Delta CNC Solution

NC300 Milling Machine Controller
G Command Guidelines

Delta CNC Solution — **NC300 Milling Machine Controller**

*We reserve the right to change the
  information in this catalogue without prior notice.

www.delta.com.tw/ia

Table of Contents

3 M Codes

4 Macro and Variables

Table of G Codes

1

This chapter lists the G codes supported by NC controller.

1.1 Milling machine G code supporting table

G code	Group	Function description
G00	01	Fast positioning command
G01	01	Linear cutting command
G02	01	Clockwise arc cutting (CW)
G03	01	Counterclockwise arc cutting (CCW)
G04	00	Pause command
G09	00	Exact stop
G10	00	Data entry setup
G11	00	Data entry setup cancelling
G15	16	Polar coordinates cancelling
G16	16	Polar coordinates
G17	02	X-Y plane selection
G18	02	Z-X plane selection
G19	02	Y-Z plane selection
G20	06	Inch input
G21	06	Metric input
G24	17	Mirror image setup
G25	17	Mirror image setup cancelling
G28	00	Homing through the reference origin point
G29	00	Homing to the starting point
G30	00	Auto homing of the second, third, and fourth reference point
G31	00	Skip function
G40	07	Tool radius compensation cancelling
G41	07	Tool radius left compensation
G42	07	Tool radius right compensation
G43	08	Tool length positive direction compensation
G44	08	Tool length negative direction compensation
G49	08	Tool length compensation cancelling
G50	11	Scale cutting cancelling
G51	11	Scale cutting
G52	00	Local coordinate system setup

(Continued)

G code	Group	Function description
G53	00	Mechanical coordinate system setup
G54	12	The first machining coordinate system selection
G55	12	The second machining coordinate system selection
G56	12	The third machining coordinate system selection
G57	12	The fourth machining coordinate system selection
G58	12	The fifth machining coordinate system selection
G59	12	The sixth machining coordinate system selection
G61	13	Exact stop mode
G64	13	Cutting mode
G65	00	Non-continuous effect macro command calling
G66	14	Continuous effect macro command calling
G67	14	Continuous effect macro command calling cancelling
G68	15	Coordinate system rotation command
G69	15	Coordinate system rotation command cancelling
G73	09	Peck drilling cycle
G74	09	Left spiral tapping cycle
G76	09	Fine boring cycle
G80	09	Constant loop cancelling
G81	09	Drilling cycle
G82	09	Countersunk drilling cycle
G83	09	Deep hole peck drilling cycle
G84	09	Right spiral tapping cycle
G85	09	Broaching cycle
G86	09	Rough boring cycle
G87	09	Rear boring cycle
G88	09	Boring cycle
G89	09	Boring cycle
G90	03	Absolute coordinate value system
G91	03	Incremental coordinate value system
G92	00	Coordinate system setup
G94	05	Feed rate setup (mm/min)
G98	10	Return to the initial point of the fixed cycle
G99	10	Return to the R point of the fixed cycle

1

(This page is intentionally left blank.)

1

G Code Description

<div style="text-align: right; font-size: 3em;">2</div>

This chapter includes G code format, its application example, and detailed descriptions.

2

G00: Fast positioning command

Command format: G00 X_Y_Z_ (This command moves all three spindles, any two or one of them as desired.)

X_Y_Z_: End point coordinates.

Command description: A G00 command moves the cutter center to position given by coordinates X, Y, and Z at fast speed. Movement speed is set by the Rapid Feed Rate key of the secondary control panel but not the F_commands. If the X, Y, and Z spindles' maximum movement speed is set at 15m/min as specified in the maximum movement speed parameter, then the actual movement speed under different Rapid Feed Rate key settings should be:
(1) 15 m/min, the maximum speed when the setup value is 100%;
(2) 7.5 m/min when the setup value is 50%;
(3) 3.75 m/min when the setup value is 25%;
(4) Spindles move at speed set by parameter values when the setup value is 0%.

The G00 command positions the spindle rather than feeds the cutting. For example, it quickly moves the spindle from mechanical origin to cutting origin and positions the X and Y axis after cutting.

[Illustrations]

See figure below for moving cutters from point A to point B at fast speed.

Expressed in absolute values: G90 G00 X92. Y35.
Expressed in incremental values: G91 G00 X62. Y-25.
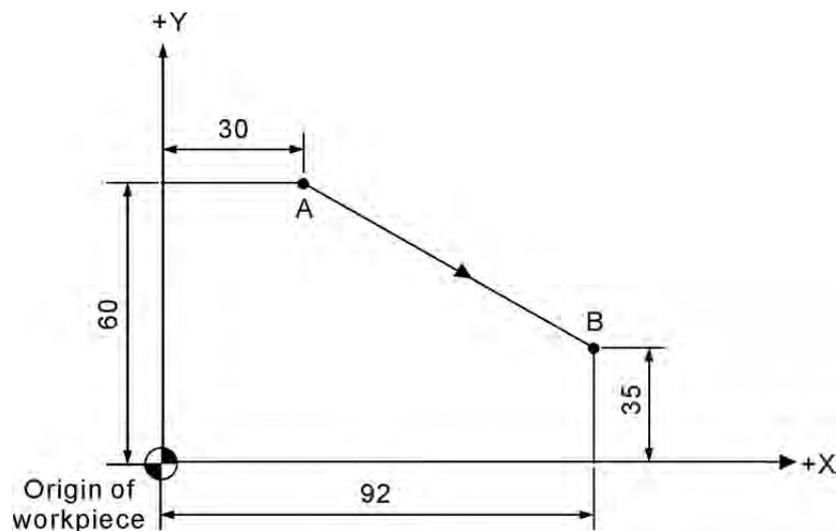
G01: Linear Cutting Command

Command format: G01 X_Y_Z_F_

X_Y_Z_: End point coordinates.

F_: Feed rate in unit of mm/min

Command description: This command enables a cutter to linear cut from the current position to a given position at F feed rate. The ending position of the cutting is given by the X, Y, and Z coordinates. You can have all three spindles, any two or one of them as desired. The feed rate is set by the F parameter along with the optional Feed Rate of the secondary control panel in unit of mm/ min.

[Illustrations]

Regarding the method of G01, see figure below for milling in direction of +Y from program origin.
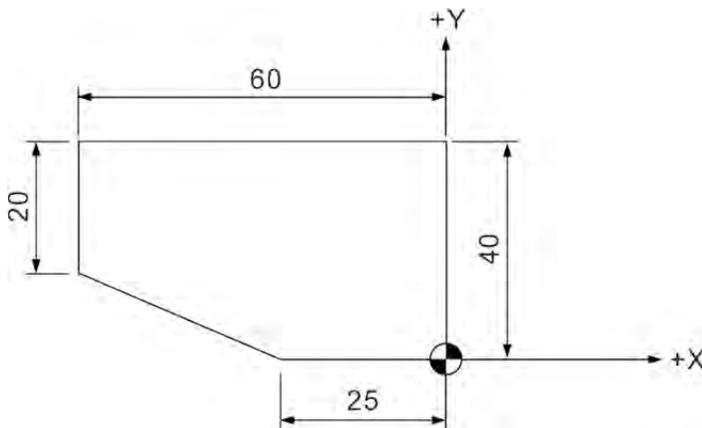

```
G00 G90 G54 X0 Y0
G90 G01 Y40.0 F80
X-60.0
G91 Y-20.0
X35.0 Y-20.0
G90 X0 Y0
```

The F parameter remains active unless being changed as shown in program codes listed above where the F parameters are omitted in subsequent statements.

G02/G03: Arc Cutting Command

2

Command format:
Arcs in the X – Y plane
G17 G02 (or G03) X_ Y_ R_ F_ or
G17 G02 (or G03) X_ Y_ I_ J_ F_
You may add parameter Z_ in the command to spiral move in the X – Y plane.

Arcs in the Z – X plane
G18 G02 (or G03) Z_ X_ R_ F_ or
G18 G02 (or G03) Z_ X_ K_ I_ F_
You may add parameter Y_ in the command to spiral move in the Z – X plane.

Arcs in the Y – Z plane
G19 G02 (or G03) Y_ Z_ R_ F_ or
G19 G02 (or G03) Y_ Z_ J_ K_ F_
You may add parameter X_ in the command to spiral move in the Y – Z plane.

G02: Clockwise (CW) arc cutting.
G03: Counterclockwise (CCW) arc cutting.
X, Y, and Z: Coordinates of the end point in absolute or incremental values determined by command G90 and G91.
R: Arc radius (expressed in Radius format).
I: The X-axis distance between arc center and starting point. The increment amount from the starting point to arc center.
J: The Y-axis distance between arc center and starting point. The increment amount from the starting point to arc center.
K: The Z-axis distance between arc center and starting point. The increment amount from the starting point to arc center.
(The I, J, and K expression is also known as the center method.)
F: The cut feeding rate in unit of mm/min.

Command description: G02 or G03 is the arc cutting command. The arc cutting direction (G02 or G03) for a three-dimensional workpiece in individual planes is shown in Figure 1. The cutting direction is defined by the right hand coordinate system. Look against the positive direction of the plane's vertical axis; G02 is for clockwise direction while G03 is for counterclockwise.

X-Y plane   Z-X plane   Y-Z plane
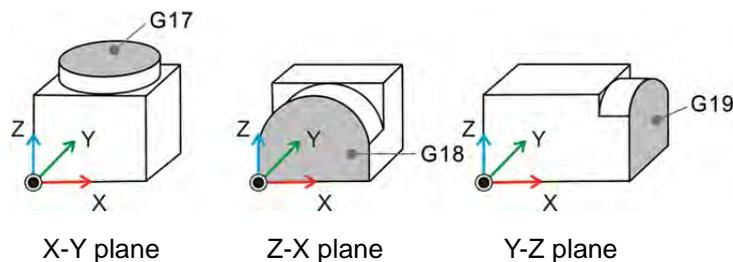
The center and radius methods are described below:

1.   Radius method: R is the radius of the arc and shows in radius value. This method represents an arc by its starting point, ending point, and arc radius. There are two equivalent arcs in a circle as shown in the figure below. For the positive R value, it is an arc of central angle ≤ 180° and an arc of central angle > 180° for the negative R value.
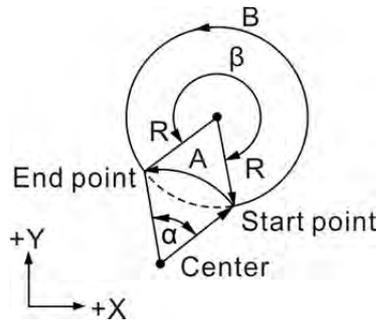
The radius method

Note: β > 180° -> the B arc with negative R; α ≤ 180° -> the A arc with positive R

In the figure above if R = 50 mm and the absolute coordinates of the end point are (100.0, 80.0), then,

(1) the arc of central angle > 180° (path B) can be expressed as "G90 G03 X100.0 Y80.0 R-50.0 F80" and;

(2) the arc of central angle ≤180° (path A) can be expressed as "G90 G03 X100.0 Y80.0 R50.0 F80".

2.   The Center method: Here parameters I, J, and K define the relative distance from the arc start point to the arc center (the end point). That is, the increments from the start point to the center in X, Y, and Z directions respectively. See the figure below for illustrations.
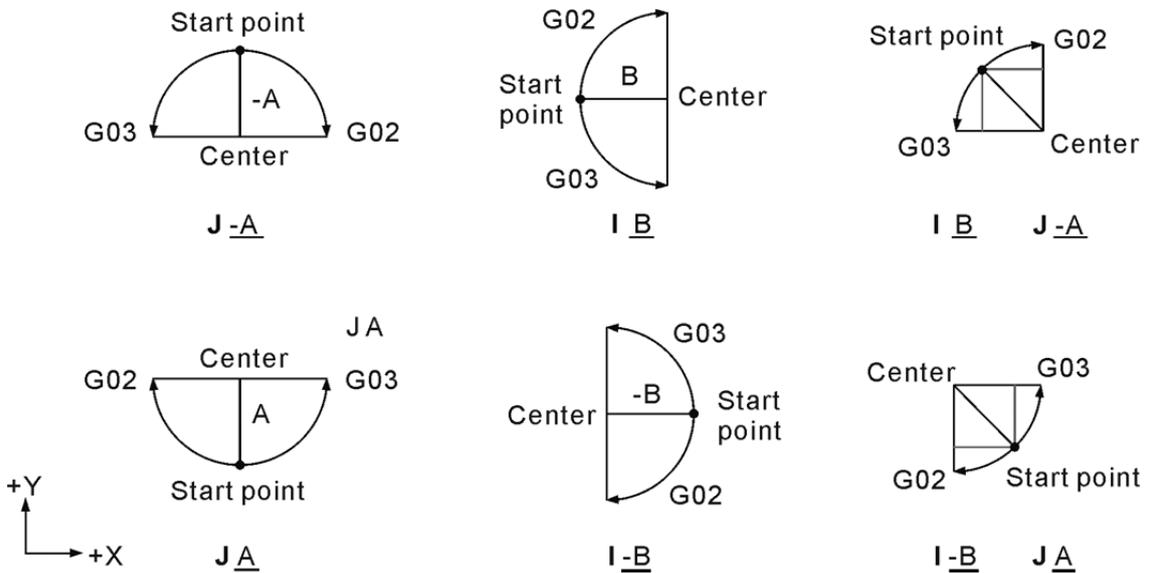
Values of parameters I and J on plane

The arc motion expressed in the radius or center method can be used in program coding for selecting conditions.

To cut milling a sphere you can use the center but not the radius method. Generating a sphere by connecting two semi-spheres bears too large a roundness deviation to be acceptable.

[Illustrations]
The command syntax for cut milling a sphere as shown in the figure below may look like "G02 I-50.0".
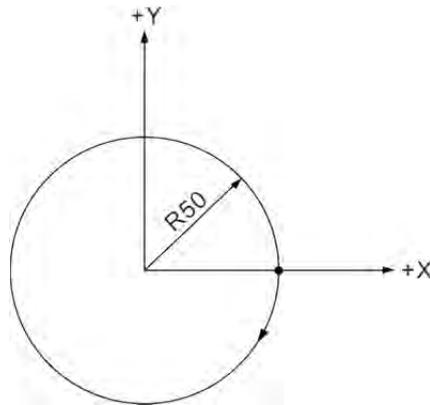
Cut milling a sphere

Use of commands G01, G02, and G03 is illustrated in the figure below. Here is the tool cut milling from program origin upward along the shape.
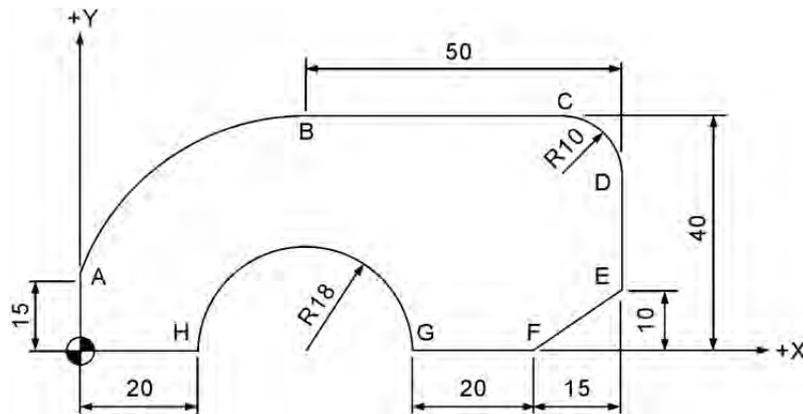

```
O0100
G90 G54 X0 Y0 S500 M3
G90 G01 Y15.0 F80                    > program origin → A
G02 X41.0 Y40.0 R41.0               > A → B
G91 G01 X40.0                        > B → C
G02 X10.0 Y-10.0 R10.0             > C → D
G01 Y-20.0                           > D → E
X-15.0 Y-10.0                        > E → F
X-20.                                > F → G
G90 G03 X20.0 R18.0                > G → H
G01 X0.0                             > H → program origin
```

Notes on G02 and G03 arc cutting:

(1)   Most systems default to G17 (X – Y plane) after power on. The G17 command can be omitted when cut milling arc in X – Y plane.

(2)   When I, J, and R parameters show in one statement only the R parameter remains active while I and J are ignored.

(3)   Parameter I0, J0, and K0 can be omitted.

(4)   When end point X, Y, and Z coordinates are omitted, it means the start and end points are the same and a sphere is to be cut and milled. The tool remains motionless for command in the radius method.

(5)   The system prompts an alarm message when the end point does not intersect with the given radius value.

(6)   For arc cutting following a linear cutting the G command must convert to command G02 or G03 and G01 for straight line cutting.

(7)   When arc cutting command (G02, G03) is not specified by R, I, J and K, its motion is the same as command G01.

G04: Pause Command

Command format: G04 X_ or
    G04 P_

Command description: This command sets up a pause of the current block. Both **X** and **P** parameters define the time of pause while **X** accepts decimal values and **P** does not.

Command scope:

Range of pause time by X parameter	
Valid values	Unit
0.001 ~ 99999.999	Seconds
Range of pause time by P parameter	
Range of valid values	Unit
1 ~ 99999999	0.001 second

[Illustrations]

G04 X1.5 or
G04 P1500

Both commands cause a pause span during program execution for 1.5 seconds.

G09: Exact Stop Command

Command format: G09 G01 X_ Y_

Command description: For tools operating in a constant feed rate, a minute round corner angle may be generated by continuous motion as the operation of the next block starts accelerating before the current one is completed. G09 command aims to eliminate the round angle at corners. With G09 in effect, the system is positioned before doing any block of motion. The next block is executed only after being positioned correctly. As a result, minor discontinuity exists between block s when G09 status is in effect. This improves precision at the cost of speed. This command applies to cutting commands (G01~G03) of the current block only.
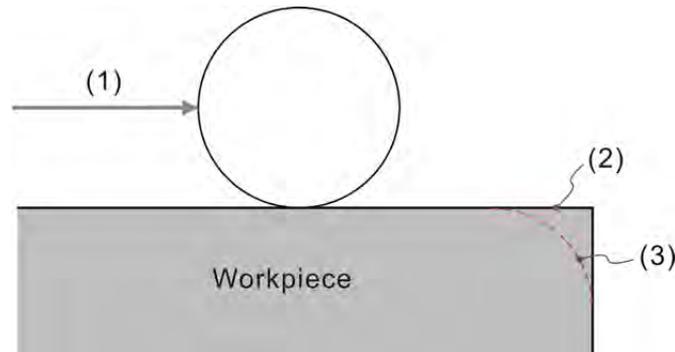
(1) Tool movement direction; (2) with G09 in use; (3) Without G09 in use

[Illustrations]

G09 G01 X100.0 F150   > Start executing the next block only after the deceleration is
                                 stopped and positioning is confirmed.

G01 Y-100

G10/G11: Data Entry Setup and Cancel

Command format: G10 L2 P_ X_ Y_ Z_
                G10 L10 P_ R_: Tool length compensation
                G10 L11 P_ R_: Tool length wear compensation
                G10 L12 P_ R_: Tool radius compensation
                G10 L13 P_ R_: Tool radius wear compensation
                G10 L20 P_ X_ Y_ Z_: Expansive workpiece coordinate system entry
                G10 L21 P_X_Y_Z_: Coordinate setting of software limit

Command description: The G10 command in the syntax of G10 L2 P_ X_ Y_ Z_ is used for workpiece coordinate system data entry. The system is set to the offset coordinate of the workpiece coordinate system when the value of parameter P is set to 0. Parameters P1~P6 correspond to G54 ~ G59 of the workpiece coordinate system while X, Y, and Z represent the origin of the given coordinates system. The P parameter in L20 command syntax can assign the values of P1 ~ P64 for the corresponding expansion workpiece coordinates system. The command format G10 L10 P_ R_ sets up the tool length compensation value where parameter P is the compensation number and R is the actual compensation value for tool radius and length. The setting of P1 from G10 L21 P is for the first forward software limit, P2 is for the first backward software limit, P3 is for the second forward software limit and P4 is for the second backward software limit.
In absolute status, the G10 command is the absolute input. It also can do data entry setup in incremental status.
In incremental status, the G10 command can input the value in incremental way.

2

[Illustrations]

G10 L10 P1 R-300.0 or
G10 L10 P2 R-100.0

Set length compensation value of Tool 1 to H-300.0 and H-100.0 for Tool 2.
P: Compensation number or workpiece coordinate number
R: Tool compensation value

The examples of absolute and incremental status are as followings:
G90 G10 L10 P1 R-300.0
Input the data to H-300.0 of Tool 1 in absolute way.

G91 G10 L10 P1 R-100.0
Input the data to Tool 1 and add H-100.0 to it in incremental way.

Note:
(1) The G10 command is non-continuous and so it is effective in a single command block. The compensation values of the offset coordinates and job coordinates system are given relative to the origin of the mechanical coordinates system. You can execute command G11 to cancel data entry settings.
(2) During program execution, the command coordinates data changed by L2 or L20 command takes effects in next motion block. The tool compensation data changed by commands L10 ~ L13 take effect only after running compensation commands (G41/G42 or G43/G44) with compensation data number (D or H) again.

G15: Polar Coordinates Command Cancelling

Command format: G15

Command description: Command G15 c cancels the status specified by the polar coordinates command and returns the program to move in the path of the original coordinates system and values.

2

G16: Polar Coordinates Command

Command syntax: G17 G16 X___ Y___ or;
                        G18 G16 Z___ X___ or;
                        G19 G16 Y___ Z___

> X__ Y__: In a G17 plane the X__ parameter specifies the radius and Y__ the angle coordinates of the polar coordinates system.
> Z__ X__: In a G18 plane the Z__ parameter specifies the radius and X__ the angle coordinates of the polar coordinates system.
> Y__ Z__: In a G19 plane the Y__ parameter specifies the radius and Z__ the angle coordinates of the polar coordinates system.

Command description: The polar coordinates command employs radius and angle as its setup format. If the first axis (X-axis) of the plane is selected for radius then the second one (Y-axis) is set for angle value. The counterclockwise angle assumes a positive value while the clockwise one assumes a negative value. When a radius specified by a negative value, the radius command value is regarded as the absolute one and the angle command value will add an additional 180 degrees. The angle values can be in absolute or incremental format.

[Illustrations]
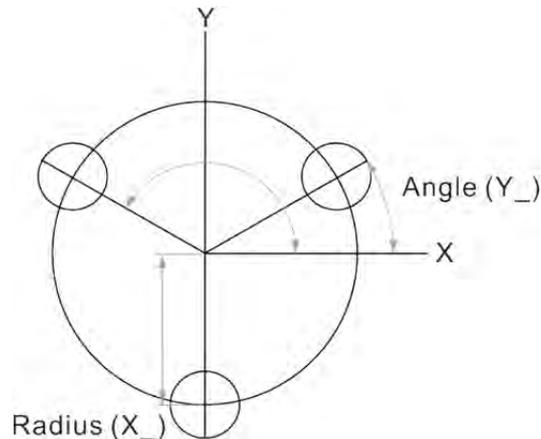
N1 G90 G16   > set up polar coordinates
N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0   > the command to loop drill at position of radius 100 and angle 30
N3 X100.0 Y150.0   > machine the second hole
N4 X100.0 Y270.0   > machine the third hole
N5 G15 G80   > cancel the polar coordinates function

2

G17/G18/G19: Plane Designation Command

Command format: X - Y plane G17 {G01~G03} X_ Y_{I_ J_ or R_}F_
Z - X plane G18 {G01~G03} Z_ X_{K_ I_ or R_}F_
Y - Z plane G19 {G01~G03} Y_ Z_{J_ K_ or R_}F_

Command description: The plane selection function selects different planes for cutting. They are not needed when all three axes move at the same time. G17 ~ G19 commands set up the active plane for straight line and arc cutting or tool compensation. The system defaults to the G17 plane after power on. If X-Y plane is desired, you do not need to set it with the G17 command.
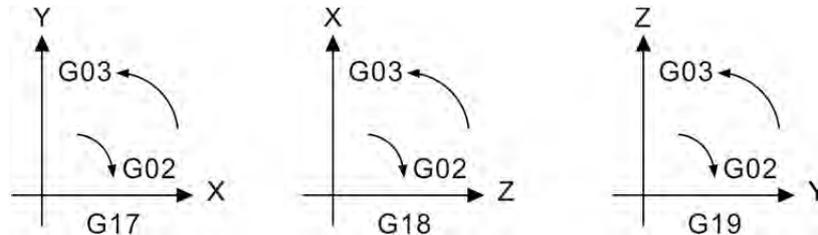
G21/G20: Metric / Inch Input Command

Command format: G21 or G20
G21: Set the metric system
G20: Set the inch system

Command description: This command sets the metric or inch unit of measure for the system. Command G21/G20 applies to the linear axis but not the rotation axis. It must be executed prior to the coordinate system setup command in a program. During the command execution, metric and inch input command cannot be switched. All relevant values which influences the system includes F value of cutting feed rate, coordinates position value, workpiece coordinates offset amount, tool compensation amount and movement distance. G21/G20 are continuous effective commands. When the program specifies a system unit, it means its unit is metric or inch unit. Moreover, G21 and G20 cannot be used in one program at the same time.

G24/G25: Mirror Image Setup Command / Cancelling Command

Command format: G24 X_ Y_ Z_
            G24: Mirror image setup command
            X_ Y_ Z_: Specify the axial direction and center of mirror image

            G25 X_ Y_ Z_

            G25: Mirror image setup cancelling command
            X_ Y_ Z_: Cancel the function of mirror image

2

Command description: When executing G24 command, it can specify X-axis or Y-axis or X-Y coordinates system as the center for mirror imaging. The system will convert the path of the original program to the mirror path. This function is applicable when the left and right paths or upper and lower paths are symmetrical. User can create a mirrored motion path on one side by converting a program path on the other side. This saves the time when programing the motion path. When cancelling this function, G25 has to specify the axis to be cancelled. For example, G25_Y means to cancel the mirror image function of Y-axis while the rest of the axes still apply to this function.

