DELTA

Smarter. Greener. Together.

Industrial Automation Headquarters
Delta Electronics, Inc.
Taoyuan Technology Center
No.18, Xinglong Rd., Taoyuan District,
Taoyuan City 33068, Taiwan
TEL: 886-3-362-6301 / FAX: 886-3-371-6301

Asia

Delta Electronics (Shanghai) Co., Ltd.
No.182 Minyu Rd., Pudong Shanghai, P.R.C.
Post code : 201209
TEL: 86-21-6872-3988 / FAX: 86-21-6872-3996
Customer Service: 400-820-9595

Delta Electronics (Japan), Inc.
Tokyo Office
Industrial Automation Sales Department
2-1-14 Shibadaimon, Minato-ku
Tokyo, Japan 105-0012
TEL: 81-3-5733-1155 / FAX: 81-3-5733-1255

Delta Electronics (Korea), Inc.
Seoul Office
1511, 219, Gasan Digital 1-Ro., Geumcheon-gu,
Seoul, 08501 South Korea
TEL: 82-2-515-5305 / FAX: 82-2-515-5302

Delta Energy Systems (Singapore) Pte Ltd.
4 Kaki Bukit Avenue 1, #05-04, Singapore 417939
TEL: 65-6747-5155 / FAX: 65-6744-9228

Delta Electronics (India) Pvt. Ltd.
Plot No.43, Sector 35, HSIIDC Gurgaon,
PIN 122001, Haryana, India
TEL: 91-124-4874900 / FAX : 91-124-4874945

Delta Electronics (Thailand) PCL.
909 Soi 9, Moo 4, Bangpoo Industrial Estate (E.P.Z),
Pattana 1 Rd., T.Phraksa, A.Muang,
Samutprakarn 10280, Thailand
TEL: 66-2709-2800 / FAX : 662-709-2827

Delta Energy Systems (Australia) Pty Ltd.
Unit 20-21/45 Normanby Rd., Notting Hill Vic 3168, Australia
TEL: 61-3-9543-3720

Americas

Delta Electronics (Americas) Ltd.
Raleigh Office
P.O. Box 12173, 5101 Davis Drive,
Research Triangle Park, NC 27709, U.S.A.
TEL: 1-919-767-3813 / FAX: 1-919-767-3969

Delta Greentech (Brasil) S/A
São Paulo Office
Rua Itapeva, 26 – 3° Andar - Bela Vista
CEP: 01332-000 – São Paulo – SP - Brasil
TEL: 55-11-3530-8642 / 55-11-3530-8640

Delta Electronics International Mexico S.A. de C.V.
Mexico Office
Vía Dr. Gustavo Baz No. 2160, Colonia La Loma,
54060 Tlalnepantla Estado de Mexico
TEL: 52-55-2628-3015 #3050/3052

EMEA

Headquarters: Delta Electronics (Netherlands) B.V.
Sales: Sales.IA.EMEA@deltaww.com
Marketing: Maketing.IA.EMEA@deltaww.com
Technical Support: iatechnicalsupport@deltaww.com
Customer Support: Customer-Support@deltaww.com
Service: Service.IA.emea@deltaww.com
TEL: +31(0)40 800 3800

BENELUX: Delta Electronics (Netherlands) B.V.
De Witbogt 20, 5652 AG Eindhoven, The Netherlands
Mail: Sales.IA.Benelux@deltaww.com
TEL: +31(0)40 800 3800

DACH: Delta Electronics (Netherlands) B.V.
Coesterweg 45, D-59494 Soest, Germany
Mail: Sales.IA.DACH@deltaww.com
TEL: +49(0)2921 987 0

France: Delta Electronics (France) S.A.
ZI du bois Challand 2, 15 rue des Pyrénées,
Lisses, 91090 Evry Cedex, France
Mail: Sales.IA.FR@deltaww.com
TEL: +33(0)1 69 77 82 60

Iberia: Delta Electronics Solutions (Spain) S.L.U
Ctra. De Villaverde a Vallecas, 265 1° Dcha Ed.
Hormigueras – P.I. de Vallecas 28031 Madrid
TEL: +34(0)91 223 74 20

C/Llull, 321-329 (Edifici CINC) | 22@Barcelona, 08019 Barcelona
Mail: Sales.IA.Iberia@deltaww.com
TEL: +34 93 303 00 60

Italy: Delta Electronics (Italy) S.r.l.
Ufficio di Milano Via Senigallia 18/2 20161 Milano (MI)
Piazza Grazioli 18 00186 Roma Italy
Mail: Sales.IA.Italy@deltaww.com
TEL: +39 02 64672538

Russia: Delta Energy System LLC
Vereyskaya Plaza II, office 112 Vereyskaya str.
17 121357 Moscow Russia
Mail: Sales.IA.RU@deltaww.com
TEL: +7 495 644 3240

Turkey: Delta Greentech Elektronik San. Ltd. Sti. (Turkey)
Şerifali Mah. Hendem Cad. Kule Sok. No:16-A
34775 Ümraniye – İstanbul
Mail: Sales.IA.Turkey@deltaww.com
TEL: + 90 216 499 9910

GCC: Delta Energy Systems AG (Dubai BR)
P.O. Box 185668, Gate 7, 3rd Floor, Hamarain Centre
Dubai, United Arab Emirates
Mail: Sales.IA.MEA@deltaww.com
TEL: +971(0)4 2690148

Egypt + North Africa: Delta Electronics
511 Cairo Business Plaza, North 90 street,
New Cairo, Cairo, Egypt
Mail: Sales.IA.MEA@deltaww.com

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

Delta CNC Milling Machine Solution G Command Guidelines

Delta CNC Milling Machine Solution G Command Guidelines

www.deltaww.com

DELTA
Smarter. Greener. Together.

Preface

Thank you for purchasing this product. Before using this product, please read through this manual carefully to ensure the correct use of the product. Please keep this manual handy for quick reference whenever needed.

This manual includes:

- G-code and command format of NC controllers
- M-code description of NC controllers
- Macros and variables of NC controllers

NC controller product features:

- Built-in 32-bit high-speed dual CPU for multi-task execution and performance improvement
- User-friendly HMI interface
- Servo Gain Auto-tuning Interface for different machine characteristics
- CNCSoft software tools to facilitate the development of customized screen images
- Front USB interface (port) to facilitate data access, data backup, and parameters copy
- Different spindle control forms for you to choose from: communication type or analog voltage type
- Serial I/O modules for flexible I/O configuration

How to use this manual:

You can use this manual as a reference for writing G-codes and using macro and variable syntax. Please read through this manual before using and setting your NC controller.

Delta technical services:

Please consult your Delta equipment distributor or Delta Customer Service Center if you counter any problems.

Safety Precautions:

- Please refer to the pin assignment when connecting and wires and ensure this product is correctly grounded.
- To avoid electric shock, do not disassemble the controller, change the wiring, or touch the power source when power is on.

Pay attention to the following safety precautions at all times during installation, wiring, operation, maintenance, and examination of the controller.

The symbols of "**DANGER**", "**WARNING**", and "**STOP**" indicate:

Danger. May cause severe or fatal injuries to personnel if the instructions are not followed.

Warning. May cause moderate injury to personnel, or lead to several damage or even malfunction of the product if the instructions are not followed.

Absolutely prohibited activities. May cause serious damage or even malfunction of the product if the instructions are not followed.

Installation

- Comply with the methods specified in this user manual for installation, or it may cause damage to the device.
- Do not expose this product to an environment containing vapor, corrosive gas, inflammable gas, or other foreign matter, or it may result in electric shock or fire.

Wiring

- Connect the ground terminal to the class-3 ground system. Ground resistance should not exceed 100Ω. Improper grounding may result in electric shock or fire.

Operation

- Correctly plan out the I/O actions with MLC Editor, or it may cause abnormal operation.
- Properly adjust the parameter settings of the machine before operation, or it may result in abnormal operation or malfunction.
- Ensure the emergency stop can be activated at any time to avoid operating the machine in an unprotected condition.

- Do not change the wiring when power is on, or it may result in personnel injury caused by electric shock.
- Never use a sharp object to touch the panel, or it may make a dent in the panel and thus cause abnormal operation of the controller.

Maintenance and Inspection

- Do not disassemble the panel or touch the internal parts of the controller when power is on, or it may result in electric shock.
- Do not touch the wiring terminals within 10 minutes after turning off the power, or the residual voltage may result in electric shock.
- Turn off the power before replacing the backup battery, and recheck the system settings afterwards.
- Do not obstruct the ventilation holes when operating the controller, or poor heat dissipation may cause malfunction.

Wiring

- Power supply: to avoid danger, use a 24 V_{DC} power supply for the controller and comply with the wire specification when wiring.
- Wire selection: use stranded wires and multi-core shielded-pair wires for all signal cables.
- Cable length: the maximum cable length for remote I/O and DMCNET is 20 m (65.62 ft); for other signal cables, 10 m (32.81 ft).
- To control the input and output signals, an additional 24 V_{DC} power supply is required for the controller I/O and remote I/O.

Wiring of Communication Circuit

- DMCNET: the wiring materials should comply with the standard specification.
- Ensure the wiring between the controller and servo drive is secure, or it may cause abnormal operation.

Note: the content of this manual may be revised without prior notice. Download the latest version at Delta's website (http://www.delta.com.tw/industrialautomation/).

(This page is intentionally left blank.)

Table of Contents

3 M-code Description

4 Macro and Variable

G-code list

1

This chapter provides the G-code list for you to quickly view all of the G-codes.

1.1   G-code list for milling system

G-code	Group	Function description
G00	01	Rapid positioning
G01	01	Linear interpolation
G02	01	Clockwise (CW) circular interpolation
G03	01	Counterclockwise (CCW) circular interpolation
G04	00	Dwell time
G05	00	Parameter group change
G09	00	Exact stop
G10	00	Data setting
G11	00	Data cancellation
G15	16	Cancel polar coordinate
G16	16	Polar coordinate
G17	02	X-Y plane selection
G18	02	Z-X plane selection
G19	02	Y-Z plane selection
G20	06	Imperial input
G21	06	Metric input
G24	17	Mirroring
G25	17	Cancel mirroring
G28	00	Return through reference point
G29	00	Return from reference point
G30	00	Return to the 2nd, 3rd, or 4th reference point
G31	00	Skip command
G40	07	Cancel tool radius compensation
G41	07	Tool radius compensation left
G42	07	Tool radius compensation right
G43	08	Tool length compensation in positive direction
G44	08	Tool length compensation in negative direction
G49	08	Cancel tool length compensation
G50	11	Cancel scaling
G51	11	Scaling
G52	00	Local coordinate system setting
G53	00	Machine coordinate system setting
G54	12	1st workpiece coordinate system selection

1

G-code	Group	Function description
G55	12	2nd workpiece coordinate system selection
G56	12	3rd workpiece coordinate system selection
G57	12	4th workpiece coordinate system selection
G58	12	5th workpiece coordinate system selection
G59	12	6th workpiece coordinate system selection
G61	13	Exact stop mode
G64	13	Cutting mode
G65	00	Macro call (one-shot)
G66	14	Continuous macro call
G67	14	Cancel continuous macro call
G68	15	Coordinate system rotation
G69	15	Cancel coordinate system rotation
G73	09	Peck drilling cycle
G74	09	Reverse tapping cycle
G76	09	Fine boring cycle
G80	09	Cancel cycle
G81	09	Drilling cycle
G82	09	Drilling cycle (counter boring)
G83	09	Deep-hole peck drilling cycle
G84	09	Tapping cycle
G85	09	Boring cycle (no dwell time)
G86	09	Boring cycle (spindle stop)
G87	09	Back boring cycle
G88	09	Boring cycle (manual retraction)
G89	09	Boring cycle (with dwell time)
G90	03	Absolute coordinates
G91	03	Incremental coordinates
G92	00	Coordinate system setting
G94	05	Feed per minute (mm/min)
G98	10	Return to start point after fixed cycle
G99	10	Return to reference point (R) after fixed cycle

1

(This page is intentionally left blank.)

G-code Description

2

This chapter introduces the G-code formats supported by the NC series controllers along with the application examples. You can learn more about G-codes in this chapter.

2

2.1 G-codes for milling system

G00: Rapid positioning

Format: G00 X_ Y_ Z_ (applicable to single-axis, double-axis, and triple-axis synchronous motion
controls)

X_ Y_ Z_: end coordinates.

Description: G00 can move the tool center to the specified X-, Y-, and Z-coordinates.

When using G00, you can adjust the moving speed with the **Rapid %** key on the
secondary control panel instead of parameter F.

Assume that the maximum speed of the X, Y, and Z axes is 15 m/min:

1. When the rapid feed rate is 100%, the axes operate at the maximum speed of
15 m/min.

2. When the rapid feed rate is 50%, the axes operate at a speed of 7.5 m/min.

3. When the rapid feed rate is 25%, the axes operate at a speed of 3.75 m/min.

4. When the rapid feed rate is 0%, the axial moving speed is determined by the
speed set in the parameter.

G00 is mainly used for rapid positioning instead of feed cutting. It is for applications
such as the tool moving from the machine origin to the cutting start point in rapid
traverse, or the tool retraction of Z axis and positioning of X and Y axes after cutting.

[Example]

The following diagram illustrates the usage of G00. The tool moves from point A to point B in
rapid traverse.

Express with absolute values: G90 G00 X92. Y35.

Express with incremental values: G91 G00 X62. Y-25.
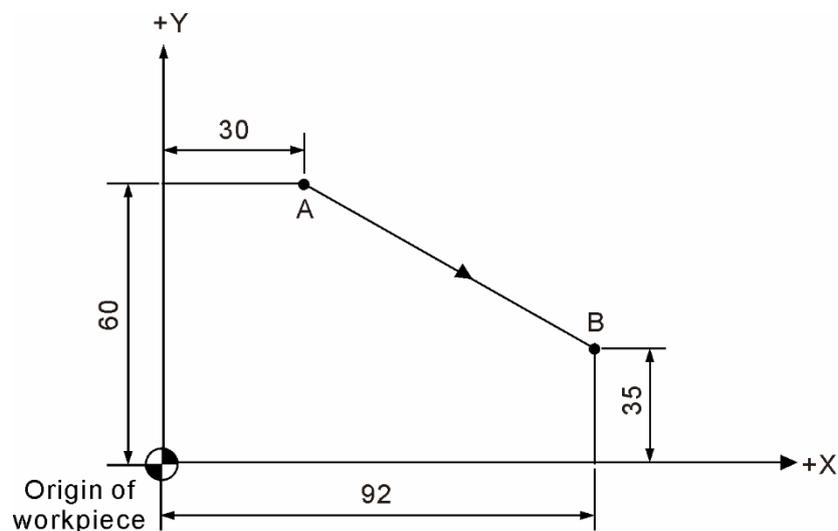
2

G01: Linear interpolation

Format: G01 X_ Y_ Z_ F_

   X_ Y_ Z_: end coordinates.

   F_: cutting feed rate in the unit of mm/min.

Description: G01 enables the tool to perform linear interpolation from the current position to the next command position at feed rate F. X-, Y-, and Z-coordinates represent the cutting end point. This command is applicable to single-axis, double-axis, or triple-axis synchronous motion controls. The feed rate is set by parameter F as well as **Rapid %** on the secondary control panel in the unit of mm/min.

[Example]

The following diagram illustrates the usage of G01. G01 enables the tool to cut towards the +Y direction from the program start point.
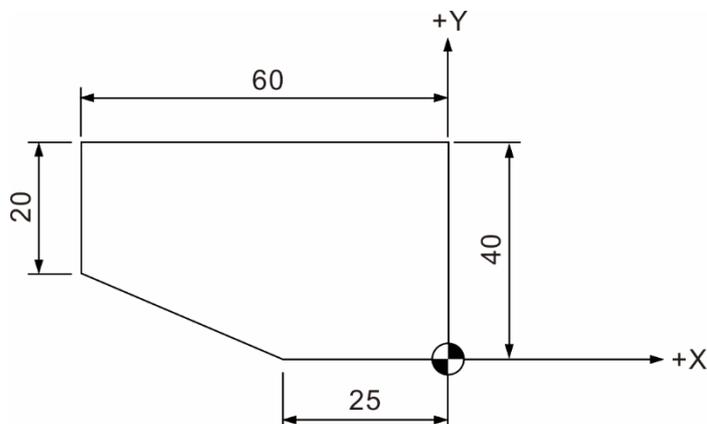

```
G00 G90 G54 X0 Y0
G90 G01 Y40.0 F80
X-60.0
G91 Y-20.0
X35.0 Y-20.0
G90 X0 Y0
```

Parameter F is continuously effective, so you do not need to set it again if the cutting speed is the same, as shown in the above program.

G01: Automatic chamfer and corner rounding

Format: G01 X_/Y_/Z_,C_ F_ (for a 45-degree angle, C)

         G01 X_/Y_/Z_,R_ F_ (for a round corner, R)

         X_Y_Z_: end coordinates.

         C_: distance from the point of the hypothetical intersection to the start or end point of the chamfer.

         R_: radius for the round corner.

         F_: cutting feed rate in the unit of mm/min.

Description: you can use this G01 command to simplify the program for chamfering or corner rounding if a 45-degree angle or round corner is required for a workpiece.

[Example]

The following diagram illustrates the G01 command for chamfering and corner rounding.
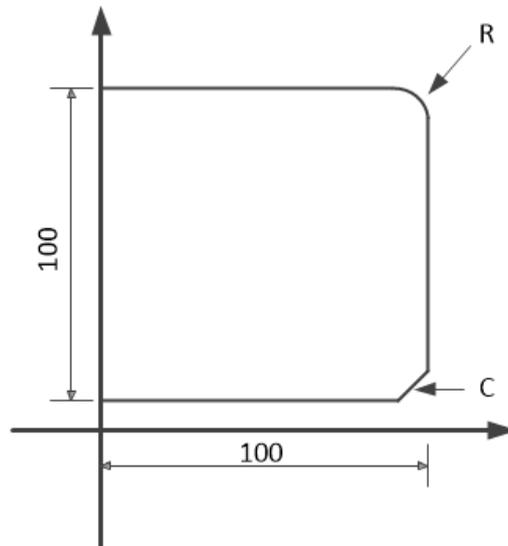
G02/G03: Circular interpolation

Format: Arcs in the X-Y plane:

        G17 G02 (G03) X_ Y_ R_ F_ or

        G17 G02 (G03) X_ Y_ I_ J_ F_

        Input parameter Z_ to generate a helical path in the X-Y plane.

        Arcs in the Z-X plane:

        G18 G02 (G03) Z_ X_ R_ F_ or

        G18 G02 (G03) Z_ X_ K_ I_ F_

        Input parameter Y_ to generate a helical path in the Z-X plane.

        Arcs in the Y-Z plane:

        G19 G02 (G03) Y_ Z_ R_ F_ or

        G19 G02 (G03) Y_ Z_ J_ K_ F_

        Input parameter X_ to generate a helical path in the Y-Z plane.

        G02: clockwise (CW) circular interpolation.

        G03: counterclockwise (CCW) circular interpolation.

        X_ Y_ Z_: end coordinates expressed with absolute (G90) or incremental values (G91).

        R_: arc radius (the format expressed with R is called radius format).

        I_: distance from the arc start point to the arc center point in X-axis direction.

        J_: distance from the arc start point to the arc center point in Y-axis direction.

        K_: distance from the arc start point to the arc center point in Z-axis direction.

          (The format expressed with I, J, and K is called center format.)

        F_: cutting feed rate in the unit of mm/min.

Description: G02 and G03 are circular interpolation commands. Since the workpiece is three-dimensional, the circular interpolation direction (G02 or G03) varies in different planes, as shown in the following diagrams. Definition: based on the right-handed coordinate system, G02 is for clockwise direction while G03 is for counterclockwise direction.
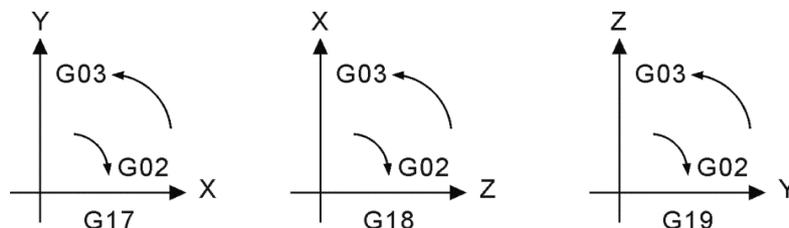
Statement expressions with center format and radius format are as follows:

1. Radius format: R is the arc radius. Specify the start point, end point, and arc radius to form an arc. There will be two arc segments as shown in the following figure. When R is a positive value, it means the central angle ≦ 180°. When R is a negative value, it means the central angle > 180°.
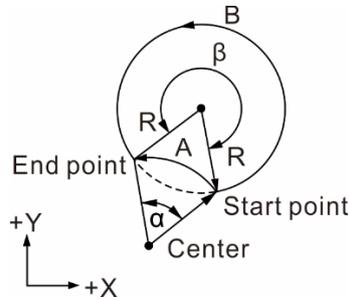
β > 180° means R is a negative value, so arc B is generated.

α ≦ 180° means R is a positive value, so arc A is generated.

In the above diagram, assume that R = 50 mm and the end coordinates in absolute values are (100.0, 80.0), then:

(1) Central angle > 180° (path B) G90 G03 X100.0 Y80.0 R-50.0 F80

(2) Central angel ≦ 180° (path A) G90 G03 X100.0 Y80.0 R50.0 F80

2. Center format: I, J, and K are the relative distances from the arc start point to the circle center, which are the incremental values from the start point to the center in the X-, Y-, and Z-axis directions, as shown in the following figure.
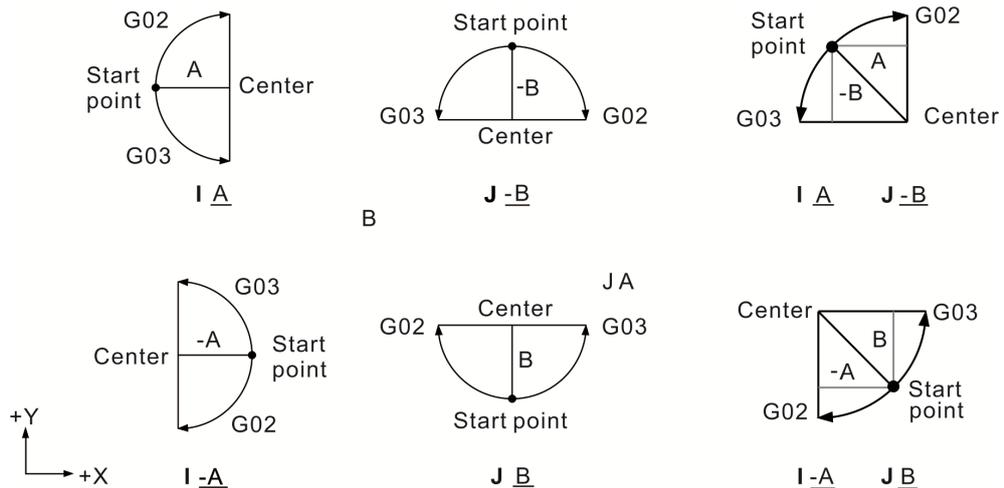
Using the command in radius or center format to perform the circular motion is necessary for making the programming easier whereas the command can only be in center format for milling a full circle. If you use the command in radius format to connect two semicircles, the roundness deviation will be too great.

2

[Example]

Refer to the following diagram. The command for milling a full circle is G02 I-50.0.
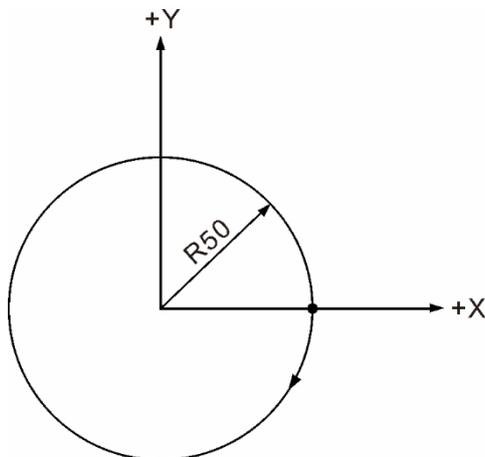
Milling a full circle

The following diagram illustrates the usage of G01, G02, and G03. Assume that the tool mills along the workpiece contour upwards from the program start point:
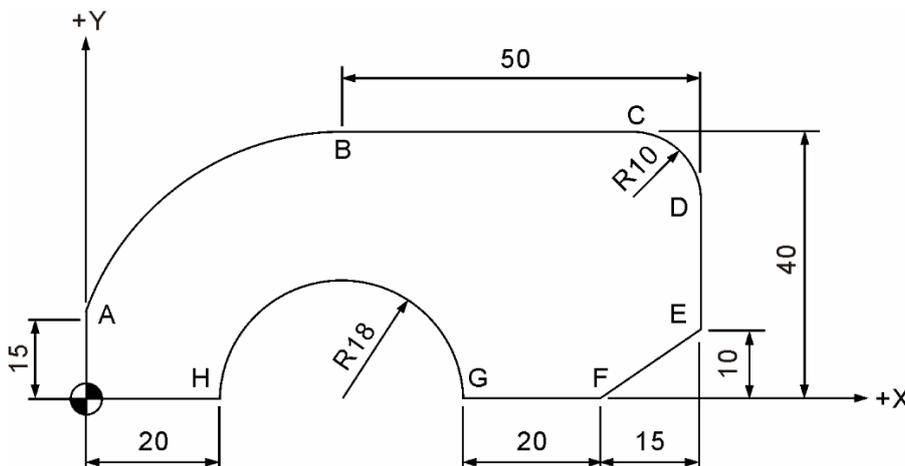

```
O0100

G90 G54 X0 Y0 S500 M3
G90 G01 Y15.0 F80              (from program start point to point A)
G02 X41.0 Y40.0 R41.0         (from point A to point B)
G91 G01 X40.0                 (from point B to point C)
G02 X10.0 Y-10.0 R10.0        (from point C to point D)
G01 Y-20.0                    (from point D to point E)
X-15.0 Y-10.0                 (from point E to point F)
X-20.                         (from point F to point G)
G90 G03 X20.0 R18.0           (from point G to point H)
G01 X0.0                      (from point H to program start point)
```

Instructions for G02 and G03 circular interpolation:

(1) After booting, the default machining plane is G17 (X-Y plane) in the milling system. Therefore, if the circular interpolation is in the X-Y plane, you can omit G17.

(2) If a block has parameters I, J, and R at the same time, only parameter R is valid.

(3) I0, J0, or K0 can be omitted.

(4) If the end X-, Y-, and Z-coordinates are not specified, it means the start and end points are the same, which cutting path will be a full circle. If the command is set in radius format, the tool does not move.

(5) An alarm message occurs when the system is unable to form an arc using the end coordinates with the specified radius.

(6) For a linear interpolation followed by a circular interpolation, you must use G02 or G03 to switch the motion. To switch to linear interpolation, you must use G01.

(7) If the circular interpolation command (G02 or G03) has no R, I, J, and K parameters specified, the motion path is the same as that of G01.

2

G04: Dwell time

Format: G04 X_ or

G04 P_

Description: this command specifies the dwell time of the current block. Parameter X sets the dwell time and this value can be a decimal. Parameter P also sets the dwell time but you can only input integers.

Setting range:

Setting range for dwell time (by parameter X)	
Setting range	0.001 - 99999.999
Unit	sec
Setting range for dwell time (by parameter P)	
Setting range	1 - 99999999
Unit	0.001 sec

[Example]

G04 X1.5

or

G04 P1500

When you use either of the above two formats, the execution result is the same: the dwell time during program execution is 1.5 seconds.

G05: Parameter group change

Format: G05 P_

    P_: parameter group number.

Description: use G05 to select the parameter group (20 groups available).

    G05 P_: set 1 - 20 to enable the group function; set G05P0 to cancel the parameter group function and restore to the default settings.

2

No.	Parameter Name	GROUP1	GROUP2	GROUP3	GROUP4
309	Nominal arc feed rate	200			
312	Cutting speed level	10			
314	G1 speed	500			
316	G00 Rapid speed	1000			
320	S curve time constant	30			

PARAMETER(PARAM GROUP) 01 N1 mm

Range: 10 ~ 50000 (mm/min)

JOG   RPD 100%   JOG 4000   S 100%   Ready

F1 SAVE | F2 DEL PAR | F3 DEL GRP | F4 WRT PAR | F5 RED PAR | F6 SRT PAR | F7 AVERAGE

You can individually set each parameter in a group. When you move the cursor to the field, the status bar below shows its setting range. Input the values in the entry row and press **Enter** to enter the values. Meanwhile, the system checks whether the range is valid; if the input value exceeds the range, the system prompts a warning window.

G09: Exact stop

Format: G09 G01 X_ Y_

Description: because the tool cuts at a constant feed rate, the cutting command execution of the next block will start before the current block execution completes. In this case, there will be a small arc generated at the corner between motion blocks. To eliminate this arc, you can use G09 to have the system confirm the tool position each time it executes a motion block. Once the tool is in the right position that is consistent with the command value, the system executes the next block. Therefore, there will be a minor discontinuity between blocks when G09 is used, which improves the precision at the cost of speed. This enables high-precision machining. This command is valid only for cutting commands (G01, G02, and G03) of a single block.
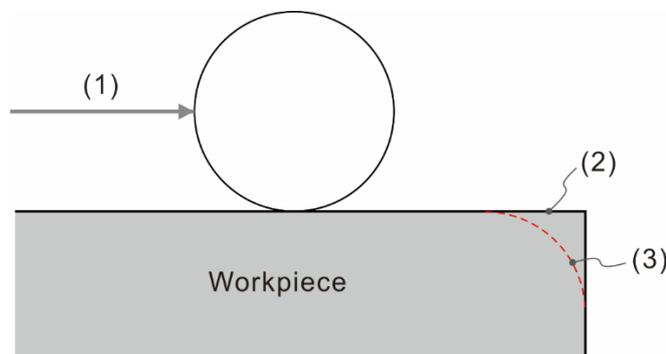
(1) Tool feeding direction; (2) With G09; (3) Without G09

[Example]

G09 G01 X100.0. F150 (The system has the tool decelerate to stop and then executes the next block after confirming the position.)
G01 Y-100.

2

G10/G11: Data setting / cancellation

Format: G10 L2 P_ X_ Y_ Z_; set the workpiece coordinate system.

G10 L10 P_ R_; set the tool length compensation.

G10 L11 P_ R_; set the tool length wear compensation.

G10 L12 P_ R_; set the tool radius compensation.

G10 L13 P_ R_; set the tool radius wear compensation.

