Smarter. Greener. Together.

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

Asia
Delta Electronics (Jiangsu) Ltd.
Wujiang Plant 3
1688 Jiangxing East Road,
Wujiang Economic Development Zone
Wujiang City, Jiang Su Province,
People's Republic of China (Post code: 215200)
TEL: 86-512-6340-3008 / FAX: 86-769-6340-7290

Delta Greentech (China) Co., Ltd.
238 Min-Xia Road, Pudong District,
ShangHai, P.R.C.
Post code : 201209
TEL: 86-21-58635678 / FAX: 86-21-58630003

Delta Electronics (Japan), Inc.
Tokyo Office
2-1-14 Minato-ku Shibadaimon,
Tokyo 105-0012, Japan
TEL: 81-3-5733-1111 / FAX: 81-3-5733-1211

Delta Electronics (Korea), Inc.
1511, Byucksan Digital Valley 6-cha, Gasan-dong,
Geumcheon-gu, Seoul, Korea, 153-704
TEL: 82-2-515-5303 / FAX: 82-2-515-5302

Delta Electronics Int'l (S) Pte Ltd
4 Kaki Bukit Ave 1, #05-05, 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

Americas
Delta Products Corporation (USA)
Raleigh Office
P.O. Box 12173,5101 Davis Drive,
Research Triangle Park, NC 27709, U.S.A.
TEL: 1-919-767-3800 / FAX: 1-919-767-8080

Delta Greentech (Brasil) S.A
Sao Paulo Office
Rua Itapeva, 26 - 3° andar Edificio Itapeva One-Bela Vista
01332-000-São Paulo-SP-Brazil
TEL: +55 11 3568-3855 / FAX: +55 11 3568-3865

Europe
Deltronics (The Netherlands) B.V.
Eindhoven Office
De Witbogt 15, 5652 AG Eindhoven, The Netherlands
TEL: 31-40-2592850 / FAX: 31-40-2592851

Delta CNC Solution

Delta CNC Solution

NC Series Command Guidelines

www.delta.com.tw/ia

Smarter. Greener. Together.

Preface

Thank you for choosing this product. Before using the product, please read through this manual carefully in order to ensure the correct use of the product. In addition, please place the manual safely for quick reference whenever is needed.

This manual includes

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

Product features

- Built-in 32-bit high-speed dual CPU for multi-task execution and performance improvement
- Friendly HMI Interface
- Servo Gain Auto-tuning Interface for different mechanism specifications
- CNC Soft 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 the user to choose from: communication type or analog voltage type
- Serial I/O modules for flexible I/O configuration

How to use this manual

This manual can be used as reference while studying NC controllers, which provides information about how to use G code, macro and variable syntax. Before using and setting your NC controller, please read through this manual carefully.

DELTA technical services

Please consult the distributors or DELTA customer service center if any problem occurs.

Safety Precautions

- Please follow the instruction of pin assignment when wiring. Ground is a must.
- When the power is being supplied, do not disconnect the controller, change the wiring or touch the power source to avoid electric shock.

Please pay close attention to the following safety precautions during inspecting, installation, operating, maintenance and troubleshooting.

The symbols of “**DANGER**”, “**WARNING**” and “**STOP**” represent:

DANGER	**It indicates the potential hazards. It is possible to cause severe injury or fatal harm if not follow the instructions.**
WARNING	**It indicates the potential hazards. It is possible to cause minor injury or lead to serious damage of the product or even malfunction if not follow the instructions.**
STOP	**It indicates the absolute prohibited activity. It is possible to damage the product or cannot be used due to malfunction if not follow the instructions.**

Installation

DANGER

- Please follow the installation instructions in this manual; otherwise it may cause damage to the equipment.
- It is prohibited to expose the product to the environment containing water, corrosive gas, inflammable gas etc. Otherwise, electric shock or fire may occur.

Wiring

DANGER

- Please connect the ground terminal to class-3 ground system (under 100 Ω). Poor grounding may result in electric shock or fire.

Operation

WARNING

- Correctly plan out the I/O actions with MLC Editor Software, or abnormal operation may occur.
- Before operation, please properly adjust the parameter settings of the machine, otherwise it may cause abnormal operation.
- Please ensure the emergency stop can be activated at any time, and avoid operating the machine in unprotected condition.

- Do not modify wiring while power is being supplied. Otherwise, it may cause personal injury due to electric shock.
- Never use a sharp-pointed object to touch the panel, as doing this might dent the screen and lead to malfunction of the controller.

Maintenance and Inspection

- While power is being supplied, do not disassemble the controller panel or touch the internal parts, otherwise electric shock may occur.
- Do not touch the ground terminal within 10 minutes after turning off the power, as the residual voltage may cause electric shock.
- Turn OFF the power first before replacing backup battery, and recheck the system settings afterwards.
- Do not block the vent holes during operation, as malfunction may easily occur due to poor ventilation.

Wiring Method

WARNING

- Power supply: In order to avoid danger, use a 24 VDC power supply for the controller and comply with the wire specification when wiring.
- Wiring materials: Use multi-stranded twisted-pair wires or multi-core shielded-pair wires to isolate all cables.
- The maximum cable length for remote I/O signals and DMCNET communication is 20 m and the maximum cable length for other signal cable is 10 m.
- To control the input and output signals, a 24 VDC power is required for the controller I/O and remote I/O.

Wiring of Communication Circuit

WARNING

- DMCNET wiring: The wiring materials should be in compliance with the standard specification.
- Please make sure the wiring between the controller and servo drive is tight and secure, as loose cables may cause abnormal operation.

Note: The content may be revised without prior notice. For the latest version, please visit Delta's website at http://www.delta.com.tw/industrialautomation/.

(This page is intentionally left blank.)

Table of Contents

1 Table of G codes

1.1 Table of G code for the milling system 1-2
1.2 Table of G code for the turning system 1-4

2 G Code Description

2.1 G codes for the milling system 2-2
2.2 G codes for the turning system 2-86

3 M Code Description

3.1 M code description 3-2

4 Macro and Variables

4.1 Variables 4-2
4.1.1 Arguments and local variables 4-3
4.1.2 System variables 4-3
4.1.3 Macro interface input and output 4-6
4.2 Variable syntax 4-8
4.3 Operational commands 4-9
4.4 Control flow 4-10
4.5 Macro call 4-12

(This page is intentionally left blank.)

1 Table of G Codes

This chapter lists the G codes supported by NC controller. You can have a quick overview of all G code function and their groups.

1.1 G code table of the milling system 1-2

1.2 G code table of the turning system 1-4

1

1.1 G code table of the milling system

G code	Group	Function description
G00	01	Fast positioning
G01	01	Linear interpolation
G02	01	Clockwise arc interpolation (CW)
G03	01	Counterclockwise arc interpolation (CCW)
G04	00	Dwell time
G09	00	Exact stop
G10	00	Data entry setup
G11	00	Data entry setup cancelling
G15	16	Polar coordinates cancelling
G16	16	Polar coordinates
G17	02	X-Y plane selection
G18	02	Z-X plane selection
G19	02	Y-Z plane selection
G20	06	Inch input
G21	06	Metric input
G24	17	Mirror image setup
G25	17	Mirror image setup cancelling
G28	00	Homing through the reference point
G29	00	Homing through start point
G30	00	Homing of the second, third, and fourth reference point
G31	00	Skip function
G40	07	Tool radius compensation cancelling
G41	07	Tool radius left compensation
G42	07	Tool radius right compensation
G43	08	Tool length positive direction compensation
G44	08	Tool length negative direction compensation
G49	08	Tool length compensation cancelling
G50	11	Scale cutting cancelling
G51	11	Scale cutting
G52	00	Local coordinate system setup
G53	00	Mechanical coordinate system setup
G54	12	The first machining coordinate system selection
G55	12	The second machining coordinate system selection

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

1

1.2 G code table of the turning system

The G codes for the turning system can be categorized into three types, A, B, and C. You can select the type via operation parameter 306 according to the requirement. Set the parameter value to 0 for type A, 1 for type B, and 2 for type C. The description in this manual is written based on G codes of type A.

Type			Group	Function description
A	B	C		
G00	G00	G00	01	Fast positioning command
G01	G01	G01	01	Linear interpolation command
G02	G02	G02	01	Clockwise arc interpolation (CW)
G03	G03	G03	01	Counterclockwise arc interpolation (CCW)
G04	G04	G04	00	Dwell time
G09	G09	G09	00	Exact stop
G10	G10	G10	00	Data entry setup
G11	G11	G11	00	Data entry setup cancelling
G17	G17	G17	02	X-Y plane selection
G18	G18	G18	02	Z-X plane selection
G19	G19	G19	02	Y-Z plane selection
G20	G20	G70	06	Inch input
G21	G21	G71	06	Metric input
G28	G28	G28	00	Homing through reference point
G29	G29	G29	00	Homing through start point
G30	G30	G30	00	Homing to the second, third, and fourth reference point
G31	G31	G31	00	Skip function
G32	G33	G33	01	Thread cutting
G34	G34	G34	01	Variable lead threading
G40	G40	G40	07	Cancel right/left compensation of tool nose radius
G41	G41	G41	07	Left compensation of tool nose radius
G42	G42	G42	07	Right compensation of tool nose radius
G50	G92	G92	00	Coordinate system setting / Max. spindle speed
G52	G52	G52	00	Local coordinate system setup
G53	G53	G53	00	Mechanical coordinate system setup
G54	G54	G54	12	The first machining coordinate system selection
G55	G55	G55	12	The second machining coordinate system selection
G56	G56	G56	12	The third machining coordinate system selection

Type			Group	Function description
A	B	C		
G57	G57	G57	12	The fourth machining coordinate system selection
G58	G58	G58	12	The fifth machining coordinate system selection
G59	G59	G59	12	The sixth machining coordinate system selection
G61	G61	G61	13	Exact stop mode
G64	G64	G64	13	Cutting mode
G65	G65	G65	00	Non-continuous effect macro command calling
G66	G66	G66	14	Continuous effect macro command calling
G67	G67	G67	14	Continuous effect macro command calling cancelling
G70	G70	G72	09	Multiple type finish turning cycle
G71	G71	G73	09	Multiple type rough turning cycle (outer diameter)
G72	G72	G74	09	Multiple type rough facing cycle
G73	G73	G75	09	Multiple type pattern repeating cycle
G74	G74	G76	09	Multiple type face pecking cycle
G75	G75	G77	09	Multiple type axial pecking cycle
G76	G76	G78	09	Multiple type thread turning cycle
G90	G77	G20	09	Axial turning cycle
G92	G78	G21	09	Threading cycle
G94	G79	G24	09	End face turning cycle
G80	G80	G80	09	Cycle cancel
G83	G83	G83	09	Face peck drilling cycle
G84	G84	G84	09	Face tapping cycle
G85	G85	G85	09	Face boring cycle
--	G90	G90	03	Absolute coordinates
--	G91	G91	03	Incremental coordinates
G98	G94	G94	05	Feeding amount by minute (mm/min)
G99	G95	G95	05	Feeding amount by revolution (mm/rev)
G96	G96	G96	17	Constant surface speed control (cycle mode)
G97	G97	G97	17	Cancel constant surface speed control (cycle mode)

(This page is intentionally left blank.)

G Codes Description 2

This chapter introduces the format of the G Codes commands supported by NC series controllers along with description of the application examples. Users can learn more about G Code through this chapter.

2.1 G codes for the milling system ··· 2-2
2.2 G codes for the turning system ··· 2-86

2

2.1 G Codes for the milling system

G00: Fast positioning command

Format: G00 X_Y_Z_ (This command I applicable to three-axis, two-axis synchronous control, and single-axis control.)
X_Y_Z_: Coordinates of the end point

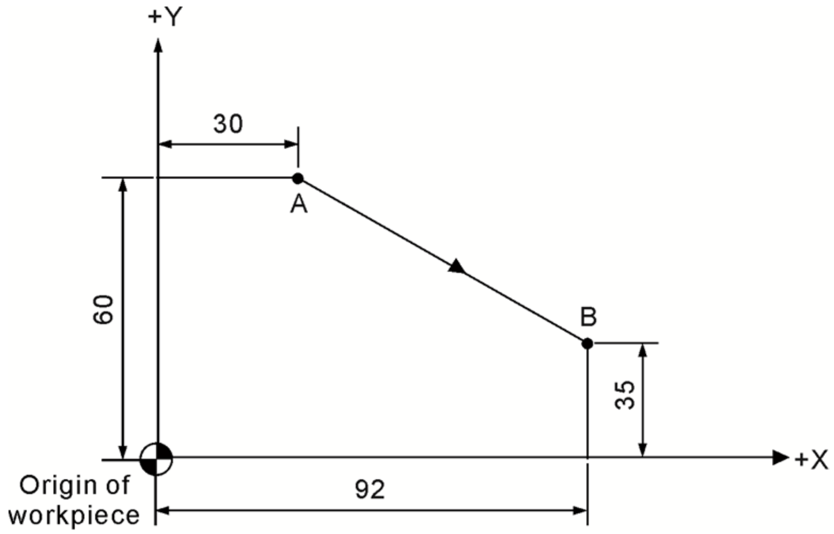
Description: The G00 command quickly moves the center of the tool to the specified coordinate position (X, Y, Z). When G00 is applied, the moving speed is not specified by value F_ in the command; instead, it is controlled by the **Rapid %** key on the secondary control panel.
If the max. moving speed of X-, Y-, and Z-axis (parameter 316) is set to 15 m/min,
(1) When the rapid feed rate is set to 100%, the moving speed will be 15 m/min.
(2) When the rapid feed rate is set to 50%, the moving speed will be 7.5 m/min.
(3) When the rapid feed rate is set to 25%, the moving speed will be 3.75 m/min.
(4) When the rapid feed rate is set to 0%, the axial moving speed will be the speed set by parameter 315.

G00 is mainly for fast positioning instead of cut feeding. It can be used in the circumstances such as positioning at start cutting point from the mechanical origin in rapid traverse or application for retraction and fast positioning of Z- and X-axis after cutting.

[Example]

The following figure illustrates the usage of G00. The tool moves from A and positioned at B in rapid traverse.

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



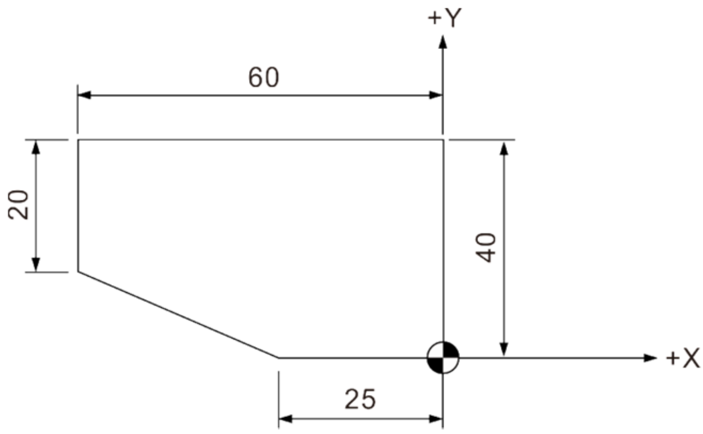
G01: Linear interpolation command

Format: G01 X_Y_Z_F_
X_Y_Z_: Coordinates of the end point
F_: Feed rate in unit of mm/min

Description: This command enables a cutter to make linear cutting from the current position to a given position at F feed rate. X, Y, Z coordinates stands for the end point of cutting. The movement can be three-axis synchronous, two-axis synchronous or single axis movement.The feed rate is set by the F parameter in unit of mm/min along with the **Rapid %** key of the secondary control panel.

[Example]

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




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

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

2

G02/G03: Arc interpolation command

Format:

Arcs in the X – Y plane
G17 G02 (or G03) X_ Y_ R_ F_ or
G17 G02 (or G03) X_ Y_ I_ J_ F_
You may add parameter Z_ in the command to allow the tool to move spirally in the X - Y plane.

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

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

G02: Clockwise (CW) arc interpolation.
G03: Counterclockwise (CCW) arc interpolation.

X, Y, and Z: Coordinates of the end point in absolute or incremental values determined by command G90 and G91.

R: Arc radius (expressed in Radius format).

I: Distance between center and starting point in X-axis direction. That is, the increments from the starting point to center.

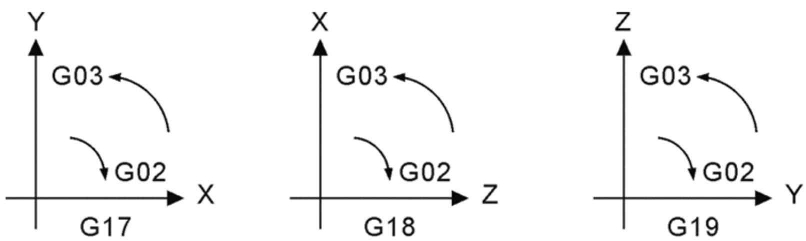
J: Distance between arc center and starting point in Y-axis direction. That is, the increments from the starting point to center.

K: Distance between arc center and starting point in Z-axis direction. That is, the increments from the starting point to center.

(The I, J, and K expression is also known as the center method.)

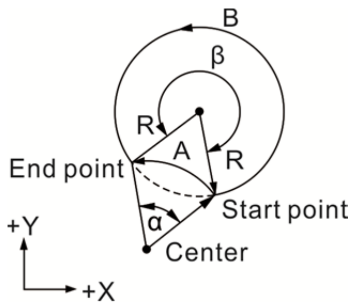
F: The cut feeding rate in unit of mm/min.

Description: G02 or G03 is the arc interpolation command. The arc interpolation direction (G02 or G03) for a three-dimensional workpiece in individual planes is shown in the figure below. G02 is for clockwise direction while G03 is for counterclockwise.



The center and radius methods are described below:

1. Radius method: R is the radius of the arc and shows in radius value. In this method, arc is made according to the start point, end point, and the arc radius. Thus, there will be two arc segments generated. See the figure below. For the positive R value, it is an arc of central angle ≤ 180° and an arc of central angle > 180° for the negative R value.



The radius method

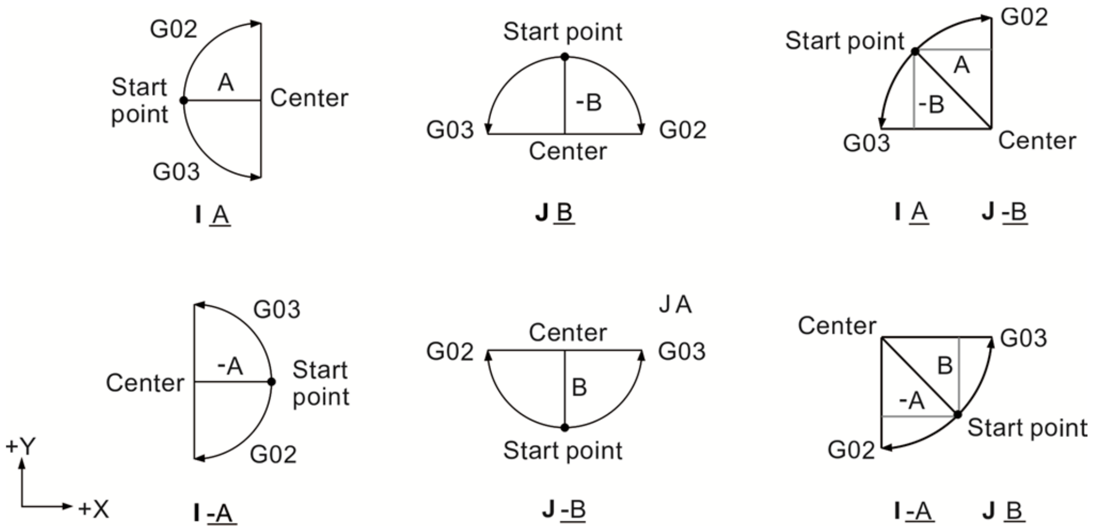
Note: β > 180° -> the B arc with negative R; α ≤ 180° -> the A arc with positive R

See the above figure. Assume that R = 50 mm and coordinates of the end point is (100.0 , 80.0), then:

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

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

2. The Center method: Parameters I, J, and K define the relative distance from the start point to the center of an arc. That is, it is the increments from the start point to the center in X, Y, and Z axis directions respectively. See the figure below for illustrations.



Values of parameters I and J on plane

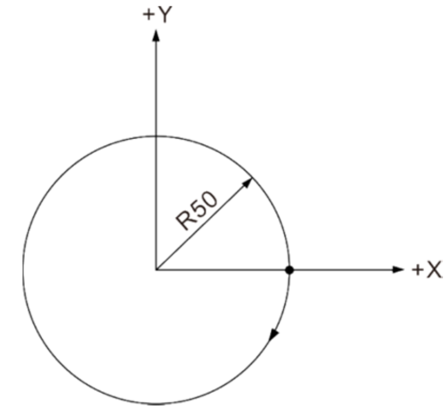
The arc motion expressed in the radius or center method can be used in program coding for selecting conditions.

To mill a whole sphere (a complete circle) of the workpiece, you have to use center method instead of radius method. It is because the roundness deviation will be too large when two semi-sphere (semi-circular) are connected together with the use of radius method.