2

[Illustrations]


```
G00 G90 G40 G49
G55 X0 Y0
G01 X5.Y10. F1000
X35. Y10.
X5. Y70.
X5. Y10.
G00 X0 Y0
G24 X0
G01 X5. Y10. F1000
X35. Y10.
X5. Y70.
X5. Y10.
G00 X0 Y0
G25 X0
M30
```
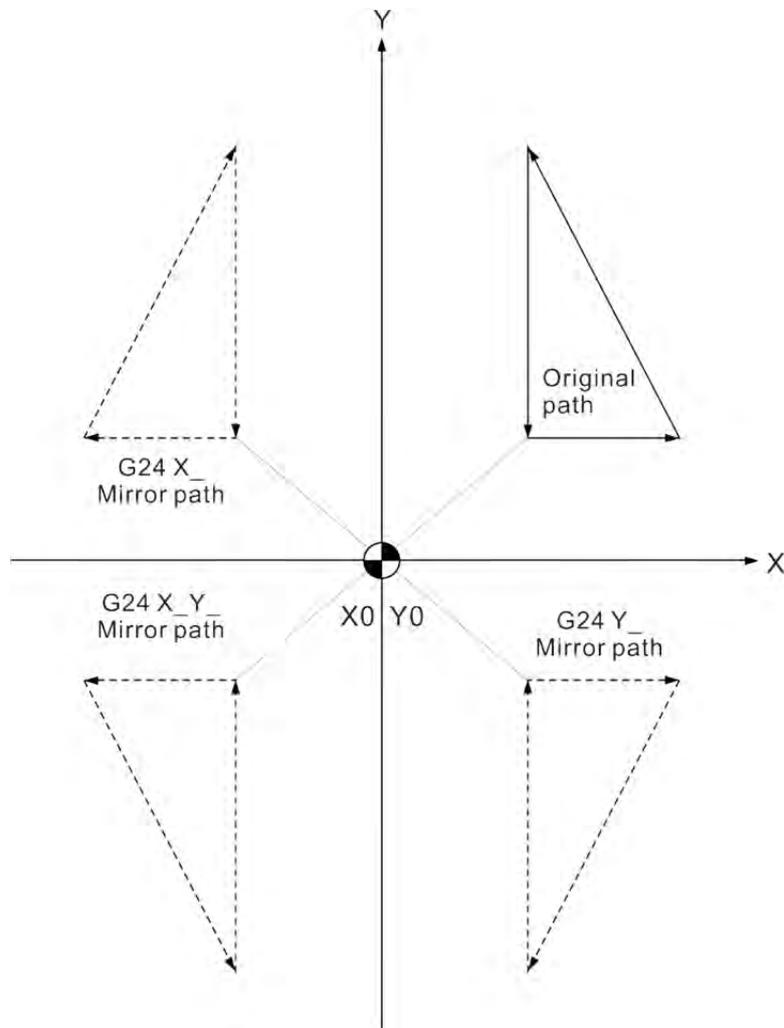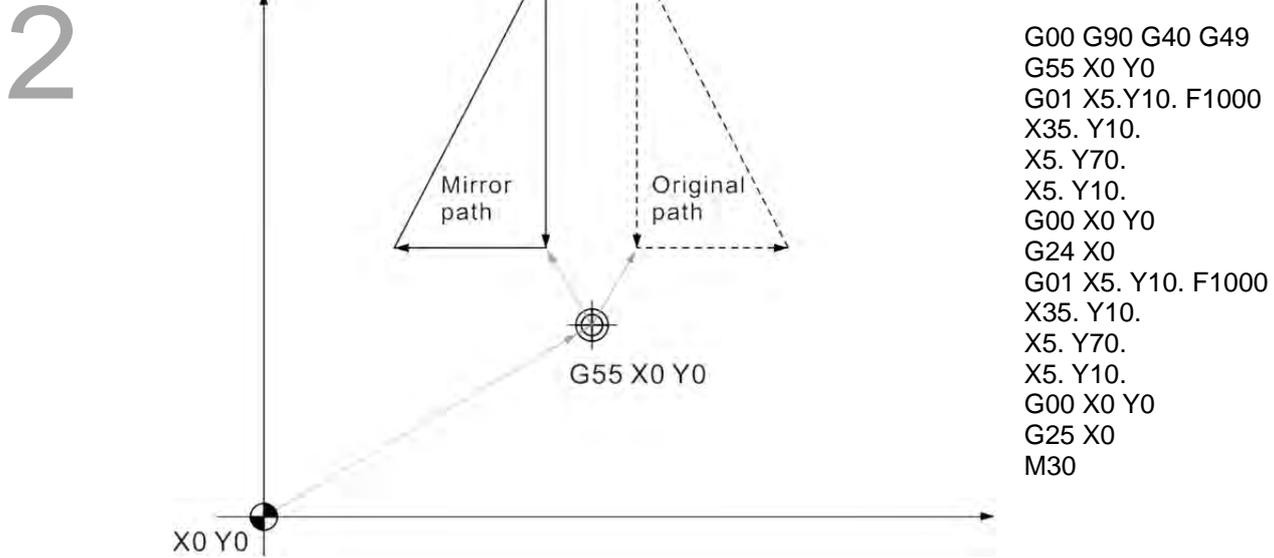
The original path is showed in dotted line, when you apply the mirror function to complete the path, the system will then complete the path showing in solid line. In this example, X-axis is specified as the mirror axis.

When applying G25 to specify the axis, it means to cancel the mirror function of the specified axis. After the mirror image function is cancelled, the motion path will become the original one.

G28: Homing to the First Reference Point

Command format: G90 G28 X_ Y_ Z_ or;
            G91 G28 X_ Y_ Z_

X_ Y_ Z_: Coordinates of reference point

2

Command description: This command instructs the tool to fast move (G00) from the reference point given by the command to the mechanical origin. It is designed to return to the mechanical origin by moving to the reference point given by the command to avoid any intersection with the workpiece or the machine equipment.

Parameter X_ Y_ Z_ of the command is for coordinates of the reference point. The undesignated axis does not set up reference point and return to the origin. When tool radius compensation (G41 or G42) has been set, it must be cancelled in advance. After executing G28 command, the system temporarily cancels the tool radius compensation function and its compensation length before moving to the reference and the mechanical origin point in sequence. It maintains the compensation-free status when returning to the mechanical zero point and resumes the tool radius compensation function automatically in next block. After executing G28 command, the tool length compensation function (G43 or G44) remains active after reaching the reference point. When returned to the mechanical origin, the tool length compensation function is cancelled and will not resume automatically in the subsequent motion block. The tool length compensation function needs be set again in the following execution blocks.

[Illustrations]
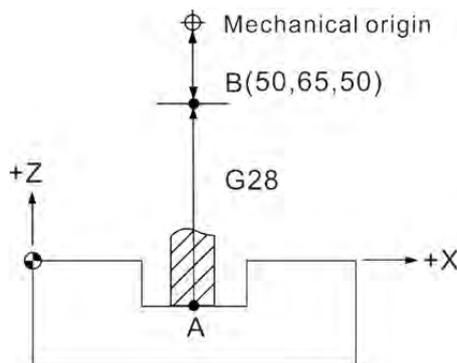
G90 G28 Z50   > Return to mechanical origin from point A through reference point B (Z-axis).

M06 T02   > Switch to tool 2.

[Illustrations]

Status of G90/G91 differs in the process of returning to the mechanical origin when command G28 is executed. See figure below for details.

G90 G28 Z50.0

G91 G28 Z50.0

G90 G28 Z0.0

G91 G28 Z0.0
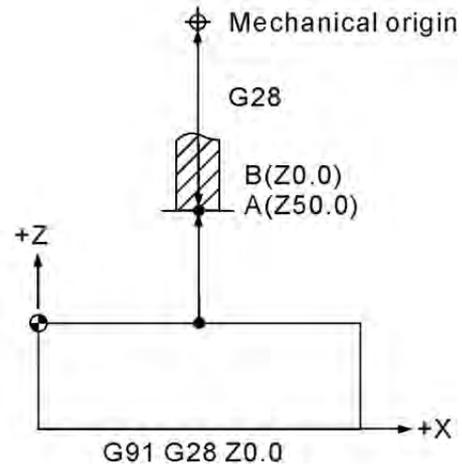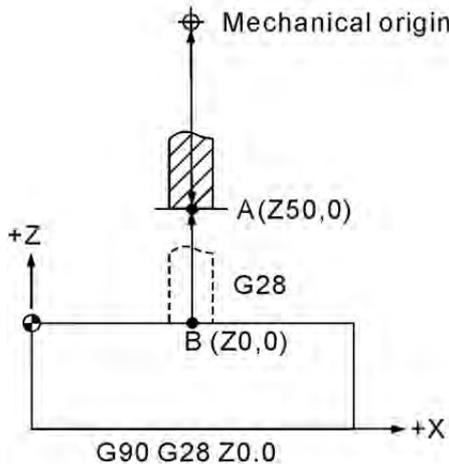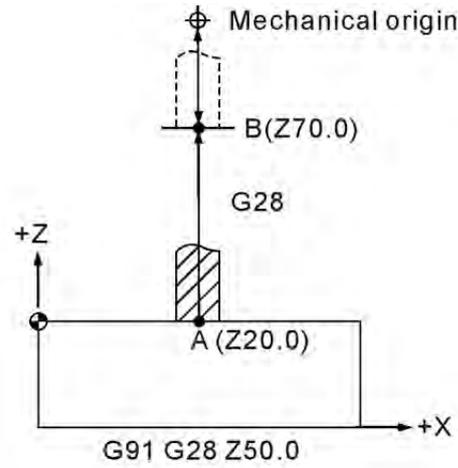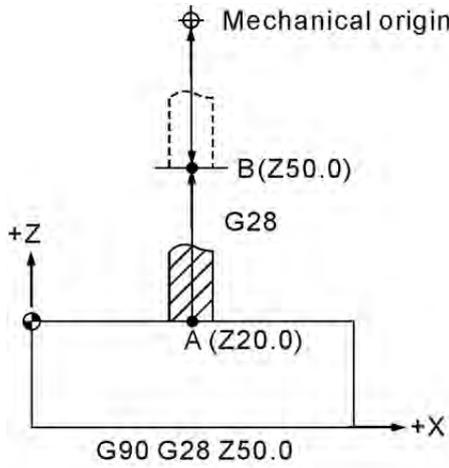
G29: Homing to the Starting Point

Command format: G90 G29 X_ Y_ Z_ or;
             G91G29 X_ Y_ Z_

2

X_ Y_ Z_: The final motion position of current block

Command description: Command G29 moves a tool from the mechanical origin to a point defined by the command through a reference point. Parameter X_Y_ Z_ assigns coordinates of the final point reached by command G29. Command G29 and G28 must be used in pair to move the tool to a reference point defined by G28 then to the final point given by G29. The actual distance from reference point to mechanical origin need not be calculated.
When running command G29 without accompanying G28 (for reference point setup), the system stops running and prompts an alarm message.

[Illustrations]
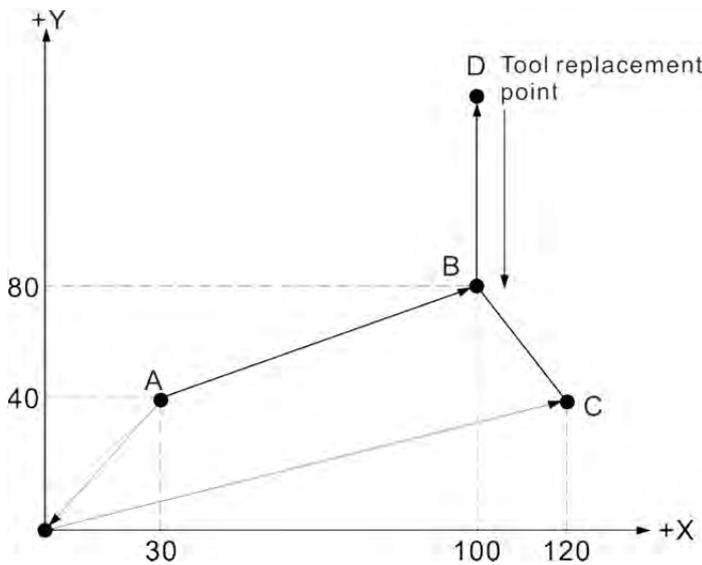
Incremental value setup (tool path A > B > D > B > C)
G28 G91 X70.0 Y40.0
M06
G29 X20.0 Y-40.0

Absolute value setup (tool path A > B > D > B > C)
G28 G90 X100.0 Y80.0
M06
G29 X120.0 Y40.0

G30: Auto Homing to the Second, Third, and Fourth Reference Point

Command syntax: G30 P2 X_ Y_ Z_ or;
G30 P3 X_ Y_ Z_ or;
G30 P4 X_ Y_ Z_

P_: Select reference point type, the second, third or fourth
X_ Y_ Z_: Coordinates of intermediate point

Command description: Parameters P2, P3, and P4 designate coordinates of reference points 2, 3 and 4. You can omit the parameter P2 if the second reference point is required.

Arguments X_Y_Z_ designate coordinates of the intermediate point. The tool moves back to the second, third, or fourth reference point from the current location through the designated intermediate point. The coordinates of the second, third, or fourth reference point are defined by the system parameters. This command is most commonly used for tool replacement. For command status in absolute value the G30 Z0.0 motion command accomplishes the required motions by moving the Z axis to the intermediate point and the second reference point in turn.

Please cancel the tool compensation (executing G40 and G49) before running commands G28 and G30. The tool radius and length compensation are cancelled automatically when executing command G28 and G30. The tool compensation function is resumed only after executing command G43/G44 before the next block is executed. The tool radius compensation function is resumed at the next block after G28 or G30.
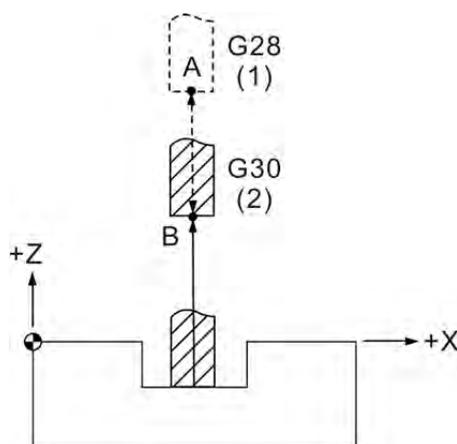
(1) Mechanical origin; (2) Second origin

As shown in the figure above, in the G90 mode the G30 Z0 command moves the Z axis to the intermediate point (the mechanical origin in this example) before being returned to the G30 (second origin) point to accomplish the second origin homing.

[Illustrations]

2

G90 G30 P2 X100. Y80.
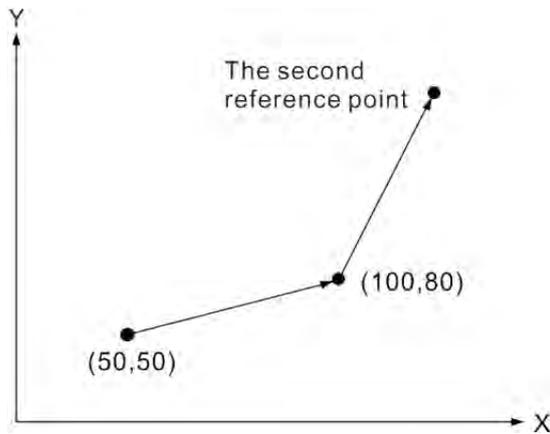
G91 G30 P2 X0. Y0. (without intermediate point)

G90 G30 Z0.0

G91 G30 Z0.0

G90 G30 Z50.0

G91 G30 Z50.0
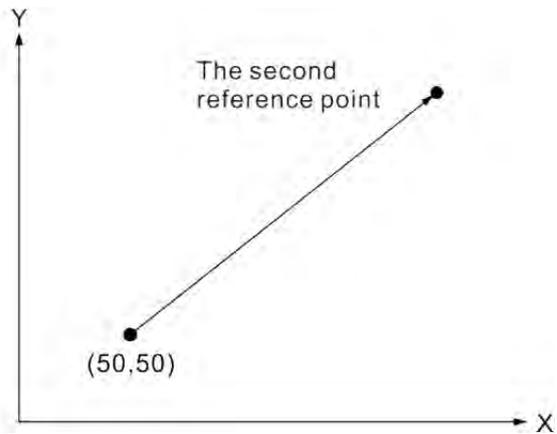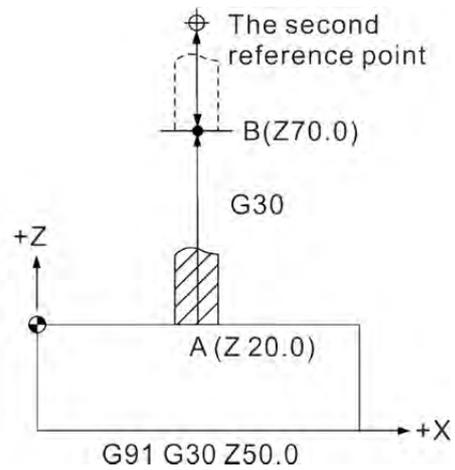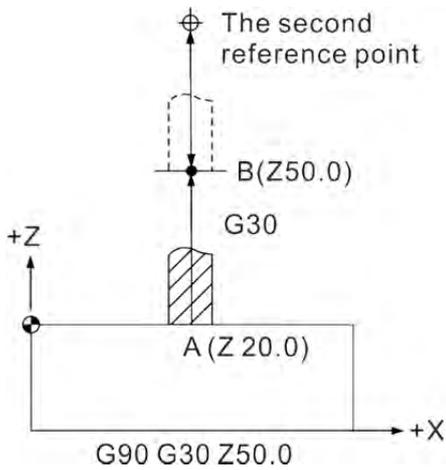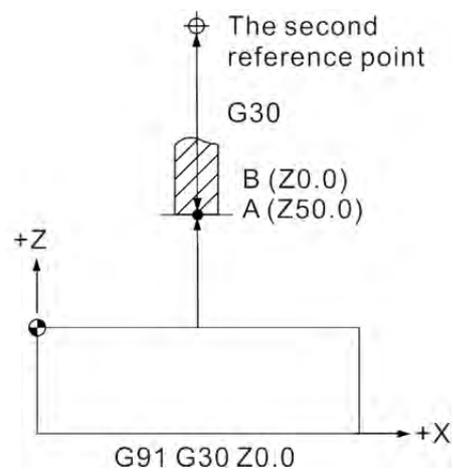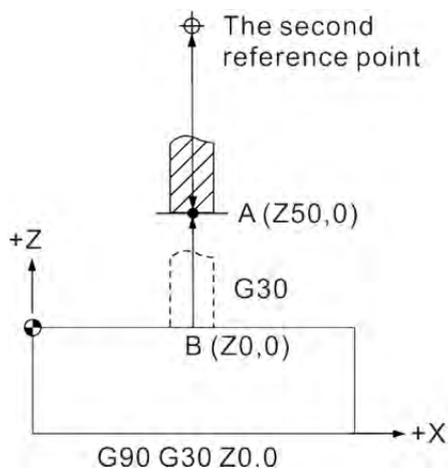
2

G31: Skip Function Command

Command format: G31 X_ Y_ Z_ F_

Command description: This command aborts the running path and jumps to execute the command in next block by accepting an external skip message when moving in a straight line along a given axis direction. This is a valid G command in a single block. Thus, the command is effective only in one block. G31 command cannot run in tool compensation mode. Please cancel tool compensation (G40) before command execution.

[Illustration 1]

G90 G00 X0 Y0
G01 G31 X80.0 F500.0
Y40.0
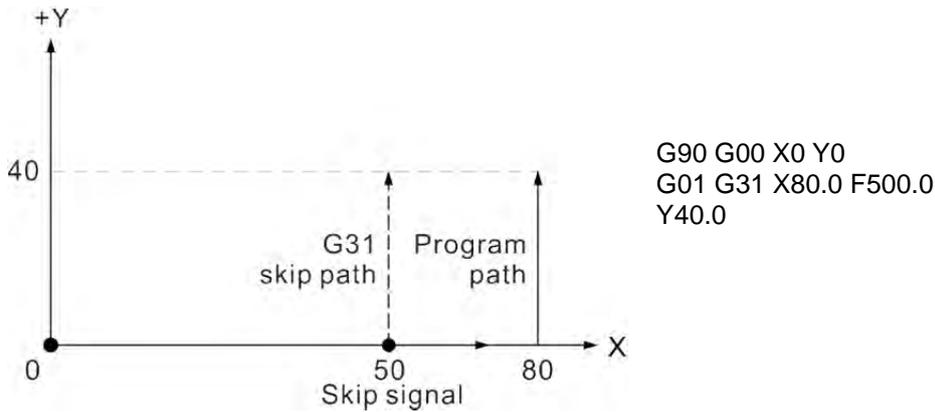
As shown in the figure above the motion path remains straight (solid line) if there is no skip signal. When a skip signal is applied, the program aborts execution of the current statement and starts executing the movements given in the next statement (dotted line).

[Illustration 2]

G90 G00 X0 Y0
G01 G90 G31 X50.0 F150.0
X80.0 Y40.0
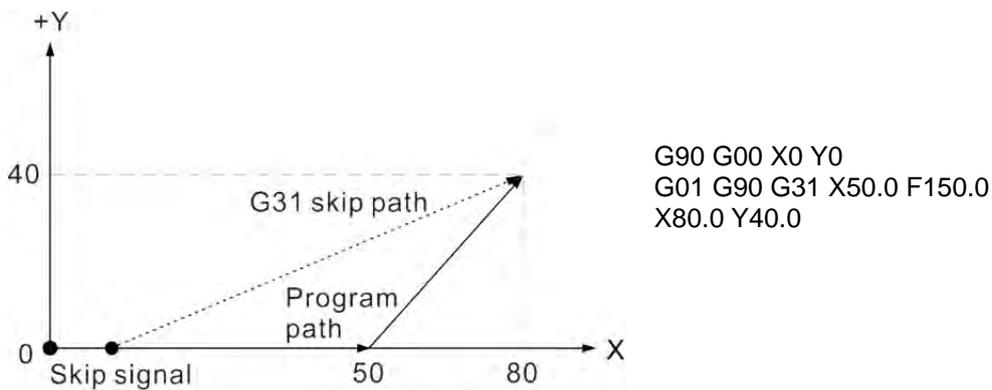
If no skip signal is applied, the actual path is shown as the solid line shown in the figure above. If a skip signal is applied, the actual path is shown as the dotted line as the tool starts executing motions of the next block from the signal input point.

G40: Tool Radius Compensation Cancelling Command

Command format: G40 or;
                                    G40 X‑ Y‑

2

Command description: This command cancels the tool compensation function when it is not
needed in the tool path. As a status command the compensation command remains active
unless cancelled by another command. Command G40 resets the tool compensation status to
avoid path error caused by tool offset for compensation.
In addition, the tool compensation function pauses when returning to the reference point in a
reference point resetting command and resumes at the next motion block. In addition, function of
tool radius compensation cannot be cancelled during arc motion.

G41/G42: Tool Radius Left and Right Compensation Command

Command format: G00 G90 G41 D‑ or;
                                    G00 G90 G42 D‑

                                    G41: Tool radius leftward compensation
                                    G42: Tool radius rightward compensation
                                    D‑: Tool radius compensation data code

Command description: For a program path without tool radius compensation, the tool cuts by
profiling the workpiece shape with the tool center. That is, the effects of tool radius are not
considered in the motion path. This leads to a machined workpiece with a size one tool diameter
smaller than required. When coding the program, transferring the tool path from the
self-calculated tool radius may easily lead to incorrect coordinate calculation and make it difficult
to control dimensions as shown in the figure below.

Tool radius: 10 mm
Contour cutting without compensation

Tool radius: 10 mm
Contour cutting without compensation
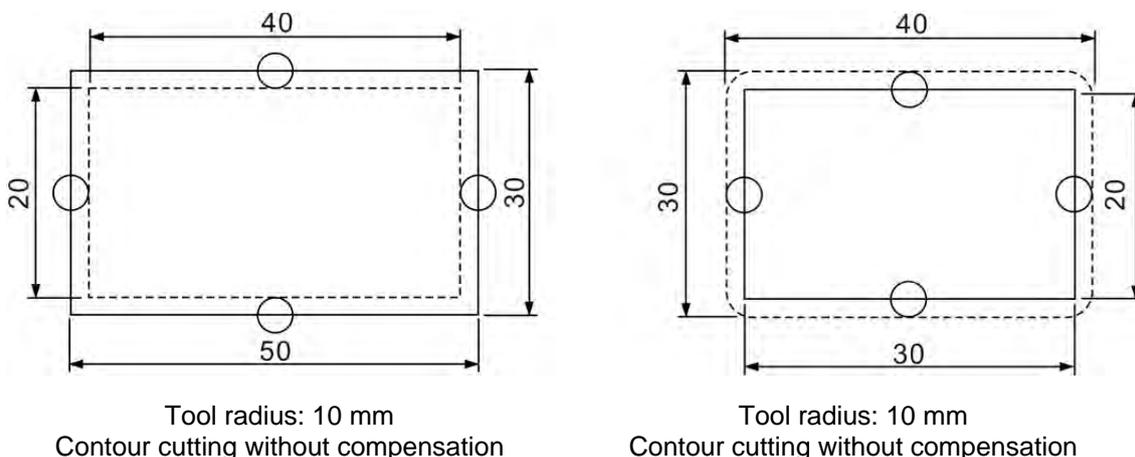
As illustrated in the figure above, a workpiece after machining may have a dimension one tool
diameter greater or smaller caused by the tool diameter when cutting along the profile of
workpiece.

The leftward or rightward compensation of a tool is defined as below: Along the cutting direction,
the tool radius shall be compensated rightward for tools moving to the right of the workpiece and
made by command G42. Otherwise, the tool radius shall be compensated leftward for tools
moving to the left of the workpiece and made by command G41.

The argument D‑ is the tool radius code represented by binary digits. This is the tool
compensation data number contained in OFS group. For example: argument D11 indicates that
the tool radius compensation number is 11. If number 11 represents value 4.0 then the tool
radius is 4.0 mm. Executing command G41 or G42 means the controller will then read the tool

2

radius data in OFS group as the compensation length based on the tool radius compensation number given by the argument D.

Note for tool radius compensation:

(1) This command may be assigned together with G00 or G01 in the same block. The tool compensation is activated only after the tool moves (enable the tool radius compensation command). It cannot work together with commands G02 and G03. The compensating tool radius in the arc path must specify the compensation in advance in the straight line motion path. While the tool radius compensation is active, it cannot be cancelled in arc path. See below for illustration.
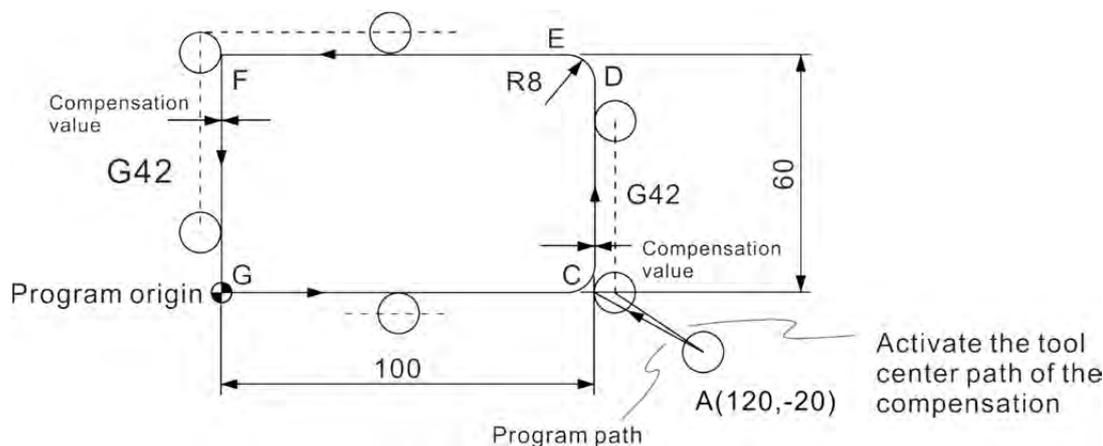
See below for program codes for moving from point A to point C and activate the tool radius rightward compensation command:

G90 G00 X120.0 Y-20.0	> fast positioning to point A
G01 G42 X100.0 Y0 D20 F80	> A → C
Y52.0	> C → D
G03 X92.0 Y60.0 R8.0	> D → E
G01 X0	> E → F
Y0	> F → G
X100.0	> G → C

(2) When designing the program, the tool compensation number, e.g. D11 and D12, defined in a program must correspond to the one contained in the compensation table. These tool radius compensation values are numbers entered by the operator in the tool entry function of the OFS group in advance.

(3) When changing compensation value from positive to negative or vice versa, the compensation direction of G41 and G42 will be changed. For example, for positive arguments in command G41, the system compensates leftward and rightward when negative arguments are given. Similarly, for positive arguments in command G42, the system compensates rightward and leftward when negative arguments are given. That is, the function direction of the G41 and G42 exchange along with the change of the positive-negative sign changes in compensation value.

(4) The tool radius compensation function is deactivated temporarily when command G28 or G29 is in active status. The compensation status resumes when executing the next motion block as the compensation status is maintained by the control system.

2

(5) After the program path is completed under the tool radius compensation mode, the command G40 should be executed to cancel the compensation status and return the tool radius center back to its actual coordinate points. That is, after a G40 command is executed in a program, the motion path cancels the compensation length by turning the leftward or rightward compensation value to the opposite direction. The G40 command should be executed after the tool is moved away from the workpiece as shown in the sample program described below:

Ø10 milling cutter
Point B is theprogram origin

"→" means the compensation vector
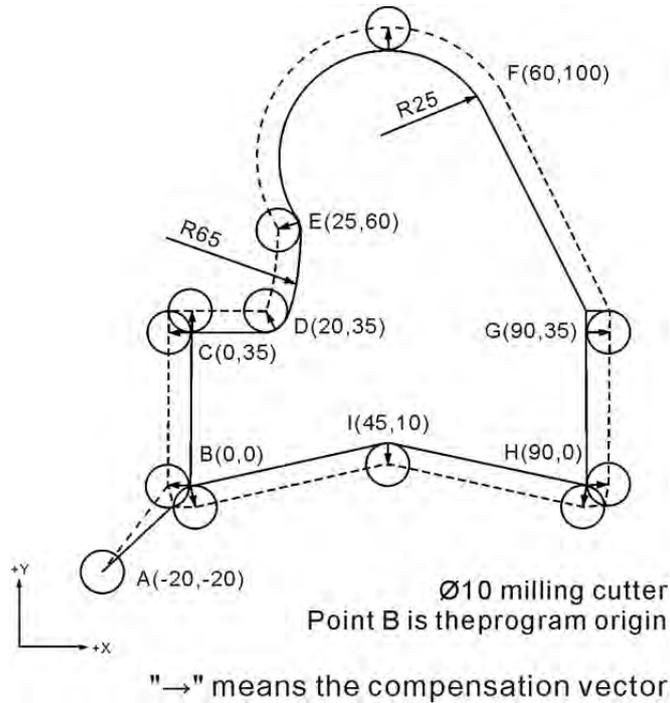
G90 G00 X-20.0 Y-20.0	> Fast positioning to point A
G01 G41 X0 Y0 D12 F80	> A → B: Start leftward compensation command G41
Y35.	> B → C
X20.	> C → D
G03 X25.0 Y60.0 R65.0	> D → E
G02 X60.0 Y100.0 R25.	> E → F
G01 X90.0 Y35.0	> F → G
G01 Y0	> G → H
X45. Y10.	> H → I
X0 Y0	> I → B
X -20. Y -20.	> B → A
G40	> Cancel compensation after the cutter is far away from the workpiece.