G10 L20 P_ X_ Y_ Z_; set the extension workpiece coordinate system.

G10 L21 P_ X_ Y_ Z_; set the software limit coordinates.

G10 L30 P_; set the spindle positioning offset.

G10 L1100 R_ S_; set the maximum and current rotation speed.

G10 L2100 X_ Y_ Z_; set the machine coordinates.

G10 L3100 P_; trigger the status M-code without stopping the motion.

G10 L4100 P_; cancel the status M-code without stopping the motion.

Description:

(1) G10 L2 P_ X_ Y_ Z_ is for workpiece coordinate system data entry. When you input 0 for P_, it means you are setting the offset coordinates of the workpiece coordinate system; P1 - P6 correspond to G54 - G59 workpiece coordinate systems; X, Y, and Z specify the position of coordinate system origin.

(2) G10 L10 - L13 P_ R_ are for setting the compensation value. P_ is the compensation number and R_ is the actual compensation data. These commands correspond to the settings for tool radius and tool length compensation value.

(3) In G10 L20, you can input 1 - 64 for P_ to set the corresponding extension workpiece coordinate systems.

(4) For G10 L21 P_: input 1 for P_ to set the first set of the software positive limit; input 2 for P_ to set the first set of the software negative limit; input 3 for P_ to set the second set of the software positive limit; input 4 for P_ to set the second set of the software negative limit.

The G10 commands mentioned above can either be expressed with absolute values (G90) or with incremental values (G91).

(5) G10 L30 P_ is for setting the spindle positioning offset. P_ sets the offset angle in the unit of 0.01 degree; G11 L30 cancels the spindle positioning offset setting (resets to Pr405 settings).

(6) In G10 L1100 R_ S_, R_ sets the maximum rotation speed and S_ sets the current rotation speed. When you select DMCNET for the spindle, you can use this command to control the analog output.

2

(7) G10 L2100 X_ Y_ Z_ is for setting the machine coordinates with the data of X, Y, and Z entered to the machine coordinates individually.

(8) G10 L3100 P_ triggers the status M-code without stopping the motion; G10 L4100 P_ cancels the status M-code without stopping the motion; P2080 - P2111 (P_ value) correspond to M2080 - M2111.

Data entry type

L command format	Argument format	Data type description
L2	P_ X_ Y_ Z_	Data entry for the workpiece coordinate system P_: 0 is the offset coordinate; 1 - 6 correspond to G54 - G59 workpiece coordinates.
L10	P_ R_	Data entry for the tool length compensation P_: 1 - 100 correspond to 1 - 100 tool length compensation data.
L11	P_ R_	Data entry for the tool length wear compensation P_: 1 - 100 correspond to 1 - 100 tool length wear data.
L12	P_ R_	Data entry for the tool radius compensation P_: 1 - 100 correspond to 1 - 100 tool radius compensation data.
L13	P_ R_	Data entry for the tool radius wear compensation P_: 1 - 100 correspond to 1 - 100 tool radius wear data.
L20	P_ X_ Y_ Z_	Data entry for the extension workpiece coordinate system P_: 1 - 64 correspond to the 1st - 64th set of extension workpiece coordinates.

L21	P_ X_ Y_ Z_	Data entry for the software limit coordinates
		P_: 1 sets the first set of the software positive limit;
		     2 sets the first set of the software negative limit;
		     3 sets the second set of the software positive limit;
		     4 sets the second set of the software negative limit.

L30	P_	P_: spindle positioning offset amount (0.01 degree).
L1100	R_ S_	R_: maximum rotation speed (rpm). S_: current rotation speed (rpm). When you select DMCNET for the spindle, you can use this command to control the analog output.
L2100	X_ Y_ Z_	Data entry of X-, Y-, and Z- machine coordinates
L3100	P_	P_: 2080 - 2111 represent M2080 - M2111.
L4100	P_	P_: 2080 - 2111 represent M2080 - M2111.

2

[Example]

G10 L10 P1 R-300.0 or

G10 L10 P2 R-100.0

The two programs above set the tool length compensation value as -300.0 for tool No.1 and -100.0 for tool No.2 separately.

P: compensation number or workpiece coordinate number.

R: compensation value for the tool.

The following programs are expressed in absolute and incremental values separately:

G90 G10 L10 P1 R-300.0

In the program above, it inputs -300.0 to the data field for the length of tool No.1 in absolute format.

G91 G10 L10 P1 R-100.0

In the program above, it increments -100.0 to the current length value of tool No.1.

Note:

(1) G10 is a one-shot command which functions only within the block where it is specified. Compensation amounts of the offset coordinates and the workpiece coordinate systems both refer to the origin of the machine coordinate system. To cancel the data entry, you can use G11.

(2) When you use L2 or L20 to change the coordinates, it functions in the next motion block. When using L10 - L13 to change the tool compensation data, you must use the compensation command (G41, G42, G43, or G44) and specify the compensation data number (D or H number) again to update the compensation value.

G15: Cancel polar coordinate

Format: G15

Description: use G15 to cancel the polar coordinate command and then the program continues the motion path with Cartesian coordinate values.

2

G16: Polar coordinate

Format: G17 G16 X_ Y_ or

G18 G16 Z_ X_ or

G19 G16 Y_ Z_

X_ Y_: in G17 plane, X_ specifies the radius for the polar coordinates and Y_ specifies the

angle for the polar coordinates.

Z_ X_: in G18 plane, Z_ specifies the radius for the polar coordinates and X_ specifies the

angle for the polar coordinates.

Y_ Z_: in G19 plane, Y_ specifies the radius for the polar coordinates and Z_ specifies the

angle for the polar coordinates.

Description: the polar coordinate command is defined with the radius and angle. If you select

G17 plane, the first axis (X axis) refers to the radius value and the second axis (Y

axis) refers to the angle degree. For the angle degree, the counterclockwise

direction of the selected plane is positive and the clockwise direction is negative. If

the radius is specified as negative value, it is regarded as an absolute value in the

command. The angle can be specified with either incremental or absolute values.

[Example]
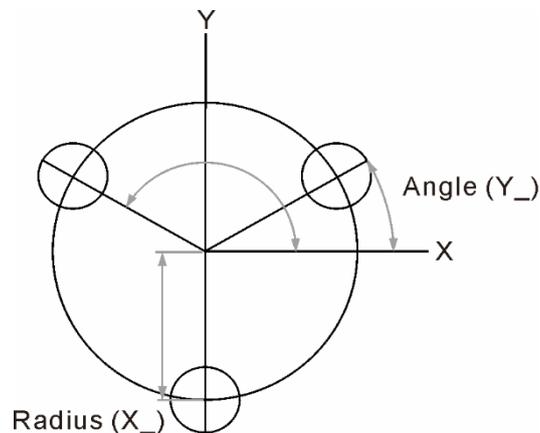

```
N1 G90 G16                                          (Use the polar coordinate command.)
N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0             (Perform the drilling cycle at the position with the
                                                     radius as 100 mm and angle as 30 degrees.)
N3 X100.0 Y150.0                                     (Drill the second hole.)
N4 X100.0 Y270.                                     (Drill the third hole.)
N5 G15 G80                                          (Cancel the polar coordinate command.)
```

G17/G18/G19: Plane designation

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_

2

Description: this command is for switching among the planes. If it is a triple-axis synchronous
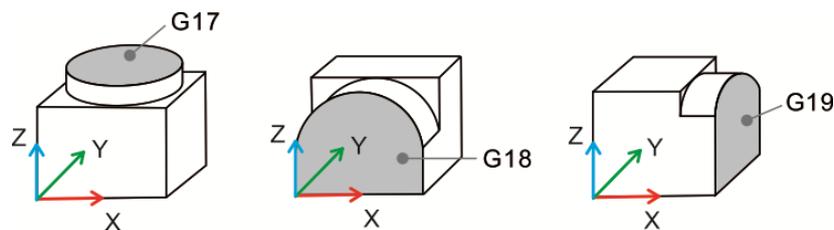
         motion, setting this command is not required. G17 - G19 can only be used in the

         condition that allows linear interpolation, circular interpolation, or tool compensation.

         The milling system's default plane after booting is G17. In this case, you do not need

         to set G17 additionally when selecting the X-Y plane for machining.

G21/G20: Metric / imperial input

Format: G21 or

         G20

         G21: metric unit settings.

         G20: imperial unit settings.

Description: you can use this command to specify the unit in metric or inch. G21 and G20 are

         only applicable to linear axes and do not affect the rotation angles of the rotation

         axes. You must input this command before starting the program; changing the

         metric or imperial settings is not allowed during program execution. This command

         changes the numeric units relevant to the system, such as the cutting feed rate (F

         value), coordinates command values, workpiece coordinate offset amount, tool

         compensation amount, and moving distance. G21 and G20 commands are

         continuously effective. Once you have specified the system unit at the beginning of

         the program, the program refers to this setting and uses metric or inch as the unit.

         You cannot use both G21 and G20 in the same program.

2

G24/G25: Mirroring / cancel mirroring

Format: G24 X_ Y_ Z_

        G24: mirroring.

        X_ Y_ Z_: specify the mirroring axis and center.

        G25 X_ Y_ Z_

        G25: cancel mirroring.

        X_ Y_ Z_: cancel the mirroring axis.

Description: you can use G24 to specify the X axis, Y axis, or the X- and Y-coordinates as the mirroring center to convert the original path to the mirror path. In this case, if the paths to be cut are left-right or up-down symmetric, you can use the program path of one side to mirror the motion path of the other side to save the time for editing the program again. To cancel mirroring with G25, you need to specify the axis to be canceled. For example, G25 Y_ cancels the Y-axis mirroring function without affecting the function in other axes.
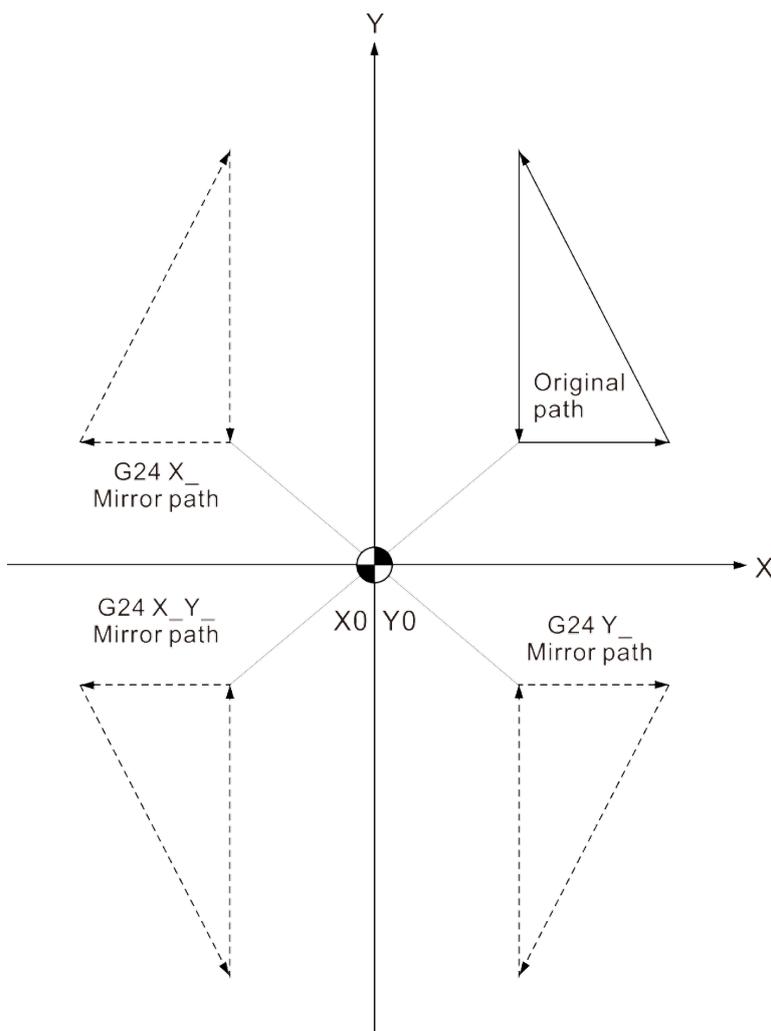
[Example]
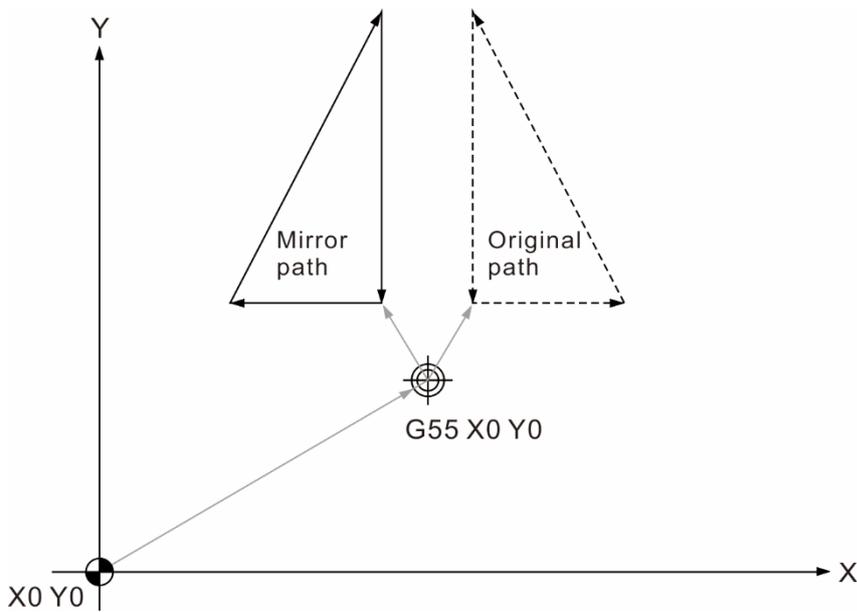

```
G00 G90 G40 G49
G55 X0 Y0
G01 X5.Y10. F1000
X35. Y10.
X5. Y70.
X5. Y10.
G00 X0 Y0
G24 X0
G01 X5. Y10. F1000
X35. Y10.
X5. Y70.
X5. Y10.
G00 X0 Y0
G25 X0
M30
```

2

The original program machines for the dotted path in the diagram above. If you use the mirroring command and run the original program, the program machines for the solid path shown in the diagram above (the specified mirroring axis is X axis in the example).

Use G25 to specify an axis to cancel the mirroring function for the corresponding axis; when the mirroring function is canceled, the program resumes its original motion path.

2

G28: Return through reference point

Format: G90 G28 X_ Y_ Z_ or

G91 G28 X_ Y_ Z_

X_ Y_ Z_: intermediate point coordinates.

Description: G28 can have the tool pass through the reference point and return to the machine origin in rapid traverse (G00). This command ensures the motion path from the current position to the specified intermediate point does not interfere with the workpiece and then returns the tool to the machine origin. X_ Y_ Z_ represents the intermediate point coordinates. The unspecified axes neither pass through the intermediate point nor return to the origin. If you have set the tool radius compensation (G41 or G42), you need to cancel the setting; otherwise, when the system is running G28, the tool radius compensation and its compensation distance setting are temporarily canceled when the tool goes to the intermediate point and then returns to the machine origin without compensation. Then, the tool radius compensation function resumes in the next motion block. When G28 is in use, the tool length compensation function (G43 or G44) remains effective when the tool reaches the reference point. Next, the tool returns to the machine origin without the tool length compensation. The tool length compensation does not resume in the following motion blocks, so you need to set it again when running the next block.

[Example]

G90 G28. Z50. (Go through the intermediate point B from point A and return to the machine origin (Z axis).)
M06 T02   (Change to tool No.2.)

[Example]

When G28 is in execution, the G90 or G91 status setting affects the process of returning to the machine origin, as shown in the following diagrams.

G90 G28 Z50.0

G91 G28 Z50.0
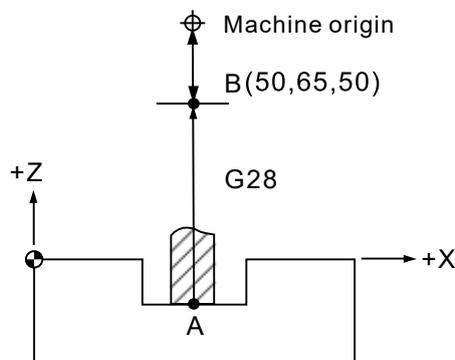

G90 G28 Z0.0

G91 G28 Z0.0
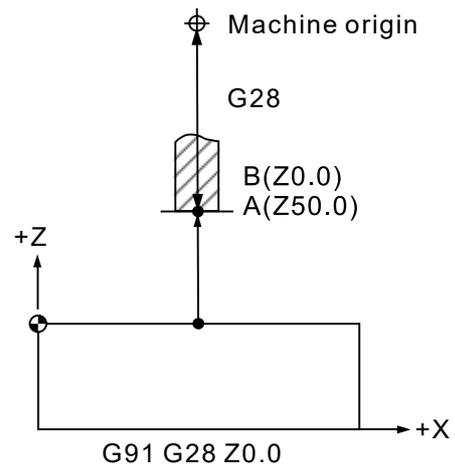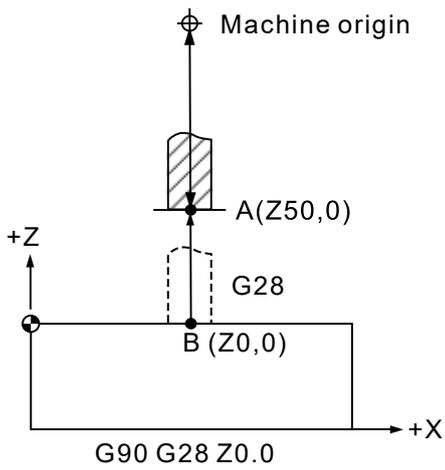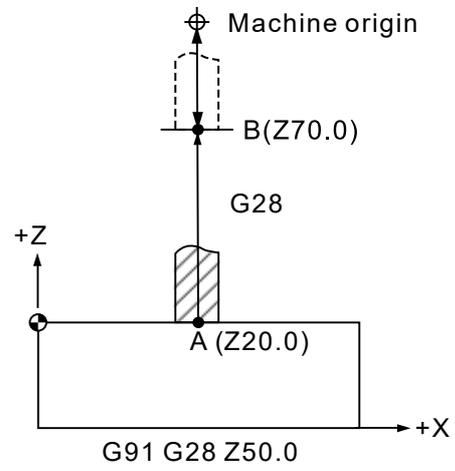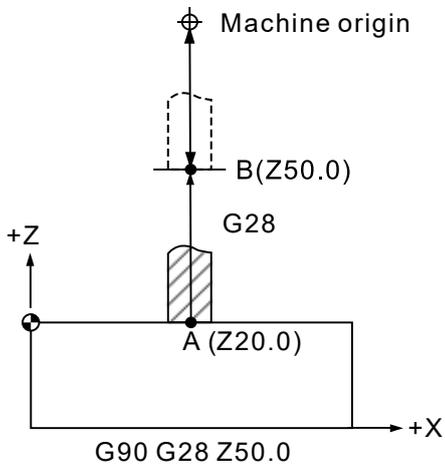
G29: Return from reference point

Format: G90 G29 X_ Y_ Z_ or

G91 G29 X_ Y_ Z_

X_ Y_ Z_: end point coordinates.

Description: G29 can have the tool pass through the intermediate point from the machine origin or any point to the specified point in the block. X_ Y_ Z_ represents the end point coordinates. G29 and G28 must be used together, so the tool moves to the intermediate point specified in G28 and then moves to the position specified in G29 without calculating the actual moving distance from the intermediate point to the machine origin. If you use G29 solely without setting the intermediate point using G28, the system displays the alarm message and stops the motion.

[Example]
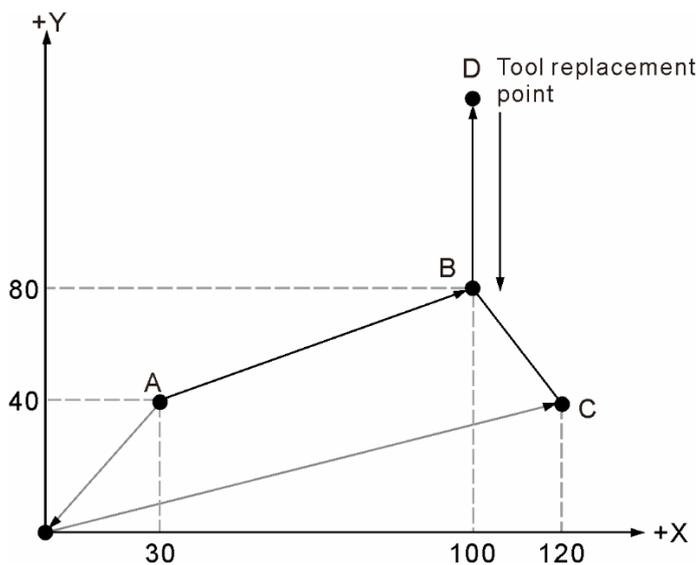
Expressed with incremental values (Tool path: A > B > D > B > C)
G28 G91 X70.0 Y40.0
M06
G29 X20.0 Y-40.0

Expressed with absolute values (Tool path: A > B > D > B > C)
G28 G90 X100.0 Y80.0
M06
G29 X120.0 Y40.0

G30: Return to the 2nd, 3rd, or 4th reference point

Format: G30 P2 X_ Y_ Z_ or

           G30 P3 X_ Y_ Z_ or

           G30 P4 X_ Y_ Z_

           P_: selection of the 2nd, 3rd, or 4th reference point.

           X_ Y_ Z_: intermediate point coordinates.

2

Description: P2, P3, and P4 commands correspond to the 2nd, 3rd, and 4th reference points respectively. When you select the 2nd reference point, you can omit P2 in the command format. X_ Y_ Z_ represents the intermediate point coordinates. The tool passes through the specified intermediate point from the current position and returns to the 2nd, 3rd, or 4th reference point, whose coordinates are set with system parameters. G30 is mostly used for tool change. When the command is set with absolute values and the motion block G30 Z0.0 is executed, the Z axis returns to the intermediate point (Z0.0) and then moves to the 2nd reference point to complete the assigned motion.

You must cancel the tool compensation setting (using G40 or G49) before using G28 or G30. If you use G30 or G28, the block including this command cancels the tool radius compensation and tool length compensation; therefore, the tool length compensation function stops in the next block and will resume until you use G43 or G44 again. The tool radius compensation function resumes in the motion block following the block including G28 or G30.
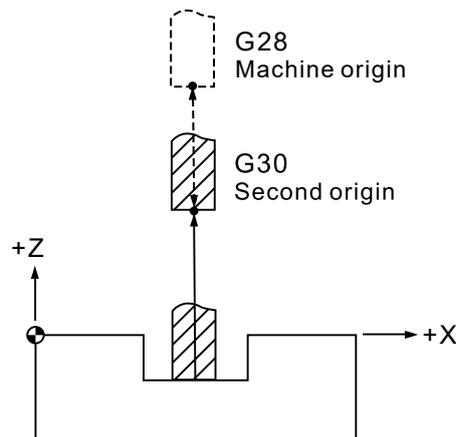
As shown in the figure above, when you use G30 Z0 in G90 mode, the Z axis first returns to the intermediate point (set to the machine origin in the example) and then goes to the coordinates specified in G30 (the second origin) to complete the homing procedure.

2

[Program illustration]

G90 G30 P2 X100. Y80.   G91 G30 P2 X0. Y0. (no intermediate point)
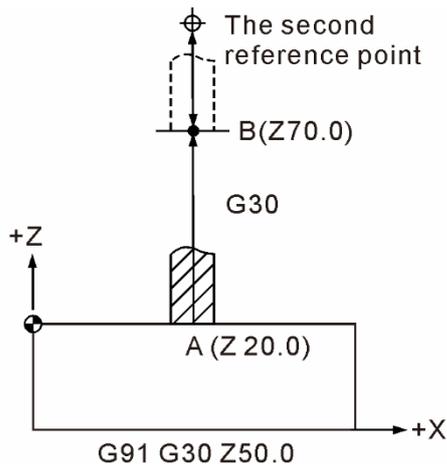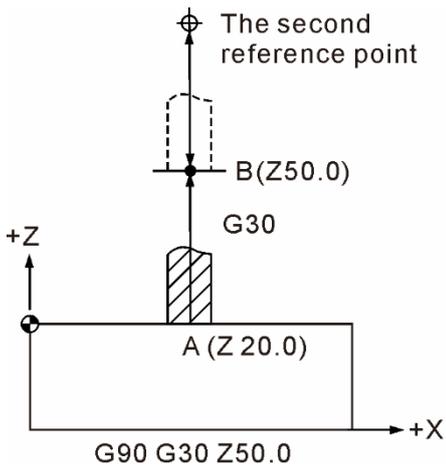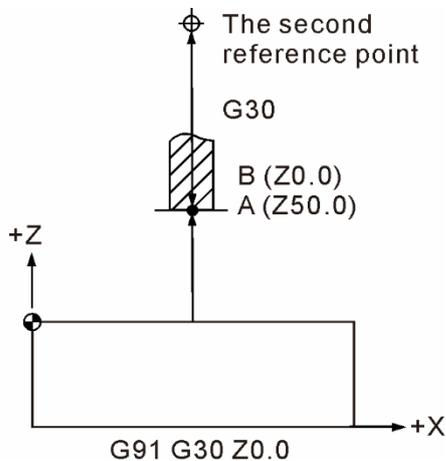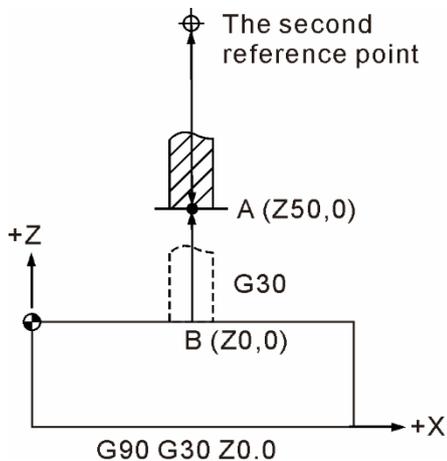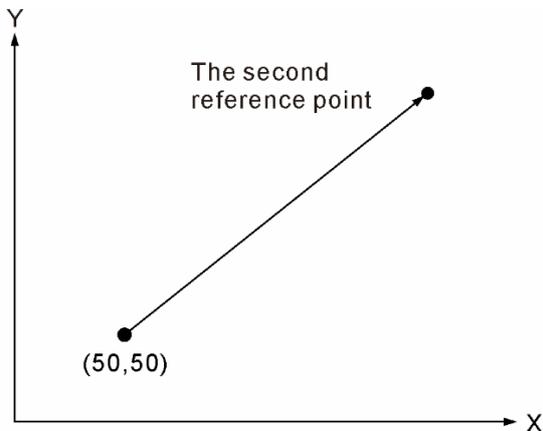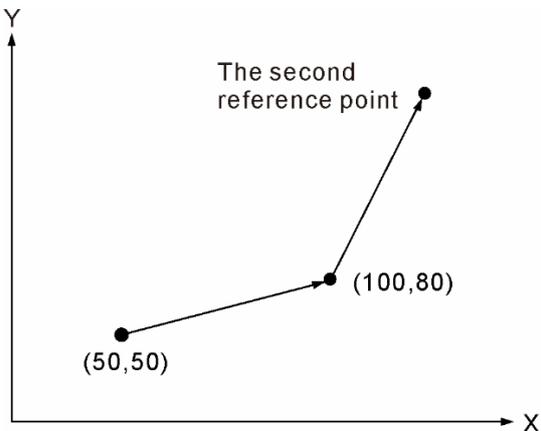
G31: Skip command

Format: G31 X_ Y_ Z_ F_ P_

Description: you can use G31 to input an external skip signal for the specified axis that is making
a linear motion, so the execution of the motion path is interrupted and the system
starts executing the next block. This G-code is a one-shot command that is effective
in one block. You cannot use G31 when the tool radius compensation (G41 or G42)
is functioning. Thus, cancel the tool radius compensation (G40) before using this
command.

Follow the instructions before using G31 skip command:

(1) You can enable the G31 high speed input 1 or 2 with the Pr46 setting.

(2) If you do not specify the P_ value in G31, the system refers to the setting of
Pr307 instead.

G31 selection range: 0 - 3;

0: no selection;

1: triggered by HSI 1;

2: triggered by HSI 2;

3: triggered by either HSI 1 or HSI 2.

(3) If you specify the P_ value in G31, the system does not refer to the setting of
Pr307. The P_ value determines which HSI to trigger;

1: triggered by HSI 1;

2: triggered by HSI 2;

3: triggered by either HSI 1 or HSI 2.

[Example 1]   [Example 2]
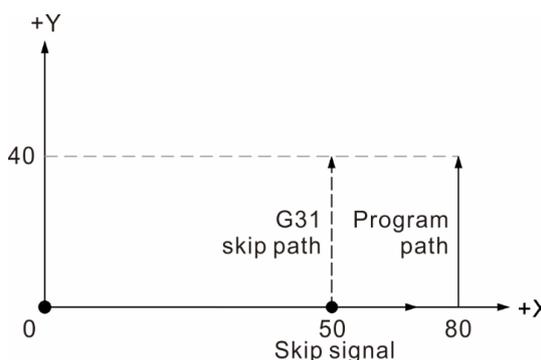
G90 G00 X0 Y0
G01 G31 X80.0 F500.0
Y40.0
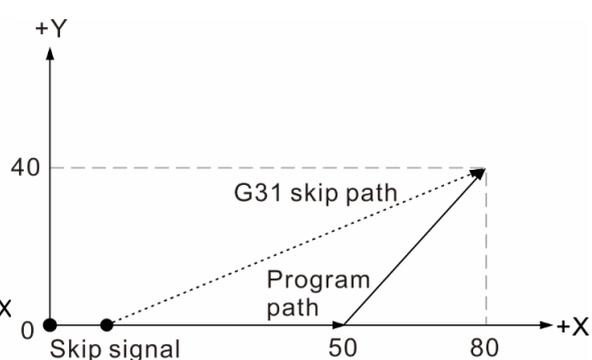
G90 G00 X0 Y0
G01 G90 G31 X50.0 F150.0
X80.0 Y40.0

The motion path in solid line shows the path without the skip command input during the process.
On the other hand, if there is a skip signal input, the program stops the current block execution
once the signal is input and starts the execution of the next block, as the path in dotted line
shows.

G40: Cancel tool radius compensation

Format: G40 or

G40 X_ Y_

Description: if the tool path does not need the tool radius compensation, you can use G40 to
cancel the compensation path. The compensation command is a status command,
so it continues to function unless you cancel it. When using the homing command,
the tool radius compensation function is temporarily canceled while the tool is
returning to the reference point. Then, the compensation function resumes in the
next motion block. Note that the tool radius compensation cancellation is not
applicable to an arc motion path.