2

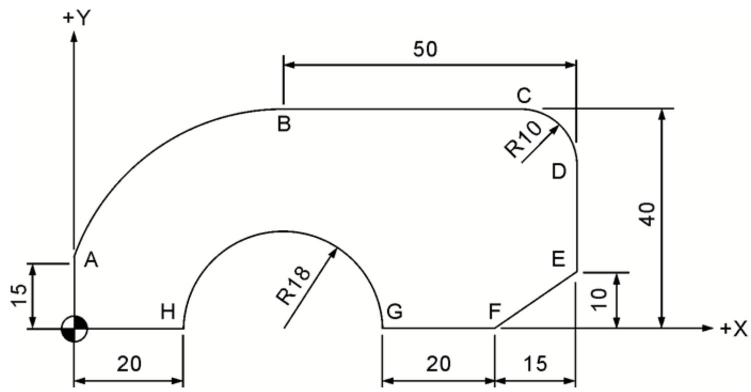
[Example]

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



Milling a complete circle of workpiece

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



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

Notes on G02 and G03 arc interpolation:

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

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

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

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

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

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

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

G04: Dwell time

Format: G04 X_ or
G04 P_

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

Command scope:

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

[Example]

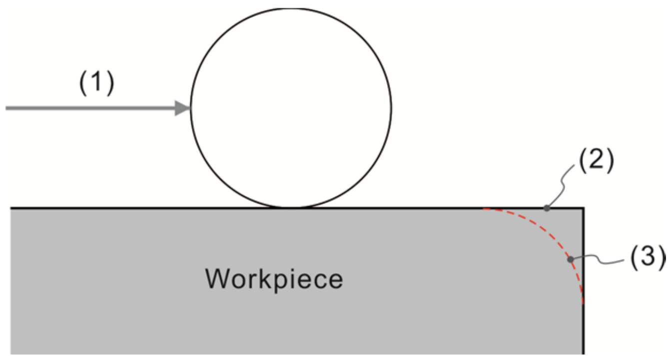
G04 X1.5 or
G04 P1500

The two formats in the example both show that the program pauses for 1.5 seconds.

G09: Exact stop command

Format: G09 G01 X_ Y_

Description: When cutting, the feed rate is constant; that is, the cutting command in the next block will be executed before the execution of the current block is complete. In this case, this mechanism will create a subtle arc at the corner. To eliminate this round corner with G09, the system will confirm the tool position every time it executes a motion block. Once the tool is in the right position which is consistent with the command value, the next block can be executed. With command G09 used, there will be minor discontinuity between blocks; this improves precision at the cost of speed. This command is only active for cutting commands (G01, G02, and G03) of a single block.



(1) Tool movement direction; (2) with G09 in use; (3) Without G09 in use

[Example]

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

G01 Y-100

2

G10/G11: Data entry setup and cancel

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

Description: The G10 command in the syntax of G10 L2 P_ X_ Y_ Z_ is used for workpiece coordinate system data entry. The system is set to the offset coordinate of the workpiece coordinate system when the value of parameter P is set to 0. Parameters P1~P6 correspond to G54 ~ G59 of the workpiece coordinate system while X, Y, and Z represent the origin of the given coordinates system. The P parameter in L20 command syntax can assign the values of P1 ~ P64 for the corresponding expansion workpiece coordinates system. The command format G10 L10 P_ R_ sets up the tool length compensation value where parameter P is the compensation number and R is the actual compensation value for tool radius and length. The setting of P1 from G10 L21 P is for the first forward software limit, P2 is for the first backward software limit, P3 is for the second forward software limit and P4 is for the second backward software limit.
When G90 is active, values specified in G10 will be in absolute form. You can also set G10 with G91 for data entry setting; thus, the values of G10 will be input in incremental form.

Data entry type

L command Format	Other argument format	Description about data type
L2	P_ X_ Y_ Z_	Data entry format of workpiece coordinate system P: 0 is the offset coordinates. 1 ~ 6 set the G54~G59 workpiece coordinates.
L10	P_ R_	Data entry format of tool length compensation P: 1 ~ 100 are paired with No. 1 ~ 100 of the tool length compensation data.
L11	P_ R_	Data entry format of tool wear compensation P: 1 ~ 100 are paired with No. 1 ~ 100 of tool wear compensation data.
L12	P_ R_	Data entry format of tool radius compensation P: 1 ~ 100 are paired with No. 1 ~ 100 of tool radius compensation data.
L13	P_ R_	Data entry format of tool wear compensation P: 1 ~ 100 are paired with No. 1 ~ 100 of tool wear compensation data.
L20	P_ X_ Y_ Z_	Data entry format of extension workpiece coordinate system P: 1 ~ 64 are paired with No. 1~ 64 of the extension workpiece coordinates system.
L21	P_ X_ Y_ Z_	Data entry format of software limit coordinates P: Set to 1 to set the 1st positive software limit; Set to 2 to set the 1st negative software limit; Set to 3 to set the 2nd positive software limit; Set to 4 to set the 2nd negative software limit.

[Example]

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

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

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

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

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

2

G15: Polar coordinates command cancelling

Format: G15

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

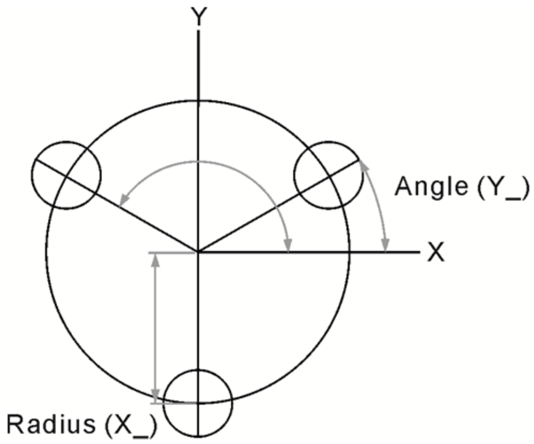
G16: Polar coordinates command

Format: G17 G16 X__ Y__ or;
G18 G16 Z___X__ or;
G19 G16 Y__ Z__

X_ Y_: In a G17 plane the X_ parameter specifies the radius and Y_ the angle coordinates of the polar coordinates system.
Z_ X_: In a G18 plane the Z_ parameter specifies the radius and X_ the angle coordinates of the polar coordinates system.
Y_ Z_: In a G19 plane the Y_ parameter specifies the radius and Z_ the angle coordinates of the polar coordinates system.

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

[Example]

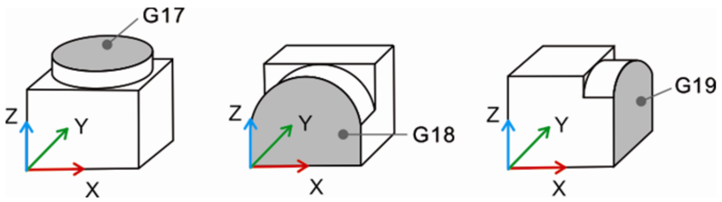


N1 G90 G16	> set up polar coordinates
N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0	> the command to loop drill at position of radius 100 and angle 30
N3 X100.0 Y150.0	> machine the second hole
N4 X100.0 Y270.0	> machine the third hole
N5 G15 G80	> cancel the polar coordinates function

G17/G18/G19 : Plane designation command

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

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



G21/G20: Metric / Inch input command

Format: G21 or G20

G21: Set the metric system
G20: Set the inch system

Description: This command sets the metric or inch unit of measure for the system. Command G21 and G20 are applicable to the linear axes but not the rotation axes. It must be executed at the beginning of a program. Please note that metric and inch setting cannot be changed during command execution. All relevant values set in the system will be referring to this unit setting, such as the F value for setting the cutting feed rate, coordinates command, offset of the tool workpiece coordinate system, tool compensation, and moving distance. G21/G20 are continuous effective commands so the system will apply the unit you set at the beginning. In addition, G21 and G20 cannot be used in one program at the same time.

2

G24/G25: Mirror image setup command / cancelling command

Format: G24 X_ Y_ Z_

G24: Mirror image setup command

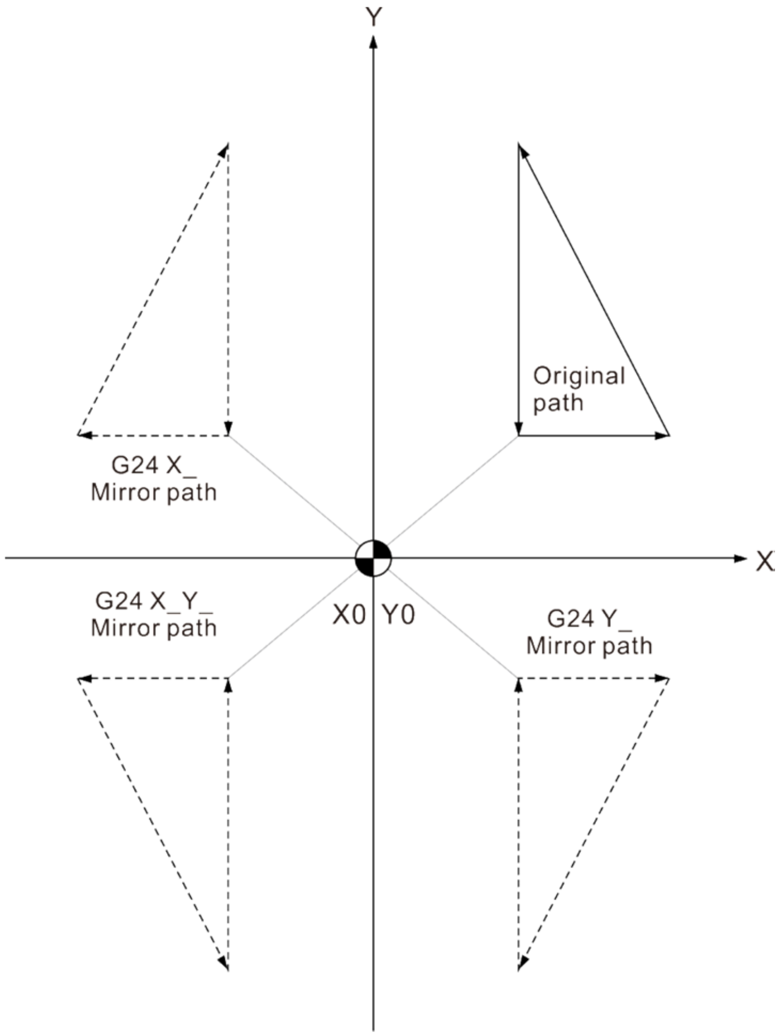
X_ Y_ Z_: Specify the axial direction and center of mirror image

G25 X_ Y_ Z_

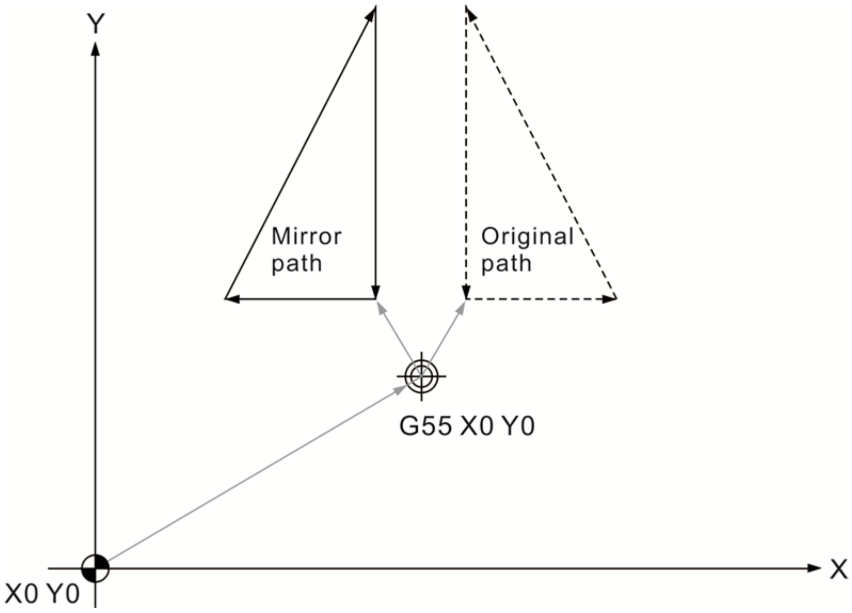
G25: Mirror image setup cancelling command

X_ Y_ Z_: Cancel the function of mirror image

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



[Illustrations]




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

The original path is showed in dotted line, when you apply the mirror function to complete the path, the system will then complete the path showing in solid line. (In this example, X-axis is specified as the mirror axis.)

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

2

G28: Homing through the reference point

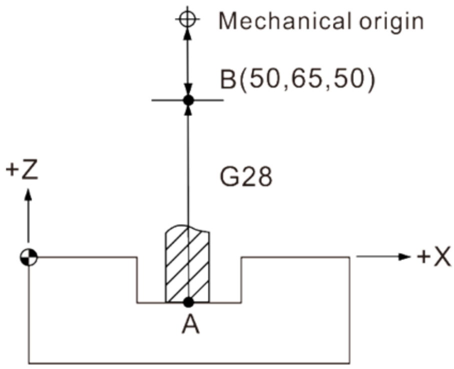
Format: G90 G28 X_ Y_ Z_ or;
G91 G28 X_ Y_ Z_

X_ Y_ Z_: Coordinates of reference point

Description: This command instructs the tool to fast move (G00) from the reference point given by the command to the mechanical origin. With this command, the tool can go to the specified reference point and then return to the mechanical origin in rapid traverse.

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

[Example]

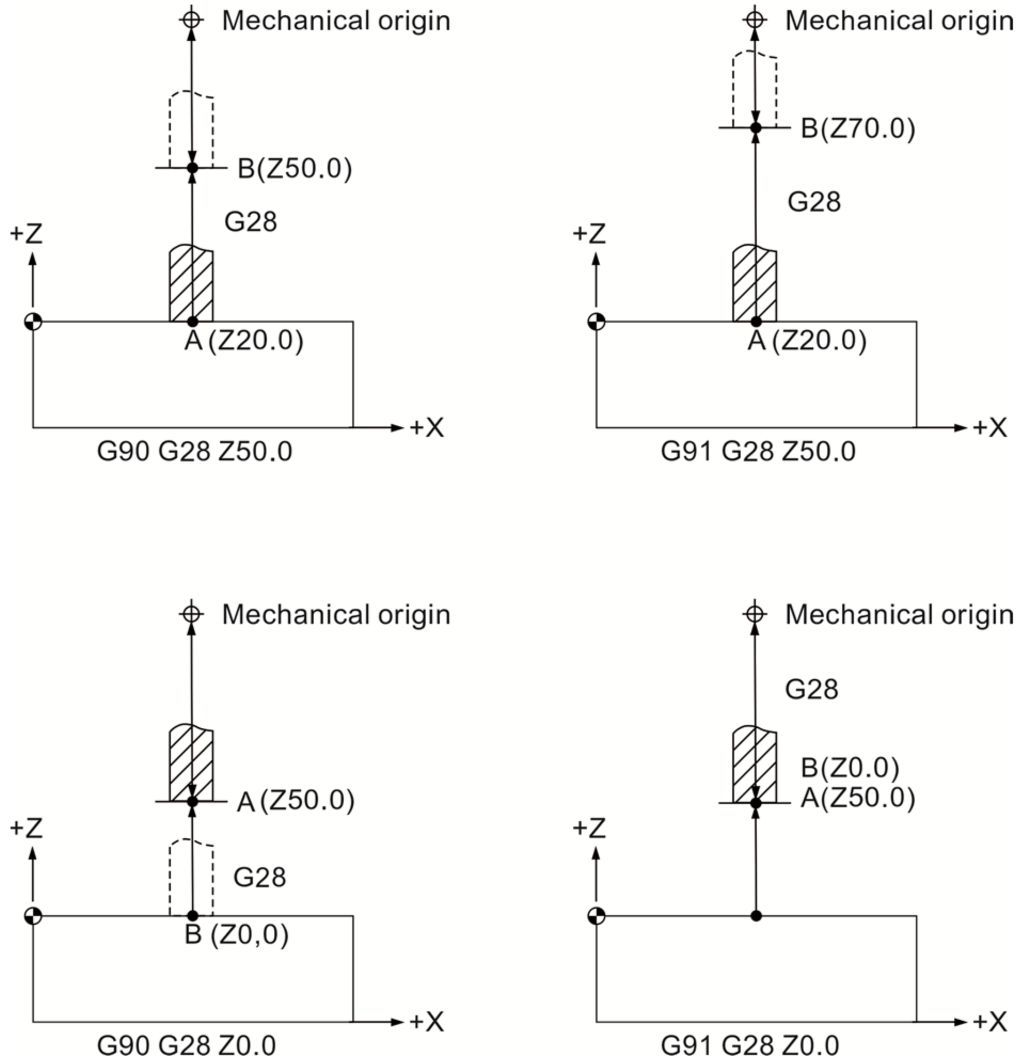


G90 G28 Z50 > Return to mechanical origin from point A through reference point B (Z-axis).

M06 T02 > Switch to tool 2.

[Example]

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



2

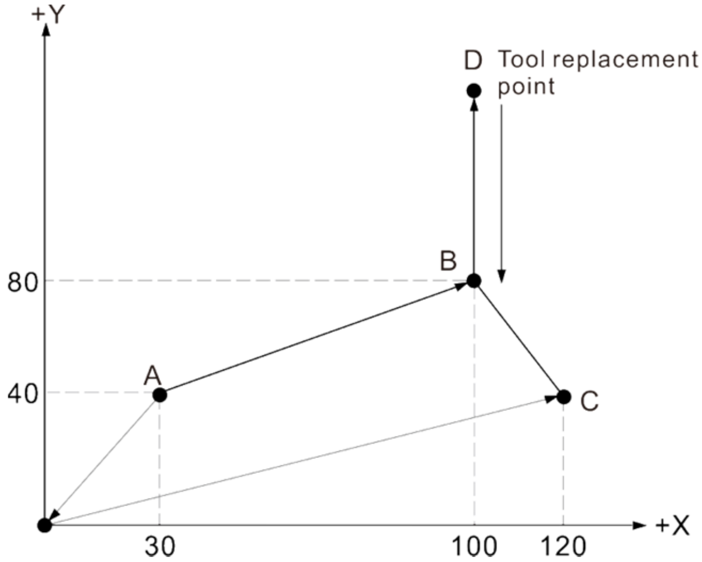
G29: Homing through the starting point

Format: G90 G29 X_ Y_ Z_ or;
G91G29 X_ Y_ Z_

X_ Y_ Z_: The final motion position of current block

Description: When G29 is applied, the tool moves from the mechanical origin or any point and pass the reference point and then go to the point specified by the block. X_Y_Z_ is the end point of G29 motion block. Please note that G29 and G28 have to be used together; that is, the tool will move to the reference point set by G28 and then go to the point specified by G29. In this case, you don't have to calculate the actual moving distance from reference point to the mechanical origin.
If G29 is executed independently without G28 that sets the reference point, an alarm will occur and the tool stops moving.

[Example]



Incremental value setup (tool path A > B > D > B > C)

```
G28 G91 X70.0 Y40.0
M06
G29 X20.0 Y-40.0
```

Absolute value setup (tool path A > B > D > B > C)

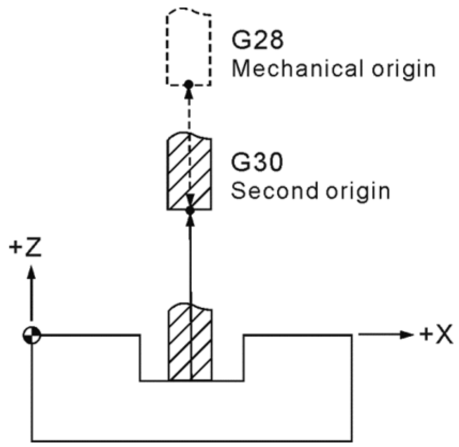
```
G28 G90 X100.0 Y80.0
M06
G29 X120.0 Y40.0
```

G30: Homing to the second, third, and fourth 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, and 4th reference point
X_ Y_ Z_: Coordinates of intermediate point

Description: Parameters P2, P3, and P4 designate coordinates of reference points 2, 3 and 4. You can omit the parameter P2 if the second reference point is required.
Arguments X_Y_Z_ designate coordinates of the intermediate point. The tool moves back to the second, third, or fourth reference point from the current location through the designated intermediate point. The coordinates of the second, third, or fourth reference point are defined by the system parameters. This command is most commonly used for tool replacement. For command status in absolute value the G30 Z0.0 motion command accomplishes the required motions by moving the Z axis to the intermediate point and the second reference point in turn.
Please cancel the tool compensation (executing G40 and G49) before running commands G28 and G30. The tool radius and length compensation are cancelled automatically when executing command G28 and G30. The tool compensation function is resumed only after executing command G43/G44 before the next block is executed. The tool radius compensation function is resumed at the next block after G28 or G30.