(6) In compensation status, the linear moving distance and inner arc cutting radius must be greater than or equal to the tool radius; otherwise, over cutting may occur due to compensation vector interference. In a case like this, the controller stops operation and prompts alarm messages. Please see the figure below for the illustration of over cutting.
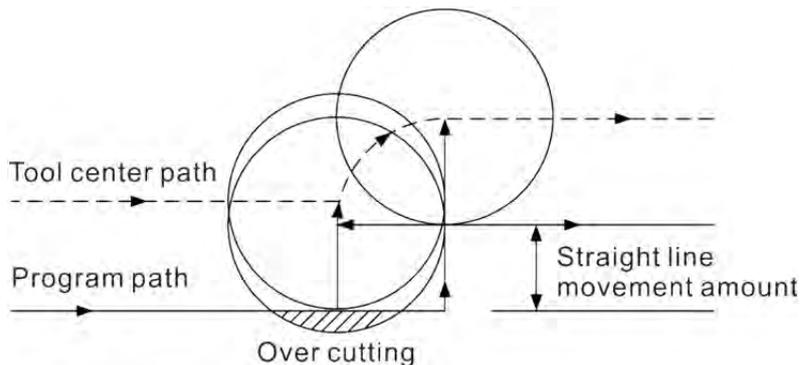
2

(7) The moving distance after cancelling the compensation should ≧ tool radius. When it is smaller than compensation vector, the cutting path will be interfered and may occur over cutting. When this happens, the controller stops operating and an alarm message pops up. See the figure below for illustration.
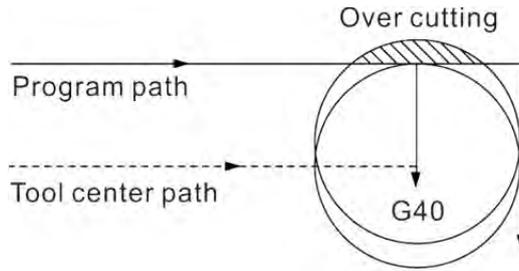
Moving distance after canceling the compensation is smaller than cutter radius

(D1 = 32.0)
G00 G90 G40 G49
G54 X-50.0 Y-50.0
G01 G42 D1 X0.0 Y0.0 F1000   > Start to do tool radius compensation
X100.0
Y100.0
X0.0
Y0.0
X50.0
Y-1.0   > Tool radius compensation is complete. Over cutting occurs.
M30

(8) With the following conditions, the function of tool radius compensation will be disabled:

A. Tool radius compensation will be disabled when it executes to the motion block after G40 command.

B. Or when it executes to the final block and no more motion block should be executed afterwards, then, the final motion block has no tool radius compensation.

Tool radius compensation path type:
The G41/G42 tool radius compensation has two types. Type A does not compensate the starting and ending points in a path while Type B compensates both points. The compensation path must take angles (180° > θ> 90°, 0 < θ < 90°) formed by blocks into consideration. When degrees of angle formed by blocks are in the range of 180° and 90° the tool moves in an inward curved angle and in an outward curved angle when the angle is in the range of 0° and 90°. See the illustration below for details.

Inward curved angle	Outward curved angle
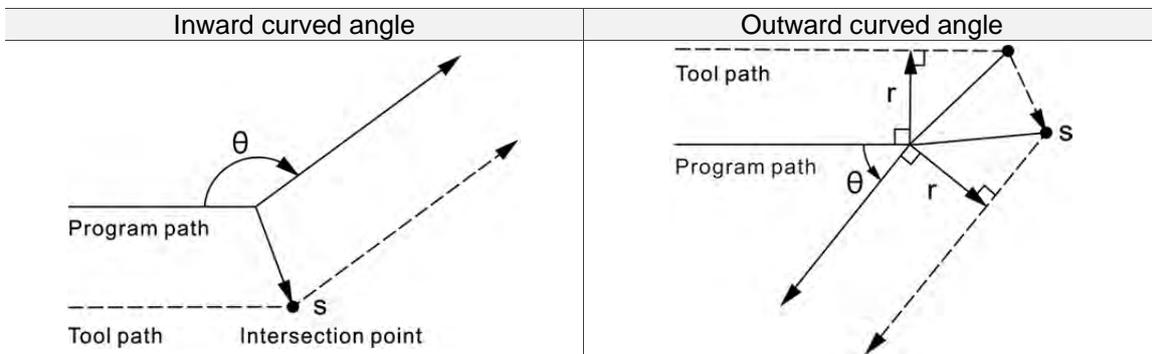
There are two compensation types: Type A and B. Type A does not compensate the starting and ending points in a path while Type B compensates both points. See the figure listed in the left for the motion tracks generated by Type A.

2

Type A: Line to line	Type B: Line to line
Type A: Line to arc	**Type B: Line to arc**
Inward curved angle: Arc to line	**Outward curved angle: Arc to line**
Inward curved angle: Arc to arc	**Outward curved angle: Arc to arc**
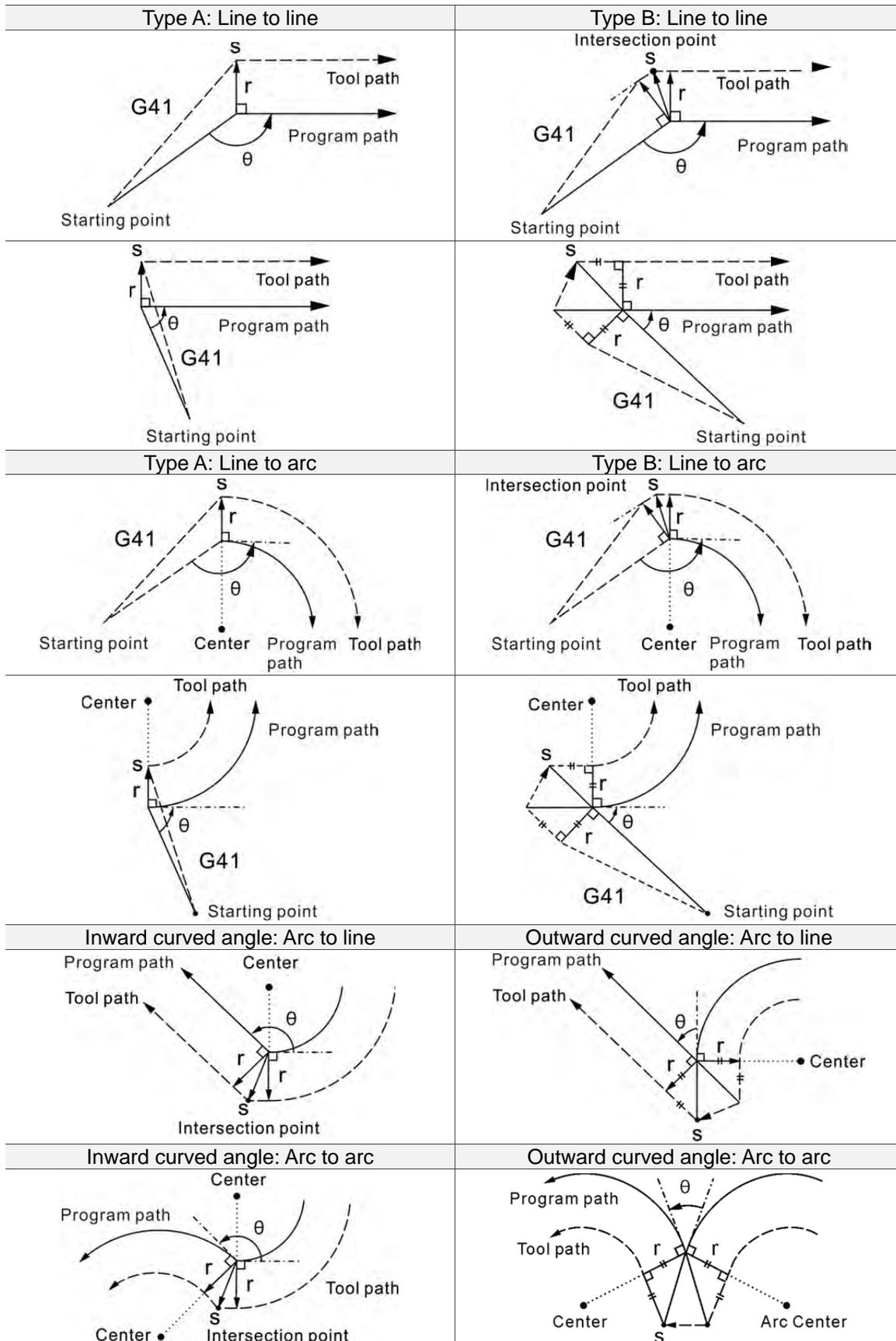
2

Compensation path toggle switch: When a motion path without compensation enters a path with compensation, the motion track of the tool center is as shown in the figure below. During the compensation period, the motion track remains active. When compensation path execution is cancelled (G40) or switched to compensation in another direction directly, the motion track will be shown as the figure below.

G40 > G41	G40 > G42
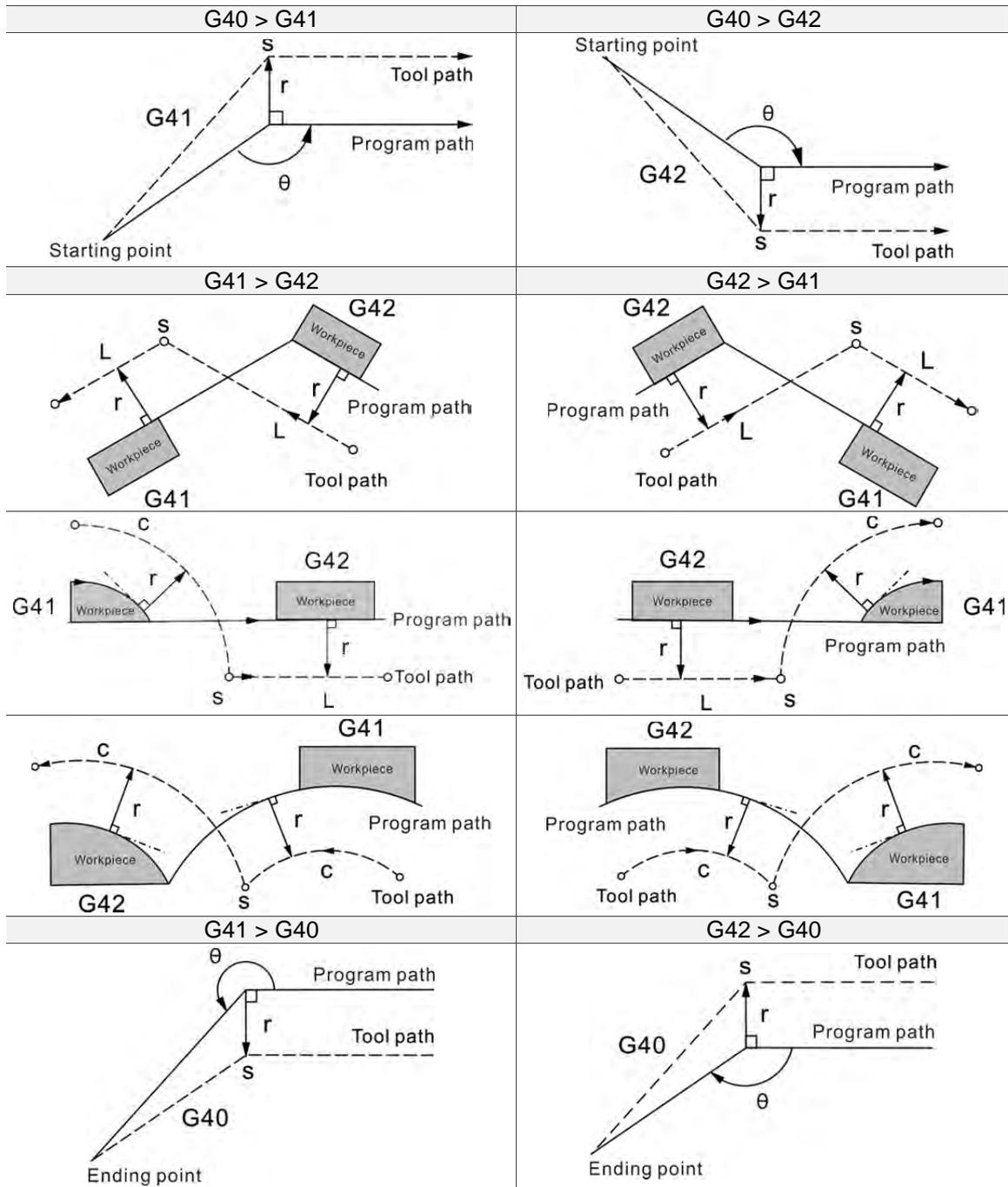
G43/G44: Tool Length Compensation Command

Command format: G43 Z_ H_
                         G44 Z_ H_

2

G43: Tool length positive compensation. For positive tool length the tool axis moves in the positive direction.
G44: Tool length negative compensation. For negative tool length the tool axis moves in the negative direction.

Command description: Most NC machine may use multiple tools with different lengths in one machining program. This command enables operators to define individual tool length ID for length compensation. It ensures machining depth complies with program specifications and simplifies the program designation tasks.

Definitions of the command are described below:

Z: The coordinate position is: zero point + tool length compensation. The zero point is the reference to the Z axis of tool coordinate.

H: Tool length compensation data ID expressed in binary digits. The tool length compensation represented by given ID will be taken as the height compensation of this program. Take H01 as an example, if the tool length compensation ID is 01 and the value represented by ID 01 is -412.8, then the tool length compensation value of this tool is -412.8 mm. After the G43 or G44 command is executed, the controller takes the value represented by the tool length compensation ID given by argument H as the tool length compensation value.

G43 Z_H_: If the value represented by the compensation ID is positive then tool compensates upward, otherwise, it compensates downward. G44 Z_H_: If the value represented by the compensation ID is positive then tool compensates downward, otherwise, it compensates upward.

Note for tool length compensation:
Both commands G43 and G44 remain active until reset by command G49 or H00. (G49: Cancel the tool length compensation. H00: Compensate with value zero.)

[Illustrations] Tool length compensation setup
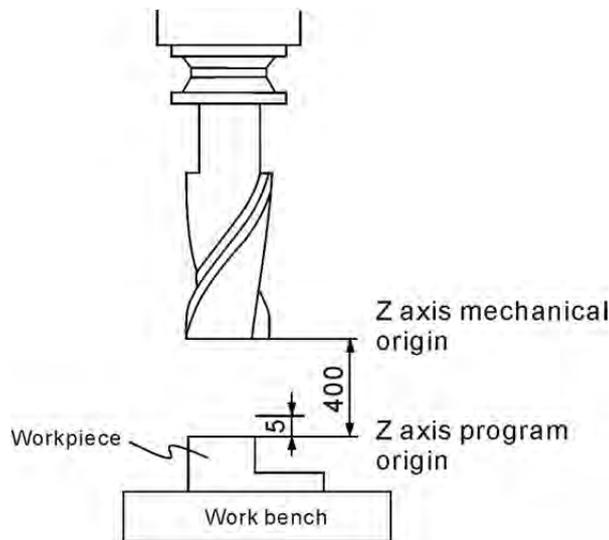
Tool length compensation

(1) **G43 Z5.0 H01**		(2) **G44 Z5.0 H01**	
ID	Data	ID	Data
01	-400.000	01	400.000
02	0	02	0
03	0	03	0

2

Note:

(1)   The system cancels the tool length compensation value automatically before executing commands G53, G28, and G30 when tool length compensation is active. Later the program runs without tool length compensation unless another H_ argument is assigned by command G43/G44.

(2)   Parameter 307 is for setting the moving mode of tool length compensation when G43/G44 and G49 execute without Z command. When the Parameter 307 is set to 0, it means G43/G44 and G49 move the tool length compensation without Z command. If the Parameter 307 is set to 1, it means when Z value is not set in G43/G44 and G49, the compensation is completed by the internal system.

(3)   The active tool length compensation (by G43 or G44) remains active when G28/G30 reaches the reference point and then cancelled when returned to the mechanical origin. The tool length compensation will not be resumed in later motion blocks.

(4)   The active tool length compensation will be cancelled by the system and switched to G49 status when commands M30 and M02 are executed successfully.

(5)   The active tool length compensation will be cancelled by the system and switched to status specified by G49 when a RESET signal is received by the system.

G49: Tool Length Compensation Cancelling Command

Command format: G49

Command description: The tool length compensation function is a status assignment. It remains active and tool length is memorized by the system after being executed. The length memory is refreshed only by reading another compensation ID. Please reset the old tool length compensation command after a new tool is selected or execute the new tool length compensation command specific to the newly selected tool. The command G49 cancels the previous tool length compensation function.

G50/G51: Scaling Up/Down Command/Cancelling Command

Command format: G51 X_ Y_ Z_ P_

> X_ Y_ Z_: Coordinates center for scaling
> P_0: Scaling ratio

Command description: Argument X_ Y_ Z_ sets the X, Y, and Z coordinates of the scaling center and P sets up the scaling ratio. This command enables the machining workpiece to cut final pieces with proportionally different dimensions. The machining path is enlarged or contracted with the coordinate's center given by argument X_ Y_ Z_ and scaling ratio by P.
The minimum value of argument P is 1 with a ration range between 0.001 and 999.999. For example, argument P100 represents to scale down to one-tenth.
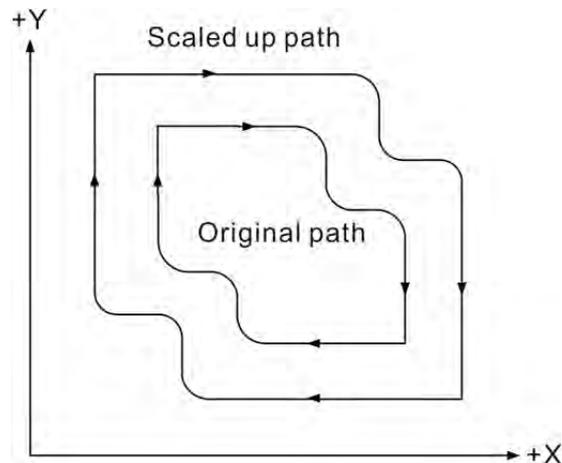
As shown in above figure the original tool path is converted into a new path by executing command G51 with scaling ratio set by argument P. The scaling function does not change tool radius compensation, tool length compensation, and tool position compensation as the compensation action and value are made after scaling. When executing commands M02 and M30 or when the NC300 controller is reset, the scaling mode will be cancelled. The reset key also can cancel scaling. Execute command G50 in program to reset the scaling function as well as return to original scale status and cut in the normal path.

G52: Local Coordinate System Setup Command

Command format: G52 X_ Y_

X_ Y_: Origin of the local coordinates system

Command description: Users can set up a sub-coordinates system based on the workpiece coordinates system for easy path coordinates designation when program coding. This sub-coordinates system, also known as the local coordinates system, can be defined as a one current workpiece coordinates system (G54~G59) by command G52 with arguments of absolute coordinates data. It is valid for absolute supplier status but not the incremental system (G91) status. Command G52 with argument zero cancels the local coordinates system setup. The tool radius compensation command is suspended when command G52 is applied.

[Illustration 1]


```
G90 G54 X0 Y0
G52 X40.0 Y40.0
G00 X20.0 Y20.0
```
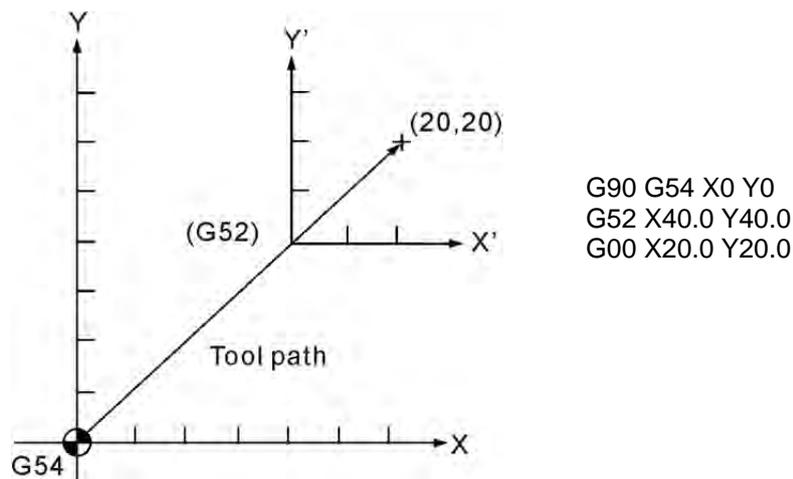
[Illustration 2]

G90 G54 G00 X10. Y10.
G52 X30. Y20.
G00 X20. Y20.--------------------------(A→B)
G56 G00 X50. Y10.-------------------(B→C)

2
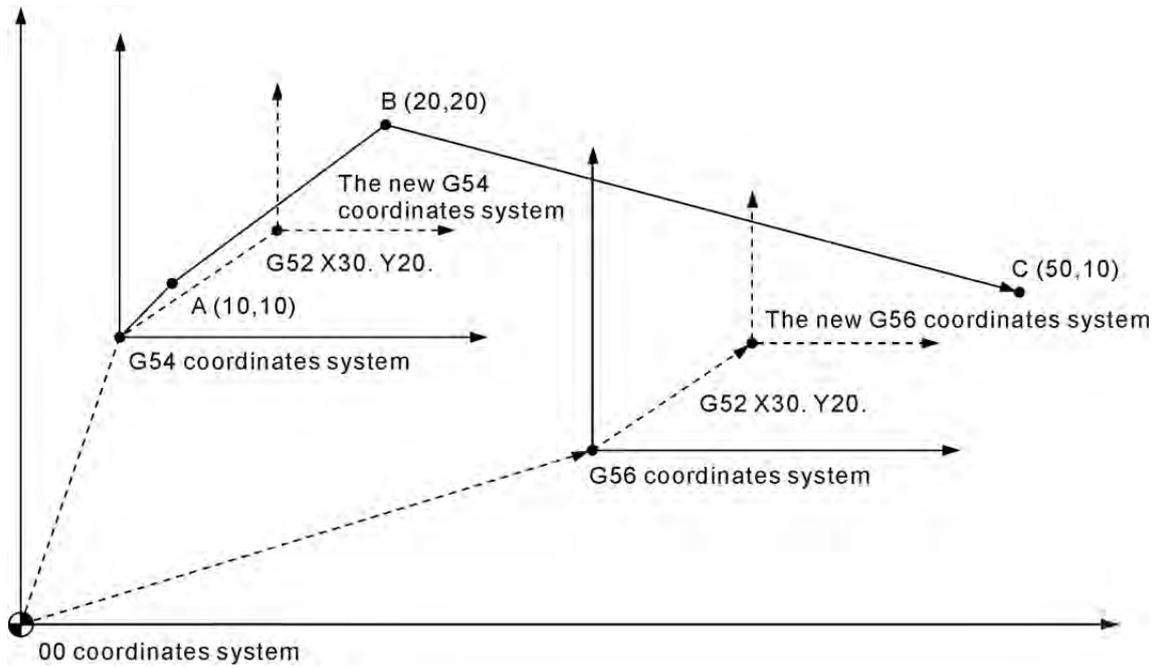
Converting the current workpiece coordinates system to another coordinates system with an active G52 command gives the converted workpiece coordinates system the same offset effects of G52.

Users can cancel a local coordinates system by executing command G52 with three argument coordinates (X, Y, and Z) in value zero. That is, an assignment of "G52 X0 Y0 Z0" command format.

G53: Mechanical Coordinate System Setup Command

Command format: G53 X_ Y_ Z_

X_ Y_ Z_: Actual arriving position of mechanical coordinates

Command description: Coordinates X, Y, and Z are the actual ending point in mechanical coordinates system specified by the program coordinates. Machine suppliers use this command to set up the tool replacement position with the reference point given in mechanical coordinates. It cannot be changed once created. The command format must be in absolute coordinates status. This command is ignored when in incremental status.

Command G53 is a non-continuous G command and is valid for the statement containing it. Before using a G53 command to set up a coordinates system the origin must be reset manually or automatically in advance. After the G53 command is executed, the system moves in G00 mode, the tool radius compensation is paused and tool length compensation cancelled automatically; the former resumes at the next motion block and the latter becomes active only by re-assigning it again.

Note:
(1)   Command G53 functions only at G90 status. It is ignored in G91 mode. However, a status command contained in the same statement of G53, G00/G01 or G90/G91, changes the status and affects the motion status of the next block.
(2)   If the statement containing command G53 also contains a specific axial command, then the axis moves to a specified point. Otherwise, there is no position command.
(3)   If both commands G53 and G28 are set in the same statement, the one read later becomes active. When command G53 is active, the motion position refers to the mechanical coordinates. If command G28 is active, then the absolute coordinates are referred to.

[Illustrations]

Example 1

G91G53X150.Y-150.   This command is ignored.

X-30.Y-30.   This command is changed to incremental motion mode.

Example 2

G90G53X50.Y-50.Z0.   Move to X50.Y-50.Z0. in actual mechanical coordinates.

Example 3

G1G53X100.Y-100.F1000   This command executes in G00 status.

X50.Y50.   The system moves in G01F1000 motion status.

G54~G59: Workpiece Coordinate System Selection Command

Command format: G90 G54 X_ Y_ Z_ or;
                           G90 G55 X_ Y_ Z_ or;
                           G90 G56 X_ Y_ Z_ or;
                           G90 G57 X_ Y_ Z_ or;
                           G90 G58 X_ Y_ Z_ or;
                           G90 G59 X_ Y_ Z_

2

Command description: Command G54 ~ G59 assigns any one of the six fundamental coordinates system as the workpiece coordinates system. A workpiece coordinates system is created by moving the tool from the mechanical origin to the desired program origin (with proper X and Y distance), registering this position data in the workpiece coordinates system setup (G54 ~ G59) in OFS group, and executing the workpiece coordinates system code, then you can set up the workpiece coordinates origin. The system also features a designation function out of 64 sets of expansive workpiece coordinates system options. This is done by assigning values to argument P_ (with valid value range of 1 ~ 64) in command G54. For example, G54 P10 X_ Y_ Z_. It means the tenth coordinates system of expanded workpiece coordinates system is used.

[Illustrations]


```
O1000
G00 G90 G55 X0 Y0
G00 X100.0
Y50.0
X0.0
Y0.0
M30
```
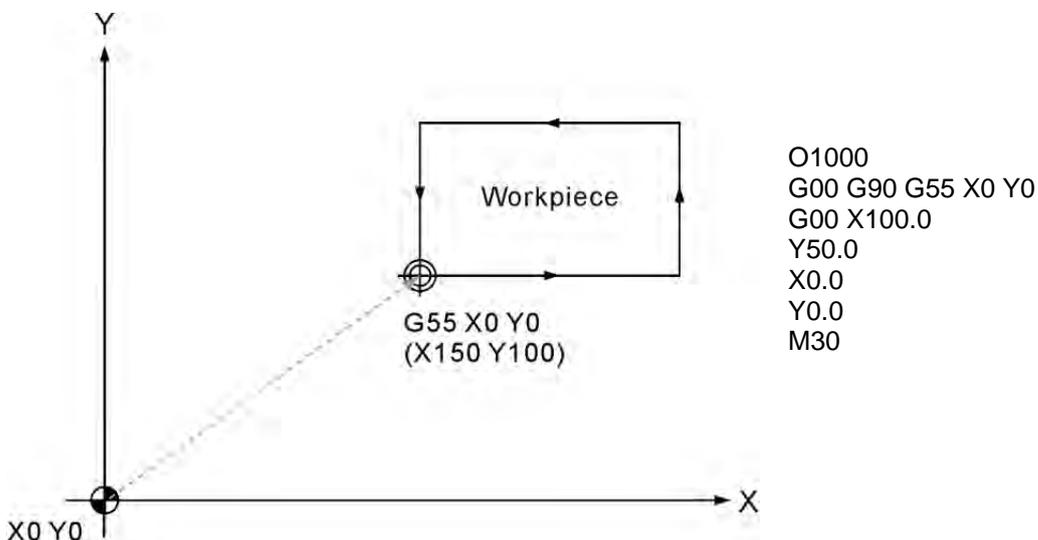
The setting of workpiece coordinates command enables easier program path calculation and design as well as creating multiple work bench coordinate systems to be used alternatively by multiple programs. As shown in the figure above, you don't need to change the program when coordinates of origin are modified. The machining process can be started simply by changing the data value of workpiece coordinates.