2

G41/G42: Tool radius compensation left / right

Format: G00 G90 G41 D_ or

G00 G90 G42 D_

G41: tool radius compensation left

G42: tool radius compensation right

D_: the number for tool radius compensation data

Description: the tool center cuts the workpiece along its contour if there is no tool radius compensation function programmed in the motion path, meaning that the motion path is not adjusted to compensate the tool radius. Thus, the actual size after machining is one radius smaller. If you take the tool radius into consideration for calculating the motion path when programming, you may get wrong coordinates and have difficulty in size control. See the following figures for illustration.
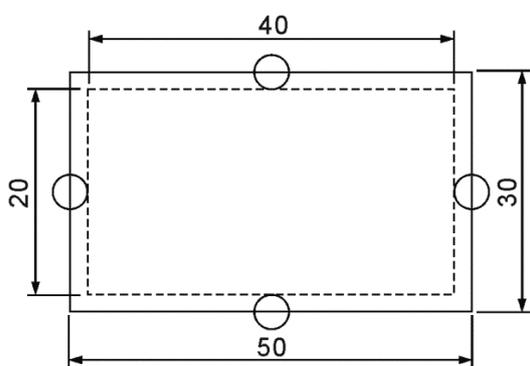
Tool radius: 10 mm   Tool radius: 10 mm

External contour milling without compensation   Internal contour milling without compensation
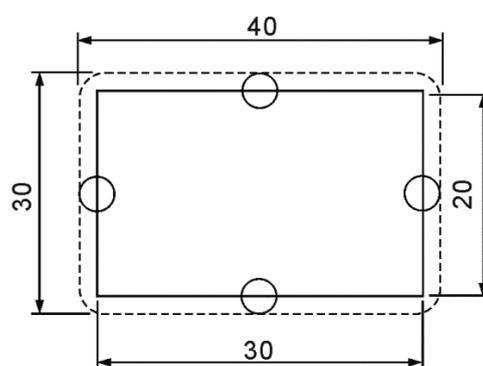
According to the figures shown above, when the tool cuts along the workpiece contour, the output size of the workpiece is one diameter larger or smaller because of the tool diameter.

You can decide when to use the tool radius compensation left or right command according to the following conditions:

■ Use G42 to set the right tool radius compensation if you look from the milling direction and the tool is on the right side of the workpiece.

■ Use G41 to set the left tool radius compensation if you look from the milling direction and the tool is on the left side of the workpiece.

D_ represents the number for tool radius compensation data, which is represented with 2 digits. This number refers to the number for tool compensation data in the [OFS group] of the controller. For example, D11 means the tool radius compensation number is No.11; if the data of No.11 is 4.0, it means the tool radius is 4.0 mm. The controller reads the tool radius data in the [OFS group] as the compensation distance according to the tool radius compensation number specified in D when using G41 or G42.

Notes for tool radius compensation:

(1) This compensation command can be used with G00 or G01 in the same block because the command takes effects only when the tool is moving (using the tool radius compensation command). It cannot be used in a block that includes G02 or G03. To use the tool radius compensation function for an arc path, you have to first set the tool radius compensation function for the linear motion path. When the compensation is active, canceling the tool radius compensation in the arc path is not allowed. See the following figure for description.
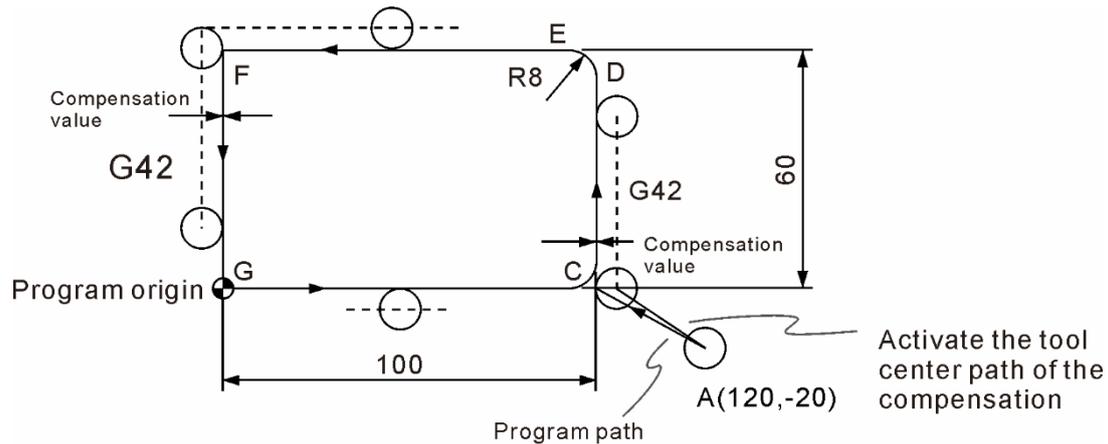
G42: tool radius compensation right

The program for enabling the tool radius compensation right command when moving the tool from point A to point C is as follows:

```
G90 G00 X120.0 Y-20.0            (Move the tool to point A in rapid traverse.)
G01 G42 X100.0 Y0 D20 F80        (from point A to point C)
Y52.0                            (from point C to point D)
G03 X92.0 Y60.0 R8.0             (from point D to point E)
G01 X0                           (from point E to point F)
Y0                               (from point F to point G)
X100.0                           (from point G to point C)
```

(2) Specify the tool radius compensation number (e.g. D11, D12, etc.) during program editing. Each tool radius compensation number corresponds to a number in the compensation data table. You need to key in the tool radius compensation value to the corresponding number in the tool setting function of the [OFS group] in advance.

(3) If there is a change in the signs (+, -) of the compensation value, the compensation direction specified in G41 and G42 will change accordingly. For example, when you assign a positive value in G41, the compensation direction is left; when you assign a negative value, the compensation direction becomes right. Likewise, when you assign a positive value in G42, the compensation direction is right; when you assign a negative value, the compensation direction becomes left.

(4) If the tool radius compensation function (continuously effective) is active and G28 or G29 is used, the compensation will be temporarily canceled. However, the system will reserve this state setting so the compensation resumes in the next motion block.

(5) When the tool radius compensation is used and the tool completes the programmed path, you must use G40 to cancel the compensation function for the tool center to return to the actual coordinates. In other words, the motion path shifts to the left or right, which is opposite to the compensation direction, to cancel the compensation distance. The best timing to use G40 to cancel the tool radius compensation is after the tool has disengaged from the workpiece. See the following program for description.

Ø10 milling cutter
Point B is the program origin
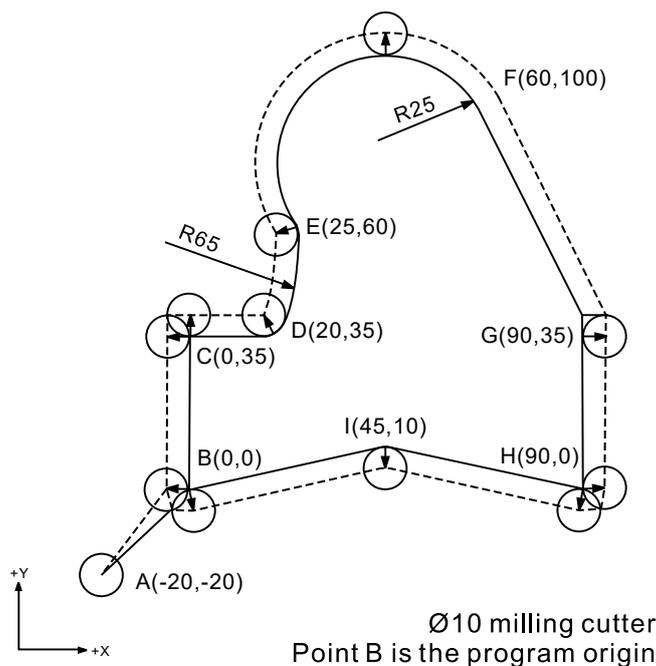
"→" means the compensation vector

G90 G00 X-20.0 Y-20.0	(Move the tool to point A in rapid traverse.)
G01 G41 X0 Y0 D12 F80	(Enable G41 tool radius compensation left when the tool moves from point A to point B.)
Y35.	(from point B to point C)
X20.	(from point C to point D)
G03 X25.0 Y60.0 R65.0	(from point D to point E)
G02 X60.0 Y100.0 R25.	(from point E to point F)
G01 X90.0 Y35.0	(from point F to point G)
G01 Y0	(from point G to point H)
X45. Y10.	(from point H to point I)
X0 Y0	(from point I to point B)
X -20. Y -20.	(from point B to point A)
G40	(Cancel the compensation after the tool has disengaged from the workpiece.)

2

(6) When compensation is enabled, the linear moving amount and inner arc interpolation radius must be $\geq$ the tool radius. Otherwise, there will be interference with the compensation vector thus causing overcutting. When this issue occurs, the controller stops running and displays the alarm message, as shown in the following figure.

Tool center path

Program path

Straight line movement amount
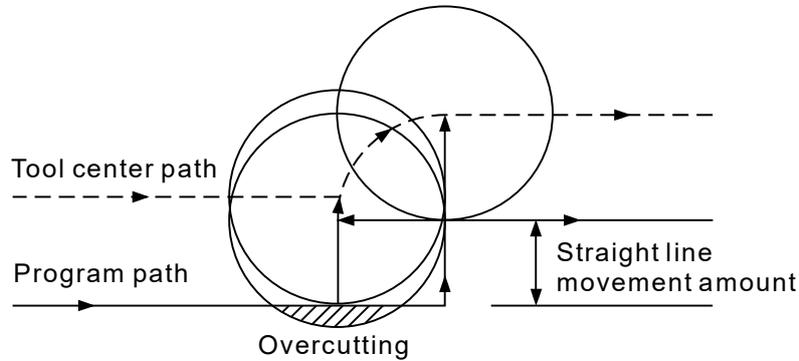
Overcutting

Linear moving amount < tool radius

(7) The canceled moving amount of the tool radius compensation must be $\geq$ the tool radius. Otherwise, there will be interference with the cutting path thus causing overcutting. When this issue occurs, the controller stops running and displays the alarm message, as shown in the following figure.

Overcutting

Program path

Tool center path

G40
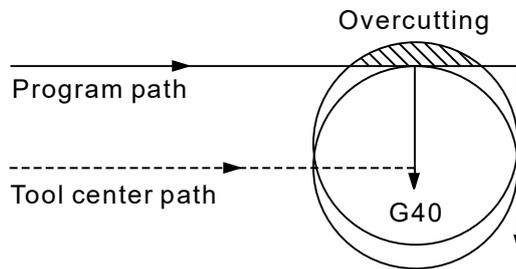
Moving distance after canceling the compensation is smaller than cutter radius

```
(D1 = 32.0)
G00 G90 G40 G49
G54 X-50.0 Y-50.0
G01 G42 D1 X0.0 Y0.0 F1000          (Start the tool radius
                                     compensation.)
X100.0
Y100.0
X0.0
Y0.0
X-50.0
Y-1.0      (End the tool radius compensation thus overcutting occurs.)
M30
```

(8) The tool radius compensation is not operable in motion blocks following the block that contains G40. Or, when you use the tool radius compensation in the last moving block without any more motion blocks following it, the tool radius compensation is not operable in the last motion block.

Tool radius compensation path for G41 and G42:

For the compensation path, you must consider the included angle θ (180° > θ > 90°, 0 < θ < 90°) formed between each block. If 180° > θ > 90°, the tool radius motion path is inward-shaped; if 0 < θ < 90°, the tool radius motion path is outward-shaped. See the following diagram for illustration.

Inward angle	Outward angle
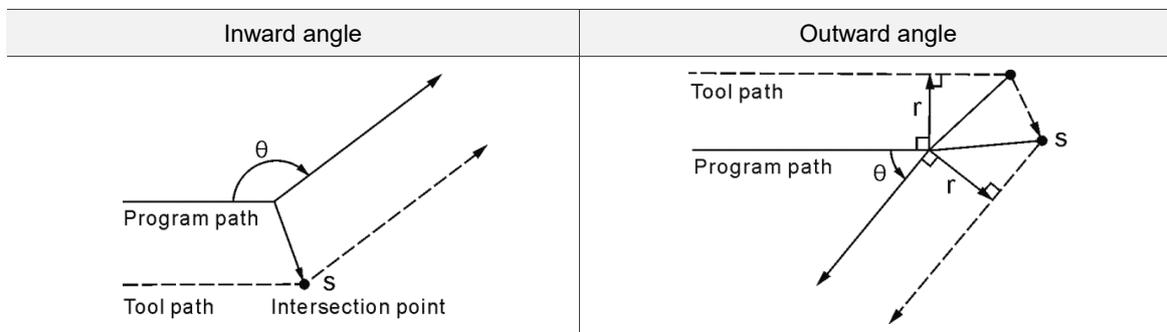
The compensation motion diagrams are shown in the following table. The controller compensates for both the start and end points of the tool path.

Line to line	

Line to arc	

Outward angle - arc to line	Outward angle - arc to arc
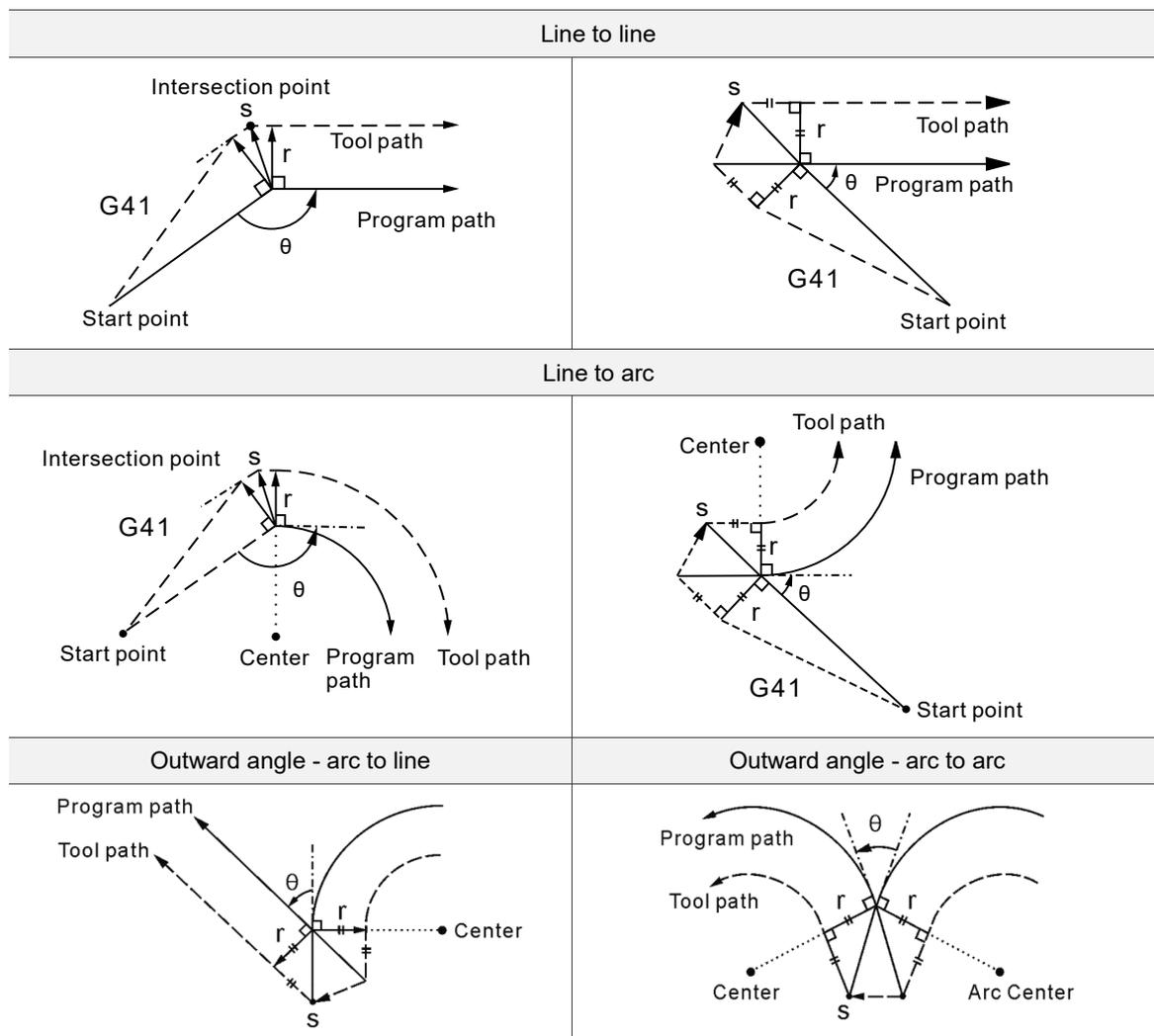
2

Compensation path change: the following diagrams show that the motion path without
compensation transits to the tool center motion path with
compensation. When the compensation command is in execution, it
continues to function for the motion path; if you use G40 to cancel
the compensation path or directly change the compensation
direction, the motion paths are shown as follows.

G40 > G41	G40 > G42

G41 > G40	G42 > G40

G41 > G42	G42 > G41

2
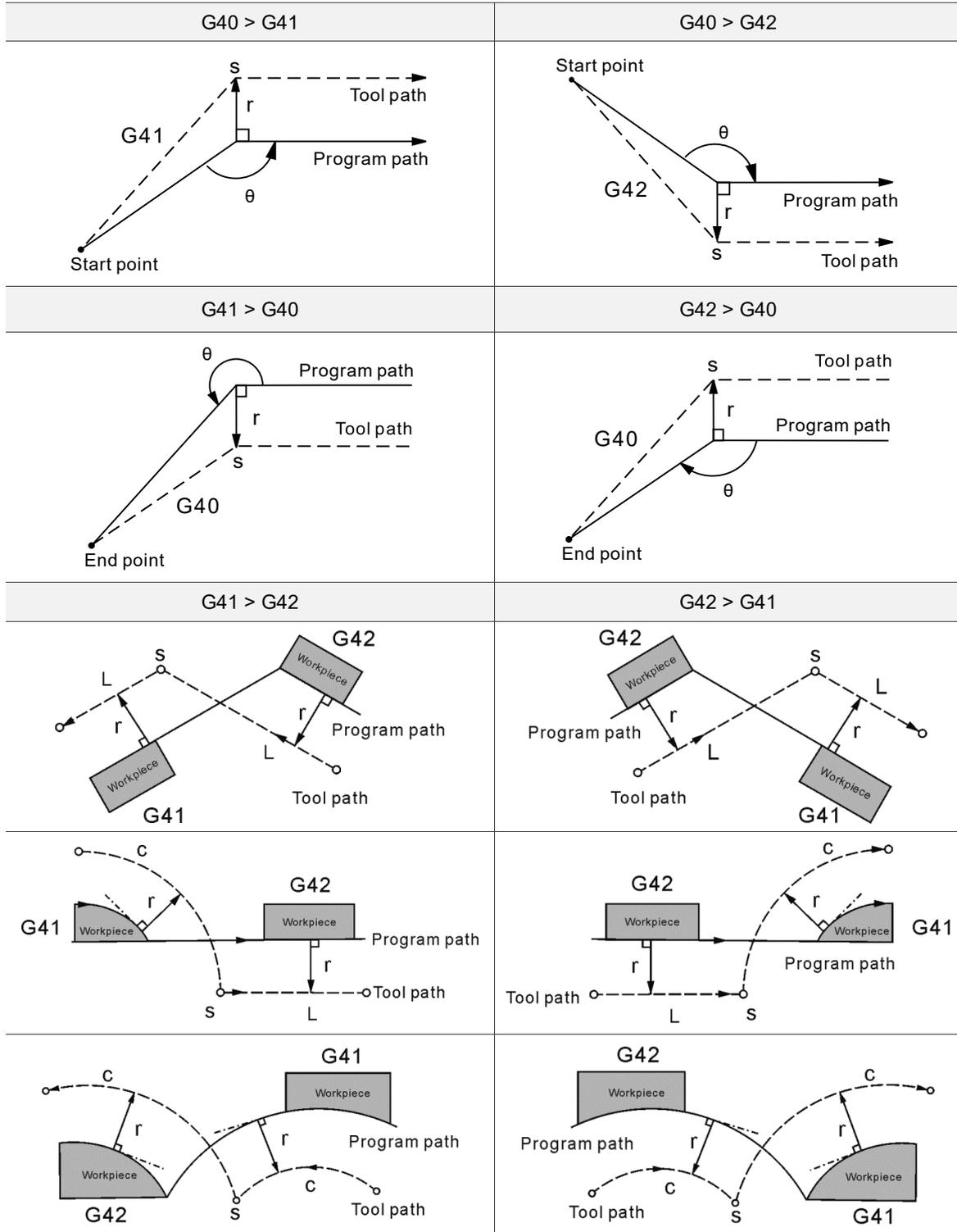
G43/G44: Tool length compensation

Format: G43 Z_ H_

G44 Z_ H_

G43: tool length compensation in positive direction. If the tool length data is a positive value, the tool axis moves towards the positive direction.

G44: tool length compensation in negative direction. If the tool length data is a positive value, the tool axis moves towards the negative direction.

Description: during the same machining process, a number of tools may be required for a numeric machine. Since the length for each tool differs from each other, you can use this command to specify the number for the length data of each tool in the program to perform the height compensation. This function compensates the length for each tool to meet the programmed depth of cut as well as simplifying the programming.

The following explains the parameters of the format:

Z: Z-axis coordinate = Z0 + tool length compensation value. Z0 is the Z-axis base point on the workpiece surface.

H: the number for the tool length compensation data, which is represented with two digits. Specify the number to designate the tool length compensation data in that number as the compensation height in this section of the program. For example, H01 means the tool length compensation number is No.01; when the data of No.01 in the tool length setting of the [OFS group] is -412.8, it means the tool length compensation data for the tool is -412.8 mm. When using G43 or G44, the controller obtains the data of the tool length compensation number specified by H as the basis for the tool length compensation.

G43 Z_ H_: when the corresponding data for the compensation number is a positive value, the tool moves up for compensation; when it is a negative value, the tool moves down for compensation. If you desire to use G44, the format is G44 Z_ H_; when the corresponding data for the compensation number is a positive value, the tool moves down for compensation; when it is a negative value, the tool moves up for compensation.

Notes for tool length compensation:

G43 and G44 are continuously effective; if you desire to cancel the tool length compensation function, use G49 or H00. (G49: cancel tool length compensation; H00: the compensation value is 0.)

[Example] Setting tool length compensation
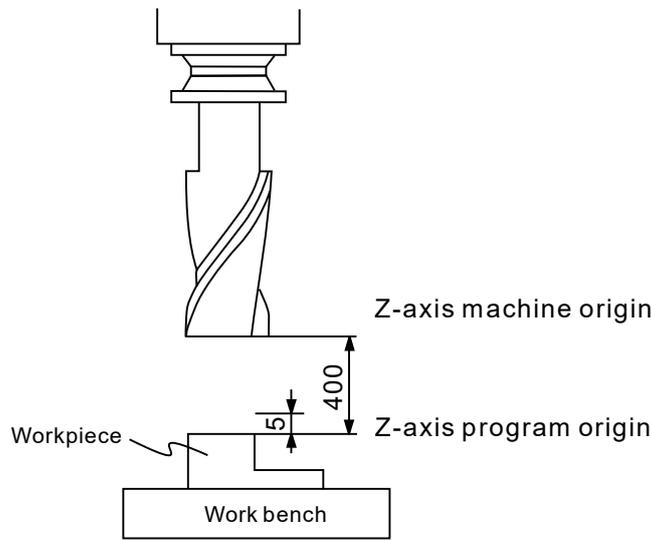
Tool length compensation

(1) G43 Z5.0 H01		(2) G44 Z5.0 H01	
No.	Data	No.	Data
01	-400.000	01	400.000
02	0	02	0
03	0	03	0

Note:

(1) When the tool length compensation is enabled, if you use G53, G28, or G30, the system cancels the tool length compensation function automatically. You need to re-specify the H_ value in the following blocks to enable the tool length compensation.

(2) Parameter 307 (Pr307) sets the tool length compensation mode for commands G43, G44, or G49 without a specified Z_ value. If you set Pr307 to 0, when you use G43, G44, or G49 without setting Z_, the tool moves by the compensation height; if you set Pr307 to 1, when you use G43, G44, or G49 without setting Z_, the tool does not move unless you set the Z_ value to move the tool for the compensation height.

(3) When the tool length compensation (G43 or G44) is enabled, if you use G28 or G30 to move the tool to the reference point, the tool length compensation function remains effective. Next, the tool returns to the machine origin without the tool length compensation. Then, the tool length compensation does not resume in the following motion blocks.

(4) When the tool length compensation is enabled, if you use the two program end commands, M30 and M02, the system cancels the tool length compensation and enables G49.

(5) When the tool length compensation is enabled, if the system receives a reset signal, it cancels the tool length compensation and enables G49.

2

G49: Cancel tool length compensation

Format: G49

Description: since the tool length compensation command is a status command, the system
reserves the tool length data after performing compensation and updates the data
until it reads other compensation numbers. In this case, you should cancel the
previous tool length compensation command before selecting different tools or use
a tool length compensation command for the tool you have changed after tool
change. Use G49 to cancel the tool length compensation function.

G51/G50: Scaling / cancel scaling

Format: G51 X_ Y_ Z_ P_ or
G51 X_ Y_ Z_ I_ J_ K_
X_ Y_ Z_: center of scaling.
I_ J_ K_: scaling factors corresponding to the X, Y, and Z axes individually.
P_: scaling factor.

Description: X_ Y_ Z_ is for setting the X-, Y-, and Z-coordinates of the center of the scaling; P_
refers to the scaling factor, and you can also specify the scaling factor for the X, Y,
and Z axes individually with the format I_ J_ K_. The command is applicable to the
machining programs for cutting products of different sizes. Set X_ Y_ Z_ as the center
of the scaling and set the scaling factor to P to scale the machining path.
The minimum value for P_ and I_ J_ K_ is 1, whose factor ranges from 0.001 to
999.999. For example, P100 means the size is reduced by 0.1 time.
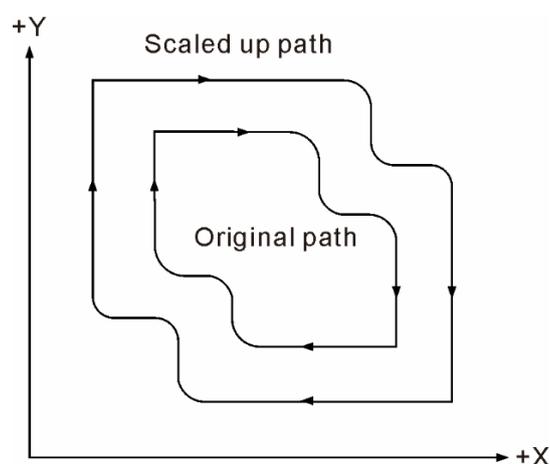
As shown in the diagram above, you specify the scaling factor to P_ in the G51
command, and the system scales the tool path generated by the original factor.
Then, the new path is generated according to the specified scaling factor. The
scaling function does not affect the compensation amount for the tool radius

2

compensation, tool length compensation, or tool position compensation because the compensation and compensation amount are calculated after scaling. When M02 or M30 is used or the NC controller is reset, the scaling is canceled. You can also press the **RESET** key to cancel the scaling. To cancel the scaling, you can use G50 in the program, then the system refers to the original factor and continues the original cutting path before scaling.

G52: Local coordinate system setting

Format: G52 X_ Y_

      X_ Y_: local coordinates system origin.

Description: during program editing, you can designate a sub-coordinate system based on the workpiece coordinates for specifying the path. And this assigned sub-coordinate system is called a local coordinate system. Set absolute values in G52 and then you can create a local coordinate system in the current workpiece coordinate system (G54 - G59). Note that G52 is valid only when it is set with absolute values instead of incremental values (G91). G52 specified with zero cancels the local coordinate system settings. When G52 is enabled, the tool radius compensation function is temporarily canceled.

[Example 1]
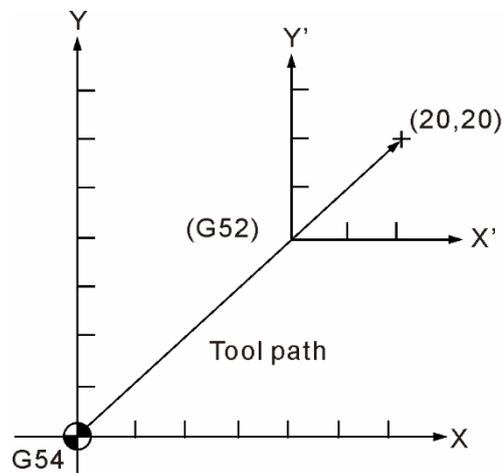
G90 G54 X0 Y0
G52 X40.0 Y40.0
G00 X20.0 Y20.0

[Example 2]

```
G90 G54 G00 X10. Y10.
G52 X30. Y20.
G00 X20. Y20.          (from point A to point B)
G56 G00 X50. Y10.      (from point B to point C)
```
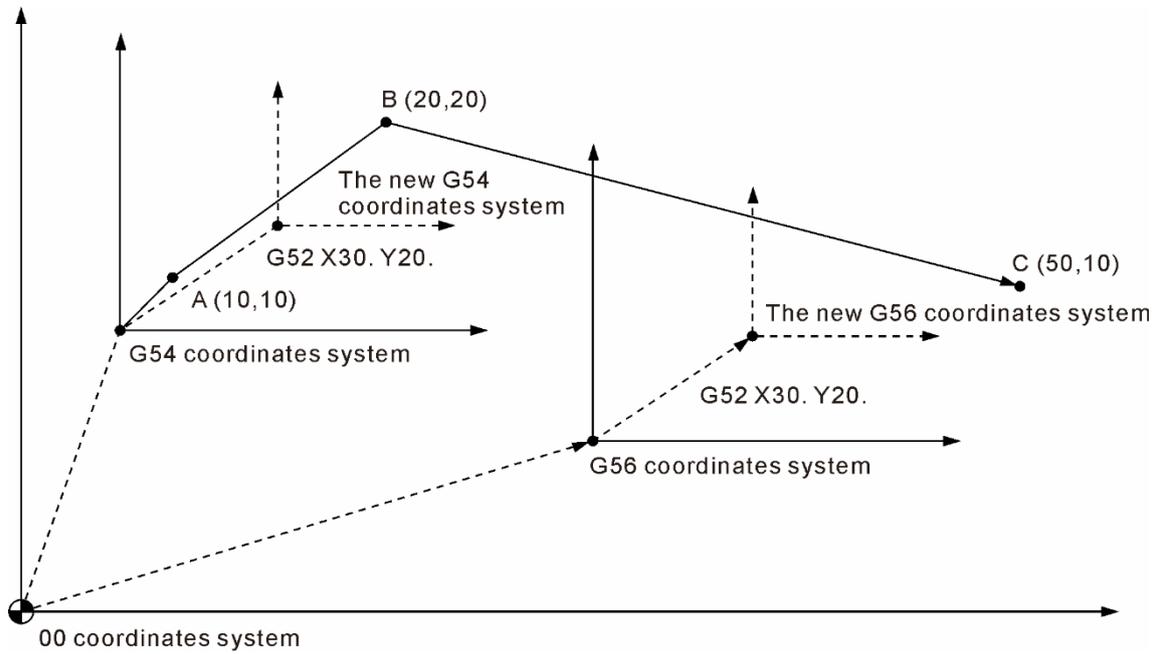

When G52 is effective, it converts the current workpiece coordinate system to another coordinate system, and the shift setting of G52 is also effective after the conversion.