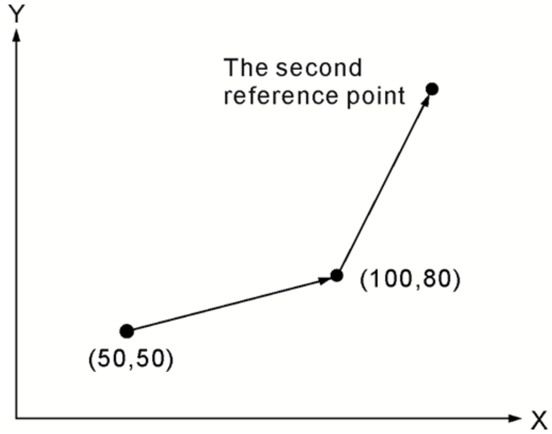


As shown in the figure above, in the G90 mode the G30 Z0 command moves the Z axis to the intermediate point (the mechanical origin in this example) before being returned to the G30 (second origin) point to accomplish the second origin homing.

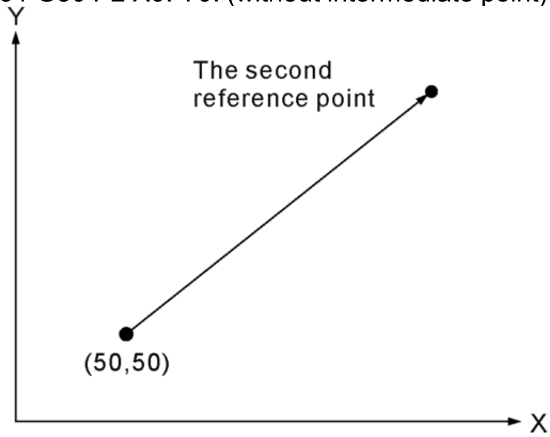
2

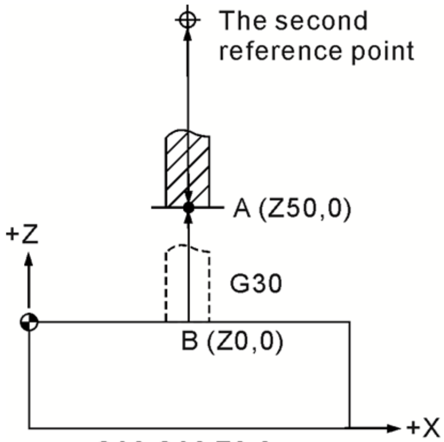
[Example]

G90 G30 P2 X100. Y80.

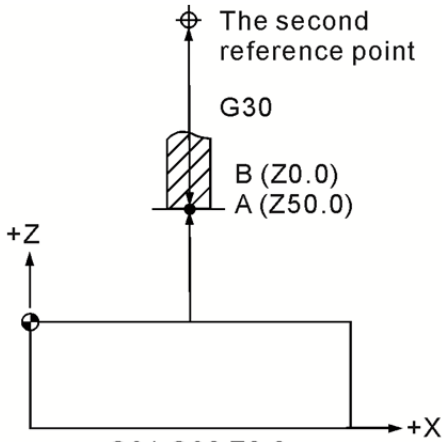


G91 G30 P2 X0. Y0. (without intermediate point)

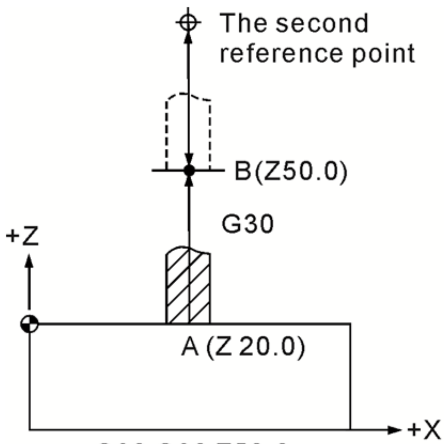




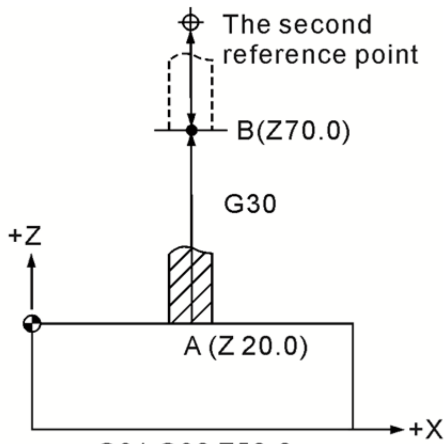
G90 G30 Z0.0



G91 G30 Z0.0



G90 G30 Z50.0



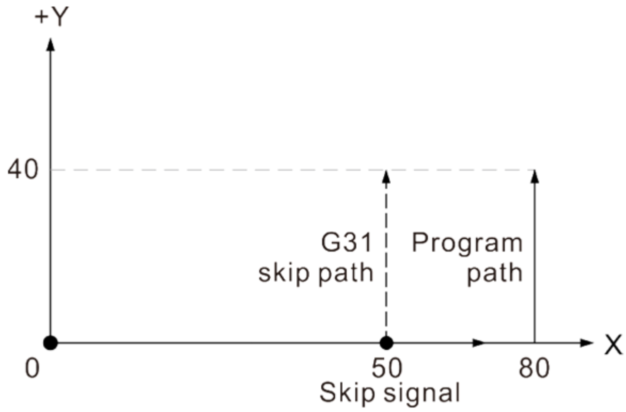
G91 G30 Z50.0

G31: Skip function command

Format: G31 X_ Y_ Z_ F_

Description: During linear motion, G31 stops running the motion path immediately via external skipping signal and the next block will be executed. This command is only valid in single block. G31 cannot be executed when tool radius compensation (G41/G42) is functioning. Thus, please cancel tool compensation (G40) before using this command.

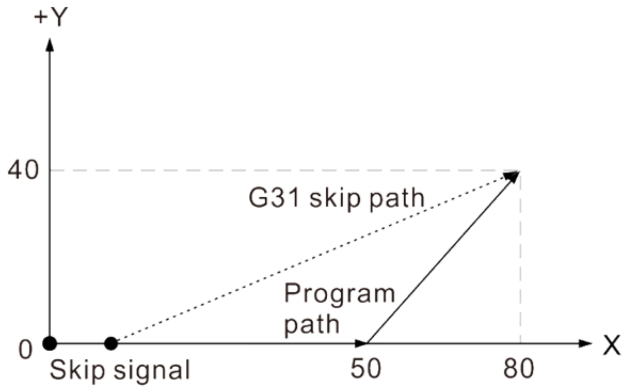
[Example 1]




```
G90 G00 X0 Y0
G01 G31 X80.0 F500.0
Y40.0
```

As shown in the figure above the motion path remains straight (solid line) if there is no skip signal. When a skip signal is applied, the program aborts execution of the current statement and starts executing the movements given in the next statement (dotted line).

[Example 2]




```
G90 G00 X0 Y0
G01 G90 G31 X50.0 F150.0
X80.0 Y40.0
```

If no skip signal is applied, the actual path is shown as the solid line shown in the figure above. If a skip signal is applied, the actual path is shown as the dotted line as the tool starts executing motions of the next block from the signal input point.

2

G40: Tool radius compensation cancelling command

Format: G40 or;
G40 X_ Y_

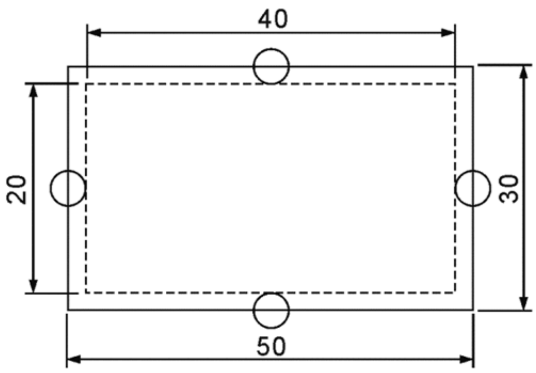
Description: This command cancels the tool compensation function when it is not needed in the tool path. As a status command the compensation command remains active unless cancelled by another command. Command G40 resets the tool compensation status to avoid path error caused by tool offset for compensation.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

G41/G42: Tool radius left and right compensation command

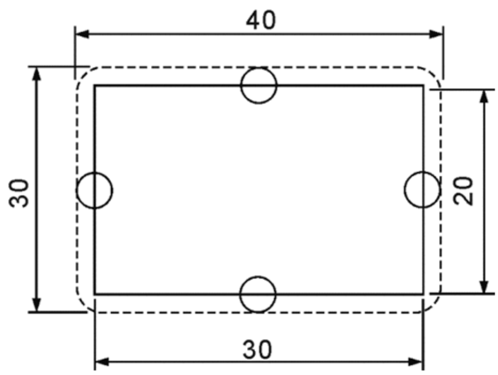
Format: G00 G90 G41 D_ or;
G00 G90 G42 D_

G41: Tool radius leftward compensation
G42: Tool radius rightward compensation
D_: Tool radius compensation data code

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



Tool radius: 10 mm
Milling without compensation for the outer contour



Tool radius: 10 mm
Contour cutting without compensation inner contour

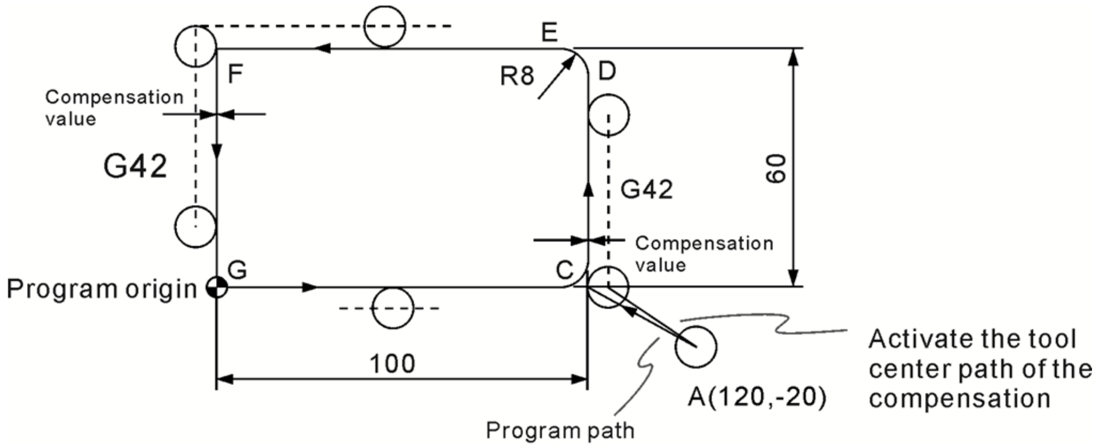
As illustrated in the figure above, a workpiece after machining may have a dimension one tool diameter greater or smaller caused by the tool diameter when cutting along the profile of workpiece.

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

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

Note for tool radius compensation:

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



Execute G42 to perform rightward radius compensation

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

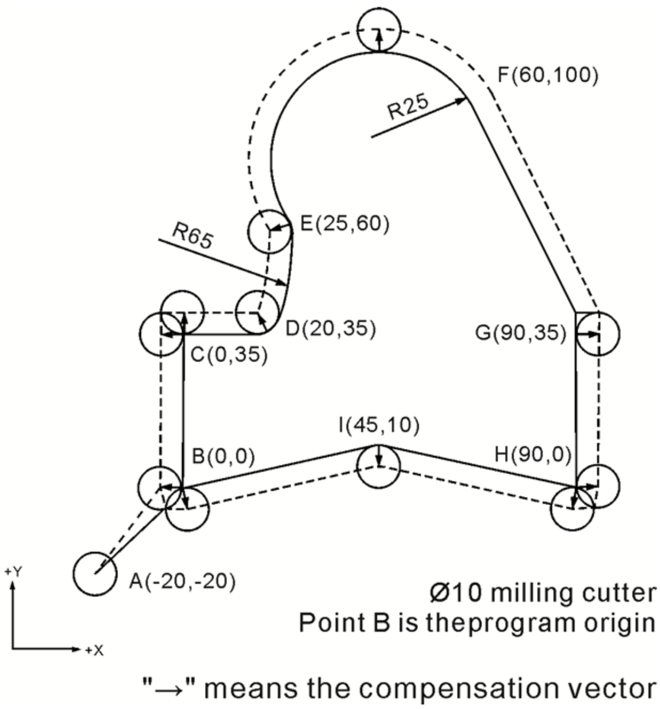
(2) When designing the program, the tool compensation number, e.g. D11 and D12, defined in a program must correspond to the one contained in the compensation table. These tool radius compensation values are numbers entered by the operator in the tool entry function of the OFS group in advance.

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

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

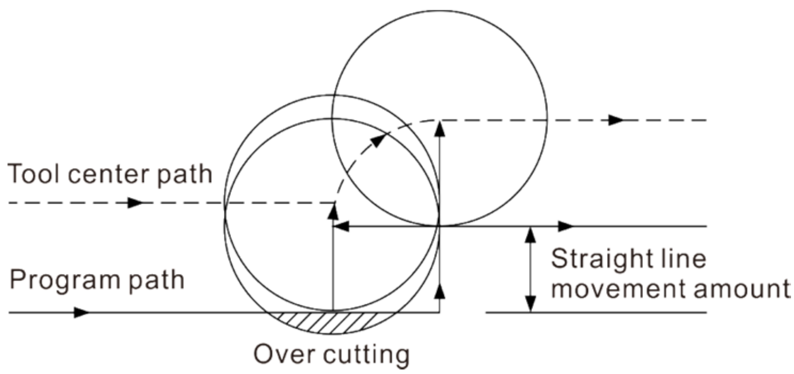
(5) After the program path is completed under the tool radius compensation mode, the command G40 should be executed to cancel the compensation status and return the tool radius center back to its actual coordinate points. That is, after a G40 command is executed in a program, the motion path cancels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

2



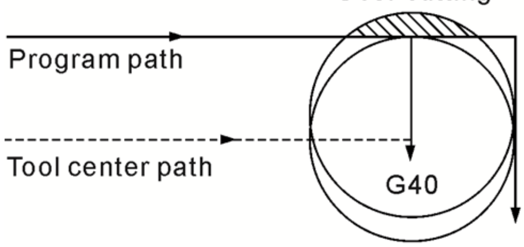
G90 G00 X-20.0 Y-20.0	> Fast positioning to point A
G01 G41 X0 Y0 D12 F80	> A → B: Start leftward compensation command G41
Y35.	> B → C
X20.	> C → D
G03 X25.0 Y60.0 R65.0	> D → E
G02 X60.0 Y100.0 R25.	> E → F
G01 X90.0 Y35.0	> F → G
G01 Y0	> G → H
X45. Y10.	> H → I
X0 Y0	> I → B
X -20. Y -20.	> B → A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

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



Straight line movement amount smaller than tool radius

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



Moving distance after canceling the compensation is smaller than cutter radius

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

(8) With the following conditions, the function of tool radius compensation will be disabled: Tool radius compensation will be disabled when it executes to the motion block after G40 command. Or when it executes to the final block and no more motion block should be executed afterwards, then, the final motion block has no tool radius compensation.

2

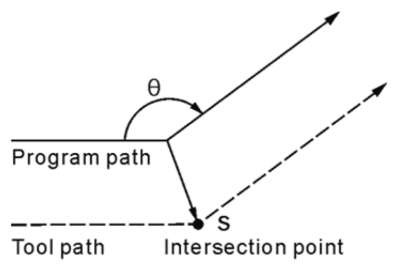
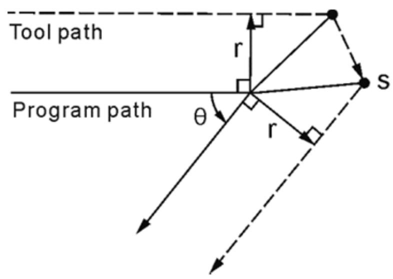
Tool radius compensation path type:

Regarding the compensation path, the angles (180° > θ> 90°, 0 < θ < 90°) generated at the intersection point (when transiting between two blocks) will determine the path type.

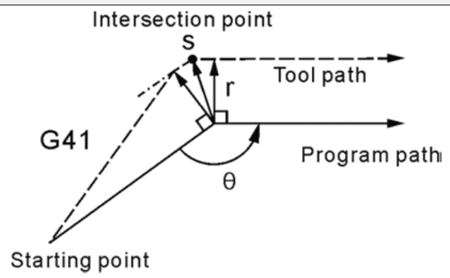
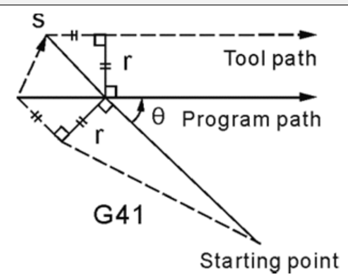
(1) When angle generated by two blocks is within the range of 90°~ 180° (180° > θ > 90°), the tool moves in an inward curved angle path.

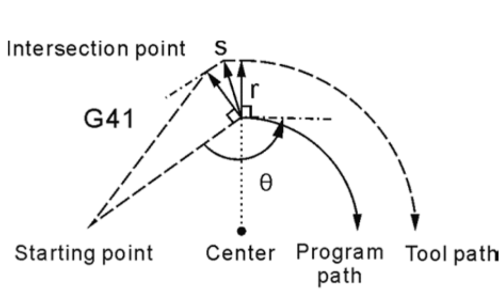
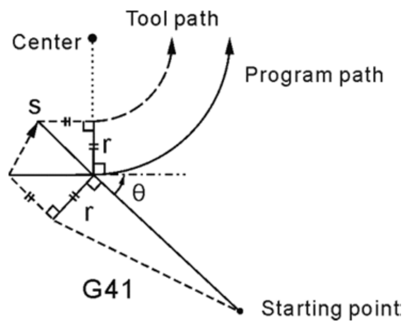
(2) When the angle generated by two blocks is within the range of 0° and 90° (0 < θ < 90°), the tool moves in an outward curved angle path.

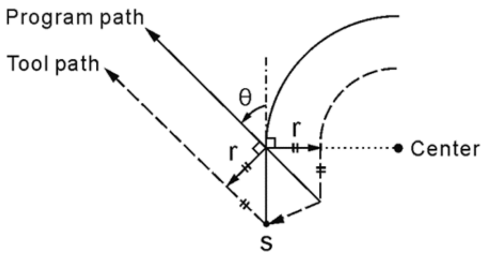
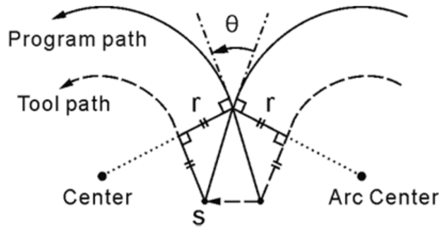
See the illustration below for details.

Inward curved angle	Outward curved angle
	

See the compensation path below. Compensation is done at both the starting and end point.

Line to line	
	

Line to arc	
	

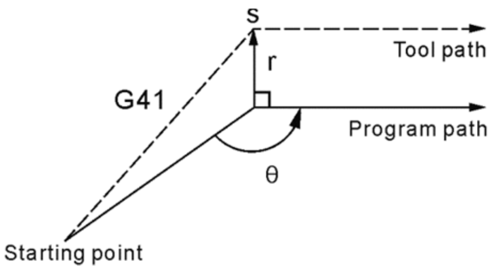
Inward curved angle: Arc to line	Outward curved angle: Arc to arc
	

Compensation path switch:

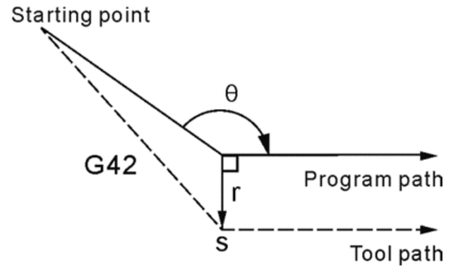
When switching from path without compensation to the one with compensation, the motion track of the tool center is as shown in the figure below (G40 >G41; G40 > G42).

During compensation, the motion track remains active. When compensation is cancelled (G40) or switched to compensation direction is changed as elaborated in the figure of G41 > G40 and G42 >G40, the motion track will be as illustrated in the figures of G41 > G42 and G42 > G41 below.

