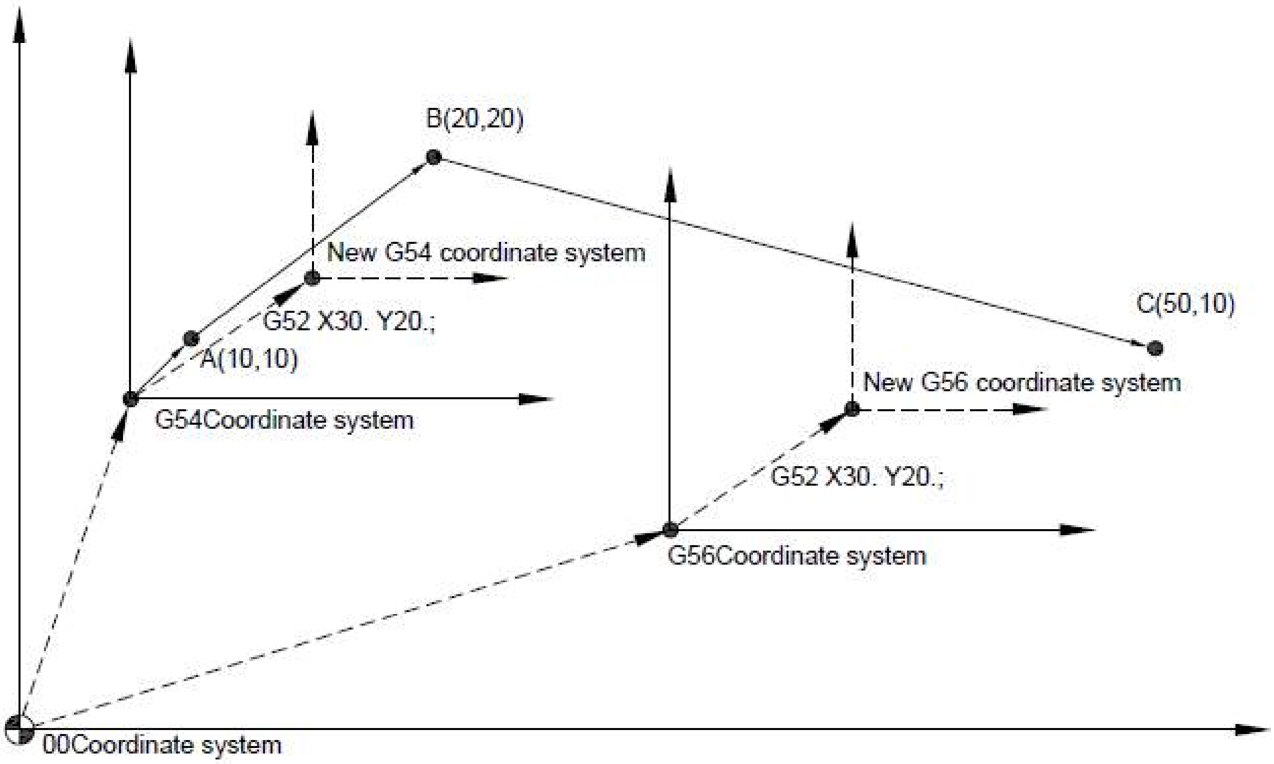
Program example:

G90 G54 G00 X10. Y10.;

G52 X30. Y20.;

G00 X20. Y20.;-- (A→B)

G56 G00 X50. Y10.; --(B→C)



The interval coordinate system set by G52 can be canceled by using two different methods. The first method is that the interval coordinate system set by G52 will be canceled upon system RESET. The second method is by executing the G52 command again with the argument set as 0.

For example:

G52 X30. Y20.;

..

G52 X0. Y0;(Cancel coordinate system G52)

Rapid positioning of the G53 machine coordinate system

Command format:

```
G53 <axis name><target position>;
```

Argument description:

Axis name	:	This is for assigning the name of axial direction to be moved, which can be any combination of X, Y, Z, A, B, C, or U, V, W. However, it must match the current axis name (axis name is set up by parameters 70464 to 70495).
Target position	:	Machine coordinate values of a target point.

Operation description:

The G53 command can be used to control tool traversal to the assigned machine coordinate location. As for the G53 command, its tool traverse method is based on rapid feeding with traverse speed set by parameters 60286 to 60317. The G53 command is valid within a single block, and it can only be used under absolute mode (G90); it is invalid under incremental mode (G91).

It must be noted that the previously assigned tool length compensation value will automatically be canceled after executing G53.

Selection of machining coordinate systems G54 to G59

Command format:

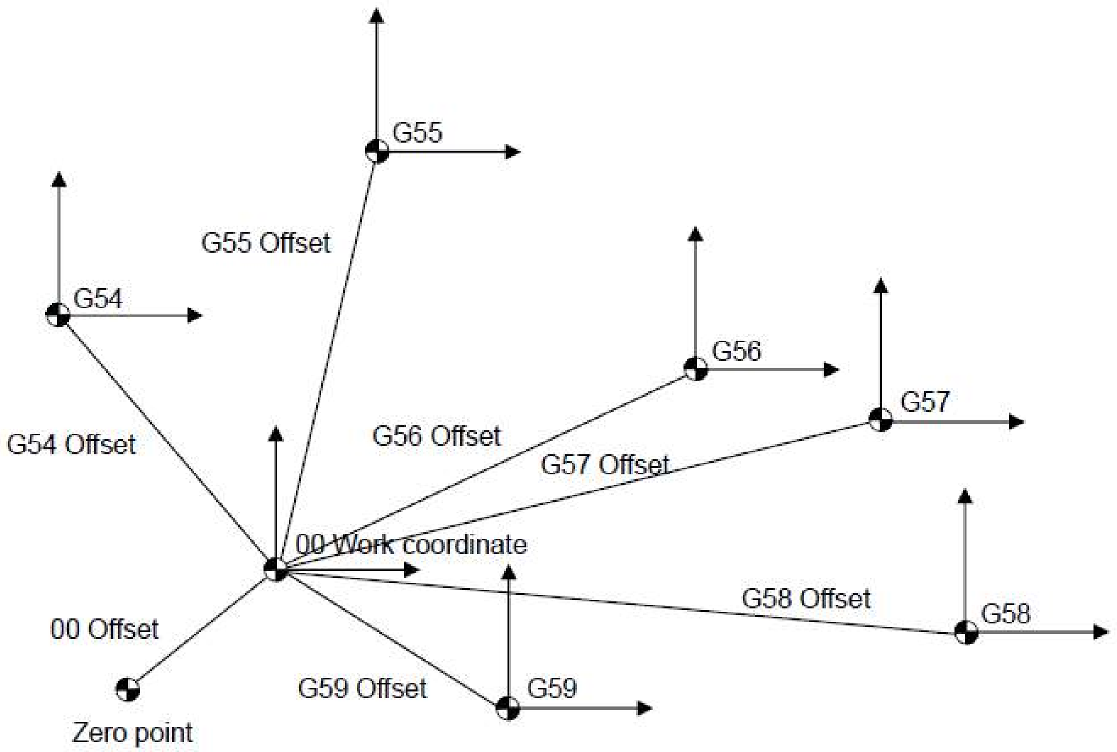
$$\begin{bmatrix} G54; \\ G55; \\ G56; \\ G57; \\ G58; \\ G59; \end{bmatrix}$$

Operation description:

In the workpiece coordinate system, the six G codes from G54 to G59 are used to represent six different coordinate systems which can be arbitrarily selected in accordance with machining requirements. The coordinate system will be restored to coordinate system G54 after system RESET.

The reference point offset for each coordinate system can be set in 〈OFFSET〉→ 〈Coordinate system setting〉, and detailed descriptions are included in the operating manual; it can also be set via the G10 command. For more information, please refer to the G10 command section.

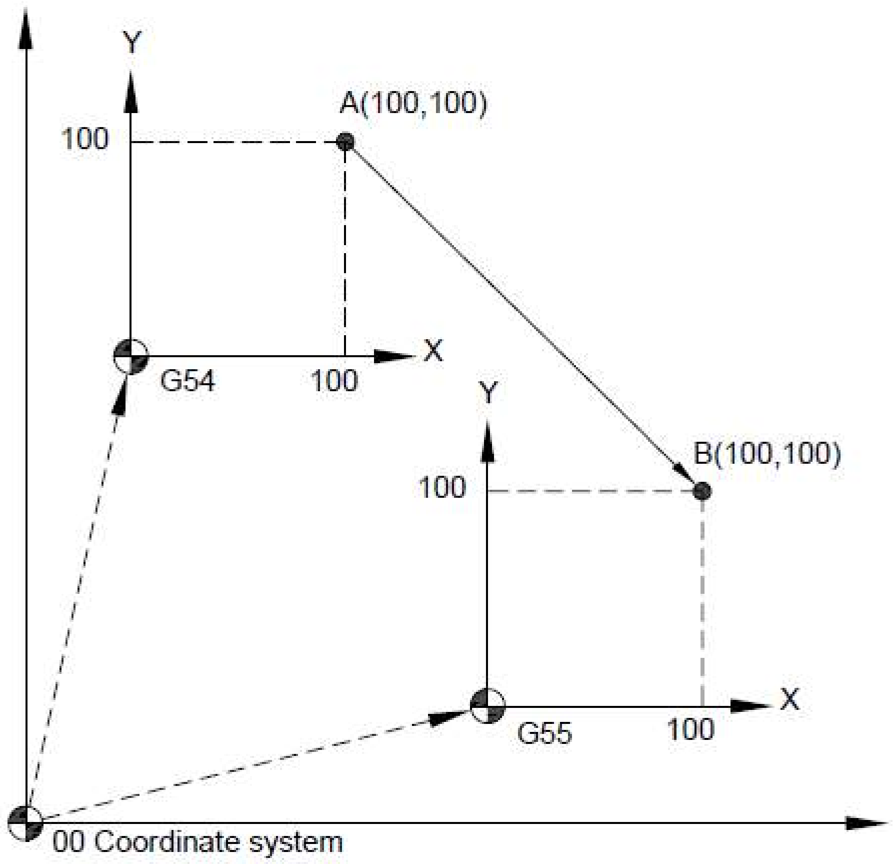
The relationship between each coordinate system is as shown below:



Program example:

G90 G54 G00 X100. Y100.;

G55 X100. Y100.;(A→B)



G61, G64 correct positioning mode, general cutting mode

Command format:

```
G61;
G64;
```

Argument description:

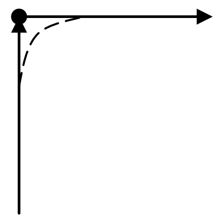
G61	:	Correct positioning mode.
G64	:	General cutting mode.

Operation description:

The function of G61 is completely identical to G09. However, G09 is only valid inside this single block, while G61 is valid after declaration until G64 (general cutting) is declared. G64 is the system default mode. It will always be in G64 mode unless G61 is declared.

As for cutting commands (G01/G02/G03), the positioning precision of each axis is set by parameters 56064 to 56095. As for rapid positioning (G00), the positioning precision of each axis is set by parameters 56096 to 56127.

Plot legends:



———— Tool path under G61 mode.

– – – – – Tool path under G64 mode.

Program example:

G61 G91 G01 Y100. F200.; ---(Correct positioning)

X100.;---(Correct positioning)

G64; -- (Stop correct positioning)

G65 macro single call

Command format 1:

```
G65 P__ L__ <Argument…>;
```

Command format 2:

```
G65 "String" L_ <Argument…>;
```

Command format 3:

```
G65 "String" P_ L_ <Argument…>;
```

Argument description:

Function 1: Use the P argument to assign a macro name.

P__	:	If the macro number (name of macro without the 4-digit number behind character "O") to be called is not entered, it will trigger the system alert 【510301 - Name of the program to be called is invalid】.
L__	:	Default value for the number of repetitions is 1 if a value was not entered.

Function 2: LNC advanced usage: a macro name is assigned by using strings.

"String"	:	An arbitrary string can be assigned, but the string length must not exceed 32 characters otherwise it will trigger system alert 【510301 - Name of the program to be called is invalid】.
L__	:	Default value for the number of repetitions is 1 if a value was not entered.

Function 3: LNC advanced usage: a macro name is assigned by a combination of strings and P argument.

"String"	:	An arbitrary string can be assigned, but the string length must not exceed 28 characters otherwise it will trigger system alert 【510301 - Name of the program to be called is invalid】.
P__	:	The number of the macro name to be assembled (macro name without the 4-digit number behind a "string").
L__	:	Default value for the number of repetitions is 1 if a value was not entered.

In addition to aforementioned P and L arguments, other arguments can be sent in via the NC address (English alphabets except for "G" and "N"), in no particular order. These arguments correspond to the local variables of the macro being called. A correspondence table is shown below:

NC address	Local variables
A	#1
B	#2
C	#3
D	#4
E	#5
F	#6
H	#8
I	#9

NC address	Local variables
J	#10
K	#11
L	#12
M	#13
O	#15
P	#16
Q	#17
R	#18

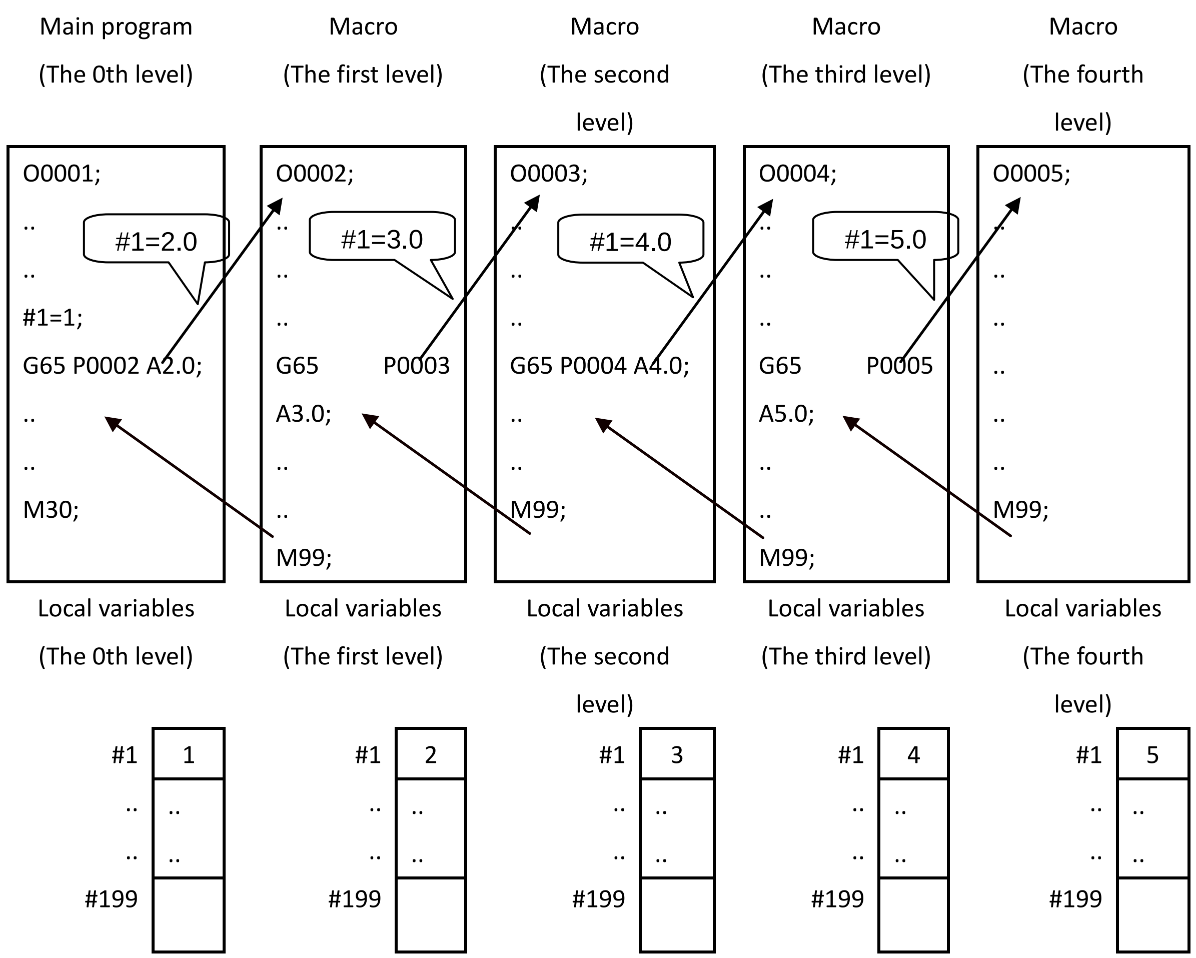
NC address	Local variables
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

```
O0001;
.
.
G65 P0008 L1 A2. B 3.;
.
.
M30;
```

```
#1=2.
#2=3.
```

```
O0008;
#3=#1+#2;
G00 X#3;(Equivalent to G00 X5.0;)
M99;
```

In the G65 single block, G65 must be edited before all other arguments. G65 can be used for nested calling. The maximum combinations of G65 and G66 can be up to 4 levels (excluding the main program at the 0th level). Each level has its own local variables as shown in the figure below:



The difference between calling a macro (G65) and calling a general subprogram (M98):

1. M98 cannot be used to assign arguments; the G65 command can be used to assign arguments.
2. M98's local variable levels are fixed; local variables of G65 are changing in accordance with the nested depth (for example, the meaning of #1 is the same before and after M98, but it is different for G65).

The combination of M98 calling level and G65, G66 can be as many as 6 levels; the calling level of G65, G66 can be as many as 4 levels.

Program example:

G65 P0008 L1 A2. B3.;---------------------------------- Command format 1, which will call a macro named O0008

G65 "HELLO" L1 A2. B3.;------------------------------- Command format 2, which will call a macro named HELLO

G65 "ABC" P0001 L1 A2. B3.;------------------------ Command format 3, which will call macro named ABC0001

G66 macro modal call

Command format 1:

G66 P__ L__ <Argument…>;

Command format 2:

G66 "String" L_ <Argument…>;

Command format 3:

G66 "String" P_ L_ <Argument…>;

Argument description:

Function 1: Use the P argument to assign a macro name.

P__ : If the macro number (name of macro without the 4-digit number behind character "O") to be called is not entered, it will trigger the system alert 【510301 - Name of the program to be called is invalid】.

L__ : Default value for the number of repetitions is 1 if a value was not entered.

Function 2: LNC advanced usage: a macro name is assigned by using strings.

"String" : An arbitrary string can be assigned, but the string length must not exceed 32 characters otherwise it will trigger system alert 【510301 - Name of the program to be called is invalid】.

L__ : Default value for the number of repetitions is 1 if a value was not entered.

Function 3: LNC advanced usage: a macro name is assigned by a combination of strings and P argument.

"String" : An arbitrary string can be assigned, but the string length must not exceed 28 characters otherwise it will trigger system alert 【510301 - Name of the program to be called is invalid】.

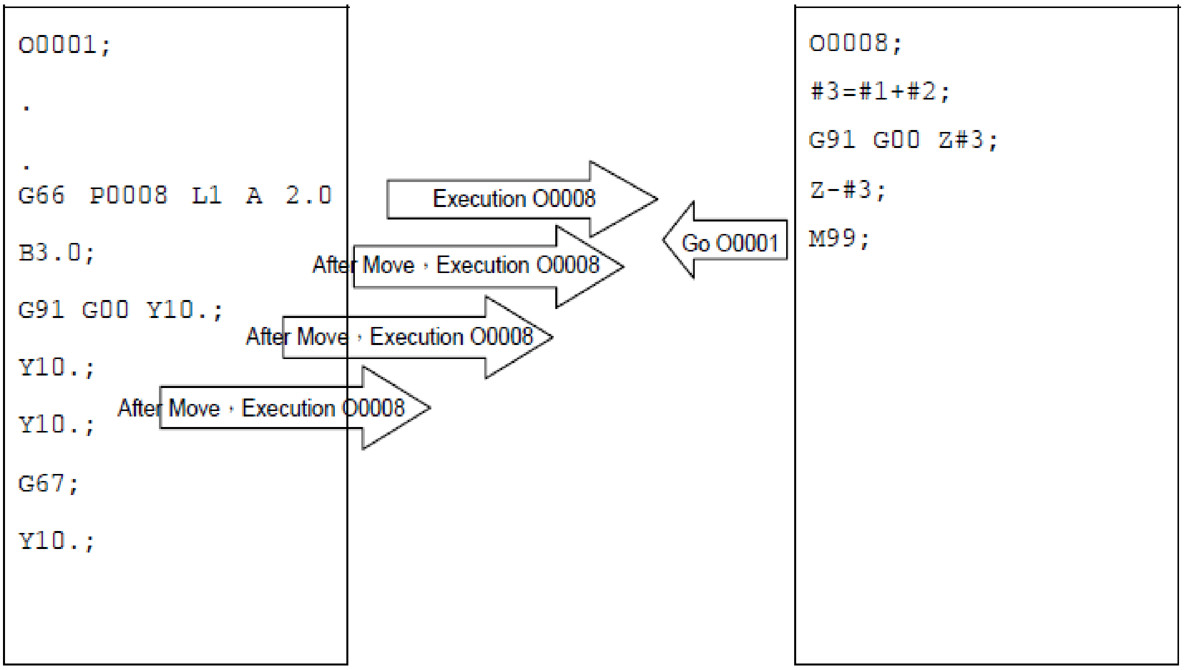
P__ : The number of the macro name to be assembled (macro name without the 4-digit number behind a "string").

L__ : Default value for the number of repetitions is 1 if a value was not entered.

In addition to aforementioned P and L arguments, other arguments can be sent in via the NC address (English alphabets except for "G" and "N"), in no particular order. These arguments correspond to the local variables of the macro being called. Please refer to the correspondence table listed under G65.

Operation description:

The difference between G66 and G65 is that the latter only calls a macro once, but the macro called by G66 will be called again after the end of every traversal single block until G67 is used to cancel the modal call.



In the G66 single block, G66 must be edited before all other arguments. Just like G65, G66 can be used for nested calling. The combination of G66 and G65 can be as high as the fourth level (excluding the main program, which is the 0th level). However, arguments of G66 (correspond to local variables of a macro) are only set once in the G66 single block, and they will not be set again in subsequent modal calls.

Program example:

G66 P0008 L1 A2. B3.;---------------------------------- Command format 1, which will call a macro named O0008

G66 "HELLO" L1 A2. B3.;------------------------------- Command format 2, which will call a macro named HELLO

G66 "ABC" P0001 L1 A2. B3.;------------------------ Command format 3, which will call macro named ABC0001

G67 Cancel macro modal call

Command format:

```
G67;
```

Operation description:

G67 is for canceling G66 macro modal call.

G68, G69 coordinate rotation

Command format:

$$\mathrm{G68}\begin{bmatrix} \mathrm{G17\ X\_\_\ \ Y\_\_} \\ \mathrm{G18\ Z\_\_\ \ X\_\_} \\ \mathrm{G19\ Y\_\_\ \ Z\_\_} \end{bmatrix}\mathrm{R\_\_;}$$

G69;

Argument description:

X__Y__ : Assign a coordinate value of rotation center in the G17 plane.

Z__X__ : Assign a coordinate value of rotation center in the G18 plane.

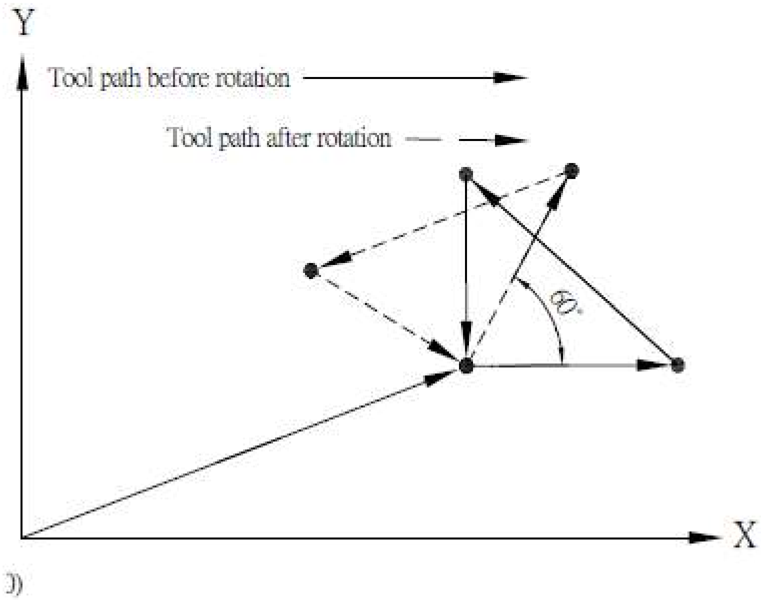
Y__Z__ : Assign a coordinate value of rotation center in the G19 plane.

If a rotation center is not assigned, the current position of G68 will be declared as the rotation center.

R__ : Rotation angle, where a positive value indicates the rotation is in the counterclockwise direction. The input unit of this argument is determined by the value set for parameter 50018. If the value of parameter 50018 is 0, the input unit of this argument is 1 degree; if the value of parameter 50018 is 3, the input unit of this argument is 0.001 degrees. If argument R__ is not assigned, the default value can be obtained by parameter 50078; and parameter 50077 can be used to determine whether the rotational angle is an absolute or incremental value.

Plot legends:

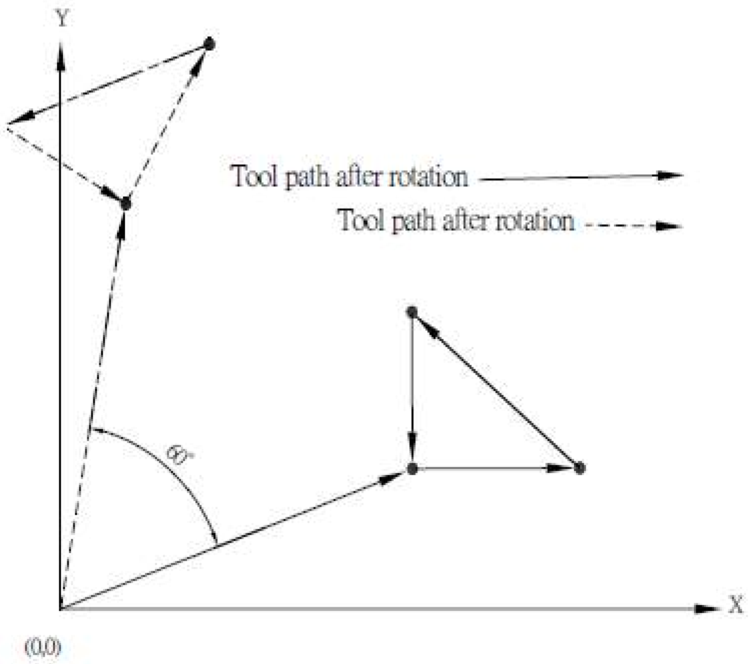
```
G90 G54 G17 G00 X0. Y0.;
G68 X20. Y10. R60.;
G01 X20. Y10. F1000.;
G91 X10.;
X-10. Y10.;
Y-10.;
G90 G69 G00 X0. Y0.;
```



If the traversal single block right behind G68 is in incremental mode (G91), the current position of G68 will be declared as the rotation center.

Plot legends:

```
G90 G54 G17 G00 X0. Y0.;
G68 X20. Y10. R60.;
G91 G01 X20. Y10. F1000.;
X10.;
X-10. Y10.;
Y-10.;
G90 G69 G00 X0. Y0.;
```



G73 High speed peck drilling cycle

Command format:

G73 X__ Y__ Z__ R__ Q__ K__ F__;

Argument description:

X__Y__	:	Coordinates of the hole position (mm).
Z__	:	Coordinates of the hole base (mm).
R__	:	Coordinate value of R point (which is the reset point) (mm).
Q__	:	Feed amount per cut (mm). Q must be assigned as (> 0), or it will trigger system alert 【610010 G73/G83 command macro - the argument for cutting feed rate is not assigned (Q should be examined)】.
K__	:	Number of repetitions.
F__	:	Feed rate (G94 mm/min).

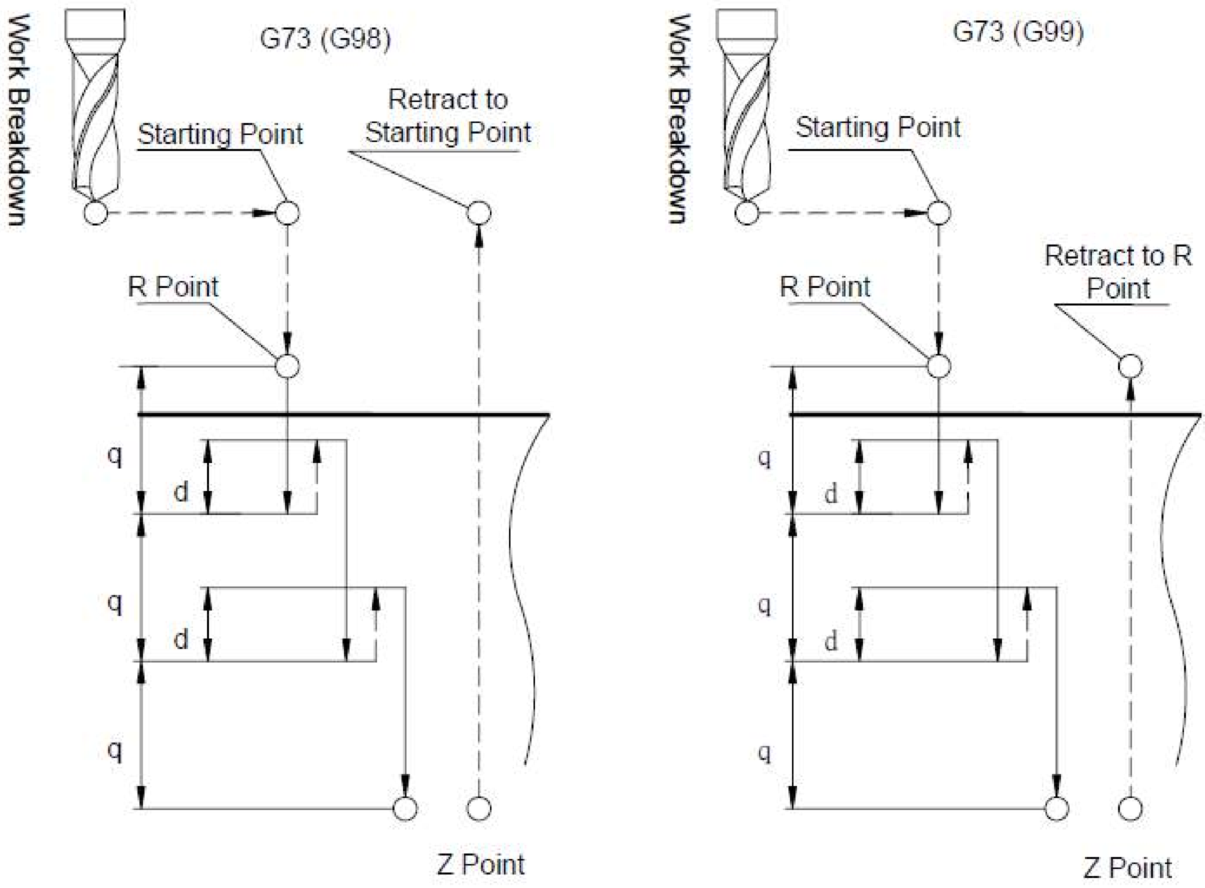
The amount of retraction for Z axis machining should be set by system parameter 160030.

Operation description (using the G17 plane as an example):

Q must be assigned while using the G73 command, or it will trigger system alert 【610010 G73/G83 command macro - the argument for cutting feed rate is not assigned (Q should be examined)】.

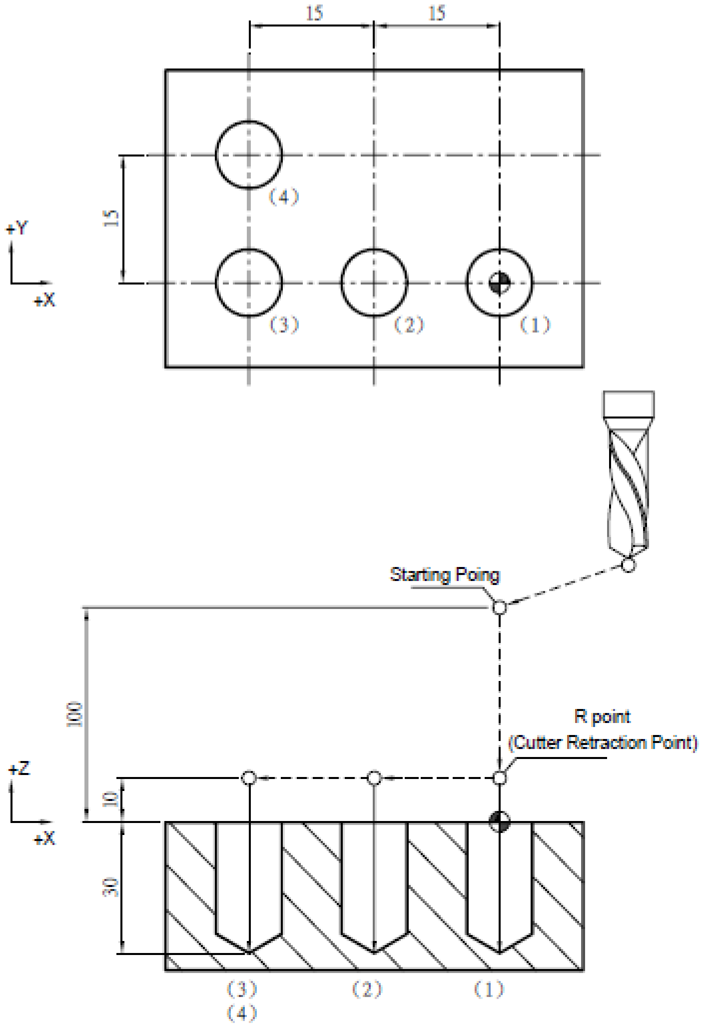
1. It is rapidly positioned to reach the hole location (X, Y, while maintaining the original tool height);
2. It is rapidly positioned to reach the R point coordinate (R);
3. based on the set cutting feed rate, spindle rotation speed, and cutting-peck drilling feed (Q);
4. Rapid tool retraction, with retraction amount set by system parameter 160030;
5. based on the set cutting feed rate, spindle rotation speed, and cutting/hole drilling (peck drilling feed + amount of peck drilling retraction);
6. Rapid tool retraction, with retraction amount set by system parameter 160030;
7. Repeat steps 5 to 6 until cutting reaches the base of the hole;
8. Under G98 mode, it is rapidly retracted to the reference point; under G99 mode, it should be rapidly retracted to the R point;
9. When K is assigned (> 1), steps 2 to 6 mentioned above should be repeated until completing the specified number of repeated drilling; otherwise the program terminates;
10. In G91 mode, argument R is for assigning the distance between R point and the reference point; argument Z is for assigning the distance between the hole base position and R point; when K is being assigned (> 1), after completion of each drilling operation (steps 2 to 8 mentioned above), a hole position offset will be implemented in accordance with the assigned X, Y before continuing with the next drilling operation.
11. The difference between G73 and G83 is that the former's amount of retraction is set by system parameter 160030, while the latter will be retracted to R point every time.