G61: Exact Stop Mode

Command format: G61

Command description: Command G61 functions the same as command G09 does, except that the later is a non-continuous (becomes active only when being assigned) status command while the former is a continuous statement. After command G61 is executed, each execution of commands G01, G02, and G03 instruct the system to decelerate to fully stop for inspection. This mode remains active until it encounters command G64 (cutting mode).

[Illustrations]

——————	Tool path in G61 mode
- - - - - - - -	Tool path in G64 mode
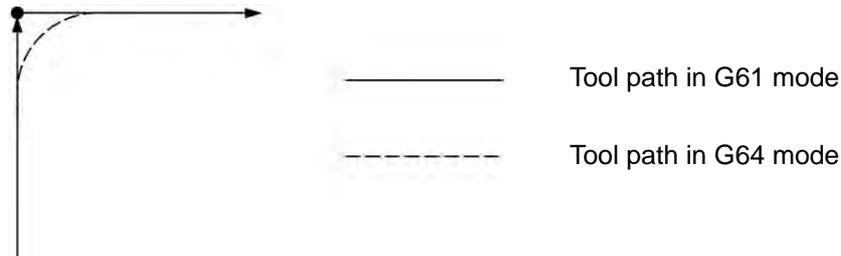
```
G61 G91 G01 Y100. F200.   -------------------------------------(Correct positioning)
X100.                     -------------------------------------(Correct positioning)
G64                       -------------------------------------(Stops correct positioning)
```

G64: Cutting Mode Command

Command format: G64

Command description: After command G64 is executed, the system remains moving at a certain speed to transit into the execution of the next motion block instead of decelerating to full stop at the end of each motion command. Normally, the initial status of the system is set to G64 cutting mode. When command G64 is used, the tool path of NC machines becomes smoother when machining. Command G64 differs from G61 in that it cuts at a constant feed rate and does not decelerate to a full stop between motion blocks.

In cases listed below the system decelerates to a full stop for positioning checking even after command G64 is assigned:

(1)   Statement with fast positioning (G00) mode
(2)   Statement with rigorous stops command (G09)
(3)   There is no movement command in the next block

G65: Non-continuous Effect Macro Calling Command

Command format: G65 P_ L_ℓ

2

          P_: Program number
          L_: Number of repetitions
          ℓ_: Value of independent variable

Command description: Command G65 calls a macro program. A macro program is used for various calculations, MLC interface data input and output, and commands for control, judgment, and branching as well as for calculation and measurements.

A macro program is a subroutine containing variables, calculation and control commands accompanied by exclusive control mechanism. These designated functions (macros) can be used in the main program by a specific macro calling command. Macros are called as command M98, except that it is non-continuous. See below for the macro calling command:

[Illustrations]

Main program	Sub program
N0010	O1010
N0020 G65 P1010 L2 A5.0 B3.0	N1020 #3=#1+#2
N0030	N1030
N0040	N1040
N0050	N1050 M99
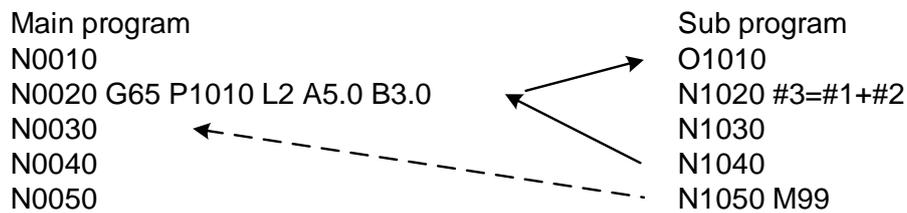
After a macro is executed, it returns to the statement next to the one containing it. That is, the program continues executing from the statement next to the G65 command. In the sample described above, the value of the #1 variable A5.0 is 5.0. Please refer to the table below.

NC position	Local variable	NC position	Local variable	NC position	Local variable
A	#1	I	#9	T	#20
B	#2	J	#10	U	#21
C	#3	K	#11	V	#22
D	#4	M	#13	W	#23
E	#5	Q	#17	X	#24
F	#6	R	#18	Y	#25
H	#8	S	#19	Z	#26

2

[Illustrations]
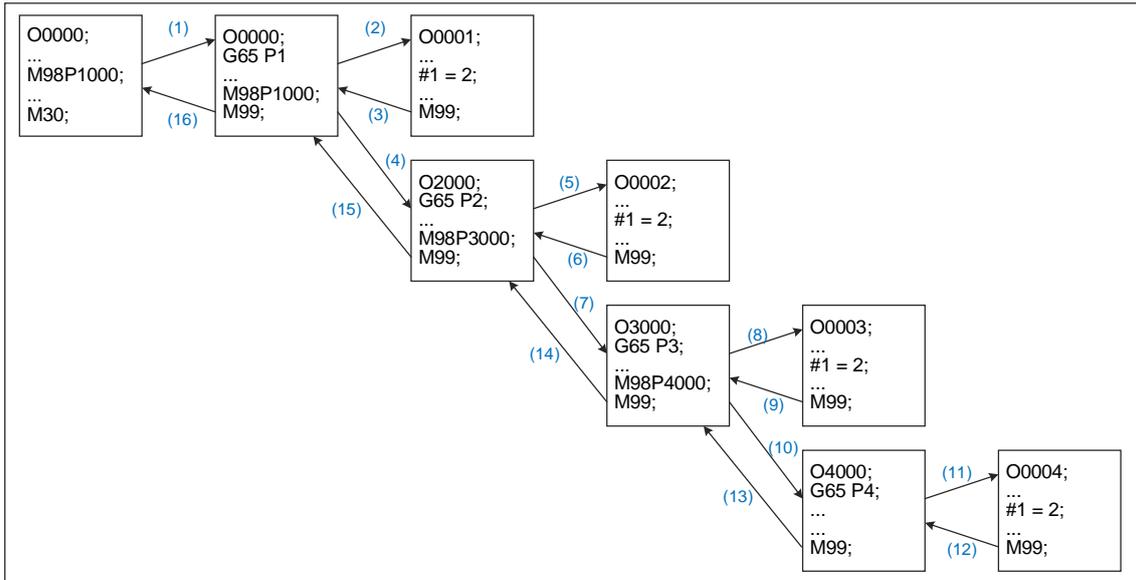
Command G65/G66 can nest macros up to 8 layers. When used together with subroutine calling command M98, the program nest layers remain at eight.

G66/G67: Continuous Effect Macro Calling Command / Cancelling Command

Command format: G66 P_ L_ℓ or;
G67

P_: program code
L_: number of repetitions
ℓ: value of independent variable

Command description: Command G66 functions the same as G65 except that instead of being active for a single block, the former keep on calling the macro in later statements until it is reset by command G67. Before being reset by command G67, the macro continues to be called.

[Illustrations]

O0001;		O0008;
...		#3 = #1 + #2;
G66 P0008 L1 A 2.0;	Execute O0008 after movement →	G91 G00 Z#23;
X3.0;		Z-#3;
G91 G00 Y10.;	Execute O0008 after movement →	M99;
Y10.;	Execute O0008 after movement →	← Back to O0001
G67;		
Y10.;		
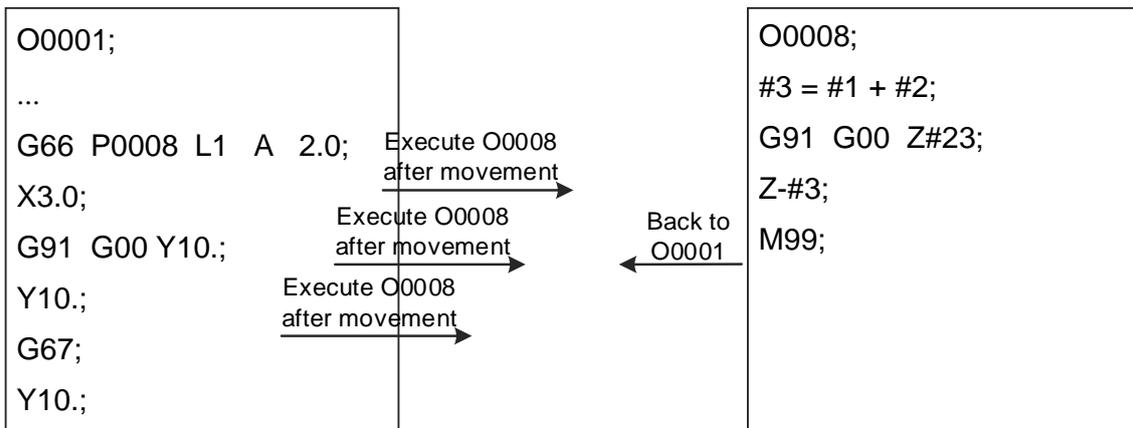
2

G68/G69: Coordinate System Rotation/Cancelling Command

Command format: G68 X_Y_R_

X_Y_: Coordinates of the rotation center
R_: Rotation angle; positive value for counterclockwise rotating and negative for clockwise.
The system rotates in units of 0.001 degree and in range from zero to 360 degrees.

Command description: The coordinates rotation command G68 rotates the original machining program at a given zero point in a specific angle. For a workpiece placed on the workbench in an angle against the program command position, users may rotate the coordinates of the machining path according to the specified rotation angle. G68 can be set with absolute command (G90) as well as incremental (G91).

[Illustrations]

Original path:
O1000
G00 G90 G55 X0 Y0
G00 X100.0
Y50.0
X0.0
Y0.0
M30

When rotates 90 degrees:
O1000
G00 G90 G55 X0 Y0
G68 X0 Y0 R90.0
G00 X100.0
Y50.0
X0.0
Y0.0
M30
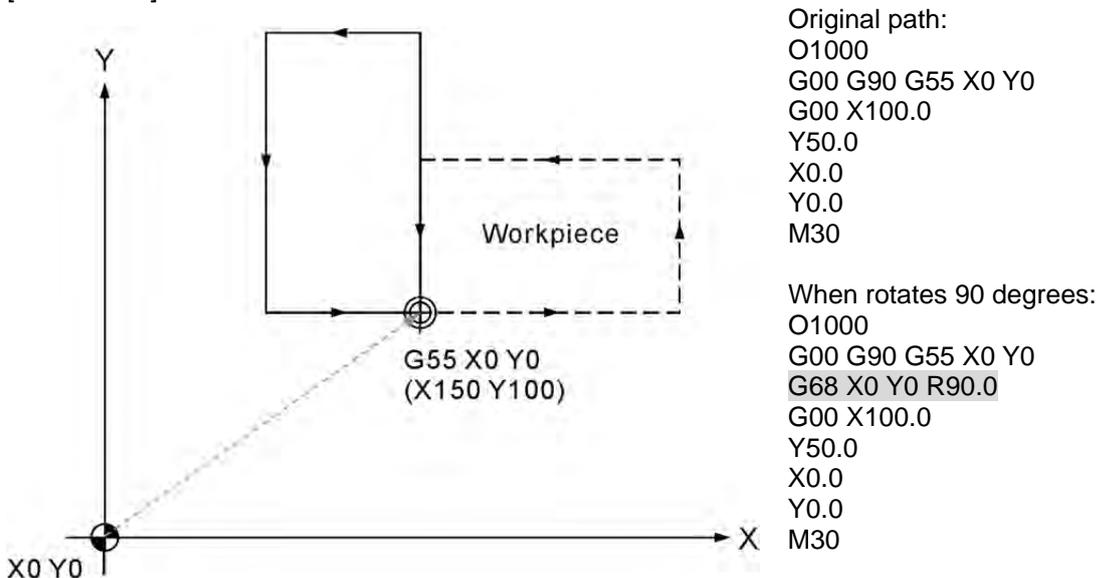
The original program (at the top of the figure) machines workpiece is in the path shown with a dotted line. If the workpiece is placed in another direction (here it is rotated 90 degrees), the program can still function without any modification by adding command G68 to rotate the program path.

Command G69 resets the rotation given by command G68. The program motion path returns back to its original motion track after the rotation effect is removed.

G73: Peck Drilling Cycle Command

Command format: G73 X_ Y_ Z_ R_ Q_ F_ K_

2

X_ Y_: Ending position of single block
Z_: Bottom of hole to be drilled
R_: Initial safety height
Q_: Depth of each peck drilling
F_: Feed rate
K_: Number of cutting loops

Command description: This command sets the machine to retreat a constant distance "d" after drilling a distance (depth) of "Q" and keeps on drilling to the desired hole depth of "Z". The back-and-forth intermittent Z-axis feeding enables easier chip dispelling during drilling deep hole. Here value Q is absolute value. Value d is the moving distance specified by parameter. If the setting value is 1 mm, it means the default value of retraction amount d is 1 mm. See the figure below for the operation specified by this command.

[Illustrations]
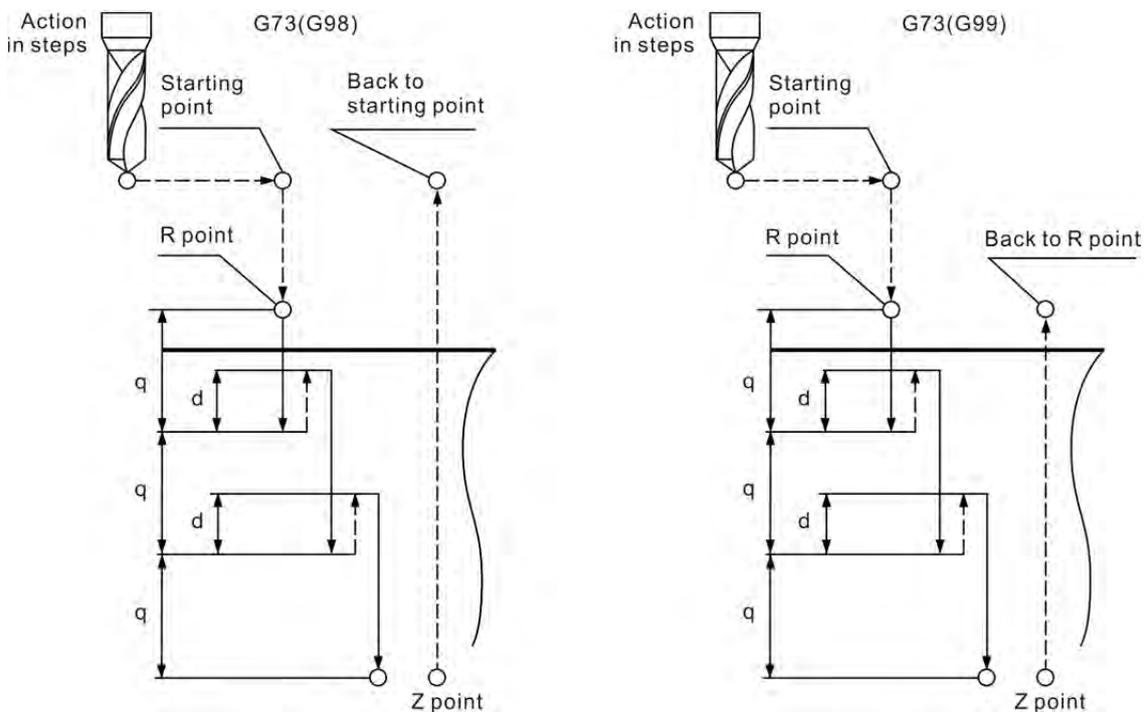
Note:
(1)   The tool radius compensation function is ignored by the peck drilling loop command.
(2)   Run command G80 to cancel the loop cutting motion.
(3)   Value Q cannot be specified as a negative value. Otherwise the alarm will display G code error.
(4)   The value of parameter K will be rounded down to the nearest whole number; e.g., K2.6 round down to K2 and K0.6 to K0.
(5)   When K value is in absolute status, it executes the cycle command for specified number of times at the origin position. When it is in incremental status, it executes the cycle command for specified number of times according to the specified distance.
(6)   K value is specified as 0. When executing this block, it only changes the command to cyclic status and moves according to the command issued by XY axis but not executing cycle command.
(7)   When K value is specified as a negative value and smaller than -1, e.g. K-1.5, it will be regarded as the result of K1.
(8)   When K value is specified as a negative value, a decimal and bigger than -1, e.g. K-0.8, it will be regarded as the result of K0.

2

[Illustrations]

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

2
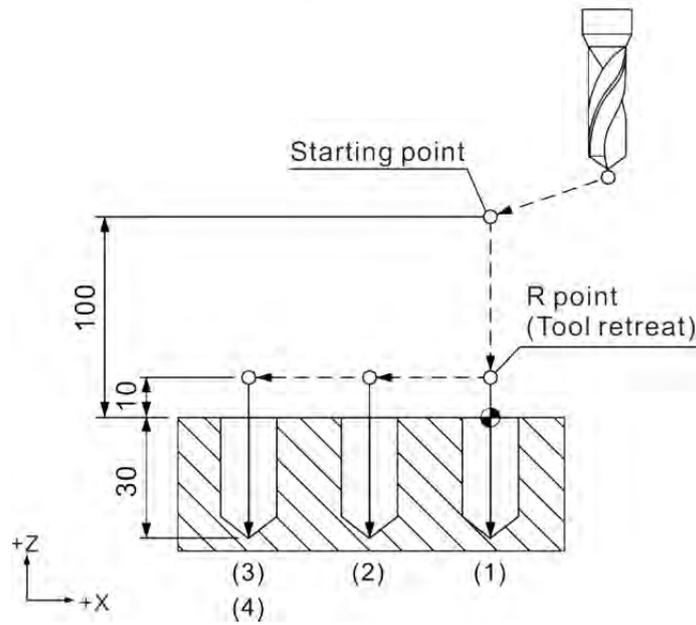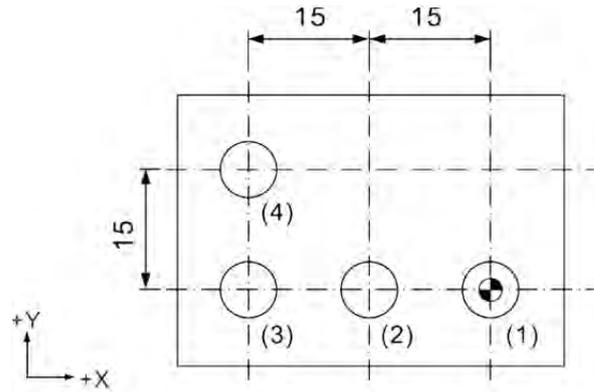
[Illustrations]

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
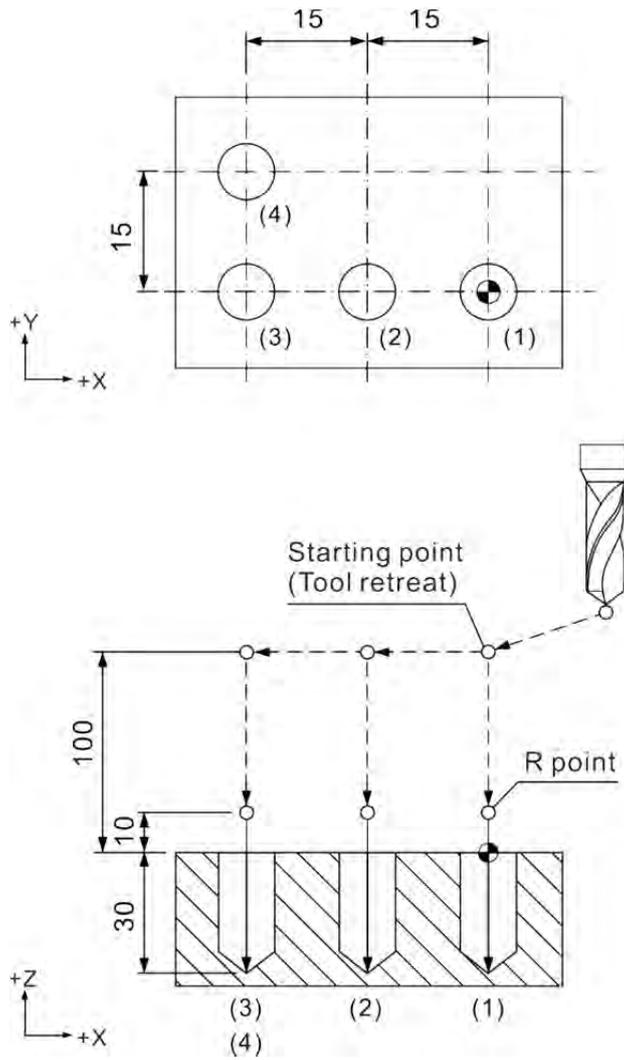
2

G74: Left Spiral Tapping Cycle Command

Command format: G74 X_ Y_ R_Q_ Z_ P_ F_ K_
           X_ Y_: Ending point of a single block
           Z_: Bottom of hole to be tapped
           R_: Initial safety height
           Q_: Peck drilling depth of each time
           P_: Pause time (in the least unit of 1/1000 second), without decimal point.
           F_: Spiral feed rate
           K_: Number of loops

           Feeding speed of tapping (mm/min) = Lead (mm/rev) × Spindle speed (rev/min).
           $F = P \times S$

Command description: This command is for left-handed threading, which requires left-handed tapping cutter and reverse turning spindle to execute command G74. It sets the machine to move as described below:
-    fast move the tool to position set by coordinates (X, Y)
-    fast position to height R
-    tapping at speed set by F till the depth of Z
-    set the spindle to turn in positive direction
-    retreat the Z-axis in positive direction to height R
-    set the spindle reverse turn after reaching height R and prepare for reverse
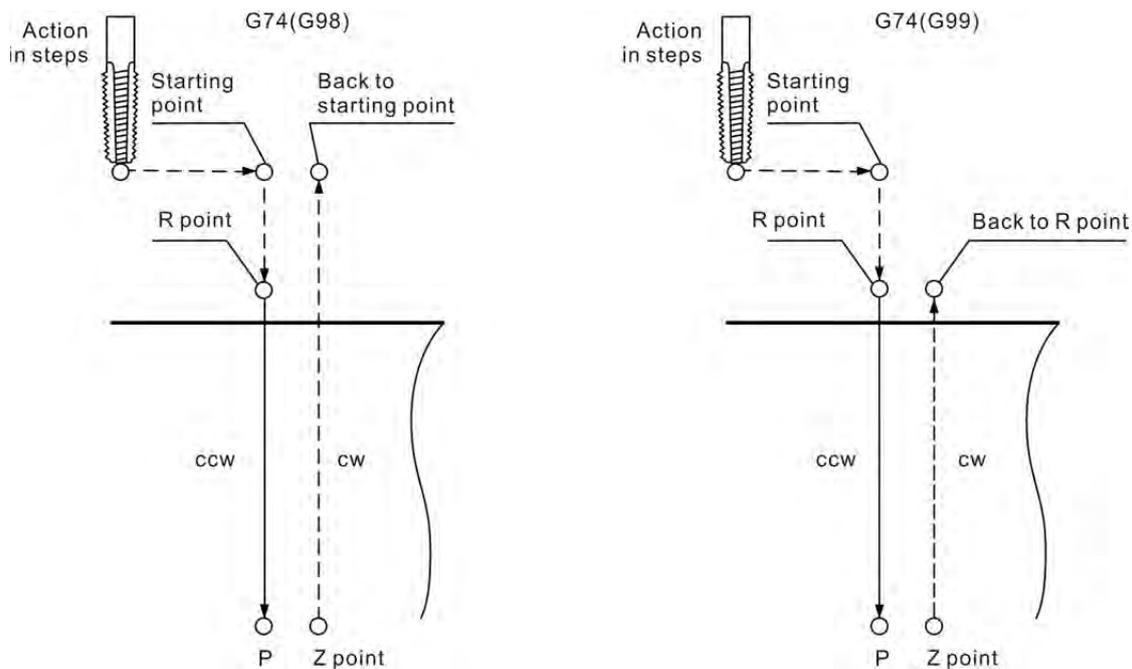
In tapping cycles the machine cuts at the speed given by parameter (100% program value) and the spindle revolution factor and feed rate of the control panel are set to inactive. Furthermore, the program stop key will be ignored to ensure thread accuracy. The stop key is active only after the thread cutting or before the tapping operation.

[Illustrations]

G17 G90 G00 G54 X0. Y0.
G00 Z100.
M29 S1000
G99 G73 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.