To cancel the local coordinate setting, set 0 for X, Y, Z in G52 (format: G52 X0 Y0 Z0).

G53: Machine coordinate system setting

Format: G53 X_ Y_ Z_

X_ Y_ Z_: actual arrival position in the machine coordinate system.

Description: the coordinates (X, Y, Z) specify the actual arriving end point in the machine
coordinate system set in the program. Machine suppliers usually use this command
to set the tool change position, which is given based on the machine coordinates.
You must set this command in the absolute format; G53 in increment format will be
omitted.

G53 is a one-shot G command and is valid for one single block. After booting and
before using G53 to set the coordinate system, you must complete the homing
procedure in auto or manual mode. When G53 is used, the tool moves in rapid
traverse (G00) and both the tool radius compensation and tool length compensation
are canceled automatically. The tool radius compensation resumes in the next
motion block while the tool length compensation will not function until you set it
again.

Note:

(1) G53 functions only in G90 mode. G53 in G91 mode will be omitted; however, the status commands
such as G00, G01, G90, or G91 in the same block will remain effective and continue to function in
the next block.

(2) If the block contains G53 and includes an axial command, the axis moves to the specified position;
otherwise, the axis does not move.

(3) When one block contains both commands G53 and G28, the command read later is effective. If G53
is effective, the movement is made based on the machine coordinates. If G28 is effective, the
movement is made based on the absolute coordinates.

[Example]

(1) G91G53X150.Y-150.   (This block is omitted.)
    X-30.Y-30.   (The motion in this block is in incremental mode.)

(2) G90G53X50.Y-50.Z0.   (Move to the actual machine coordinates X50.Y-50.Z0.)

(3) G1G53X100.Y-100.F1000   (Execute this block with G00 rapid traverse.)
    X50.Y50.   (The motion in this block is changed to G01F1000.)

G54 - G59: Workpiece coordinate system selection

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_

Description: commands G54 - G59 allow you to assign any one of the 6 general coordinate systems as the workpiece coordinate system. To create a workpiece coordinate system, you can first move the tool from the machine origin to the program origin (X, Y). Then, input the position data in the [OFS group] for the workpiece coordinates system settings (G54 - G59). Next, use the workpiece coordinate system ID to set the workpiece coordinate origin. In addition, the system also provides 64 extension workpiece coordinate systems for selection. You can designate one by setting the P_ value within the range of 1 - 64 in G54. For example, if G54 P10 X_ Y_ Z_ is set, it means the 10th coordinate system in the extension workpiece coordinate system is used.

[Example]

O1000
G00 G90 G55 X0 Y0
G00 X100.0
Y50.0
X0.0
Y0.0
M30
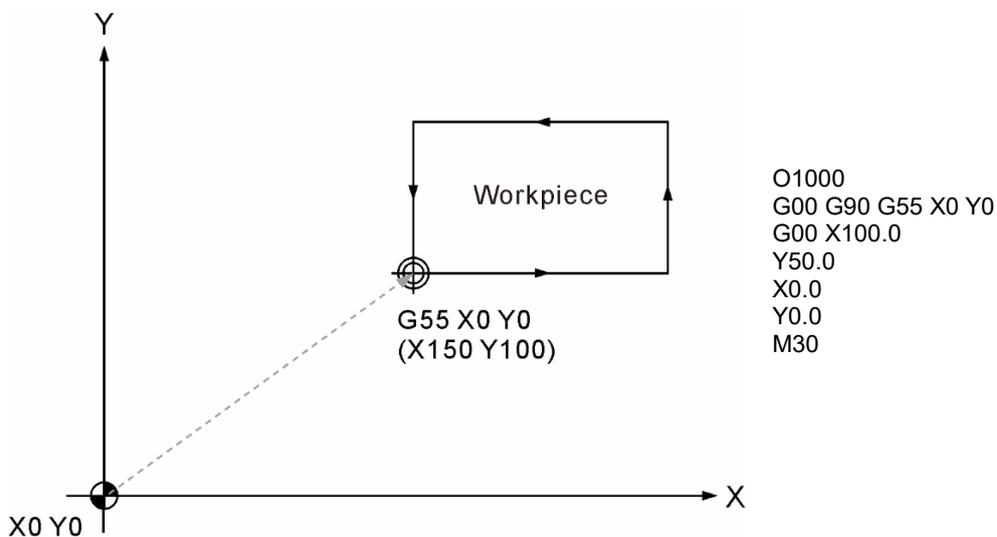
With the workpiece coordinate system settings, you can easily calculate and design the programming path, and create multiple coordinate systems on the work platform for switching among programs. As shown in the figure above, when the coordinate origin is changed, you can simply edit the workpiece coordinate data instead of creating another program to carry out machining operations.

G61: Exact stop mode

Format: G61

Description: G61 and G09 function the same except that G09 is a one-shot command (only becomes effective when specified) while G61 is not. After G61 is used, each time the system executes G01, G02, and G03, it decelerates to stop for inspection. You can use G64 (cutting mode) to cancel G61 mode or it remains effective.

[Example]

—————————	Tool path in G61 mode
— — — — — — —	Tool path in G64 mode

G61 G91 G01 Y100. F200.	(Exact positioning)
X100.	(Exact positioning)
G64	(Stop exact positioning.)
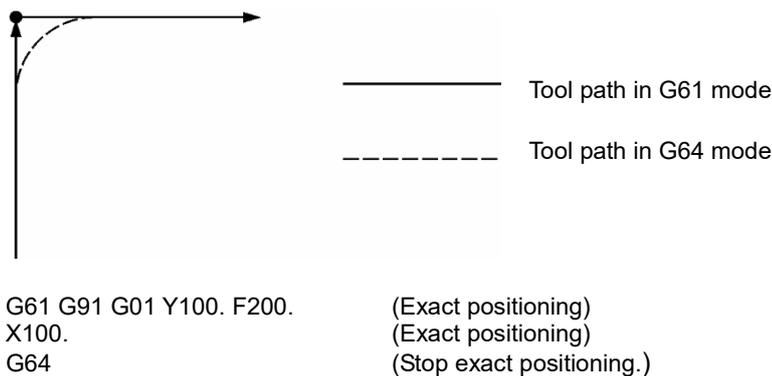
G64: Cutting mode

Format: G64

Description: when G64 is used and during the transition between blocks, the tool moves at a constant speed instead of decelerating to full stop at the end of each motion block. Normally, the initial status of the system is set to G64 cutting mode. Unlike the motion status of G61, G64 enables the tool to cut at a constant feed rate and the tool does not decelerate to stop between motion blocks. However, deceleration to full stop for inspection takes place in the following circumstances when G64 is in use:

(1)   The system executes the block that contains G00 rapid positioning command.

(2)   The system executes the block that contains G09 exact stop command.

(3)   The next block does not contain motion commands.

G65: Macro call (one-shot)

Format: G65 P_ L_ ℓ_

        P_: program number.

        L_: number of repetitions.

        ℓ_: independent variable.

Description: you can use G65 to call a macro program. Macro programs are used for commands such as operations, MLC interface data input and output, control, making statement and discrepancy, and can enable the system to perform calculations and measurements. In the main program, these macro programs are active only when they are called by specified macro programs. This command works the same way as M98 does except that G65 is a one-shot command.

[Example]

```
Main program                                    Subprogram
N0010                                            O1010
N0020 G65 P1010 L2 A5.0 B3.0                      N1020 #3=#1+#2
N0030                                            N1030
N0040                                            N1040
N0050                                            N1050 M99
```
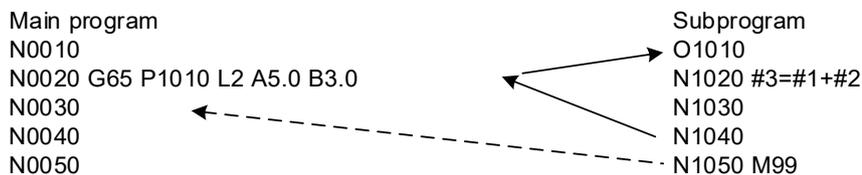
After the macro program is executed, the program returns to the block following G65 in the main program, which means the block following G65 is executed. See the example above, A5.0 means the value of variable #1 is 5.0. Refer to the following table.

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

[Program illustration]

```
O0000;        (1)    O1000;        (2)    O0001;
...          ─────▶  G65P1;       ─────▶  ...
M98P1000;            ...                  #1=2;
...          ◀─────  M98P2000;    ◀─────  ...
M30;          (16)   M99;          (3)    M99;

                          (4)    O2000;        (5)    O0002;
                        ─────▶   G65P2;       ─────▶  ...
                                 ...                  #1=2;
                         (15)    M98P3000;    ◀─────  ...
                                 M99;          (6)    M99;

                               (7)    O3000;        (8)    O0003;
                             ─────▶   G65P3;       ─────▶  ...
                                      ...                  #1=2;
                              (14)    M98P4000;    ◀─────  ...
                                      M99;          (9)    M99;

                                    (10)   O4000;        (11)   O0004;
                                  ─────▶   G65P4;       ─────▶  ...
                                           ...                  #1=2;
                                   (13)    ...                  ...
                                           M99;          (12)   M99;
```
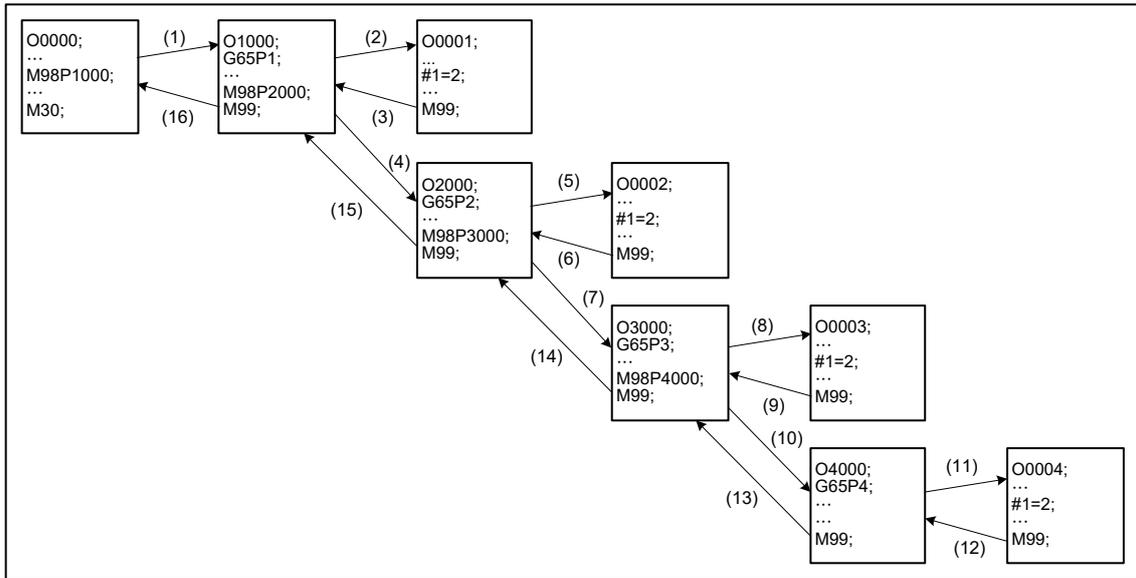
Commands G65 and G66 can nest macros up to 8 layers. When using with M98 subprogram call, the maximum layers it can call remains 8.

G66/G67: Continuous macro call / cancel continuous macro call

Format: G66 P_ L_ ℓ_ or

G67

P_: program number.

L_: number of repetitions.

ℓ_: independent variable.

Description: G66 and G65 function the same except that G65 is a one-shot command. When G66 is used, each block will start the macro program call unless there is a G67 command that cancels G66. If G67 is not used, then the macro call command continues to function.

[Example]
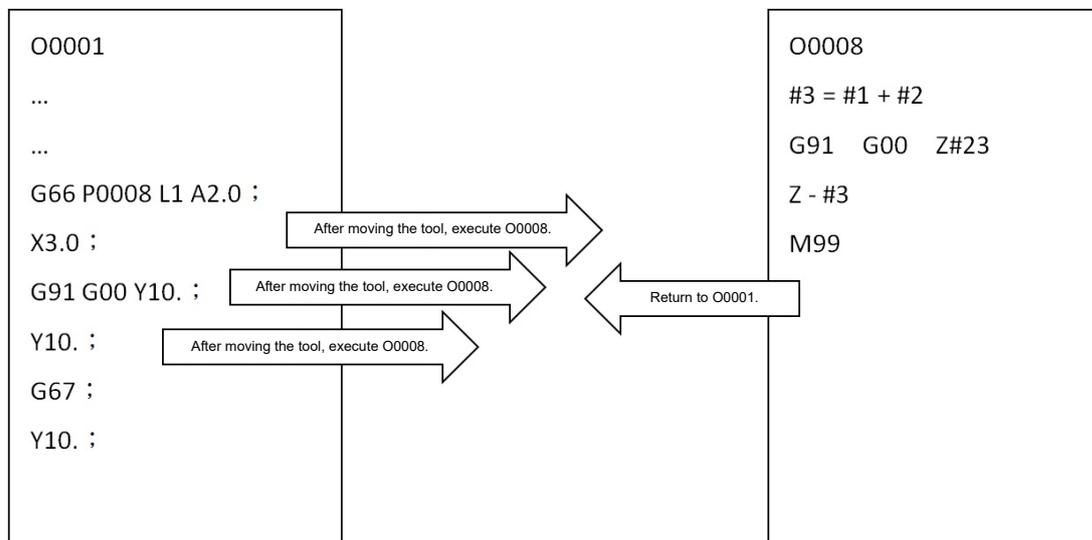
2

G68/G69: Coordinate system rotation / cancel coordinate system rotation

Format: G68 X_ Y_ R_

        X_Y_: center of rotation.

        R_: rotation angle. If the rotation direction is counterclockwise, R_ is a positive value; if the rotation direction is clockwise, R_ is a negative value. The minimum unit of rotation is 0.001°, whose setting range is 0° - 360°.

Description: you can use G68 to assign the specified coordinates as the center of rotation for converting the path of the original machining program by the specified rotation angle. Thus, when the workpiece is positioned at an angle that is different from the direction of the original machining program, you can use G68 to simplify the complicated calculation for the program path and save the time for creating another program. G68 can be enabled in absolute mode (G90) as well as in incremental mode (G91).

[Example]

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

When the path is rotated by 90°:
O1000
G00 G90 G55 X0 Y0
G68 X0 Y0 R90.0
G00 X100.0
Y50.0
X0.0
Y0.0
M30
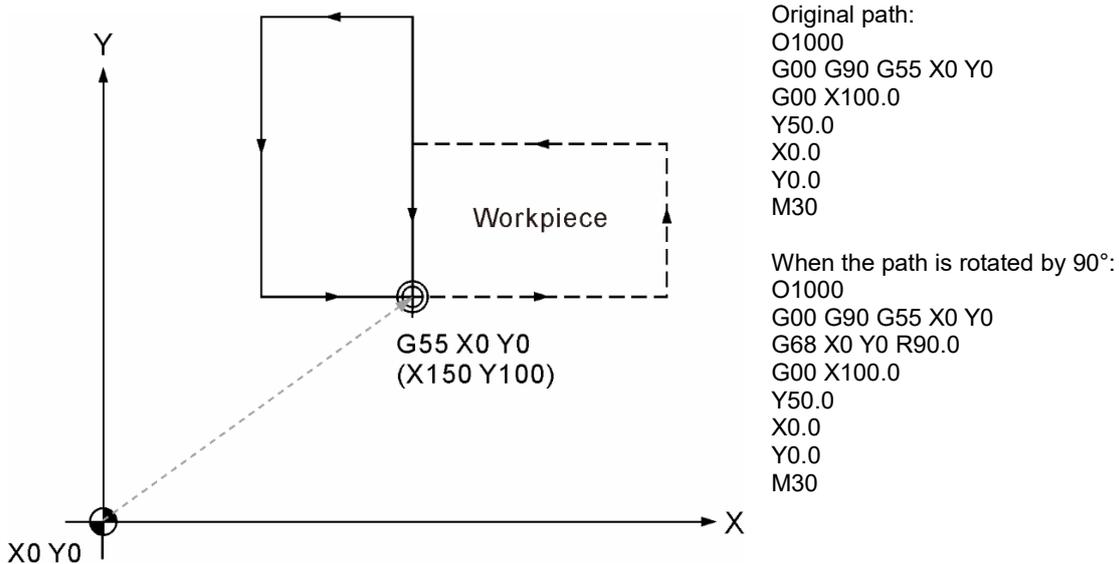
Use the upper program if the original program machines for the dotted path in the diagram above. If the workpiece is positioned in a different direction, you should have created a new program path. However, you can use G68 to rotate the program path, so a new program is not required (in the example above, the coordinate system and program path rotate by 90°). G69 cancels the function of G68: when the rotation function is canceled, the system restores the motion path to the original motion path.

G73: Peck drilling cycle

Format: G73 X_ Y_ Z_ R_ Q_ F_ K_

        X_ Y_: end position of a single block.

        Z_: Z-axis coordinate of the bottom of the hole.

        R_: initial point level.

        Q_: depth of each drill.

        F_: cutting feed rate.

        K_: number of cycle repetitions.

Description: each time the tool drills by the distance (depth) of Q, it retracts by a fixed distance d in rapid traverse, and then this motion continues until reaching point Z. The intermittent Z-axis feed allows a small retraction amount to be set in the deep hole drilling, which simplifies the chip evacuation. Q is an absolute value. d is the moving distance specified by the parameter; when d = 1 mm, it means the default retraction amount is 1 mm. See the following diagrams for the motion assigned by G73.

[Program illustration]
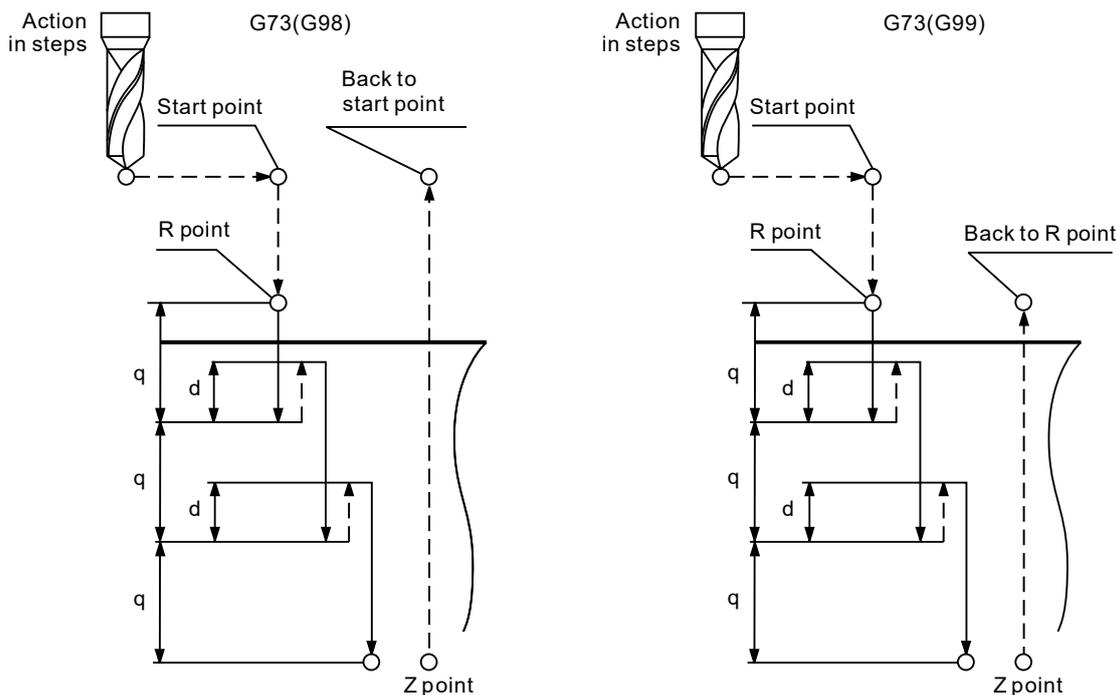
2

Note:

(1) The tool radius compensation function is canceled when the drilling cycle command is enabled.

(2) Use G80 to cancel the motion status of cycle commands.

(3) Q can only be a positive value. If you set a negative value to Q, a G-code format error occurs.

(4) If you set a decimal to K, the value is unconditionally rounded down. For example, K2.6 becomes K2 and K0.6 becomes K0.

(5) When you specify an absolute value for K, the tool repeats the motion for the specified times at the same position; when you specify an incremental value, the tool repeats the incremental equidistant motion according to the specified distance and times.

(6) When you use G73 with K specified as 0, the tool moves according to the specified coordinates (X_ Y_) without repeating the motion.

(7) If you specify a negative value to K, such as K-1.5, the result is the same as that of K1.

(8) If you specify negative decimals to K, such as K-0.8, the result is the same as that of K0.

[Example]

M03 S1000

G17 G90 G00 G54 X0. Y0.

G00 Z100.

G99 G73 X0. Y0. Z-30. R10. Q4. K1 F100. --(1)

X-15. --(2)

X-30. --(3)

X-30. Y15. --(4)

G80 G91 G28 X0. Y0. Z0.

M05

2

2
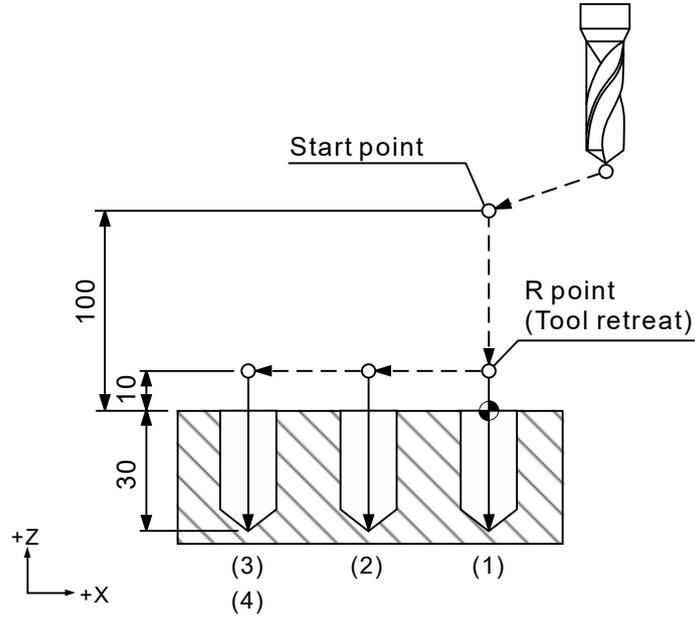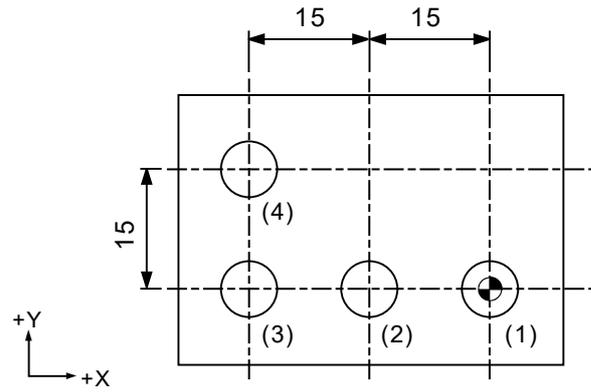
[Example]

M03 S1000
G17 G90 G00 G54 X0. Y0.
G00 Z100.
G98 G73 X0. Y0. Z-30. R10. Q4. K1 F100.   --(1)
X-15.   --(2)
X-30.   --(3)
X-30. Y15.   --(4)
G80 G91 G28 X0. Y0. Z0.
M05
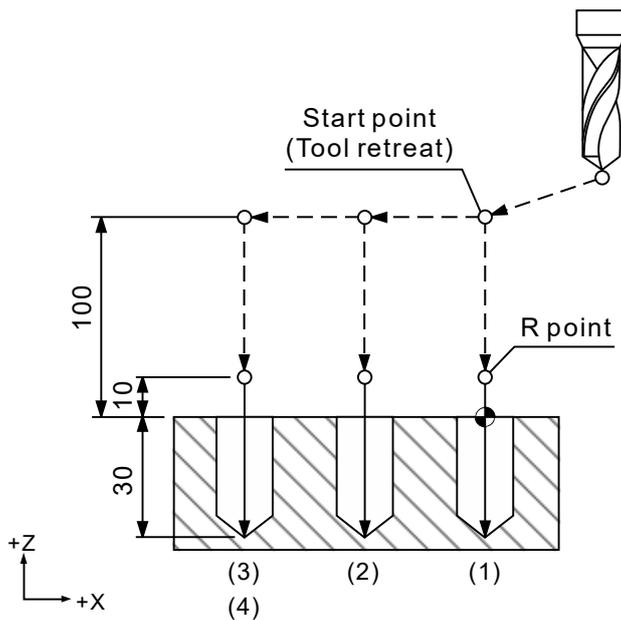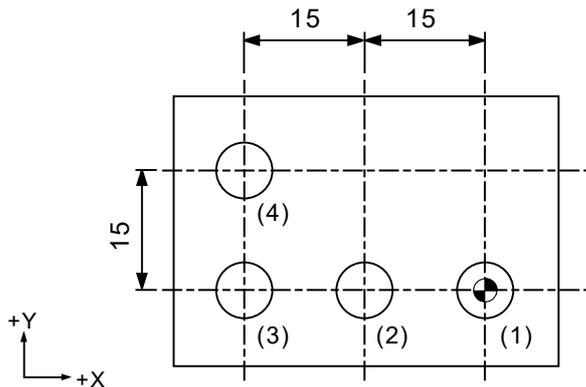
G74: Reverse tapping cycle

Format: G74 X_ Y_ R_ Q_ Z_ P_ F_ K_

X_ Y_: end position of a single block.

Z_: Z-axis coordinate of the bottom of the hole.

R_: initial point level.

Q_: depth of each tap.

P_: dwell time (unit: 1/1000 seconds). No decimals.

F_: cutting feed rate for the thread.

K_: number of cycle repetitions.

Cutting feed rate for the thread (mm/min) = lead (mm/rev) x spindle speed (rev/min).

F = P x S

Description: G74 is used for the left-hand thread machining application, which requires the left-hand threading taps and spindle rotation in reverse (CCW) direction. The motion steps for this command are as follows: move the tool center to the designated X- and Y-coordinates in rapid traverse then to the point R level, and then perform thread cutting at the specified feed rate F until the tool reaches the specified depth Z. After that, the spindle rotates in forward (CW) direction and the tool retracts to the point R level in positive direction in the meantime. After reaching the point R level, the spindle rotates in the preset reverse direction.

G74(G98)	G74(G99)
Action in steps / Start point / Back to start point / R point / ccw / cw / P / Z point	Action in steps / Start point / R point / Back to R point / ccw / cw / P / Z point
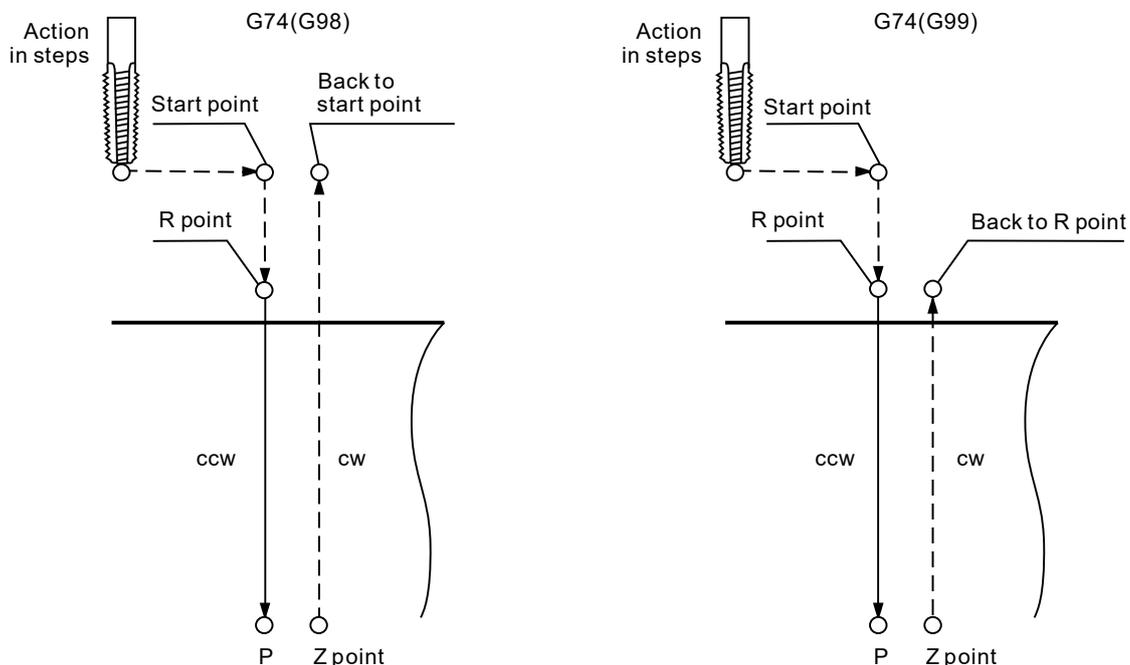
When the tapping cycle is enabled, the tool cuts at the feed rate, 100%, specified by the program. In this case, you cannot adjust the spindle speed rate or the cutting feed rate on the control panel. Moreover, when the tapping cycle command is enabled, you must disable the function of the **FEED HOLD** key before finishing the thread cutting or tapping to ensure the correct pitch.