Plot legends:

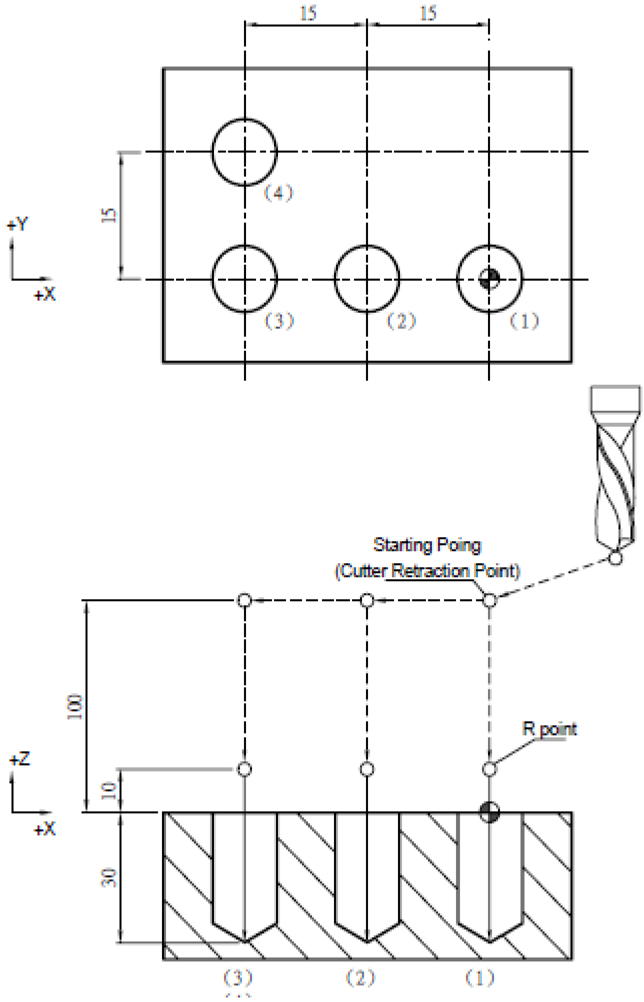


Program example:

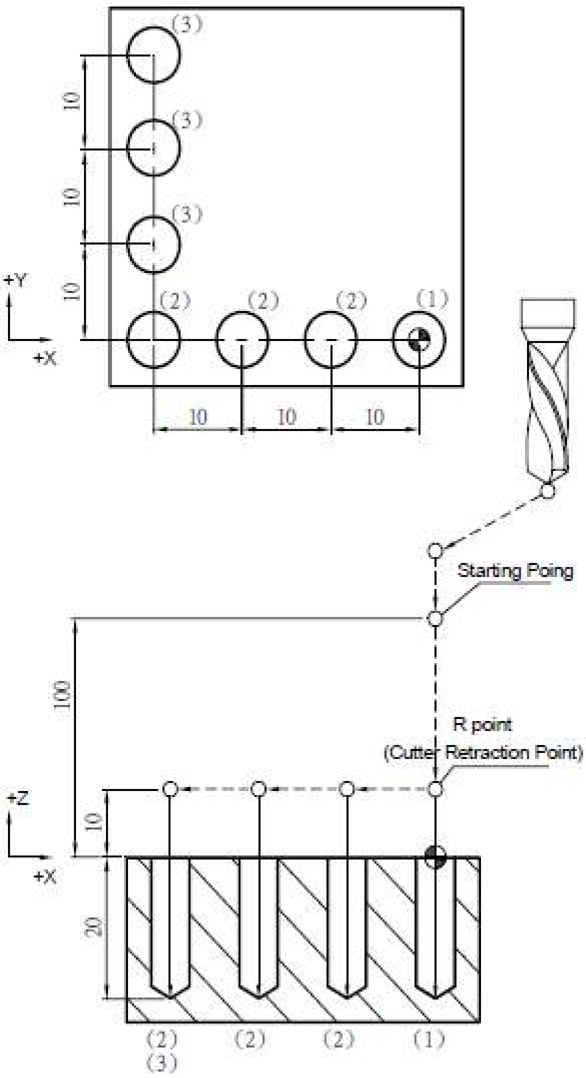
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G73 X0. Y0. Z-30. R10. Q4. K1 F100.;----------------------------------------(1)
X-15.; -----------------------------------------------------------------------(2)
X-30.; -----------------------------------------------------------------------(3)
X-30. Y15.;-------------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



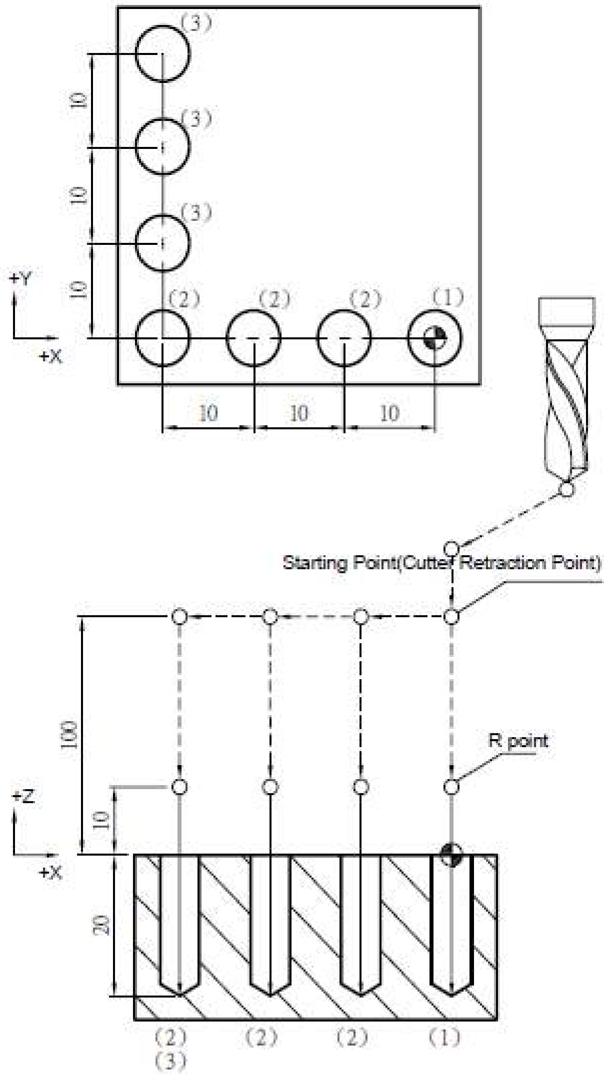
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G73 X0. Y0. Z-30. R10. Q4. K1 F100.;----------------------------------------------------------------------------------(1)
X-15.; ----------------------------------------------------------------------------------------------------------------------(2)
X-30.; ----------------------------------------------------------------------------------------------------------------------(3)
X-30. Y15.;------------------------------------------------------------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



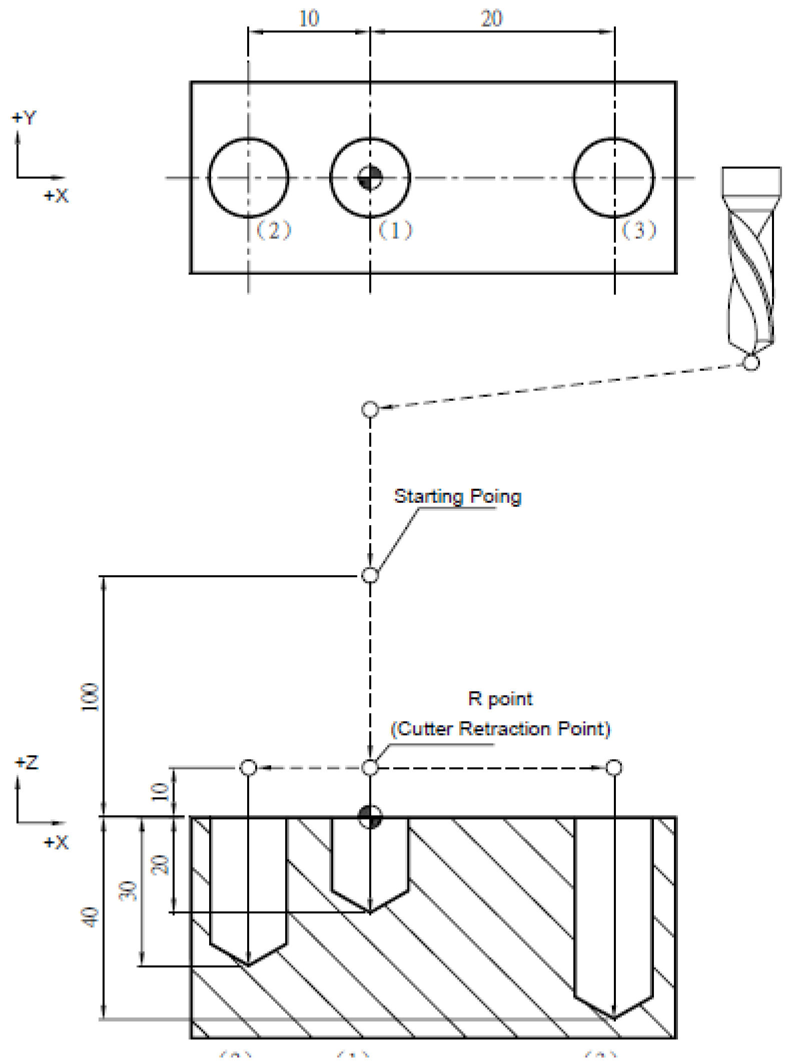
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G43 G00 H01 Z150.;
G00 Z100.;
G99 G73 X0. Y0. Z-20. R10. Q4. K1 F100.;-------------------------------------------------------------------------------(1)
G91 X-10. K3 ; -----------------------------------------------------------------------------------------------------------(2)
Y10. K3 ;----------------------------------------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



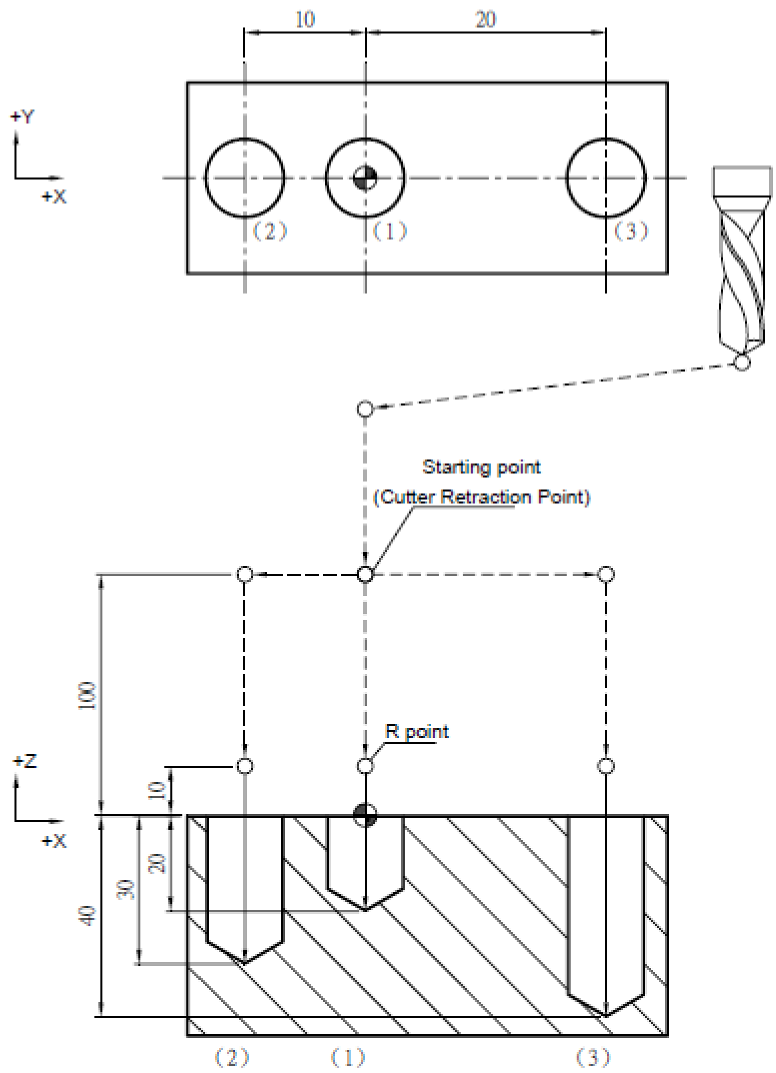
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G43 G00 H01 Z150.;
G00 Z100.;
G98 G73 X0. Y0. Z-20. R10. Q4. K1 F100.;------------------------------------------(1)
G91 X-10. K3; ---------------------------------------------------------------------(2)
Y10. K3; --------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0.Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G73 X0. Y0. Z-20. R10. Q4. K1 F100.;------------------------------------------------------------------------------(1)
X-10. Z-30.; ---------------------------------------------------------------------------------------------------------(2)
X20. Z-40.; ----------------------------------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G73 X0. Y0. Z-20. R10. Q4. K1 F100.;--------------------------------------------------------------------------------(1)
X-10. Z-30.; -----------------------------------------------------------------------------------------------------------(2)
X20. Z-40.; ------------------------------------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



G74 Left-hand thread tapping cycle

Command format:

G74 X__ Y__ Z__ R__ P__ K__ F__;

Command format 2:

G74 X__ Y__ Z__ R__ P__ Q__ K__ F__;

Argument description:

Function 1: Left-hand thread tapping cycle.

X__Y__	:	Coordinates of the hole position (mm).
Z__	:	Coordinates of the hole base (mm).
R__	:	Coordinate value of R point (which is the reset point) (mm).
P__	:	Pause time at hole base (1/1000 second), minimum unit, no decimal point allowed.
K__	:	Number of repetitions.
F__	:	Cutting feed rate (G94 mm/min, G95 mm/rev).

Function 2: Left-hand thread peck tapping cycle.

X__Y__	:	Coordinates of the hole position (mm).
Z__	:	Coordinates of the hole base (mm).
R__	:	Coordinate value of R point (which is the reset point) (mm).
P__	:	Pause time at hole base (1/1000 second), minimum unit, no decimal point allowed.
Q__	:	Feed amount per cut (mm).
K__	:	Number of repetitions.
F__	:	Cutting feed rate (G94 mm/min, G95 mm/rev).

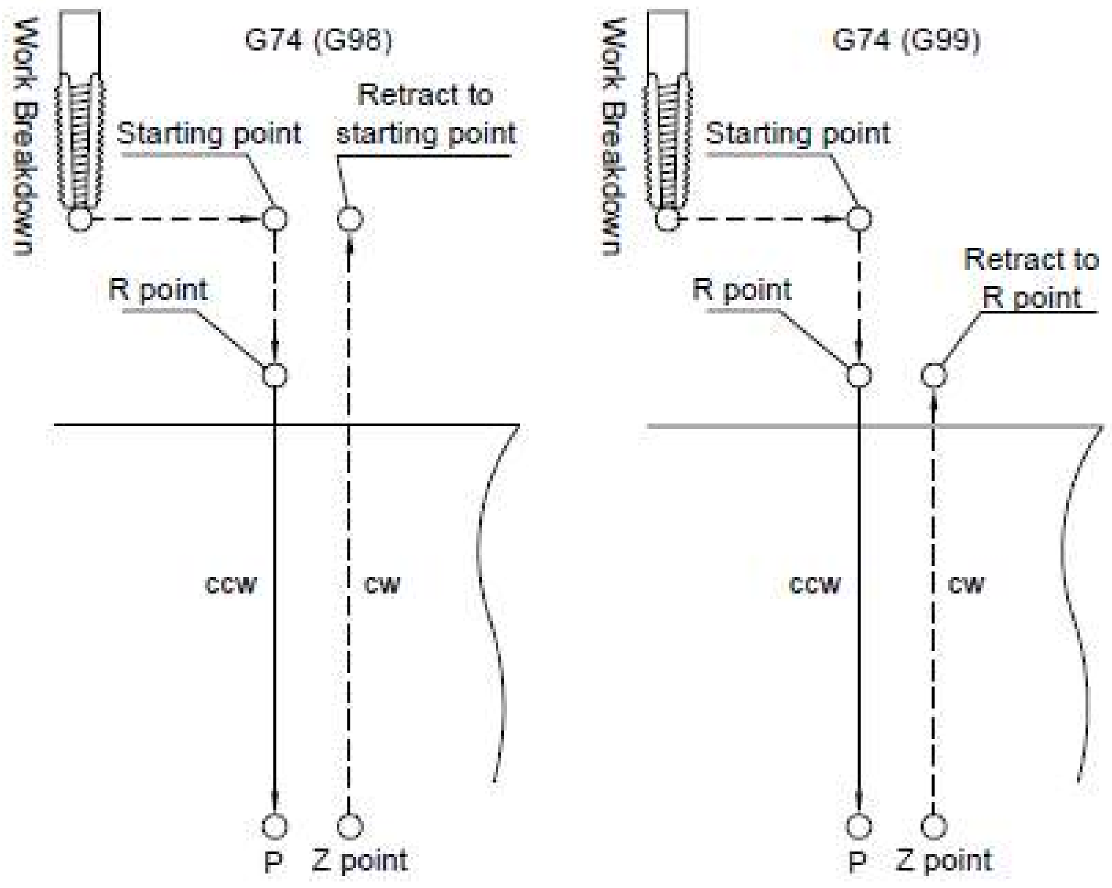
If the M29 command is added in front of G74, it will become right-hand thread rigid tapping cycle. The peck tapping mode of the peck tapping cycle can be set by parameter 160041 (0: High speed, 1: Normal). When high speed is selected, each peck tapping will only be retracted by the amount set in parameter 160042; if normal is selected, peck tapping is retracted to R point every time.

Operation description (using the G17 plane as an example):

1. It is rapidly positioned to reach the hole location (X, Y, while maintaining the original tool height);
2. It is rapidly positioned to reach the R point coordinate (R);
3. Tapping starts with the spindle in reverse rotation;

4. Cutting should reach the hole base position based on a set cutting feed rate and spindle rotation speed (Z);
5. Spindle stops; if P is assigned, it is the pause period at the hole base position;
6. Spindle is in forward rotation based on a set cutting feed rate and spindle rotational speed for the cutting to reach R point;
7. Tapping ends, spindle stops; if P is assigned, it is the pause period at the R point position;
8. Under G98 mode, it is rapidly retracted to the reference point;
9. If K(> 1) is assigned, steps 2 to 8 mentioned above should be repeated until completing the specified number of repeated tapping; otherwise the program terminates;
10. In G91 mode, argument R is for assigning the distance between R point and the reference point; argument Z is for assigning the distance between the hole base position and R point; when K is being assigned (> 1), after the completion of each tapping operation (steps 2 to 8 mentioned above), a hole position offset will be implemented in accordance with the assigned X, Y before continuing with the next tapping operation.
11. Under G94 mode, cutting feed rate F is rotation speed (S) × thread pitch (PITCH).

Plot legends:



Function 2 operation description (using the G17 plane as an example):

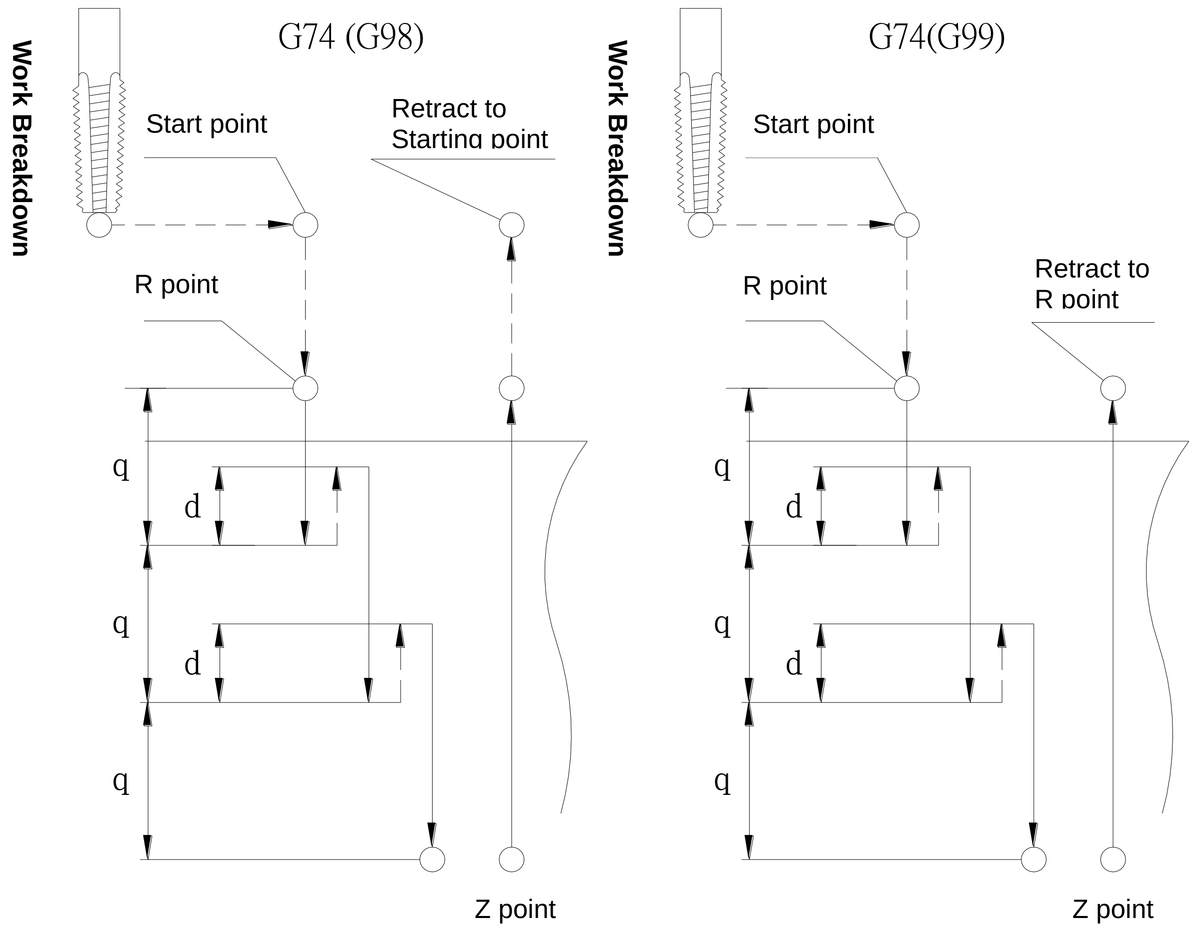
1. It is rapidly positioned to reach the hole location (X, Y, while maintaining the original tool height);
2. It is rapidly positioned to reach the R point coordinate (R);
3. Tapping starts with the spindle in reverse rotation;
4. based on the set cutting feed rate, spindle rotation speed, and cutting-peck feed;
5. Spindle stops; if P is assigned, it is the pause period at this point position;
6. Spindle in forward rotation; if parameter 160041 is set as high speed peck tapping mode, the amount of retraction will be set by system parameter 160042; if it is set as normal peck tapping mode, then it will be rapidly retracted to R point;
7. Spindle stops; if P is assigned, it is the pause period at this point position;
8. Spindle in reverse rotation; if parameter 160041 is set as normal peck tapping mode, it will be rapidly positioned to a certain height away from the previous machining point, and this height will be set by system parameter 160042;
9. Cutting feed, with the feed rate equal to (amount of peck tapping tool feed + the set value of system parameter 160042);

10. Spindle stops; if P is assigned, it is the pause period at the hole base position;
11. Spindle in forward rotation; rapidly retracted to R point;

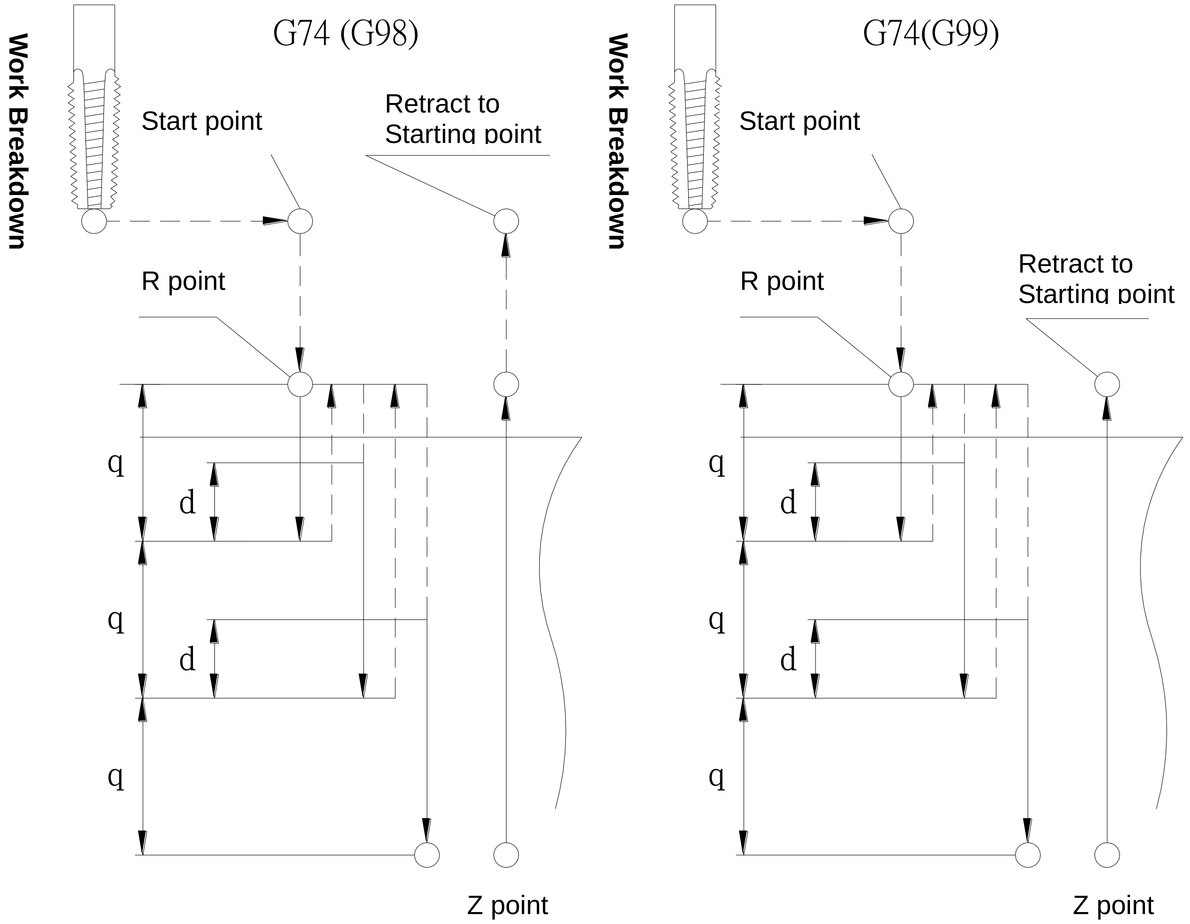
12. Steps 8 to 11 will be repeated until cutting reaches the base of the hole;
13. Spindle is in forward rotation based on the set cutting feed rate and spindle rotational speed for the cutting to reach R point position;
14. Tapping ends, spindle stops; if P is assigned, it is the pause period at the R point position;
15. Under G98 mode, it is rapidly retracted to the reference point;
16. If K(> 1) is assigned, steps 2 to 8 mentioned above should be repeated until completing the specified number of repeated tapping; otherwise the program terminates;
17. In G91 mode, argument R is for assigning the distance between R point and the reference point; argument Z is for assigning the distance between the hole base position and R point; when K is being assigned (> 1), after completion of each tapping operation (steps 2 to 8 mentioned above), a hole position incremental offset will be implemented in accordance with the assigned X, Y before continuing with the next tapping operation.
18. Under G94 mode, cutting feed rate F is rotation speed (S) × thread pitch (PITCH).

Plot legends:

Parameter 160041 is for setting high speed peck tapping mode

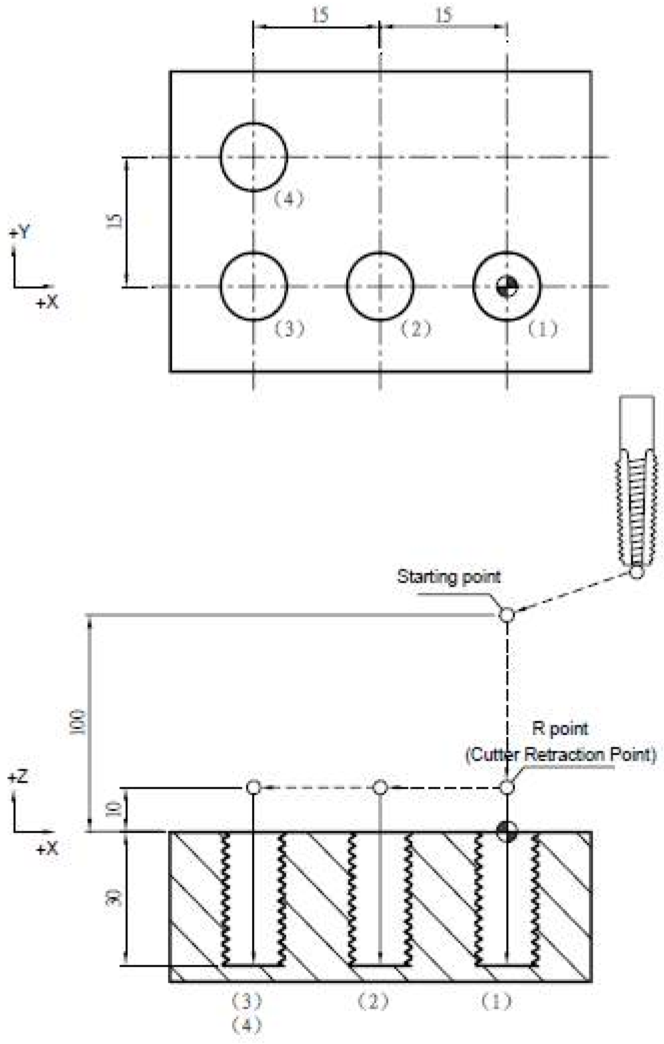


Parameter 160041 is for setting general peck tapping mode

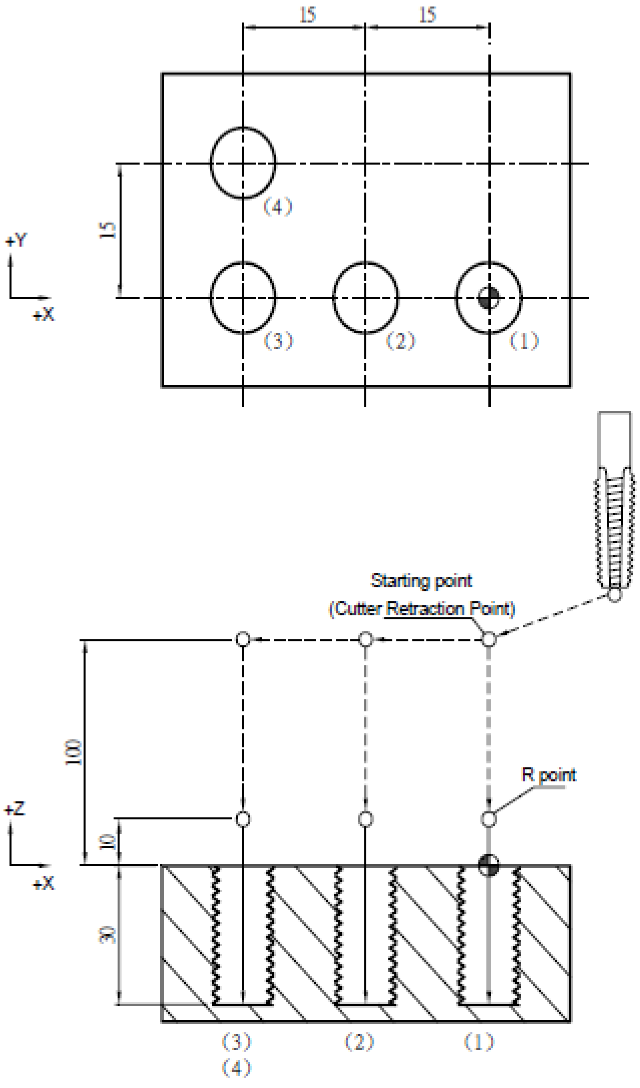


Program example:

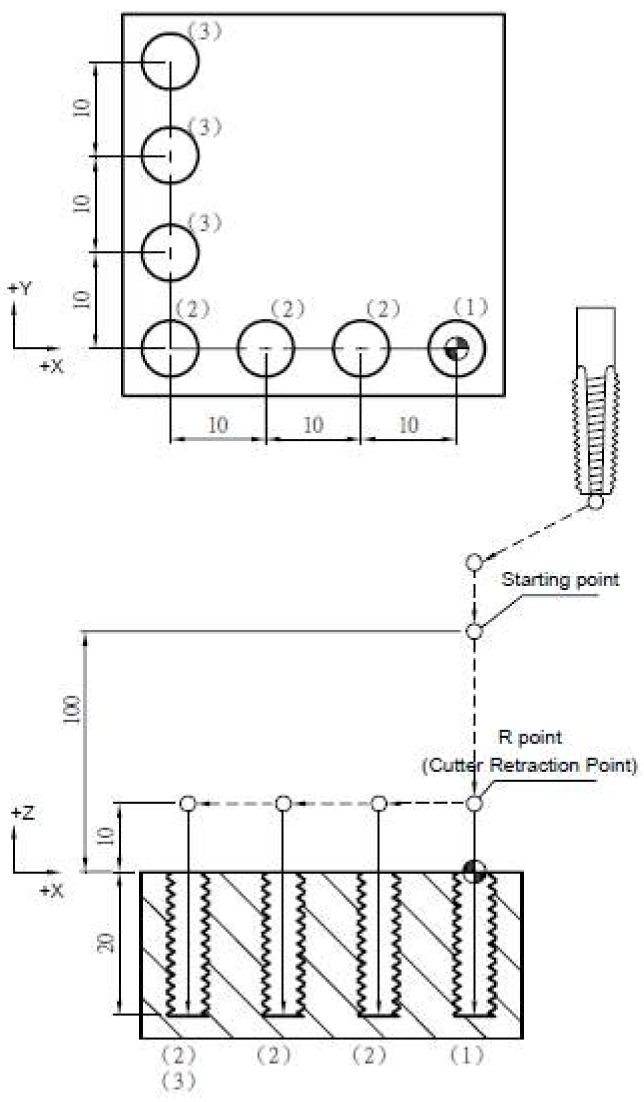
```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G99 G74 X0. Y0. Z-30. R10. P1000 K1 F1000.; ---------------------------(1)
X-15.; ---------------------------------------------------------------(2)
X-30.; ---------------------------------------------------------------(3)
X-30. Y15.;-----------------------------------------------------------(4)
M28;
G91 G80 G28 X0. Y0. Z0.;
```



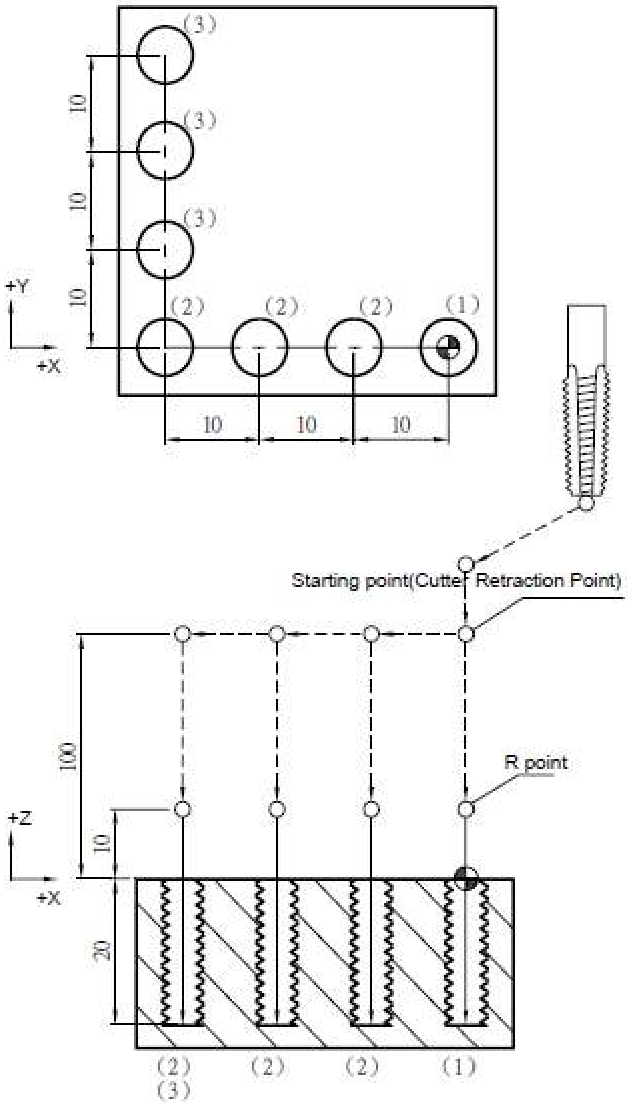
```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G98 G74 X0. Y0. Z-30. R10. P1000 K1 F1000.; --------------------------------------------------(1)
X-15.; ---------------------------------------------------------------------------------------(2)
X-30.; ---------------------------------------------------------------------------------------(3)
X-30. Y15.;-----------------------------------------------------------------------------------(4)
M28;
G91 G80 G28 X0. Y0. Z0.;
```