2
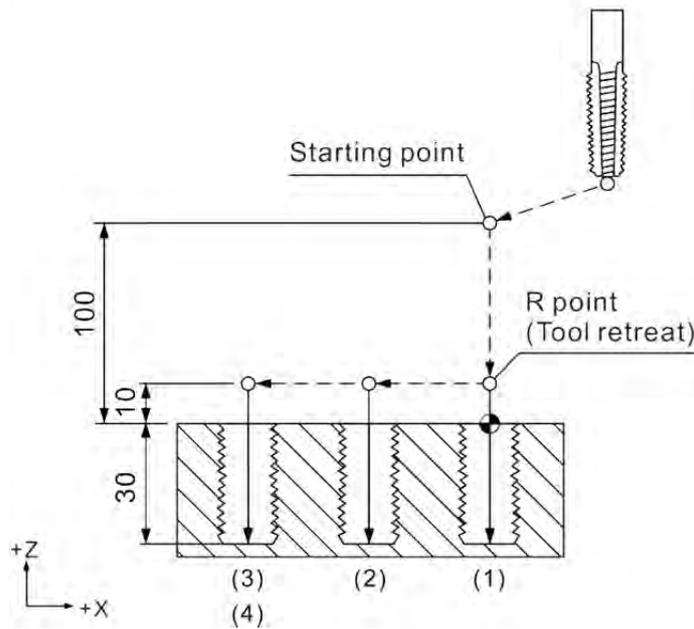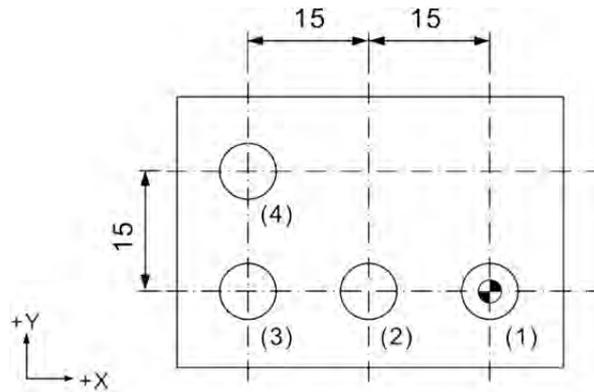
[Illustrations]

G17 G90 G00 G54 X0. Y0.
G00 Z100.
M29 S1000
G99 G73 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.
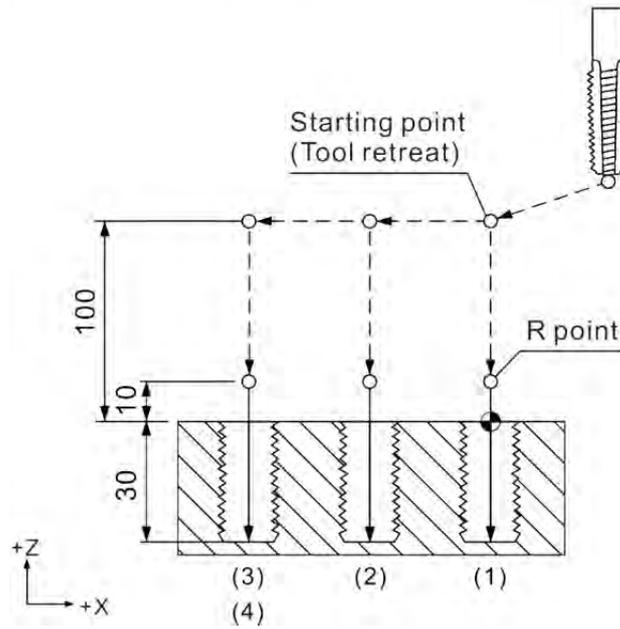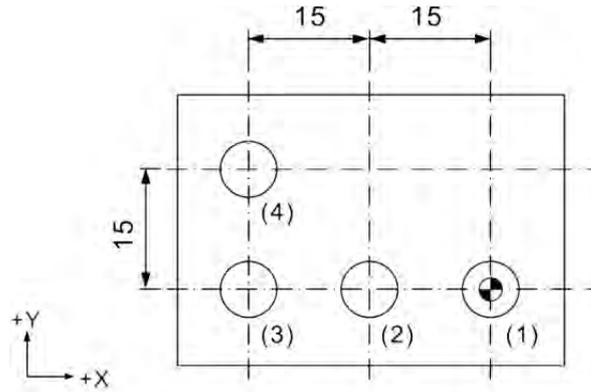
G76: Fine Boring Cycle Command

Command format: G76 X_ Y_ R_ P_ Z_ Q_ F_K_

2

X_ Y_: Ending position of a single block
R_: Initial safety height
P_: Pause time (in the least unit of 1/1000 second), without decimal point.
Z_: Bottom of hole to be tapped
Q_: Offset distance
F_: Feed rate
K_: Number of loops

Command description: This command is for fine boring. When cutting into the given depth, the spindle stops turning and be in position after pausing for the time specified by P_. Then, the tool center moves away from the workpiece surface for a distance set by program Q_, thus the tool makes no contact with the workpiece surface. In this way, the tool is able to return to height R or staring point without scratching the workpiece. The tool center offsets to the distance set by parameter Q_ and then the resumes turning after the Z-axis returns to the original starting position.
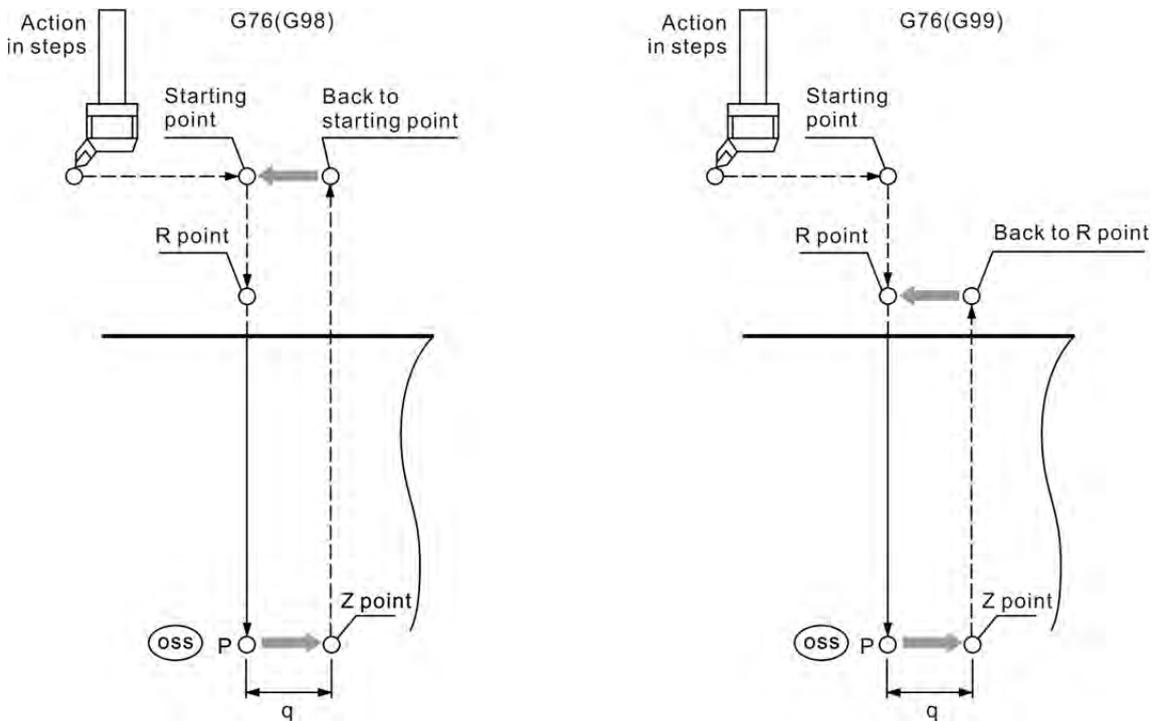
Figure 1: Fine boring cycle
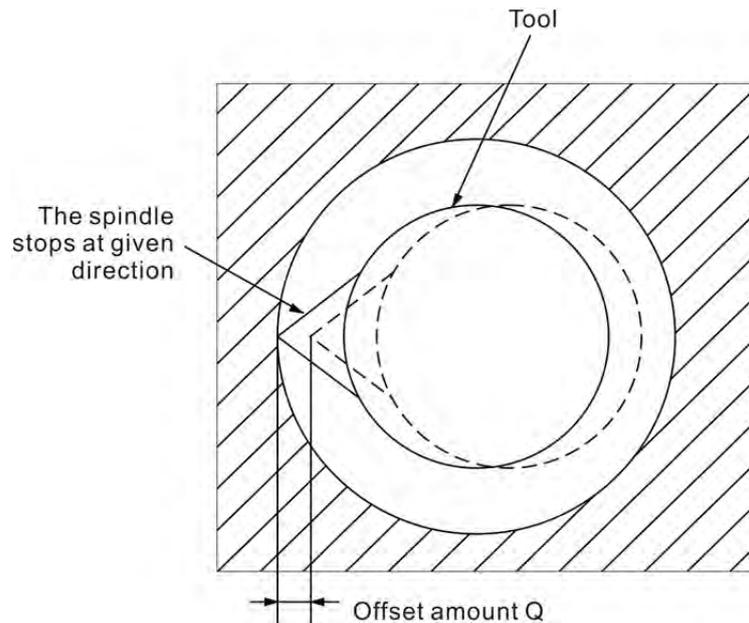
Figure 2: Offset amount for fine boring cycle

See below for movements set by this command (refer to Figure 1):
The firstly boring cutter fast positions to point (X, Y) and height R. Then, it cuts to depth Z with F feed rate before the spindle stops and positions. The cutter tip points to the positioning direction; offset the boring cutter for distance Q away from the hole wall (Figure 2). In this way, the cutter can move out of the hole without scratching the machining surface. Then, the tool center moves back to the original center position after the boring cutter moves to point R or the starting point. Finally, the spindle resumes turning.

The offset amount shown in Figure 2 is given by parameter Q. The value of parameter Q must be positive (if a negative value is given its absolute value will be used instead; to offset 1.0 mm, set the parameter to Q1.0). The offset direction can be set by coordinates (+X, +Y) or (-X, -Y). Parameter Q should not be set too big as the tool center may collide with the workpiece.

The Q value for number of constant cycle is a status value and applies to the cutting amount of commands G73 and G83 as well as the offset amount of command G87. Set proper Q value for commands G73, G76, G83, G87 to avoid improper collision for commands G76 and G87 or cutting for commands G73 and G83.

[Illustrations]

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

2

2
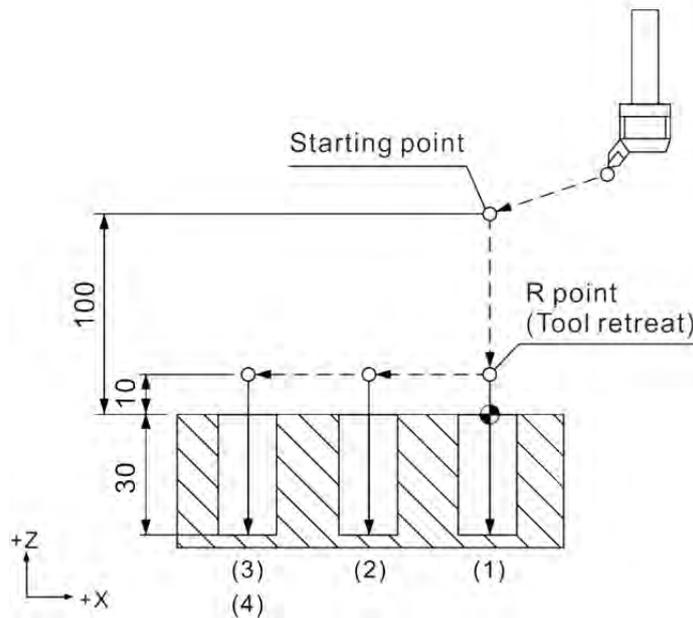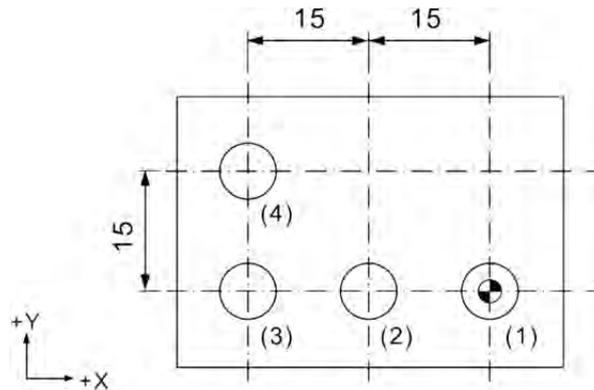
[Illustrations]

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
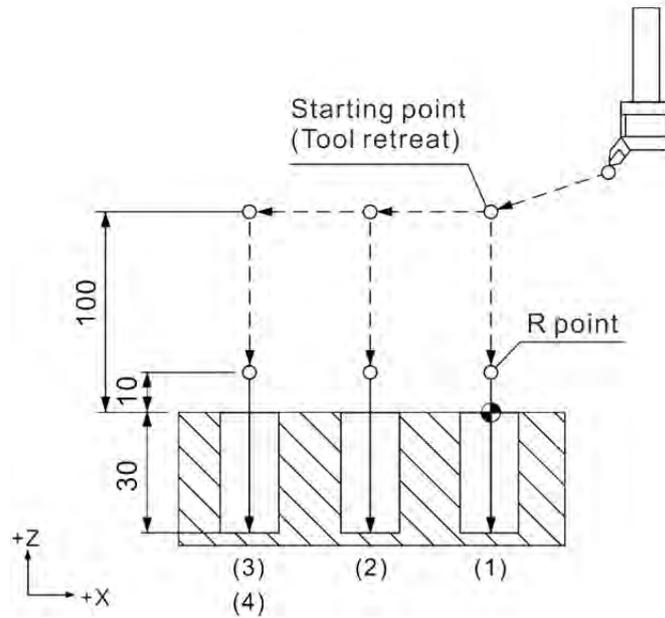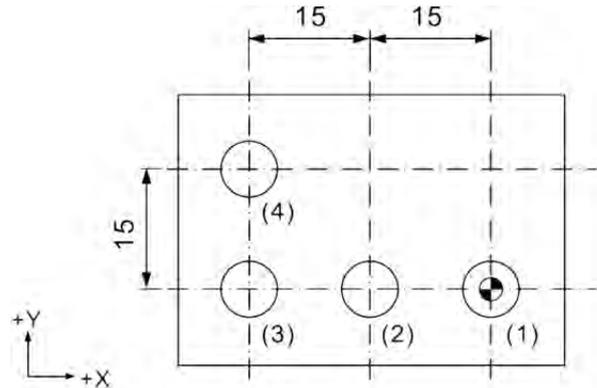
G80: Cycle Cancelling Command

Command format: G80

Command description: All cycle commands are status commands. Please cancel the cycle command status before resuming normal cutting operations. Command G80 cancels the cycle status set by commands G73, G74, G76 and G81 ~ G89.

[Illustrations]
G17 G90 G00 G54 X0. Y0.
Z100.
G99 G73 X0. Y0. Z-20. R10. Q4. K1 F100.
G80
G17 G90 G00 G54 X0. Y0.   ---------------(Cancel cycles set by command G73)
Z100.

G81: Drilling Cycle Command

Command format: G81 X_ Y_ Z_ R_ F_K_

           X_ Y_: Ending position of single block
           Z_: Bottom of hole to be cut
           R_: Initial safety height
           F_: Feed rate
           K_: Number of cycles

Command description: Command G81 is used for drilling loops of general purpose. It drills to given depth Z in one shot without any retreat. The drilling cycle is completed by returning the cutter to point R or the initial point after being drilled to given depth Z with G00 movement. See the figure below for reference.
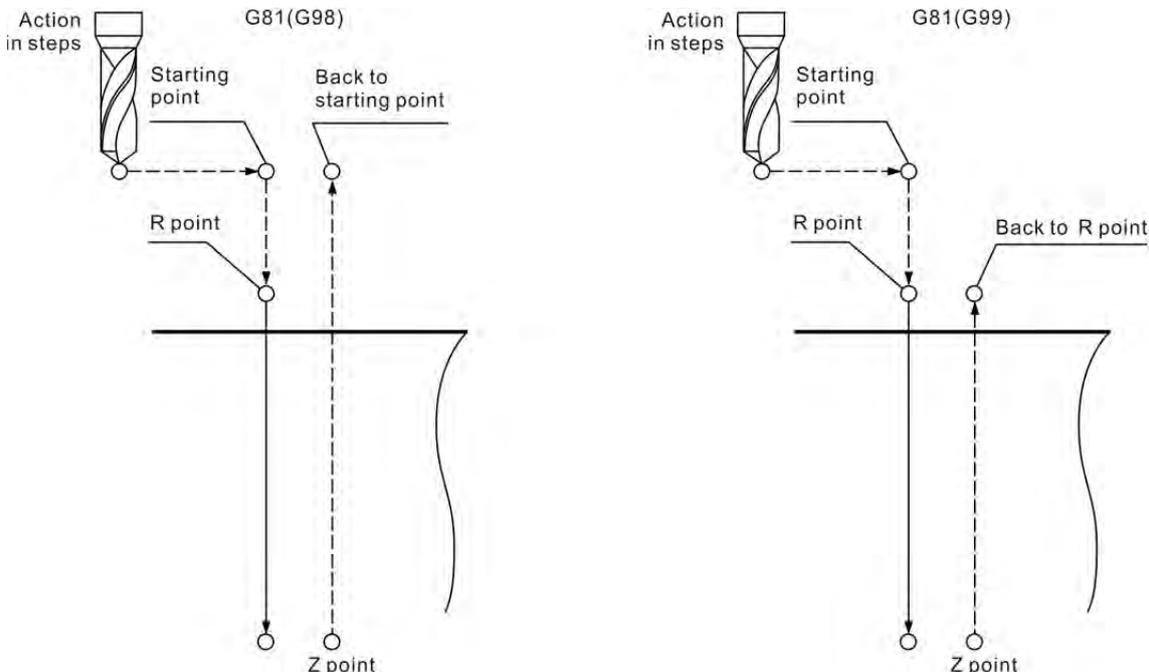
2

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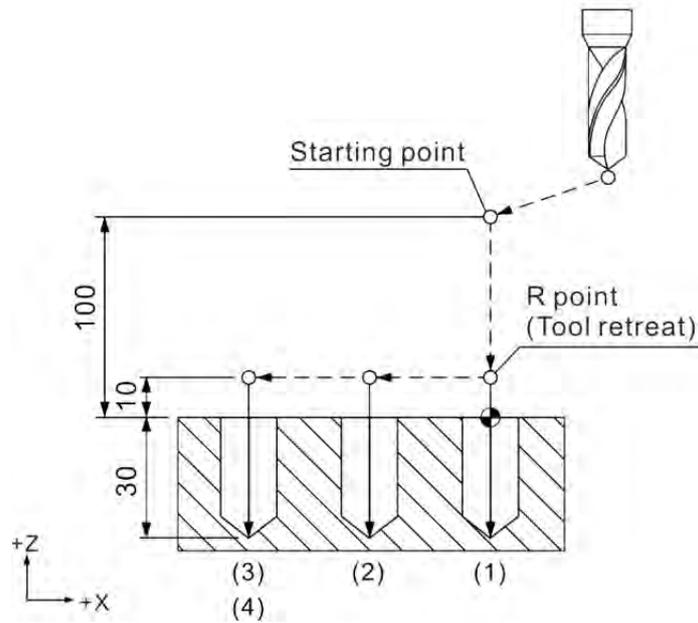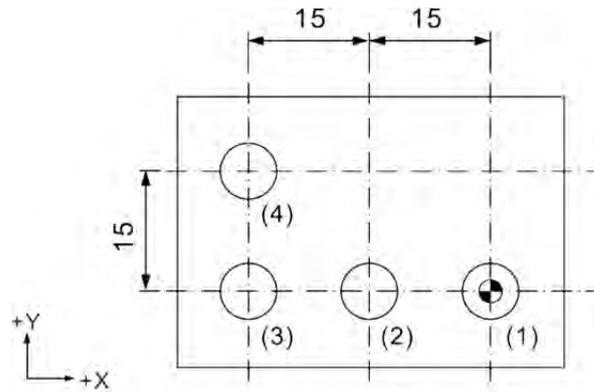
[Illustrations]
M03 S1000
G17 G90 G00 G54 X0. Y0.
G00 Z100.
G99 G81 X0. Y0. Z-30. R10. K1 F100.   --(1)
X-15.   --(2)
X-30.   --(3)
X-30. Y15.   --(4)
G80 G91 G28 X0. Y0. Z0.
M05

2
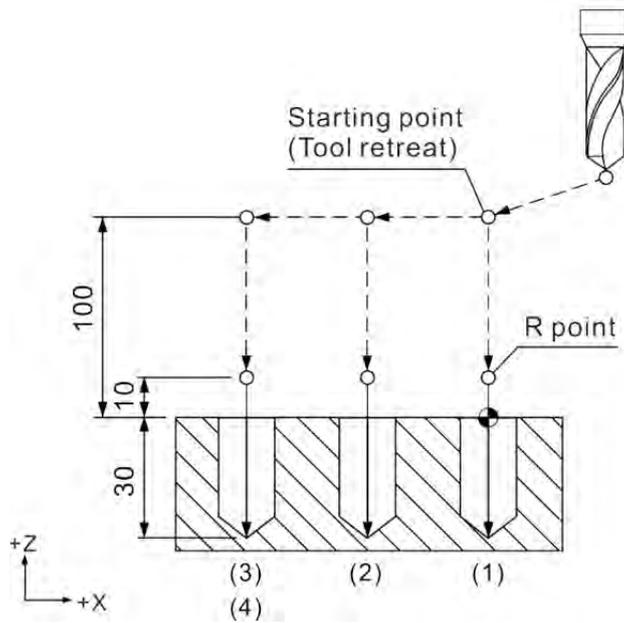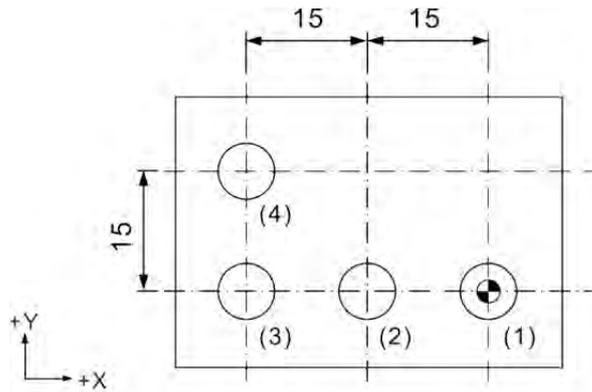
G82: Countersunk Drilling Cycle Command

Command format: G82 X_ Y_ R_ Z_ P_ F_K_

X_ Y_: Ending position of single block
Z_: Bottom of hole to be cut
R_: Initial safety height
P_: Pause time (in the least unit of 1/1000 second), without decimal point.
F_: Feed rate
K_: Number of cycles

Command description: Command G82 functions the same as command G81 except that the former can designate pause time P_ in a single block. That is, command G82 differs from G81 only in that it can set up a pause time P after cutting to the bottom of the hole. Command G82 is ideal for drilling counter bore, spot facing, and for machining holes that need precise depth. Command G82 can set the tool to stay within a specified time span after cutting to the bottom of the hole for a more smooth hole wall and accurate hole depth.
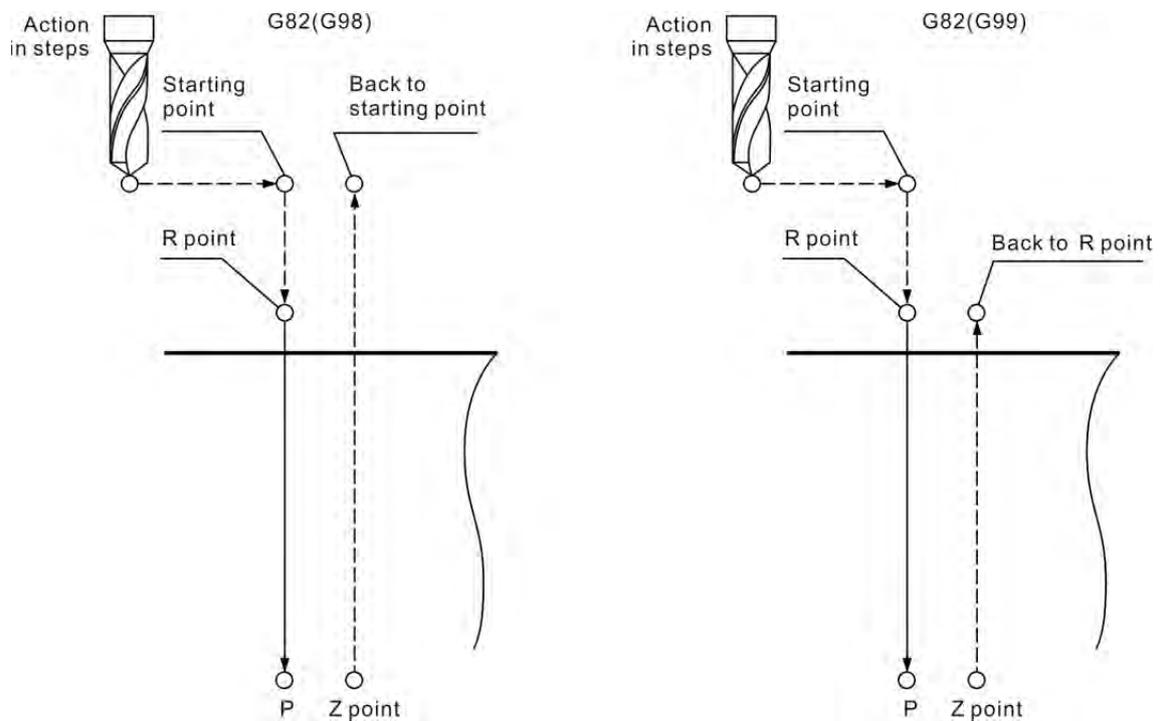
[Illustrations]
M03 S1000
G17 G90 G00 G54 X0. Y0.
G00 Z100.
G99 G82 X0. Y0. Z-30. R10. P1000 K1 F100.   ---(1)
X-15.   ---(2)
X-30.   ---(3)
X-30. Y15.   ---(4)
G80 G91 G28 X0. Y0. Z0.
M05

2

2
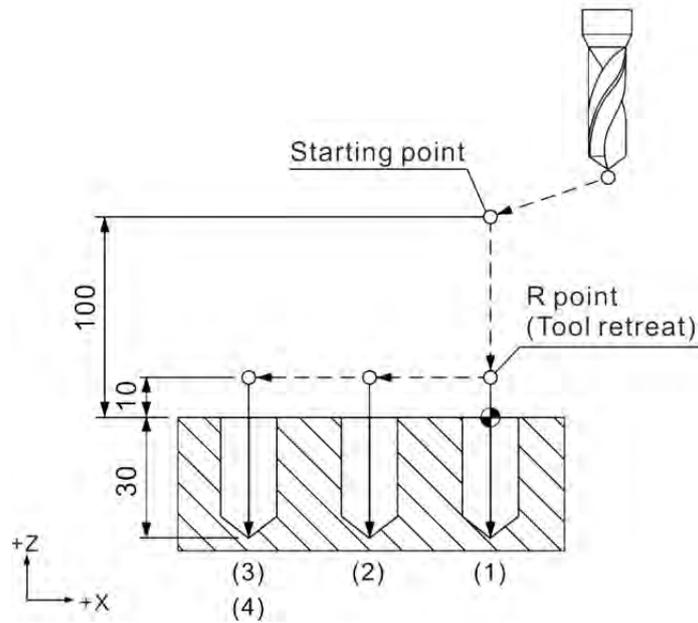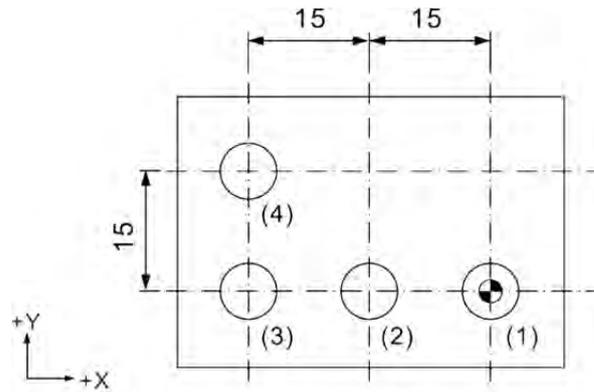
[Illustrations]
M03 S1000
G17 G90 G00 G54 X0. Y0.
G00 Z100.
G99 G82 X0. Y0. Z-30. R10. P1000 K1 F100.   --(1)
X-15.   --(2)
X-30.   --(3)
X-30. Y15.   --(4)
G80 G91 G28 X0. Y0. Z0.
M05
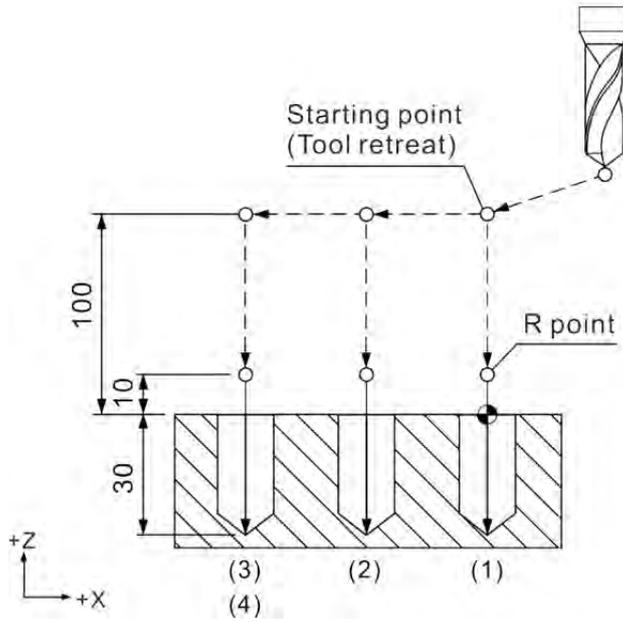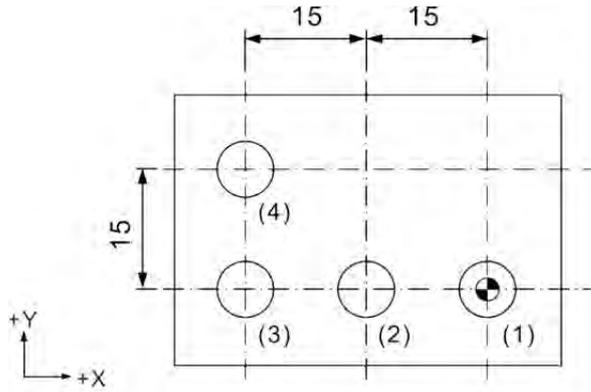
G83: Deep Hole Peck Drilling Cycle Command

Command format: G83 X_ Y_ R_ Q_ Z_ F_ K_

2

X_ Y_: Ending position of single block
Z_: Bottom of hole to be cut
R_: Initial safety height
Q_: Peck cutting depth
F_: Feed rate
K_: Number of cycles

Command description: Command G83 functions the same as command G73 except that the former returns back to height R after each deep drilling so that cutting debris can be fully removed and cutting fluid may flow into the hole for better cooling.