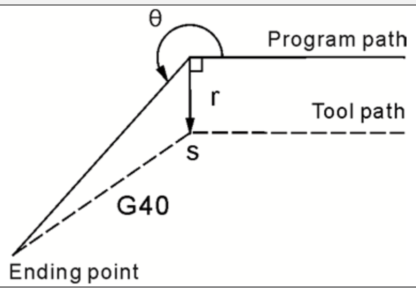
G40 > G41



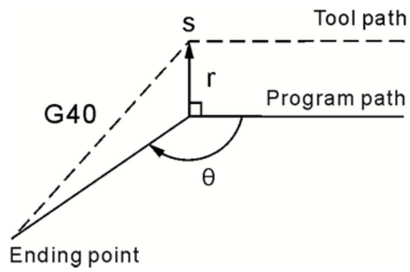
G40 > G42



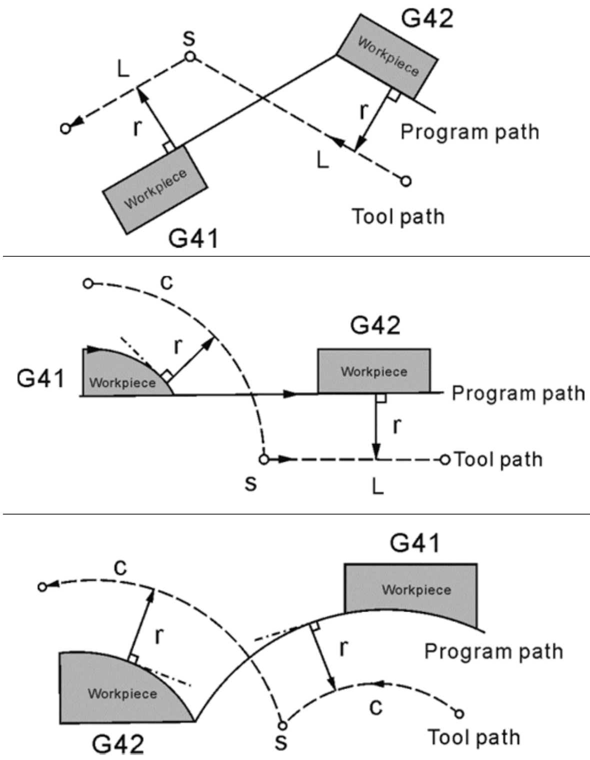
G41 > G40



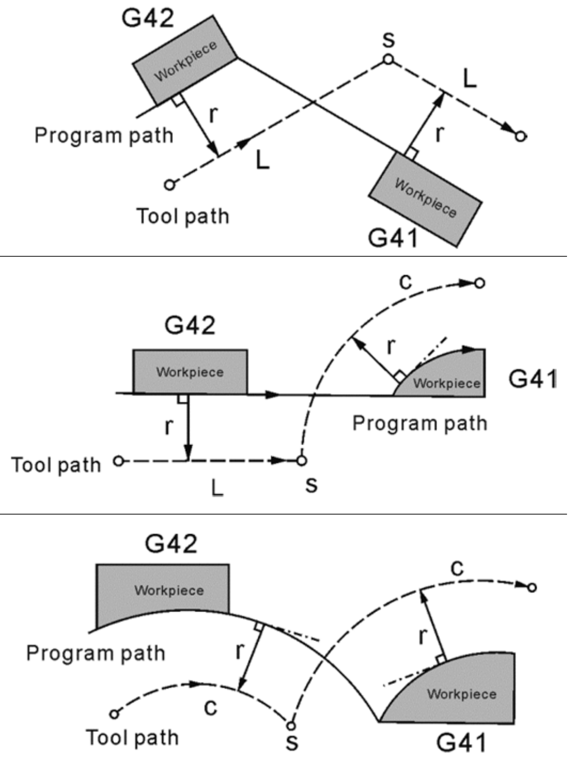
G42 > G40



G41 > G42



G42 > G41



2

G43/G44: Tool length compensation command

Format: G43 Z_ H_
G44 Z_ H_

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

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

Definitions of the command are described below:

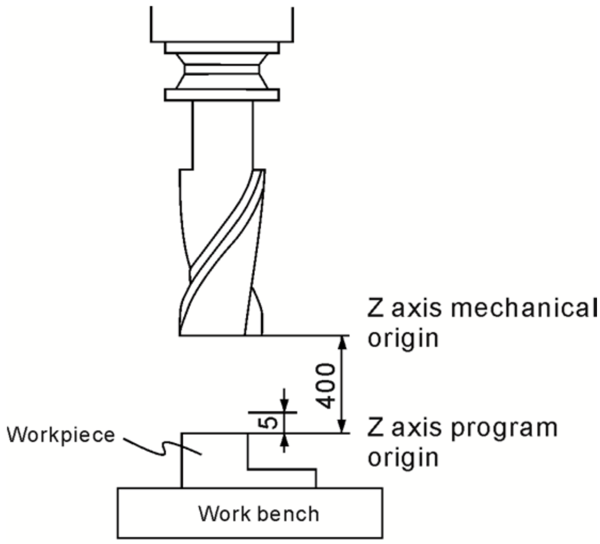
Z: The coordinate position is: zero point + tool length compensation. The zero point is the reference to the Z axis of tool coordinate.

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

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

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

[Example] Tool length compensation setup



Tool length compensation

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

Note:

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

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

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

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

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

G49: Tool length compensation cancelling command

Format: G49

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

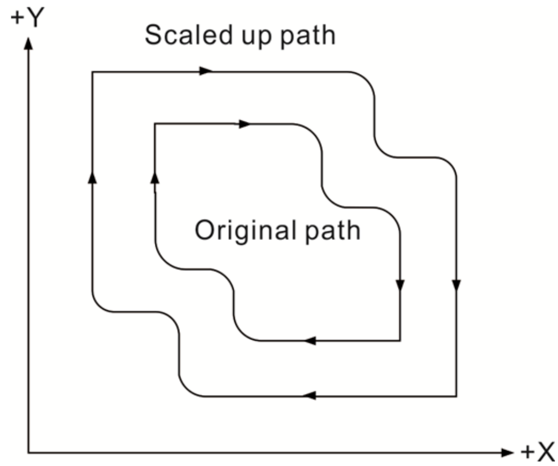
2

G50/G51: Scaling up/down command/cancelling command

Format: G51 X_ Y_ Z_ P_ or
G51 X_ Y_ Z_ I_J_K_

X_ Y_ Z_: Coordinates center for scaling
I_ J_ K_: The scaling ratio of axis X, Y, Z
P_: Scaling ratio

Description: Argument X_ Y_ Z_ sets the X, Y, and Z coordinates of the scaling center and P sets up the scaling ratio. Arguments I_ J_ K set the scaling ratio of axis X, Y, and Z. This command enables the machining workpiece to cut final pieces with proportionally different dimensions. The machining path is enlarged or contracted with the coordinate's center given by argument X_ Y_ Z_ and scaling ratio by P.
The minimum value of argument P and I_ J_ K is 1 with a ratio ranges between 0.001 and 999.999. For example, argument P100 represents to scale down to one-tenth.



As shown in above figure the original tool path is converted into a new path by executing command G51 with scaling ratio set by argument P. That is, a new tool path is generated according to the settings scaling ratio. The scaling function does not change tool radius compensation, tool length compensation, and tool position compensation as the compensation action and value are made after scaling. When executing commands M02 and M30 or when the NC300 controller is reset, the scaling mode will be cancelled. The reset key also can cancel scaling. Execute command G50 in program to reset the scaling function as well as return to original scale status and cut in the normal path.

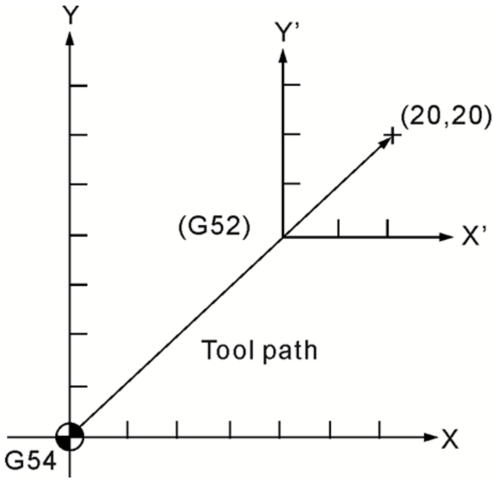
G52: Local coordinate system setup command

Format: G52 X_ Y_

X_ Y_: Origin of the local coordinates system

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

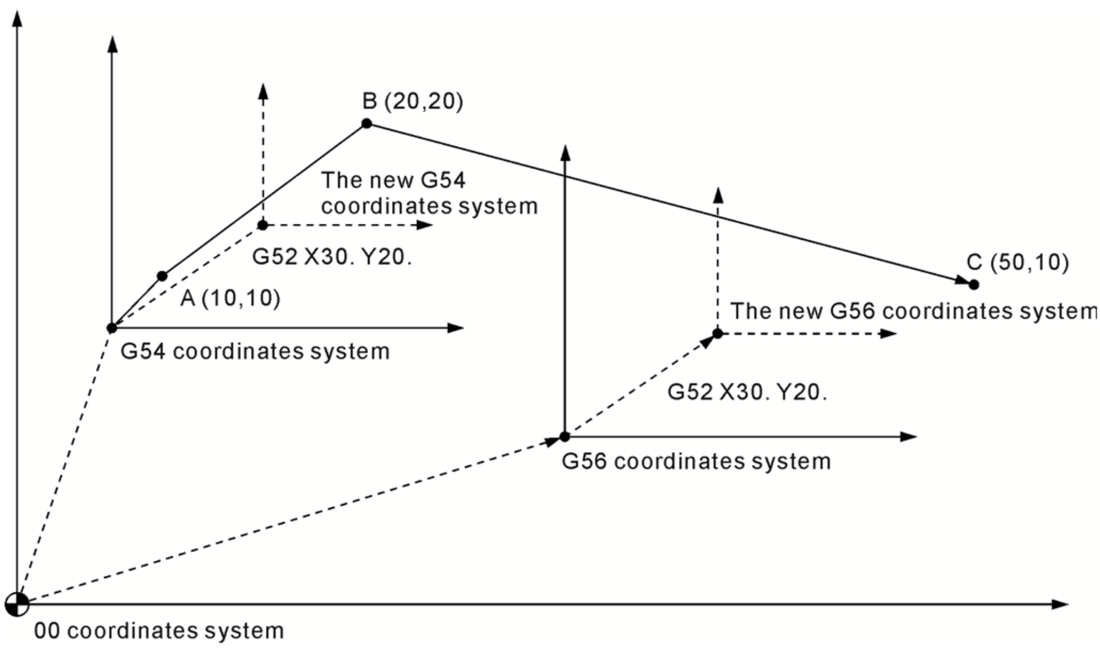
[Example 1]



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.----------------------------(A→B)
G56 G00 X50. Y10.---------------------(B→C)



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

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

G53: Mechanical coordinate system setup command

Format: G53 X_ Y_ Z_

X_ Y_ Z_: Actual arriving position of mechanical coordinates

Description: Coordinates X, Y, and Z are the actual ending point in mechanical coordinates system specified by the program coordinates. Machine suppliers use this command to set up the tool replacement position with the reference point given in mechanical coordinates, which is based on the mechanical coordinate system. The command format must be in absolute coordinates status. This command is ignored when in incremental status.

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

Note:

(1) Command G53 functions only at G90 status. It is ignored in G91 mode. However, a status command contained in the same statement of G53, G00/G01 or G90/G91, changes the status and affects the motion status of the next block.

(2) If the statement containing command G53 also contains a specific axial command, then the axis moves to a specified point. Otherwise, there is no position command.

(3) If both commands G53 and G28 are set in the same statement, the one read later becomes active. When command G53 is active, the motion position refers to the mechanical coordinates. If command G28 is active, then the absolute coordinates are referred to.

[Illustrations]

Example 1

G91G53X150.Y-150.	This command is ignored.
X-30.Y-30.	This command is changed to incremental motion mode.

Example 2

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

Example 3

G1G53X100.Y-100.F1000	This command executes in G00 status.
X50.Y50.	The system moves in G01F1000 motion status.

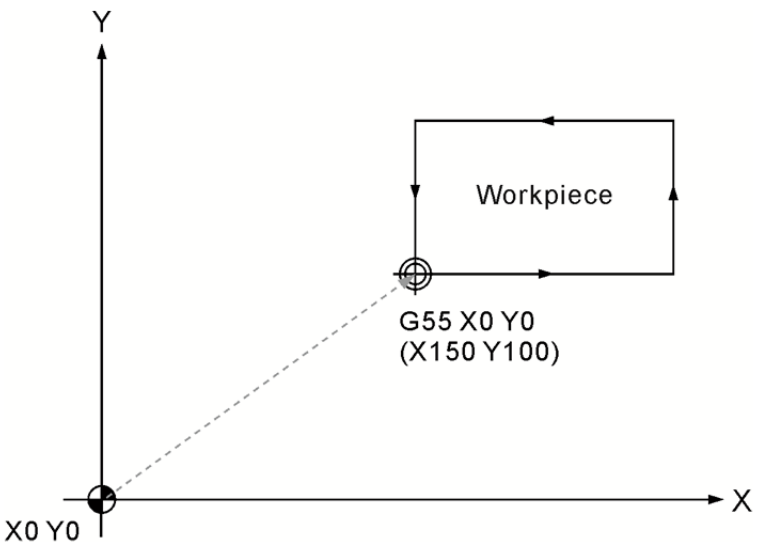
2

G54~G59: Workpiece coordinate system selection command

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

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

[Example]




```
O1000
G00 G90 G55 X0 Y0
G00 X100.0
Y50.0
X0.0
Y0.0
M30
```

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

G61: Exact stop mode

Format: G61

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

[Example]


```
G61 G91 G01 Y100. F200.   ---------------------------------------(Correct positioning)
X100.                     ---------------------------------------(Correct positioning)
G64                       ---------------------------------------(Stops correct positioning)
```

G64: Cutting mode command

Format: G64

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

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

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

G65: Non-continuous effect macro calling command

Format: G65 P_ L_ℓ

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

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

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

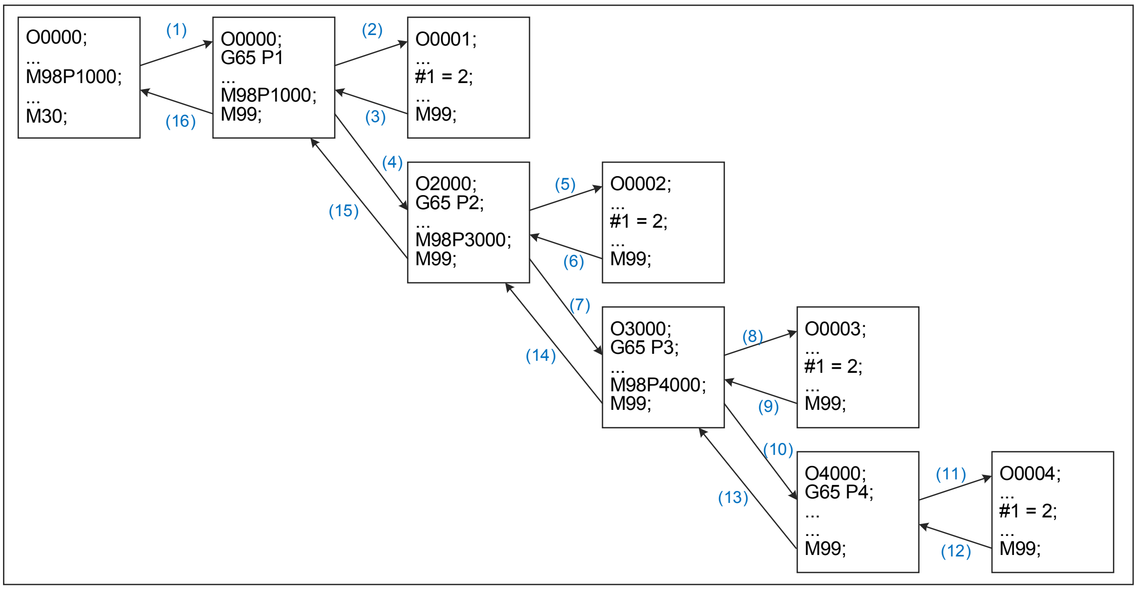
[Example]

```
Main program                         Subroutine
N0010                                O1010
N0020 G65 P1010 L2 A5.0 B3.0         N1020 #3=#1+#2
N0030                                N1030
N0040                                N1040
N0050                                N1050 M99
```

After a macro is executed, it returns to the statement next to the one containing it. That is, the program continues executing from the statement next to the G65 command. In the sample described above, the value of the #1 variable A5.0 is 5.0. Please refer to the table below.

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

[Illustrations]



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

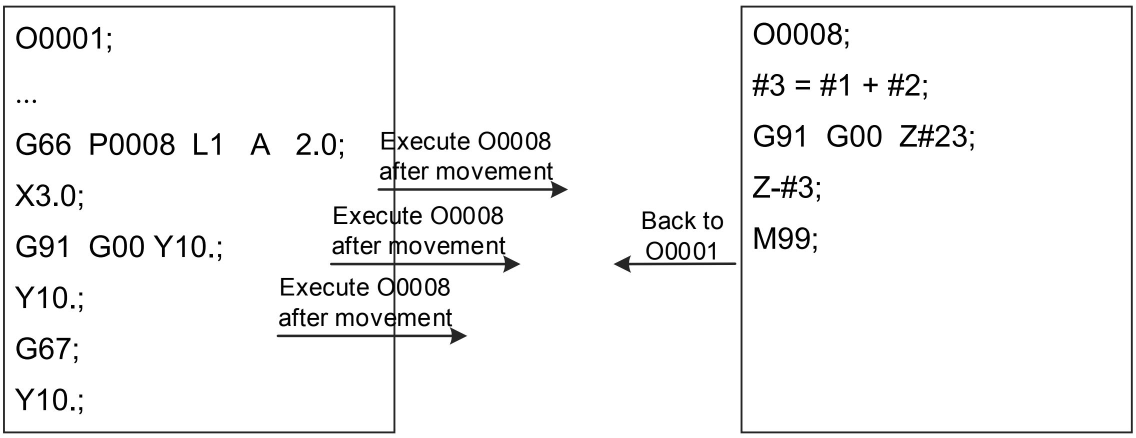
G66/G67: Continuous effect macro calling command / cancelling command

Format: G66 P_ L_ℓ or;
G67

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

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

[Example]



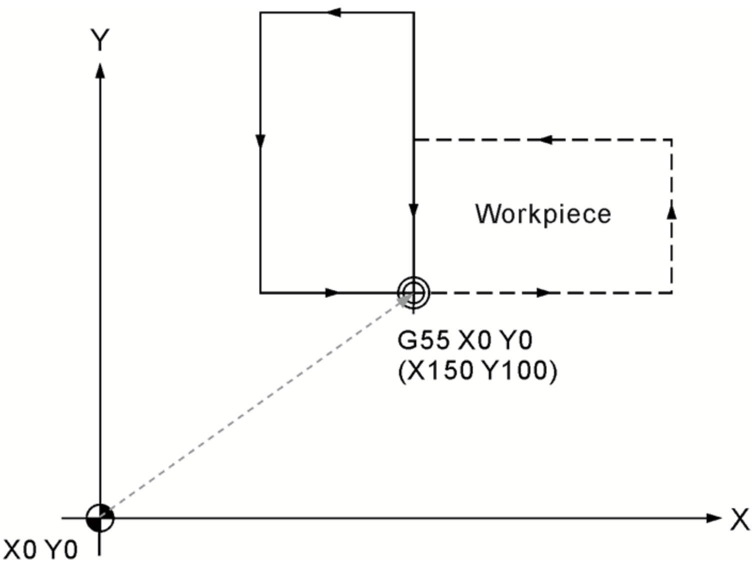
G68/G69: Coordinate system rotation/cancelling command

Format: G68 X_Y_R_

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

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

[Example]



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

When rotates 90 degrees:
O1000
G00 G90 G55 X0 Y0
G68 X0 Y0 R90.0
G00 X100.0
Y50.0
X0.0
Y0.0
M30

The original program (at the top of the figure) machines workpiece is in the path shown with a dotted line. If the workpiece is placed in another direction (here it is rotated 90 degrees), the program can still function without any modification by adding command G68 to rotate the program path.