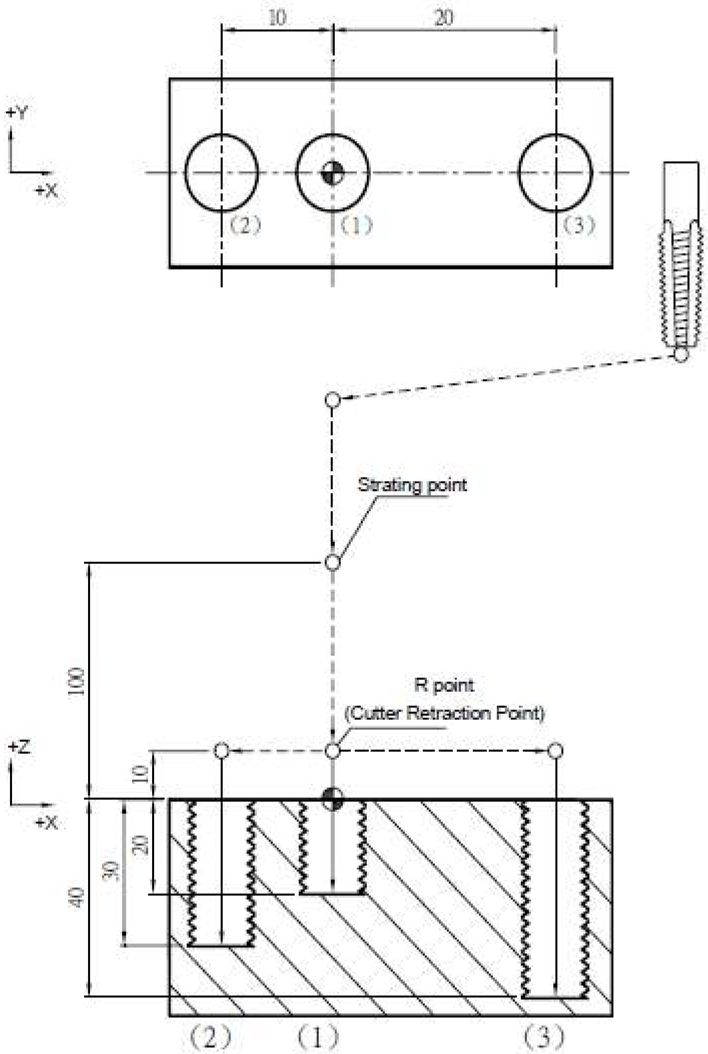
```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G99 G74 X0. Y0. Z-20. R10. P1000 K1 F1000.; -----------------------------------------------(1)
G91 X-10. K3; ------------------------------------------------------------------------------(2)
Y10. K3; -----------------------------------------------------------------------------------(3)
M28;
G91 G80 G28 X0. Y0. Z0.;
```



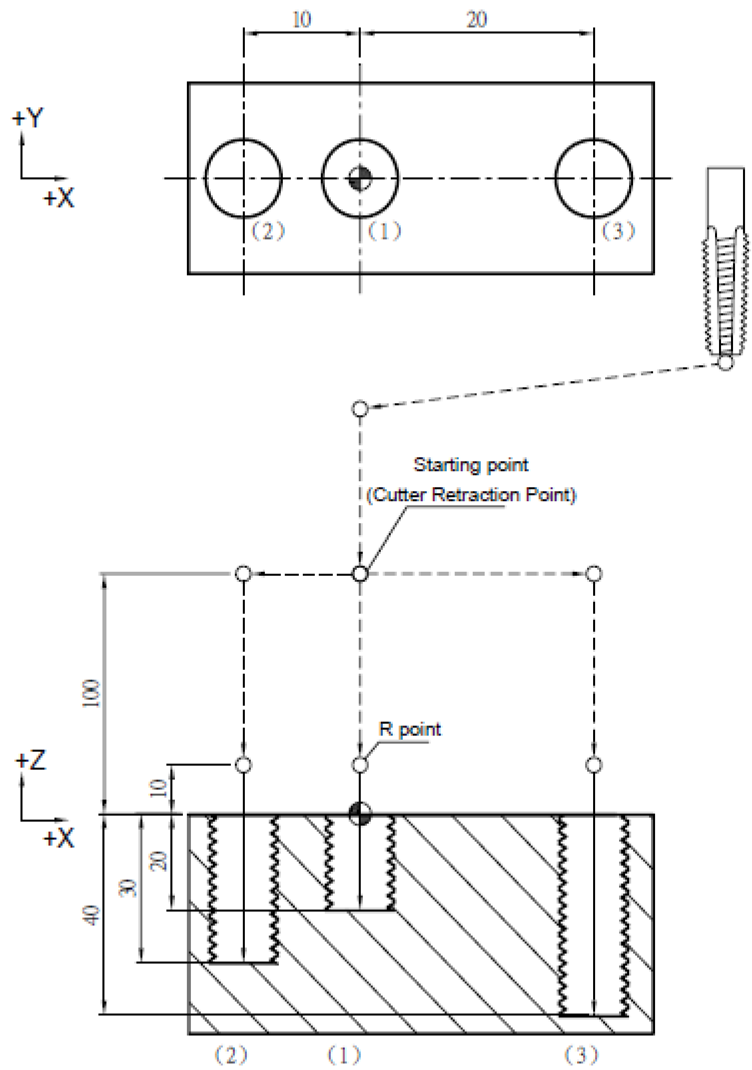
```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G98 G74 X0. Y0. Z-20. R10. P1000 K1 F1000.; --------------------------------------------------------------------------(1)
G91 X-10. K3 ; ----------------------------------------------------------------------------------------------------------------(2)
Y10. K3 ;---------------------------------------------------------------------------------------------------------------------------(3)
M28;
G91 G80 G28 X0. Y0. Z0.;
```



```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G99 G74 X0. Y0. Z-20. R10. P1000 K1 F1000.; ----------------------------------------------------(1)
X-10. Z-30.; ------------------------------------------------------------------------------------(2)
X20. Z-40.; -------------------------------------------------------------------------------------(3)
M28;
G91 G80 G28 X0. Y0. Z0.;
```



```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G98 G74 X0. Y0. Z-20. R10. P1000 K1 F1000.; ------------------------------------------(1)
X-10. Z-30.; ---------------------------------------------------------------------------(2)
X20. Z-40.; ----------------------------------------------------------------------------(3)
M28;
G91 G80 G28 X0. Y0 Z0.;
```



G76 precision boring cycle

Command format:

```
G76 X__ Y__ Z__ R__ P__ Q__ K__ F__;
```

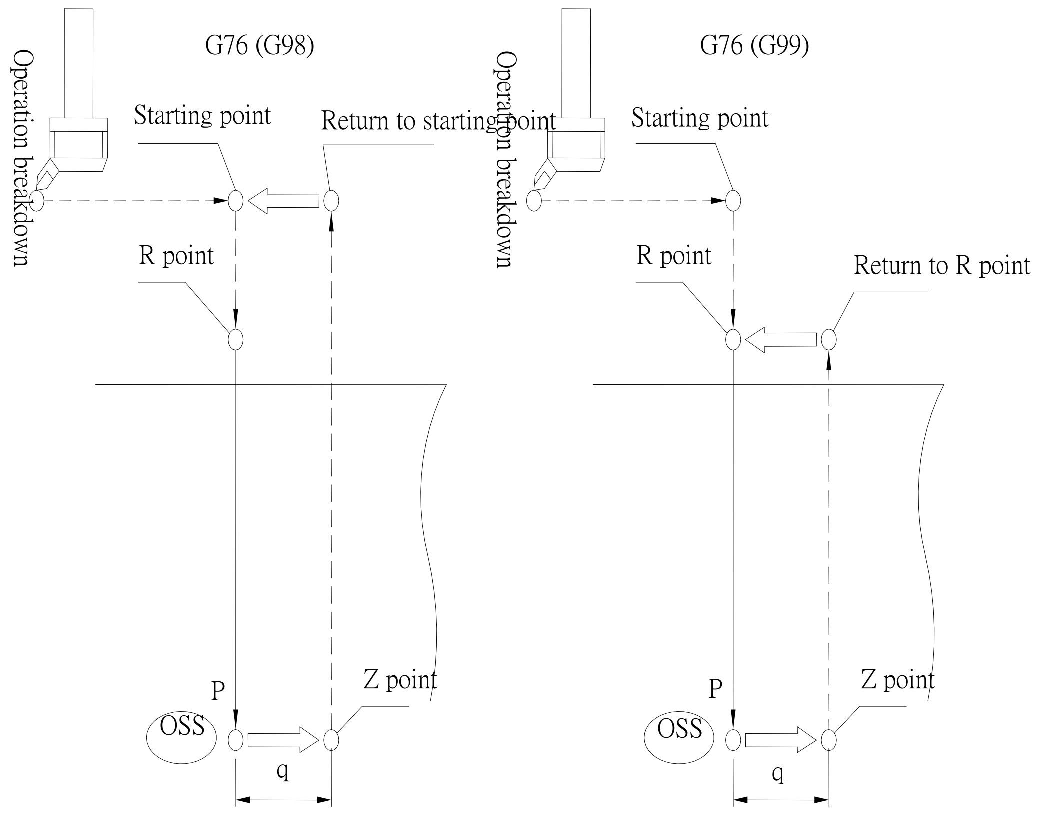
Argument description:

X__Y__	:	Coordinates of the hole position (mm).
Z__	:	Coordinates of the hole base (mm).
R__	:	Coordinate value of R point (which is the reset point) (mm).
Q__	:	Amount of offset at hole base (mm), with offset direction set by system parameter 170003.
K__	:	Number of repetitions.
F__	:	Feed rate (G94 mm/min).

Operation description (Using the G17 plane as an example):

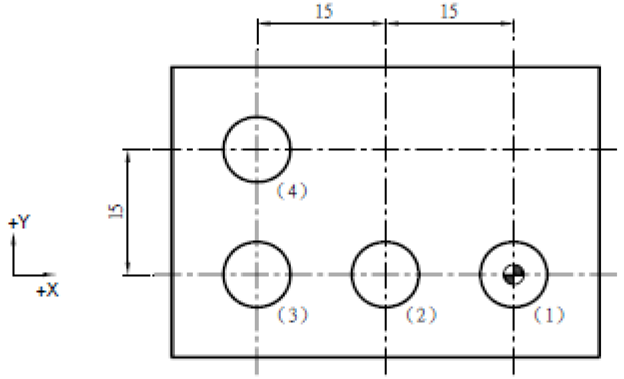
1. It is rapidly positioned to reach the hole location (X, Y, while maintaining the original tool height);
2. It is rapidly positioned to reach the R point coordinate (R);
3. Cutting should reach the hole base position based on a set cutting feed rate and spindle rotation speed (Z);
4. If P is assigned, it is the pause period at the hole base position;
5. Spindle stops, executes the positioning of M19;
6. Tool offset, with offset distance set by argument Q, and offset direction set by system parameter 160010;
7. Under G98 mode, it is rapidly retracted to the reference point; under G99 mode, it is rapidly retracted to the coordinates of R point;
8. Tool offset, which will return to the original hole coordinate position (by performing steps which are opposite of step 6 mentioned above);
9. Cancel spindle positioning status, spindle rotates;
10. When K is assigned (> 1), steps 2 to 9 mentioned above should be repeated until completing the specified number of repeated boring; otherwise the program terminates;
11. In G91 mode, argument R is for assigning the distance between R point and the reference point; argument Z is for assigning the distance between the hole base position and R point; when K is being assigned (> 1), after completion of each boring operation (steps 2 to 9 mentioned above), a hole position incremental offset will be implemented in accordance with the assigned X, Y before continuing with the next boring operation.

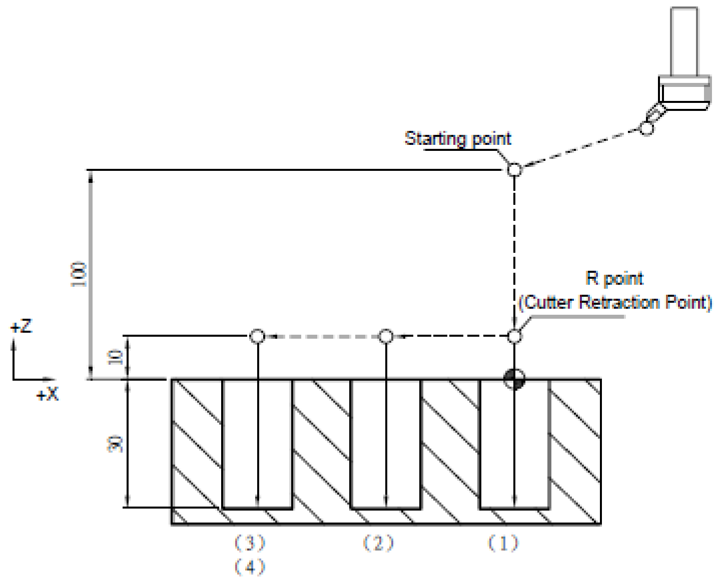
Plot legends:



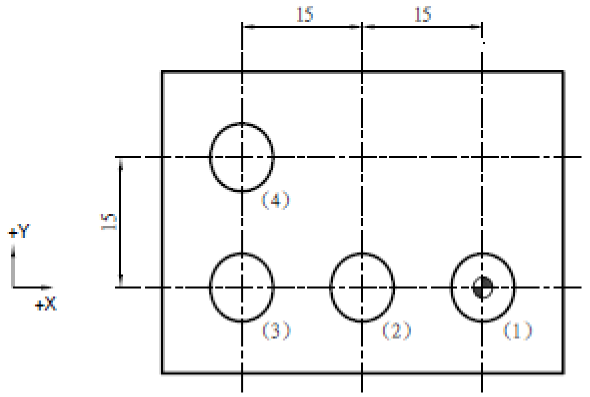
Program example:

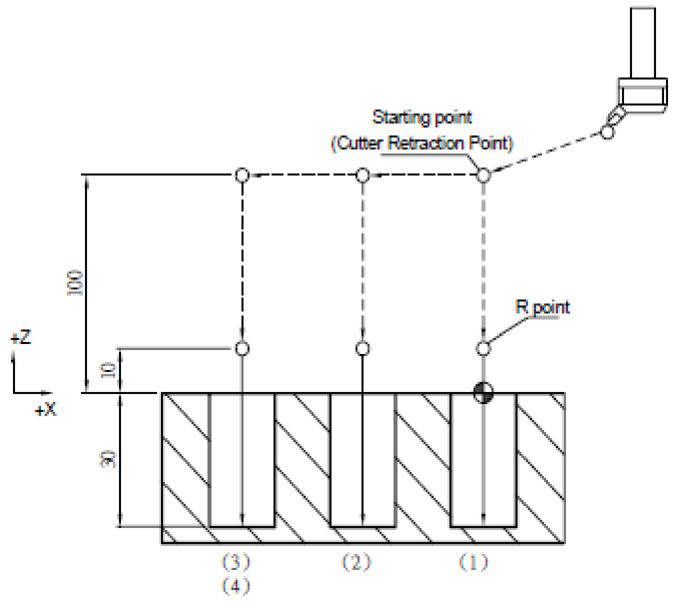
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G76 X0. Y0. Z-30. R10. P1000 Q5. K1 F100.; -----------------------------------------------------------------------(1)
X-15.; ---------------------------------------------------------------------------------------------------------------------------(2)
X-30.; ---------------------------------------------------------------------------------------------------------------------------(3)
X-30. Y15.;------------------------------------------------------------------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



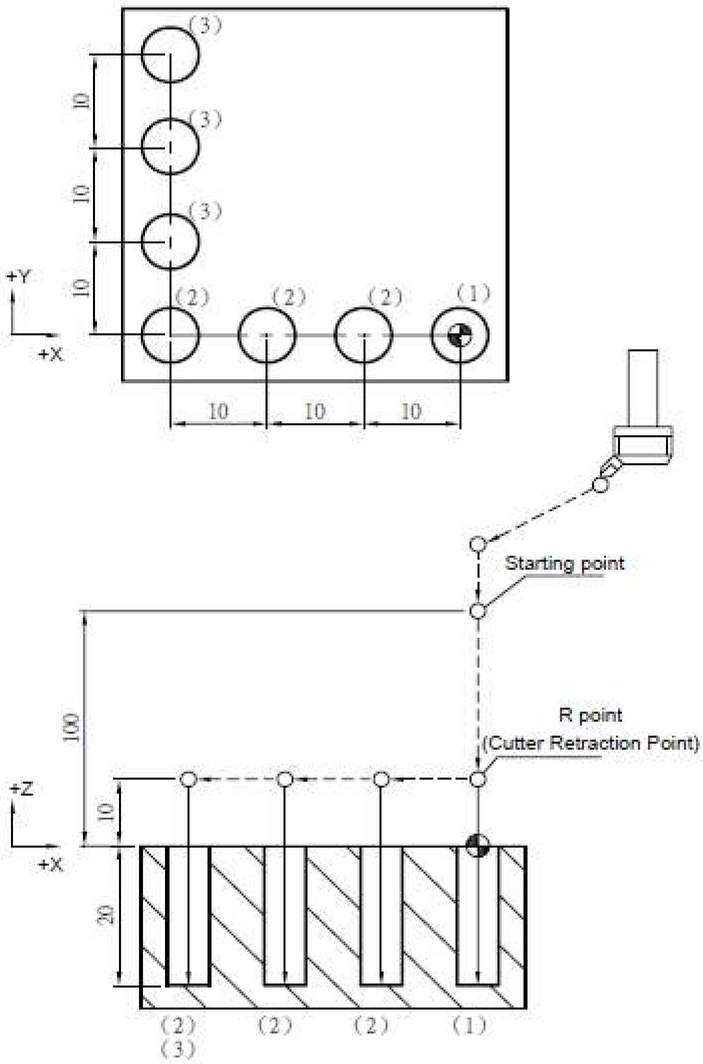


```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.
G98 G76 X0. Y0. Z-30. R10. P1000 Q5. K1 F100.; --------------------(1)
X-15.; ------------------------------------------------------------(2)
X-30.; ------------------------------------------------------------(3)
X-30. Y15.; -------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```

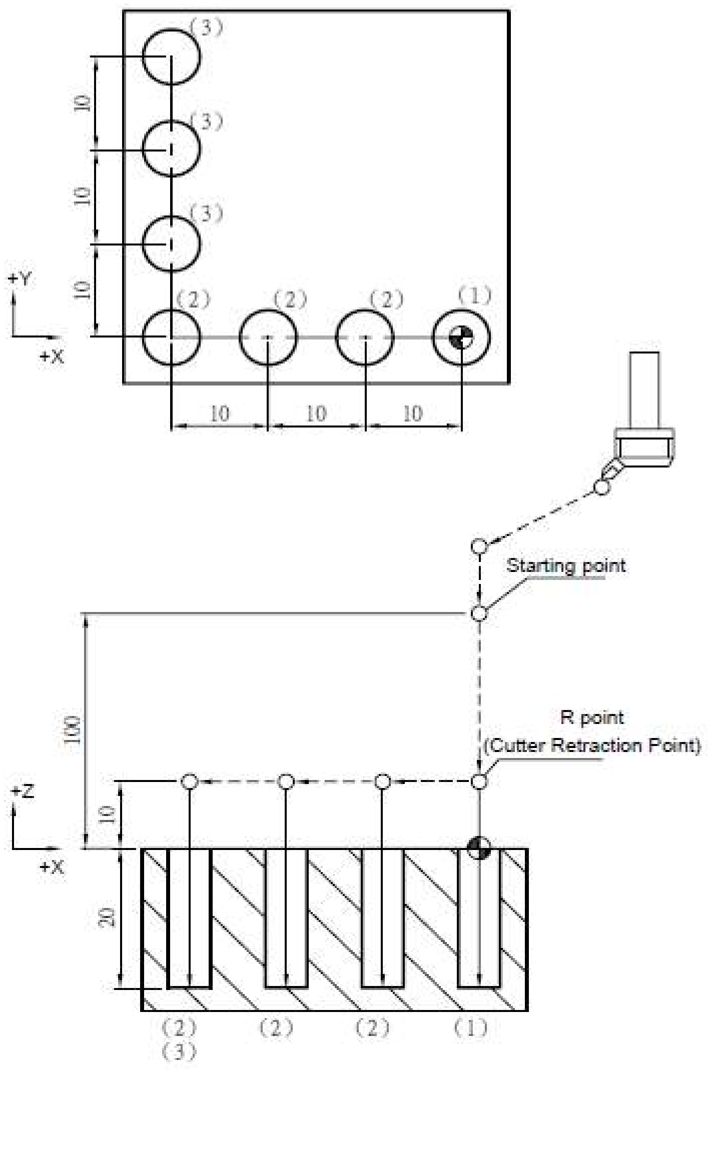




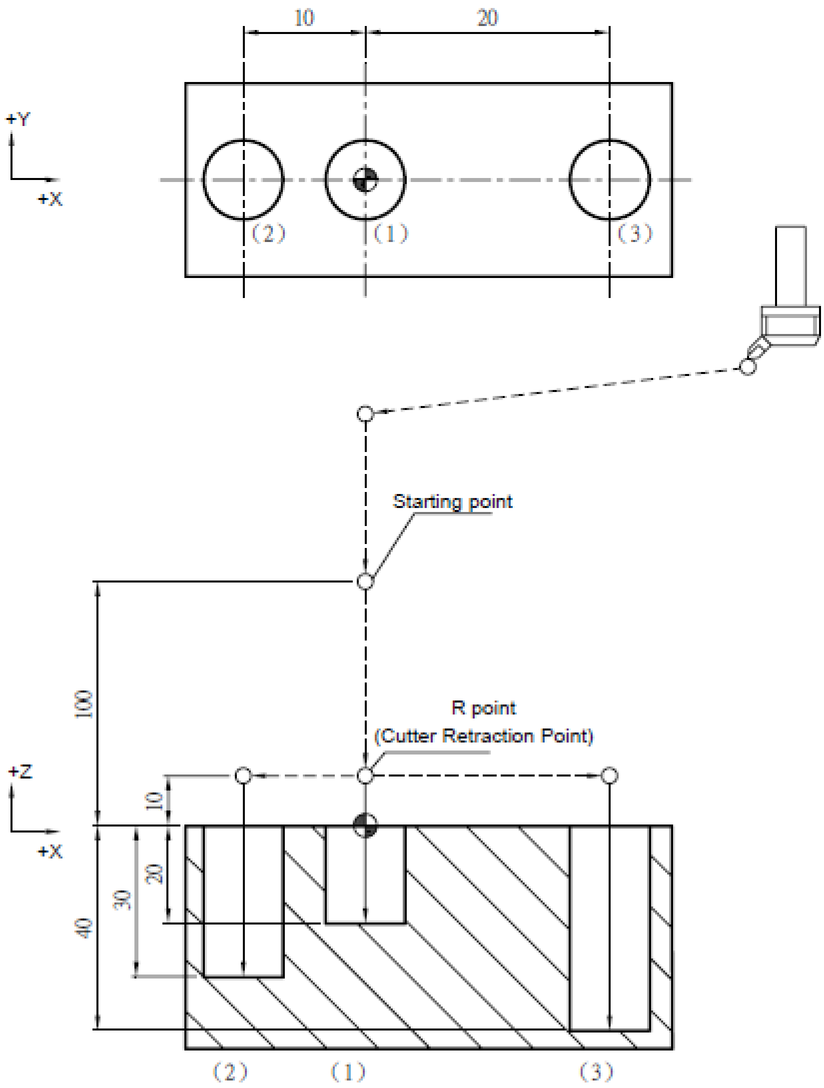
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G76 X0. Y0. Z-20. R10. P1000 Q5. K1 F100.; ----------------------------------------------------------------------(1)
G91 X-10. K3; ------------------------------------------------------------------------------------------------------------(2)
Y10. K3; -----------------------------------------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



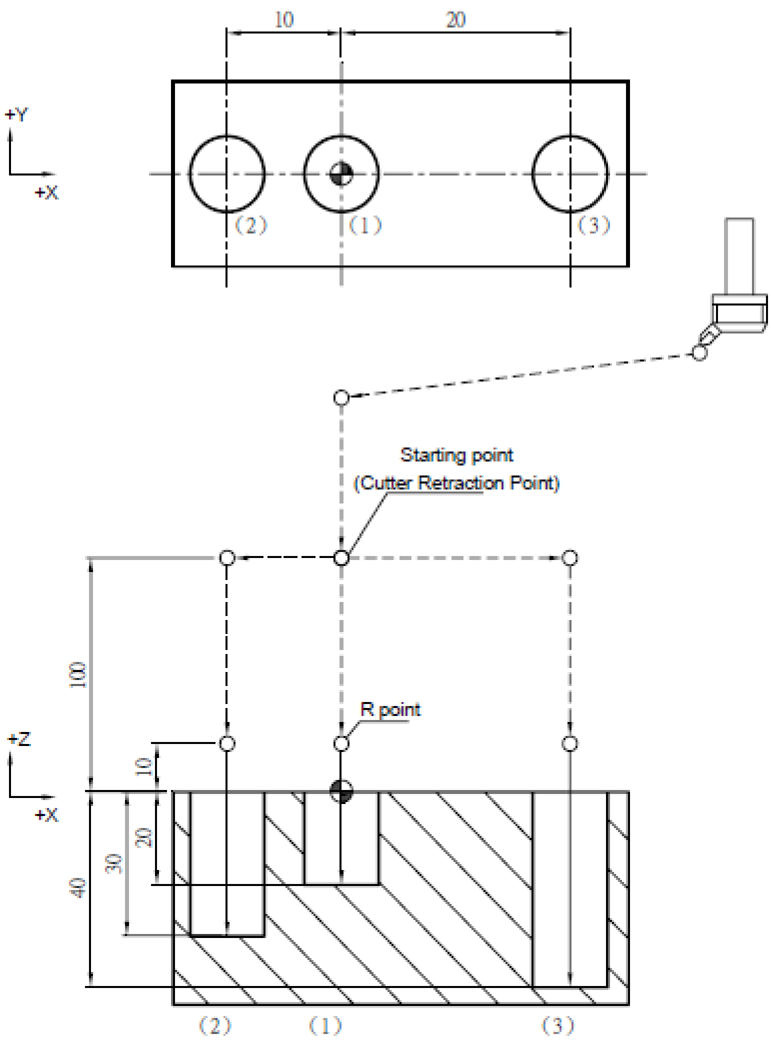
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G76 X0. Y0. Z-20. R10. P1000 Q5. K1 F100.; -------------------------------------------------------------------------(1)
G91 X-10. K3; -------------------------------------------------------------------------------------------------------------(2)
Y10. K3; ------------------------------------------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G76 X0. Y0. Z-20. R10. P1000 Q5. K1 F100.; ----------------------------------------(1)
X-10. Z-30.; ----------------------------------------------------------------------------(2)
X20. Z-40.; -----------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G76 X0. Y0. Z-20. R10. P1000 Q5. K1 F100.; ----------------------------------------------------------------------(1)
X-10. Z-30.;-------------------------------------------------------------------------------------------------------------(2)
X20. Z-40.;--------------------------------------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



G80 cancel fixed cycle cutting mode (canned cycle)

Command format:

G80;

Argument description:

It can be used for canceling fixed cycle cutting modes G73, G74, G76, G81 to G89 (canned cycle).

In addition to G80, traverse commands G00, G01, G02, G03 can also be used directly to cancel the aforementioned fixed cycle cutting mode (canned cycle).

Program example:

```
G17 G90 G00 G54 X0. Y0.;
Z100.;
G99 G73 X0. Y0. Z-20. R10. Q4. K1 F100.;
G80; ------------------------------------------------------------------------------------------------------------(Cancel G73 cycle)

G17 G90 G00 G54 X0. Y0.;
Z100.;
G99 G73 X0. Y0. Z-20. R10. Q4. K1 F100.;
G00 Z100.;------------------------------------------------------------------------------------------------------(Cancel G73 cycle)
```

G81 Drilling cycle

Command format:

G81 X__ Y__ Z__ R__ K__ F__;

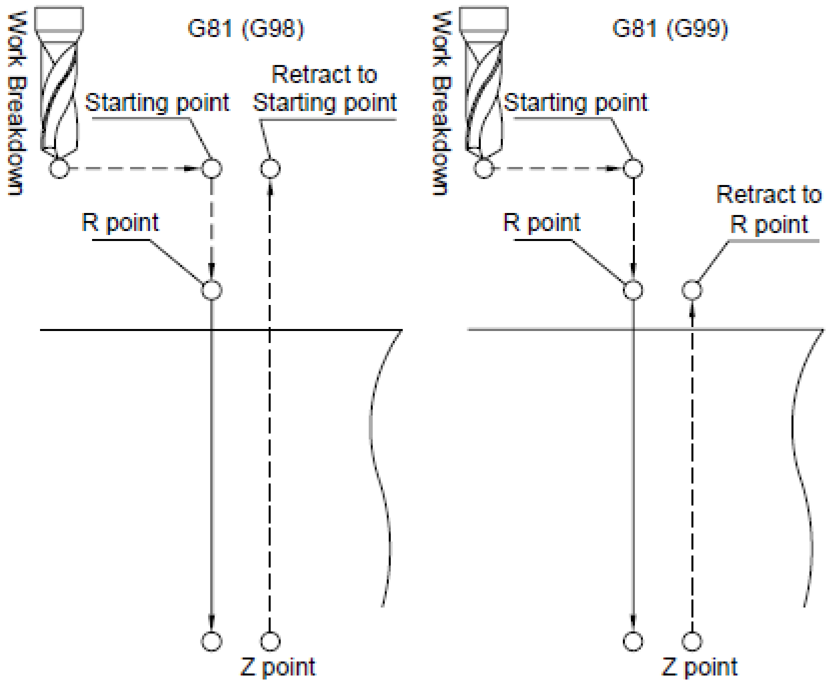
Argument description:

X__Y__	:	Coordinates of the hole position (mm).
Z__	:	Coordinates of the hole base (mm).
R__	:	Coordinate value of R point (which is the reset point) (mm).
K__	:	Number of repetitions.
F__	:	Feed rate (G94 mm/min).

Operation description (using the G17 plane as an example):

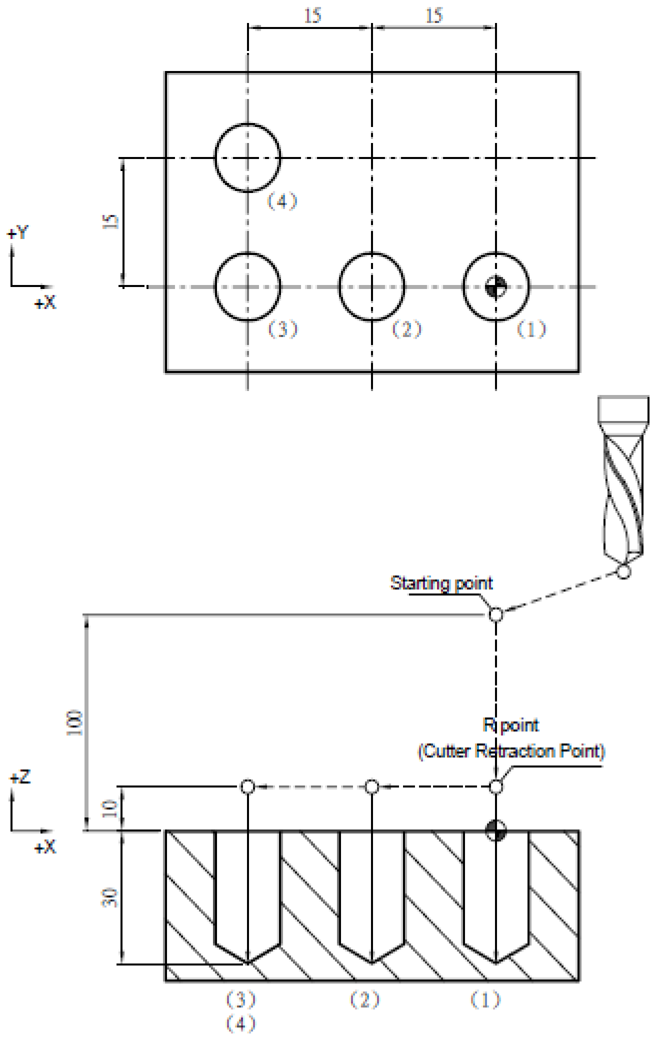
1. It is rapidly positioned to reach the hole location (X, Y, while maintaining the original tool height);
2. It is rapidly positioned to reach the R point coordinate (R);
3. Cutting should reach the hole base position based on a set cutting feed rate and spindle rotation speed (Z);
4. Under G98 mode, it is rapidly retracted to the reference point; under G99 mode, it is rapidly retracted to the R point;
5. If K (> 1) is assigned, steps 2 to 4 mentioned above should be repeated until completing the specified number of repeated drilling; otherwise the program terminates;
6. In G91 mode, argument R is for assigning the distance between R point and the reference point; argument Z is for assigning the distance between the hole base position and R point; If K is assigned (> 1), after completion of each drilling operation (steps 2 to 5 mentioned above), a hole position offset will be implemented in accordance with the assigned X, Y before continuing with the next drilling operation.
7. The difference between G81 and G82 is that the latter can be used to assign the pause time at hole base.

Plot legends:

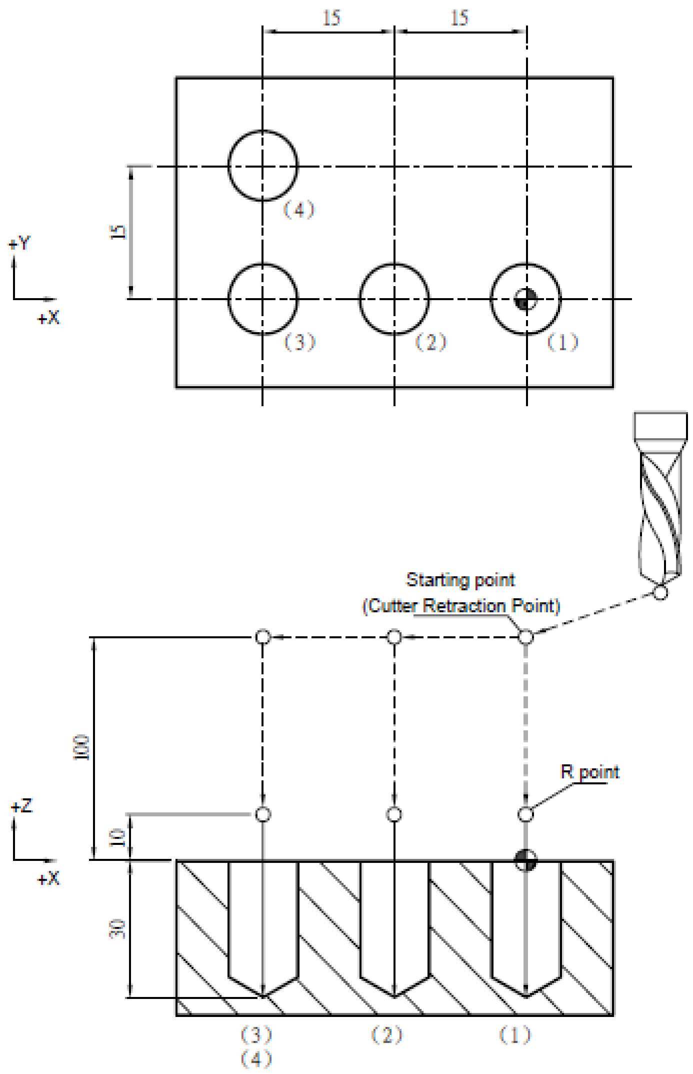


Program example:

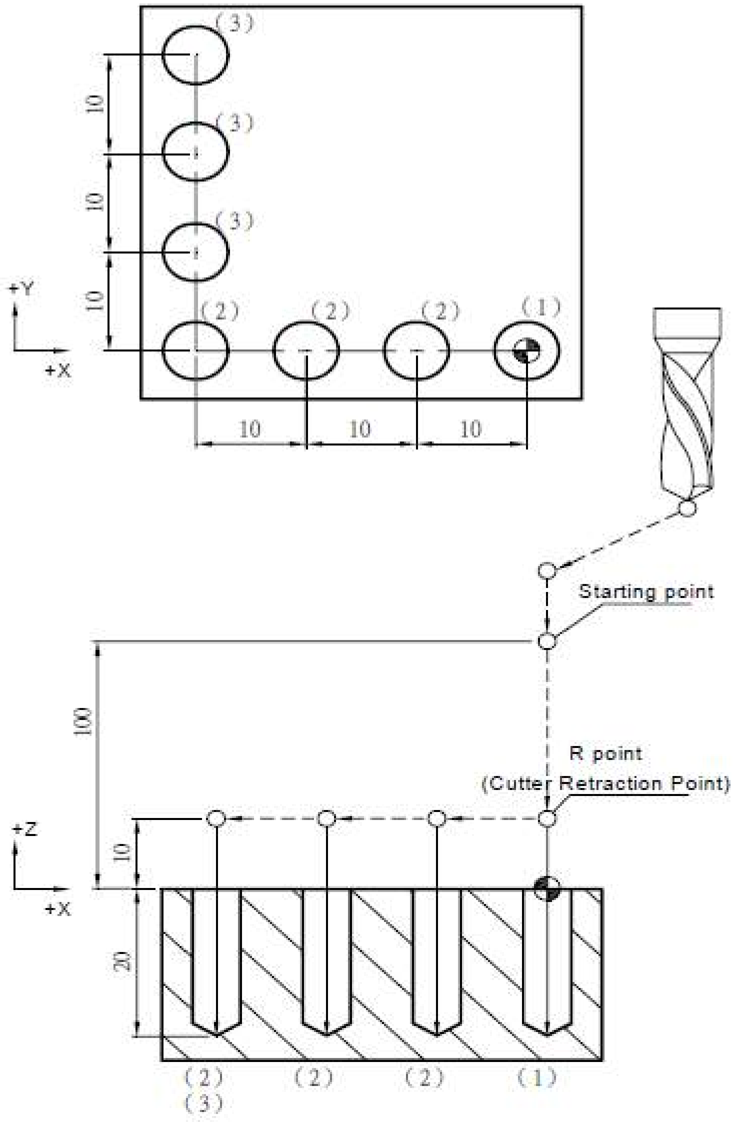
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G81 X0. Y0. Z-30. R10. K1 F100.; --------------------------------------------------(1)
X-15.; -------------------------------------------------------------------------------(2)
X-30.; -------------------------------------------------------------------------------(3)
X-30. Y15.; --------------------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



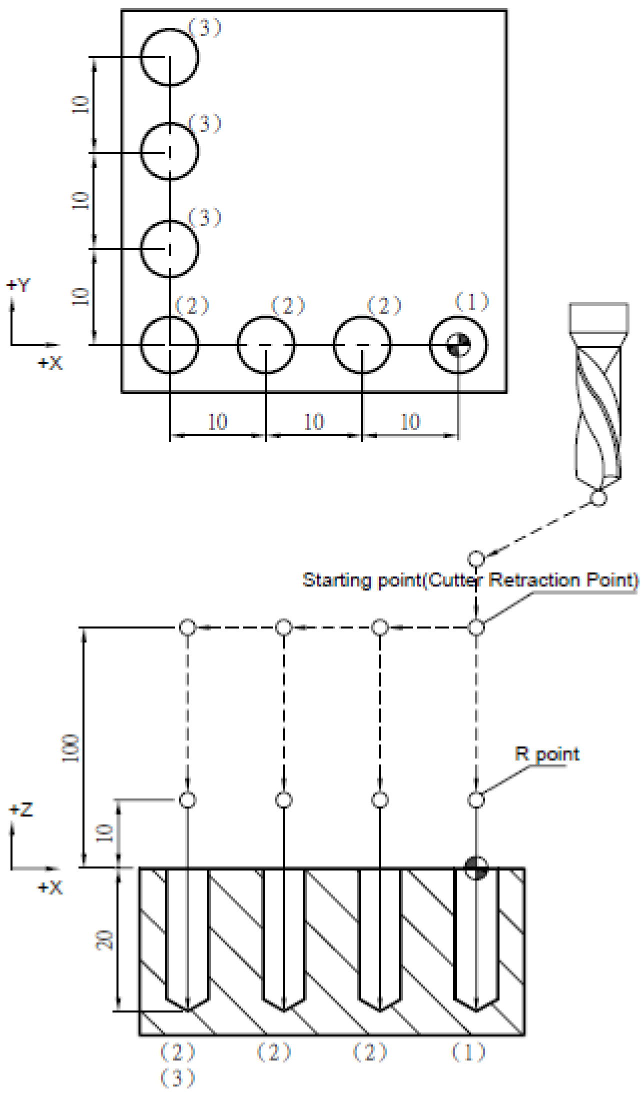
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.
G98 G81 X0. Y0. Z-30. R10. K1 F100.; ------------------------------------------(1)
X-15.; --------------------------------------------------------------------------(2)
X-30.; --------------------------------------------------------------------------(3)
X-30. Y15.;----------------------------------------------------------------------(4)
G91 G80 G28 X0. Y0. Z0.;
M05;
```