The machine is set to move as described below:

The tool returns back to point R after cutting to depth Q. Then, it is positioned to a point higher than the last drill ending point of distance "d". Next, it starts drilling for depth "q+d" in the next drilling process. And it drills to given position Z before returning back to position R or the initial point to finish drilling a deep hole.
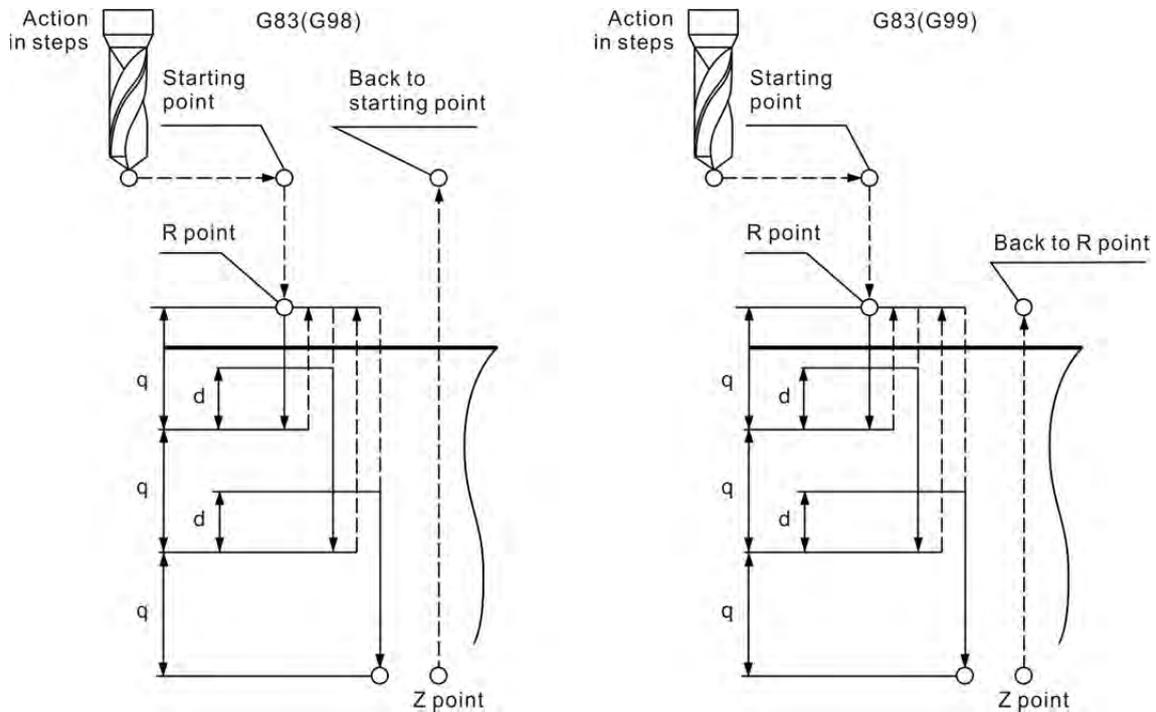
2

[Illustrations]

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
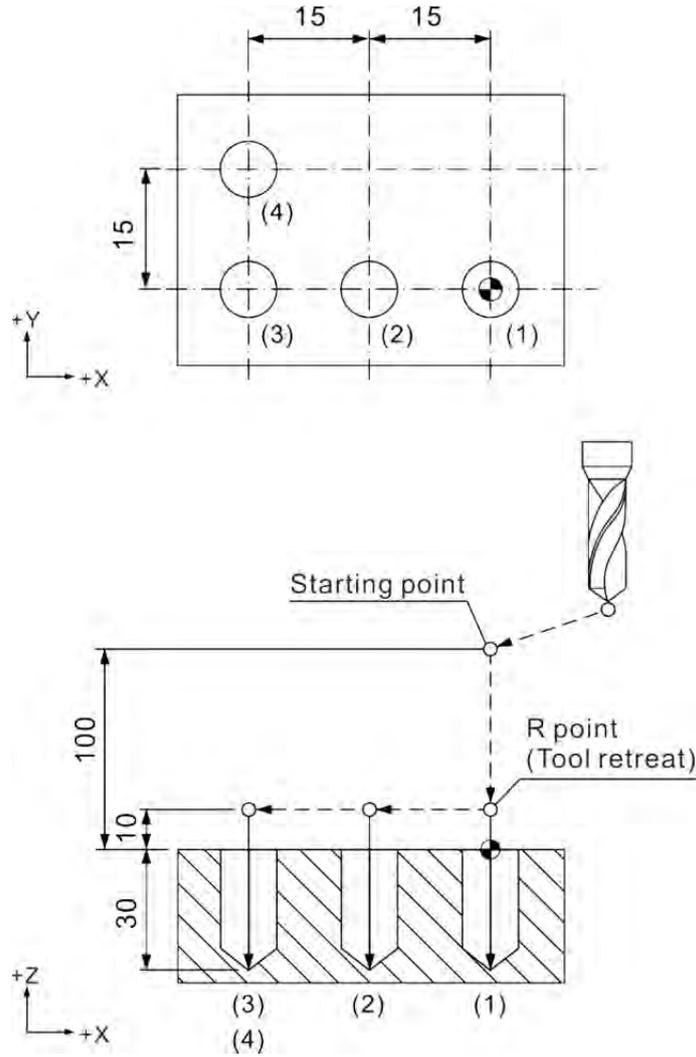
[Illustrations]

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
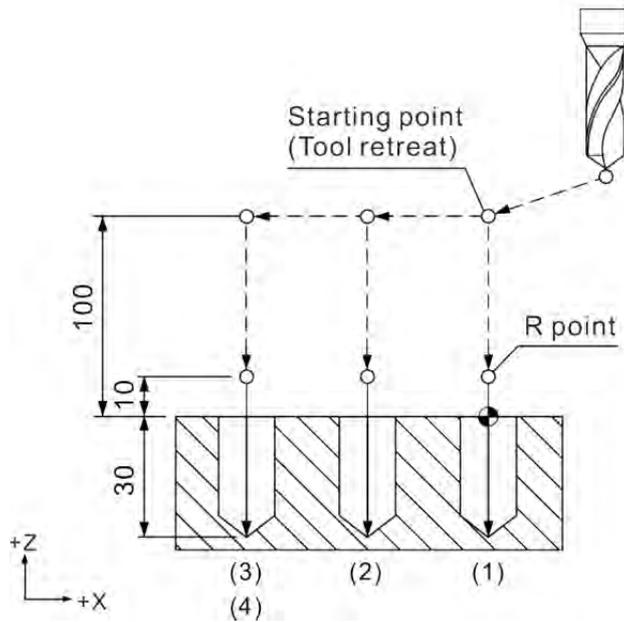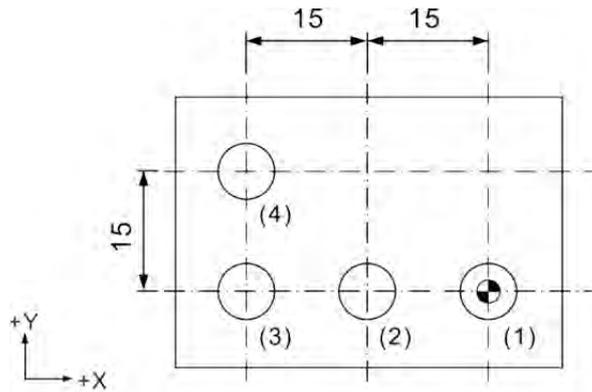
2

G84: Right Spiral Tapping Cycle Command

Command format: G84 X_ Y_ R_Q_Z_ P_ F_ K_

X_ Y_: Ending position of single block
Z_: Bottom of hole to be cut
R_: Initial safety height
Q_: Peck drilling depth of each time
P_: Pause time (in the least unit of 1/1000 second), without decimal point.
F_: Spiral feed rate
K_: Number of cycles

Command description: Command G84 taps right-handed threads. It functions the same as command G74 except that the turning direction of G84 is opposite to that of command G74. All the remaining operations and notes are the same as command G74.
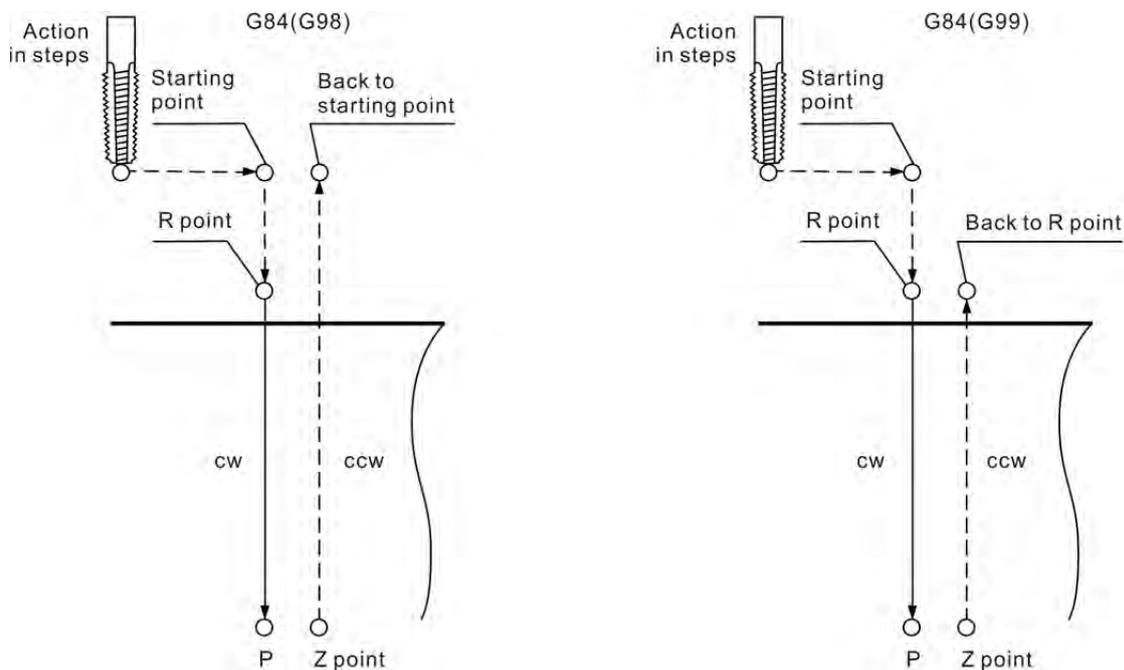
[Illustrations]

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.
M05

2
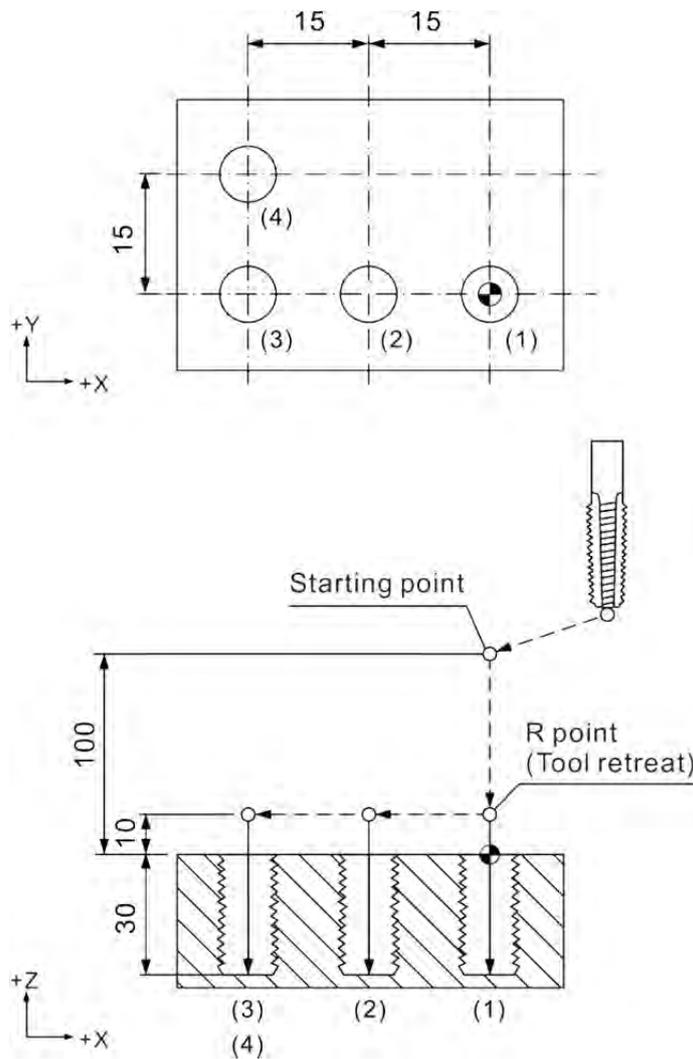
[Illustrations]

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.
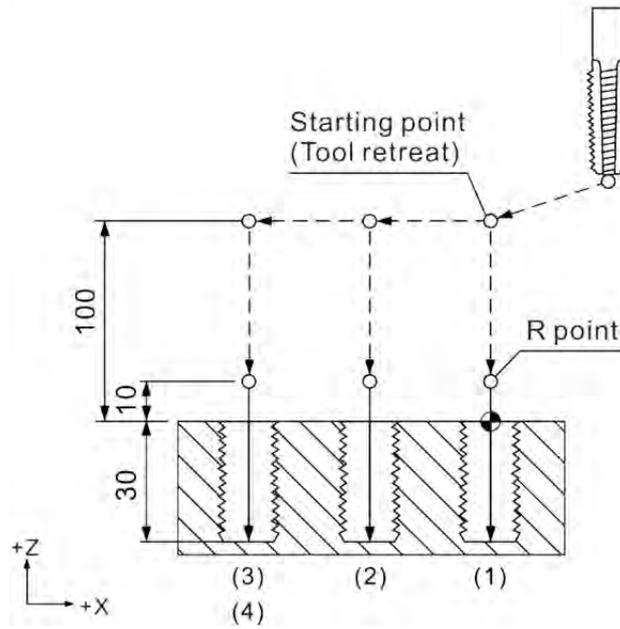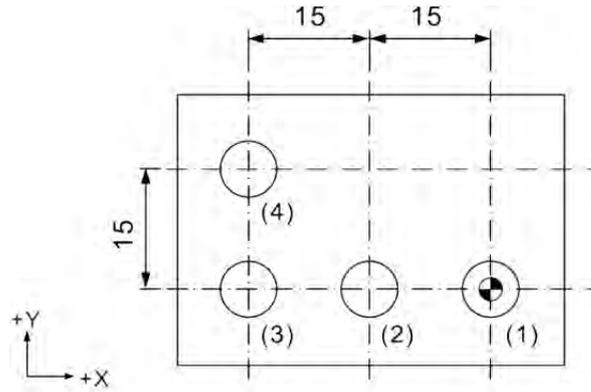
G85: Broaching Cycle Command

Command format: G85 X_ Y_ R_ Z_ F_ K_

2

                        X_ Y_: Ending position of single block
                        Z_: Cutting depth position
                        R_: Initial safety height
                        F_: Feed rate
                        K_: Number of cycles

Command description: Command G85 improves boring precision significantly. It usually works with reaming or boring cutter for holes with high reaming or boring accuracy.
See below for the movements set by this command:
The tool cuts to given depth Z from point R in given feed speed F before lifting the up to point R at the same feed speed of F. Then, it returns to point R (G99) or initial point (G98).
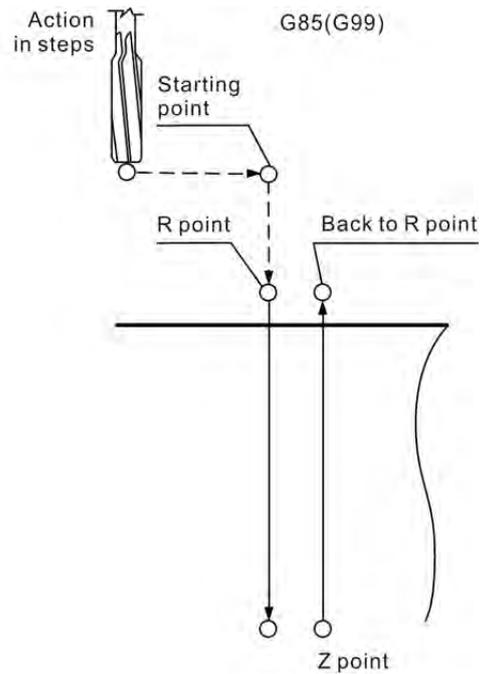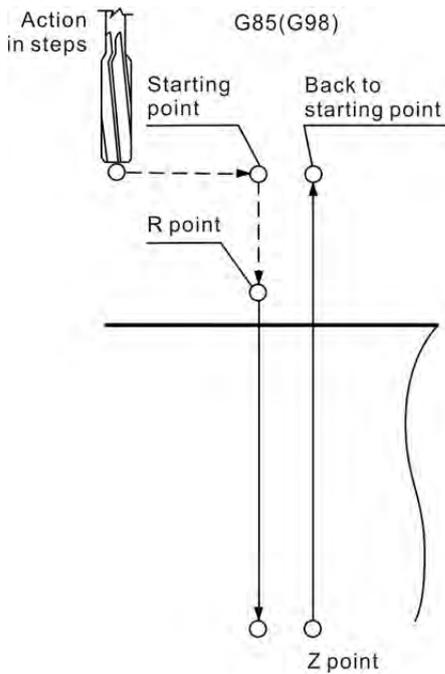
2

[Illustrations]

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

2
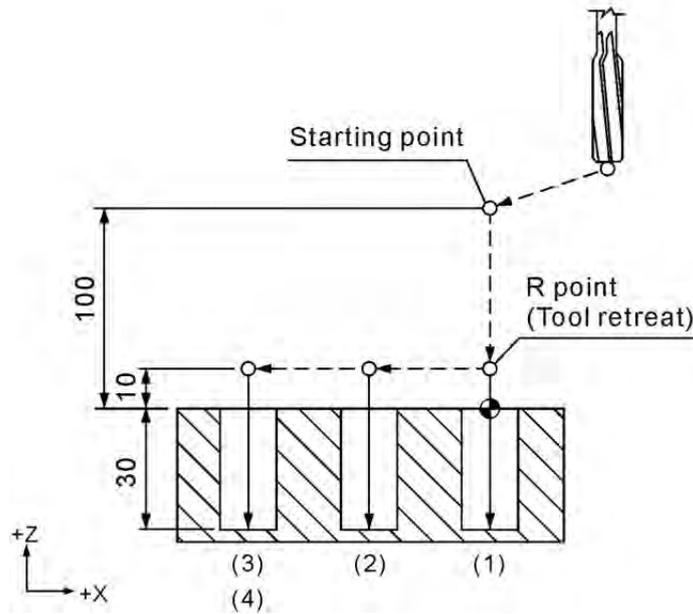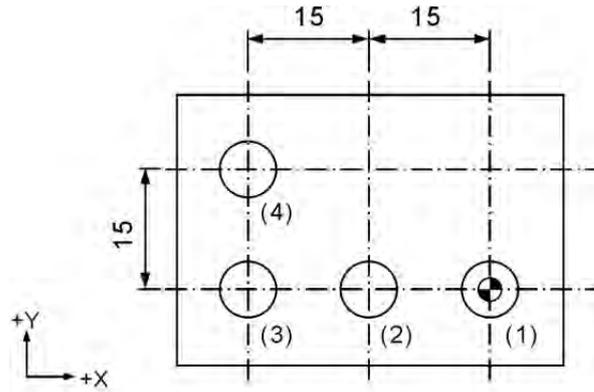
[Illustrations]

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
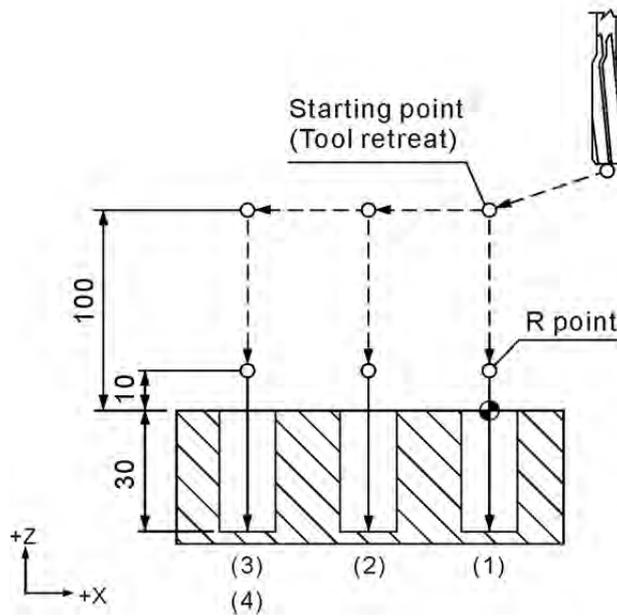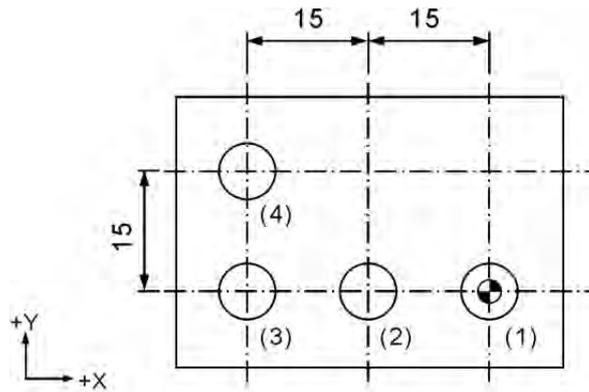
2

G86: Rough Boring Cycle Command

Command format: G86 X_ Y_ R_ Z_ F_ K_

X_Y_: Ending position of single block
Z_: Bottom of hole to be cut
R_: Initial safety height
F_: Feed rate
K_: Number of cycles

Command description: See the figure below for the motions set by this command. After bored to given depth Z, the spindle stops turning and lifts to the initial height with fast feeding command G00. A full boring cycle is completed after the spindle returns to the initial height and resumes positive direction turning. There is no Q parameter in this command and so it lacks any offset or peck cutting motion. As the tool keeps contacting the working surface constantly and is not turning when lifting off the hole, the hole wall may be left with minor scratches. Command G86 is thus usually used for rough boring.
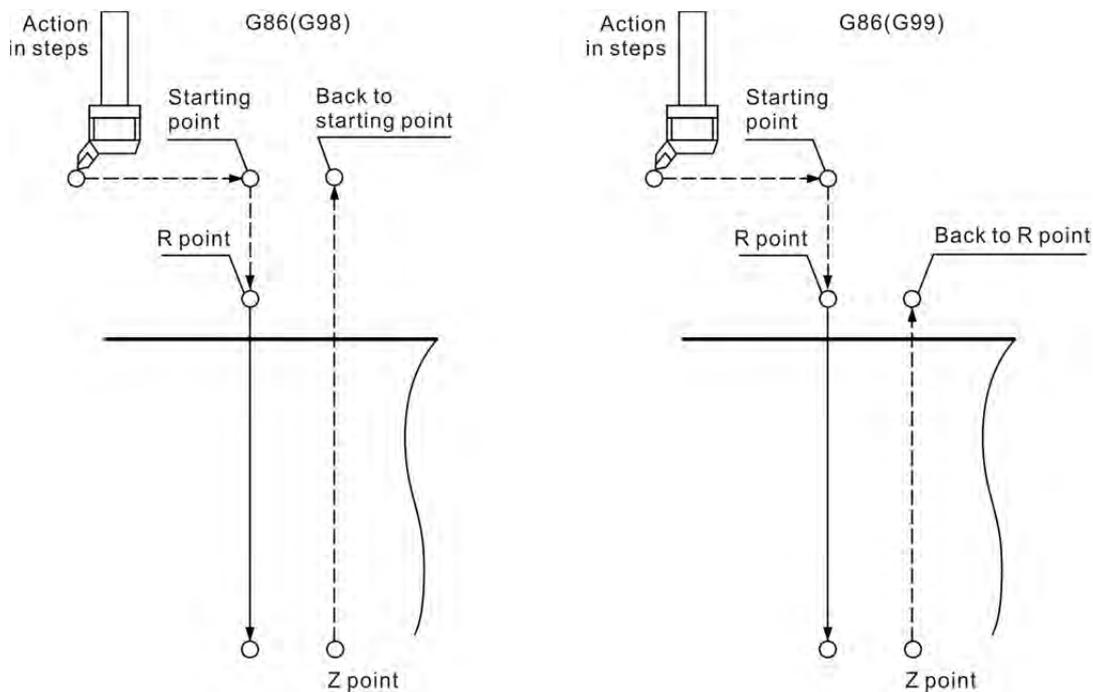
[Illustrations]

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

2

2
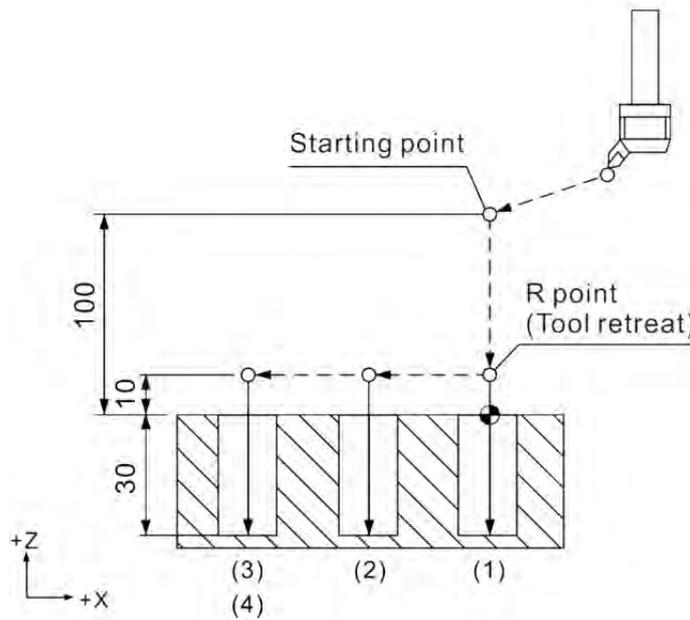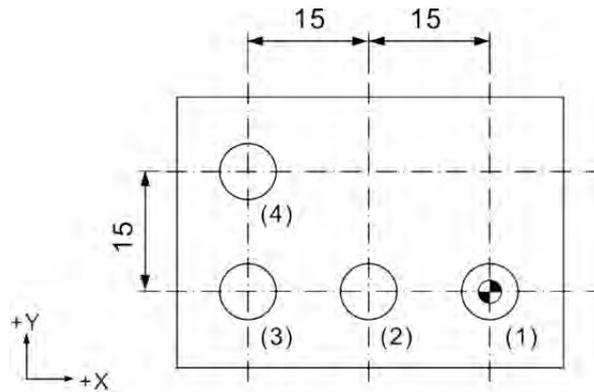
[Illustrations]

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
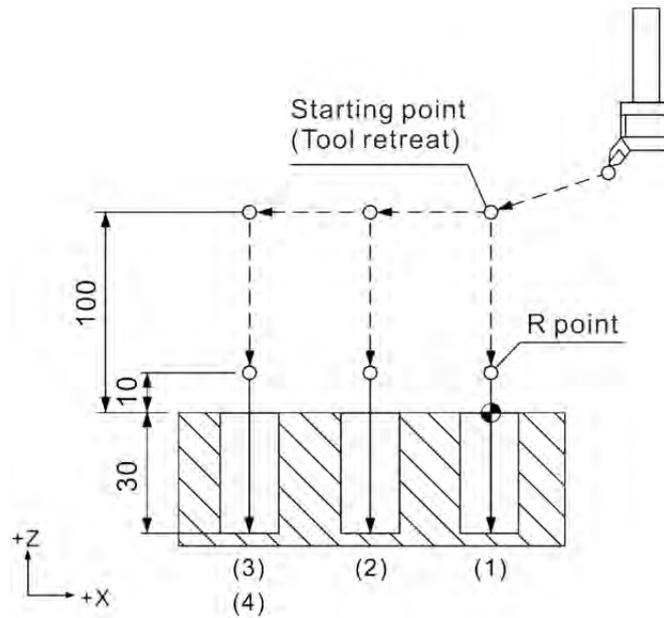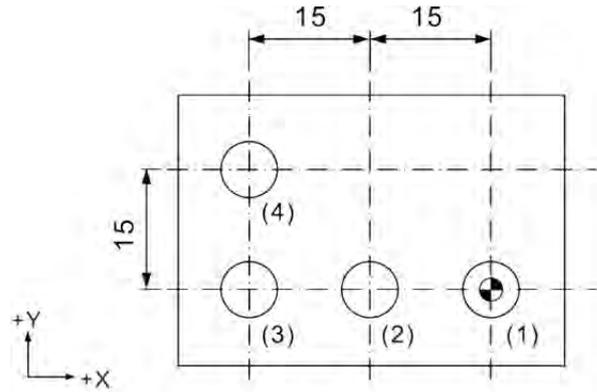
G87: Rear Boring Cycle Command

Command format: G87 X_ Y_ R_ Z_ Q_ P_ F_ K_

2

X_Y_: Ending position of single block
Z_: Bottom of hole to be cut
R_: Initial safety height
Q_: Offset distance
P_: Pause time (in the least unit of 1/1000 second), without decimal point.
F_: Feed rate
K_: Number of cycles

Command description: The tool fast positions to the position set by coordinates (X, Y) before the spindle moves to the position the cutter tip in a given direction. The tool center then offsets a distance of value Q. In this way, the cutter blade does not contact the hole wall when moving to height R. After moving to height R and complete fast positioning, the tool center moves to the original position given by coordinates (X, Y) and the spindle starts to turn forward. The tool now starts cutting from point R to point Z. The tool center then offsets a distance Q and the spindle positions after reaching point Z. The tool then returns back to the Z-axis initial point with fast positioning and resets the offset after reaching the initial point. The offset value Q equals the one set by command G76. Please note that command G87 does not support operation in G99 mode.