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

2

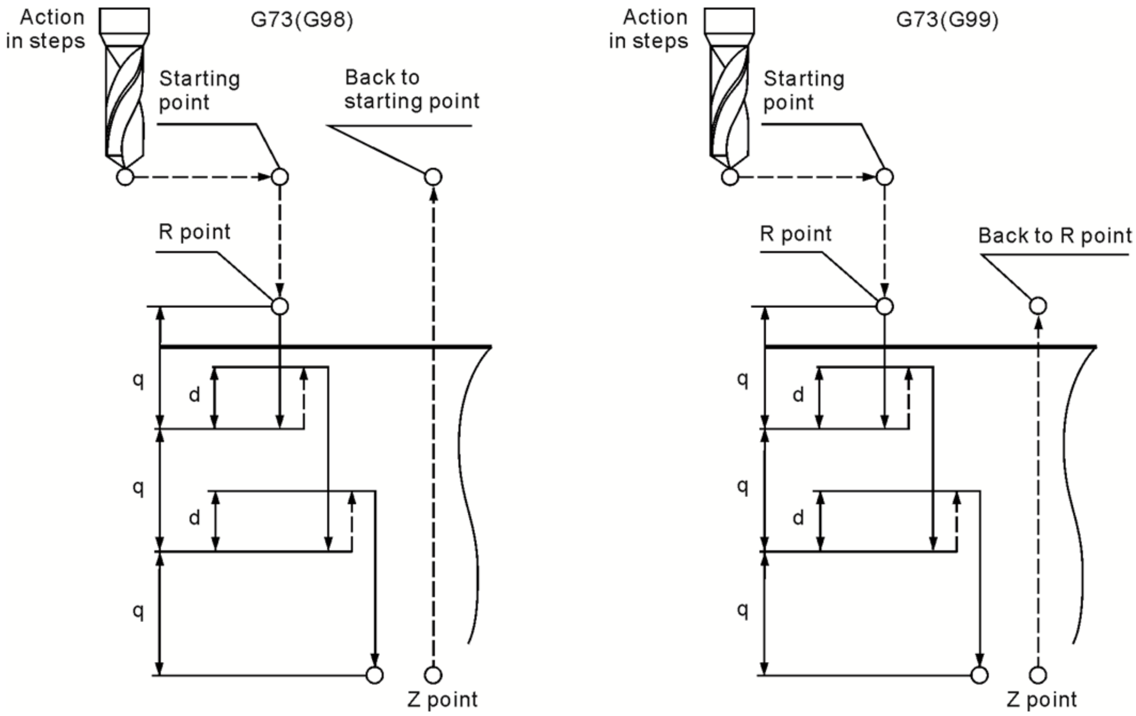
G73: Peck drilling cycle command

Format: G73 X_ Y_ Z_ R_ Q_ F_ K_

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

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

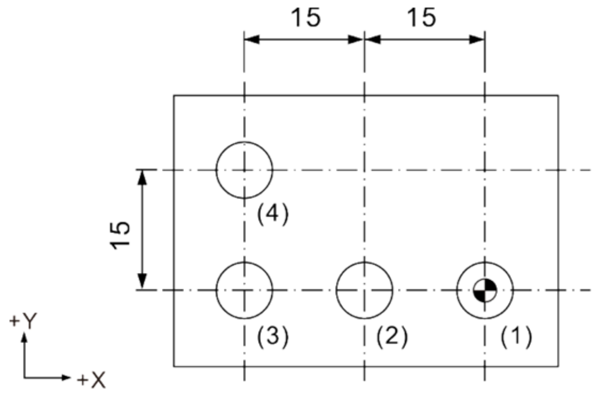
[Example]

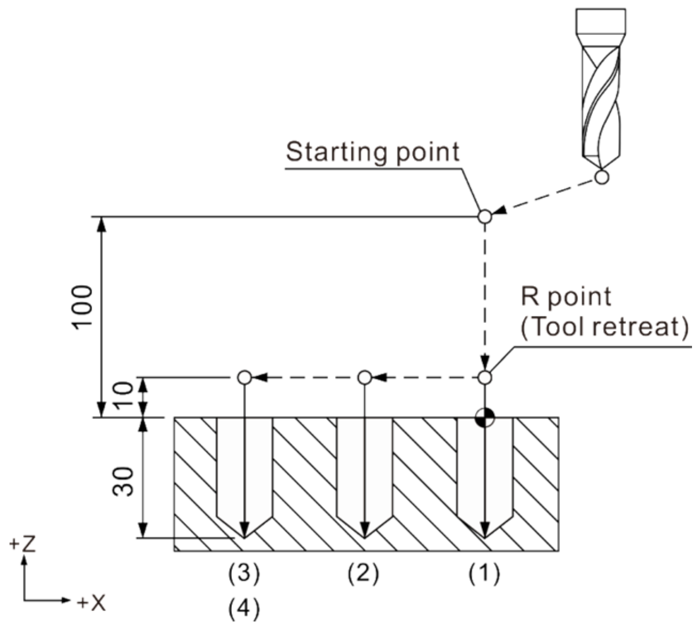


Note:
(1) The tool radius compensation function is ignored by the peck drilling loop command.
(2) Run command G80 to cancel the loop cutting motion.
(3) Value Q cannot be specified as a negative value. Otherwise the alarm will display G code error.
(4) The value of parameter K will be rounded down to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

[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
```

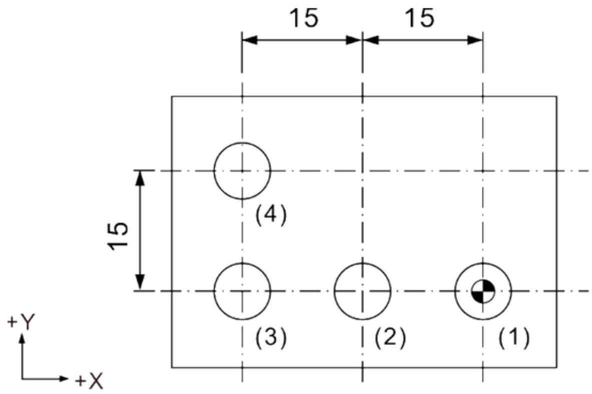


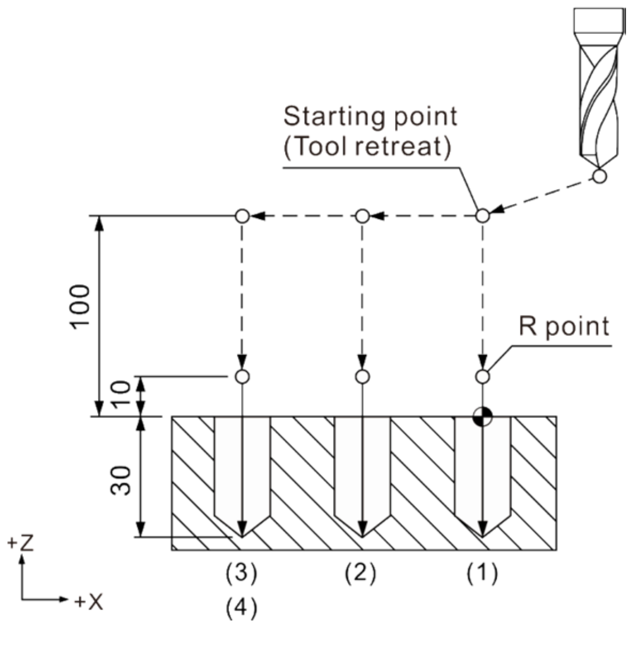


2

[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
```





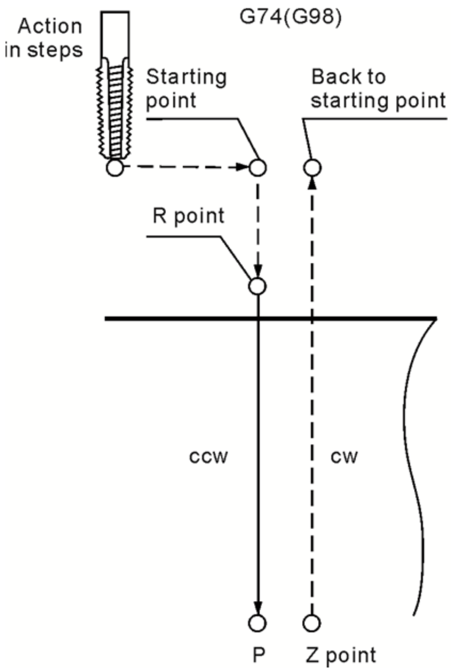
G74: Left spiral tapping cycle command

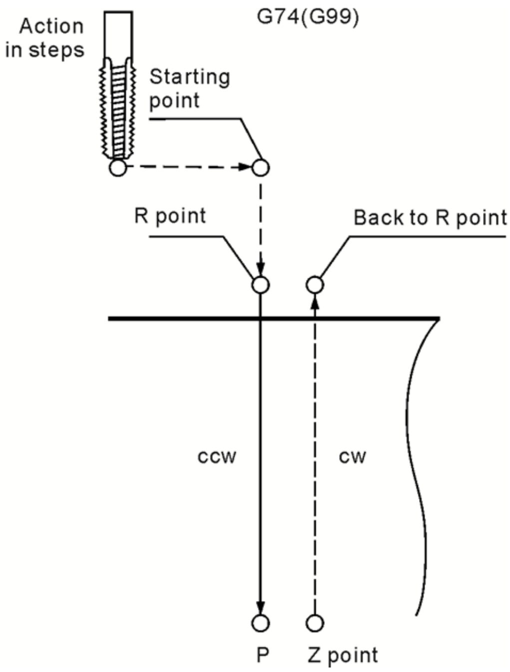
Format: G74 X_ Y_ R_Q_ Z_ P_ F_ K_

X_ Y_: Ending point of a single block
Z_: Bottom of hole to be tapped
R_: Initial safety height
Q_: Peck drilling depth of each time
P_: Pause time (in the least unit of 1/1000 second), without decimal point.
F_: Spiral feed rate
K_: Number of loops
Feeding speed of tapping (mm/min) = Lead (mm/rev) × Spindle speed (rev/min).
F = P × S

Description: This command is for left-handed threading, which requires left-handed tapping cutter and reverse turning spindle to execute command G74. It sets the machine to move as described below:

- fast move the tool to position set by coordinates (X, Y)
- fast position to height R
- tapping at speed set by F till the depth of Z
- set the spindle to turn in positive direction
- retreat the Z-axis in positive direction to height R
- set the spindle reverse turn after reaching height R and prepare for reverse

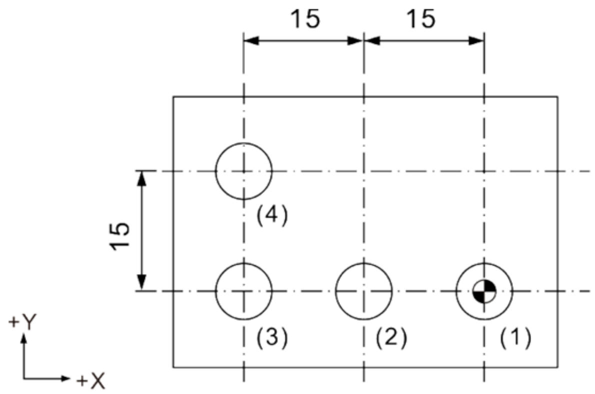


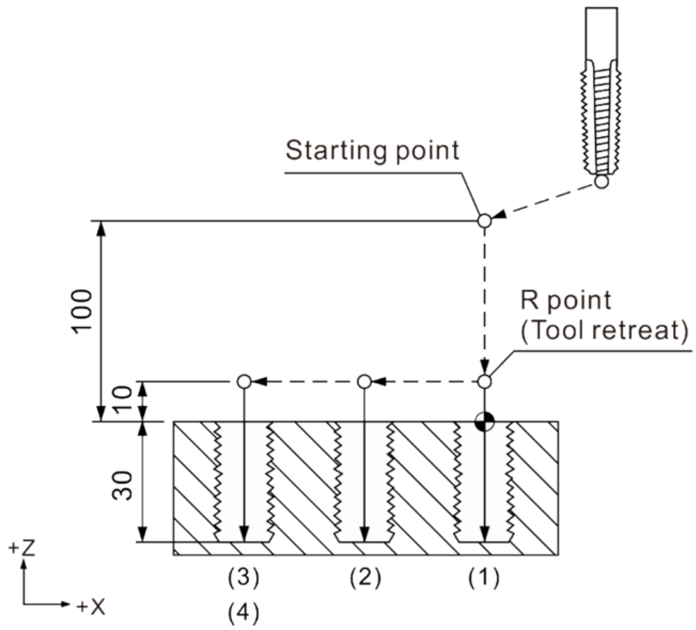


In tapping cycles the machine cuts at the speed given by parameter (100% program value) and the spindle revolution factor and feed rate of the control panel are set to inactive. Furthermore, the program stop key will be ignored to ensure thread accuracy. The stop key is active only after the thread cutting or before the tapping operation.

[Example]

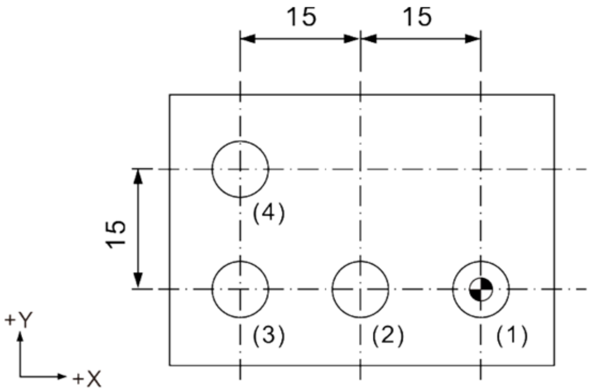
```
G17 G90 G00 G54 X0. Y0.
G00 Z100.
M29 S1000
G99 G73 X0. Y0. Z-30. R10. P1000 K1 F1000.   --------------------------------------------------------(1)
X-15.                                        --------------------------------------------------------(2)
X-30.                                        --------------------------------------------------------(3)
X-30. Y15.                                   --------------------------------------------------------(4)
M28
G91 G80 G28 X0. Y0. Z0.
```

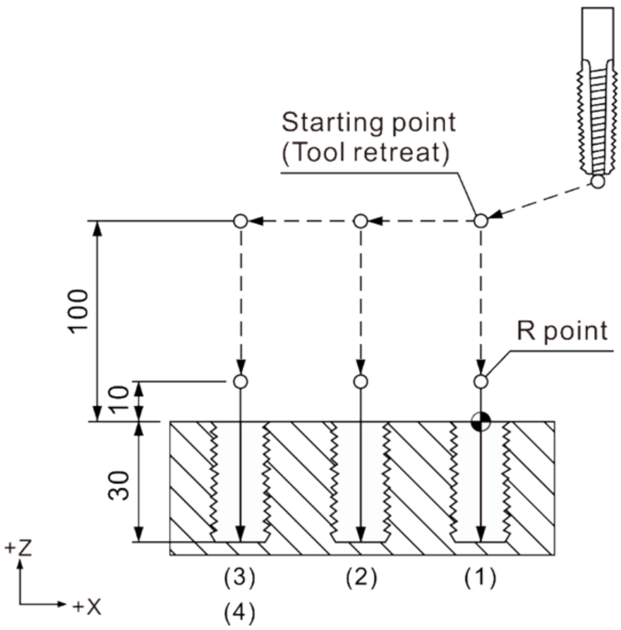




[Example]

```
G17 G90 G00 G54 X0. Y0.
G00 Z100.
M29 S1000
G99 G73 X0. Y0. Z-30. R10. P1000 K1 F1000.   --------------------------------------------------------(1)
X-15.                                        --------------------------------------------------------(2)
X-30.                                        --------------------------------------------------------(3)
X-30. Y15.                                   --------------------------------------------------------(4)
M28
G91 G80 G28 X0. Y0. Z0.
```





2

G76: Fine boring cycle command

Format: G76 X_ Y_ R_ P_ Z_ Q_ F_K_

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

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

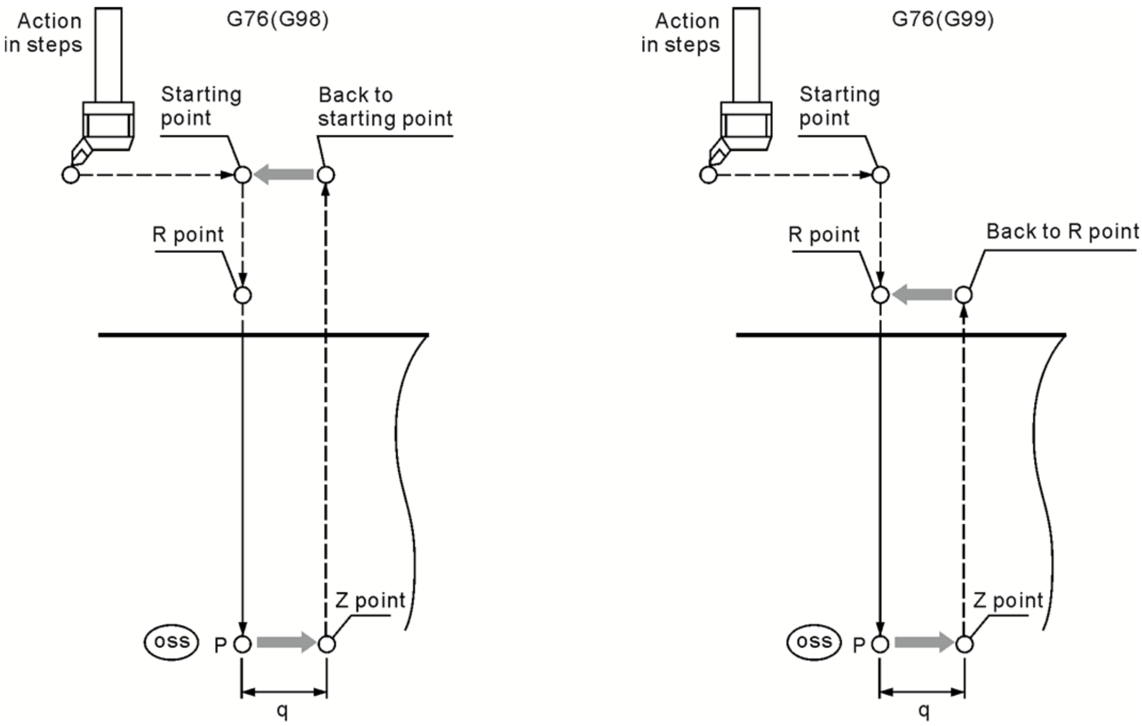


Figure 1: Fine boring cycle

2

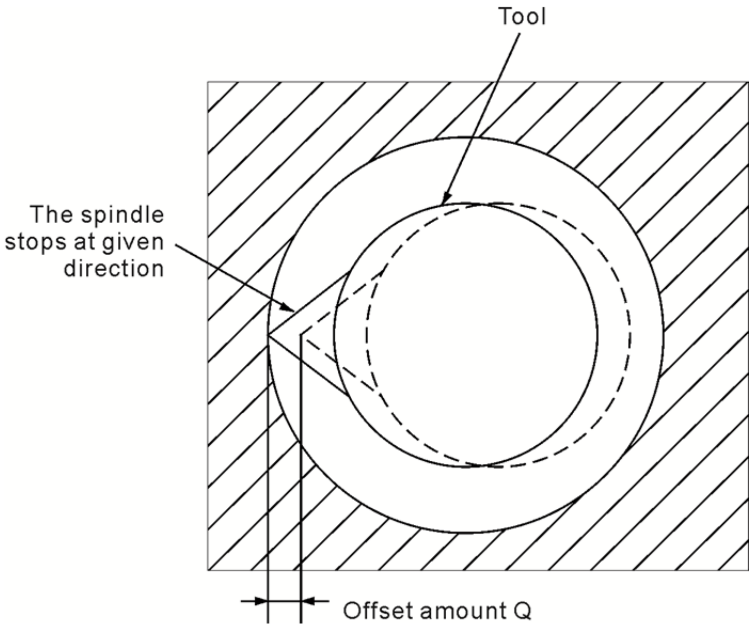


Figure 2: Offset amount for fine boring cycle

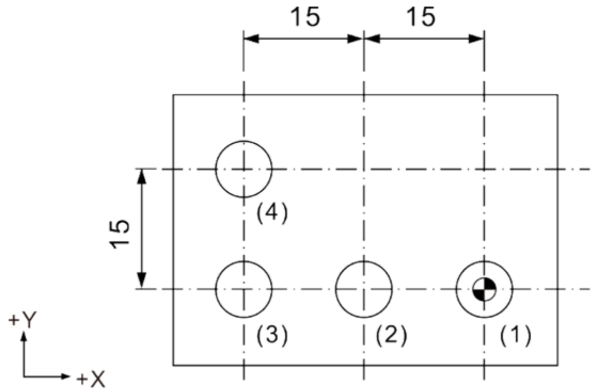
See below for movements set by this command (refer to Figure 1):
The firstly boring cutter fast positions to point (X, Y) and height R. Then, it cuts to depth Z with F feed rate before the spindle stops and positions. The cutter tip points to the positioning direction; offset the boring cutter for distance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

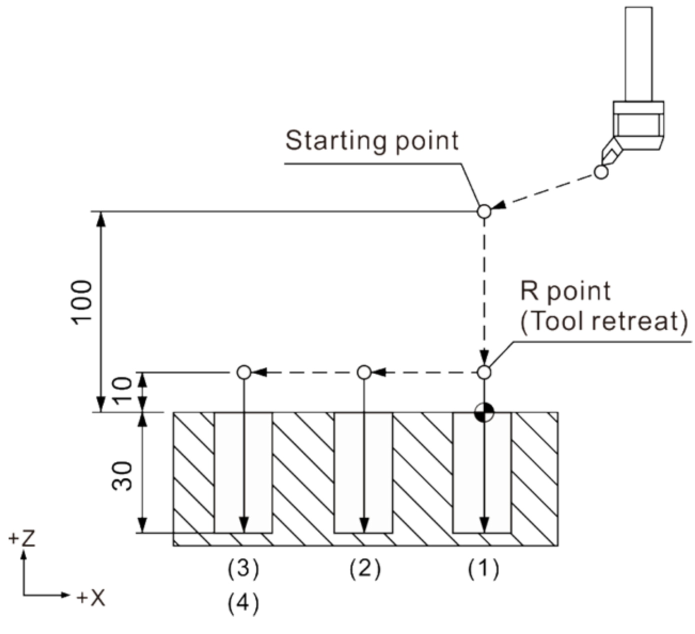
The offset amount shown in Figure 2 is given by parameter Q. The value of parameter Q must be positive (if a negative value is given its absolute value will be used instead; to offset 1.0 mm, set the parameter to Q1.0). The offset direction can be set by coordinates (+X, +Y) or (-X, -Y). Parameter Q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

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

[Example]