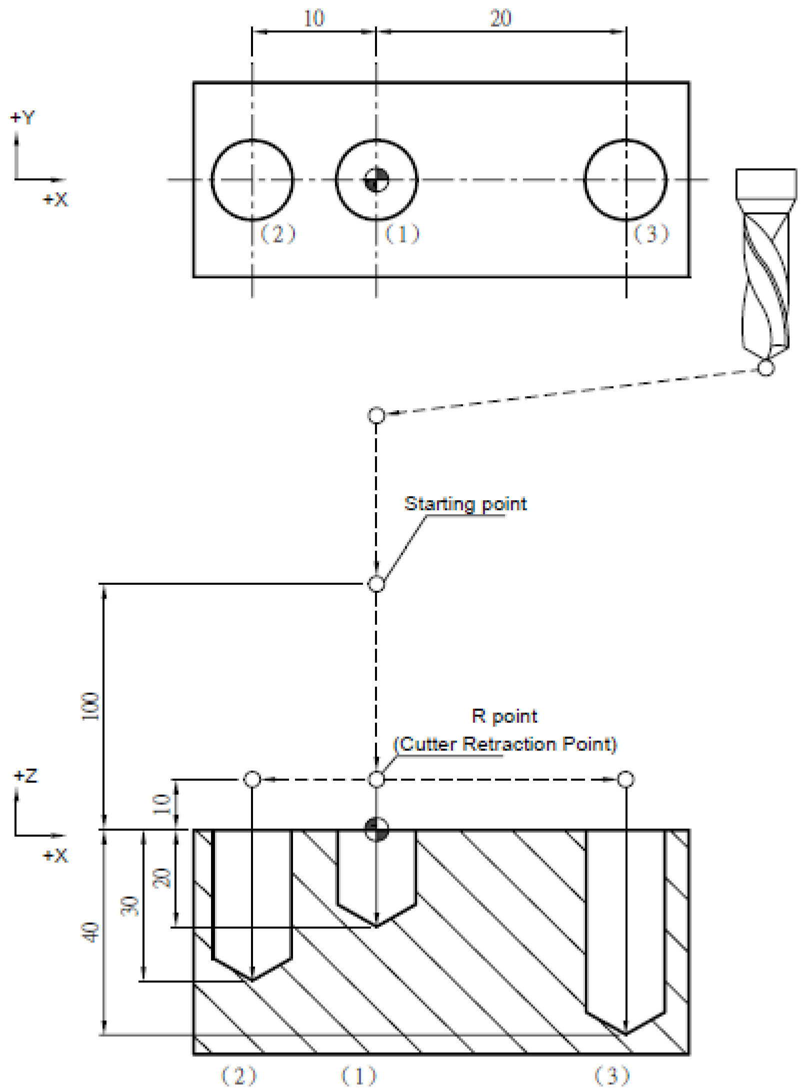
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G81 X0. Y0. Z-20. R10. K1 F100.; ---------------------------------------------------------------------------------------(1)
G91 X-10. K3; --------------------------------------------------------------------------------------------------------------------(2)
Y10. K3; ------------------------------------------------------------------------------------------------------------------------(3)
G91 G80 G28 X0. Y0. Z0.;
M05;
```



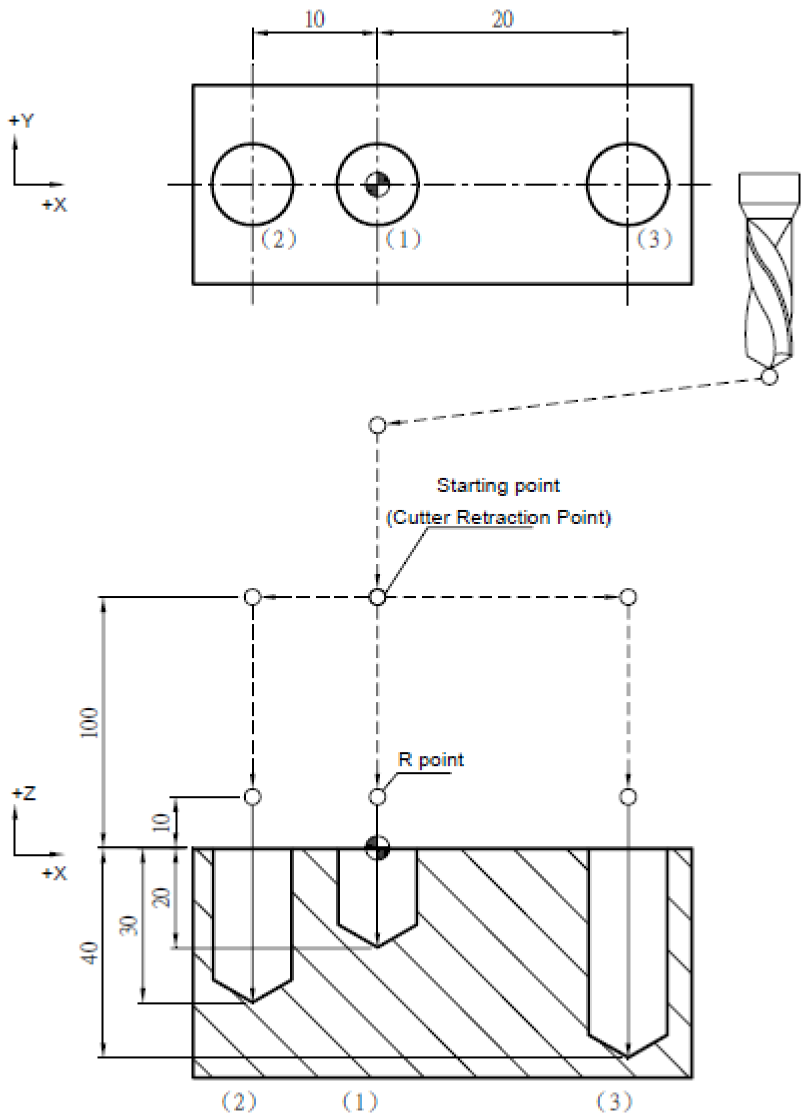
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G81 X0. Y0. Z-20. R10. K1 F100.; ----------------------------------------(1)
G91 X-10. K3; -----------------------------------------------------------------(2)
Y10. K3; -----------------------------------------------------------------------(3)
G91 G80 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G81 X0. Y0. Z-20. R10. K1 F100.; -----------------------------------------------------(1)
X-10. Z-30.; -----------------------------------------------------------------------------(2)
X20. Z-40.; ------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G81 X0. Y0. Z-20. R10. K1 F100.; --------------------------------------------------(1)
X-10. Z-30.; --------------------------------------------------------------------------(2)
X20. Z-40.; ---------------------------------------------------------------------------(3)
G91 G80 G28 X0. Y0. Z0.;
M05;
```



G82 Drilling cycle

Command format:

```
G82 X__ Y__ Z__ R__ P__ K__ F__;
```

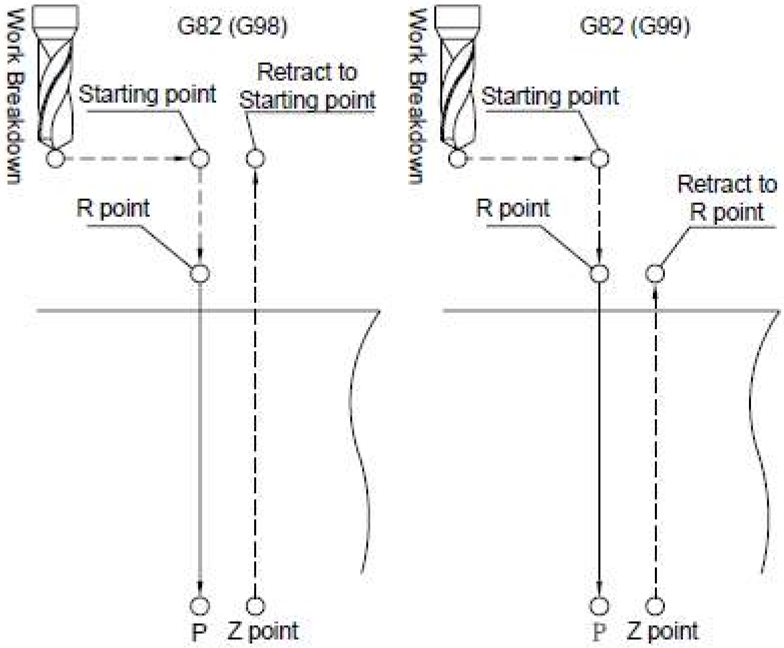
Argument description:

X__Y__	:	Coordinates of the hole position (mm).
Z__	:	Coordinates of the hole base (mm).
R__	:	Coordinate value of R point (which is the reset point) (mm).
P__	:	Pause time at hole base (1/1000 second), minimum unit, no decimal point allowed.
K__	:	Number of repetitions.
F__	:	Cutting feed rate (G94 mm/min).

Operation description (Using the G17 plane as an example):

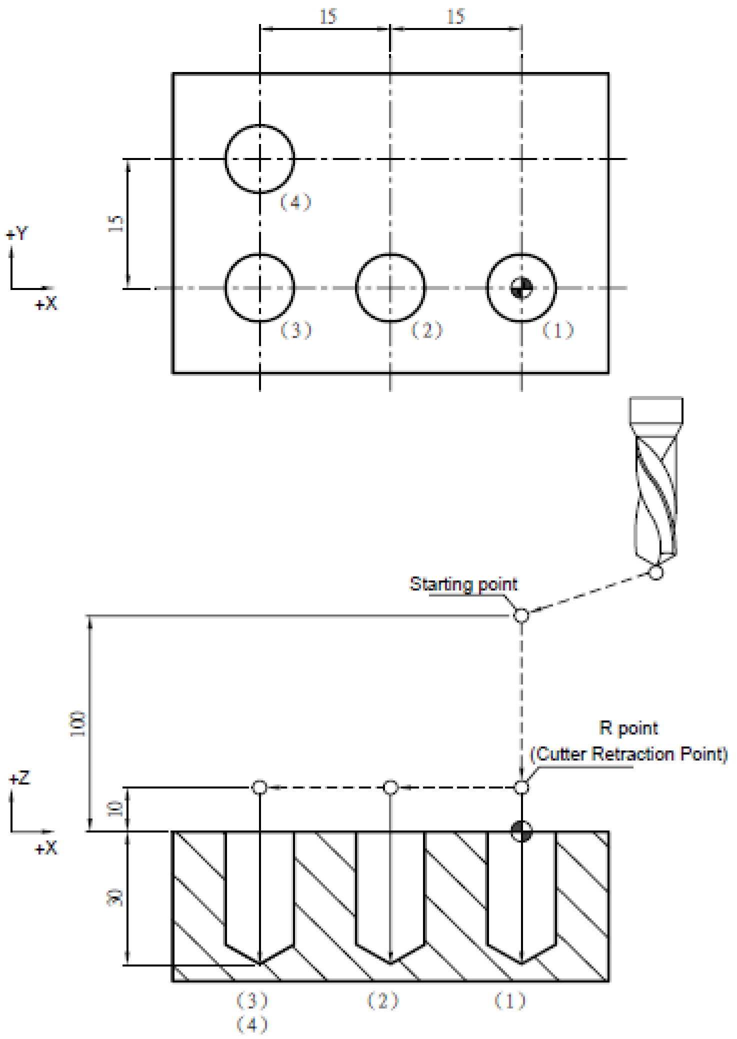
1. It is rapidly positioned to reach the hole location (X, Y, while maintaining the original tool height);
2. It is rapidly positioned to reach the R point coordinate (R);
3. Cutting should reach the hole base position based on a set cutting feed rate and spindle rotation speed (Z);
4. If P is assigned, it is the pause period at the hole base position;
5. Under G98 mode, it is rapidly retracted to the reference point; under G99 mode, it is rapidly retracted to the R point;
6. If K (> 1) is assigned, steps 2 to 5 mentioned above should be repeated until completing the specified number of repeated drilling; otherwise the program terminates;
7. In G91 mode, argument R is for assigning the distance between R point and the reference point; argument Z is for assigning the distance between the hole base position and R point; If K is assigned (> 1), after executing a drilling operation (steps 2 to 5 mentioned above), a hole position offset will be implemented in accordance with the assigned X, Y before continuing with the next drilling operation.
8. The difference between G81 and G82 is that the latter can be used to assign the pause time at hole base.

Plot legends:

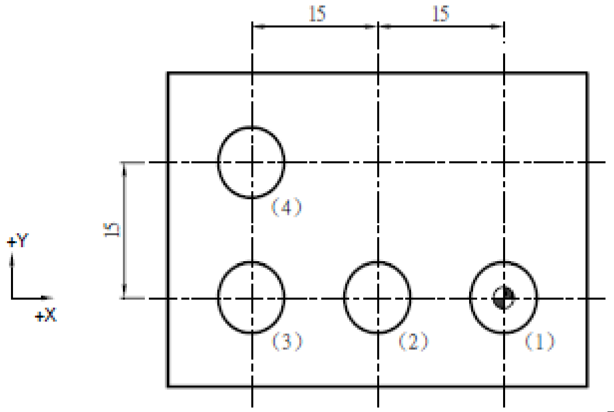


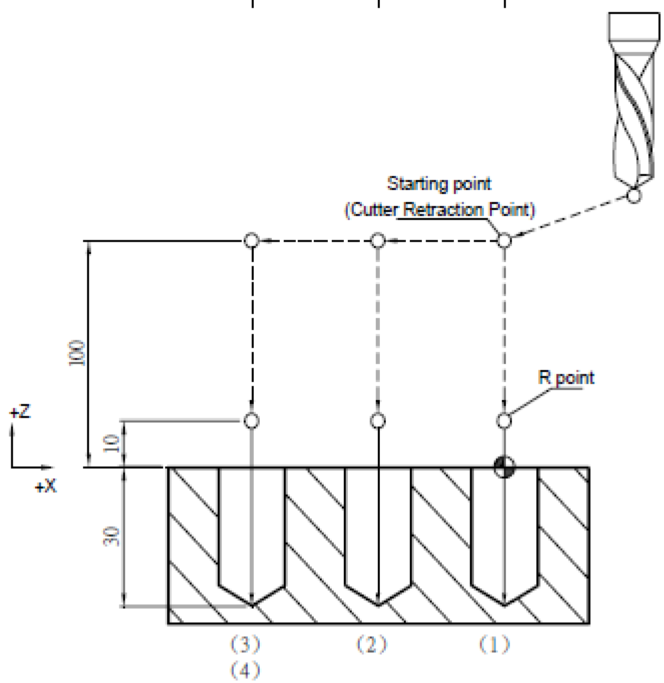
Program example:

```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G82 X0. Y0. Z-30. R10. P1000 K1 F100.;-----------------------------------(1)
X-15.; ------------------------------------------------------------------------(2)
X-30.; ------------------------------------------------------------------------(3)
X-30. Y15.;--------------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```

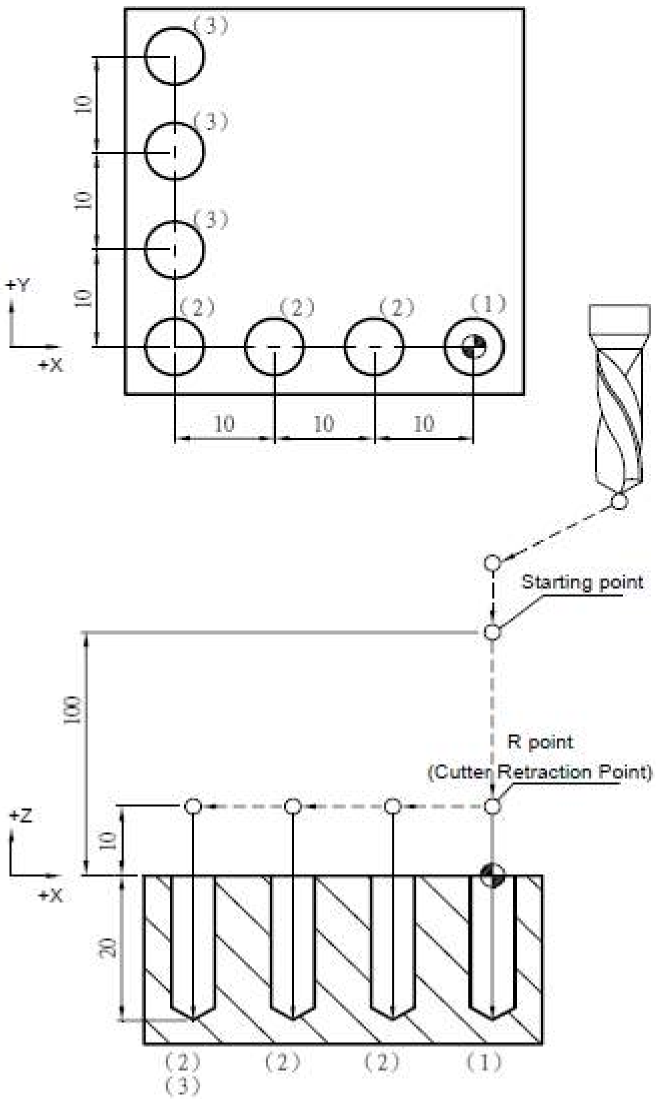


```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G82 X0. Y0. Z-30. R10. P1000 K1 F100.;-----------------------------------------------(1)
X-15.; ------------------------------------------------------------------------------------(2)
X-30.; ------------------------------------------------------------------------------------(3)
X-30. Y15.;--------------------------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```

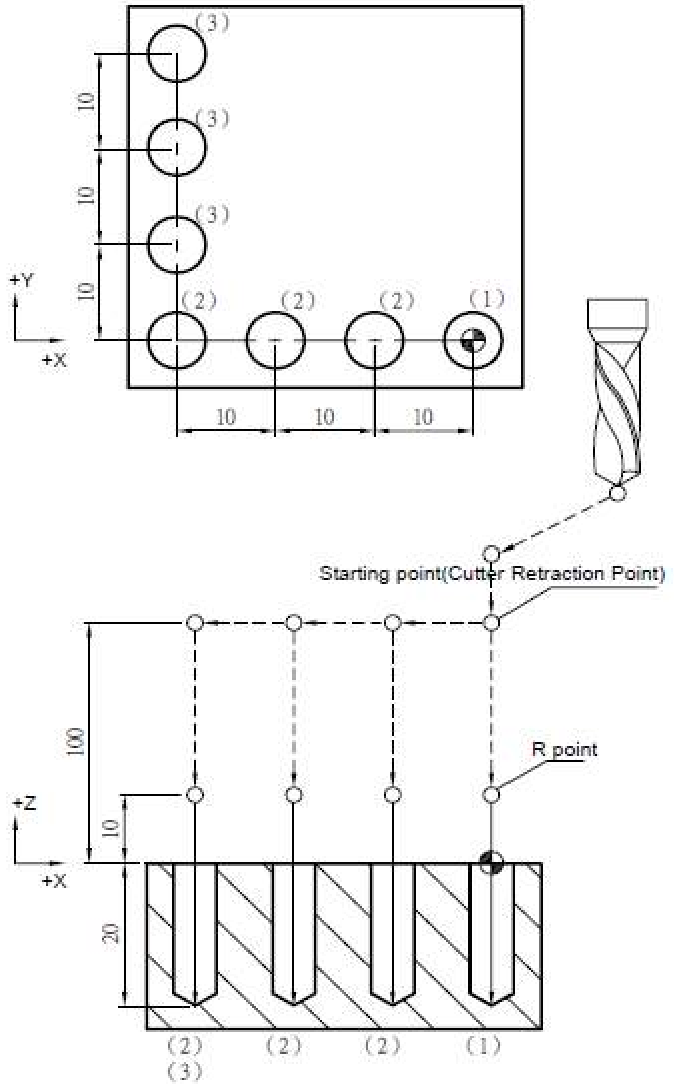




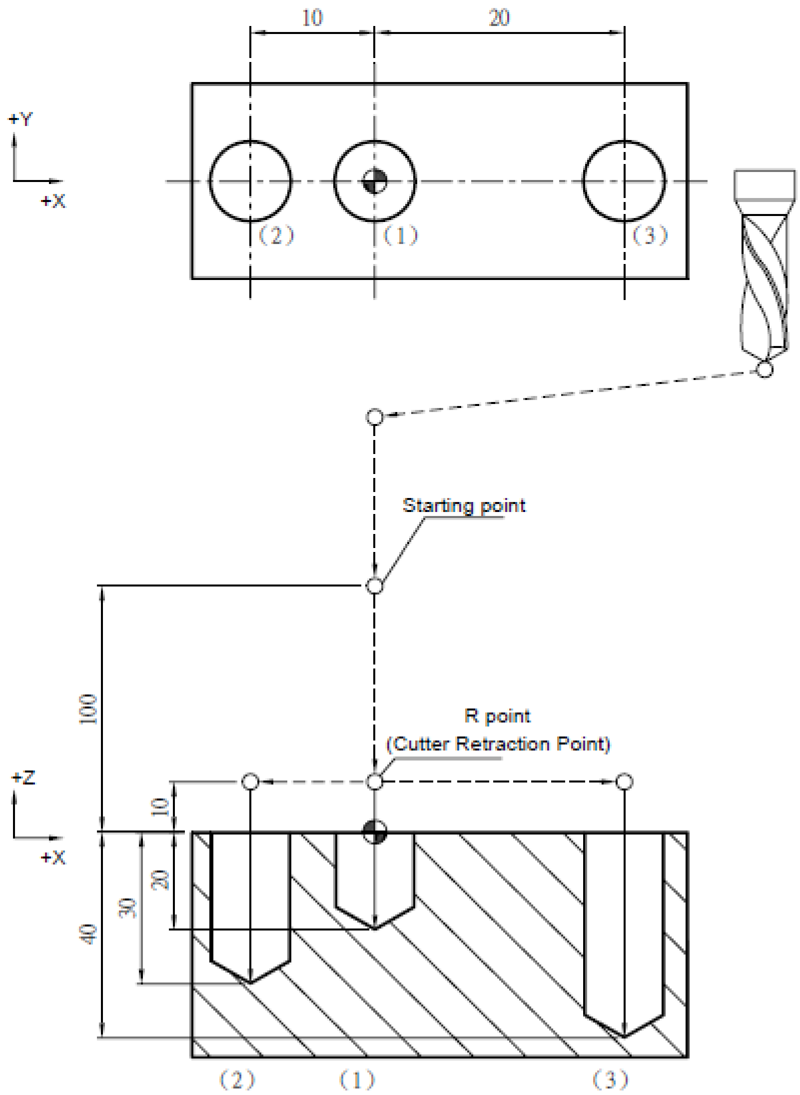
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G82 X0. Y0. Z-20. R10. P1000 K1 F100.;---------------------------------------------------------------------------(1)
G91 X-5. K3;-------------------------------------------------------------------------------------------------------------(2)
Y5. K3; -------------------------------------------------------------------------------------------------------------------(3)
G91 G80 G28 X0. Y0. Z0.;
M05;
```



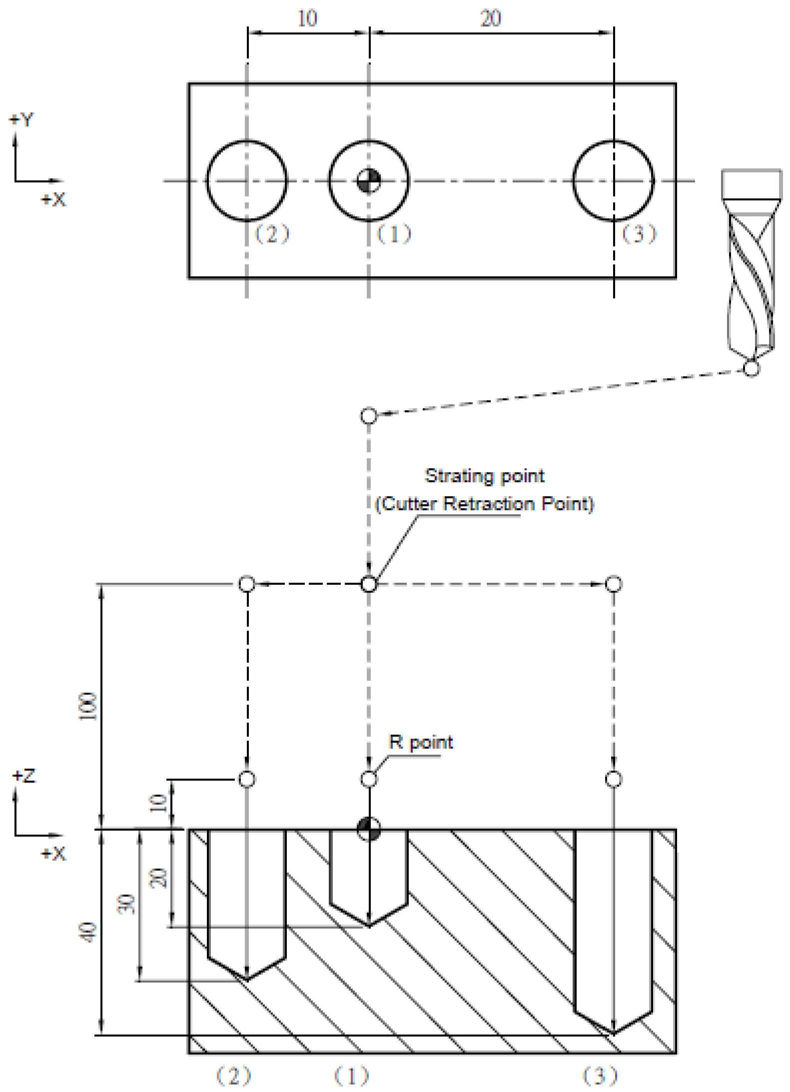
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G82 X0. Y0. Z-20. R10. P1000 K1 F100.;-----------------------------------(1)
G91 X-10. K3; ---------------------------------------------------------------(2)
Y10. K3; --------------------------------------------------------------------(3)
G91 G80 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G82 X0. Y0. Z-20. R10. P1000 K1 F100.;-----------------------------------------------(1)
X-10. Z-30.;-----------------------------------------------------------------------------(2)
X20. Z-40.;------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0;.
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G82 X0. Y0. Z-20. R10. P1000 K1 F100.;-----------------------------------------------(1)
X-10. Z-30.;-----------------------------------------------------------------------------(2)
X20. Z-40.;------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



G83 Peck drilling cycle

Command format:

G83 X__ Y__ Z__ R__ Q__ K__ F__;

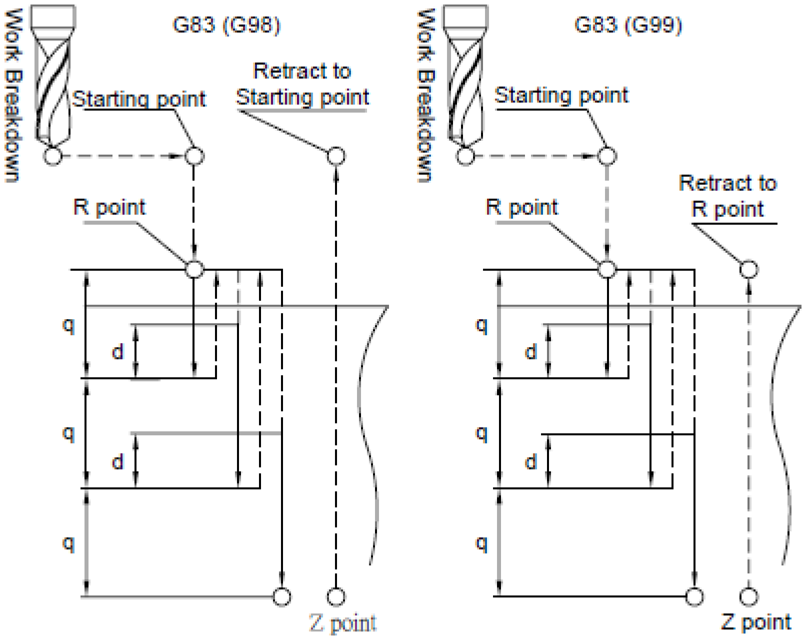
Argument description:

X__Y__	:	Coordinates of the hole position (mm).
Z__	:	Hole depth (mm).
R__	:	Coordinate value of R point (which is the reset point) (mm).
Q__	:	Feed amount per cut (mm). Q must be assigned as (> 0), or it will trigger system alert 【610010 G73/G83 command macro - the argument for cutting feed rate is not assigned (Q should be examined)】.
K__	:	Number of repetitions.
F__	:	Feed rate (G94 mm/min).

Operation description (using the G17 plane as an example):

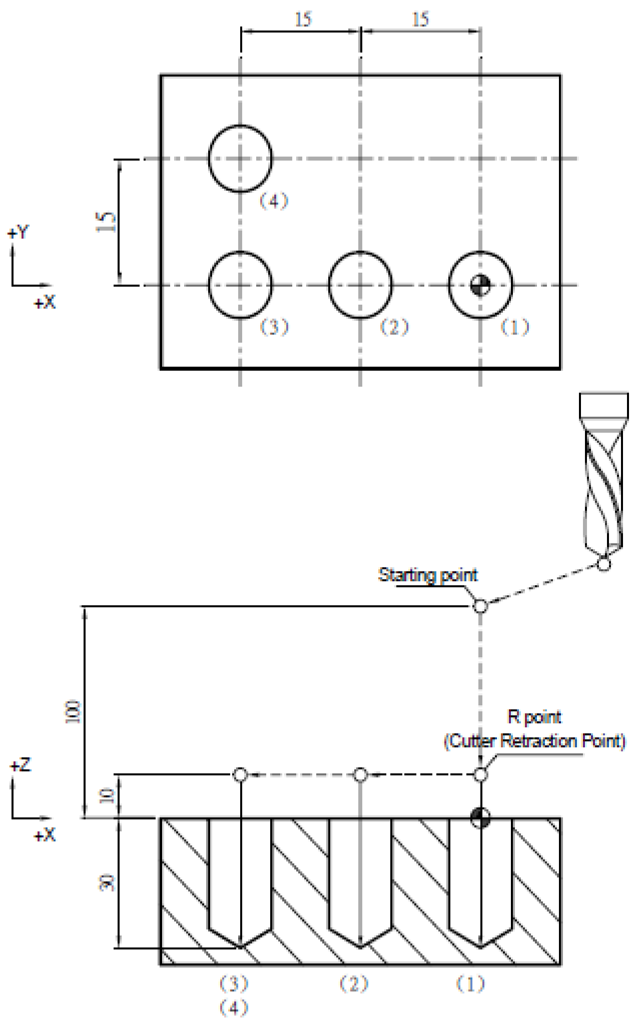
1. It is rapidly positioned to reach the hole location (X, Y, while maintaining the original tool height);
2. It is rapidly positioned to reach the R point coordinate (R);
3. Based on the set cutting feed rate, spindle rotational speed, and cutting-peck tool feed rate;
4. Rapidly retract to R point;
5. Rapidly positioned to a certain height away from the previous machining point, and this height is set by system parameter 160031;
6. Cutting feed, with the feed rate equal to (amount of peck drilling tool feed + the set value of system parameter 160031);
7. Rapidly retract to R point;
8. Repeat steps 5 to 7 until cutting reaches the base of the hole;
9. Under G98 mode, it is rapidly retracted to the reference point; under G99 mode, it is rapidly retracted to the R point;
10. If K (> 1) is assigned, steps 2 to 9 mentioned above should be repeated until completing the specified number of repeated drilling; otherwise the program terminates;
11. In G91 mode, argument R is for assigning the distance between R point and the reference point; argument Z is for assigning the distance between the hole base position and R point; If K is assigned (> 1), after executing a drilling operation (steps 2 to 9 mentioned above), a hole position offset will be implemented in accordance with the assigned X, Y before continuing with the next drilling operation;
12. The difference between G73 and G83 is that the former's amount of retraction is set by system parameter 160031, while the latter will be retracted to R point every time.

Plot legends:

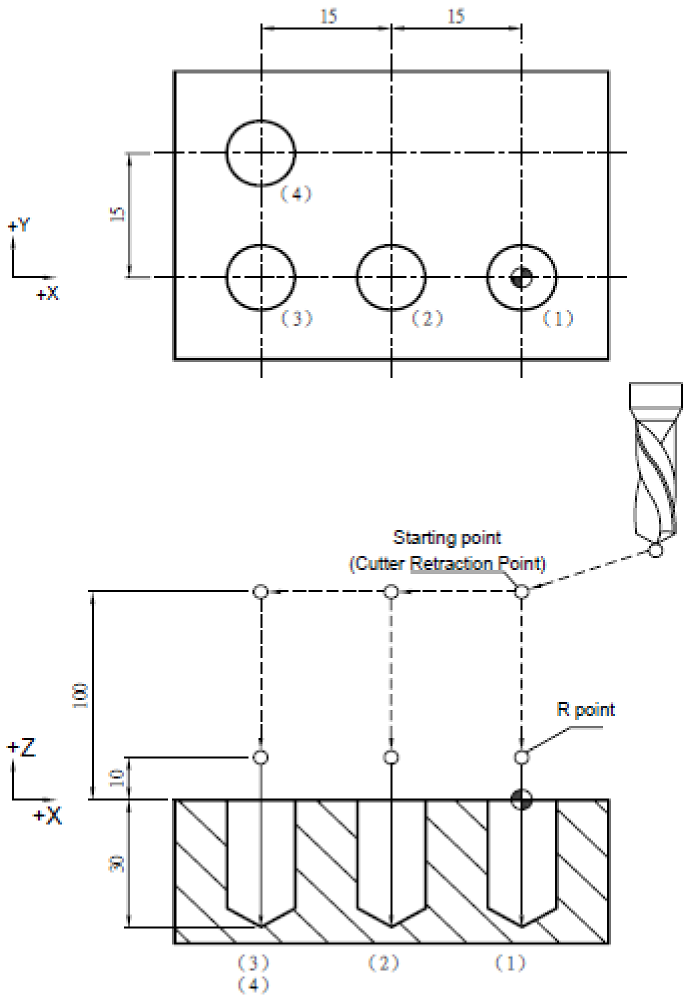


Program example:

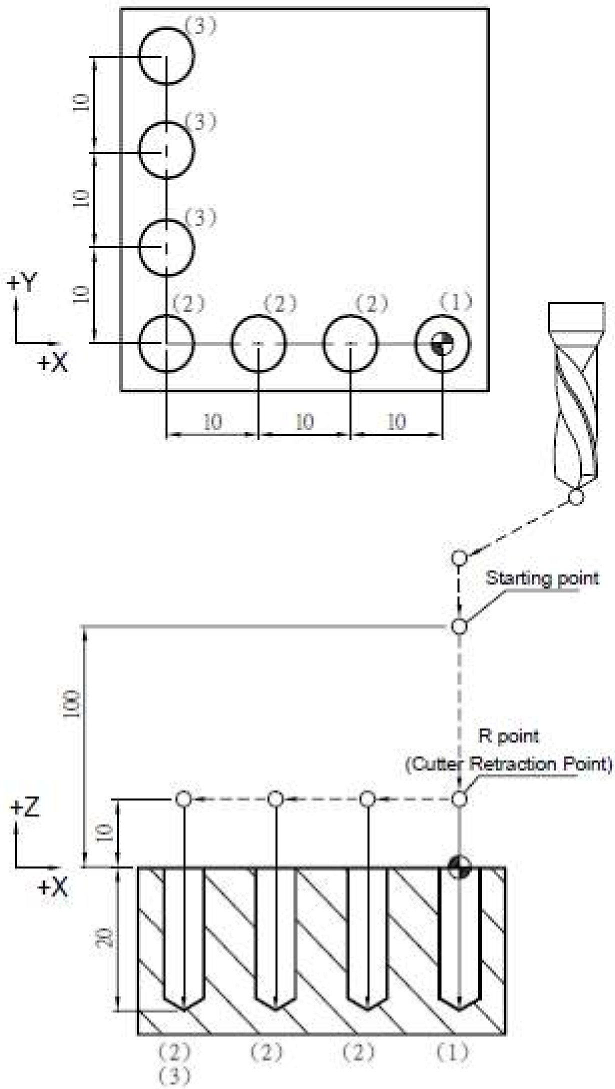
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G83 X0. Y0. Z-30. R10. Q4 K1 F100.; ----------------------------------------(1)
X-15.; ---------------------------------------------------------------------------(2)
X-30.; ---------------------------------------------------------------------------(3)
X-30. Y15.;-----------------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



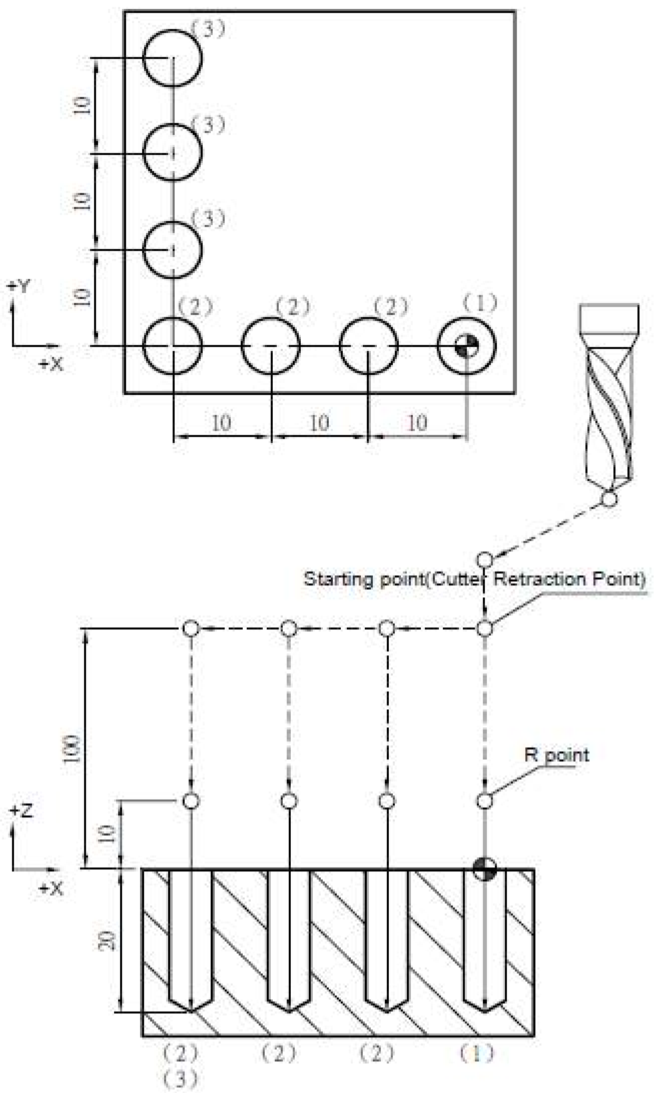
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G83 X0. Y0. Z-30. R10. Q4 K1 F100.; --------------------------------------------------------------------------------(1)
X-15.; ------------------------------------------------------------------------------------------------------------------------(2)
X-30.; ------------------------------------------------------------------------------------------------------------------------(3)
X-30. Y15.;----------------------------------------------------------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



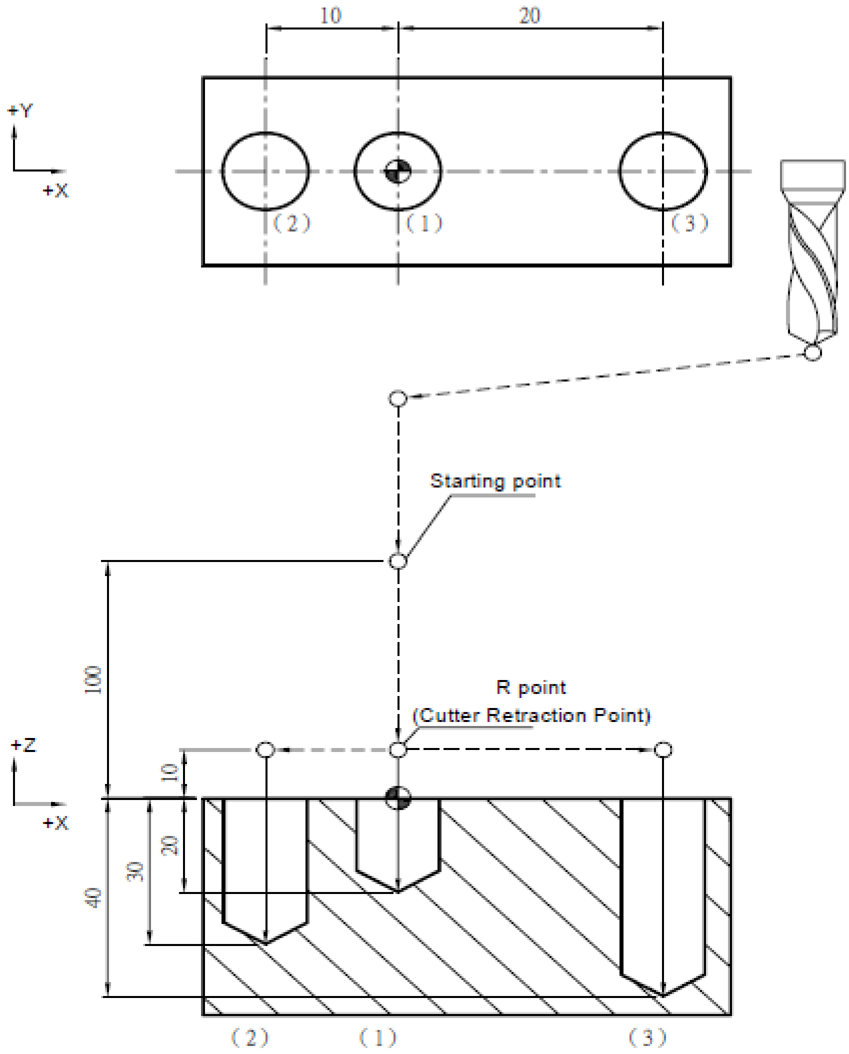
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G83 X0. Y0. Z-20. R10. Q4 K1 F100.; -------------------------------------------------(1)
G91 X-10. K3; ---------------------------------------------------------------------------(2)
Y10. K3; --------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



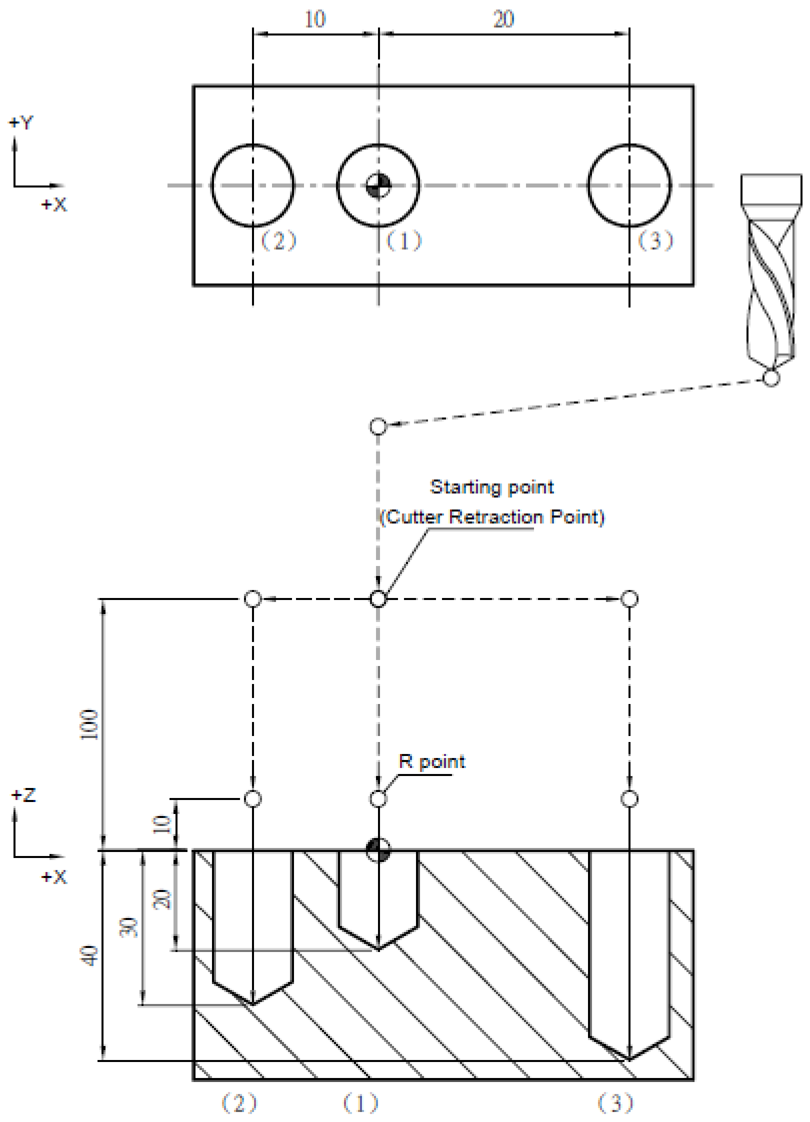
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G83 X0. Y0. Z-20. R10. Q4 K1 F100.; ---------------------------------------------------------------------------(1)
G91 X-10. K3; -------------------------------------------------------------------------------------------------------(2)
Y10. K3; ------------------------------------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G83 X0. Y0. Z-20. R10. Q4 K1 F100.; ----------------------------------------(1)
X-10. Z-30.; --------------------------------------------------------------------(2)
X20. Z-40.; ---------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G83 X0. Y0. Z-20. R10. Q4 K1 F100.; ------------------------------------------(1)
X-10. Z-30.; ---------------------------------------------------------------------(2)
X20. Z-40.; ----------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



G84 Right-hand thread tapping cycle

Command format 1:

```
G84 X__ Y__ Z__ R__ P__ K__ F__;
```

Command format 2:

```
G84 X__ Y__ Z__ R__ P__ Q__ K__ F__;
```

Argument description:

Function 1: Right-hand thread tapping cycle.

X__Y__	:	Coordinates of the hole position (mm).
Z__	:	Coordinates of the hole base (mm).
R__	:	Coordinate value of R point (which is the reset point) (mm).
P__	:	Pause time at hole base (1/1000 second), minimum unit, no decimal point allowed.
K__	:	Number of repetitions.
F__	:	Cutting feed rate (G94 mm/min, G95 mm/rev).

Function 2: Right-hand thread peck tapping cycle.

X__Y__	:	Coordinates of the hole position (mm).
Z__	:	Coordinates of the hole base (mm).
R__	:	Coordinate value of R point (which is the reset point) (mm).
P__	:	Pause time at hole base (1/1000 second), minimum unit, no decimal point allowed.
Q__	:	Feed amount per cut (mm).
K__	:	Number of repetitions.
F__	:	Cutting feed rate (G94 mm/min, G95 mm/rev).

The peck tapping mode of the peck tapping cycle can be set by parameter 160041 (0: High speed, 1: Normal). When high speed is selected, each peck tapping will only be retracted by the amount set in parameter 160042; if normal is selected, peck tapping is retracted to R point every time.

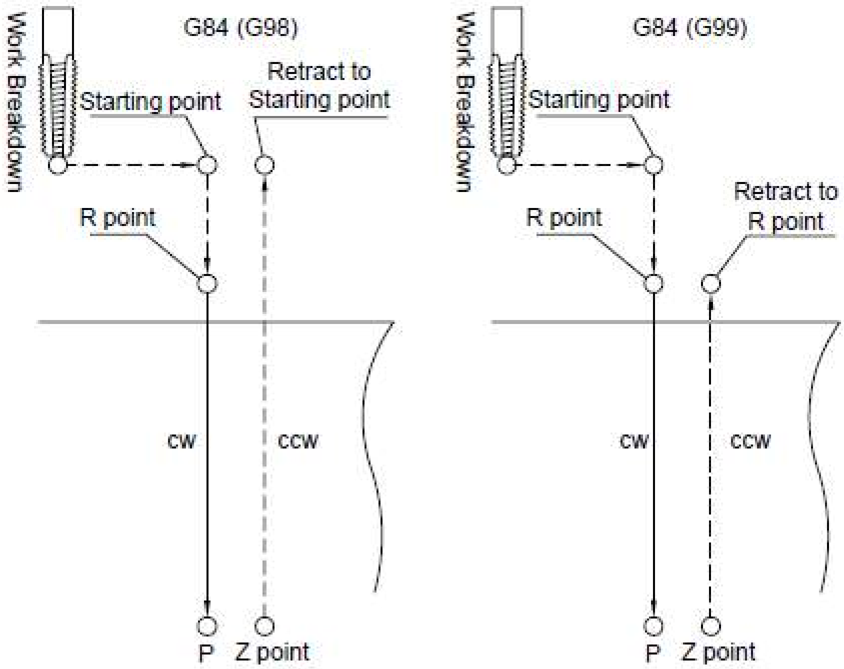
Function 1 operation description (using the G17 plane as an example):

1. It is rapidly positioned to reach the hole location (X, Y, while maintaining the original tool height);
2. It is rapidly positioned to reach the R point coordinate (R);
3. Tapping starts with the spindle in forward rotation;
4. Cutting should reach the hole base position based on a set cutting feed rate and spindle rotation

speed (Z);

5. Spindle stops; if P is assigned, it is the pause period at the hole base position;
6. Spindle is in reverse rotation based on a set cutting feed rate and spindle rotational speed for the cutting to reach R point;
7. Tapping ends, spindle stops; if P is assigned, it is the pause period at the R point position;
8. Under G98 mode, it is rapidly retracted to the reference point;
9. If K (> 1) is assigned, steps 2 to 8 mentioned above should be repeated until completing the specified number of repeated drilling; otherwise the program terminates;
10. In G91 mode, argument R is for assigning the distance between R point and the reference point; argument Z is for assigning the distance between the hole base position and R point; when K is being assigned (> 1), after completion of each tapping operation (steps 2 to 8 mentioned above), a hole position incremental offset will be implemented in accordance with the assigned X, Y before continuing with the next tapping operation.
11. Under G94 mode, cutting feed rate F is rotation speed (S) × thread pitch (PITCH).

Plot legends:

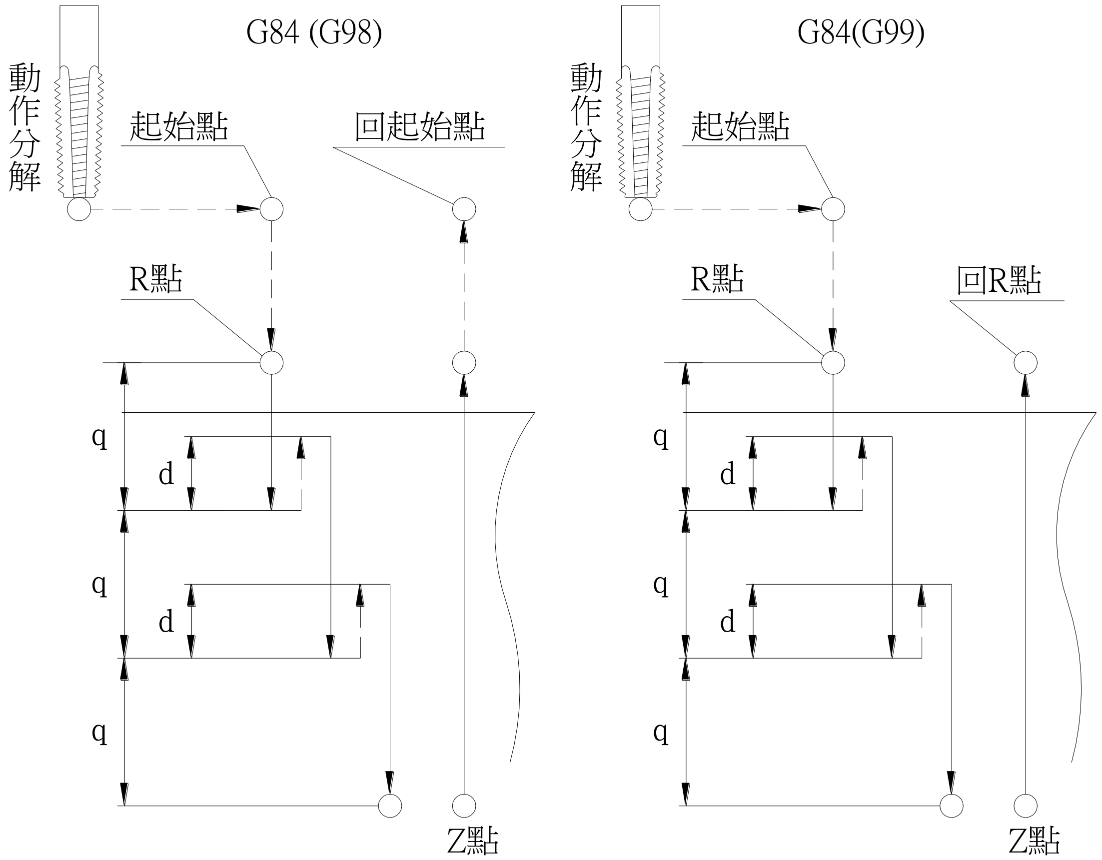


Function 2 operation description (using the G17 plane as an example):

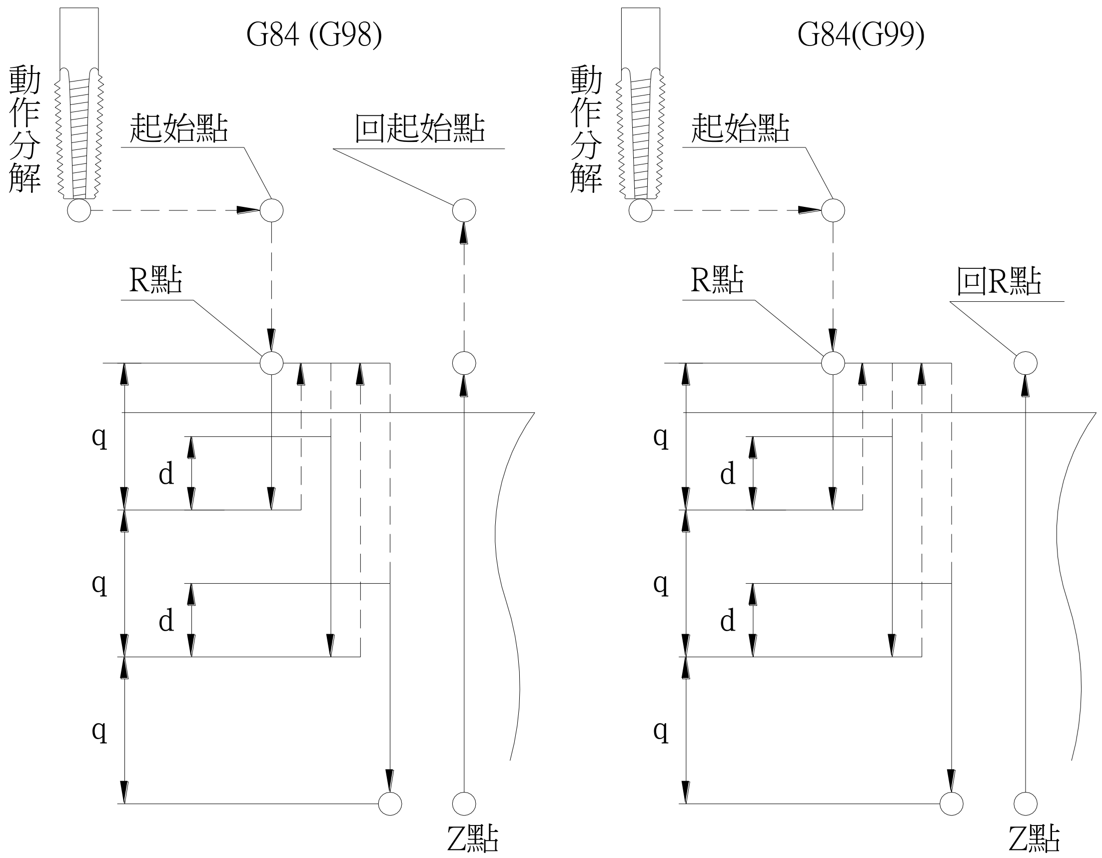
19. It is rapidly positioned to reach the hole location (X, Y, while maintaining the original tool height);
20. It is rapidly positioned to reach the R point coordinate (R);
21. Tapping starts with the spindle in forward rotation;
22. based on the set cutting feed rate, spindle rotation speed, and cutting-peck feed;
23. Spindle stops; if P is assigned, it is the pause period at this point position;
24. Spindle in reverse rotation; if parameter 160041 is set as high speed peck tapping mode, the amount of retraction will be set by system parameter 160042; if it is set as normal peck tapping mode, then it will be rapidly retracted to R point;
25. Spindle stops; if P is assigned, it is the pause period at this point position;
26. Spindle is in forward rotation; if parameter 160041 is set as normal peck tapping mode, it will be rapidly positioned to a certain height away from the previous machining point, and this height will be set by system parameter 160042;
27. Cutting feed, with the feed rate equal to (amount of peck tapping tool feed + the set value of system parameter 160042);
28. Spindle stops; if P is assigned, it is the pause period at the hole base position;
29. Spindle in reverse rotation; rapidly retracted to R point;
30. Steps 8 to 11 will be repeated until cutting reaches the base of the hole;
31. Spindle is in reverse rotation based on a set cutting feed rate and spindle rotational speed for the cutting to reach R point;
32. Tapping ends, spindle stops; if P is assigned, it is the pause period at the R point position;
33. Under G98 mode, it is rapidly retracted to the reference point;
34. If K(> 1) is assigned, steps 2 to 8 mentioned above should be repeated until completing the specified number of repeated tapping; otherwise the program terminates;
35. In G91 mode, argument R is for assigning the distance between R point and the reference point; argument Z is for assigning the distance between the hole base position and R point; when K is being assigned (> 1), after completion of each tapping operation (steps 2 to 8 mentioned above), a hole position incremental offset will be implemented in accordance with the assigned X, Y before continuing with the next tapping operation.
36. Under G94 mode, cutting feed rate F is rotation speed (S) × thread pitch (PITCH).

Plot legends:

Parameter 160041 is for setting high speed peck tapping mode

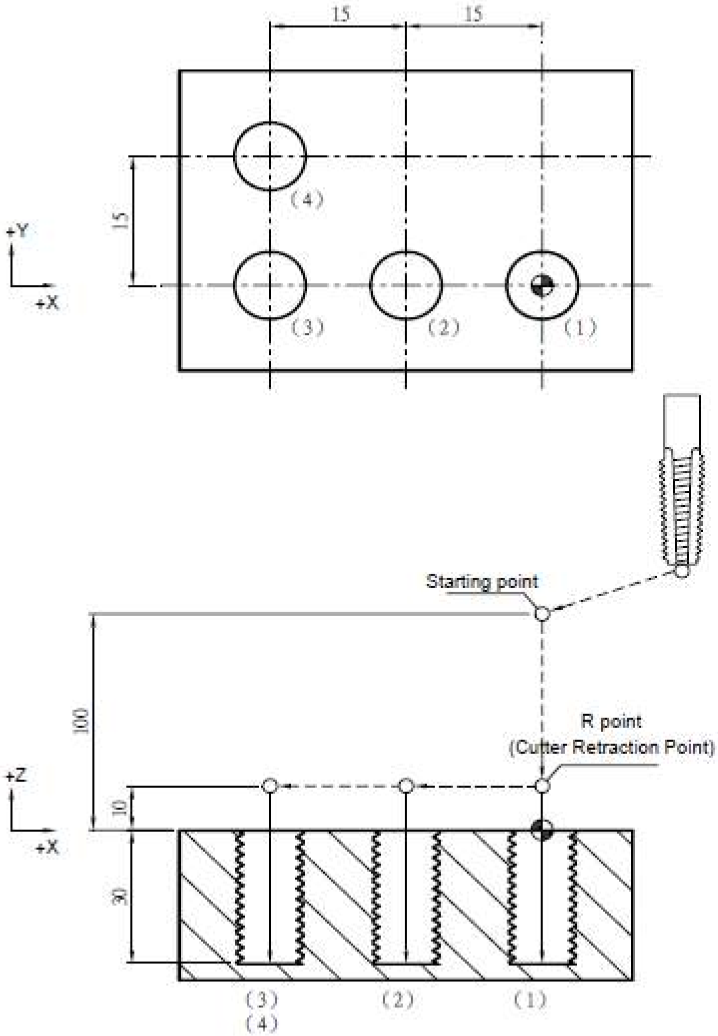


Parameter 160041 is for setting general peck tapping mode

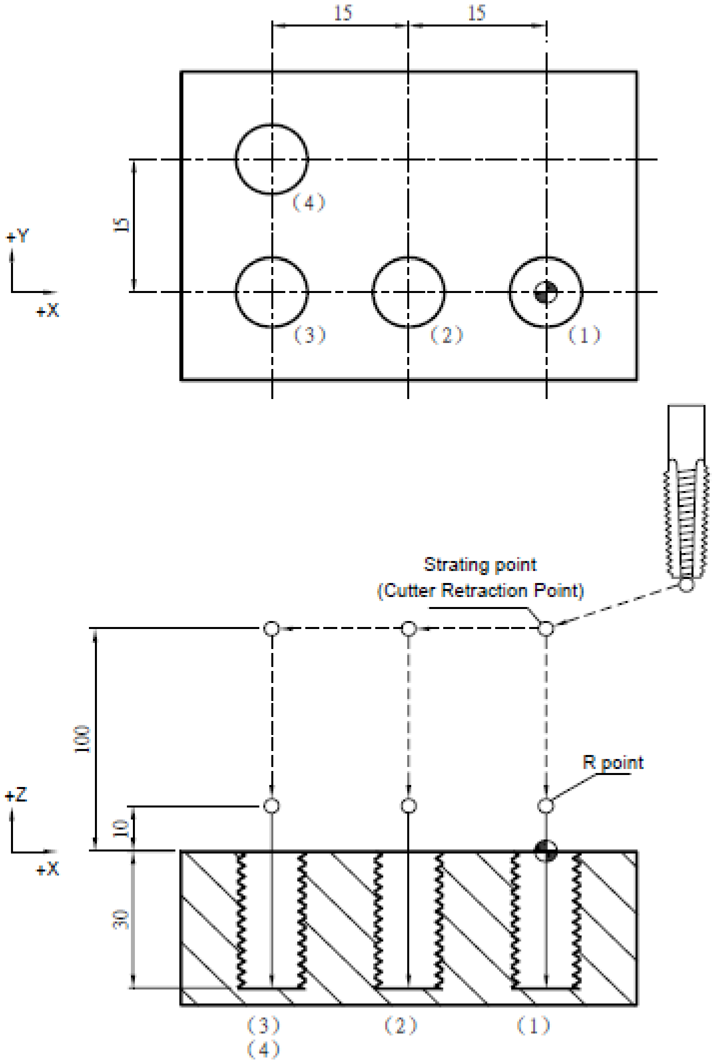


Program example:

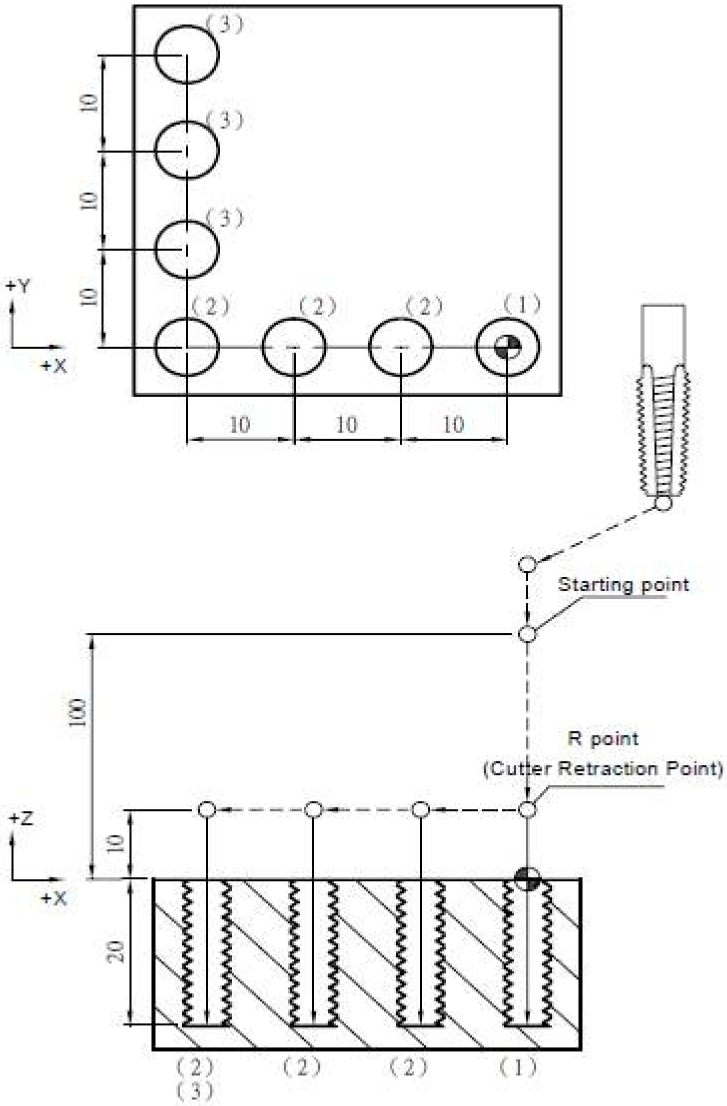
```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G99 G84 X0. Y0. Z-30. R10. P1000 K1 F1000.; --------------------------------------------------------------------------(1)
X-15.; ---------------------------------------------------------------------------------------------------------------(2)
X-30.; ---------------------------------------------------------------------------------------------------------------(3)
X-30. Y15.;-----------------------------------------------------------------------------------------------------------(4)
M28;
G80 G91 G28 X0. Y0. Z0.;
```



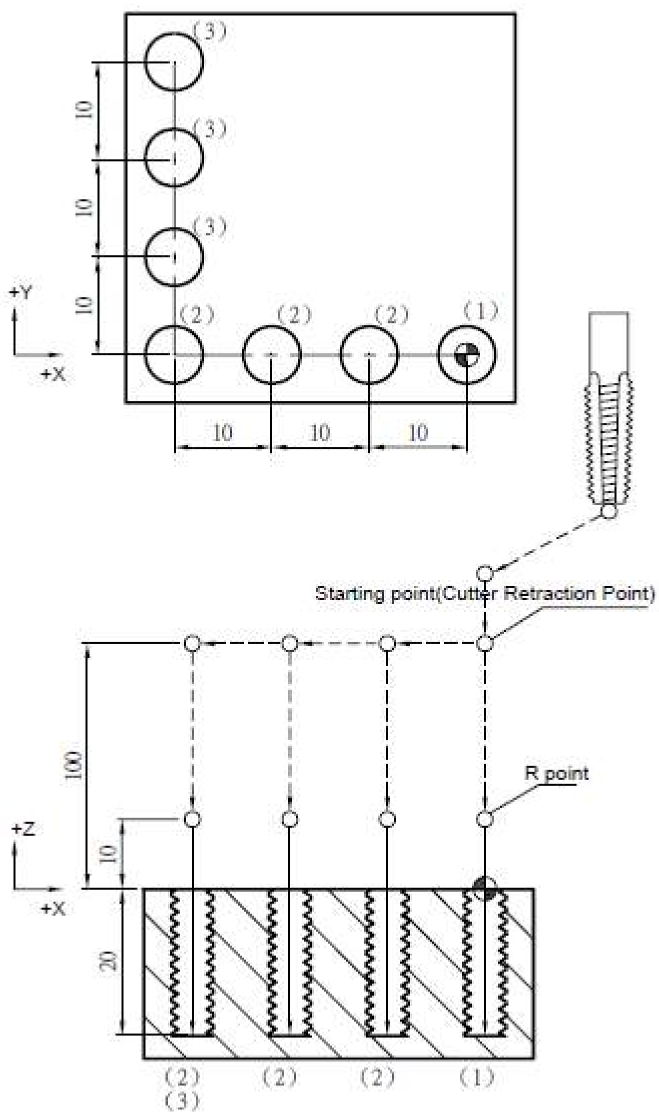
```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G98 G84 X0. Y0. Z-30. R10. P1000 K1 F1000.; ------------------------------------------------(1)
X-15.; ----------------------------------------------------------------------------------------(2)
X-30.; ----------------------------------------------------------------------------------------(3)
X-30. Y15.; -----------------------------------------------------------------------------------(4)
M28;
G80 G91 G28 X0. Y0. Z0.;
```



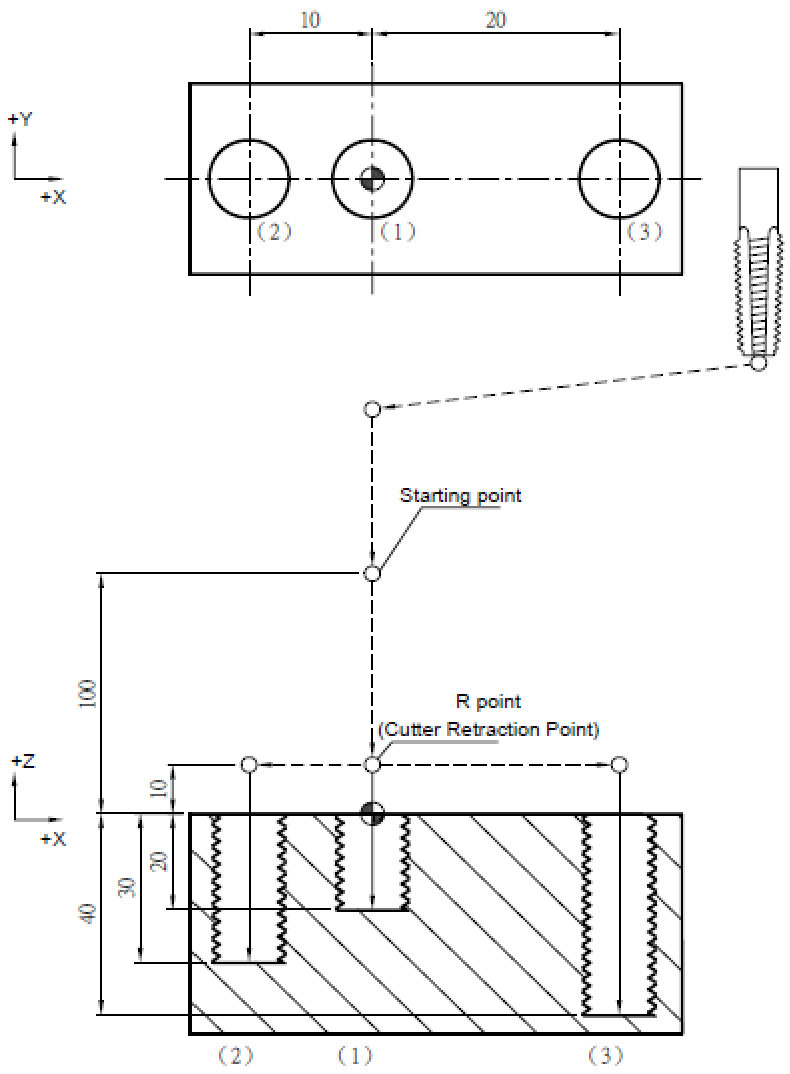
```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G99 G84 X0. Y0. Z-20. R10. P1000 K1 F1000.; ----------------------------------------------------------------------------(1)
G91 X-10. K3; ---------------------------------------------------------------------------------------------------------------------(2)
Y10. K3; -----------------------------------------------------------------------------------------------------------------------------(3)
M28;
G80 G91 G28 X0. Y0. Z0.;
```



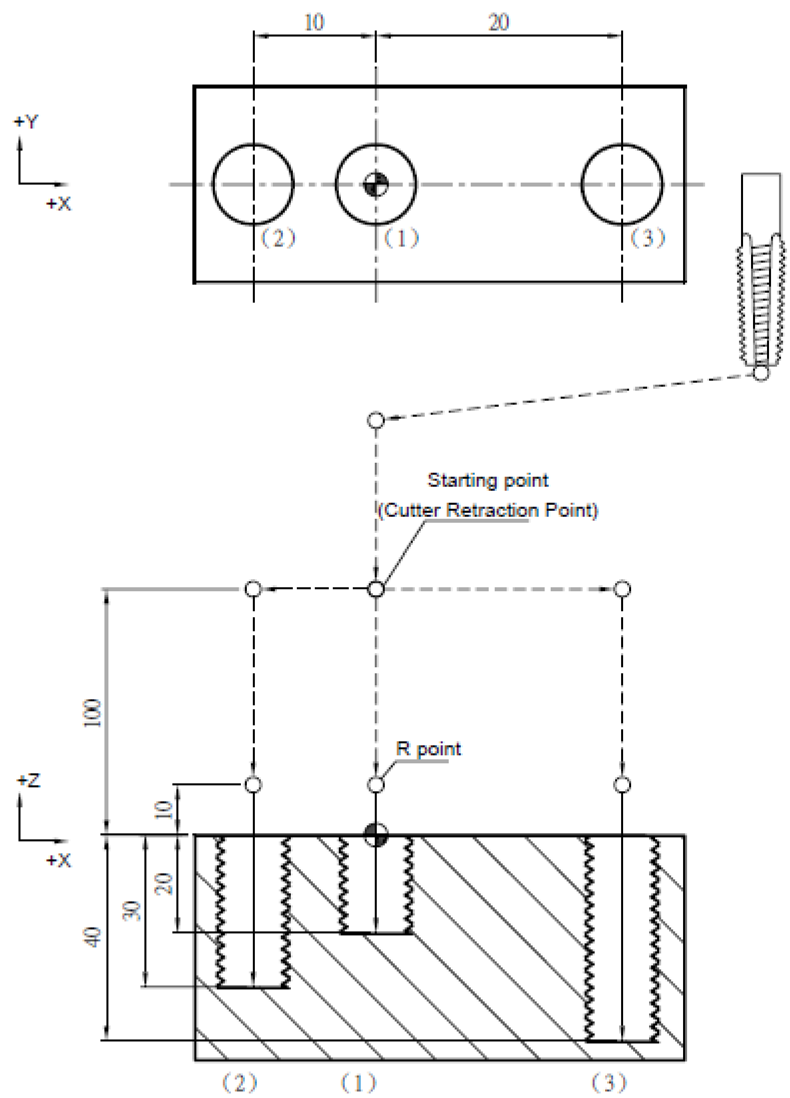
```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G98 G84 X0. Y0. Z-20. R10. P1000 K1 F1000.; ---------------------------------------------------------------------------(1)
G91 X-10. K3; ---------------------------------------------------------------------------------------------------------------------(2)
Y10. K3; ---------------------------------------------------------------------------------------------------------------------------(3)
M28;
G80 G91 G28 X0. Y0. Z0.;
```