2
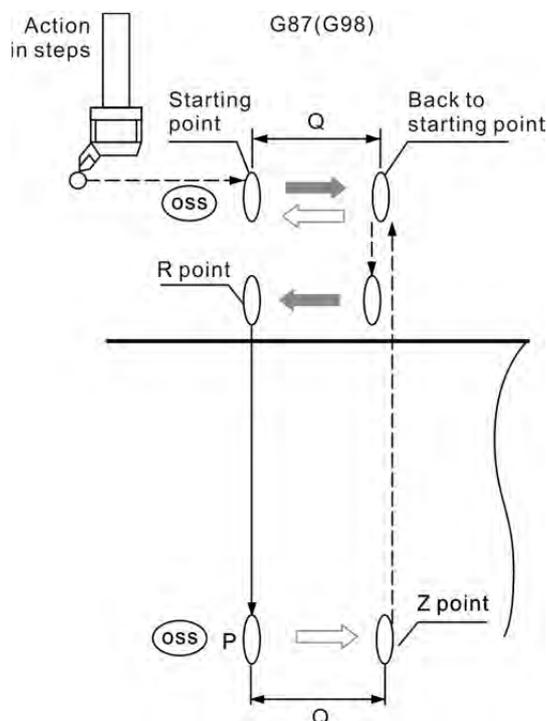
[Illustrations]

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
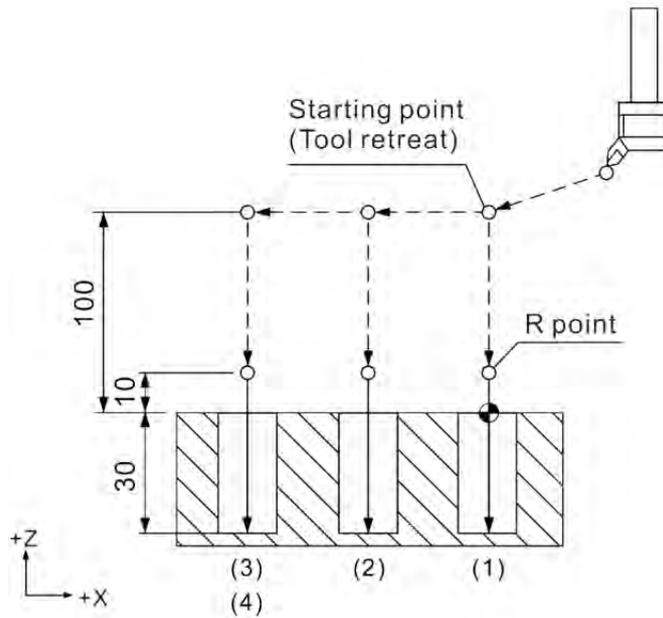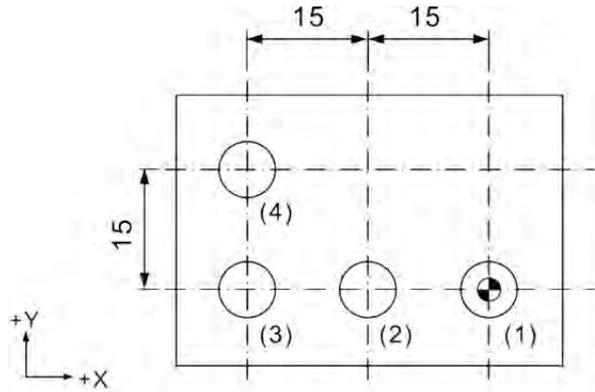
G88: Boring Cycle Command

Command format: G88 X_ Y_ R_ Z_ P_ F_ K_

2

X_Y_: Ending position of single block
Z_: Bottom of hole to be cut
R_: Initial safety height
P_: Pause time (in the least unit of 1/1000 second), without decimal point.
F_: Feed rate
K_: Number of cycles

Command description: Command G88 sets the tool to cut from point R to given point Z and pause at point Z for the time specified in parameter P. The spindle stops turning and execution (same as executing command M00) after a pause period and fast retreats the tool to point R or the initial point only after the Cycle start key is pressed.
In addition, the operator may manually move (by switching the system to **MPG mode**) the tool upward in Z+ direction after cutting to point Z. Switch the system mode back to **Auto** and press the program execution key to resume program control. After the Z-axis is lifted to point R (G99) or the initial point (G98), a full cycle of command G88 is accomplished. This command is used for blind hole boring.
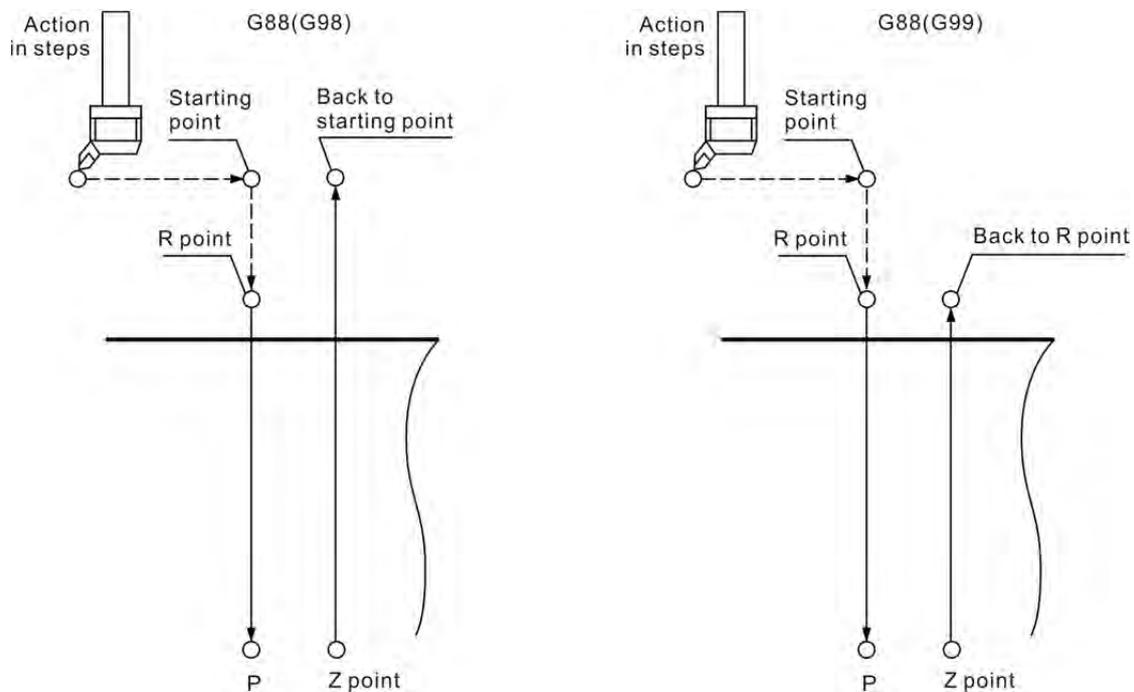
2

[Illustrations]

```
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
```
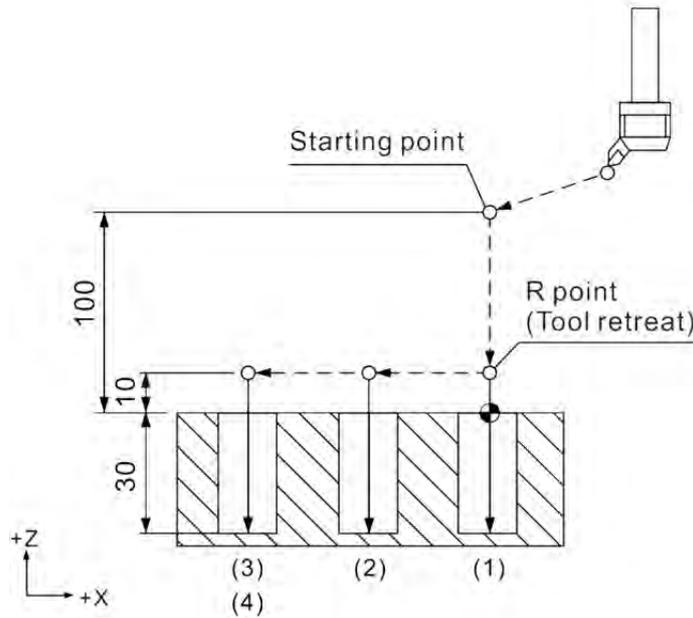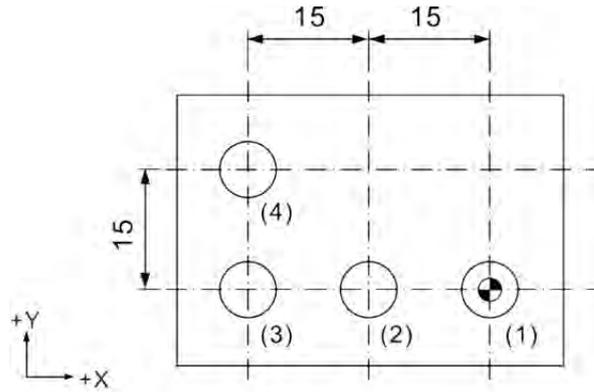
[Illustrations]

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
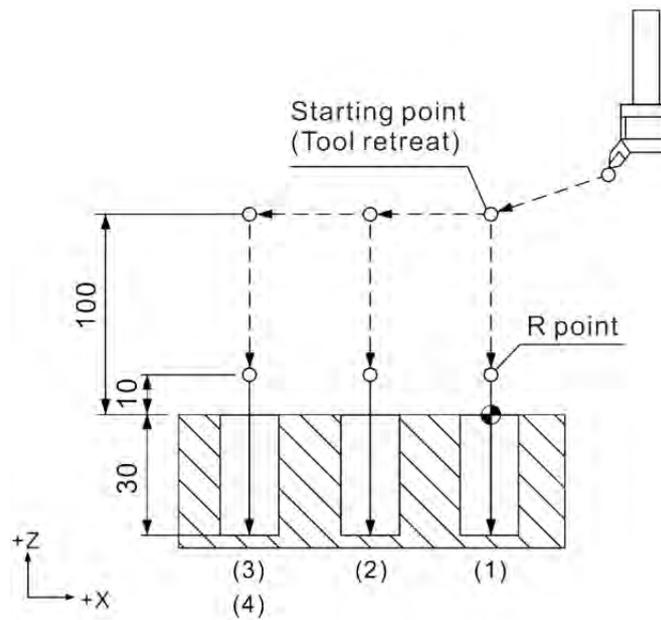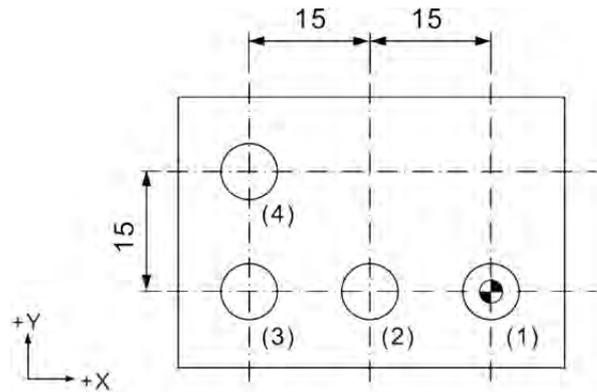
2

G89: Boring Cycle Command

Command format: G89 X_ Y_ R_ Z_ P_ F_ K_

2

X_Y_: Ending position of single block
Z_: Bottom of hole to be cut
R_: Initial safety height
P_: Pause time (in the least unit of 1/1000 second), without decimal point.
F_: Feed rate
K_: Number of cycles

Command description: Command G89 is for blind hole reaming. It functions the same as command G85 except that a pause P can be added at point Z. Both its downward and upward movement of the tool in the Z-axis is in F feed rate. With the added pause at point Z, the machining accuracy for depth and diameter of the hole can be greatly improved by keeping the tool staying for a while when cut to position Z.
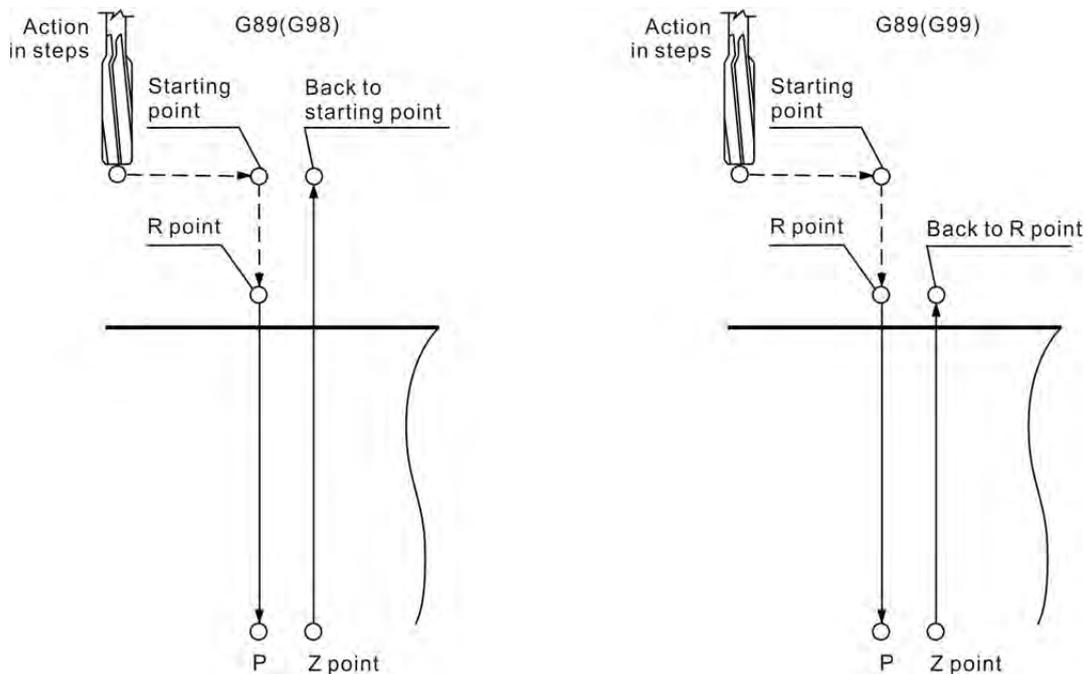
[Illustrations]

M03 S1000
G17 G90 G00 G54 X0. Y0.
G00 Z100.
G99 G89 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

2

2
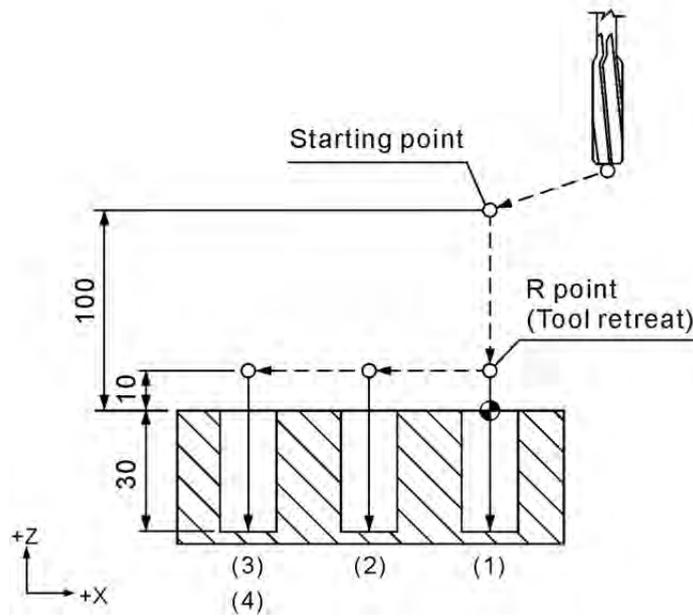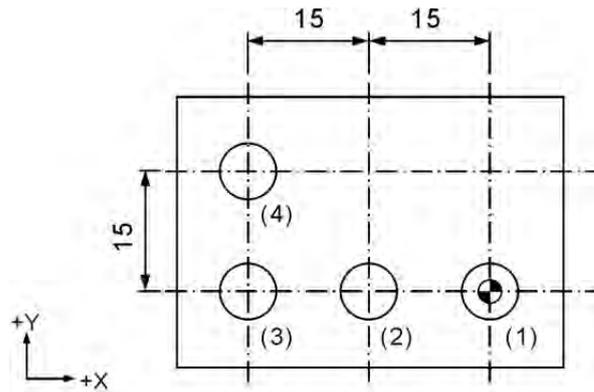
[Illustrations]

M03 S1000
G17 G90 G00 G54 X0. Y0.
G00 Z100.
G99 G89 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
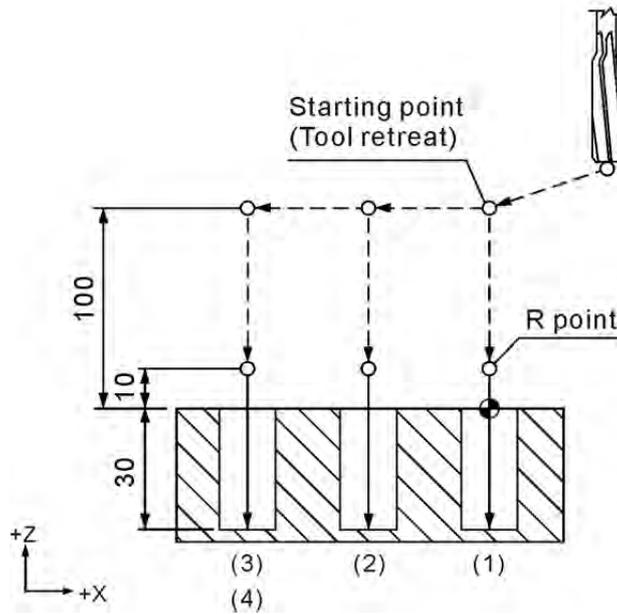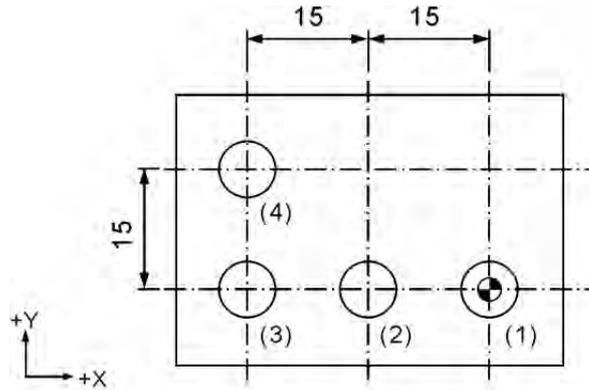
G90: Absolute Coordinate System Command

Command syntax: G90 X_ Y_ Z_

Command description: Command G90 is a status command of continuous effects. When this command is executed, all axial commands and coordinate works with their absolute values specified. That is, the tool moves relative to a reference point of the origin in the workpiece coordinate system. Each axial command after command G90 moves an actual distance relative to the origin of the workpiece coordinates.

2

[Illustrations]

From point 1 of coordinates (X10,Y10) to point two (X30,Y30) the tool moves an actual distance of X20 and Y20 as shown in the figure below.
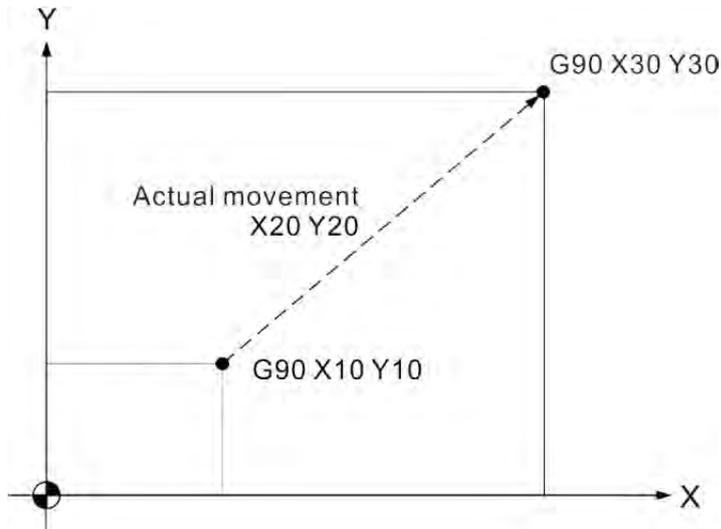

```
O1000
G01 G90 X10.0 Y10.0 F500.0
X30.0 Y30.0
```

G91: Incremental Coordinate System Command

Command format: G91 X_ Y_ Z_

Command description: The increment command G91 sets all axial movements in a motion program of a single block. It will move or rotate incrementally from the current position to the specified position. The G91 command is a status command that cancels command G90 once it is activated.

[Illustrations]

From point 1 of coordinates (X10, Y10) to point two (X20, Y20) the tool moves an actual distance of X20 and Y20 to point (X30, Y30) of the actual mechanical coordinates as shown in the figure below.


```
O1001
G01 G91 X10.0 Y10.0
F500.0
X20.0 Y20.0
```
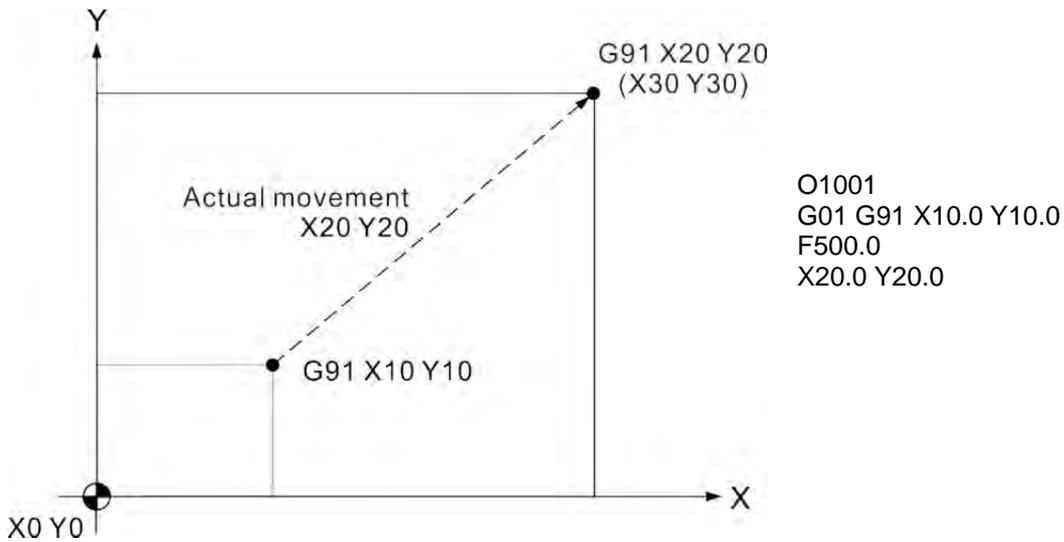
Example of using G90 and G91 commands together:
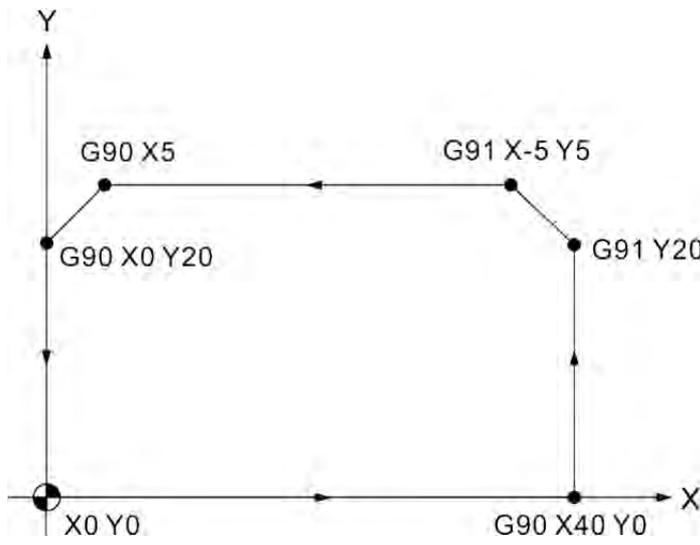

```
O0010
G01 G90 X0 Y0 F1000
X40.0
G91 Y20.0
X-5.0 Y5.0
G90 X5.0
X0 Y20.0
Y0
M30
```

2

2

G92: Coordinate System Setup Command

Command format: G92 X_ Y_ Z_

Command description: Command G92 X0 Y0 Z0 sets the current tool position as the zero point of an absolute coordinates system. The absolute command in the program calculates positions relative to this point. Values of absolute coordinates and current positions are all refreshed with new values given by parameters of command G92 X_ Y_ Z_ (as long as any of the three parameters X, Y and Z have values assigned to them).

Note:
(1)   When G92 is active, the program will keep running until encountering commands for end of program (M02 / M30) and then the status set by G92 will be canceled.
(2)   To reset the status set by command G92, press the RESET key.

[Illustrations]


```
O0005
G00 G90 G54 X0.0 Y0.0
G00 G92 X0.0 Y0.0
G01 X100.0 Y0.0 F1000.0
X100.0 Y50.0
X0.0 Y50.0
X0.0 Y0.0
M30
```
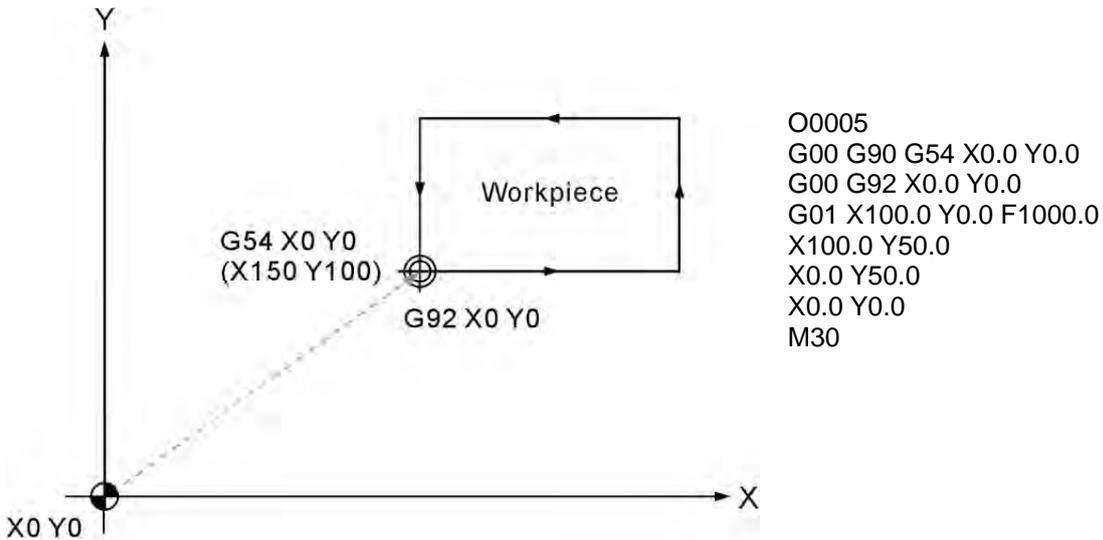
G94: Feed Rate (mm/min) Setup Command

Command format: G94 G01 X_ Y_ Z_ F_

Command description: The feed rate set by command G94 is in the unit of mm/min. This status set command remains effective. It sets the tool to cut at the speed given by parameter F in the unit of mm/min. G94 can be executed along with the motion block simultaneously. It can be executed in single block alone as well. It is commonly used by most milling machines for feed rate calculation.
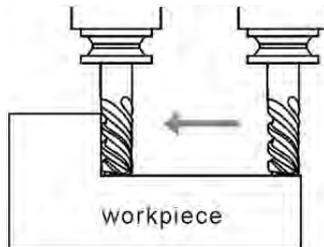
Feed rate in unit of maintenance/min or inch/min.