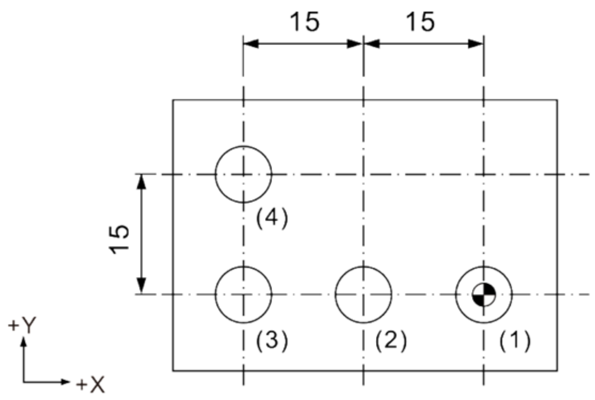
```
M03 S1000
G17 G90 G00 G54 X0. Y0.
G00 Z100.
G99 G76 X0. Y0. Z-30. R10. P1000 Q5. K1 F100.   -----------------------------------------------(1)
X-15.                                           -----------------------------------------------(2)
X-30.                                           -----------------------------------------------(3)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

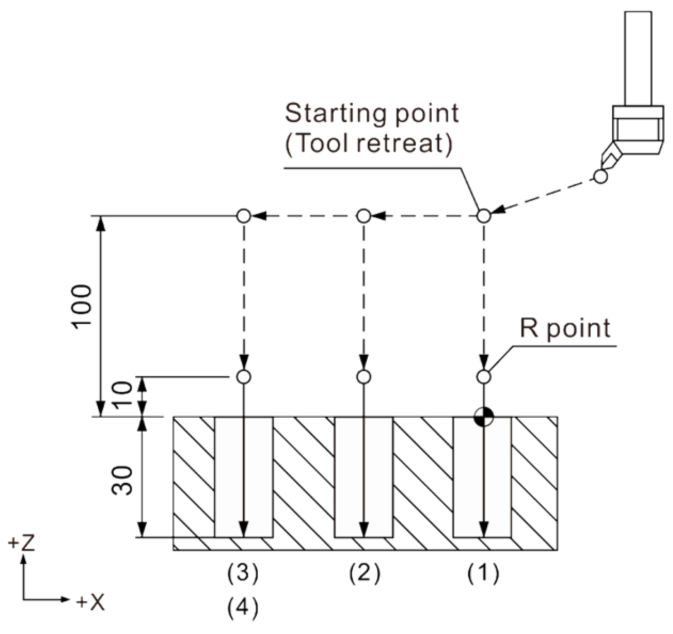




[Example]

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





2

G80: Cycle cancelling command

Format: G80

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

[Illustrations]

G17 G90 G00 G54 X0. Y0.

Z100.

G99 G73 X0. Y0. Z-20. R10. Q4. K1 F100.

G80

G17 G90 G00 G54 X0. Y0. ---------------(Cancel cycles set by command G73)

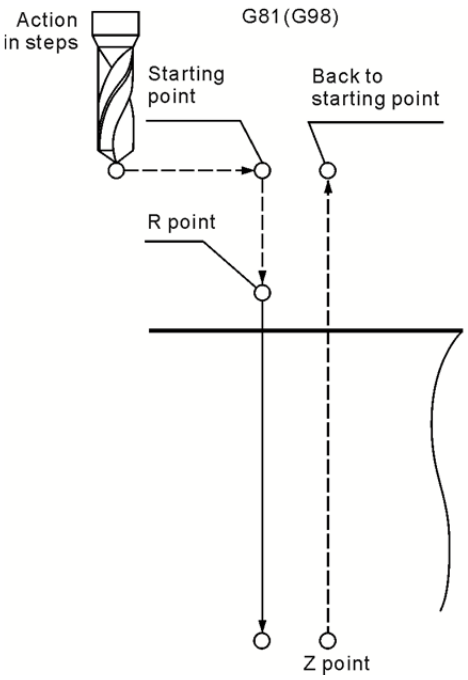
Z100.

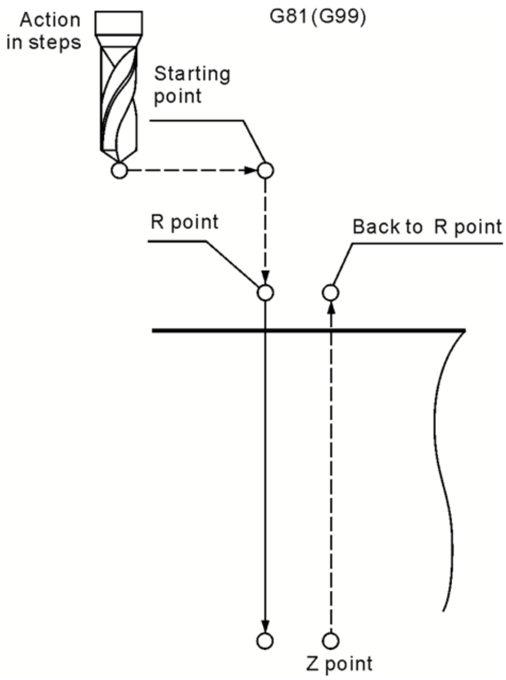
G81: Drilling cycle command

Format: G81 X_ Y_ Z_ R_ F_K_

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

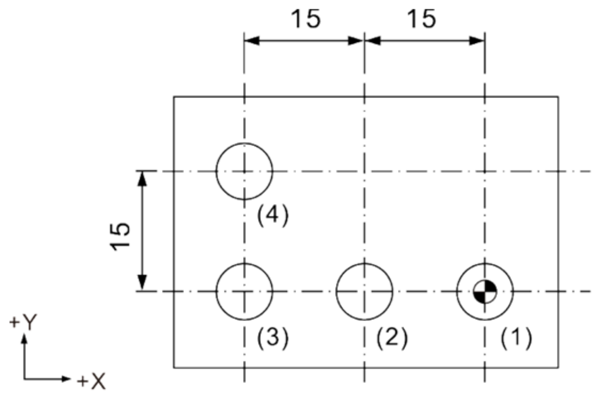
Description: Command G81 is used for drilling loops of general purpose. It drills to given depth Z in one shot without any retreat. The drilling cycle is completed by returning the cutter to point R or the initial point after being drilled to given depth Z with G00 movement. See the figure below for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

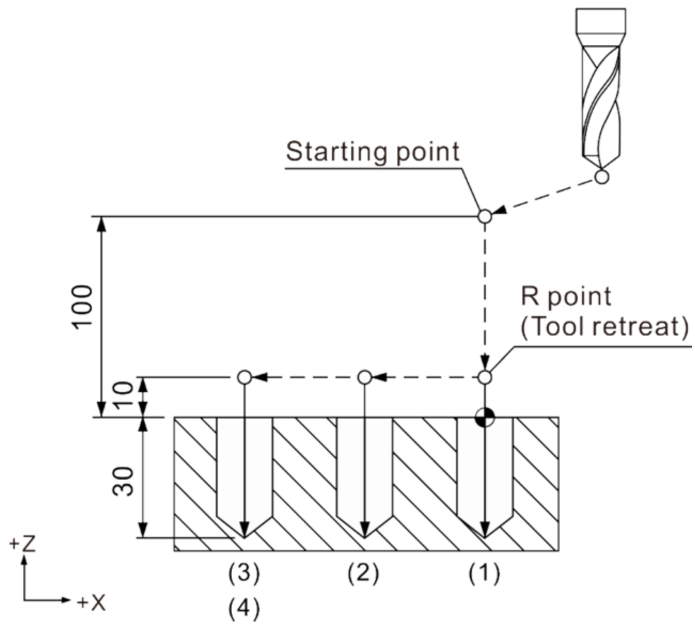




[Example]

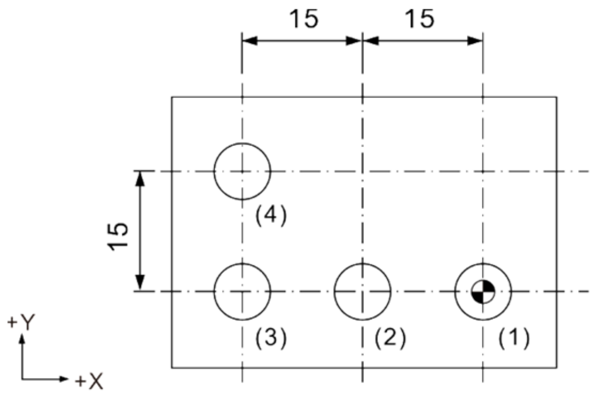
```
M03 S1000
G17 G90 G00 G54 X0. Y0.
G00 Z100.
G99 G81 X0. Y0. Z-30. R10. K1 F100.    ----------------------------------------------------------(1)
X-15.                                  ----------------------------------------------------------(2)
X-30.                                  ----------------------------------------------------------(3)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

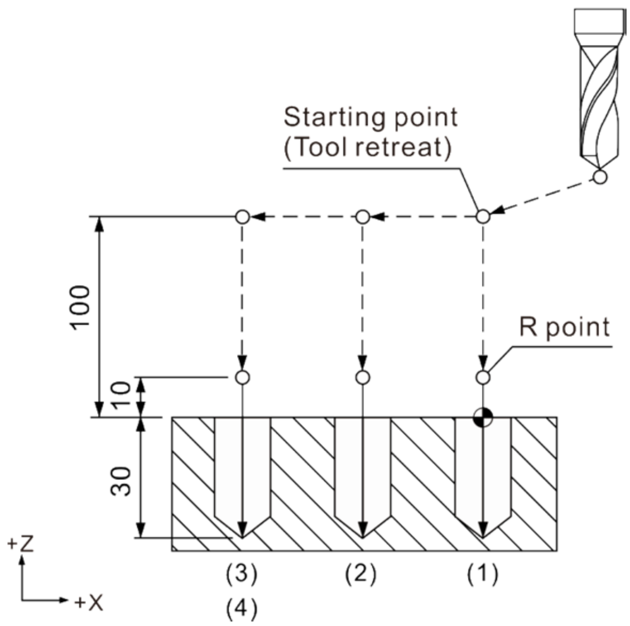




[Example]

```
M03 S1000
G17 G90 G00 G54 X0. Y0.
G00 Z100.
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
```



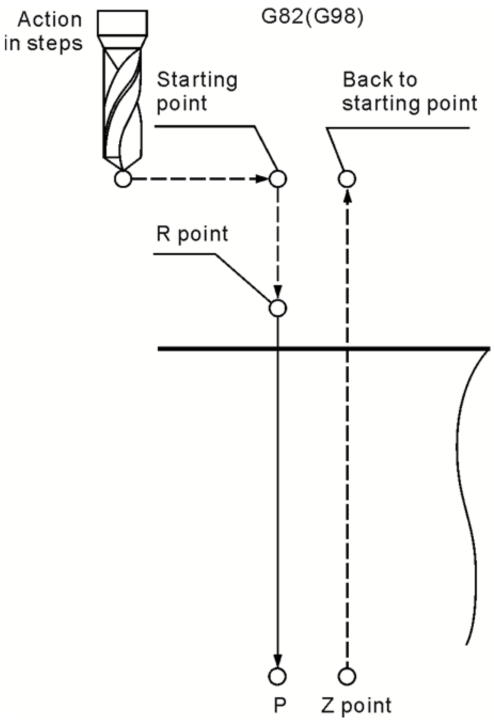


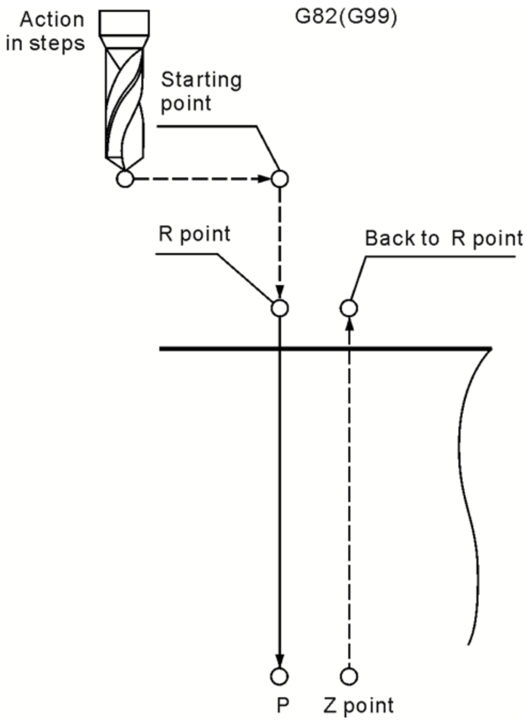
G82: Countersunk drilling cycle command

Format: G82 X_ Y_ R_ Z_ P_ F_K_

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

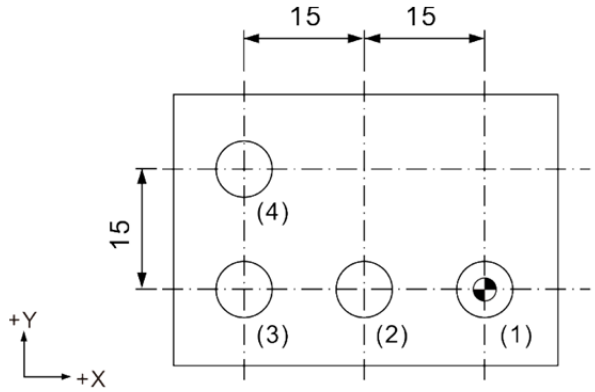
Description: Command G82 functions the same as command G81 except that the former can designate pause time P_ in a single block. That is, command G82 differs from G81 only in that it can set up a pause time P after cutting to the bottom of the hole. Command G82 is ideal for drilling counter bore, spot facing, and for machining holes that need precise depth. Command G82 can set the tool to stay within a specified time span after cutting to the bottom of the hole for a more smooth hole wall and accurate hole depth.

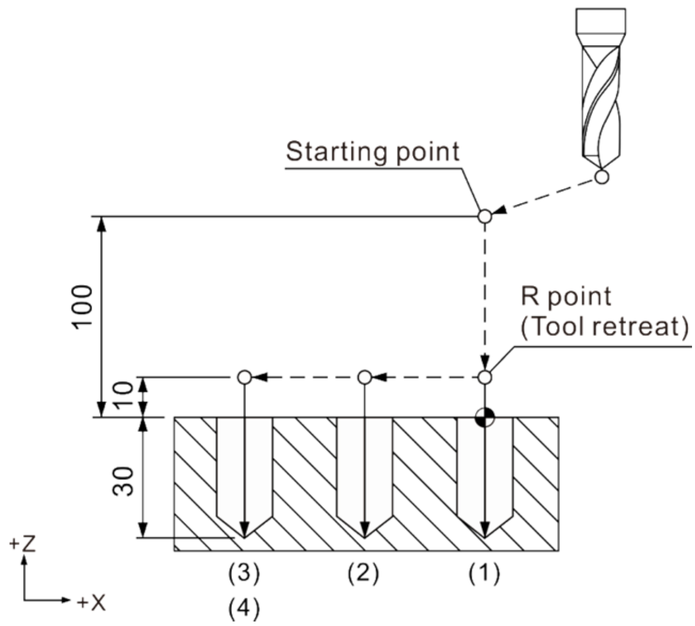




[Example]

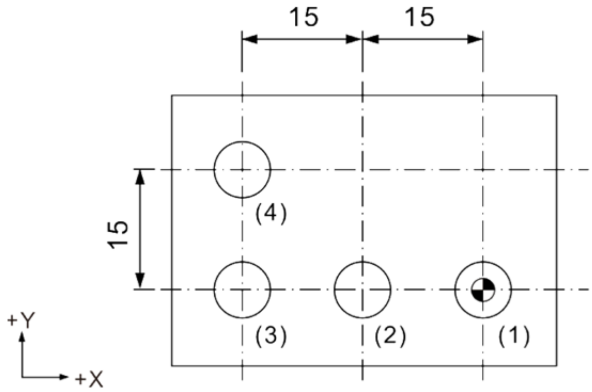
```
M03 S1000
G17 G90 G00 G54 X0. Y0.
G00 Z100.
G99 G82 X0. Y0. Z-30. R10. P1000 K1 F100.   ---------------------------------------------------------(1)
X-15.                                       ---------------------------------------------------------(2)
X-30.                                     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

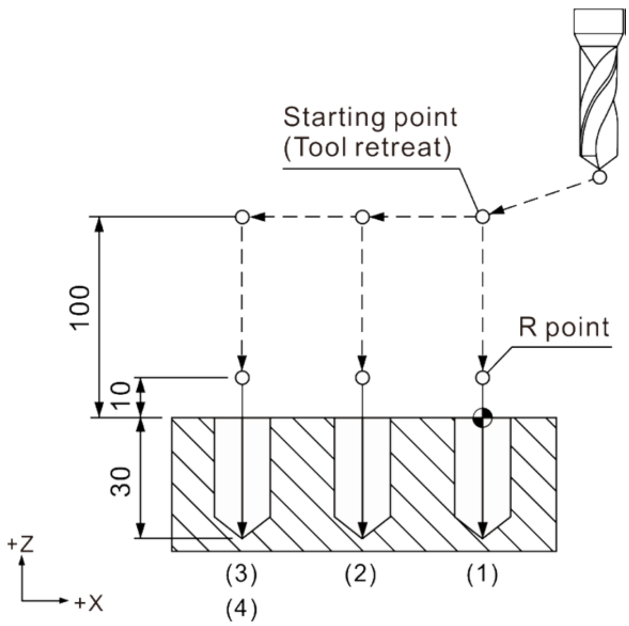




[Example]

```
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

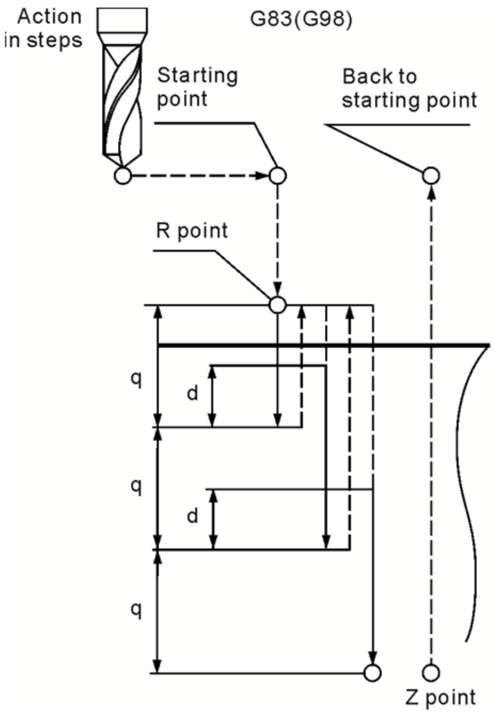
G83: Deep hole peck drilling cycle command