2

[Example]

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.
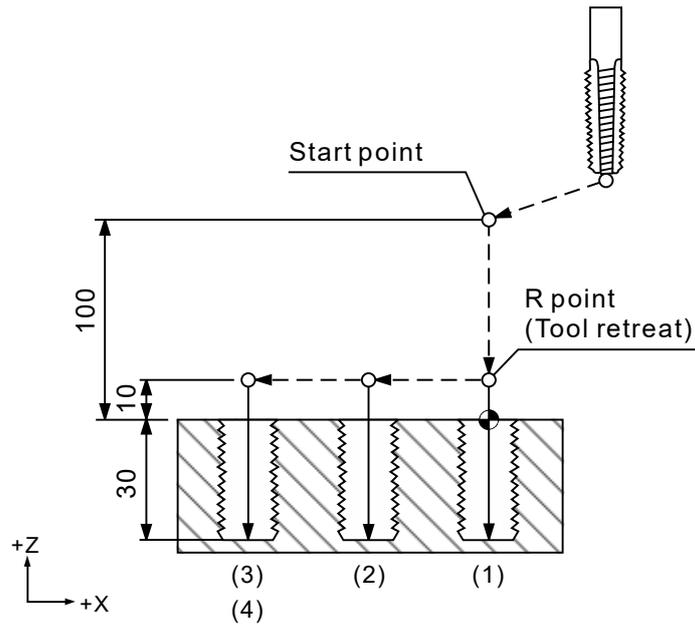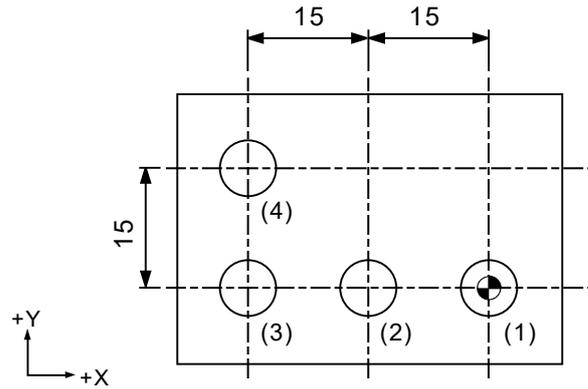
[Example]

G17 G90 G00 G54 X0. Y0.
G00 Z100.
M29 S1000
G98 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.
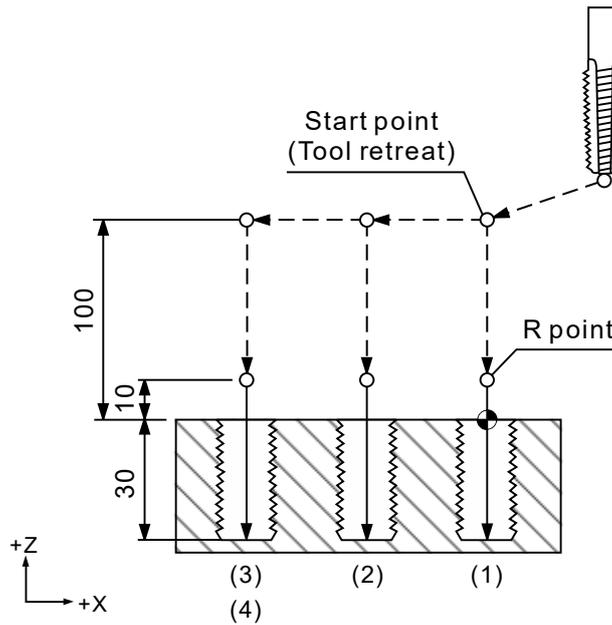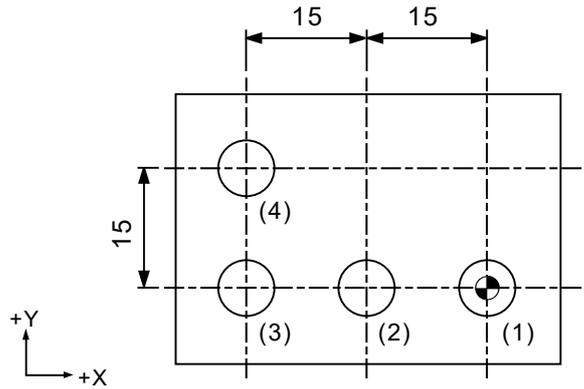
G76: Fine boring cycle

Format: G76 X_ Y_ R_ P_ Z_ Q_ F_ K_

        X_ Y_: end position of a single block.

        R_: initial point level.

        P_: dwell time (unit: 1/1000 seconds). No decimals.

        Z_: Z-axis coordinate of the bottom of the hole.

        Q_: shift amount.

        F_: cutting feed rate.

        K_: number of cycle repetitions.

Description: this command is used for fine boring. When the tool cuts until reaching the specified point, it dwells for the time specified for P_, and the spindle stops in an oriented position. Then, the tool center shifts from the workpiece surface by the specified distance Q_ to disengage from the workpiece. This prevents the tool from scratching the workpiece when retracting the tool to the point R level or the start point. After the tool retracts to point R or the start point, the tool center shifts back by the amount Q_ and returns to the start point in Z axis, then the spindle continues to rotate.
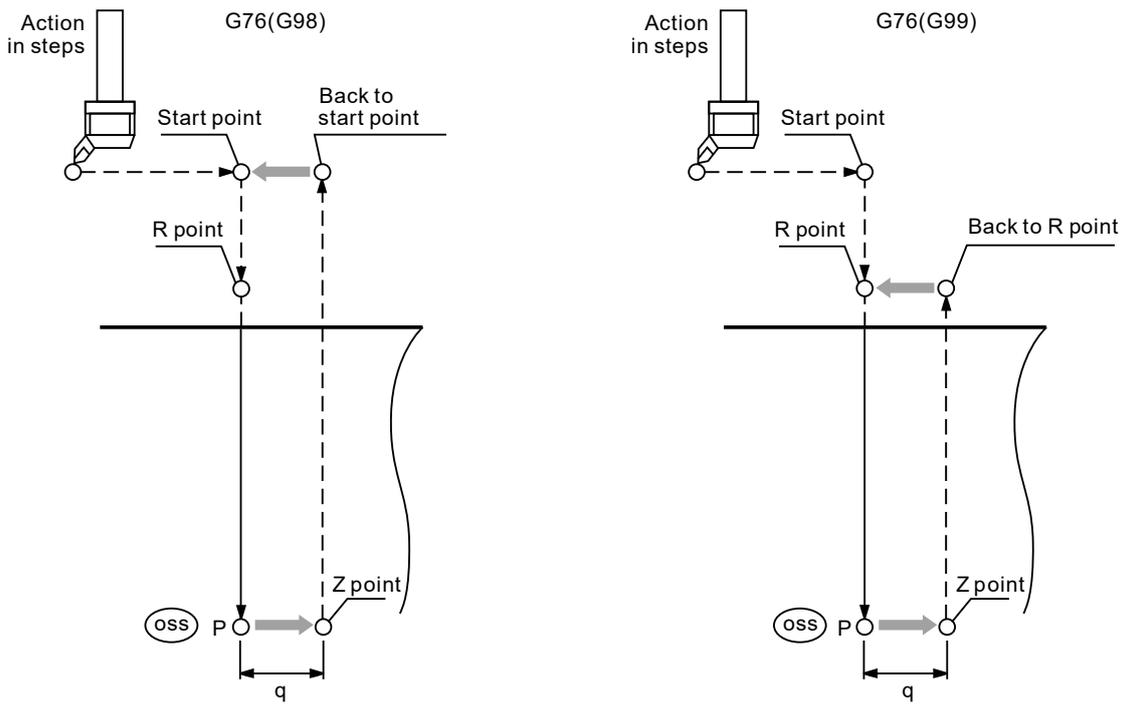
Figure 1 Fine boring cycle

2
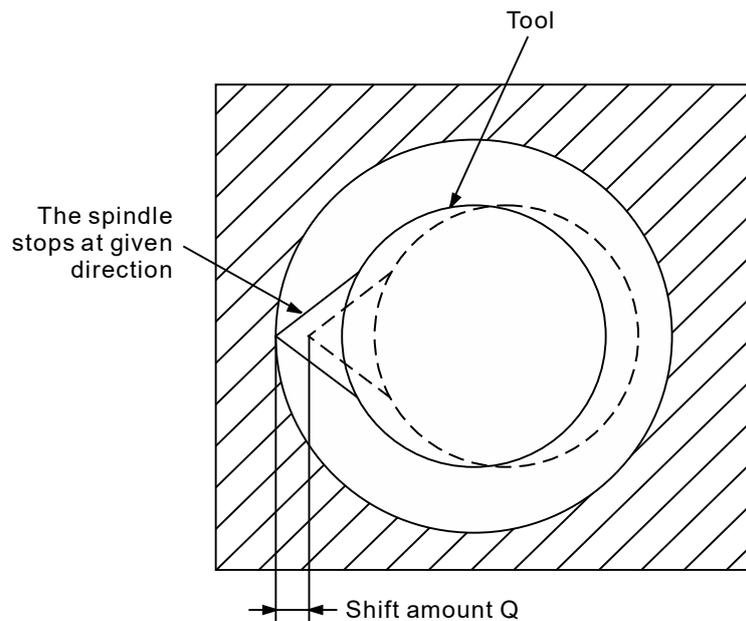
Figure 2 Fine boring cycle - shift amount

As shown in Figure 1, the boring tool moves to the specified X- and Y-coordinates in rapid traverse then to the point R level, and then bores at the specified feed rate F until reaching the specified depth Z. After that, the spindle stops in an oriented position to align the tool tip to the fixed direction, and then the tool center shifts by the specified distance Q to disengage the tool tip from the hole wall (as shown in Figure 2). This prevents the tool from scratching the machining surface when retracting the tool from the hole. When the system retracts the boring tool to point R or the start point, the tool center returns to the original center and the spindle continues rotating.

The shift amount shown in Figure 2 is specified by Q. The Q value must be positive (if you set a negative value to Q, the negative sign is omitted. So, you should set Q1.0 if the desired shift distance is 1.0 mm), and you can set +X, +Y, -X, or -Y as the fixed shift direction with the parameter. Setting the Q value within a reasonable range to prevent the tool from colliding with the workpiece.

The Q value of the fixed cycle functions is a status value, and the Q value can be used as the depth of each drill for G73 and G83, and as the shift amount for G87. As a result, when using commands G73, G76, G83, and G87, you should set a proper Q value to prevent the tool from colliding with the workpiece (G76 and G87) or avoid improper cutting amount (G73 and G83).

2

[Example]

M03 S1000

G17 G90 G00 G54 X0. Y0.

G00 Z100.

G99 G76 X0. Y0. Z-30. R10. P1000 Q5. K1 F100.　　--(1)

X-15.　　--(2)

X-30.　　--(3)

X-30. Y15.　　--(4)

G80 G91 G28 X0. Y0. Z0.

M05

[Example]

M03 S1000

G17 G90 G00 G54 X0. Y0.

G00 Z100.

G98 G76 X0. Y0. Z-30. R10. P1000 Q5. K1 F100.   --(1)

X-15.   --(2)

X-30.   --(3)

X-30. Y15.   --(4)

G80 G91 G28 X0. Y0. Z0.

M05

2

2
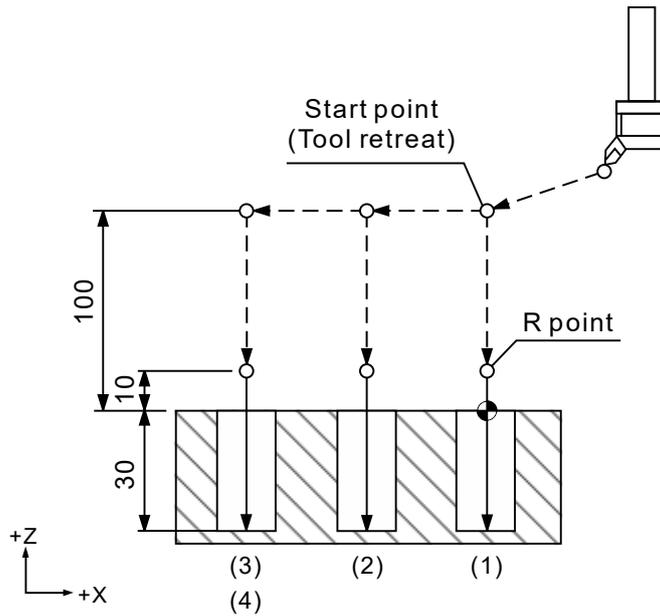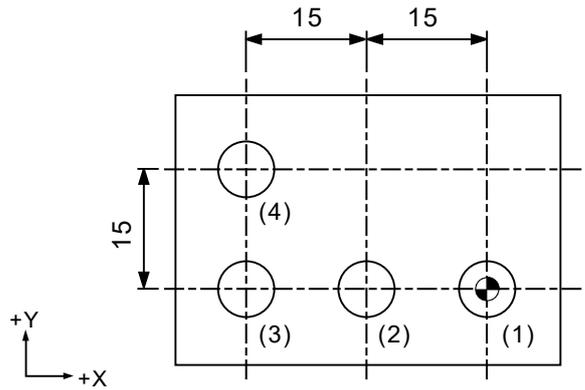
G80: Cancel cycle

Format: G80

Description: all cycle commands are status commands. You must first cancel the cycle command
to restore to the general motion mode. Use G80 to cancel the cycle functions set by
G73, G74, G76, and G81 - G89.

[Example]

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

2

G81: Drilling cycle

Format: G81 X_ Y_ Z_ R_ F_ K_

      X_ Y_: end position of a single block.

      Z_: Z-axis coordinate of the bottom of the hole.

      R_: initial point level.

      F_: cutting feed rate.

      K_: number of cycle repetitions.

Description: G81 is a drilling cycle command used for general applications. The system does not retract the tool during the drilling operation; instead, it retracts the tool to point R or the start point in rapid traverse (G00) after drilling to the specified depth Z to complete a cycle. See the following diagram for the motion assigned by G81.

2
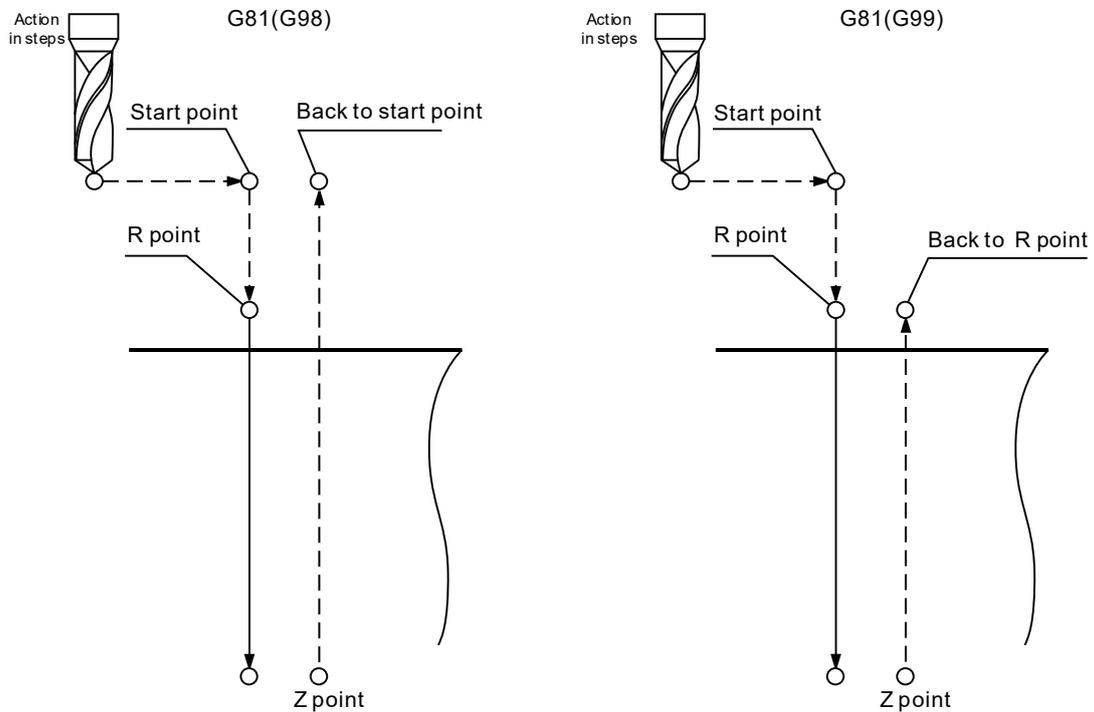
[Example]

M03 S1000

G17 G90 G00 G54 X0. Y0.

G00 Z100.

G99 G81 X0. Y0. Z-30. R10. K1 F100.   --(1)

X-15.   --(2)

X-30.   --(3)

X-30. Y15.   --(4)

G80 G91 G28 X0. Y0. Z0.

M05
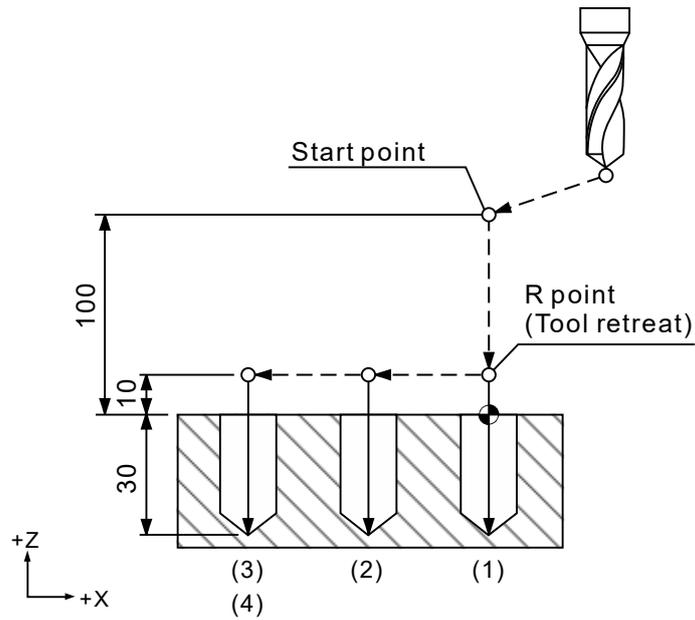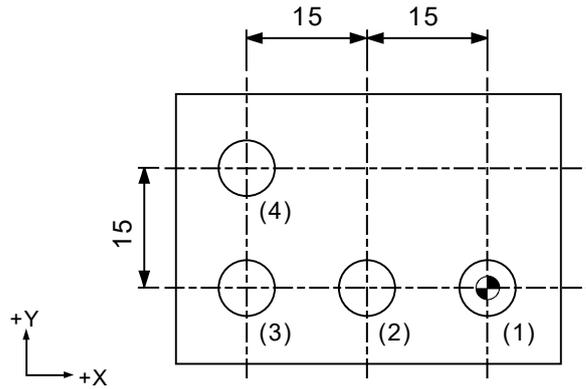
[Example]

M03 S1000

G17 G90 G00 G54 X0. Y0.

G00 Z100.

G98 G81 X0. Y0. Z-30. R10. K1 F100. --(1)

X-15. --(2)

X-30. --(3)

X-30. Y15. --(4)

G80 G91 G28 X0. Y0. Z0.

M05

2
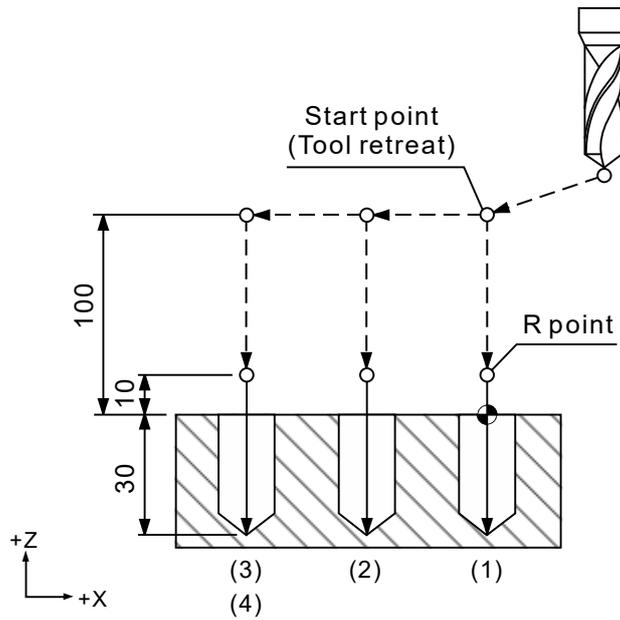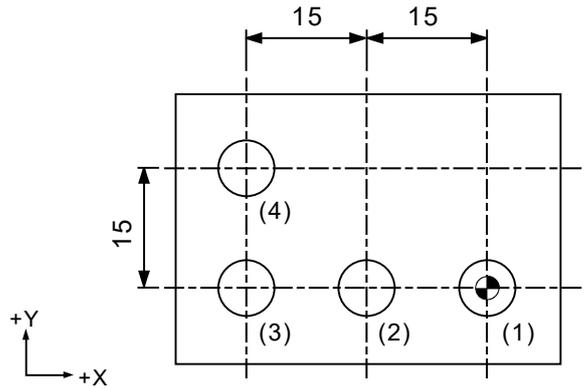
2

G82: Drilling cycle (counter boring)

Format: G82 X_ Y_ R_ Z_ P_ F_K_

         X_ Y_: end position of a single block.

         Z_: Z-axis coordinate of the bottom of the hole.

         R_: initial point level.

         P_: dwell time (unit: 1/1000 seconds). No decimals.

         F_: cutting feed rate.

         K_: number of cycle repetitions.

Description: G82 and G81 function the same except that you can use G82 to specify the dwell time for P_ in the block when the tool cuts until reaching the bottom of the hole. This command is used for counter bore, spot face, and hole machining that requires precise depth. This command specifies the dwell time when the tool cuts to the bottom of the hole to make a smoother hole wall and more precise hole depth.

[Example]

M03 S1000

G17 G90 G00 G54 X0. Y0.

G00 Z100.

G99 G82 X0. Y0. Z-30. R10. P1000. K1 F100.   --(1)

X-15.   --(2)

X-30.   --(3)
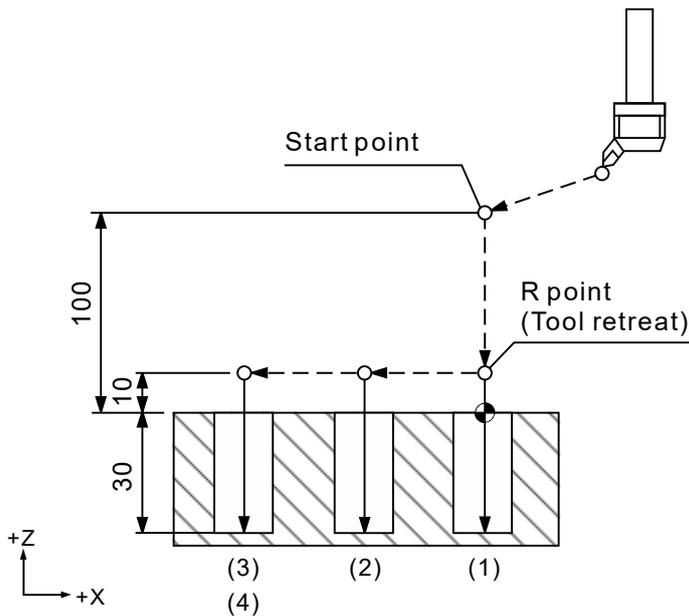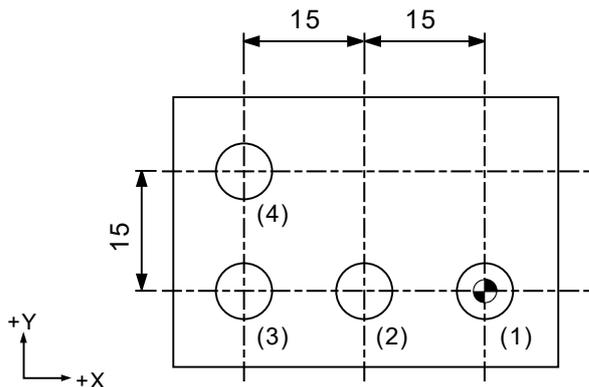
X-30. Y15.   --(4)

G80 G91 G28 X0. Y0. Z0.

M05

2

2
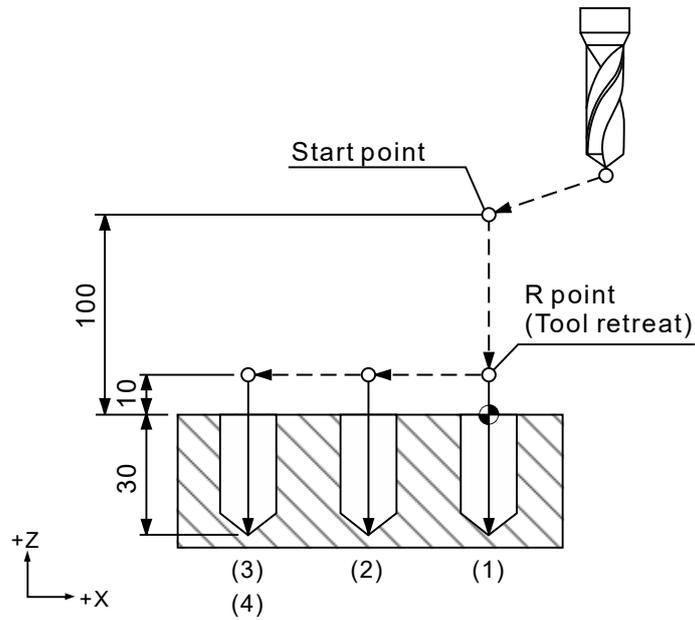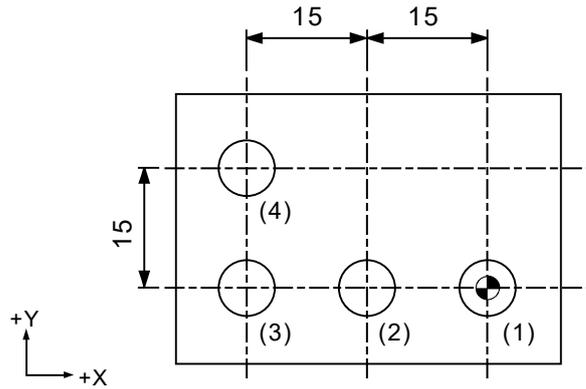
[Example]

M03 S1000

G17 G90 G00 G54 X0. Y0.

G00 Z100.

G98 G82 X0. Y0. Z-30. R10. P1000. K1 F100.   ---(1)

X-15.   ---(2)

X-30.   ---(3)

X-30. Y15.   ---(4)

G80 G91 G28 X0. Y0. Z0.

M05
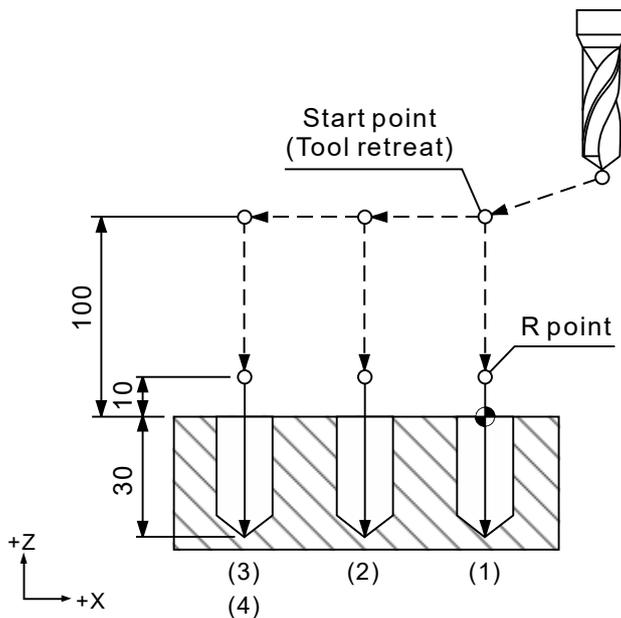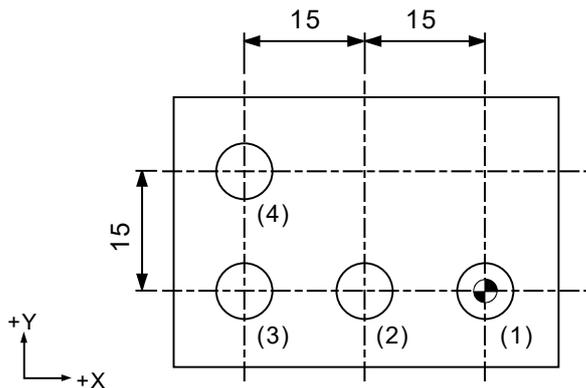
G83: Deep-hole peck drilling cycle

Format: G83 X_ Y_ R_ Q_ Z_ F_ K_

      X_ Y_: end position of a single block.

      Z_: Z-axis coordinate of the bottom of the hole.

      R_: initial point level.

      Q_: depth of each drill.

      F_: cutting feed rate.

      K_: number of cycle repetitions.