```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G99 G84 X0. Y0. Z-20. R10. P1000 K1 F1000.; -----------------------------------(1)
X-10. Z-30.; ------------------------------------------------------------------(2)
X20. Z-40.; -------------------------------------------------------------------(3)
M28;
G80 G91 G28 X0. Y0. Z0.;
```



```
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
M29 S1000;
G98 G84 X0. Y0. Z-20. R10. P1000 K1 F1000.; -------------------------------------------(1)
X-10. Z-30.; ----------------------------------------------------------------------------(2)
X20. Z-40.; -----------------------------------------------------------------------------(3)
M28;
G80 G91 G28 X0. Y0. Z0.;
```



G85 Reaming cycle

Command format:

G85 X__ Y__ Z__ R__ K__ F__;

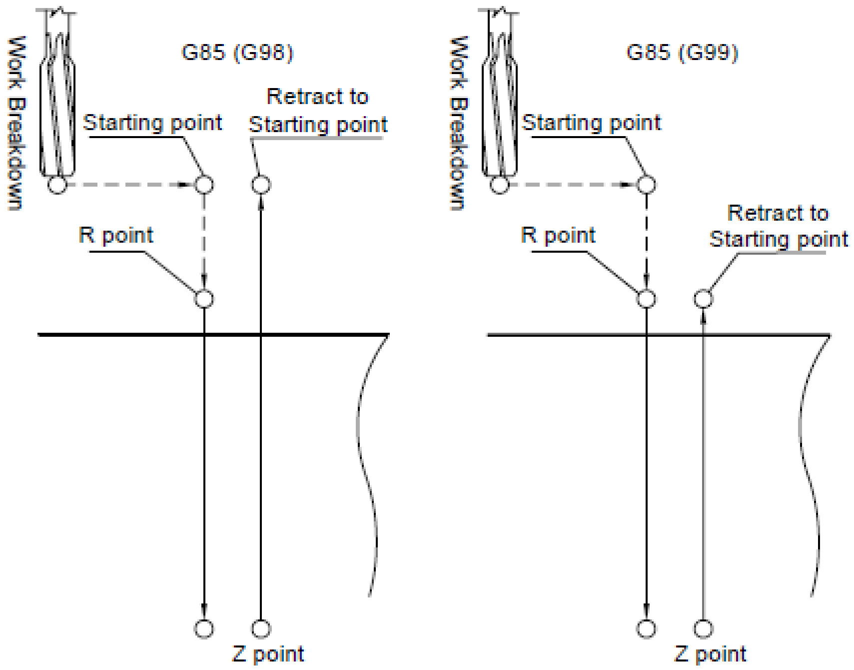
Argument description:

X__Y__	:	Coordinates of the hole position (mm).
Z__	:	Coordinates of the hole base (mm).
R__	:	Coordinate value of R point (which is the reset point) (mm).
K__	:	Number of repetitions.
F__	:	Cutting feed rate (G94 mm/min).

Operation description (using the G17 plane as an example):

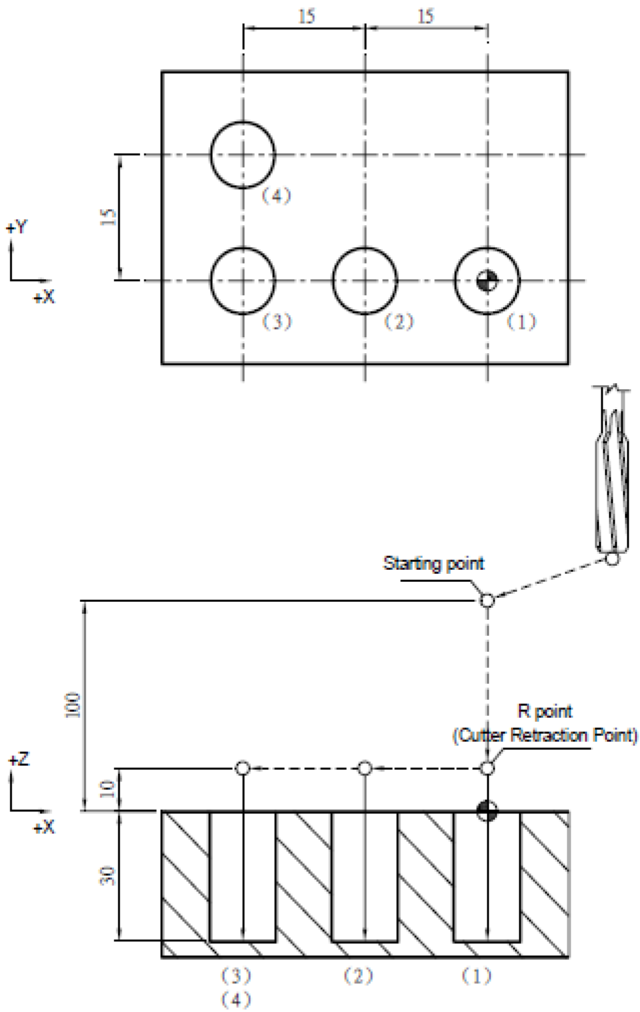
1. It is rapidly positioned to reach the hole location (X, Y, while maintaining the original tool height);
2. It is rapidly positioned to reach the R point coordinate (R);
3. Cutting should reach the hole base position based on a set cutting feed rate and spindle rotation speed (Z);
4. based on a set cutting feed rate and spindle rotational speed, it will be retracted to the R point position;
5. Under G98 mode, it is rapidly retracted to the reference point.
6. When K is assigned (> 1), steps 2 to 4 mentioned above should be repeated until completing the specified number of repeated reaming; otherwise the program terminates;
7. In G91 mode, argument R is for assigning the distance between R point and the reference point; argument Z is for assigning the distance between the hole base position and R point; If K is assigned (> 1), after executing a reaming operation (steps 2 to 4 mentioned above), a hole position offset will be implemented in accordance with the assigned X, Y before continuing with the next reaming operation.
8. The difference between G85 and G89 is that the latter can be used to assign the pause time at hole base.

Plot legends:

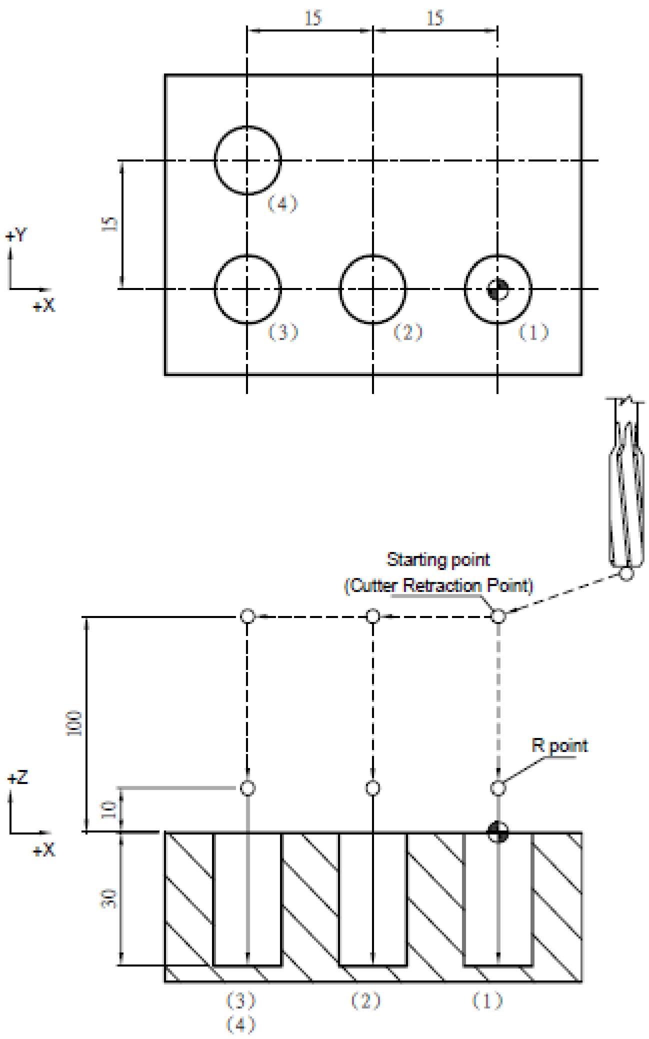


Program example:

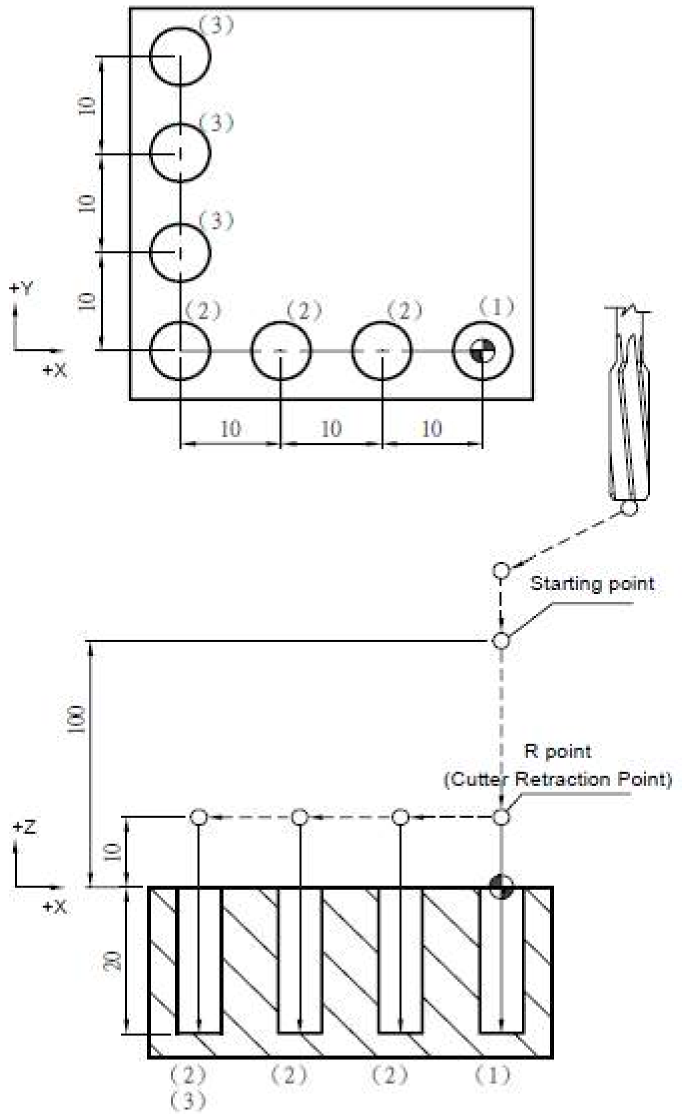
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G85 X0. Y0. Z-30. R10. K1 F100.; ----------------------------------------(1)
X-15.; ---------------------------------------------------------------------(2)
X-30.; ---------------------------------------------------------------------(3)
X-30. Y15.;-----------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



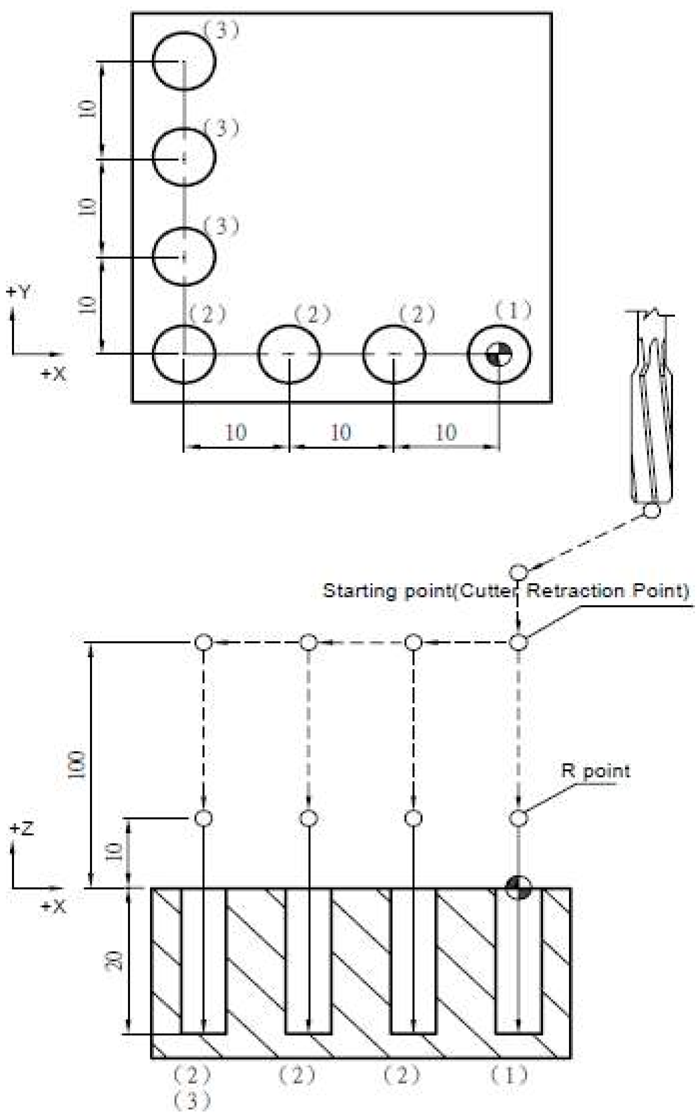
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G85 X0. Y0. Z-30. R10. K1 F100.; ----------------------------------------(1)
X-15.; -------------------------------------------------------------------(2)
X-30.; -------------------------------------------------------------------(3)
X-30. Y15.; --------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



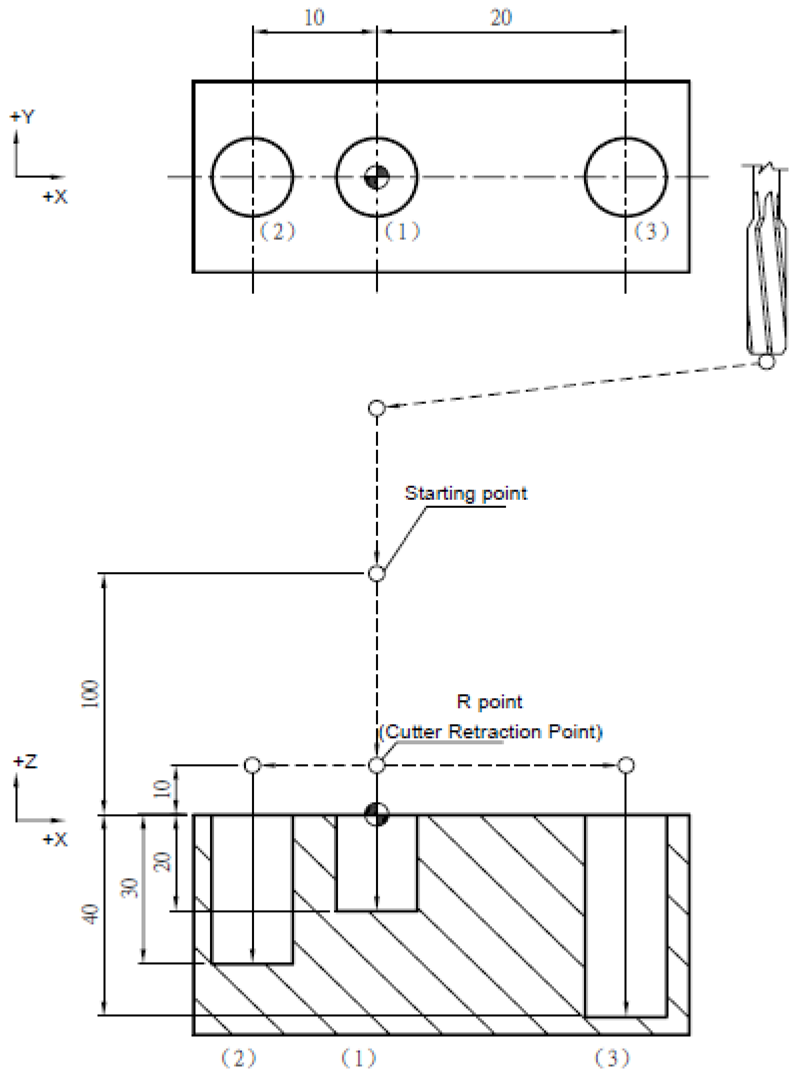
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G85 X0. Y0. Z-20. R10. K1 F100.; ------------------------------------------(1)
G91 X-10. K3; ----------------------------------------------------------------(2)
Y10. K3; ---------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



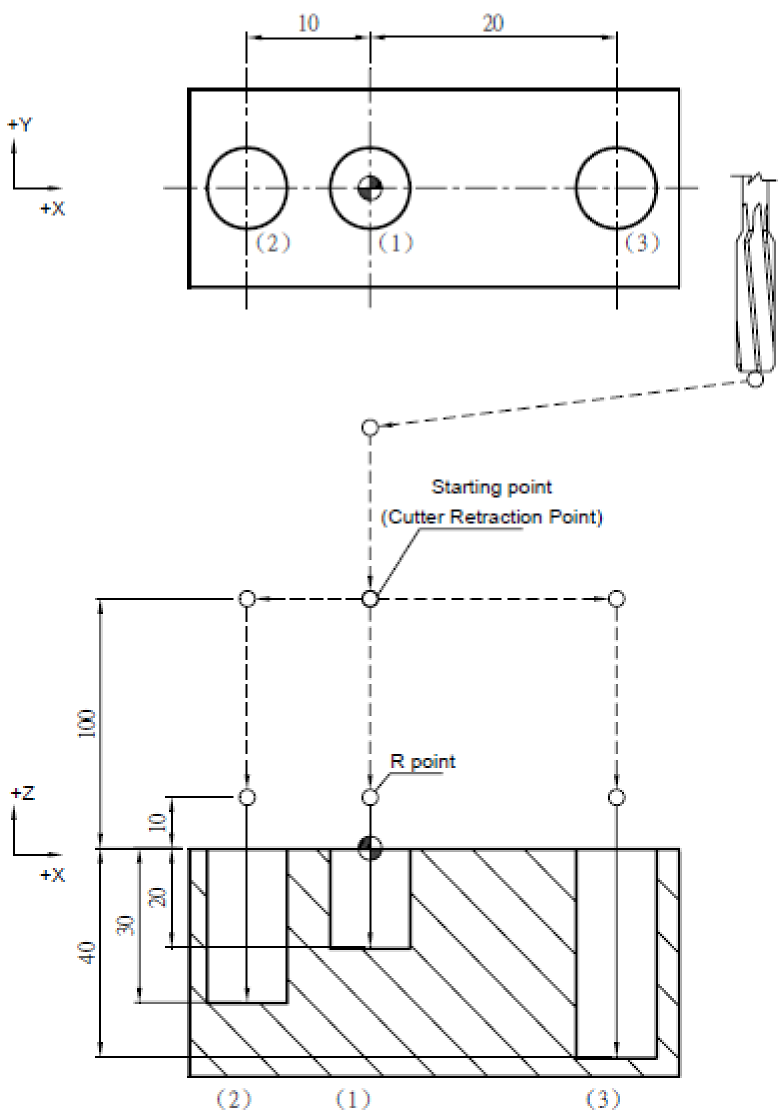
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G85 X0. Y0. Z-20. R10. K1 F100.; ----------------------------------------(1)
G91 X-10. K3; ---------------------------------------------------------------(2)
Y10. K3; --------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G85 X0. Y0. Z-20. R10. K1 F100.; ---------------------------------------------------------------------------------------(1)
X-10. Z-30.; ----------------------------------------------------------------------------------------------------------------------(2)
X20. Z-40.; -----------------------------------------------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G85 X0. Y0. Z-20. R10. K1 F100.; ------------------------------------------------------------------------------------------(1)
X-10. Z-30.; -----------------------------------------------------------------------------------------------------------------(2)
X20. Z-40.; ------------------------------------------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



G86 Boring cycle

Command format:

```
G86 X__ Y__ Z__ R__ K__ F__;
```

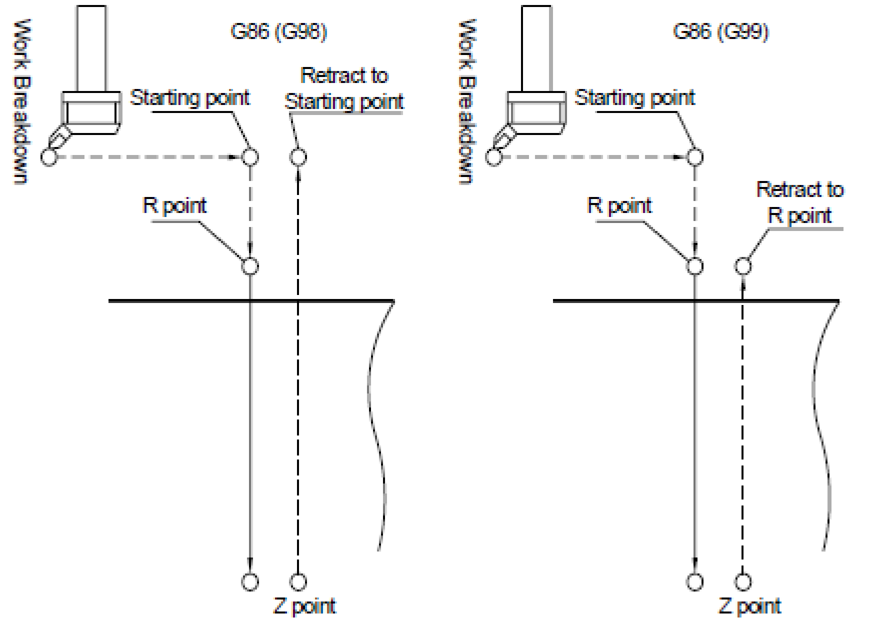
Argument description:

X__Y__	:	Coordinates of the hole position (mm).
Z__	:	Coordinates of the hole base (mm).
R__	:	Coordinate value of R point (which is the reset point) (mm).
K__	:	Number of repetitions.
F__	:	Feed rate (G94 mm/min).

Operation description (using the G17 plane as an example):

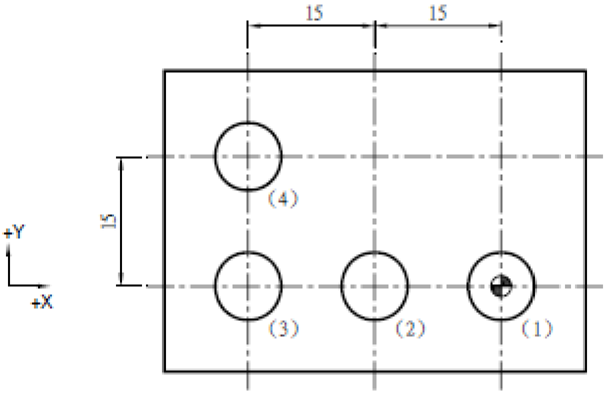
1. It is rapidly positioned to reach the hole location (X, Y, while maintaining the original tool height);
2. It is rapidly positioned to reach the R point coordinate (R);
3. Cutting should reach the hole base position based on a set cutting feed rate and spindle rotation speed (Z);
4. Spindle rotation is stopped;
5. Under G98 mode, it is rapidly retracted to the reference point; under G99 mode, it is rapidly retracted to the R point;
6. Spindle in forward rotation;
7. When K is assigned (> 1), steps 2 to 6 mentioned above should be repeated until completing the specified number of repeated boring; otherwise the program terminates;
8. In G91 mode, argument R is for assigning the distance between R point and the reference point; argument Z refers to the distance between R point and the hole base; if K is assigned (> 1), after the completion of every boring action (steps 2 to 6 mentioned above), a hole position offset will be implemented in accordance with the assigned X, Y before continuing with the next boring action.
9. The difference between G86 and G88 is that the latter can be used to assign the pause time at hole base.

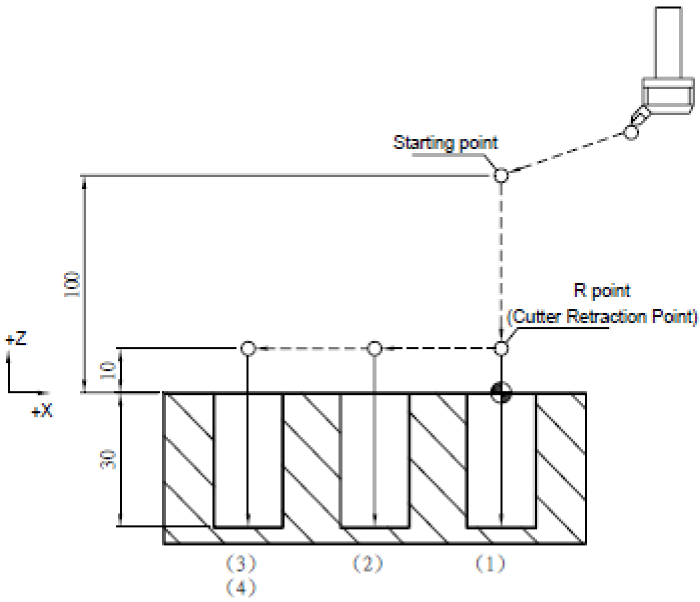
Plot legends:



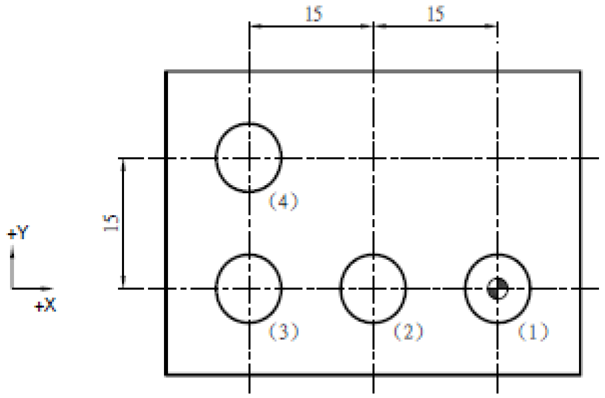
Program example:

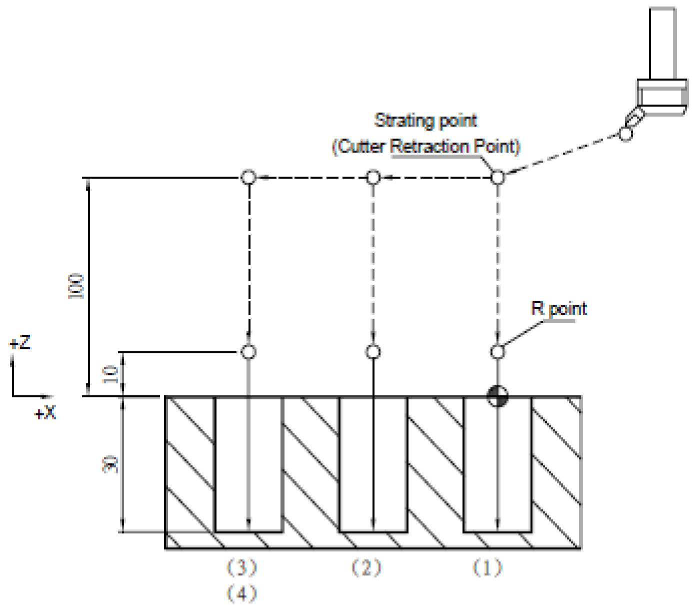
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G86 X0. Y0. Z-30. R10. K1 F100.; ----------------------------------------(1)
X-15.; -------------------------------------------------------------------------(2)
X-30.; -------------------------------------------------------------------------(3)
X-30. Y15.; --------------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



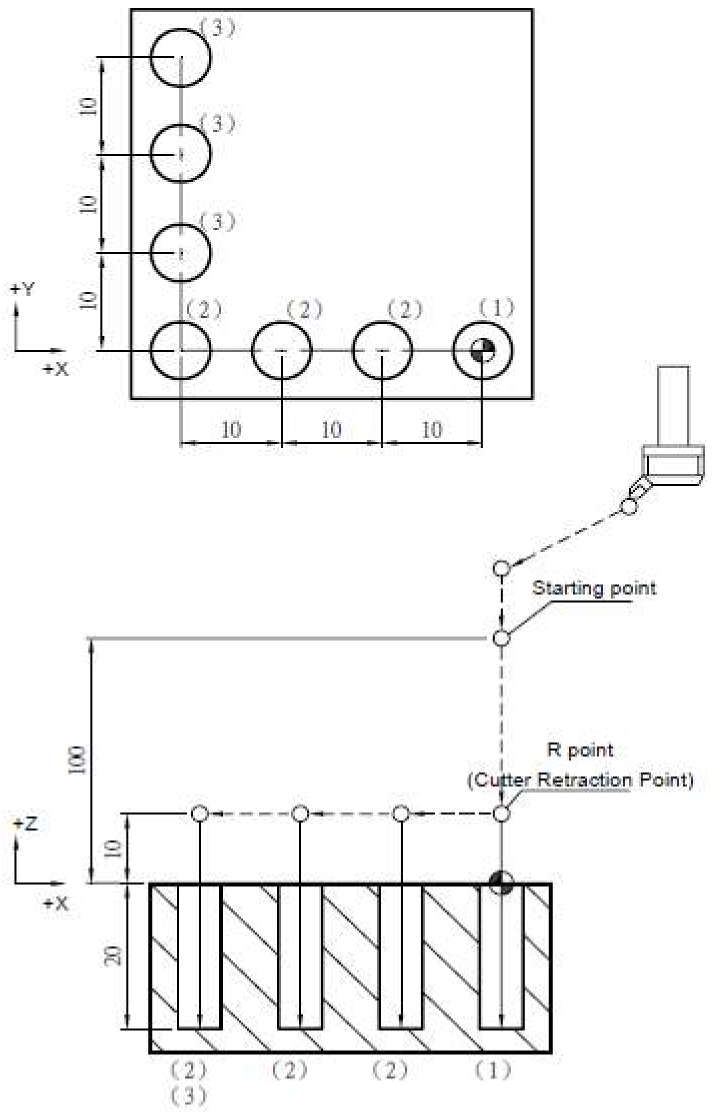


```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G86 X0. Y0. Z-30. R10. K1 F100.; ------------------------------------(1)
X-15.; ------------------------------------------------------------------(2)
X-30.; ------------------------------------------------------------------(3)
X-30. Y15.; -------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```