G98: Return to the Initial Point of the Fixed Cycle

Command format: G98 G8_ X_ Y_ Z_ R_ F_

Command description: Command G98/G99 is a status command for returning the tool back to a specified height after the execution of a fixed loop command ends. Command G98 returns the tool back to its initial point after a cycle command. After the execution of a fixed cycle command ends, the G98 command in the program returns the tool back to its initial point of this fixed cycle command. Command G98 can be reset by command G99. At the system initialization phase, you may set command G98 to direct the system on how to resume the initial points as illustrated in the figure below.

2
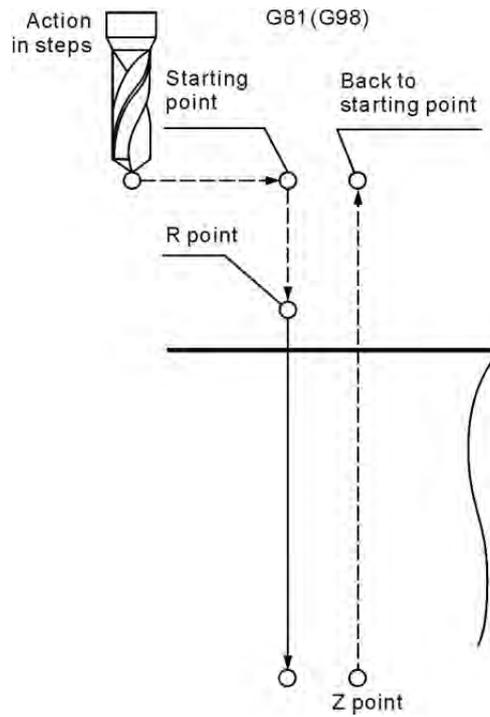
G99: Return to the R Point of the Fixed Cycle

Command format: G99 G8_ X_ Y_ Z_ R_ F_

Command description: Command G98/G99 is a status command for returning the tool back to a specified height after the execution of a fixed cycle command ends. Command G99 returns the tool back to its reference point (R) after a cycle command. Please use command G98, instead of G99, to return the tool back to its initial point after a cycle command.
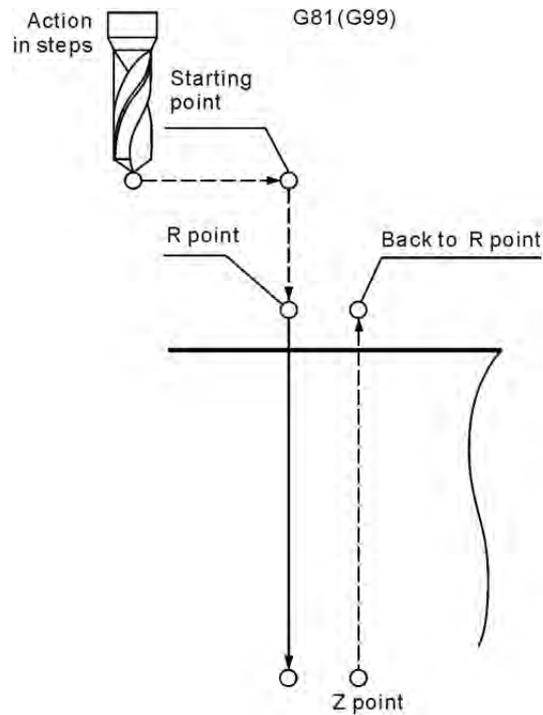
M Codes

<div style="text-align: right; font-size: 3em;">3</div>

This chapter describes functions of general auxiliary M codes.

3

The auxiliary function M codes are used in turning the on and off functions of the machine. Its format is a capital letter M suffixed with three digits. Some system-defined M codes can be executed for program control without any MLC coding.

3.1 M00 program stop (non-optional)

Command description: When an M00 command is executed when program running, the program stops immediately after the line containing the M00 command. This means that, the program stops at where the M00 command is executed. To resume the program execution, press the program execution key once again. The M00 command is used for inspecting tools or workpiece's appearance and dimensions when cutting.

3.2 M01 program stop (optional)

Command format: M01;

Command description: M01 command functions the same way as M00 command does except that it does not stop a program from running on its own. It comes effect only when the optional stop key on the secondary control panel is pressed. If the optional stop is not enabled, the program ignores M01 command and keeps on running until executing an M00 command or program ending command with optional stop function enabled.

3.3 M02 end of a program

Command format: M02;

Command description: M02 command is commonly placed at the end of a machining program to instruct the controller that the running program is to be ended. For an M02 command within a program, the content that follows M02 will be ignored and is regarded as program finished; the cursor stops at the block of M02 command.

3.4 M30 end of a program, return to start

Command format: M30;

Command description: M30 command is commonly placed at the end of a program to instruct the
controller that the running program is to be ended. When an M30
command is located within a program, the program stops running after
executing M30 and then returns the cursor back to the beginning of the
program.

M30 and M02 commands function the same way except that M02
command keeps the cursor at the block of M0 command while M30
command returns the cursor back to the beginning of the program.

3.5 M98 subroutine call

Command format: M98 P__ L__;

Command description: You may group fixed or repetitive actions in a program into a subroutine
to simplify the structure and reduce the length of the program. The main
program calls a subroutine, which in turn can call another subroutine in
the next level, up to eight layers. When the controller reads an M98
command, it jumps to the designated subroutine and executes it
according to the setting number of times.

P__: indicates program code of the subroutine; L__: Indicates the number
of times the subroutine is to be executed.

[Illustrations]

Main program	Sub program
N0010	O1010
N0020 M98 P1010	N1020
N0030	N1030
N0040	N1040
N0050	N1050
N0060	N1060
N0070	N1070
N0080	N1080
N0090	N1090
N0100	N1100 M99
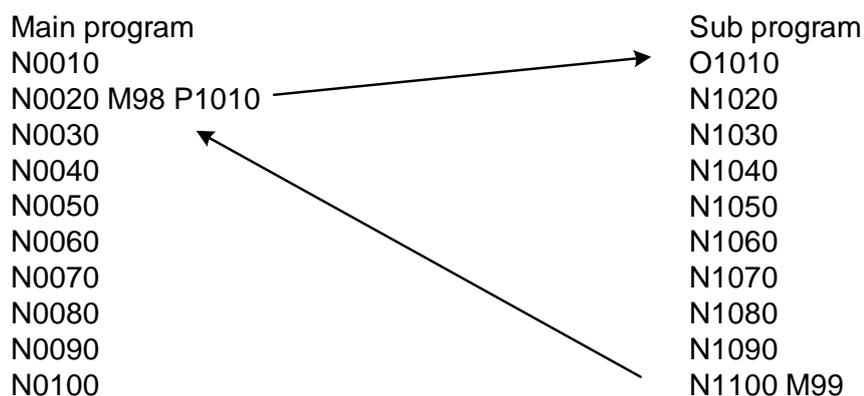
3.6 M99 return from a subroutine

Command format: M99;

Command description: M98 command can run a subroutine from the main program. If the user desires to return to the main program, issue M99 command again. Thus, after executing to the block of M99, the subroutine will call the next block and then start to execute the command.

[Illustrations]

Main program	Sub program
N0010	O1010
N0020 M98 P1010	N1020
N0030	N1030
N0040	N1040
N0050	N1050
N0060	N1060
N0070	N1070
N0080	N1080
N0090	N1090
N0100	N1100 M99
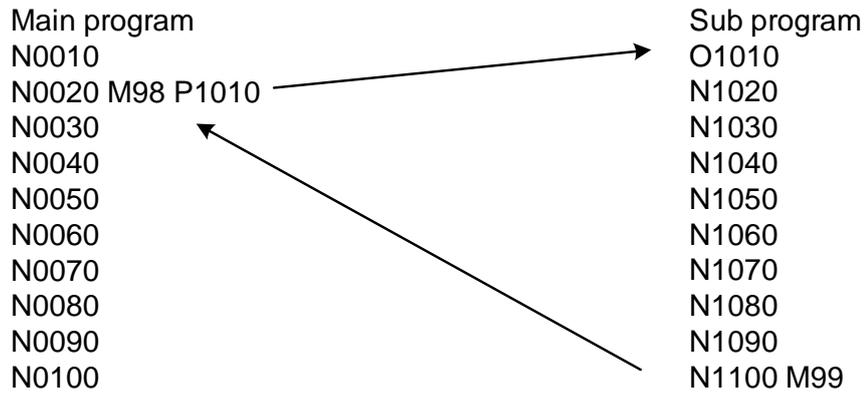
3.7 Table of commonly used M codes

The commonly used **M** codes are summarized in the table below. The functions of an **M** code are defined in the MLC except those pre-defined by the system. Actual functions of the **M** codes vary with individual machines.

M code	Function	Remark
M00	Program stop (non-optional)	System defined
M01	Program stop (optional)	System defined
M02	End of program	System defined
M03	Spindle on clockwise	
M04	Spindle on counterclockwise	
M05	Spindle stop	
M06	Tool change	
M08	Coolant On	
M09	Coolant Off	
M19	Spindle positioning	
M20	Release spindle positioning	
M28	Cancel rigid tapping	
M29	Rigid Tapping	
M30	End of program, return to start	System defined
M98	Subroutine call	System defined
M99	Return from a subroutine	System defined

(This page is intentionally left blank.)

3

Marco and Variables

4

Macros and variables are essential parts in program execution. The operational command and examples will be elaborated in this chapter.

4

4.1 Variables

You can assign values of variables contained in an NC program. You can calculate and modify multiple variable values in advance to adapt a program for a range of conditions. A variable is formatted as a symbol "#" suffixed with a variable number.

Variable types

Type	No.	Usage	Read	Write
Local	#1~#50	Local variables are used by sub-routines or macros for data registry or calculation. They can be referred by arguments. The calling subroutines can nest up to 8 layers with calling arguments at each layer referred to some local variables.	★	★
Global	#51~#250	Global variables are used by subroutines or macros for data registry or calculation.	★	★
Maintain	#1601~#1800	Systems variables are used for reading and writing internal data of the system during an NC operation. This is the power outage maintain variable.	★	★
MLC bit output	#1801~#1832	Read MLC message status (MLC->NC) with variable number (Variable #1801~#1832 for bit and #1833~#1848 for word).	★	
MLC word output	#1833~#1848		★	
MLC bit input	#1864~#1895	Write MLC message status (NC->MLC) with variable number (Variable #1864~#1895 for bit and #1896~#1911 for word).		★
MLC word input	#1896~#1911			★

4.1.1 Arguments and local variables

Except for the G, L, N, O and P codes, all the others can be used as designated arguments. When called by subroutine G65 and G66, they can be sent as local variables.

#1	#2	#3	#4	#5	#6	#7	#8	#9	#10
A	B	C	D	E	F		H	I	J
#11	#12	#13	#14	#15	#16	#17	#18	#19	#20
K		M				Q	R	S	T
#21	#22	#23	#24	#25	#26				
U	V	W	X	Y	Z				

4.1.2   Systems variables

Systems variables are used for reading and writing internal systems data during an NC operation. These MLC outputs are used for data exchanges between the NC program and MLC where "Special M" stands for bit and "Special D" for word.

(1) G code group message:

Variable No.	Function	Read	Write
#2000~2019	G code group	★	
#2020	F code, the F value of NC machining speed.	★	
#2021	H code, the H value of NC tool length compensation number.	★	
#2022	D code, the D value of NC tool diameter compensation number.	★	
#2023	T code, the T value of NC tool number.	★	
#2024	S code, the S value of NC spindle rotation speed.	★	

Various mode messages during program execution (for read only)

Variable	Function
#2000	G04, G09, G10, G11
#2001	Interpolation mode: G00, 01, 02, 03
#2002	Plane selection: G17, 18, 19
#2003	Absolute/incremental: G90, 91
#2004	Stroke check: G22, 23
#2005	Feed designation: G94
#2006	Imperial/metric: G21
#2007	Tool diameter compensation: G40, 41, 42
#2008	Tool length compensation: G43, 44, 49
#2009	Fixed cycle: G73, 74, 76, 80, 81, 82, 83, 84, 85, 86, 87, 88, 89
#2010	Homing position: G98, 99
#2011	Scaled cutting: G50, 51
#2012	Workpiece coordinates: G54, 55, 56, 57, 58, 59
#2013	Cutting mode: G61, 64
#2014	Macro calling: G66, 67
#2015	Coordinates rotation: G68, 69
#2016	Polar coordinates: G15, 16

(2) Position relevant messages:

Variables #2100 ~ #2214 can read coordinate values shown in the table below (read only).

Position message / Spindle	Mechanical coordinates / Absolute coordinates	Relative coordinates / Coordinates of end of single block	Jumping mechanical coordinates / Jumping absolute coordinates	Deviation between restart position and absolute coordinates / Deviation between restart position and tool
X	#2100	#2180	#2148	#2196
	#2116	#2132	#2164	#2212
Y	#2101	#2181	#2149	#2197
	#2117	#2133	#2165	#2213
Z	#2102	#2182	#2150	#2198
	#2118	#2134	#2166	#2214
A	#2103	#2183	#2151	#2199
	#2119	#2135	#2167	#2215
B	#2104	#2184	#2152	#2200
	#2120	#2136	#2168	#2216
C	#2105	#2185	#2153	#2201
	#2121	#2137	#2169	#2217

(3) Job coordinates messages:

Variables #3000 ~ #3262 can read offset and job coordinates (read only).

Position message / Spindle	Offset coordinates	Workpiece coordinates G54 / Workpiece coordinates G57	Workpiece coordinates G55 / Workpiece coordinates G58	Workpiece coordinates G56 / Workpiece coordinates G59
X	#3000	#3001	#3002	#3003
		#3004	#3005	#3006
Y	#3128	#3129	#3130	#3131
		#3132	#3133	#3134
Z	#3256	#3257	#3258	#3259
		#3260	#3261	#3262
A	#3384	#3385	#3386	#3387
		#3388	#3389	#3390
B	#3512	#3513	#3514	#3515
		#3516	#3517	#3518
C	#3640	#3641	#3642	#3643
		#3644	#3645	#3646

(4) Other messages:

Variable No.	Function	Read	Write
#2300	Single block **I**, for arc command	★	
#2301	Single block **J**, for arc command	★	
#2302	Single block **K**, for arc command	★	
#2303	Timing after system power on	★	
#2304	Tool number at spindle 1 (dual tool magazine)	★	
#2305	Tool number at spindle 2 (dual tool magazine)	★	
#2500	Tool number fetched from tool magazine 1		★
#2501	Tool number fetched from tool magazine 2		★
#5000 ~ #5013	Log of latest used **M** codes (14 sets)	★	
#5014 ~ #5015	Log of latest used **T** codes (2 sets)	★	
#5016	Search the latest **S** code again	★	
#6000	System macro warnings in range of 1 ~ 1000. Message can be edited by the Screen Editor.		★
#6001~	Tool length	★	★
#6501~	Tool radius	★	★
#7001~	Wear in length	★	★
#7501~	Wear in radius	★	★
#8001~	Tool life span	★	★
#8600~	Program timing in unit of ms	★	★

4

4.1.3   Macro interface input and output

You can use variables #1801 ~ #1911 to get interface messages and read or write the MLC message status. Variable values can be in Bit or Word format. For Bit message types, the variable value can be digit 0 or 1.It can be any numeric for the Word type.

MLC Bit output, read MLC message status (MLC > NC)			
Read MLC message status	Macro output point	Read MLC message status	Macro output point
#1801	M1024	#1817	M1040
#1802	M1025	#1818	M1041
#1803	M1026	#1819	M1042
#1804	M1027	#1820	M1043
#1805	M1028	#1821	M1044
#1806	M1029	#1822	M1045
#1807	M1030	#1823	M1046
#1808	M1031	#1824	M1047
#1809	M1032	#1825	M1048
#1810	M1033	#1826	M1049
#1811	M1034	#1827	M1050
#1812	M1035	#1828	M1051
#1813	M1036	#1829	M1052
#1814	M1037	#1830	M1053
#1815	M1038	#1831	M1054
#1816	M1039	#1832	M1055

MLC Word output, read MLC message status (MLC > NC)			
Read MLC message status	Macro output register	Read MLC message status	Macro output register
#1833	D1024	#1841	D1032
#1834	D1025	#1842	D1033
#1835	D1026	#1843	D1034
#1836	D1027	#1844	D1035
#1837	D1028	#1845	D1036
#1838	D1029	#1846	D1037
#1839	D1030	#1847	D1038
#1840	D1031	#1848	D1039

MLC Bit input, write MLC message status (NC > MLC)			
Write MLC message status	Macro input point	Write MLC message status	Macro input point
#1864	M2080	#1880	M2096
#1865	M2081	#1881	M2097
#1866	M2082	#1882	M2098
#1867	M2083	#1883	M2099
#1868	M2084	#1884	M2100
#1869	M2085	#1885	M2101
#1870	M2086	#1886	M2102
#1871	M2087	#1887	M2103
#1872	M2088	#1888	M2104
#1873	M2089	#1889	M2105
#1874	M2090	#1890	M2106
#1875	M2091	#1891	M2107
#1876	M2092	#1892	M2108
#1877	M2093	#1893	M2109
#1878	M2094	#1894	M2110
#1879	M2095	#1895	M2111

MLC Word input, write MLC message status (NC > MLC)			
Write MLC message status	Macro input register	Write MLC message status	Macro input register
#1896	D1336	#1904	D1344
#1897	D1337	#1905	D1345
#1898	D1338	#1906	D1346
#1899	D1339	#1907	D1347
#1900	D1340	#1908	D1348
#1901	D1341	#1909	D1349
#1902	D1342	#1910	D1350
#1903	D1343	#1911	D1351

4.2 Variable syntax

Numeric values required by a program can be assigned by using variables. This empowers the program for better flexibility and universality as you can use the variables for mathematical operations.

(1)   You can specify the scope of local variables:

i: the ith variable (for 1<= i <=50)

(2)   You can define the variable number with expressions:

For variable # [A], the value of A must be in range between A and the upper limit of the system variable number, that is 1 <= A <= the upper limit of the systems variable number. Value of A shall not be smaller than 0 or be a negative number.

#[<expression>]	description
# [#20]	(Valid)
#[#20Δ3]	(Valid) where Δ represents operators =, +, -, *, and /
##20	(Invalid), there cannot be two variable symbols (#) in a sequence.
#[#20] = …	(Valid)
#20 = …	(Valid)
#[#20 - #10] = …	(Valid), no operator can precede an equal (=) symbol
#[- #20]= …	(Valid)

(3)   Symbols precedes a variable

-#<variable number>	Pre-conditions
Z-#20 is the same as Z-10.1	#20=10.1
#20 is the same as G1	#20=1

(4)   Define with variables

#20 = 10

#20 = #5

#20 = #5 + #2

(5)   Conditional expressions

IF [#20==1]

4.3 Calculation commands

You can subject variables to a variety of calculations and use the result as a value for another variable, as a combination of several variables or as an alternative to other variables.

#i, #j, and #k can be replaced by constants.

Command	Symbol	Usage	Definition
Elementary arithmetic	+	#i = #j + #k	Addition
	-	#i = #j - #k	Subtraction
	*	#i = #j * #k	Multiplication
	/	#i = #j / #k	Division
	=	#i = #j	Substitution
	[ ]	#i = #j * [#p + #q]	Parenthesis
Function	SIN	#i = SIN [#k]	Sine
	ASIN	#i = ASIN [#k]	Arcsine
	COS	#i = COS [#k]	Cosine
	ACOS	#i = ACOS [#k]	Arccosine
	TAN	#i = TAN [#k]	Tangent
	ATAN	#i = ATAN [#k]	Arctangent
	ATAN2	#i = ATAN2 [#m, #n]	Arctangent; the angle is adjacent side #m and diagonal side #n
	ABS	#i = ABS [#k]	Absolute value
	FIX	#i = FIX [#k]	Round down
	FUP	#i = FUP [#k]	Round up
	ROUND	#i = ROUND[#k]	Round off
	SQRT	#i = SQRT [#k]	Squared value
	POW	#i = POW [#m, #n]	#m to the power of #n
	BIT	#i = BIT [#m, #n]	The value of the #n bit of a binary #m
Logic Operation	&	#i = #j & #k	AND logic
			#i = #j	#k	OR logic
	∧	#I = #j^#k	XOR logic
	!	#i = ! #j	NOT logic
Constants	PI	PI = π	Pi
	TRUE	TRUE = 1	Return value 1 when the IF statement is true
	FALSE	FALSE=0	Return value 0 when the IF statement is true

4

4.4 Flow commands

Change the program flow with WHILE ~ ENDW and IF ~ GOTO commands.

4.4.1 Repetitive execution

When the WHILE expression is true: The program loops from the statement under WHILE to the statement above ENDW. Otherwise, the program jumps to the statement under ENDW for execution.

[WHILE]

⟩

ENDW

Illustration:

WHILE [#80<=360.]   (Enters the cycle for repetitive execution when #80 is less than or

equal to 360.)

WHILE [#60>=20.]   (Enters the second cycle for execution when #60 is greater than or

equal to 20.)

#60=#60-2.

ENDW   (The second cycle ends when ENDW is encountered.)

#80=#80+15.

#50=#50-0.05

ENDW   (The first cycle ends when the second ENDW is encountered.)

4.4.2   Branch conditions

IF [discriminant] GOTON (N is the sequence number in the procedure)
When the IF statement is true, the program branches (GOTO) to statement number N for
execution. Otherwise, the program executes the next block again.

GOTON (It jumps to N un-conditionally when being used independently.)
N from GOTON has to be existed in the same procedure. If not, alarm will occur (B61C).

∫

N10 #12=#10 #13=#11+2;

IF〔#2=1〕GOTO200;

#12=#10-#3;

When #2 = 1,
branch to N200.

#13=#11-#4;

N200 X#12 Z#13;

#5=#5+2;

∫

Note 1: When searching the sequence number in branch, it starts from the beginning. If not, an alarm
(B61C) occurs. If there are more than one sequence number, it executes from the first searched one.

Types of conditional expressions:

Conditional expressions	Description	Examples		
#j > #k	#j greater than #k	#i = #j > #k	TRUE	returns value: #i=1
			FALSE	returns value: #i=0
#j < #k	#j less then #k	#i = #j < #k	TRUE	returns value: #i=1
			FALSE	returns value: #i=0
#j == #k	#j equal to #k	#i = #j == #k	TRUE	returns value: #i=1
			FALSE	returns value: #i=0
#j >= #k	#j greater than or equal to #k	#i = #j >= #k	TRUE	returns value: #i=1
			FALSE	returns value: #i=0
#j <= #k	#j less than or equal to #k	#i = #j <= #k	TRUE	returns value: #i=1
			FALSE	returns value: #i=0
#j != #k	#j not equal to #k	#i = #j != #k	TRUE	returns value: #i=1
			FALSE	returns value: #i=0

4

Illustration:

#100 = 1.234;   (Assign value 1.234 to #100)

#100 = #101;   (Assign value of #100 to #101)

#100 = [[#101+#102]/2.0];   (Assign value of #100 to be the average of #101 and #102)

#100 = #102+2;   (Assign value of #100 to be the sum of #102 and 2)

#100 = SIN[#102];

X-#100   (Assign the negative value of #100 to X coordinate)

G1X#100Y#101;   (Assign the value of #100 to X coordinate and #101 to Y

                                       coordinate)

G1X[#100];   (Assign the value of #100 to X coordinate)

G1X[#100+#101];   (Assign the sum of values of #100 and #101 to X coordinate)

G2X [#100*SIN [#102]];   (Assign the multiplication of value of #100 by SIN#102 to X

                                       coordinate)

G1Z#100F#102S#103;   (Assign the value of #100 to Z coordinate, #102 to F, and

                                       #103 to S)

4.5   MACRO calling

(1)   A Macro calls G code

Macro function	G code number	Remarks
O9010	0 ~ 1000	
O9011	0 ~ 1000	
O9012	0 ~ 1000	
O9013	0 ~ 1000	
O9014	0 ~ 1000	
O9015	0 ~ 1000	Set G to 0 when not called by a macro
O9016	0 ~ 1000	
O9017	0 ~ 1000	
O9018	0 ~ 1000	
O9019	0 ~ 1000	

Restrictions: In a macro program that uses a macro to call the G, M or T code, the parameter G code is set as a general one and cannot call another macro.

(2)   A Macro calls M code

Macro function	M code number	Remark
O9020	0 ~ 1000	
O9021	0 ~ 1000	
O9022	0 ~ 1000	
O9023	0 ~ 1000	
O9024	0 ~ 1000	
O9025	0 ~ 1000	Set M to 0 when not called by a macro
O9026	0 ~ 1000	
O9027	0 ~ 1000	
O9028	0 ~ 1000	
O9029	0 ~ 1000	

Restrictions: In a macro program that uses a macro to call a G, M or T code, the parameter M code is set as a general one and cannot call another macro.

(3)   A Macro calls T code

Macro function	T code number	Remarks
O9000	0: disabled	Set T to 0 when not called by a macro.
	else: enabled	The T code number is set to local variable ＃20

Restrictions: In a macro program that uses a macro to call G, M or T code, the parameter T code is set as a general one and cannot call another macro.

4

List of NC variables

Number	Description	Read	write
#1 ~ #50	Local variable	★	★
#51 ~ #250	Global variable	★	★
#1601 ~ #1800	System variable		
#1801 ~ #1832	MLC logic output: MLC > NC at macro input point M1024_32	★	
#1833 ~ #1848	MLC data output: MLC > NC at macro input point D1024_16	★	
#1864 ~ #1895	MLC logic input: NC > MLC at macro output point M2080_32		★
#1896 ~ #1911	MLC data input: NC > MLC at macro output point D1336_16		★
#2000 ~ 2019	G code group	★	
#2020	F code	★	
#2021	H code	★	
#2022	D code	★	
#2023	T code	★	
#2024	S code	★	
#2100 ~ 2102	X ~ Z spindle mechanical coordinates	★	
#2116 ~ 2118	X ~ Z spindle absolute coordinates	★	
#2132 ~ 2134	X ~ Z spindle single block ending point coordinates	★	
#2148 ~ 2150	X ~ Z spindle jumping mechanical coordinates	★	
#2164 ~ 2166	X ~ Z spindle jumping absolute coordinates	★	
#2180 ~ 2182	X ~ Z spindle relative coordinates	★	
#2196 ~ 2198	X ~ Z spindle absolute coordinates for restart	★	
#2212 ~ 2214	X ~ Z spindle offset coordinates for restart	★	
#2300	Single block I	★	
#2301	Single block J	★	
#2302	Single block K	★	
#2303	Spindle tool number	★	
#2304	Timing after system power on	★	
#3000	X spindle offset coordinates	★	
#3001~	X spindle workpiece coordinates G54 ~ G59	★	
#3128	Y spindle offset coordinates	★	
#3129~	Y spindle workpiece coordinates G54 ~ G59	★	
#3256	Z spindle offset coordinates	★	

(Continued)

Number	Description	Read	write
#3257~	Z spindle workpiece coordinates G54 ~ G59	★	
#5000 ~ #5013	Latest used M codes (14 sets)	★	
#5014 ~ #5015	Latest used T codes (2 sets)	★	
#5016	Search the latest S code again	★	
#6000	System macro alarm		★
#6001~	Tool length	★	★
#6501~	Tool radius	★	★
#7001~	Wear in length	★	★
#7501~	Wear in radius	★	★
#8001~	Tool life span	★	★

(This page is intentionally left blank.)

4