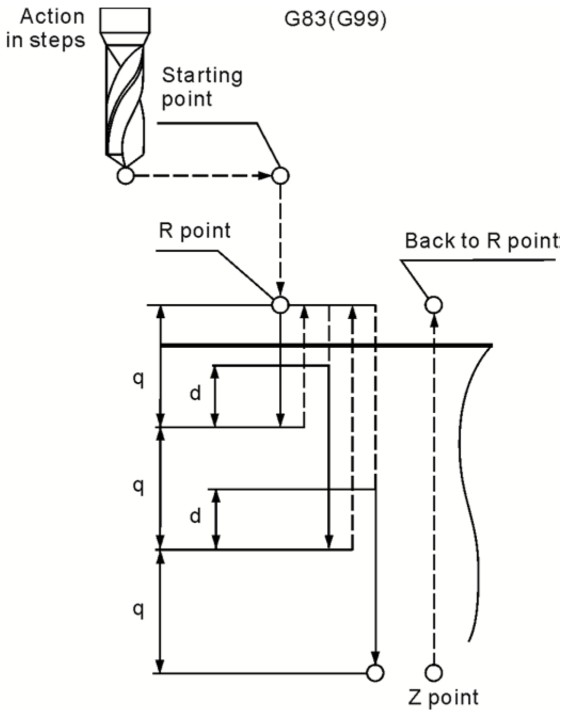
Format: G83 X_ Y_ R_ Q_ Z_ F_ K_

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

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

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

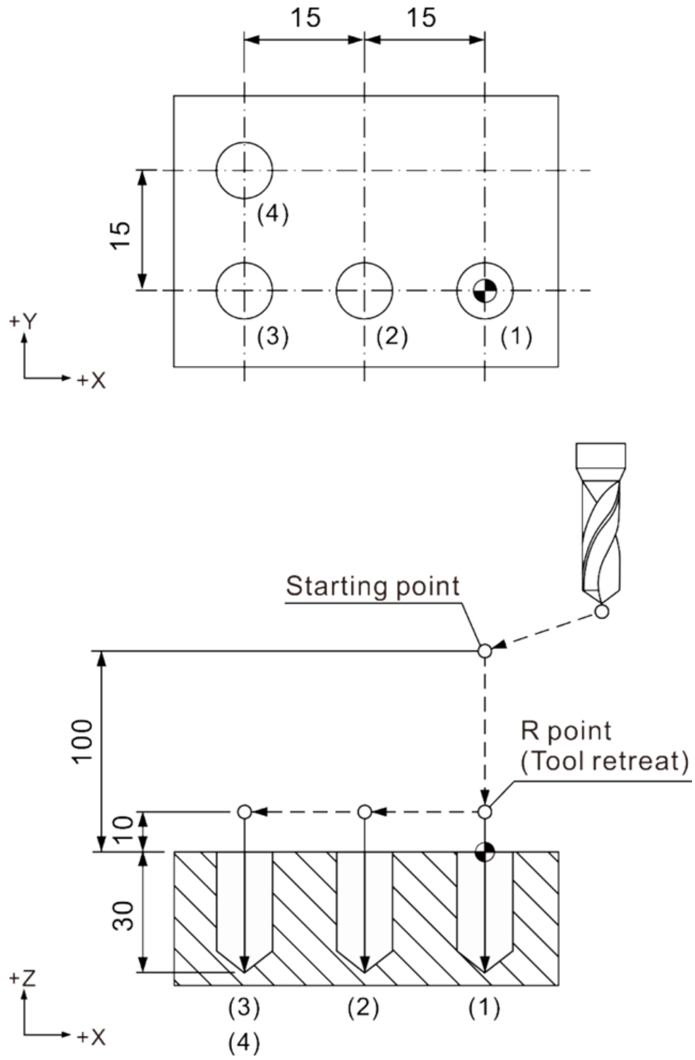




[Example]

```
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
```

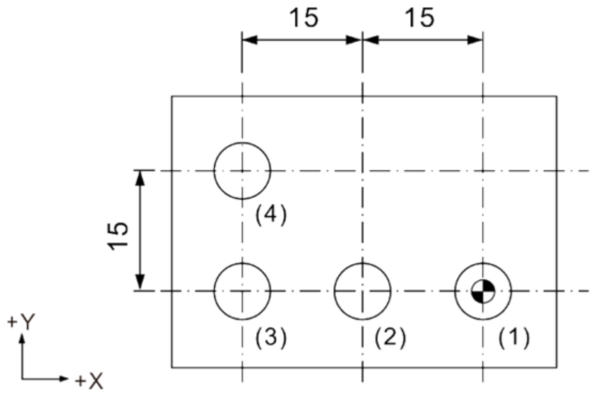
2

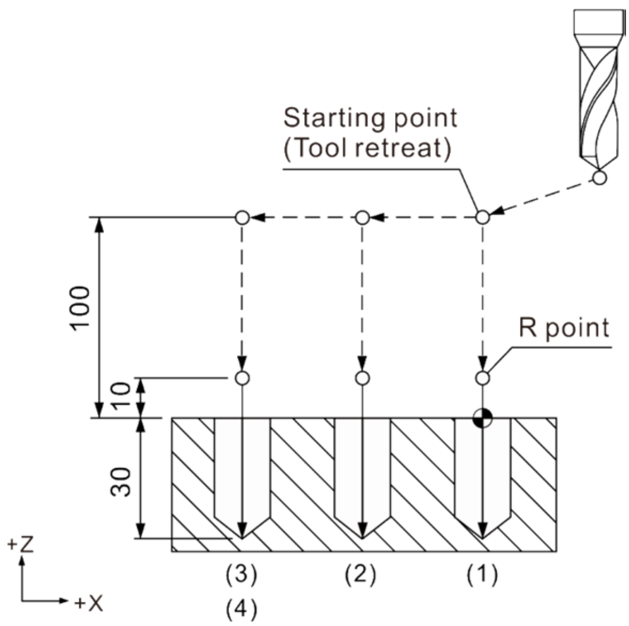


2

[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
```





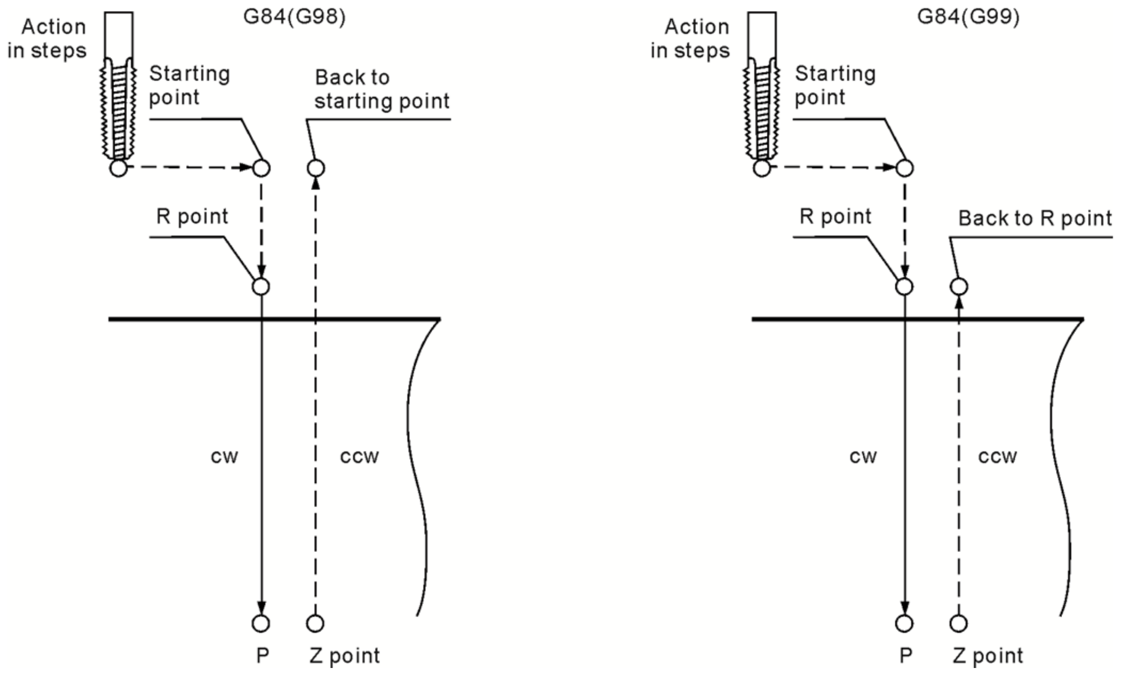
G84: Right spiral tapping cycle command

Format: G84 X_ Y_ R_Q_Z_ P_ F_ K_

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

2

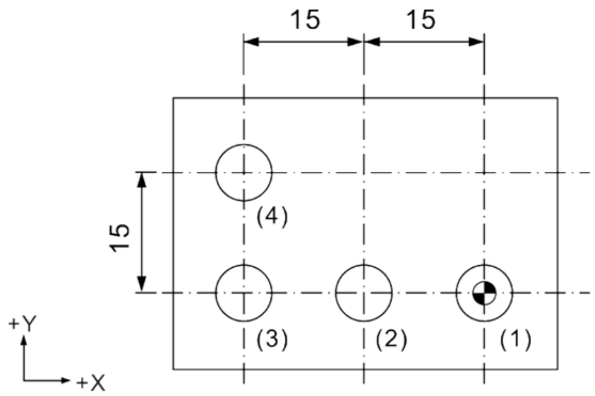
Description: Command G84 taps right-handed threads. It functions the same as command G74 except that the turning direction of G84 is opposite to that of command G74. All the remaining operations and notes are the same as command G74.

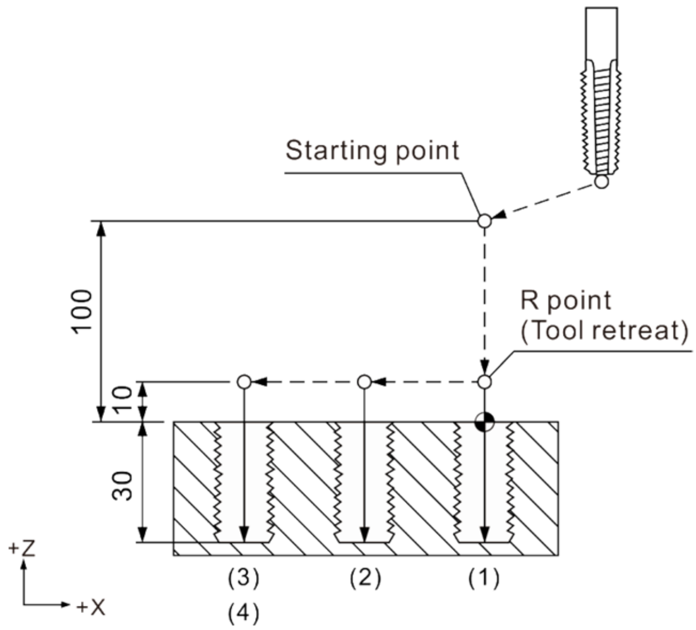


2

[Example]

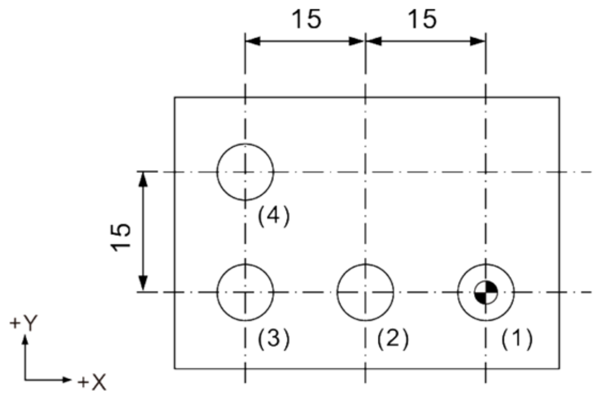
```
G17 G90 G00 G54 X0. Y0.
G00 Z100.
M29 S1000
G99 G84 X0. Y0. Z-30. R10. P1000 K1 F1000.    -----------------------------------------------------(1)
X-15.                                         -----------------------------------------------------(2)
X-30.                                         -----------------------------------------------------(3)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

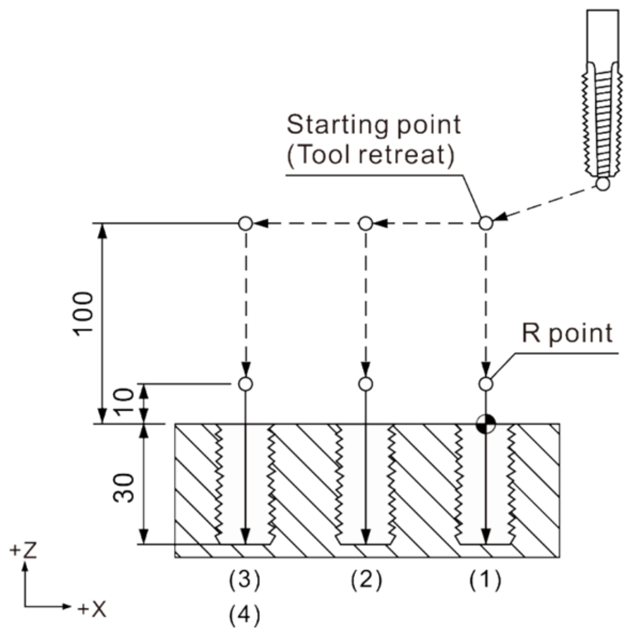




[Example]

```
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.
```





2

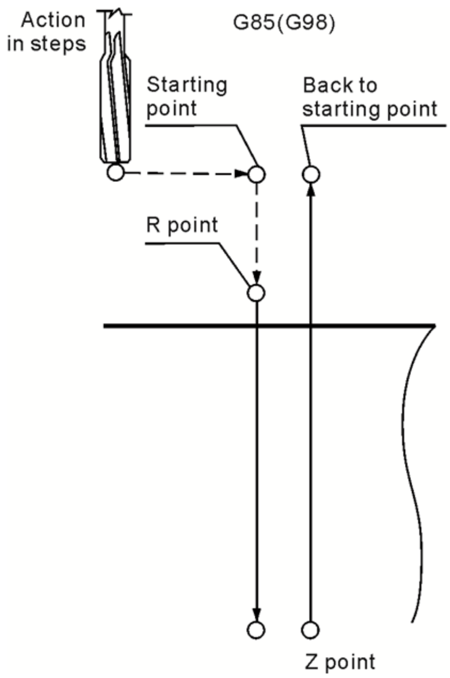
G85: Broaching cycle command

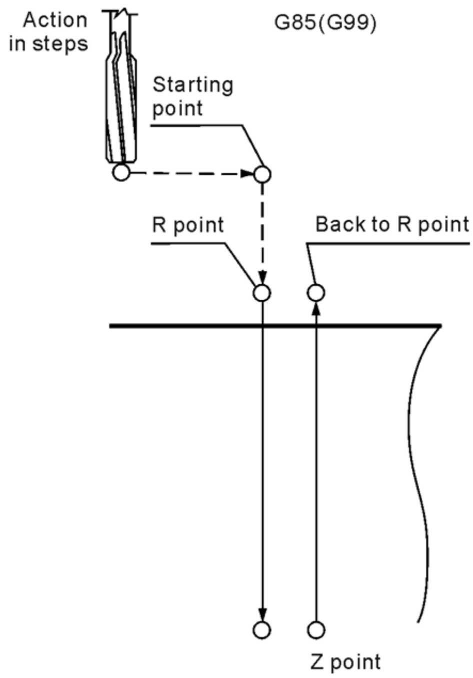
Format: G85 X_ Y_ R_ Z_ F_ K_

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

Description: Command G85 improves boring precision significantly. It usually works with reaming or boring cutter for holes with high reaming or boring accuracy.

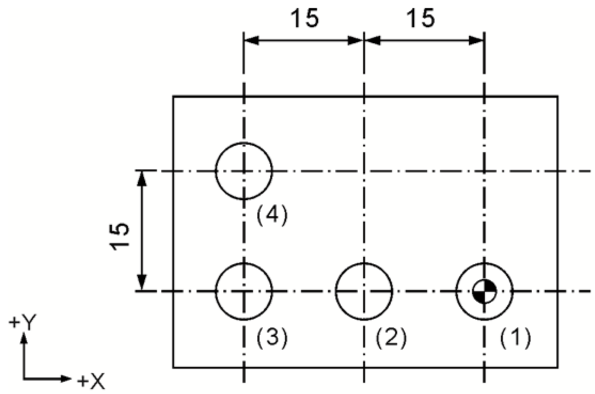
See below for the movements set by this command:
The tool cuts to given depth Z from point R in given feed speed F before lifting the up to point R at the same feed speed of F. Then, it returns to point R (G99) or initial point (G98).

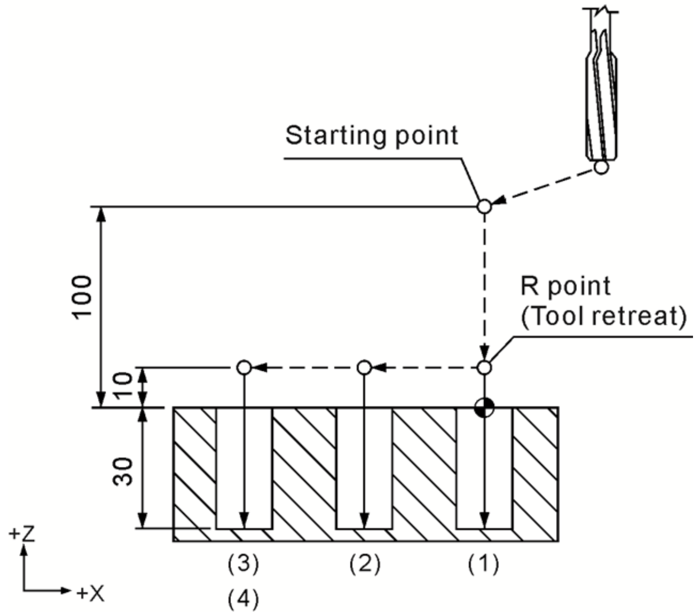




[Example]

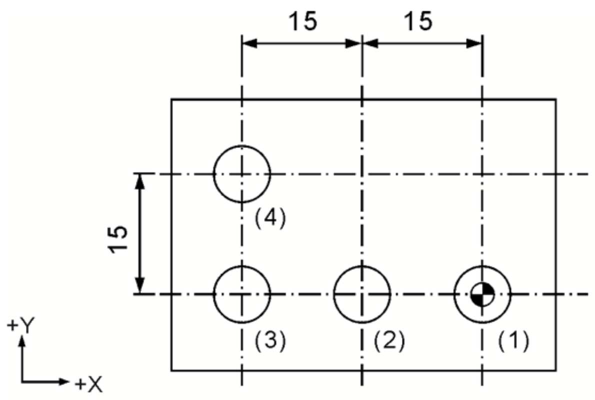
```
M03 S1000
G17 G90 G00 G54 X0. Y0.
G00 Z100.
G99 G85 X0. Y0. Z-30. R10. K1 F100.    -------------------------------------------------------(1)
X-15.                                  -------------------------------------------------------(2)
X-30.                                  -------------------------------------------------------(3)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

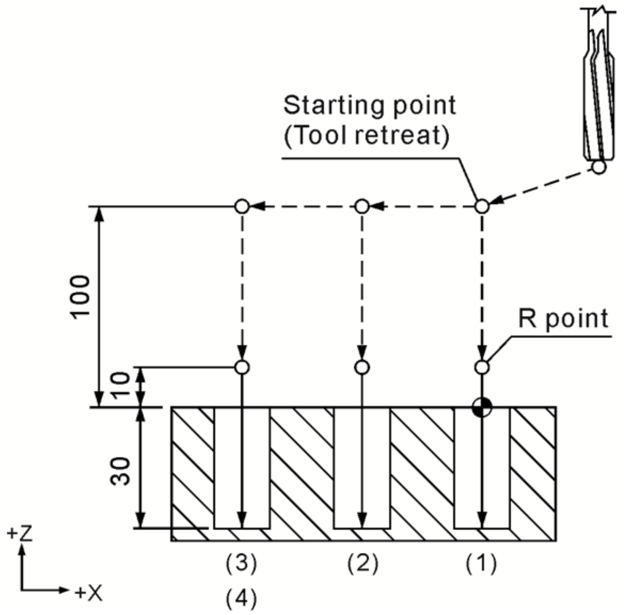




[Example]

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





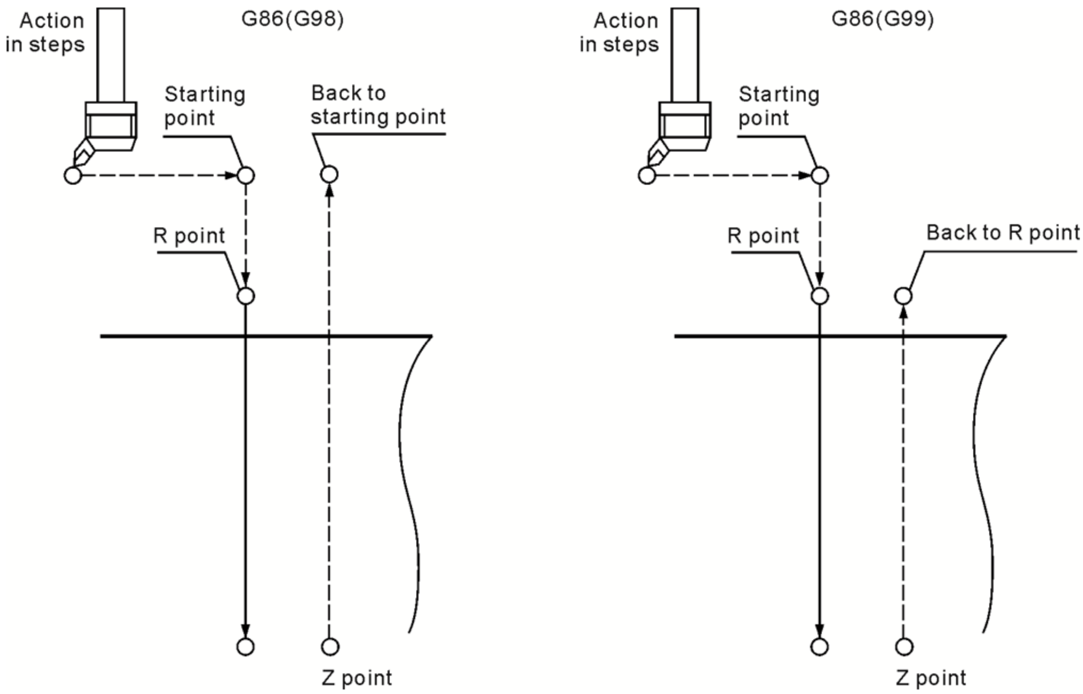
2

G86: Rough boring cycle command

Format: G86 X_ Y_ R_ Z_ F_ K_

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

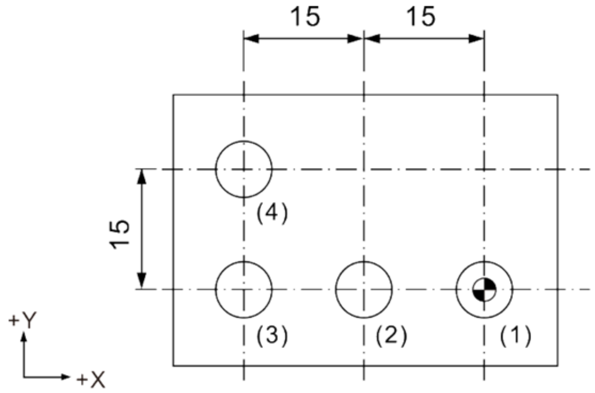
Description: See the figure below for the motions set by this command. After bored to given depth Z, the spindle stops turning and lifts to the initial height with fast feeding command G00. A full boring cycle is completed after the spindle returns to the initial height and resumes positive direction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

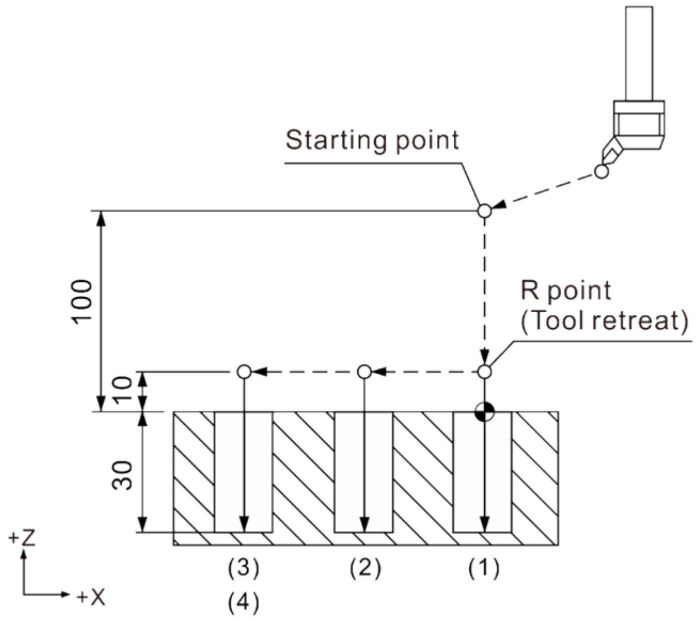


2

[Example]

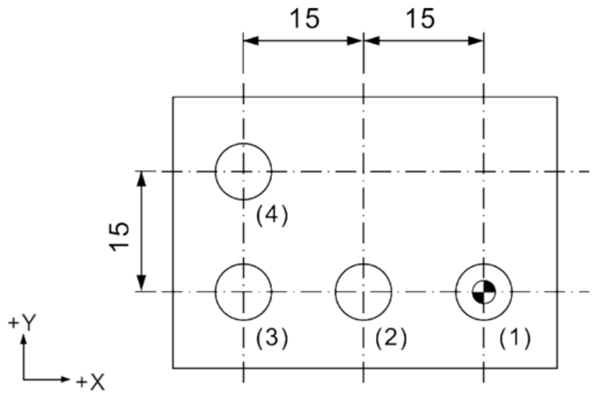
```
M03 S1000
G17 G90 G00 G54 X0. Y0.
G00 Z100.
G99 G86 X0. Y0. Z-30. R10. K1 F100.   ----------------------------------------------------(1)
X-15.                                 ----------------------------------------------------(2)
X-30.                                 ----------------------------------------------------(3)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