Description: G83 and G73 function the same except that G83 retracts the tool to the point R level
each time it performs a drilling operation, which allows more coolant to flow into the
hole and evacuate more chips. The following describes the motion steps: each time
the tool drills for the depth Q_, it retracts to point R and moves to the point which is
higher than the previous drilling endpoint by the distance d in rapid traverse and
then starts the next drilling with the depth of drilling as q+d. The motion continues
until the tool reaches the specified position Z. Then, the tool retracts to point R or the
start point to complete the deep-hole drilling cycle.
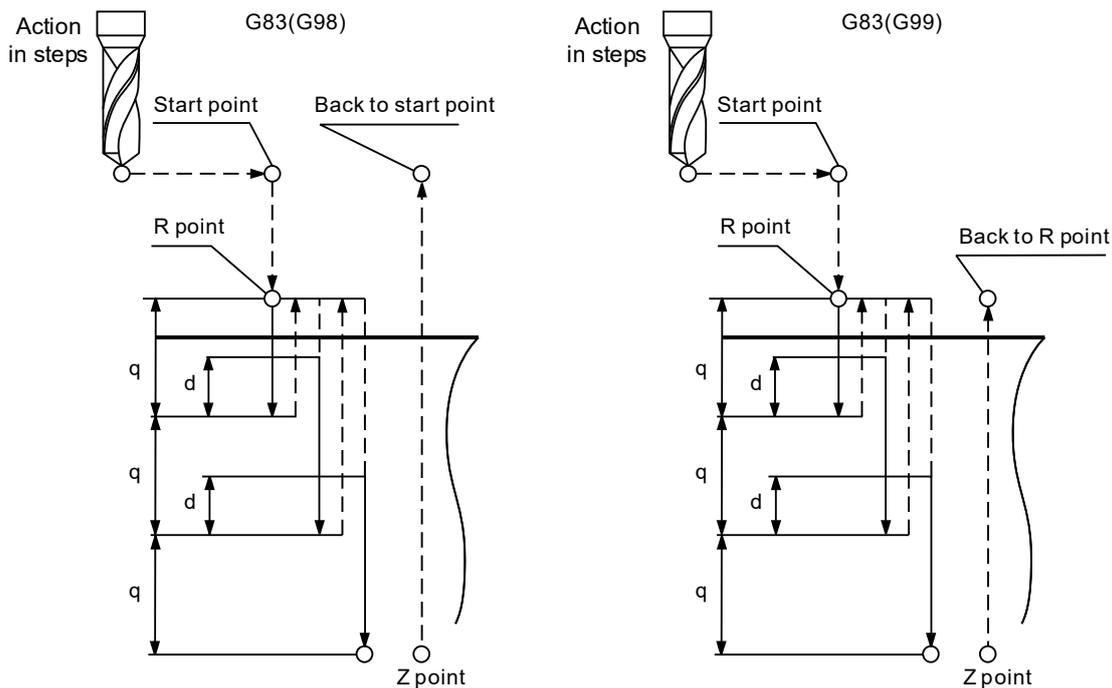
2

[Example]

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

[Example]

M03 S1000

G17 G90 G00 G54 X0. Y0.

G00 Z100.

G99 G83 X0. Y0. Z-30. R10. Q4 K1 F100.   --(1)

X-15.   --(2)

X-30.   --(3)

X-30. Y15.   --(4)

G80 G91 G28 X0. Y0. Z0.

M05
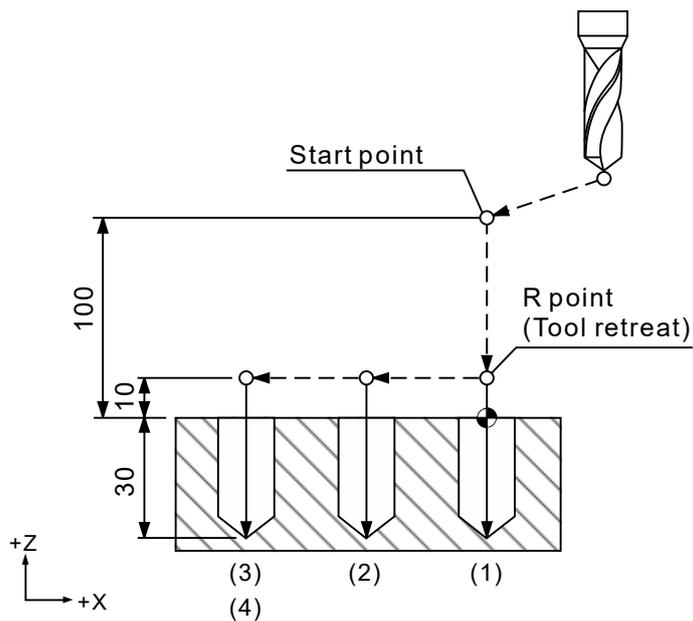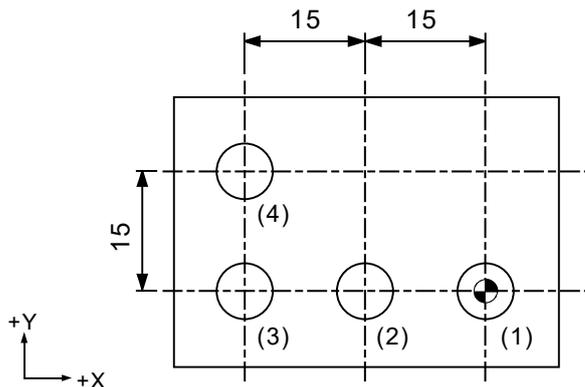
2
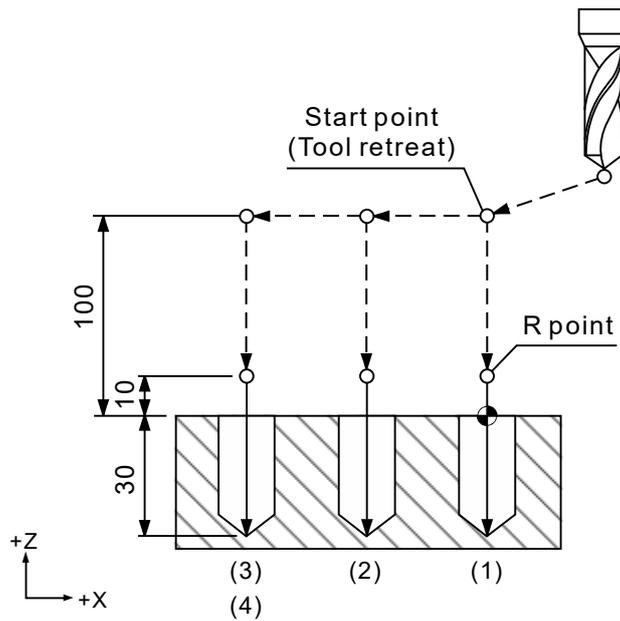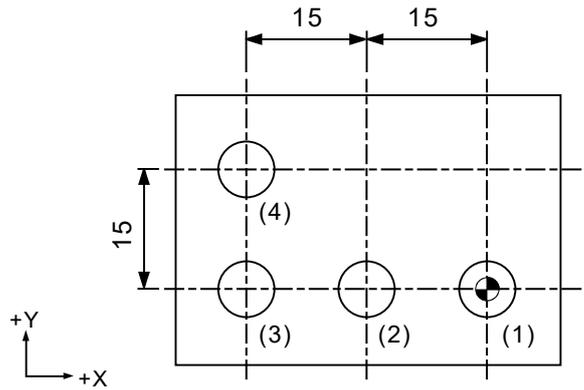
2

G84: Tapping cycle

Format: G84 X_ Y_ R_ Q_ Z_ P_ F_ K_

X_ Y_: end position of a single block.
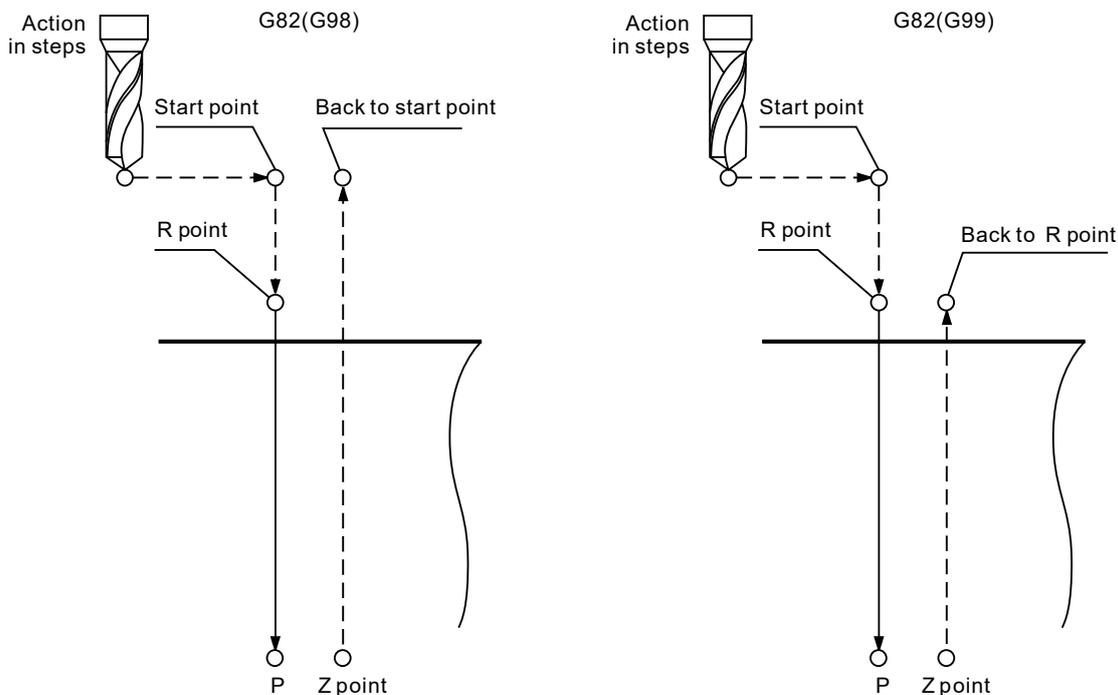
Z_: Z-axis coordinate of the bottom of the hole.

R_: initial point level.

Q_: depth of each tap.

P_: dwell time (unit: 1/1000 seconds). No decimals.

F_: cutting feed rate for the thread.

K_: number of cycle repetitions.

Description: use G84 with right-hand tap to perform the thread cutting cycle. G84 and G74 function the same except that the spindle rotation direction for G84 is opposite to that of G74.

[Example]

G17 G90 G00 G54 X0. Y0.
G00 Z100.
M29 S1000
G99 G84 X0. Y0. Z-30. R10. P1000 K1 F1000.　　---(1)
X-15.　　---(2)
X-30.　　---(3)
X-30. Y15.　　---(4)
M28
G80 G91 G28 X0. Y0. Z0.
M05

2

2
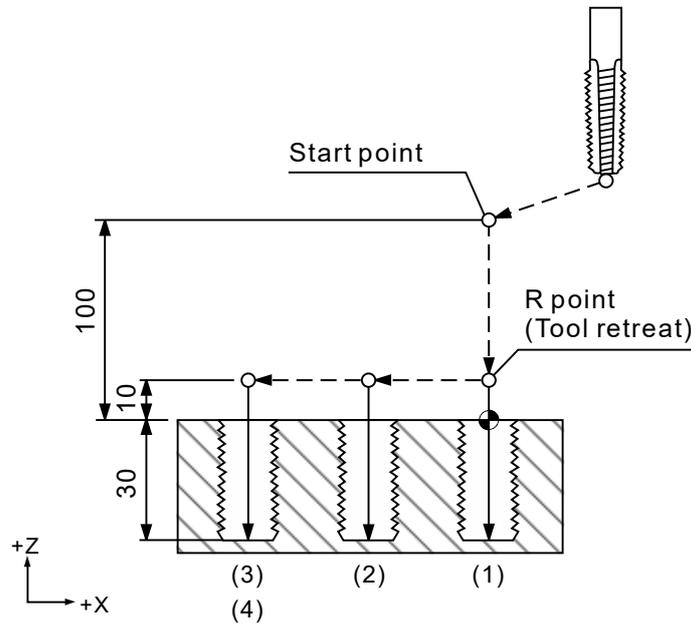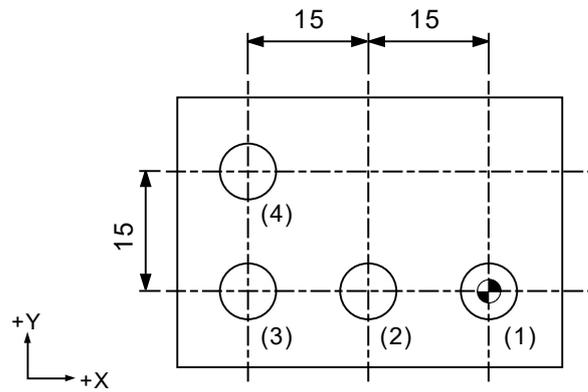
[Example]

G17 G90 G00 G54 X0. Y0.
G00 Z100.
M29 S1000
G98 G84 X0. Y0. Z-30. R10. P1000 K1 F1000.   ---(1)
X-15.   ---(2)
X-30.   ---(3)
X-30. Y15.   ---(4)
M28
G80 G91 G28 X0. Y0. Z0.
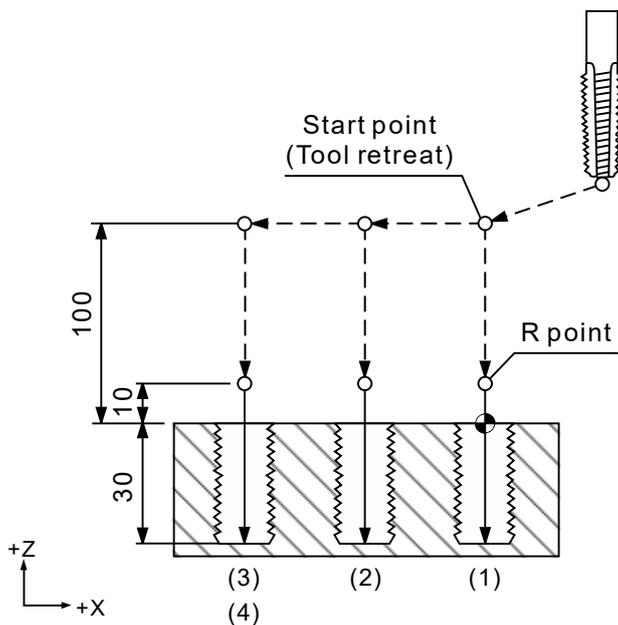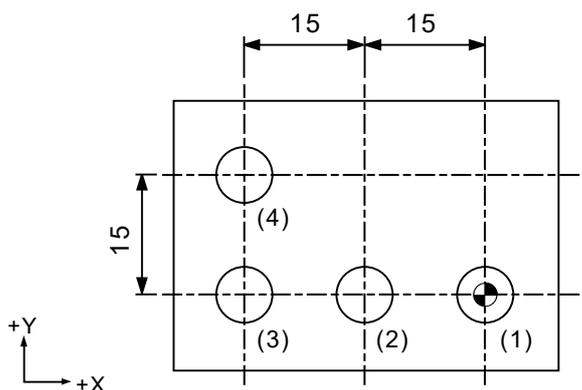
G85: Boring cycle (no dwell time)

Format: G85 X_ Y_ R_ Z_ F_ K_

      X_ Y_: end position of a single block.

      Z_: Z-axis coordinate of the bottom of the hole.

      R_: initial point level.

      F_: cutting feed rate.

      K_: number of cycle repetitions.

Description: this command increases precision of the hole cutting. You can use G85 to cut with reamers or boring bars for applications that require high precision in hole diameters such as reaming or boring. The command is used in the following steps: the tool cuts at the specified feed rate F until reaching the specified depth Z, and then retracts to point R at the same speed. Finally, the tool returns to point R (G99) or the start point (G98) with G00 rapid positioning command.

2
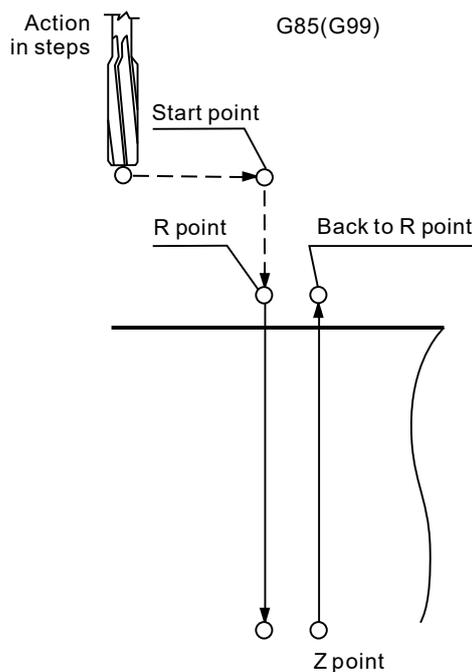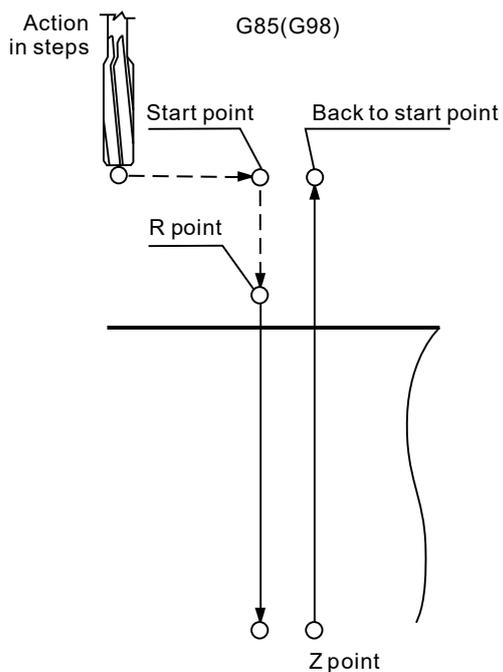
[Example]

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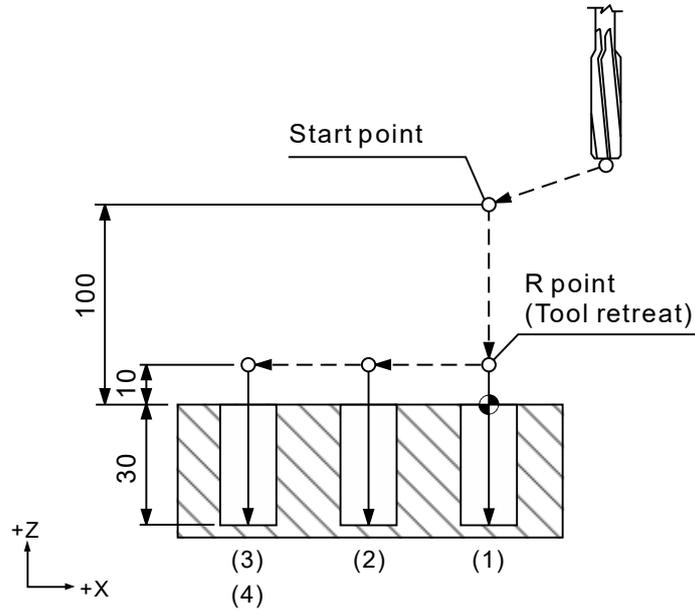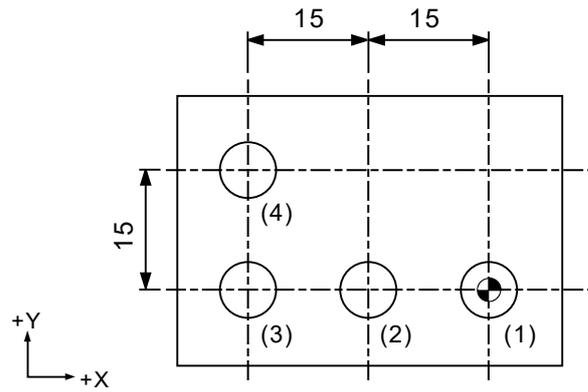
[Example]

M03 S1000

G17 G90 G00 G54 X0. Y0.

G00 Z100.

G98 G85 X0. Y0. Z-30. R10. K1 F100.   --(1)

X-15.   --(2)

X-30.   --(3)

X-30. Y15.   --(4)

G80 G91 G28 X0. Y0. Z0.

M05

2
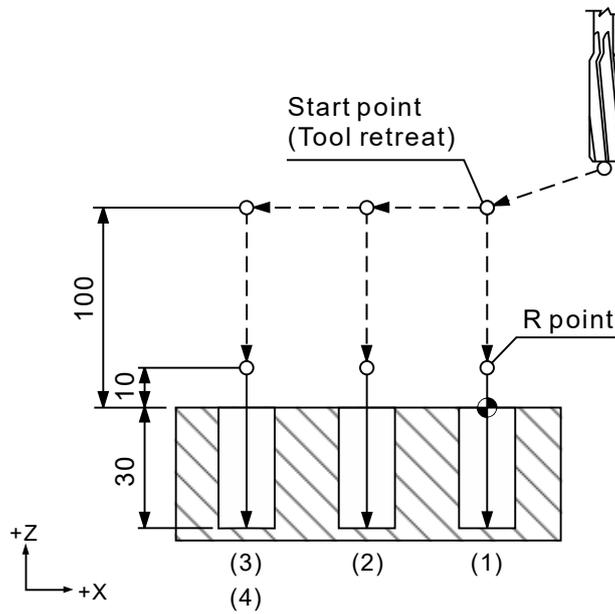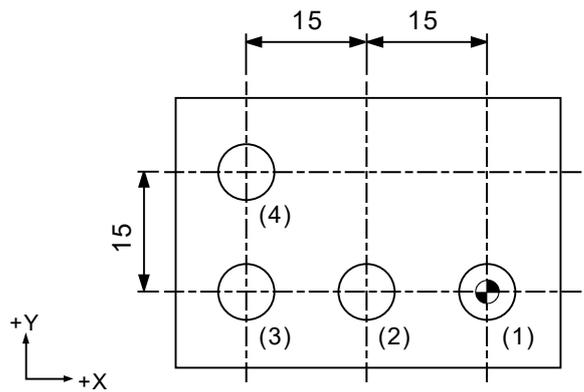
2

G86: Boring cycle (spindle stop)

Format: G86 X_ Y_ R_ Z_ F_ K_

      X_ Y_: end position of a single block.

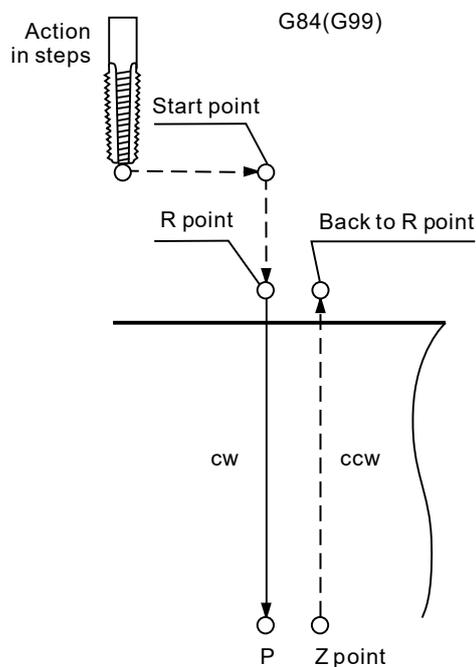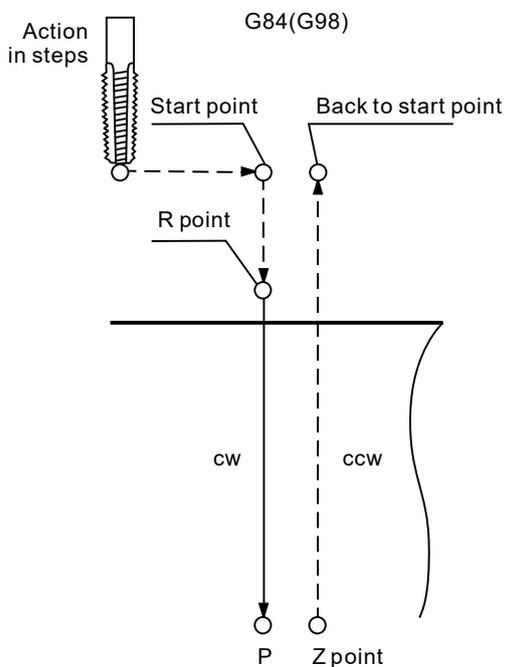   Z_: Z-axis coordinate of the bottom of the hole.

      R_: initial point level.

      F_: cutting feed rate.

      K_: number of cycle repetitions.

Description: the following figures illustrate the command. The tool cuts until reaching the specified position Z, and then the spindle stops rotating and the tool retracts with G00 rapid positioning command. After the tool returns to the initial level, the spindle starts rotating in clockwise direction and thus completes a boring cycle. The tool does not shift or tap for a fixed amount since no Q value is specified in the command. Since the tool and workpiece are in contact, the tool retraction causes slight line scratches on the hole wall because the tool is not rotating. As a result, this cycle command is usually used for rough boring applications.
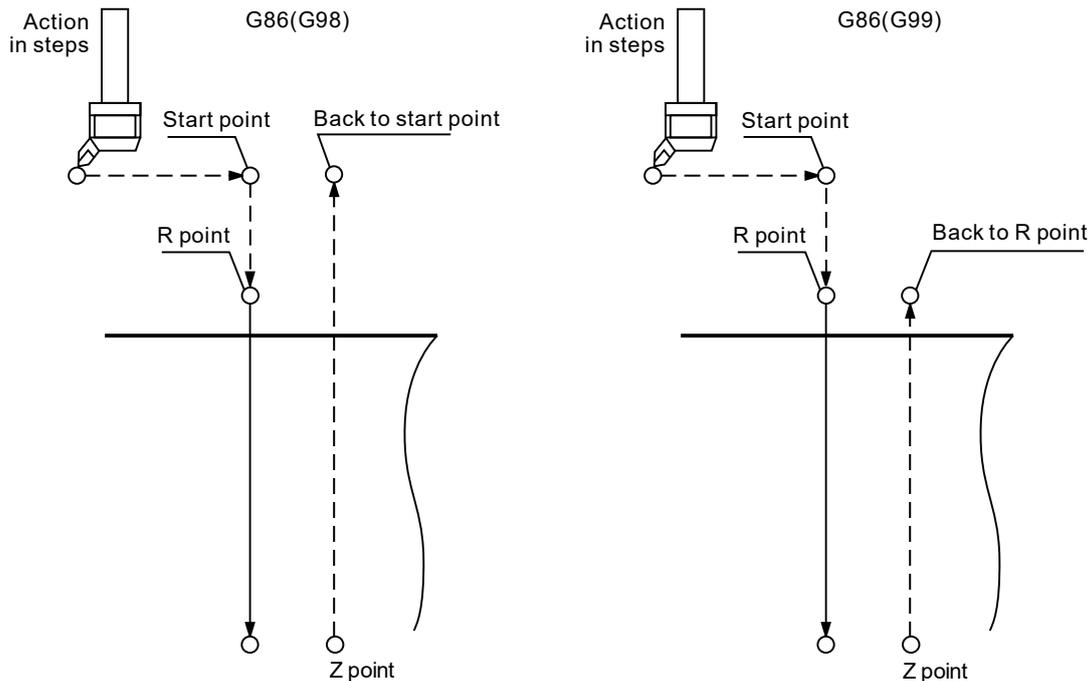
[Example]

M03 S1000

G17 G90 G00 G54 X0. Y0.

G00 Z100.

G99 G86 X0. Y0. Z-30. R10. K1 F100.   --(1)

X-15.   --(2)

X-30.   --(3)

X-30. Y15.   --(4)

G80 G91 G28 X0. Y0. Z0.

M05

2

2
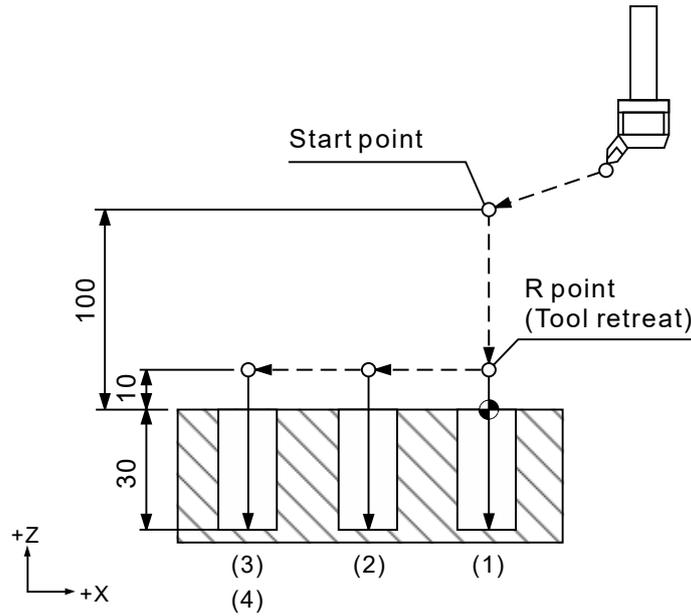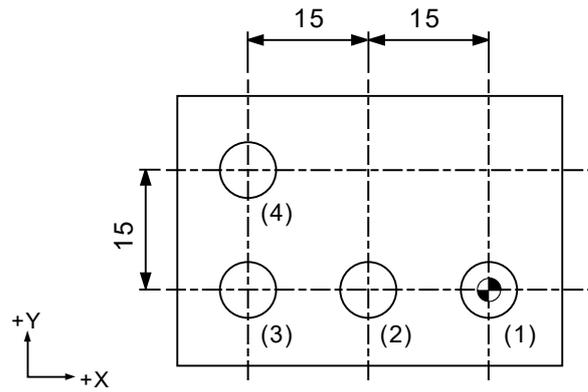
[Example]

M03 S1000
G17 G90 G00 G54 X0. Y0.
G00 Z100.
G98 G86 X0. Y0. Z-30. R10. K1 F100.   ---(1)
X-15.   ---(2)
X-30.   ---(3)
X-30. Y15.   ---(4)
G80 G91 G28 X0. Y0. Z0.
M05
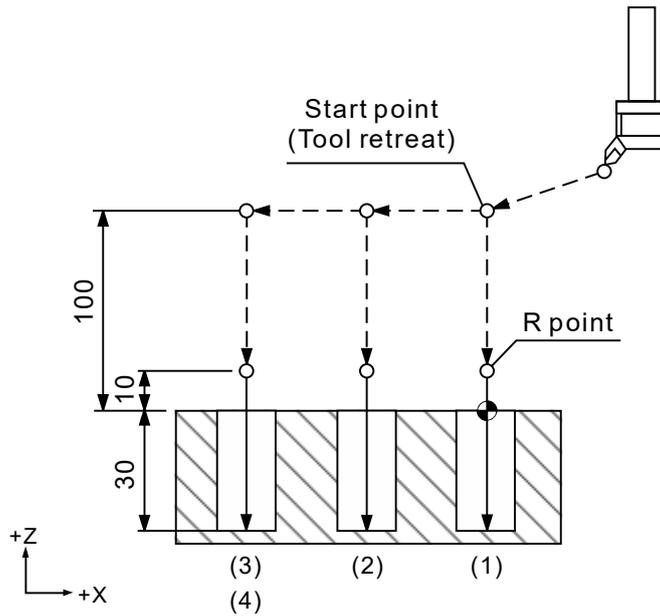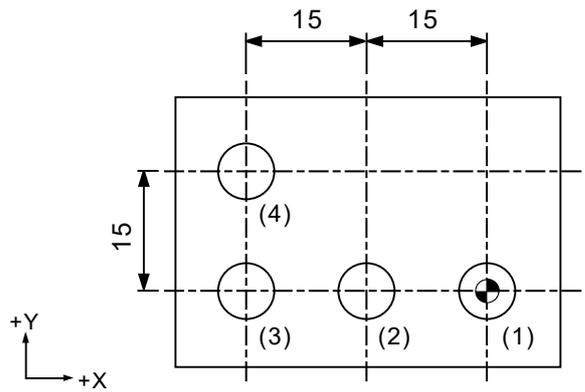
G87: Back boring cycle

Format: G87 X_ Y_ R_ Z_ Q_ P_ F_ K_

       X_ Y_: end position of a single block.