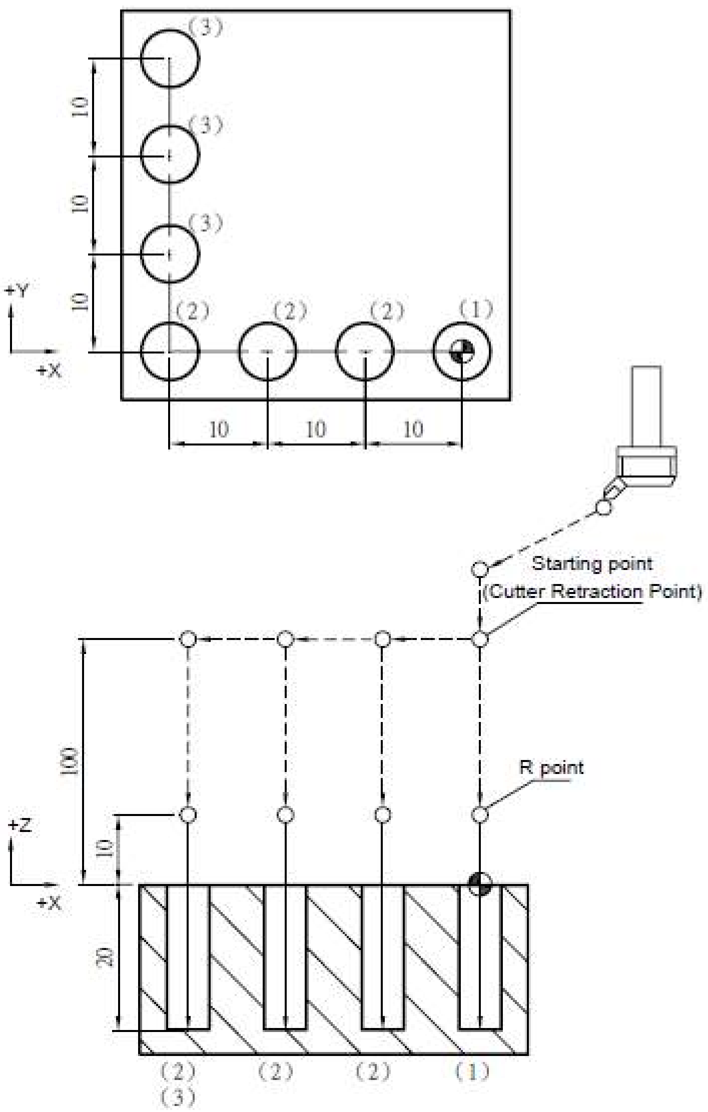




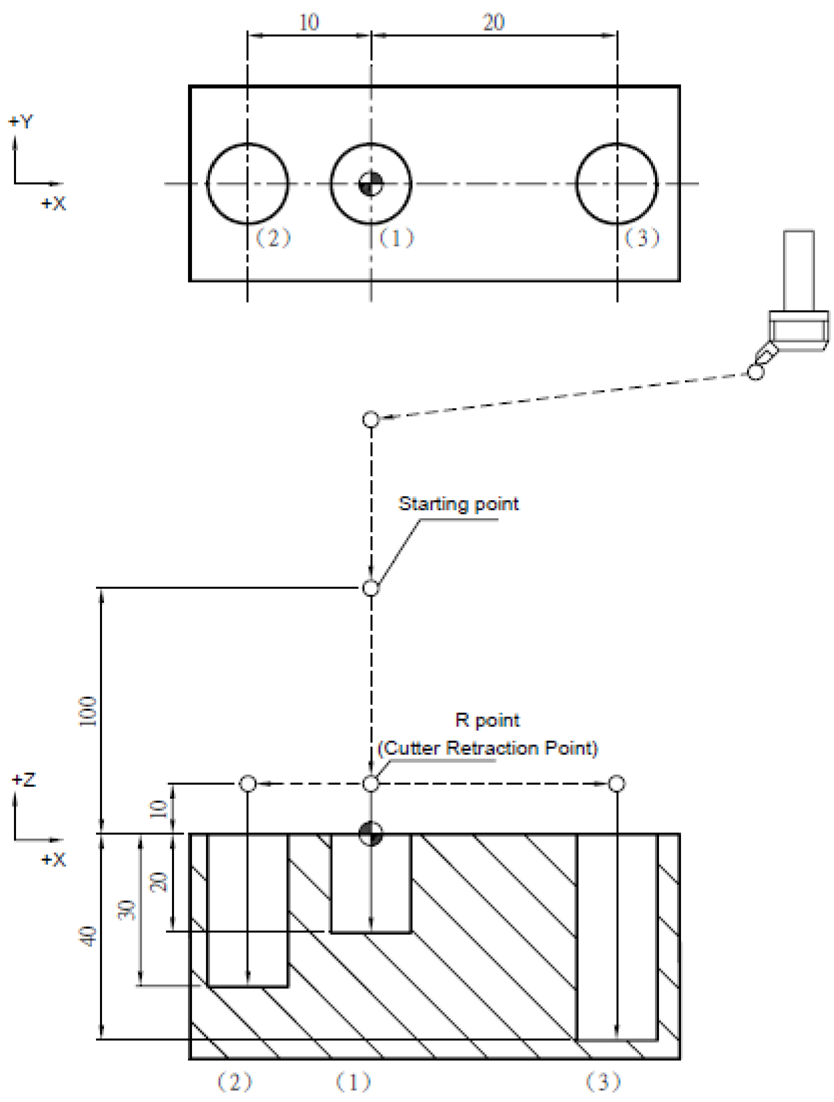
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G86 X0. Y0. Z-20. R10. K1 F100.; ----------------------------------------------------(1)
G91 X-10. K3; -----------------------------------------------------------------------------(2)
Y10. K3; ----------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



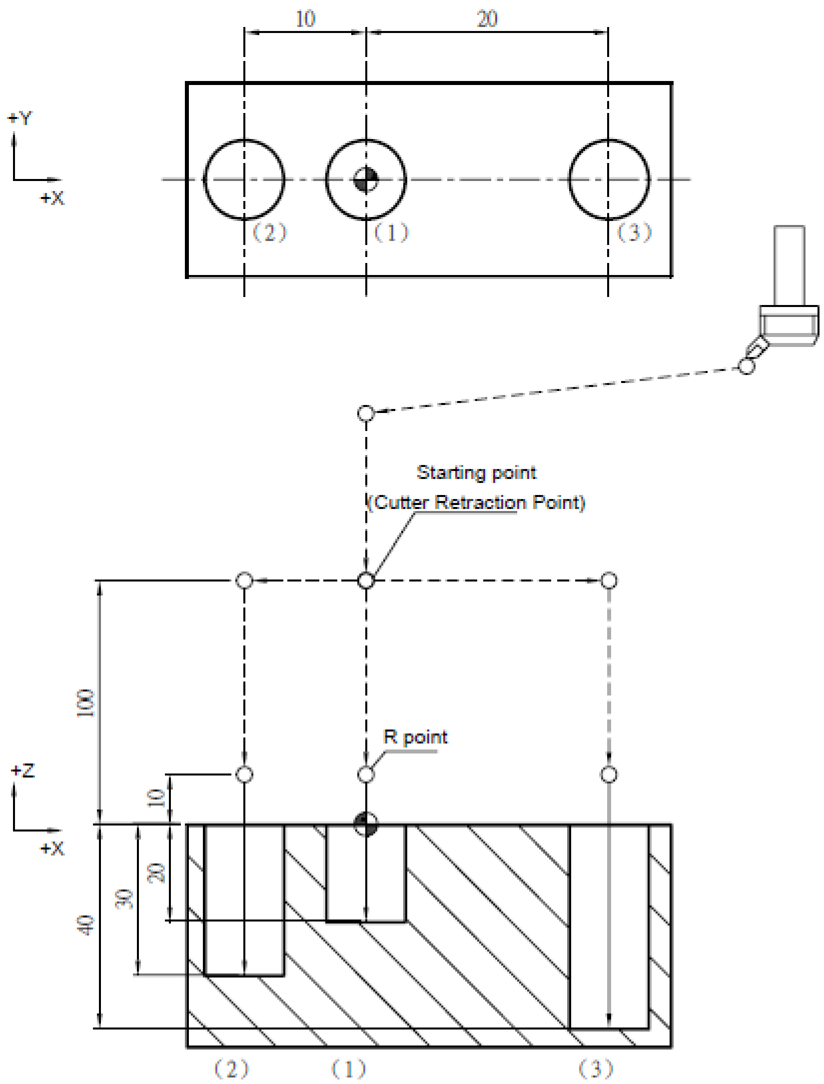
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G86 X0. Y0. Z-20. R10. K1 F100.; ---------------------------------------------------------------------------------------(1)
G91 X-10. K3; -----------------------------------------------------------------------------------------------------------------(2)
Y10. K3; ------------------------------------------------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G86 X0. Y0. Z-20. R10. K1 F100.; --------------------------------------------------------------------------------------(1)
X-10. Z-30.; ------------------------------------------------------------------------------------------------------------------(2)
X20. Z-40.; ------------------------------------------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G86 X0. Y0. Z-20. R10. K1 F100.; --------------------------------------(1)
X-10. Z-30.; ---------------------------------------------------------------(2)
X20. Z-40.; ----------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



G87 Back boring cutting

Command format:

G87 X__ Y__ Z__ R__ P__ Q__ K__ F__;

Argument description:

X__Y__	:	Coordinates of the hole position (mm).
Z__	:	Coordinates of the hole base (mm).
R__	:	Coordinate value of R point (which is the reset point) (mm).
P__	:	Pause time at hole base (1/1000 second), minimum unit, no decimal point allowed.
Q__		Amount of offset at hole base (mm), with offset direction set by system parameter 170003.
K__	:	Number of repetitions.
F__	:	Cutting feed rate (G94 mm/min).

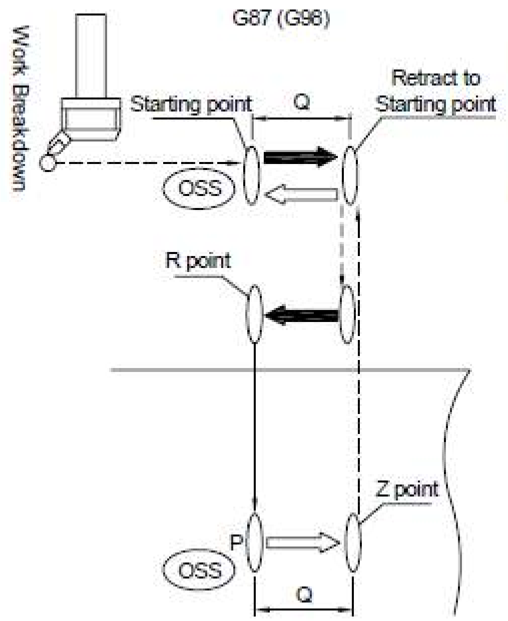
Operation description (Using the G17 plane as an example):

1. It is rapidly positioned to reach the hole location (X, Y, while maintaining the original tool height);
2. Execute M19 spindle positioning;
3. Tool offset, with offset distance set by argument Q, and offset direction set by system parameter 160010;
4. It is rapidly positioned to reach the R point coordinate (R);
5. Tool offset, which will return to the original hole coordinate position (by performing steps which are opposite of step 3 mentioned above);
6. Cancel spindle positioning status, spindle in forward rotation;
7. Cutting should reach the hole base position based on a set cutting feed rate and spindle rotation speed (Z);
8. If P is assigned, it is the pause period at the hole base position;
9. Spindle stops, executes the positioning of M19;
10. Tool offset, with offset distance set by argument Q, and offset direction set by system parameter 160010;
11. Under G98 mode, it is rapidly retracted to the reference point; under G99 mode, lateral movement before returning to the reference point is prohibited for it will result in workpiece interference, and the system operation will be carried out under G98 mode;
12. Tool offset, which will return to the original hole coordinate position (by performing steps which are opposite of step 10 mentioned above);
13. Cancel spindle positioning status, spindle rotates;
14. When K is assigned (> 1), steps 2 to 13 mentioned above should be repeated until completing the

specified number of repeated boring; otherwise the program terminates;

15. In G91 mode, argument R is for assigning the distance between R point and the reference point; argument Z is for assigning the distance between the hole base position and R point; when K is being assigned (> 1), after completion of each boring operation (steps 2 to 15 mentioned above), a hole position incremental offset will be implemented in accordance with the assigned X, Y before continuing with the next boring operation.

Plot legends:

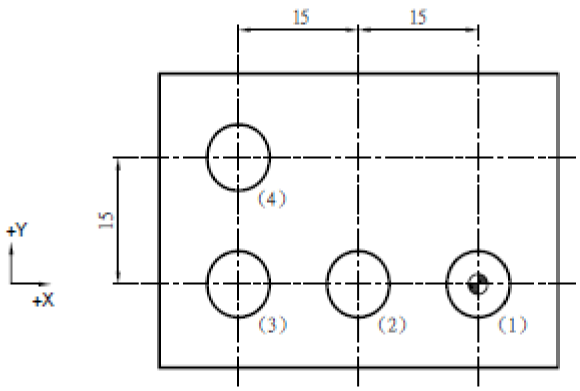


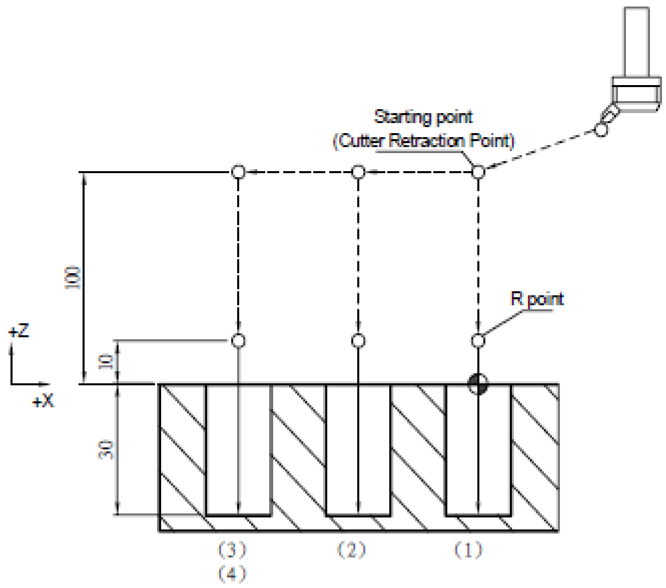
G87 (G99)

Not allowed

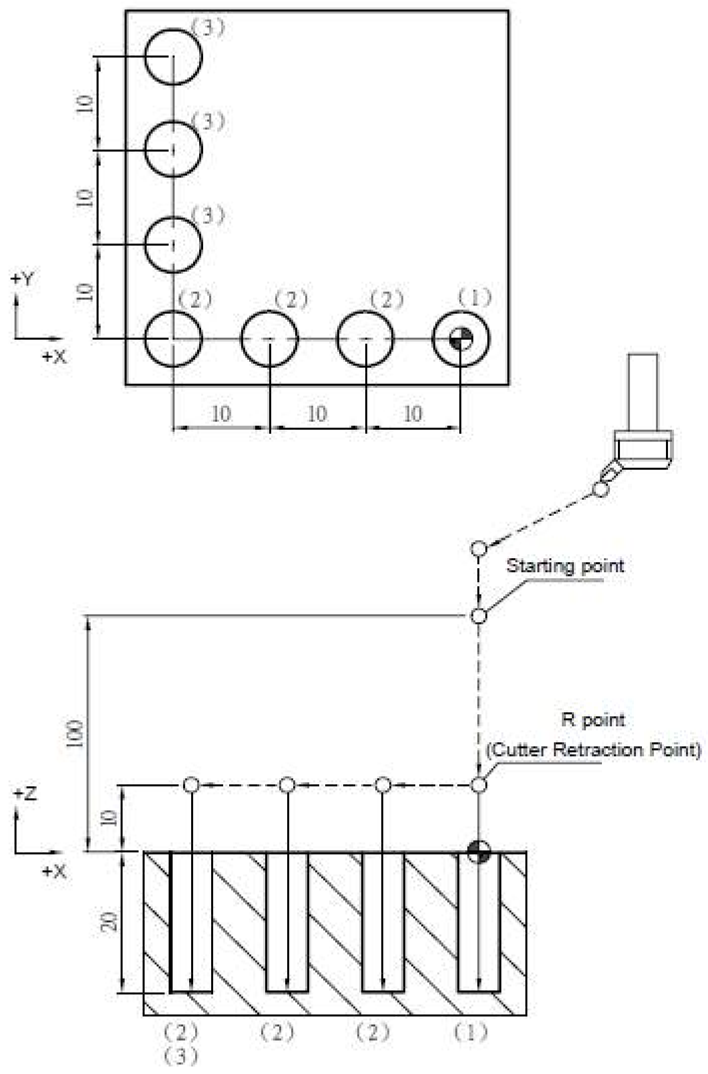
Program example:

```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G87 X0. Y0. Z-30. R10. P1000 Q5. K1 F100.; -----------------------------------------------------------------------(1)
X-15.; -------------------------------------------------------------------------------------------------------------------(2)
X-30.; -------------------------------------------------------------------------------------------------------------------(3)
X-30. Y15.;--------------------------------------------------------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```





```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G87 X0. Y0. Z-20. R10. P1000 Q5. K1 F100.; ------------------------------------------------------------------------(1)
G91 X-10. K3; --------------------------------------------------------------------------------------------------------------------(2)
Y10. K3; ---------------------------------------------------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



M03 S1000;

G17 G90 G00 G54 X0. Y0.;

G00 Z100.;

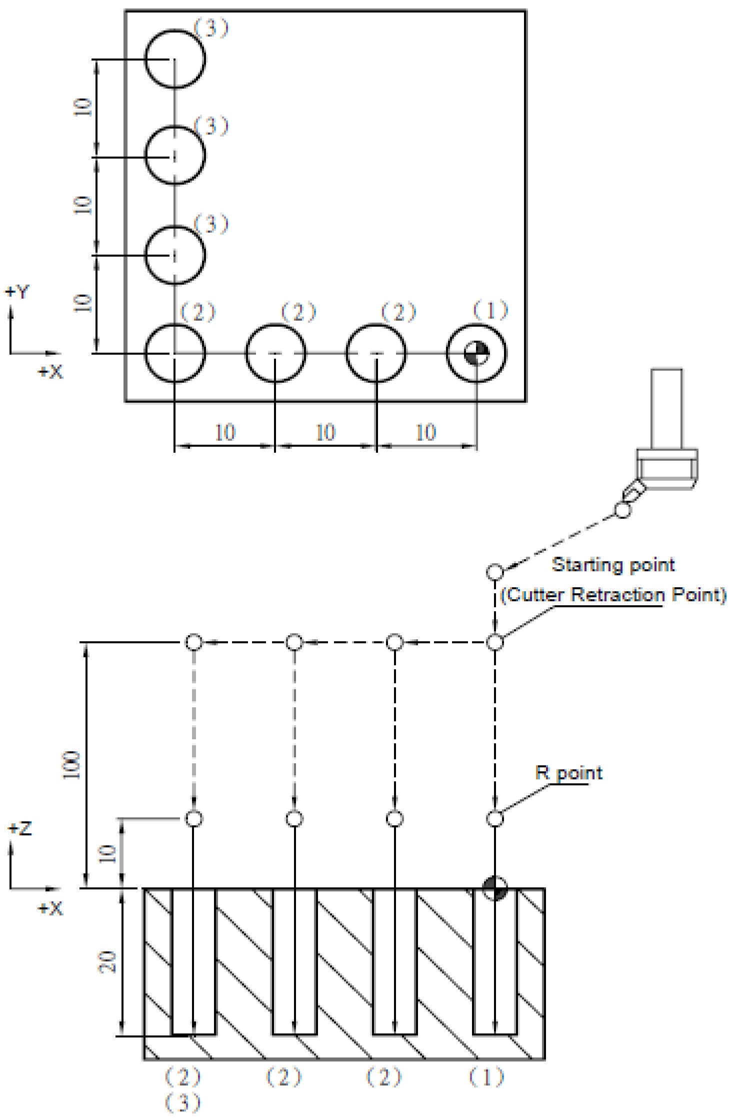
G98 G87 X0. Y0. Z-20. R10. P1000 Q5. K1 F100.; ---(1)

X-10. Z-30.; --(2)

X20. Z-40.; ---(3)

G80 G91 G28 X0. Y0 Z0.;

M05;



G88 Boring cycle

Command format:

G88 X__ Y__ Z__ R__ P__ K__ F__;

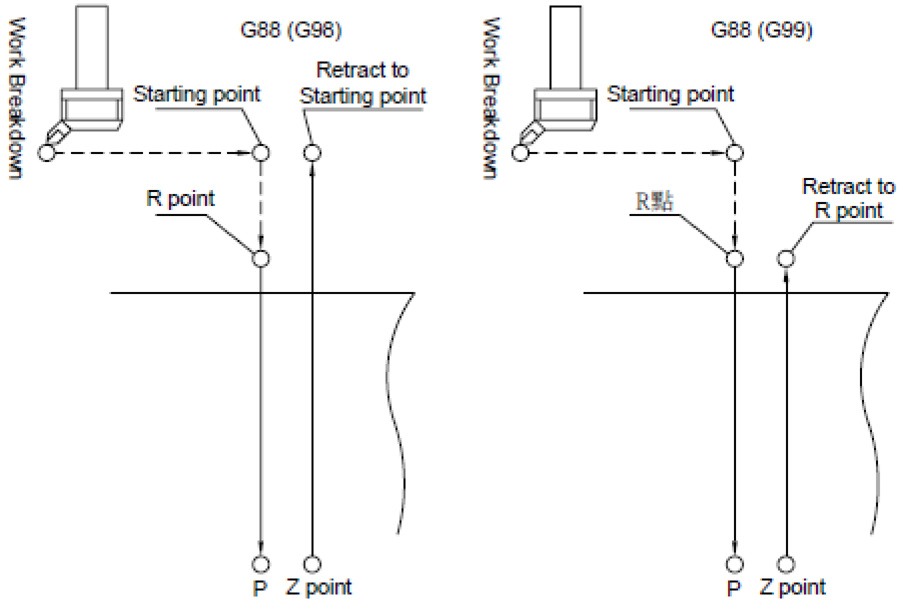
Argument description:

X__Y__	:	Coordinates of the hole position (mm).
Z__	:	Coordinates of the hole base (mm).
R__	:	Coordinate value of R point (which is the reset point) (mm).
P__	:	Pause time at hole base (1/1000 second), minimum unit, no decimal point allowed.
K__	:	Number of repetitions.
F__	:	Cutting feed rate (G94 mm/min).

Operation description (using the G17 plane as an example):

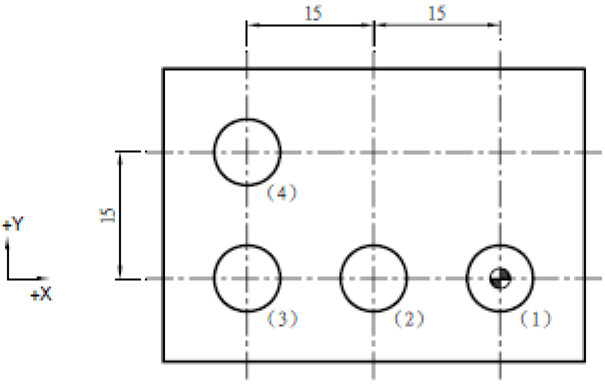
1. It is rapidly positioned to reach the hole location (X, Y, while maintaining the original tool height);
2. It is rapidly positioned to reach the R point coordinate (R);
3. Cutting should reach the hole base position based on a set cutting feed rate and spindle rotation speed (Z);
4. If P is assigned, it is the pause period at the hole base position;
5. Spindle rotation is stopped;
6. Under G98 mode, it is retracted to the reference point based on the cutting feed rate; in G99 mode, it is retracted to the R point based on the cutting feed rate;
7. Spindle in forward rotation;
8. When K is assigned (> 1), steps 2 to 7 mentioned above should be repeated until completing the specified number of repeated boring; otherwise the program terminates;
9. In G91 mode, argument R is for assigning the distance between R point and the reference point; argument Z is for assigning the distance between the hole base position and R point; when K is being assigned (> 1), after completion of each boring operation (steps 2 to 7 mentioned above), a hole position incremental offset will be implemented in accordance with the assigned X, Y before continuing with the next boring operation.
10. The difference between G86 and G88 is that the latter can be used to assign the pause time at hole base.

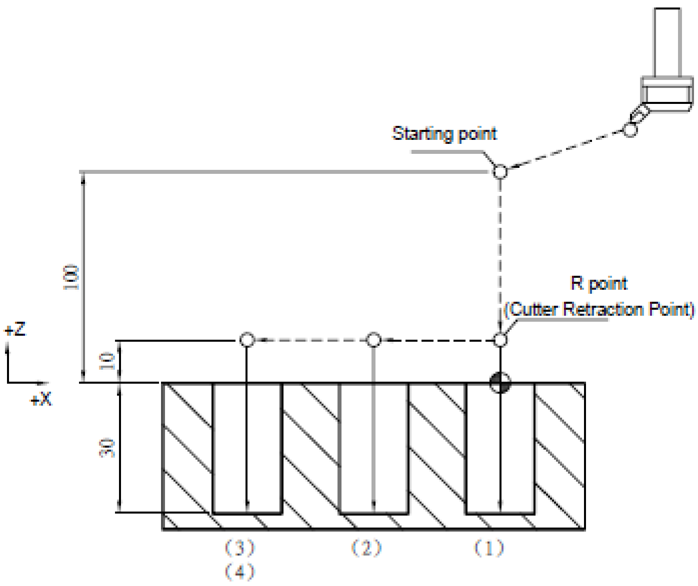
Plot legends:



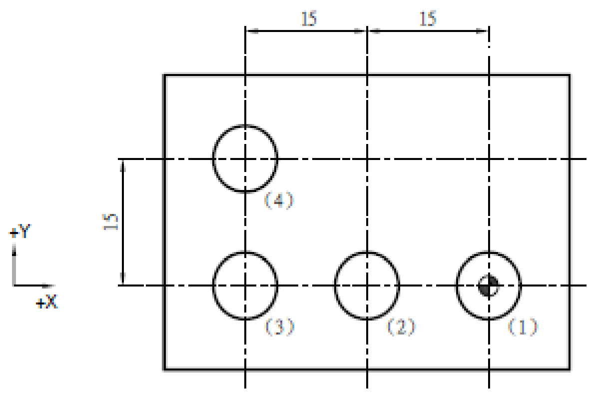
Program example:

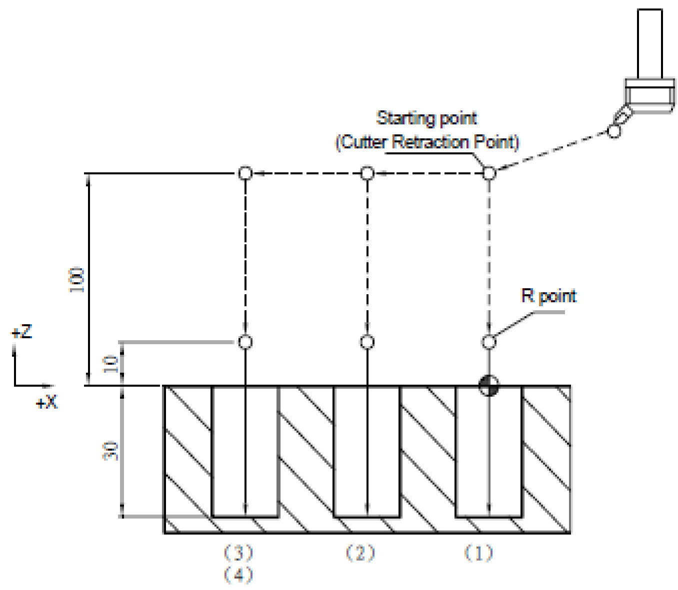
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G88 X0. Y0. Z-30. R10. P1000 K1 F100.;-----------------------------------------------------------------------------(1)
X-15.; ------------------------------------------------------------------------------------------------------------------------(2)
X-30.; ------------------------------------------------------------------------------------------------------------------------(3)
X-30. Y15.;------------------------------------------------------------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