       Z_: Z-axis coordinate of the bottom of the hole.

       R_: initial point level.

       Q_: shift amount.

       P_: dwell time (unit: 1/1000 seconds). No decimals.

       F_: cutting feed rate.

       K_: number of cycle repetitions.

Description: the system has the tool move to the specified X- and Y-coordinates in rapid traverse, stops the spindle in an oriented position to align the tool tip to the fixed direction, and shifts the tool center by the shift amount specified by Q. In this case, the blade does not touch the hole wall when the tool moves to the point R level.
After moving to the point R level in rapid traverse, the tool center moves to the specified X- and Y-coordinates, and in the meantime the spindle rotates in clockwise direction. At this time, the tool starts cutting the workpiece from point R to point Z. After reaching point Z, the system shifts the tool center by the amount Q and positions the spindle, then it has the tool move back to the start point on Z axis in rapid traverse. After returning to the start point, the tool center shifts back to the original position by the shift amount Q. The value and direction of the shift amount Q is the same as that of G76. In addition, you cannot use G87 with G99 (Return to reference point after fixed cycle).

2
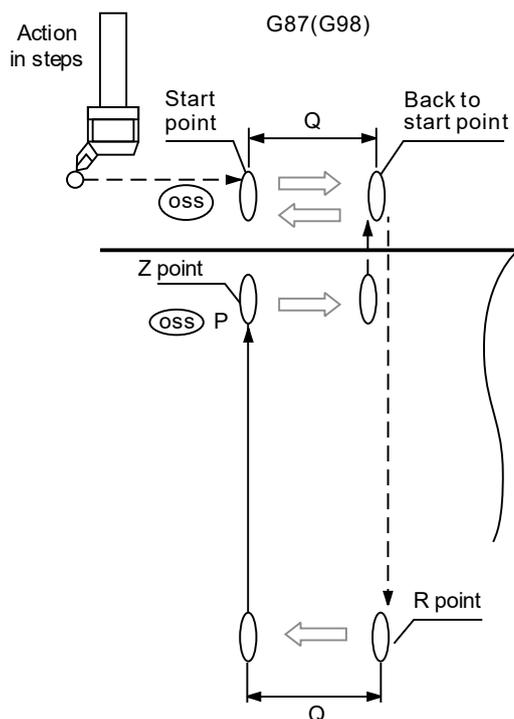
[Example]

M03 S1000

G17 G90 G00 G54 X0. Y0.

G00 Z100.

G98 G87 X0. Y0. Z-30. R10. P1000 Q5. K1 F100.   ---(1)

X-15.   ---(2)

X-30.   ---(3)

X-30. Y15.   ---(4)

G80 G91 G28 X0. Y0. Z0.

M05
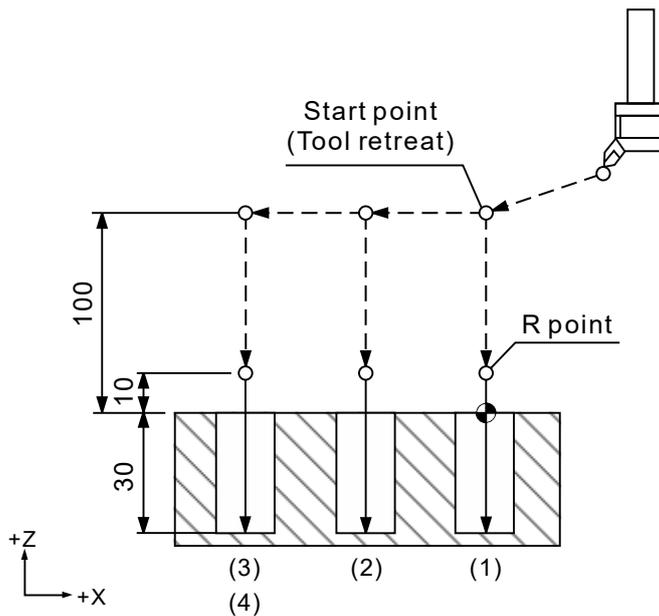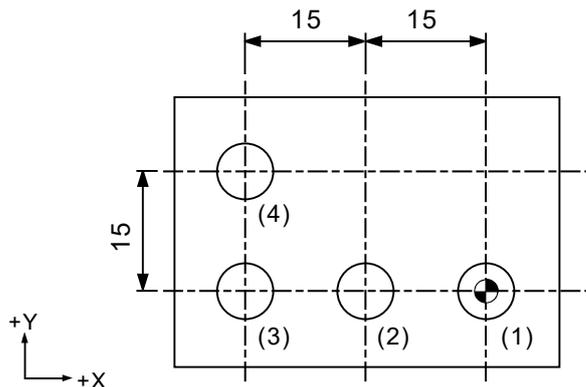
G88: Boring cycle (manual retraction)

Format: G88 X_ Y_ R_ Z_ P_ F_ K_

       X_ Y_: end position of a single block.

       Z_: Z-axis coordinate of the bottom of the hole.

       R_: initial point level.

       P_: dwell time (unit: 1/1000 seconds). No decimals.

       F_: cutting feed rate.

       K_: number of cycle repetitions.

Description: when G88 is used, the tool cuts from point R to the specified point Z and dwells for the specified time P at point Z. After that, the spindle stops rotating and the program stops (the same effect as using M00). Then, when you press **Cycle start**, the tool retracts to point R or the start point in rapid traverse. In addition, when the tool cuts to point Z, you can retract the tool in positive direction manually by setting the system to MPG mode. To resume the program, set the system to Auto mode and press **Cycle start**. At this time, the system retracts the tool to point R (G99) or the start point (G98) to complete G88 boring cycle. This command is applicable for boring blind holes.

2
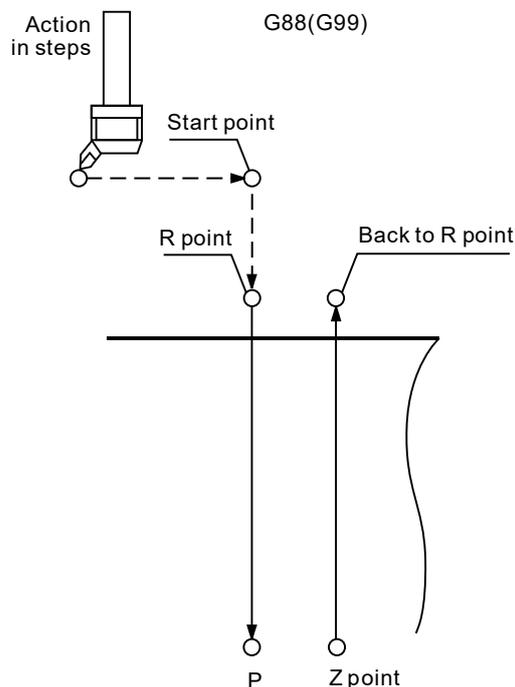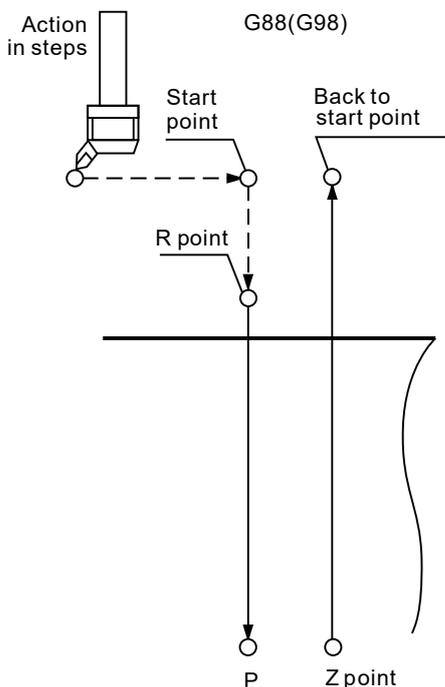
[Example]

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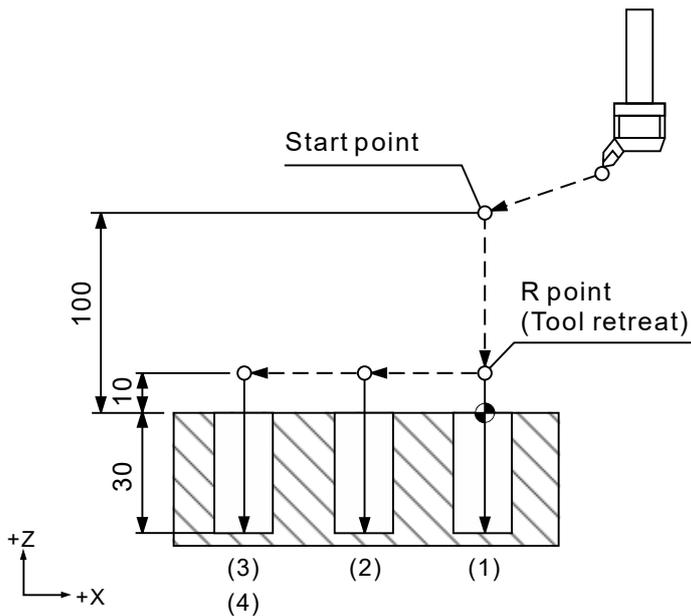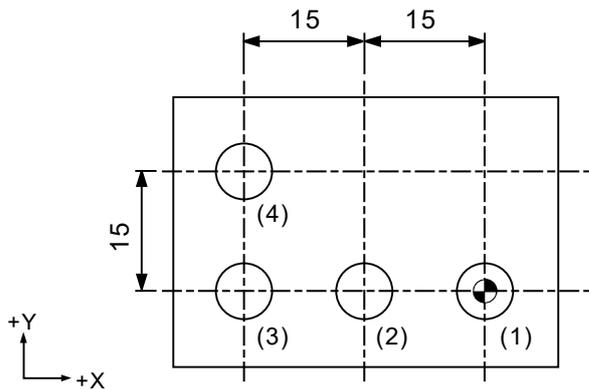
[Example]

M03 S1000

G17 G90 G00 G54 X0. Y0.

G00 Z100.

G98 G88 X0. Y0. Z-30. R10. P1000 K1 F100.   --(1)

X-15.   --(2)

X-30.   --(3)

X-30. Y15.   --(4)

G80 G91 G28 X0. Y0. Z0.

M05

2
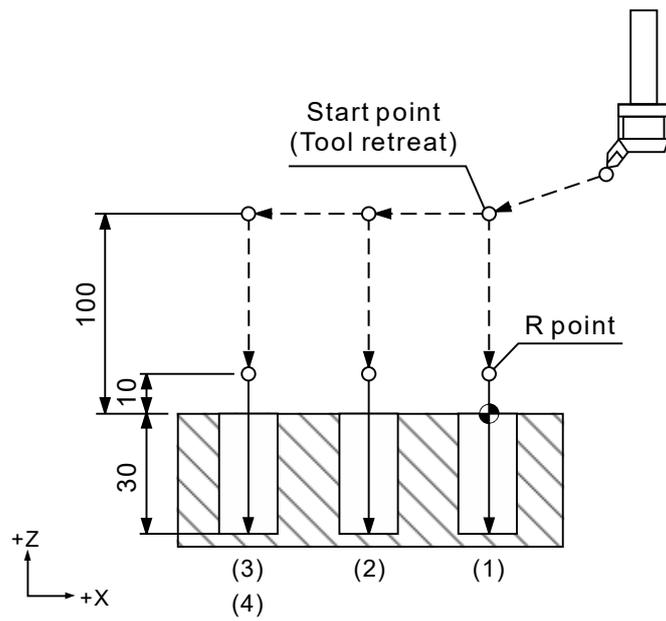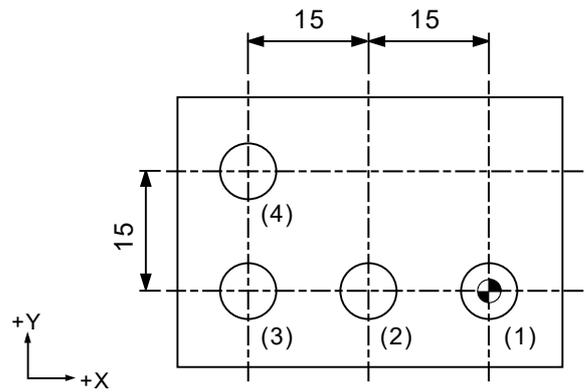
2

G89: Boring cycle (with dwell time)

Format: G89 X_ Y_ R_ Z_ P_ F_ K_

      X_ Y_: end position of a single block.

      Z_: Z-axis coordinate of the bottom of the hole.

      R_: initial point level.

      P_: dwell time (unit: 1/1000 seconds). No decimals.

      F_: cutting feed rate.

      K_: number of cycle repetitions.

Description: G89 is mainly used for reaming blind holes. Except that the tool dwells for the time specified by P at point Z, G89 and G85 function similarly by feeding and retracting the tool at the feed rate F. You can get more precise hole depth and hole diameter because you set the dwell time in the command to stop the tool after it reaches point Z.
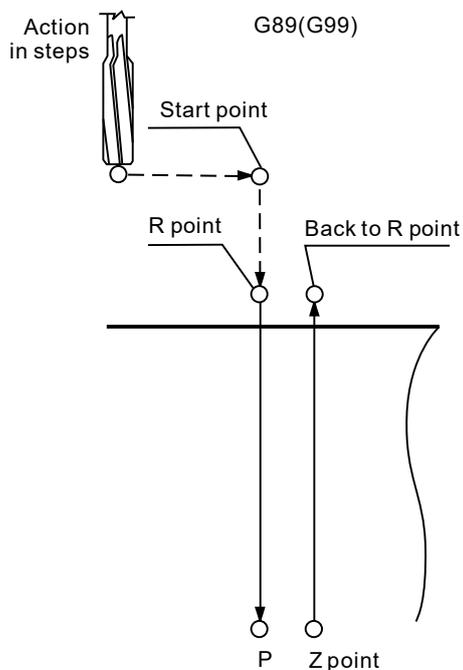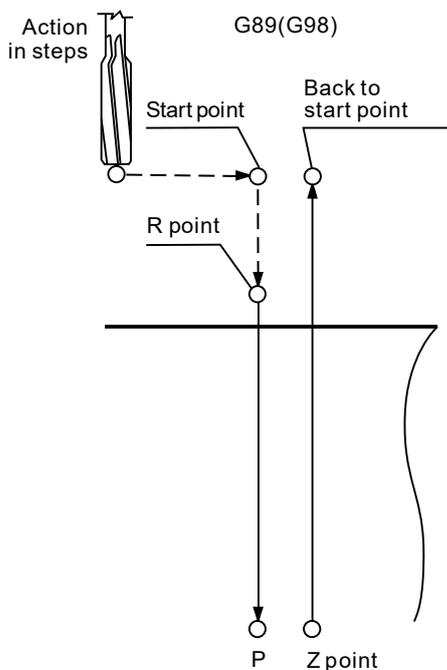
[Example]

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
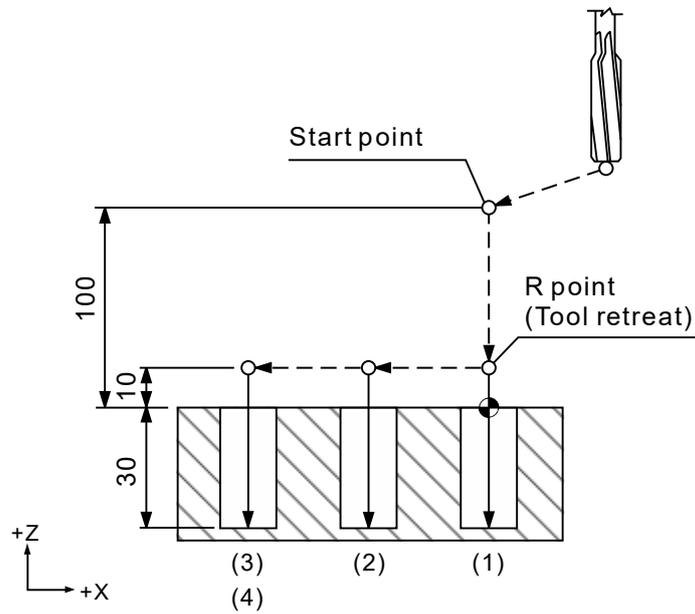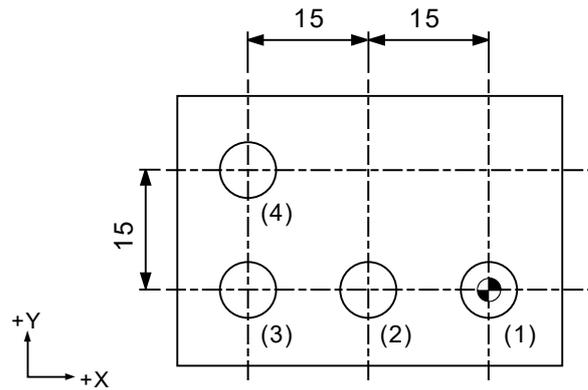
[Example]

M03 S1000

G17 G90 G00 G54 X0. Y0.

G00 Z100.

G98 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
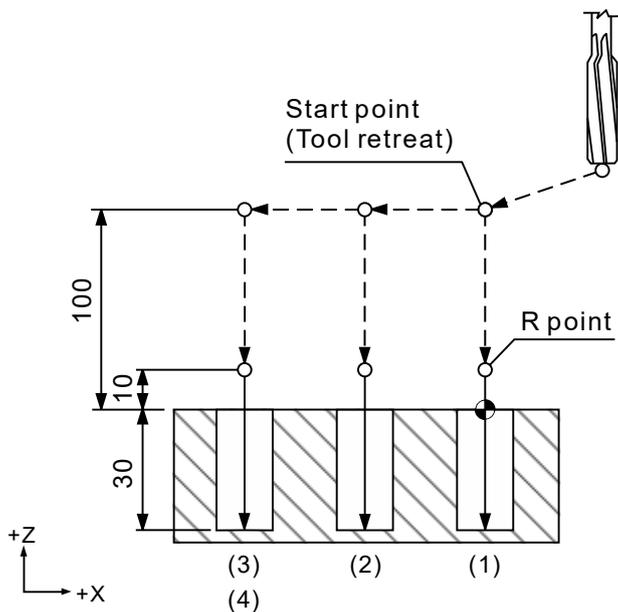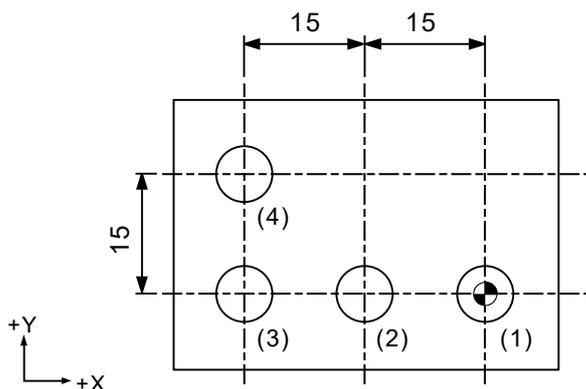
G90: Absolute coordinates

Format: G90 X_ Y_ Z_

Description: this command is continuously effective. When you use this command, it means you assign all axial commands in the absolute format. In other words, the tool moves based on the workpiece coordinate origin. After you send an axial movement command, the tool calculates the actual required moving distance according to the workpiece coordinate origin.

[Example]

When the first point is (10, 10) and the second point is (30, 30), the actual moving distance is (20, 20), as shown in the following diagram.

O1000
G01 G90 X10.0 Y10.0 F500.0
X30.0 Y30.0
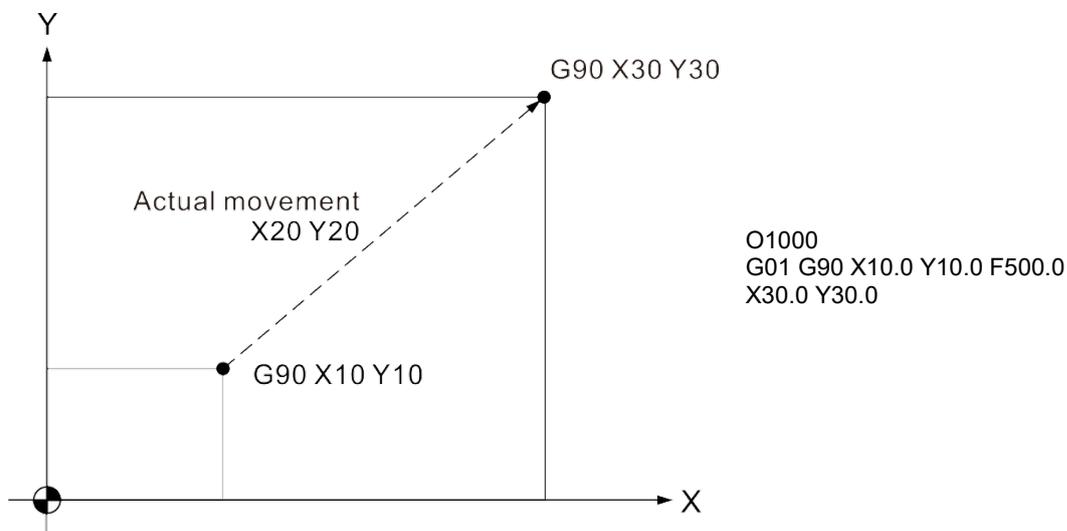
2

G91: Incremental coordinates

Format: G91 X_ Y_ Z_

Description: when you use G91, values assigned for the axial movements in the block are
incremental. The tool makes incremental movements or incremental rotations from
the current position to the specified position. G91 is a status command.
If you use G91, G91 immediately replaces G90.

[Example]

When the first point is (10, 10) and the second point is (20, 20), the actual moving distance is (20, 20), and the actual machine coordinates is (30, 30), as shown in the following diagram.

O1001
G01 G91 X10.0 Y10.0 F500.0
X20.0 Y20.0
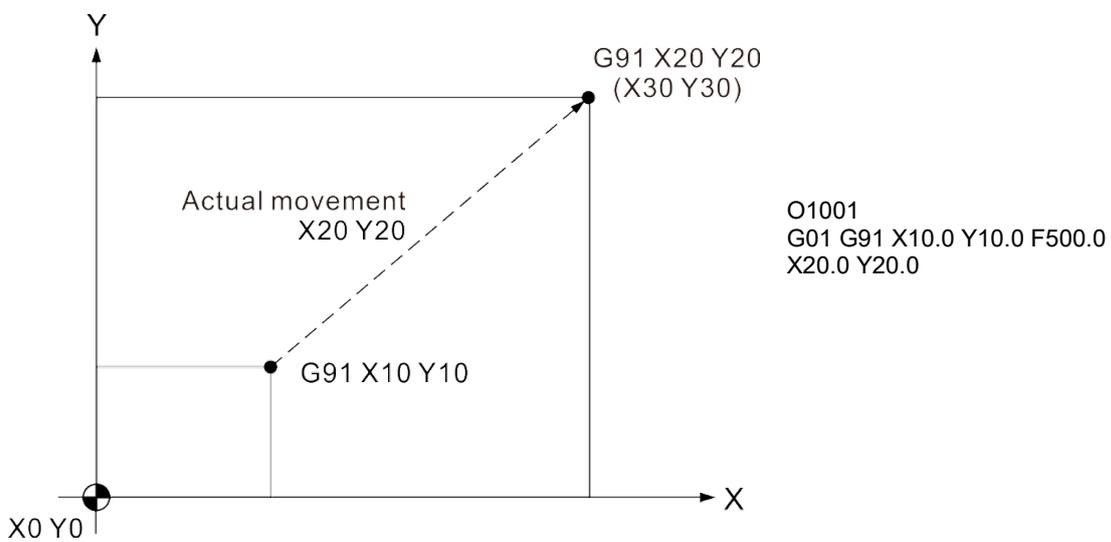
[Example of using both G90 and G91]
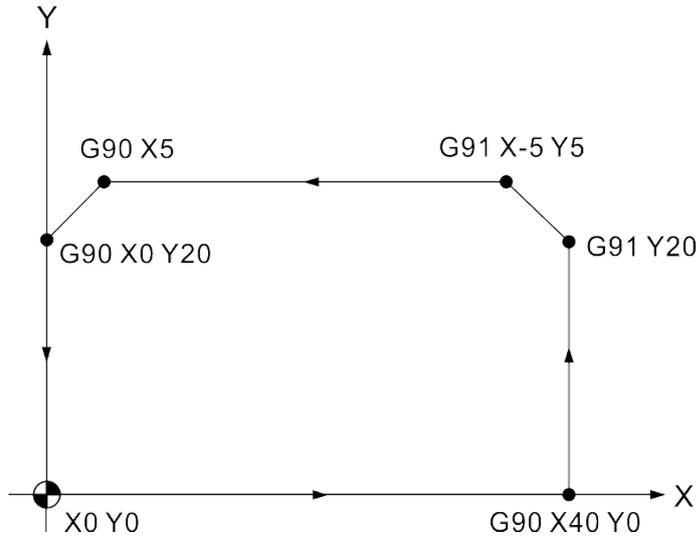

```
O0010
G01 G90 X0 Y0 F1000
X40.0
G91 Y20.0
X-5.0 Y5.0
G90 X5.0
X0 Y20.0
Y0
M30
```

2

2

G92: Coordinate system setting

Format: G92 X_ Y_ Z_

Description: G92 X0 Y0 Z0 command can set the current tool position as the absolute coordinate system origin. The absolute commands in this program refer to this origin when calculating the position. In G92 X_ Y_ Z_, if you have specified the value for X, Y, or Z, the absolute coordinates and the current position display will update according to the G92 command.

Note:

(1) G92 continues to function unless M02 or M30 is used (program end command).

(2) Pressing the **RESET** key can cancel the G92 setting.

[Example]


```
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
```
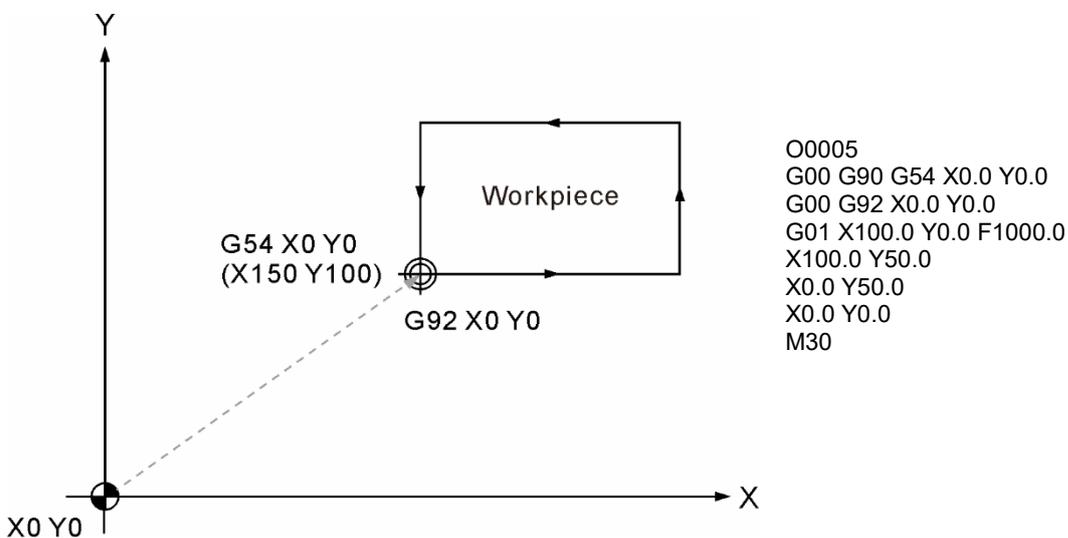
G94: Feed per minute (mm/min)

Format: G94 G01 X_ Y_ Z_ F_

Description: the unit for G94 feed per minute is mm/min. This means the tool performs cutting each minute at speed F_. You can use G94 with motion blocks at the same time and use G94 independently in one block. This command is continuously effective and its format is set based on the tool feeding calculation for the milling system.

2
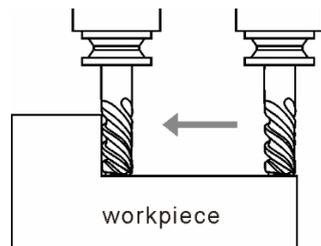
Feed per minute (mm/min or inch/min)

G98: Return to start point after fixed cycle

Format: G98 G8_ X_ Y_ Z_ R_ F_

Description: G98 and G99 are status commands used for moving the tool back to the specified height after a fixed cycle command. G98 is used for retracting the tool to the specified start point after a fixed cycle. After completing the cutting of a fixed cycle, you can use G98 to return the tool to the start point of the fixed cycle. Since G98 is a status command, you can only cancel it with G99. The default homing command is G98. See the following figure for illustration.
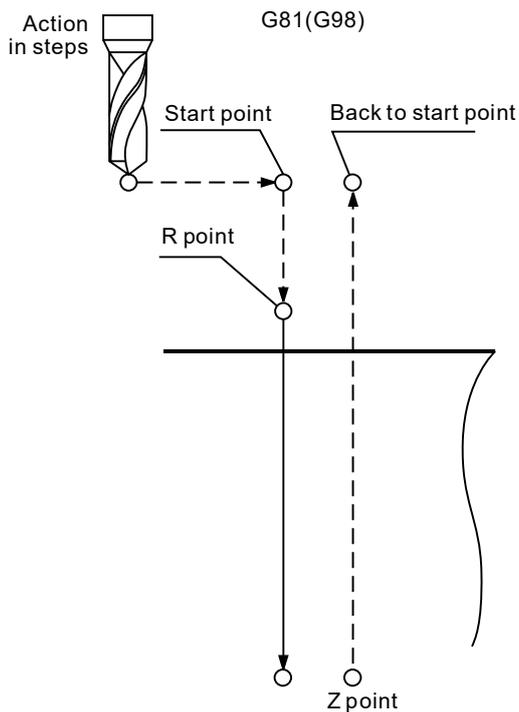
G99: Return to reference point (R) after fixed cycle

Format: G99 G8_ X_ Y_ Z_ R_ F_

Description: G98 and G99 are status commands used for moving the tool back to the specified height after a fixed cycle command. G99 is used for retracting the tool to the reference (R) point after a fixed cycle. Use G98 to cancel G99 if you desire to retract the tool to the start point.
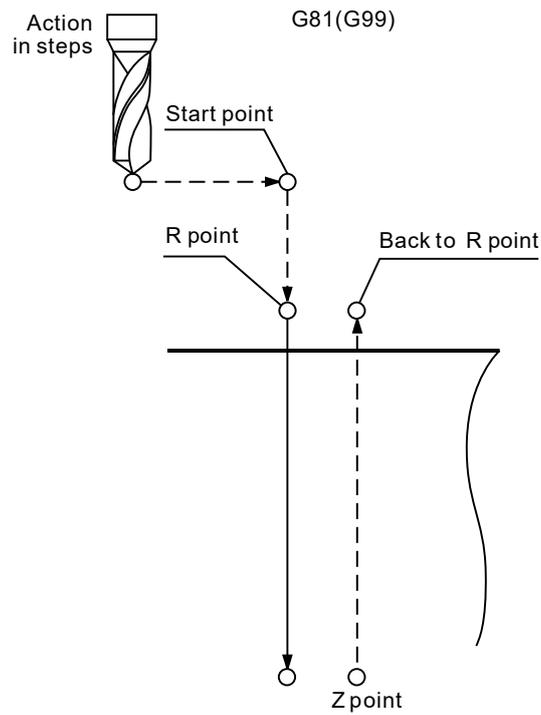
(This page is intentionally left blank.)