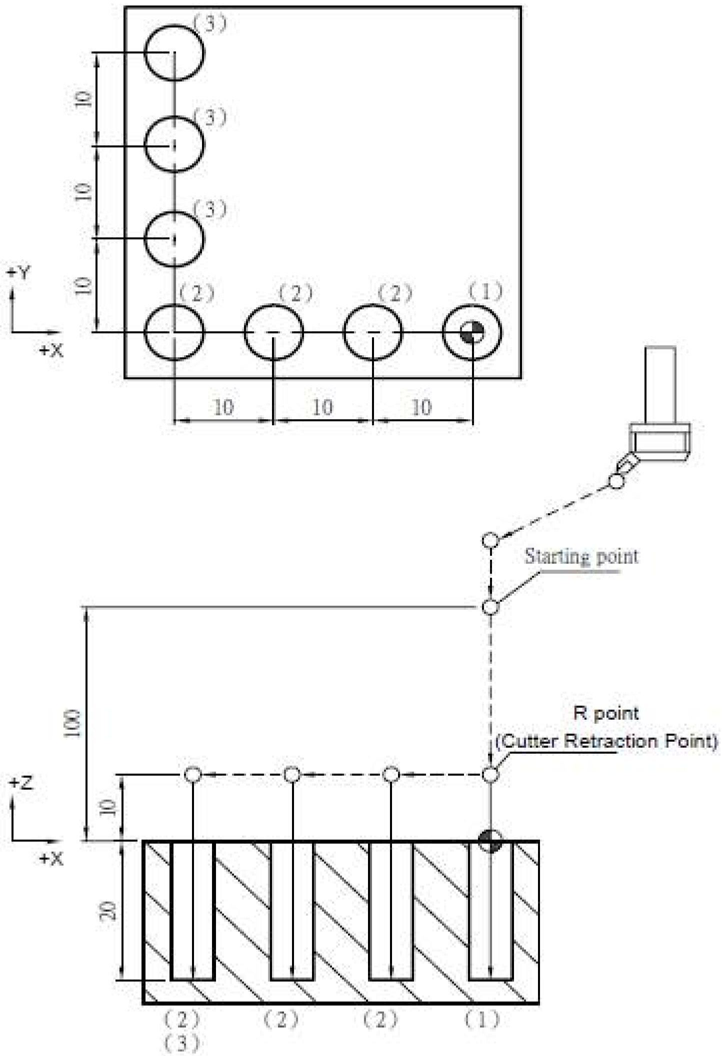


```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G88 X0. Y0. Z-30. R10. P1000 K1 F100.;--------------------------------------(1)
X-15.; ---------------------------------------------------------------------------(2)
X-30.; ---------------------------------------------------------------------------(3)
X-30. Y15.;-----------------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```

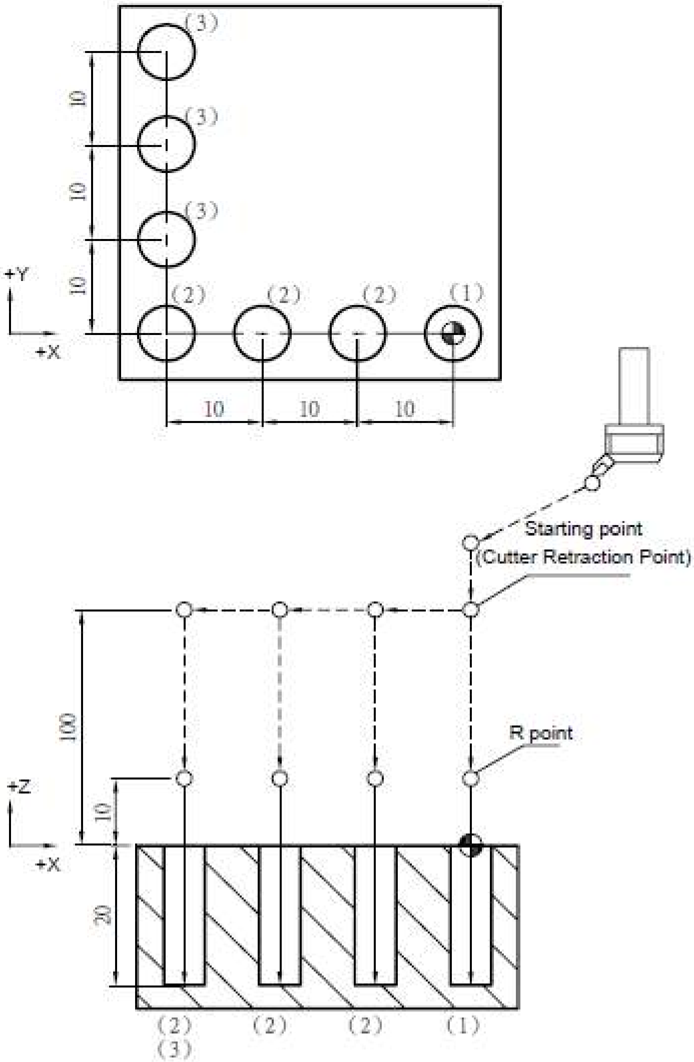




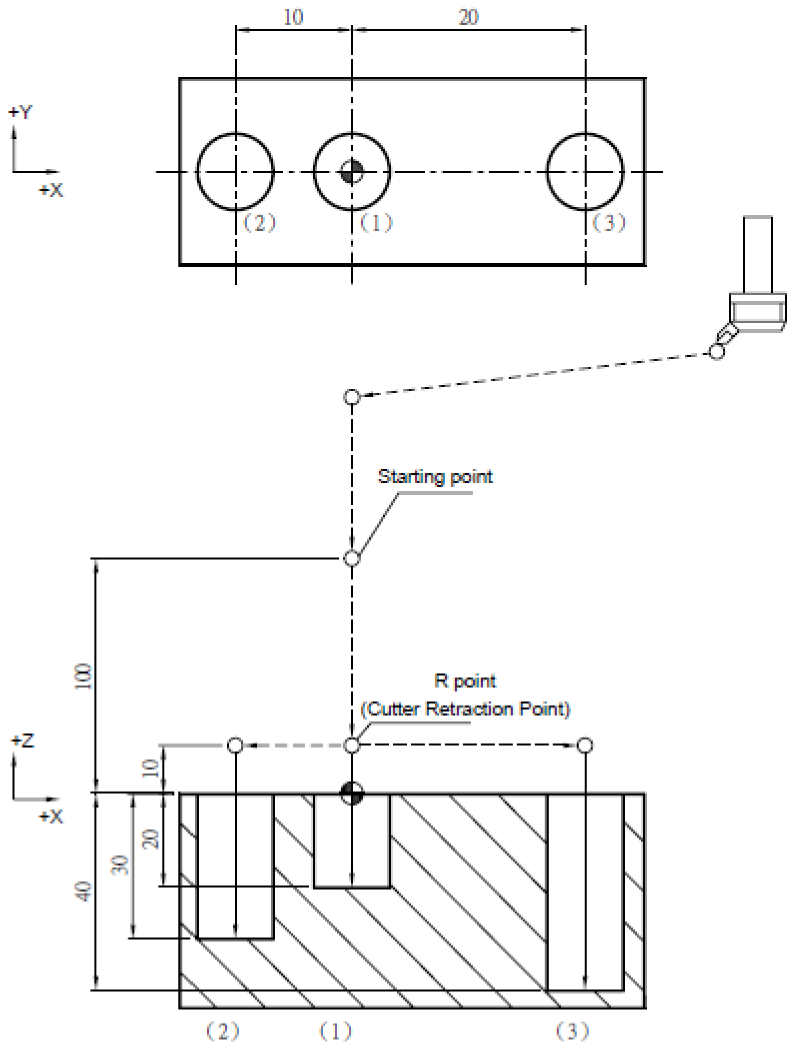
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G88 X0. Y0. Z-20. R10. P1000 K1 F100.;-----------------------------------------(1)
G91 X-10. K3; -----------------------------------------------------------------------(2)
Y10. K3; ----------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



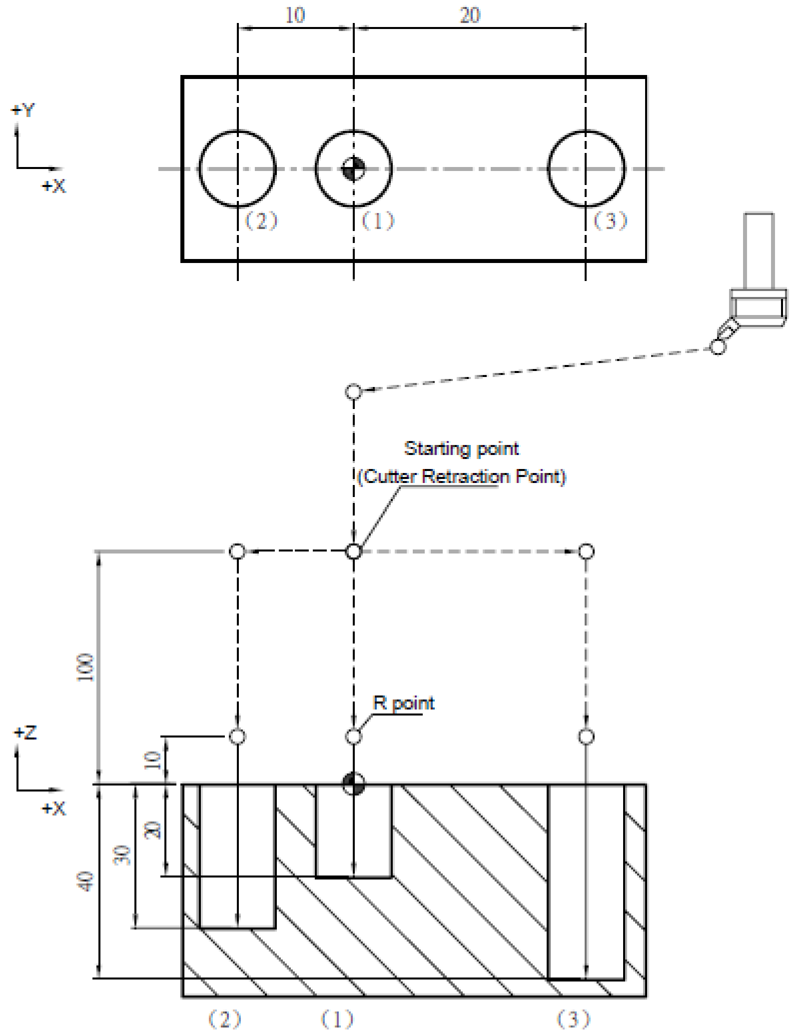
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G88 X0. Y0. Z-20. R10. P1000 K1 F100.;------------------------------(1)
G91 X-10. K3; ----------------------------------------------------------(2)
Y10. K3; ---------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G88 X0. Y0. Z-20. R10. P1000 K1 F100.;------------------------------(1)
X-10. Z-30.;-------------------------------------------------------------(2)
X20. Z-40.;--------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G88 X0. Y0. Z-20. R10. P1000 K1 F100.;------------------------------------------------------------------------------(1)
X-10. Z-30.;----------------------------------------------------------------------------------------------------------------(2)
X20. Z-40.;-----------------------------------------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



G89 Reaming cycle

Command format:

G89 X__ Y__ Z__ R__ P__ K__ F__;

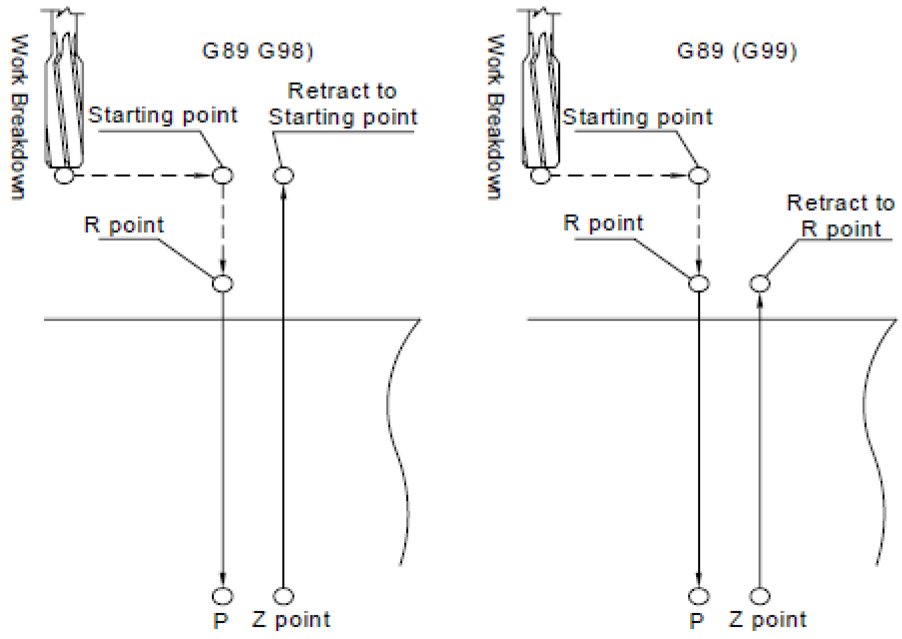
Argument description:

X__Y__	:	Coordinates of the hole position (mm).
Z__	:	Coordinates of the hole base (mm).
R__	:	Coordinate value of R point (which is the reset point) (mm).
P__	:	Pause time at hole base (1/1000 second), minimum unit, no decimal point allowed.
K__	:	Number of repetitions.
F__	:	Cutting feed rate (G94 mm/min).

Operation description (using the G17 plane as an example):

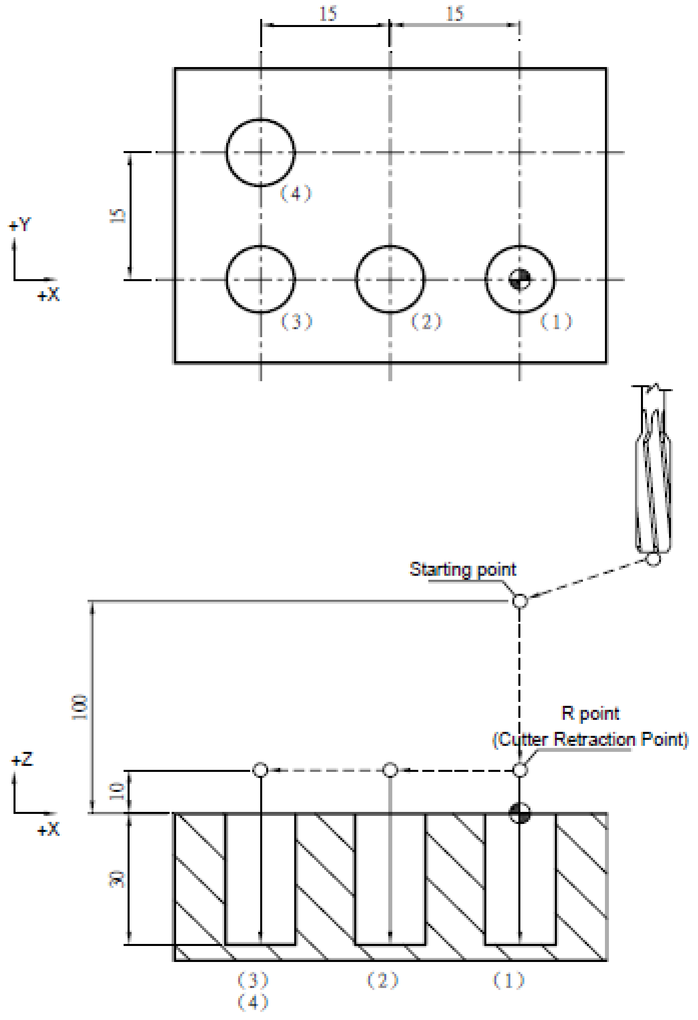
1. It is rapidly positioned to reach the hole location (X, Y, while maintaining the original tool height);
2. It is rapidly positioned to reach the R point coordinate (R);
3. Cutting should reach the hole base position based on a set cutting feed rate and spindle rotation speed (Z);
4. If P is assigned, it is the pause period at the hole base position;
5. Based on a set cutting feed rate and spindle turning velocity, it will be retracted to R point;
6. Under G98 mode, it is rapidly retracted to the reference point.
7. When K is assigned (> 1), steps 2 to 5 mentioned above should be repeated until completing the specified number of repeated reaming; otherwise the program terminates;
8. In G91 mode, argument R is for assigning the distance between R point and the reference point; argument Z is for assigning the distance between the hole base position and R point; If K is assigned (> 1), after executing a reaming operation (steps 2 to 5 mentioned above), a hole position offset will be implemented in accordance with the assigned X, Y before continuing with the next reaming operation.
9. The difference between G85 and G89 is that the latter can be used to assign the pause time at hole base.

Plot legends:

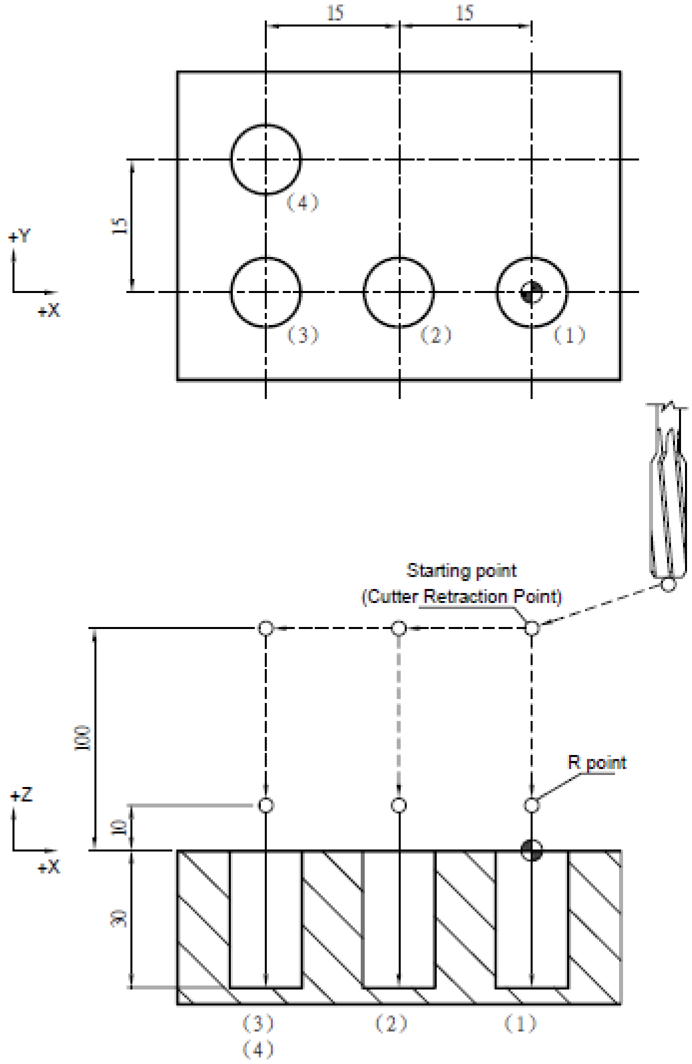


Program example:

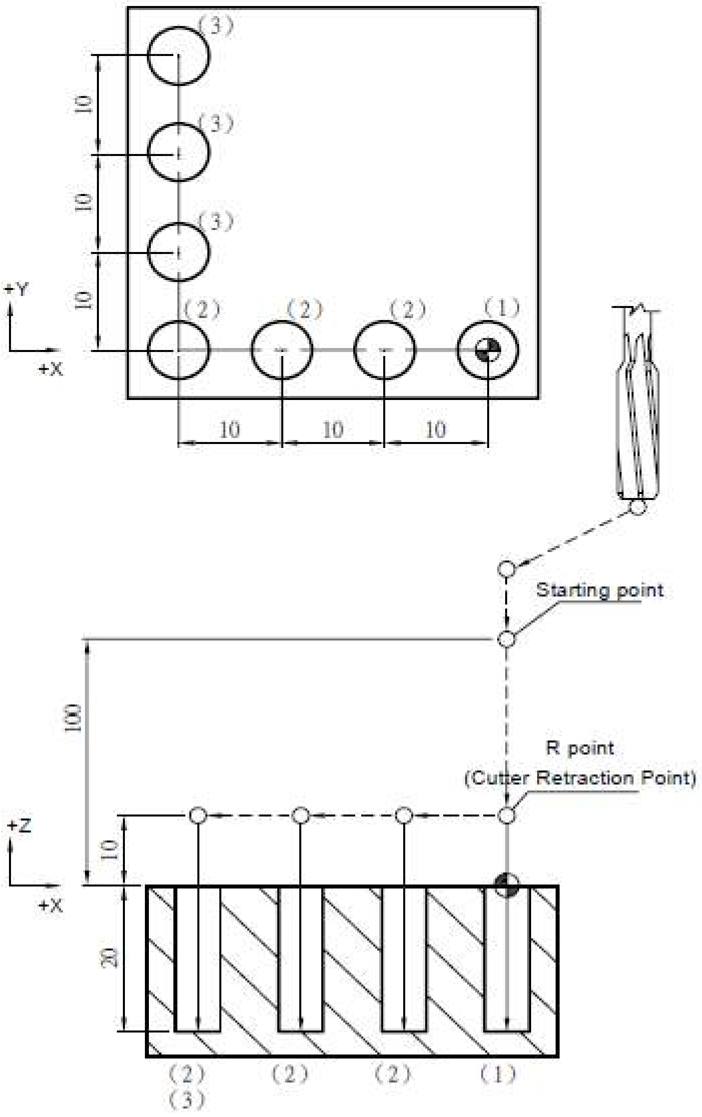
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G82 X0. Y0. Z-30. R10. P1000 K1 F100.;-----------------------------------------------------------------------------(1)
X-15.; -------------------------------------------------------------------------------------------------------------------(2)
X-30.; -------------------------------------------------------------------------------------------------------------------(3)
X-30. Y15.;----------------------------------------------------------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



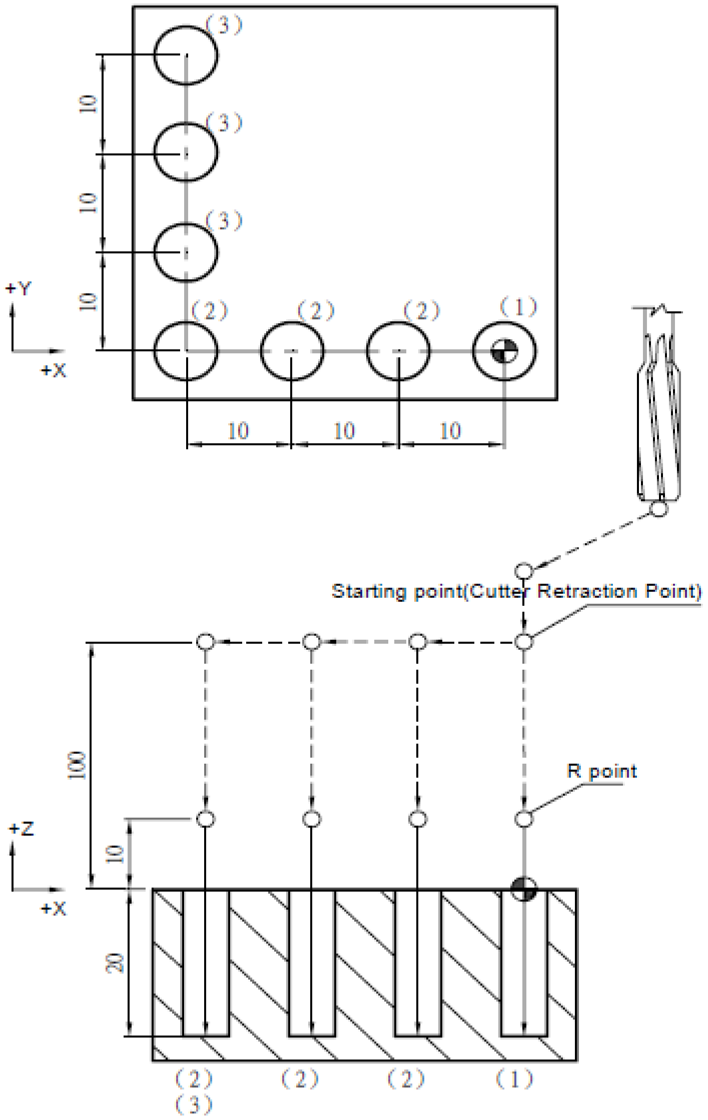
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G82 X0. Y0. Z-30. R10. P1000 K1 F100.;---------------------------------------------------------------------------(1)
X-15.; ---------------------------------------------------------------------------------------------------------------(2)
X-30.; ---------------------------------------------------------------------------------------------------------------(3)
X-30. Y15.;-----------------------------------------------------------------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



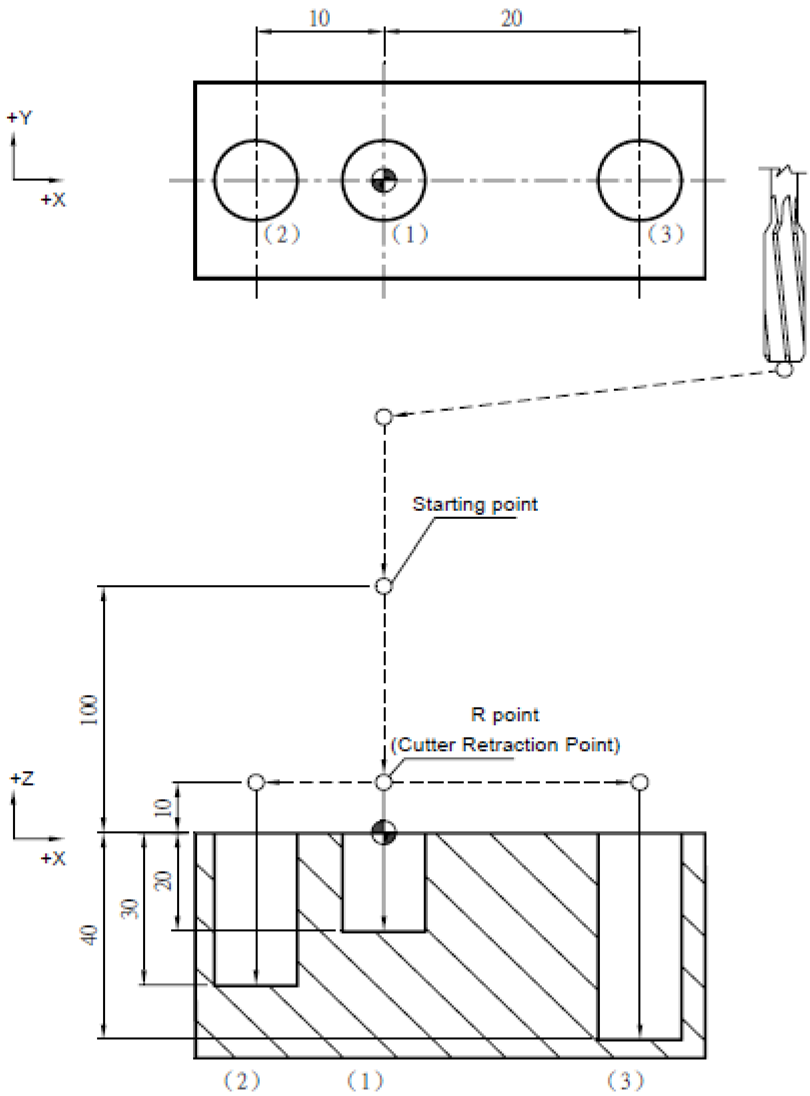
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G82 X0. Y0. Z-20. R10. P1000 K1 F100.;-------------------------------------------------------------------------------(1)
G91 X-10. K3; -------------------------------------------------------------------------------------------------------------(2)
Y10. K3; ------------------------------------------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



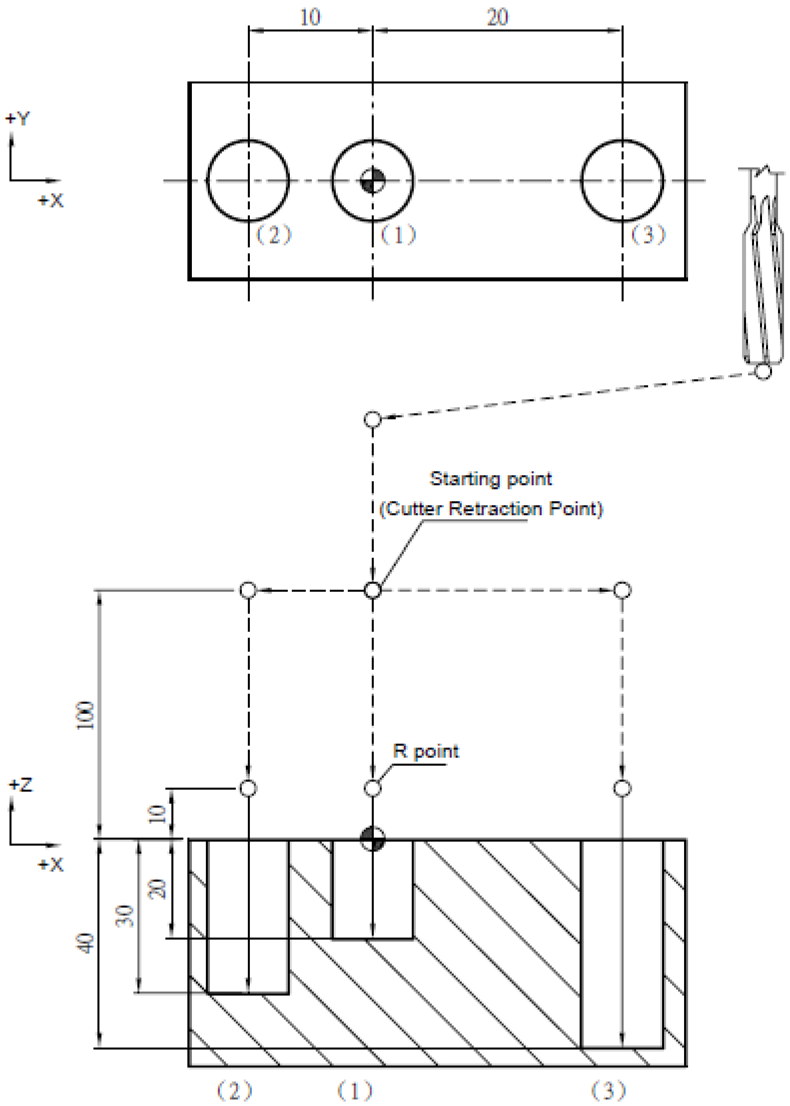
```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G82 X0. Y0. Z-20. R10. P1000 K1 F100.;----------------------------------(1)
G91 X-10. K3; ---------------------------------------------------------------(2)
Y10. K3; --------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G99 G82 X0. Y0. Z-20. R10. P1000 K1 F100.;------------------------------------------(1)
X-10. Z-30.;------------------------------------------------------------------------(2)
X20. Z-40.;-------------------------------------------------------------------------(3)
G91 G80 G28 X0. Y0. Z0.;
M05;
```



```
M03 S1000;
G17 G90 G00 G54 X0. Y0.;
G00 Z100.;
G98 G82 X0. Y0. Z-20. R10. P1000 K1 F100.;--------------------------------------------------------(1)
X-10. Z-30.;--------------------------------------------------------------------------------------(2)
X20. Z-40.;---------------------------------------------------------------------------------------(3)
G80 G91 G28 X0. Y0. Z0.;
M05;
```



G90, G91 Absolute, incremental mode

Command format:

```
G90;
G91;
```

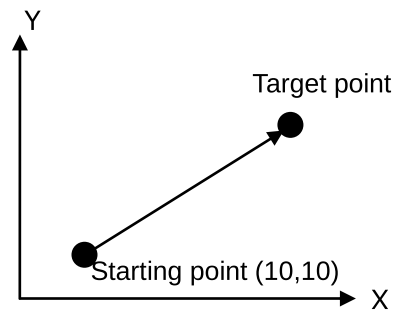
Argument description:

G90: Absolute mode - Under this state, the workpiece program assigns the coordinates of a target point.

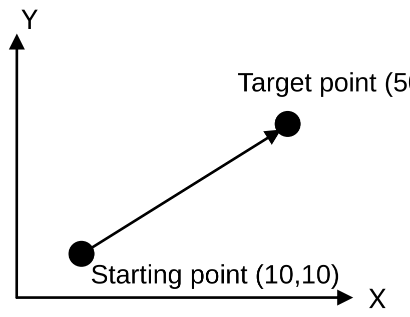
G91: Incremental mode - Under this state, the workpiece program assigns the traverse distance of a target point.

Program example:

G90 X50. Y40.;



G91 X40. Y30.;



G92 coordinate value setting

Command format:

G92 <Axis name><New coordinate value>;

Argument description:

Axis name : This is for assigning the name of axial direction to be set up, which can be any combination of X, Y, Z, A, B, C, or U, V, W. However, it must match the current axis name (axis name is set up by parameters 70464 to 70495).

New coordinate value : New coordinate value for the current tool position.

Operation description:

The G92 command can be used for setting the current position as the assigned coordinate value, and the offset amount between new and old coordinate values will affect all coordinate systems G54 to G59. Once G92 is set, the traversal command of absolute mode is calculated based on the coordinate system after offset. The offset amount set by G92 will be canceled after system RESET.

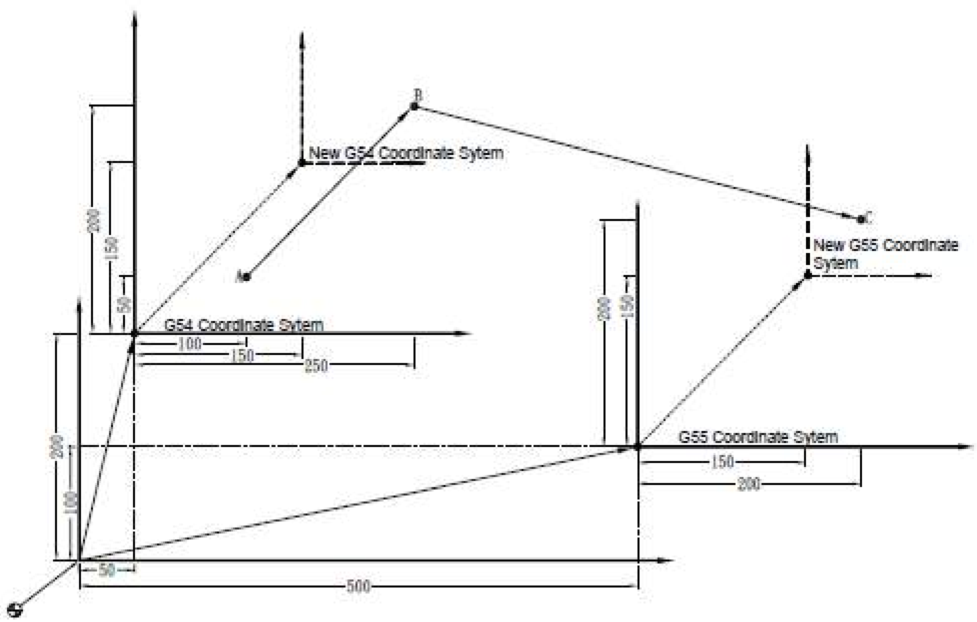
Program example:

G90 G54 G00 X100. Y50.; --(Point A)

G92 X-50. Y-100.; ------------------(Causing the offset of reference points of all coordinate systems (150,150))

X100. Y50.; --- (A→B)

G55 G00 X50. Y50.; ---(B→C)



G94, G95 Feed rate per minute, feed rate per revolution

Command format:

```
G94 F___;
G95 F___;
```

Argument description:

G94	:	Feed rate per minute, unit: mm/min or inch/min.
G95	:	Feed per revolution, unit: mm/rev or inch/rev.

Operation description:

It is for setting the unit of F code assigned in the cutting feed command (G01/G02/G03/G31).

G98, G99 return point settings

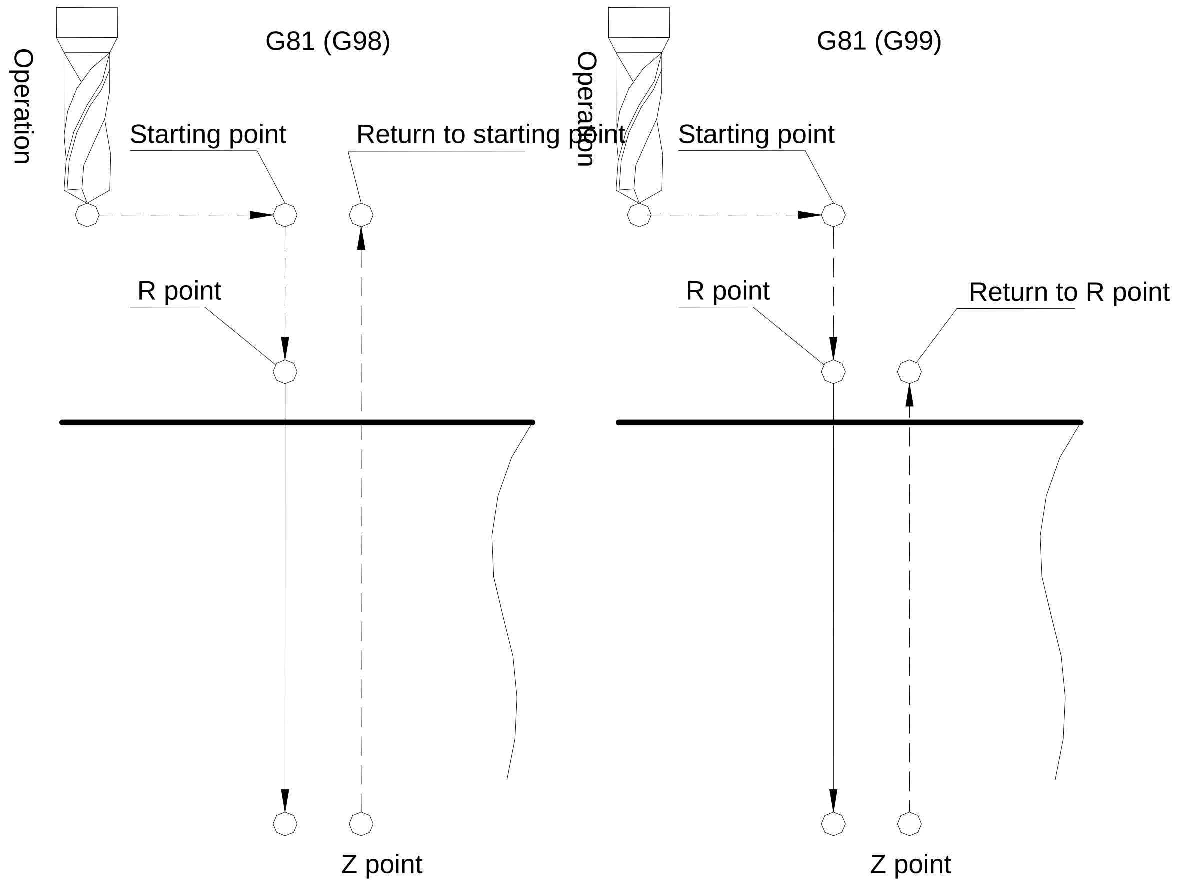
Command format:

G98;
G99;

Argument description:

G98	:	Set the return point as the reference point under the fixed cycle cutting mode (CANNED CYCLE).
G99	:	Set the return point as the R point under the fixed cycle cutting mode (CANNED CYCLE).

Plot legends:



4 Instructions for using the supplementary function (M code)

The supplementary function is used for controlling the machine's ON and OFF functions. The command format is the M code followed by a one or two digit number. The next M code to be introduced is a built-in supplementary code in the controller with fixed functions which are not determined by the machine maker. This type of M code includes M00, M01, M02, M30, M98, and M99. In other words, these functions have nothing to do with the composition of the LADDER program.

(1) M00: Program pause

When the CNC executes to the M00 command, program execution will be paused in order to allow the operator to perform size inspections and take corrective measures; the program start key 〈CYCLE START〉 must be pressed again for the program to resume.

(2) M01: Optional program pause

The M01 function is similar to M00; however, M01 is controlled by the 〈Optional pause〉 key on the panel: When the indicator light is ON and program execution reaches M01, the program will be paused; when the indicator light is OFF, M01 will be invalid.

(3) M02: Program termination

When the CNC executes to this command, machining will end. To resume program execution, the 〈RESET〉 key must be pressed before pressing the 〈CYCLE START〉 key to restart the machining process.

(4) M30: Program terminates and the cursor returns to the beginning

This function is for ending the program, which is the same as M02. However, the cursor in the code checking page will return to the beginning of the program.

(5) M98: Subprogram call

Command format 1:

```
M98 P__ L__ H__;
```

Command format 2:

```
M98 "String" L__ H__;
```

Command format 3:

```
M98 "String" P__ L__ H__;
```

Command format 4:

```
M98 L__ H__;
```

Argument description:

Function 1: Use the P argument to assign the name of a subprogram.

P__	:	The number of the subprogram to be called (name of the subprogram without the 4-digit number behind character "O")
L__	:	Default value for the number of repetitions is 1 if a value was not entered.
H__	:	Jump to serial number. Execution will start from a specified serial number when a subprogram is called.

Function 2: LNC advanced usage, a subprogram name is assigned by using strings.

"String"	:	An arbitrary string can be assigned, but the string length must not exceed 32 characters otherwise it will trigger system alert 【510301 - Name of the program to be called is invalid】.
L__	:	Default value for the number of repetitions is 1 if a value was not entered.
H__	:	Jump to serial number. Execution will start from a specified serial number when a subprogram is called.

Function 3: LNC advanced usage, a subprogram name is assigned by a combination of strings and the P argument.

"String"	:	An arbitrary string can be assigned, but the string length must not exceed 28 characters otherwise it will trigger system alert 【510301 - Name of the program to be called is invalid】.

P__	:	The number of the subprogram name to be assembled (macro name without the 4-digit number behind the "string").
L__	:	Default value for the number of repetitions is 1 if a value was not entered.
H__	:	Jump to serial number. Execution will start from a specified serial number when a subprogram is called.

Function 4: No subprogram name has been assigned.

L__	:	Default value for the number of repetitions is 1 if a value was not entered.
H__	:	Jump to serial number. Execution will start from a specified serial number when a subprogram is called.

When no subprogram name has been assigned, the subprogram called will be the file itself. Usually, this method can be used in coordination with H_ to jump to the line number to execute a certain section of the program and then return. As shown in the example figure below.

```
G90 G0 X0 Y0
G0 X10
M98 H10
G0 X20
M30

N10 G0 Y10
G0 Y20
M99
```

The difference between calling a macro (G65) and calling a general subprogram (M98):

3. M98 cannot be used to assign arguments; the G65 command can be used to assign arguments.
4. M98's local variable levels are fixed; local variables of G65 are changing in accordance with the nested depth (for example, the meaning of #1 is the same before and after M98, but it is different for G65).

The combination of M98 calling level and G65, G66 can be as many as 6 levels; the calling level of G65, G66 can be as many as 4 levels.

(6) M99: Returning to the main program after a subprogram is finished

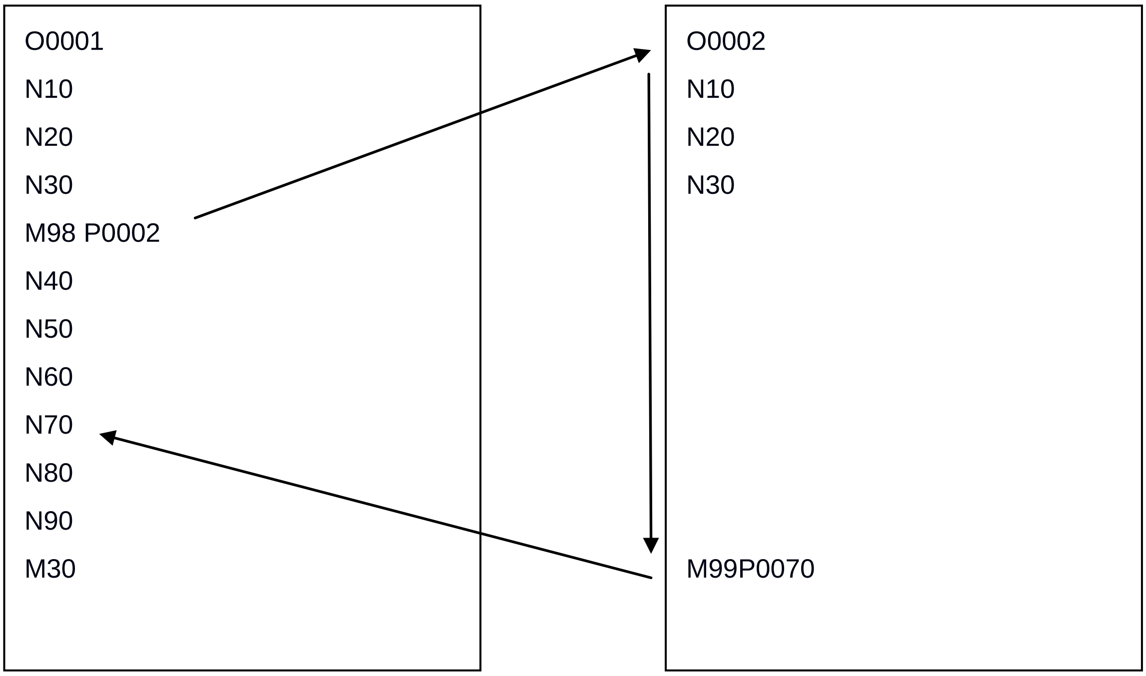
1. After NC has executed to M99 in the main program, it will return to the front of the program and restart program execution. M99 must serve as the program termination in the subprogram, and program execution must return to the main program.
2. **Command format:** M99 P__ ;

   P__: Specify serial number upon return

   If M99 P_ is used in the main program, the system will look for the serial number assigned by M99 and then start execution

```
N10
N20
M99 P0050
N30
N40
N50
N60
N70
N80
N90
M30
```

If M99 P is used at the end of a subprogram, then the execution of the main program will start at the serial number assigned by M99 after execution of the subprogram is completed.


O0001
N10
N20
N30
M98 P0002
N40
N50
N60
N70
N80
N90
M30
O0002
N10
N20
N30
M99P0070

The overview table of M code is shown below. In addition to M00, M01, M02, M30, M98, and M99, other functions on this table are designed by the LADDER diagram program; the functions of these M codes are not assigned by the system, so they may vary with respect to different machines. Users should verify the command specifications for their machine (functions listed in this table are factory standard LADDER version functions).

M code	Function		Remark
M00	Program stop	Program stop	CNC
M01	Optional stop	Optional stop	CNC
M02	End of program	End of program	CNC
M03	Spindle turning CW	Spindle CW	
M04	Spindle turning CCW	Spindle CCW	
M05	Spindle stop	Spindle stop	
M08	Coolant ON	Coolant ON	
M09	Coolant OFF	Coolant OFF	
M30	Terminate program & restart	Program rewind	CNC
M98	Calling a subroutine	Calling of subprogram	CNC
M99	Returning from a subroutine	End of subprogram	CNC

Table 4-1