2

M-code Description

3

The auxiliary function M-codes are used for switching on and off the machine functions. This chapter describes the functions of commonly used M-codes. Please note that the functions of M-codes vary with individual machines.

3.1   M-code Description

The M-code format is a capital M followed by numeric digits, which range is 0 - 65534. The controller has some system-defined M-codes that do not require MLC coding to run and are usually used for program control. The following table is the commonly used M-codes. Except the system-defined codes, you need to use MLC to specify the M-code functions.

M-code	Function	Note
M00	Program stop (non-optional)	System-defined
M01	Program stop (optional)	System-defined
M02	End of program	System-defined
M03	Spindle On - clockwise	MLC
M04	Spindle On - counterclockwise	MLC
M05	Spindle stop	MLC
M06	Tool change	MLC
M08	Coolant On	MLC
M09	Coolant Off	MLC
M19	Spindle positioning	MLC
M20	Cancel spindle positioning	MLC
M29	Rigid tapping	MLC
M30	Program end with return to program start position	System-defined
M96	Program interruption	System-defined
M97	Cancel program interruption	System-defined
M98	Subprogram call	System-defined
M99	Return from subprogram	System-defined

M00: Program stop (non-optional)

Format: M00

Description: when there is an M00 command in the block, the program stops after executing this block. To resume the program execution, re-press **Cycle Start**. You can use this command to inspect the tools or workpiece appearance and dimensions when cutting.

M01: Program stop (optional)

Format: M01

Description: when M01 is executed and functioning, its function is the same as M00 except that M01 has to work with the **Optional stop** key on the secondary control panel. If the **Optional stop** key is disabled and there is an M01 command in the block, the controller automatically skips M01 and continues to run the program.

M02: End of program

Format: M02

Description: M02 is usually placed at the end of a machining program to notify the controller that the program has ended. If you place an M02 in the middle of the program, when the controller executes M02, the program stops at M02 and regards the program as ended; meanwhile, the cursor stops at the block that includes M02.

M30: End of program with return to program start position

Format: M30

Description: M30 is usually placed at the end of a machining program to notify the controller that the program has ended. If you place an M30 in the middle of the program, when the controller executes M30, the program stops at M30 and regards the program as ended; meanwhile, the cursor returns to the program start position. M30 and M02 commands are similar. The only difference is that M02 stops the cursor at the block that includes M02 whereas M30 returns the cursor to the program start position.

3

M96: Program interruption

Format: M96 P_

    M96: enable program interruption.

    P_: the subprogram code to be interrupted.

Description: when there is an M96 command in the block, the controller enables the monitoring
            and continues executing the program until the special M-code M1078 is triggered to
            have the main program stop and call the subprogram. When the controller executes
            M99 in the subprogram, it returns to the interrupted block in the main program and
            then carries on the execution.

Note:

1. You can check if the M96 command is executed with the special M-code M2216 when M1078 is
    triggered.

2. You can only trigger the interrupt signal in the main program.

3. When G02 or G03 is executed, if you trigger the interrupt signal to jump to the subprogram, the
    controller has the tool move to the endpoint of the G02 or G03 path by linear interpolation after
    executing the subprogram and then carries on the machining.

4. You cannot use M96 when the following functions are enabled:

    Polar coordinate: G16

    Mirroring: G24

    Tool radius compensation: G41 and G42

    Scaling: G51

[Example]

Main program	Subprogram
N0010	O1010
N0020 M96 P1010	N1020
N0030	N1030
N0040	N1040
N0050	N1050
N0060	N1060
N0070	N1070
N0080	N1080
N0090 M97	N1090
N0100	N1100 M99
N0110	
N0120	Subprogram
N0130	O1130
N0140 M96 P1130	N1140
N0150	N1150
N0160	N1160
N0170	N1170 M99 P0170
N0180 M97	N1180

Valid
The interrupt signal can be triggered in these blocks.

Invalid
The interrupt signal cannot be triggered in these blocks.

Valid
The interrupt signal can be triggered in these blocks.

M97: Cancel program interruption

Format: M97

Description: if you no longer need to use the M96 command, place an M97 command in the
block; when it is executed, M96 is canceled.

3

M98: Subprogram call

Format: M98 P_ H_ L_

  P_: subprogram code.

  H_: the block number to jump to.

  L_: number of call for the subprogram.

Description: if there are fixed or highly repetitive actions, you can write these actions into a
subprogram to simplify the whole program. The main program can call a
subprogram; a subprogram can call another subprogram with up to eight
consecutive layers. When the controller reads an M98 command, it jumps to the
designated subprogram and executes the subprogram commands in the specified
block repeating for the set times.

[Example]

Normally used as: M98 P_ H_ L_

```
      Main program                         Subprogram
        N0010                                O1010
        N0020  M98  P1010  H1040 L2          N1020
        N0030                                N1030
        N0040                                N1040
        N0050                                N1050
        N0060                                N1060
        N0070                                N1070
        N0080                                N1080
        N0090                                N1090
        N0100                                N1100  M99 P0140
        N0110
        N0120
        N0130
        N0140
        N0150
        N0160  M30
```
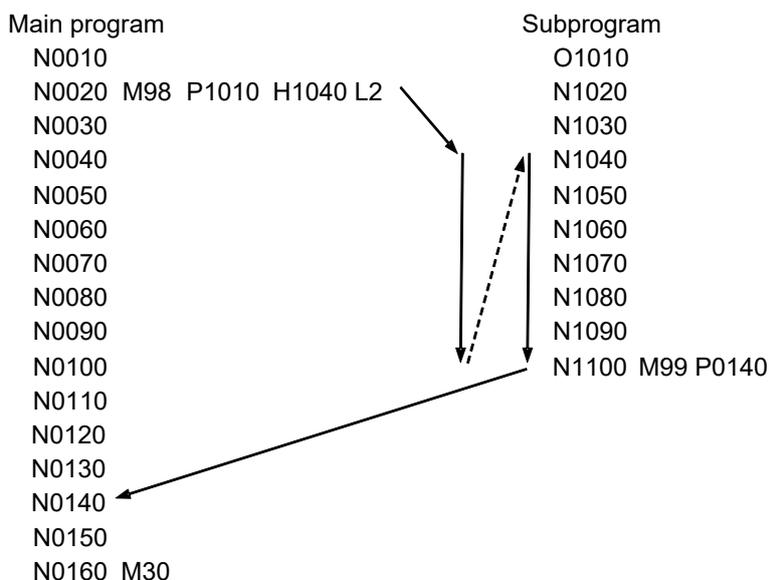
Can be used as: M98 H_ L_: when the main program and subprogram are written in the same file,

you can use the M98 command in this format in which H refers to

the block specified with the sequence number N.

```
Main program
N0010
…
…
N0050   M98   H0130 L2
…
…
N0090   M30
…
…
N0130
…
…
N0160   M99
```

M99: Return from subprogram

3
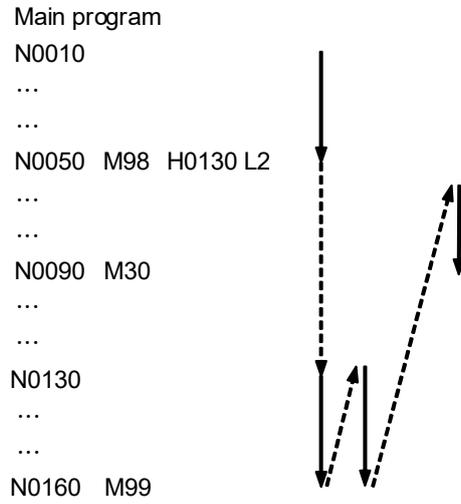
Format: M99 or M99 P_

　　　M99: return to the main program.

　　　P_: the block number to return to.

Description: after M98 is executed, to return to the main program, you need to use M99 to have the cursor return to the block following the block that has called the subprogram and then carry on the execution in the main program. You can use M99 P_ to have the cursor jump to the specified block in the main program and then carry on the execution.

[Example]

```
Main program                    Subprogram
  N0010                           O1010
  N0020 M98 P1010                 N1020
  N0030                           N1030
  N0040                           N1040
  N0050                           N1050
  N0060                           N1060
  N0070                           N1070
  N0080                           N1080
  N0090                           N1090
  N0100                           N1100 M99
  N0110
  N0120                         Subprogram
  N0130                           O1130
  N0140 M98 P1130                 N1140
  N0150                           N1150
  N0160                           N1160
  N0170                           N1170 M99 P0170
  N0180                           N1180
```

(This page is intentionally left blank.)

3

(This page is intentionally left blank.)

Macro and Variable

4
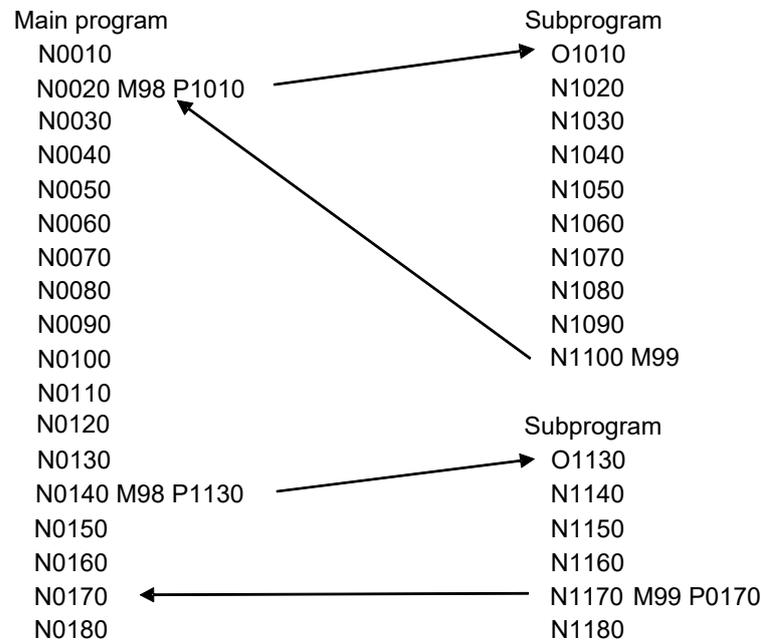
This chapter provides descriptions about the system variables and operation commands and examples of the macro syntax.

4.1 Variables

When performing variable operations in the NC program, you can use variables to replace the NC program code to batch modify the values, which makes the program editing and variable calculation easier. A variable is formatted as a symbol "#" suffixed with a variable number. Types of variables are as follows.

Variable type	No.	Function	Read	Write
Local	#1 - #50	For data storage or operation in the subprogram or macro program. The arguments can correspond to the local variables.	★	★
Global	#51 - #250	For data storage or operation in the subprogram or macro program.	★	★
Maintain (non-volatile)	#1601 - #1800	For reading and writing the system internal data during NC operation. These are non-volatile variables.	★	★
Extension (non-volatile)	#10001 - #10450	For reading and writing the system internal data during NC operation. These are non-volatile variables.	★	★
MLC bit output	#1801 - #1832	For reading the MLC signal status (MLC > NC). #1801 - #1832 for bit format and #1833 - #1848 for word format.	★	
MLC word output	#1833 - #1848		★	
MLC bit input	#1864 - #1895	For writing the MLC signal status (NC > MLC). #1864 - #1895 for bit format and #1896 - #1911 for word format.		★
MLC word input	#1896 - #1911			★

4.1.1 Arguments and local variables

Except for the G, L, N, O, and P-codes, all the other variable codes can be used as arguments. When these arguments are used for G65 and G66 subprogram call, they are 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

The system variables are used for reading and writing the system internal data during NC operations. The MLC output and input are used for the data exchange between the NC program and the MLC. The special M-codes correspond to the bit format and special D-codes correspond to the word format.

(1) G-code group data

No.	Function	Read	Write
#2000 - #2019	G-code group	★	
#2020	F-code, the NC machining speed F	★	
#2021	H-code, the NC number H for tool length offset	★	
#2022	D-code, the NC number D for tool radius offset	★	
#2023	T-code, the NC tool number T	★	
#2024	S-code, the NC spindle speed S	★	

The modal logic data for program execution (read-only)

No.	Function
#2000	G04, G09, G10, and G11
#2001	Interpolation mode: G00, G01, G02, and G03
#2002	Plane selection: G17, G18, and G19
#2003	Absolute / increment designation: G90 and G91
#2004	Stored stroke check : G22 and G23
#2005	Feed rate setting: G94
#2006	Imperial / metric setting: G20 and G21
#2007	Tool radius compensation: G40, G41, and G42
#2008	Tool length compensation: G43, G44, and G49
#2009	Cycle: G73, G74, G76, G80, G83, G84, G85, G86, G87, G88, and G89
#2010	Homing position: G98 and G99
#2011	Scaling: G50 and G51
#2012	Workpiece coordinates: G54, G55, G56, G57, G58, and G59
#2013	Cutting mode: G61 and G64
#2014	Macro call: G66 and G67
#2015	Coordinate system rotation: G68 and G69
#2016	Polar coordinates: G15 and G16

(2) Position related data

You can use variables #2100 - #2220 to read the following coordinates (read-only).

Axis / Position data	Machine coordinates / Absolute coordinates	Relative coordinates / Block end coordinates	Machine coordinates when G31 skip command is triggered / Absolute coordinates when G31 skip command is triggered	Absolute coordinates of breakpoint search line / Offset between machine coordinates of breakpoint search line and current machine coordinates
X axis	#2100	#2180	#2148	#2196
	#2116	#2132	#2164	#2212
Y axis	#2101	#2181	#2149	#2197
	#2117	#2133	#2165	#2213
Z axis	#2102	#2182	#2150	#2198
	#2118	#2134	#2166	#2214
A axis	#2103	#2183	#2151	#2199
	#2119	#2135	#2167	#2215
B axis	#2104	#2184	#2152	#2200
	#2120	#2136	#2168	#2216
C axis	#2105	#2185	#2153	#2201
	#2121	#2137	#2169	#2217
U axis	#2106	#2186	#2154	#2102
	#2122	#2138	#2170	#2218
V axis	#2107	#2187	#2155	#2203
	#2123	#2139	#2171	#2219
W axis	#2108	#2188	#2156	#2204
	#2124	#2140	#2172	#2220

(3) Workpiece coordinate data

You can use variables #3000 - #4030 to read the offset coordinates and workpiece coordinates (read-only).

Position data Axis	Offset coordinates	Workpiece coordinates G54	Workpiece coordinates G55	Workpiece coordinates G56
		Workpiece coordinates G57	Workpiece coordinates G58	Workpiece coordinates G59
X axis	#3000	#3001	#3002	#3003
		#3004	#3005	#3006
Y axis	#3128	#3129	#3130	#3131
		#3132	#3133	#3134
Z axis	#3256	#3257	#3258	#3259
		#3260	#3261	#3262
A axis	#3384	#3385	#3386	#3387
		#3388	#3389	#3390
B axis	#3512	#3513	#3514	#3515
		#3516	#3517	#3518
C axis	#3640	#3641	#3642	#3643
		#3644	#3645	#3646
U axis	#3768	#3769	#3770	#3771
		#3772	#3773	#3774
V axis	#3896	#3897	#3898	#3899
		#3900	#3901	#3902
W axis	#4024	#4025	#4026	#4027
		#4028	#4029	#4030

(4) G54 - G59 Individual offset coordinates

No.	Description	Read
#3065 - #3070	G54 - G59 X-axis individual offset coordinates	★
#3193 - #3198	G54 - G59 Y-axis individual offset coordinates	★
#3321 - #3326	G54 - G59 Z-axis individual offset coordinates	★
#3449 - #3454	G54 - G59 A-axis individual offset coordinates	★
#3577 - #3582	G54 - G59 B-axis individual offset coordinates	★
#3705 - #3710	G54 - G59 C-axis individual offset coordinates	★
#3833 - #3838	G54 - G59 U-axis individual offset coordinates	★
#3961 - #3966	G54 - G59 V-axis individual offset coordinates	★
#4089 - #4094	G54 - G59 W-axis individual offset coordinates	★

(5) Others

No.	Function	Read	Write
#2300	The variable corresponding to single block I when an arc command is executed.	★	
#2301	The variable corresponding to single block J when an arc command is executed.	★	
#2302	The variable corresponding to single block K when an arc command is executed.	★	
#2303	Timer starts after system power on	★	
#2304	Tool number of Spindle 1 (dual tool magazine)	★	
#2305	Tool number of Spindle 2 (dual tool magazine)	★	
#2500	Write the tool number for tool magazine 1		★
#2501	Write the tool number for tool magazine 2		★
#4514	The latest N-label (sequence number) when the program is executed.	★	
#5000 - #5013	Breakpoint search function: recently used M-codes (14 sets), #5000 - #5013 from the latest to the oldest	★	
#5014 - #5015	Breakpoint search function: recently used T-codes (2 sets), #5014 - #5015 from the latest to the oldest	★	
#5016	Breakpoint search function: the last used S-code	★	
#6000	System macro alarm with the value range of 1 - 1000. The messages are edited with the Screen Editor.		★
#6001 - #6064	Tool length	★	★
#6501 - #6564	Tool radius	★	★
#7001 - #7064	Tool length wear	★	★
#7501 - #7564	Tool radius wear	★	★
#8001 - #8064	Tool life	★	★
#8600	Program timer in the unit of ms	★	★

4.1.3   Macro interface input / output

You can use variables #1801 - #1911 to get the interface data as well as read and write the MLC signal status. Variable values can be in bit or word format. For the bit-type signals, the variables can only be 1 or 0; for the word-type signals, the variables can be any value.

MLC bit output; read the MLC signal status (MLC > NC)

Read MLC signal status	Macro output point	Read MLC signal 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 the MLC signal status (MLC > NC)

Read MLC signal status	Macro output register	Read MLC signal 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

4

MLC bit input; write the MLC signal status (NC > MLC)

Write MLC signal status	Macro input point	Write MLC signal 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 the MLC signal status (NC > MLC)

Write MLC signal status	Macro input register	Write MLC signal 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

The numeric values in the NC program can be replaced by variables. This empowers the program for better flexibility and universality as you can use the variables for mathematical operations.

(1) The allowable range for the local variables:

#i: the ith variable (when $1 \leqq i \leqq 50$)

(2) Define the variable number with formula:

When calculating #[A], the range of A must be $1 \leq A \leq$ System's maximum variable number.

The value of A must be a positive value and no smaller than 0.

#[<Calculation formula>]	Description
#[#20]	(Correct)
#[#20Δ3]	(Correct) when Δ = + - * /
##20	(Incorrect) There are two consecutive # symbols.
#[#20] = …	(Correct)
#20 = …	(Correct)
#[#20 - #10] = …	(Correct) Placing mathematical operators before an equal sign is not allowed.
#[- #20] = …	(Correct)

(3) Place a symbol before a variable:

- #<Variable number>	Preceding condition
Z-#20 equals Z-10.1	#20 = 10.1
G#20 equals G1	#20 = 1 (When G#20 is used, it can only be written with the program.)

(4) Use variable for definition:

#20 = 10   (#20 equals 10)

#20 = #5   (#20 equals #5)

#20 = #5 + #2   (#20 equals #5 + #2)

(5) Conditional expression:

IF[#20==1]   (If #20 equals 1, the condition is satisfied.)

4.3   Operation commands

When the system executes all kinds of calculations among variables, the operation commands can turn the calculation result into a variable, or combine and replace the result into other variables. You can use constants to replace #i, #j, and #k.

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]	Parentheses
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 angle = adjacent side #m / 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]	Square root
	POW	#i = POW [#m, #n]	#m to the power of #n
	BIT	#i = BIT [#m, #n]	Value of the #nth 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
Constant	PI	PI = π	Pi
	TRUE	TRUE = 1	Return 1 when the statement is true.
	FALSE	FALSE = 0	Return 0 when the statement is false.

4.4   Control flow

When the WHILE [statement] is true, the program executes the block that follows WHILE and repetitively executes the set of statements until it executes the first ENDW. Otherwise, the program jumps to the block of code following ENDW for execution.

```
WHILE [statement]

{

ENDW
```

Example:

```
WHILE[#80<=360.]        (Enter the loop for repetitive execution when #80 is less than or equal to 360.)


      WHILE[#60>=20.]   (Enter the inner loop when #60 is greater than or equal to 20.)
            #60=#60-2.
      ENDW              (Execute the first ENDW and the inner loop ends.)
            #80=#80+15.
            #50=#50-0.05


ENDW                    (Execute the second ENDW and the outer loop ends.)
```

Branch conditions:

When the IF [statement] is true, the program switches the execution from GOTO to statement N. Otherwise, the program executes the next block of code following the IF statement, as shown in the following example .

```
IF [Statement]   GOTO N   (Execute the IF statement, conditionally jump to statement N for execution.)

GOTO N                    (When GOTO N is used independently, unconditionally jump to statement N.)
```

The N in "GOTO N" must be used in the same program or an alarm will occur.

```
                ∫
        N10 #12=#10
        #13=#11+2;
        IF[#2=1]GOTO200;  ┐
        #12=#10-#3;       │    When #2 = 1,
        #13=#11-#4;       │    branch to N200.
        N200 X#12 Z#13;  ◄┘
        #5=#5+2;
                ∫
```
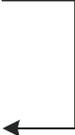
Note: when searching the sequence number in the branch, the system searches from the start of the program. An alarm will occur if the target number is not found. If the same sequence number repeats in the program, the system will execute the block of code that is first found.

Types of conditional statements:

Condition	Description	Example	
#j > #k	#j is greater than #k	#i = #j > #k	TRUE; return value: #i = 1 FALSE; return value: #i = 0
#j < #k	#j is less then #k	#i = #j < #k	TRUE; return value: #i = 1 FALSE; return value: #i = 0
#j == #k	#j is equal to #k	#i = #j == #k	TRUE; return value: #i = 1 FALSE; return value: #i = 0
#j >= #k	#j is greater than or equal to #k	#i = #j >= #k	TRUE; return value: #i = 1 FALSE; return value: #i = 0
#j <= #k	#j is less than or equal to #k	#i = #j <= #k	TRUE; return value: #i = 1 FALSE; return value: #i = 0
#j != #k	#j is not equal to #k	#i = #j != #k	TRUE; return value: #i = 1 FALSE; return value: #i = 0

Example:

#100 = 1.234;	(Define: #100 is 1.234)
#100 = #101;	(Define: #100 equals #101)
#100 = [[#101+#102]/2.0];	(Define: #100 is #101 plus #102 and then divided by 2)
#100 = #102+2.;	(Define: #100 is #102+2)
#100 = SIN[#102];	(Define: #100 is the value of SIN of #102)

X-#100	(The X-coordinate is the negative value of #100)
G1X#100Y#101;	(The X-coordinate is #100; the Y-coordinate is #101)
G1X[#100];	(The X-coordinate is #100)
G1X[#100+#101];	(The X-coordinate is #100 + #101)
G2X[#100*SIN[#102]];	(The X-coordinate is #100 multiplied by SIN[#102])
G1Z#100F#102S#103;	(The Z-coordinate is #100; F is #102; S is #103)

4.5   Use M-code, S-code, and T-code to call macro

(1) To use the G-code to call a macro, go to [CONFIG] to set the number to be called. The corresponding setting is as follows.

Macro function	G-code number	Note
O9010	0 - 1000	
O9011	0 - 1000	
O9012	0 - 1000	
O9013	0 - 1000	
O9014	0 - 1000	Set the G-code number to 0 if not using the macro call function.
O9015	0 - 1000	
O9016	0 - 1000	
O9017	0 - 1000	
O9018	0 - 1000	
O9019	0 - 1000	

Constraint: if you use a macro to call G-codes, M-codes, or T-codes, the nested macro will not be executed (the G-codes in this macro are treated as general G-codes).

(2) To use the M-code to call a macro, go to [CONFIG] to set the number to be called. The corresponding setting is as follows.

Macro function	M-code number	Note
O9020	0 - 1000	
O9021	0 - 1000	
O9022	0 - 1000	
O9023	0 - 1000	
O9024	0 - 1000	Set the M-code number to 0 if not using the macro call function.
O9025	0 - 1000	
O9026	0 - 1000	
O9027	0 - 1000	
O9028	0 - 1000	
O9029	0 - 1000	

Constraint: if you use a macro to call G-codes, M-codes, or T-codes, the nested macro will not be executed (the M-codes in this macro are treated as general M-codes).

4

(3) To use the T-code to call a macro, go to [CONFIG] to enable the function. The setting is as follows.

Macro function	T-code number	Note
O9000	0: disabled else: enabled	Set the T-code number to 0 if not using the macro call function. The T-code number is defined as local variable #20.

Constraint: if you use a macro to call G-codes, M-codes, or T-codes, the nested macro will not be executed (the T-codes in this macro are treated as general T-codes).

The variable definition is as follows.

No.	Description	Read	Write
#1 - #50	Local variables	★	★
#51 - #250	Global variables	★	★
#1601 - #1800	Maintain variables (non-volatile)	★	★
#10001 - #10450	Extension variables (non-volatile)	★	★
#1801 - #1832	MLC logic output points. MLC > NC macro input points: M1024 - M1055 (32 points in total)	★	
#1833 - #1848	MLC data output points. MLC > NC macro input points: D1024 - D1039 (16 points in total)	★	
#1864 - #1895	MLC logic input points. NC > MLC macro output points: M2080 - M2111 (32 points in total)		★
#1896 - #1911	MLC data input points. NC > MLC macro output points: D1336 - D1351 (16 points in total)		★
#2000 - #2019	G-code group	★	
#2020	F-code	★	
#2021	H-code	★	
#2022	D-code	★	
#2023	T-code	★	
#2024	S-code	★	
#2100 - #2108	Machine coordinates of X - W axes	★	
#2116 - #2124	Absolute coordinates of X - W axes	★	
#2132 - #2140	End coordinates of X - W axes in single block	★	
#2148 - #2156	Machine coordinates of X - W axes when G31 skip command is triggered	★	
#2164 - #2172	Absolute coordinates of X - W axes when G31 skip command is triggered	★	
#2180 - #2188	Relative coordinates of X - W axes	★	
#2196 - #2204	Absolute coordinates of breakpoint search line (X - W axes)	★	
#2212 - #2220	Offset between machine coordinates of breakpoint search line and current machine coordinates (X - W axes)	★	
#2300	The variable corresponding to single block I when an arc command is executed.	★	

4

No.	Description	Read	Write
#2301	The variable corresponding to single block J when an arc command is executed.	★	
#2302	The variable corresponding to single block K when an arc command is executed.	★	
#2303	Timer starts after system power on	★	
#2304	Tool number of Spindle 1 (dual tool magazine)	★	
#2305	Tool number of Spindle 2 (dual tool magazine)	★	
#2500	Write the tool number for tool magazine 1		★
#2501	Write the tool number for tool magazine 2		★
#3000	X-axis offset coordinate	★	
#3001 - #3006	X-axis workpiece coordinate G54 - G59	★	
#3007 - #3070	X-axis workpiece coordinate G54P1 - G54P64	★	
#3128	Y-axis offset coordinate	★	
#3129 - #3134	Y-axis workpiece coordinate G54 - G59	★	
#3135 - #3198	Y-axis workpiece coordinate G54P1 - G54P64	★	
#3256	Z-axis offset coordinate	★	
#3257 - #3262	Z-axis workpiece coordinate G54 - G59	★	
#3263 - #3326	Z-axis workpiece coordinate G54P1 - G54P64	★	
#3384	A-axis offset coordinate	★	
#3385 - #3390	A-axis workpiece coordinate G54 - G59	★	
#3391 - #3454	A-axis workpiece coordinate G54P1 - G54P64	★	
#3512	B-axis offset coordinate	★	
#3513 - #3518	B-axis workpiece coordinate G54 - G59	★	
#3519 - #3582	B-axis workpiece coordinate G54P1 - G54P64	★	
#3640	C-axis offset coordinate	★	
#3641 - #3646	C-axis workpiece coordinate G54 - G59	★	
#3647 - #3710	C-axis workpiece coordinate G54P1 - G54P64	★	
#3768	U-axis offset coordinate	★	
#3769 - #3774	U-axis workpiece coordinate G54 - G59	★	
#3775 - #3838	U-axis workpiece coordinate G54P1 - G54P64	★	
#3896	V-axis offset coordinate	★	
#3897 - #3902	V-axis workpiece coordinate G54 - G59	★	
#3903 - #3966	V-axis workpiece coordinate G54P1 - G54P64	★	
#4024	W-axis offset coordinate	★	
#4025 - #4030	W-axis workpiece coordinate G54 - G59	★	
#4031 - #4094	W-axis workpiece coordinate G54P1 - G54P64	★	
#5000 - #5013	Breakpoint search function: recently used M-codes (14 sets), #5000 - #5013 from the latest to the oldest	★	

No.	Description	Read	Write
#5014 and #5015	Breakpoint search function: recently used T-codes (2 sets), #5014 - #5015 from the latest to the oldest	★	
#5016	Breakpoint search function: the last used S-code	★	
#6000	System macro alarm		★
#6001 - #6064	Tool length	★	★
#6501 - #6564	Tool radius	★	★
#7001 - #7064	Tool length wear	★	★
#7501 - #7564	Tool radius wear	★	★
#8001 - #8064	Tool life	★	★
#8600	Program timer in the unit of ms	★	★

Revision History

Release date	Version	Chapter	Revision contents
March, 2019	V1.0 (First edition)		

For relevant information about [Milling Machine Solution G Command Guidelines], please refer to:

(1) Delta CNC Milling Machine Solution - Operation and Maintenance Manual
(2) Delta CNC Solution – NC Series MLC Application Manual

(This page is intentionally left blank.)