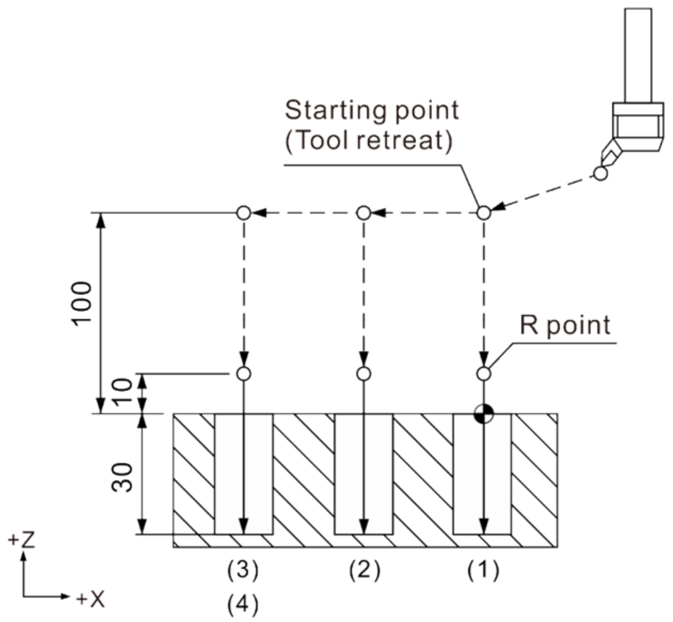




[Illustrations]

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





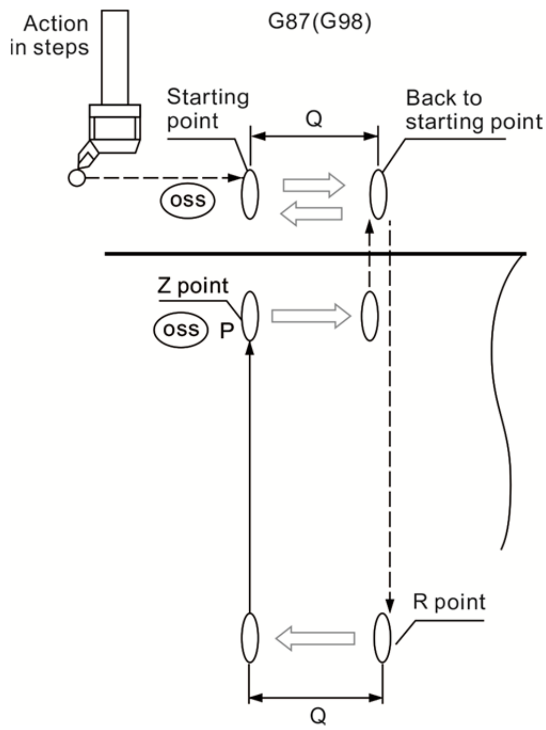
2

G87: Rear boring cycle command

Format: G87 X_ Y_ R_ Z_ Q_ P_ F_ K_

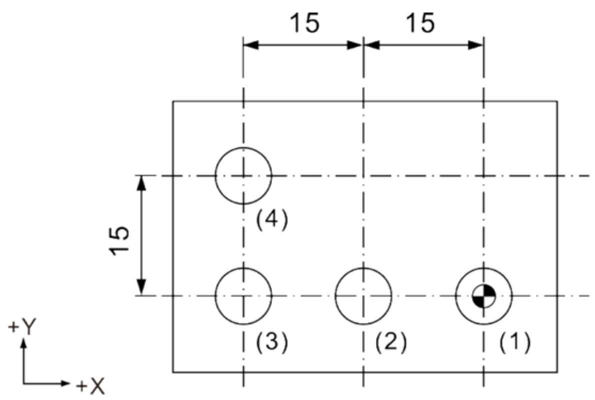
X_Y_: Ending position of single block
Z_: Bottom of hole to be cut
R_: Initial safety height
Q_: Offset distance
P_: Pause time (in the least unit of 1/1000 second), without decimal point.
F_: Feed rate
K_: Number of cycles

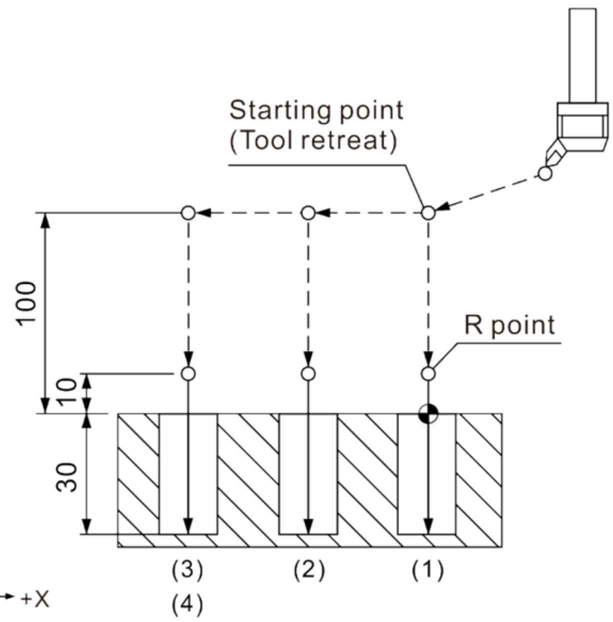
Description: The tool fast positions to the position set by coordinates (X, Y) before the spindle moves to the position the cutter tip in a given direction. The tool center then offsets a distance of value Q. In this way, the cutter blade does not contact the hole wall when moving to height R. After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



[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
```



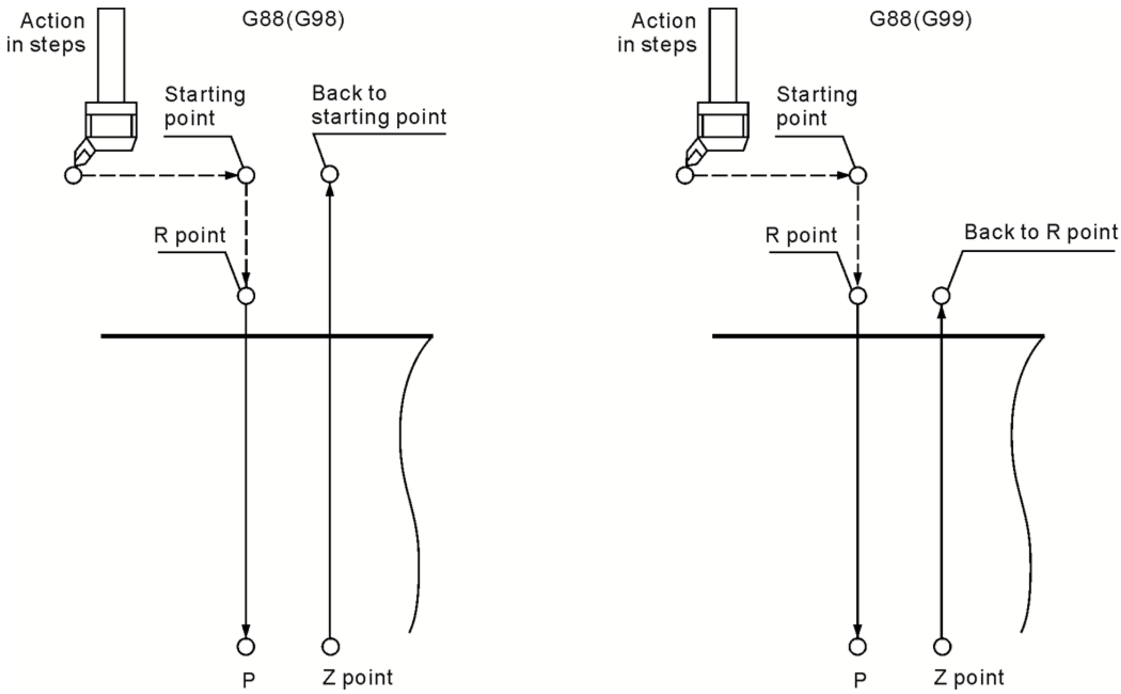
2

G88: Boring cycle command

Format: G88 X_ Y_ R_ Z_ P_ F_ K_

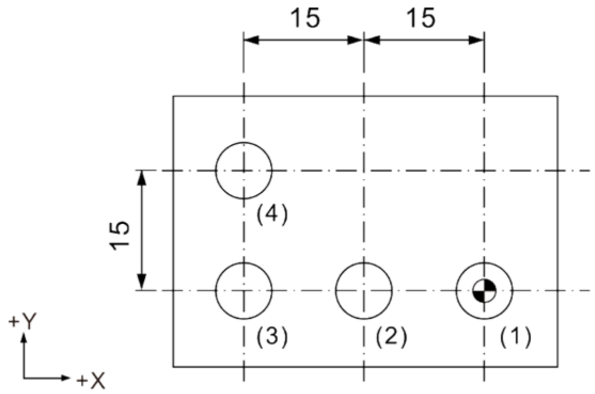
X_Y_: Ending position of single block
Z_: Bottom of hole to be cut
R_: Initial safety height
P_: Pause time (in the least unit of 1/1000 second), without decimal point.
F_: Feed rate
K_: Number of cycles

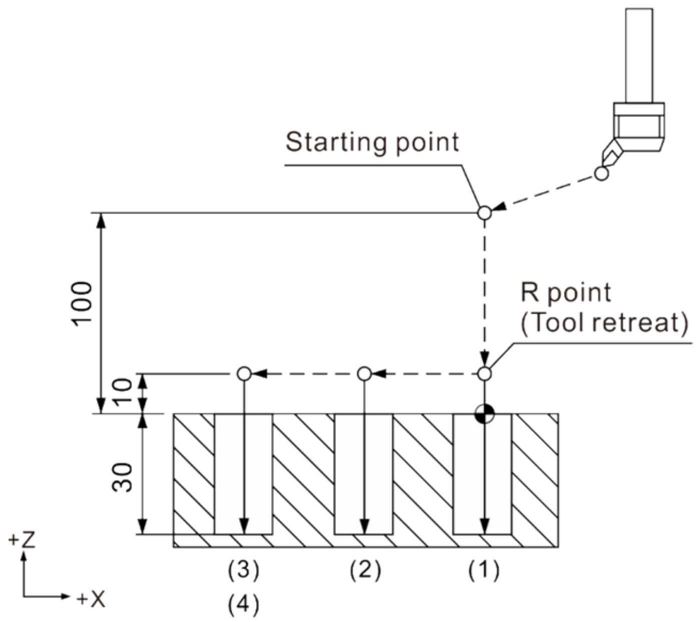
Description: Command G88 sets the tool to cut from point R to given point Z and pause at point Z for the time specified in parameter P. The spindle stops turning and execution (same as executing command M00) after a pause period and fast retreats the tool to point R or the initial point only after the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



[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
```

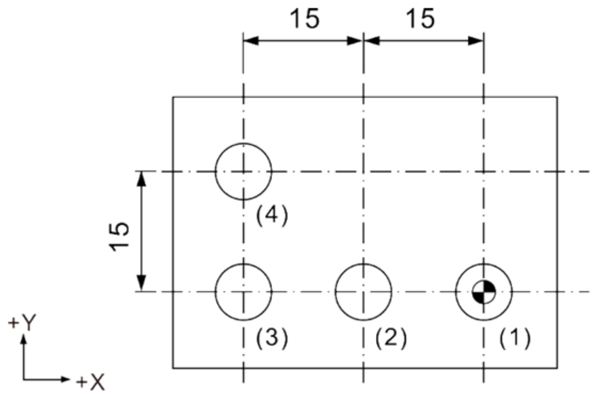


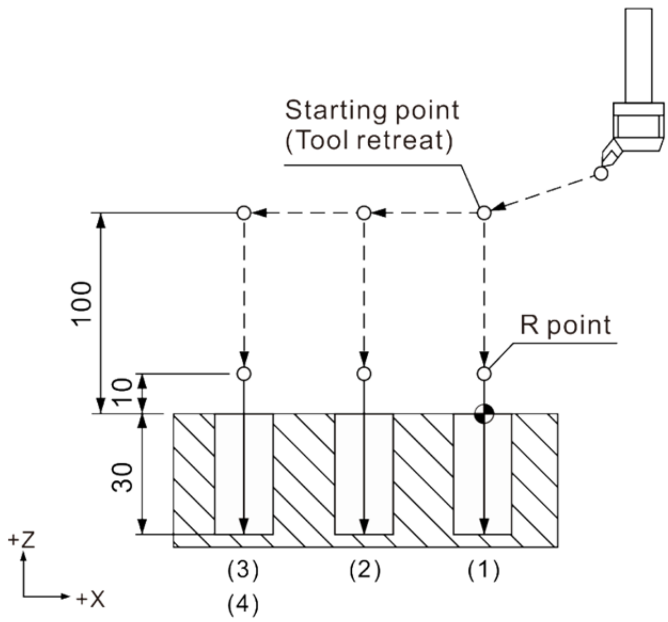


2

[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
```



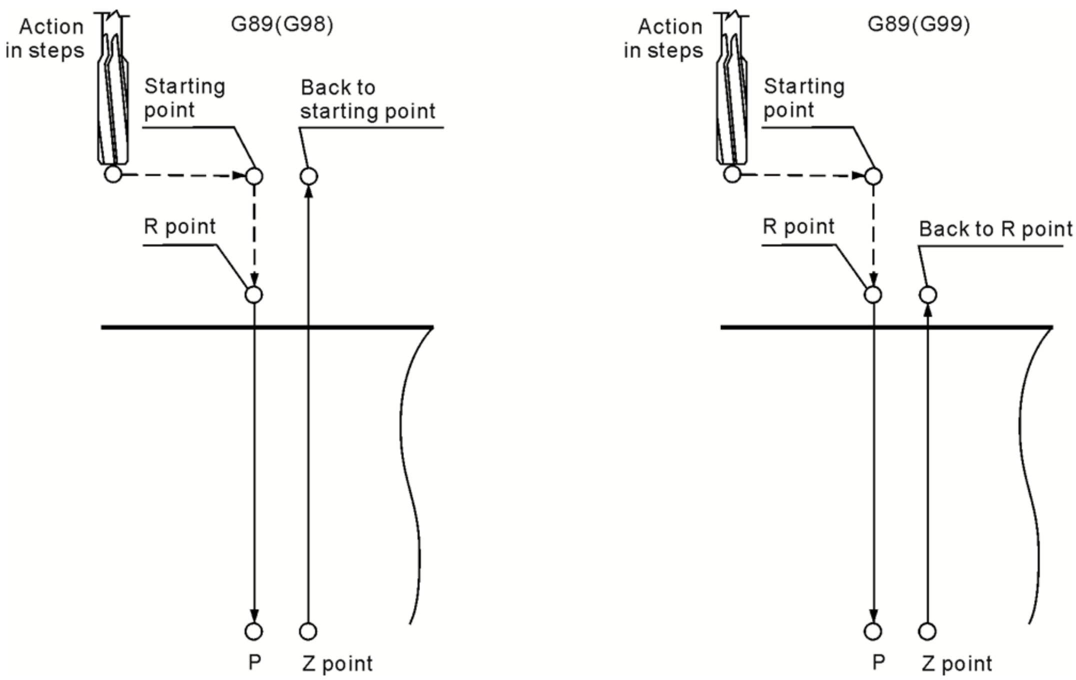


G89: Boring cycle command

Format: G89 X_ Y_ R_ Z_ P_ F_ K_

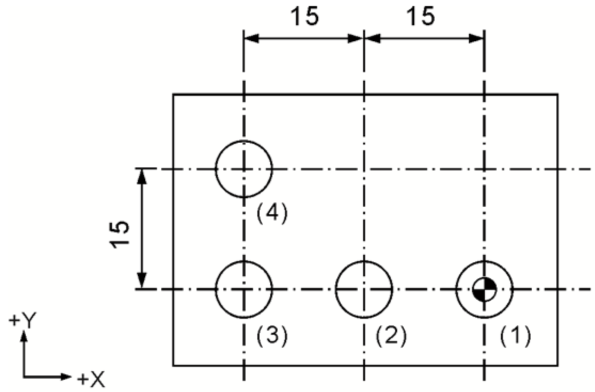
X_Y_: Ending position of single block
Z_: Bottom of hole to be cut
R_: Initial safety height
P_: Pause time (in the least unit of 1/1000 second), without decimal point.
F_: Feed rate
K_: Number of cycles

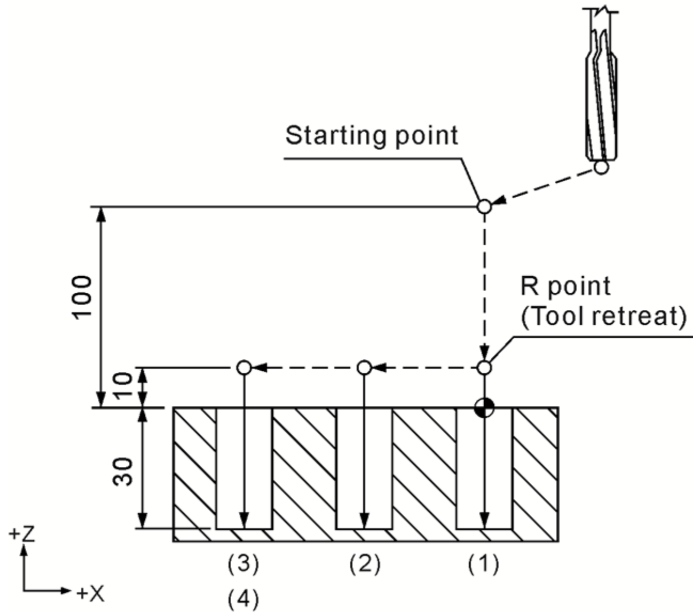
Description: Command G89 is for blind hole reaming. It functions the same as command G85 except that a pause P can be added at point Z. Both its downward and upward movement of the tool in the Z-axis is in F feed rate. With the added pause at point Z, the machining accuracy for depth and diameter of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



[Example]

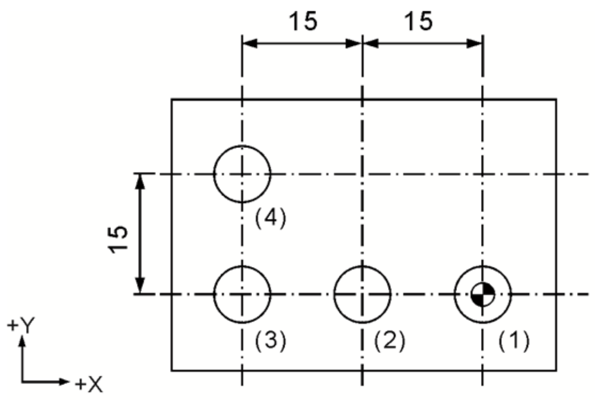
```
M03 S1000
G17 G90 G00 G54 X0. Y0.
G00 Z100.
G99 G89 X0. Y0. Z-30. R10. P1000 K1 F100.    ---------------------------------------------------(1)
X-15.                                         ---------------------------------------------------(2)
X-30.                                         ---------------------------------------------------(3)
X-30. Y15.                                    ---------------------------------------------------(4)
G80 G91 G28 X0. Y0. Z0.
M05
```

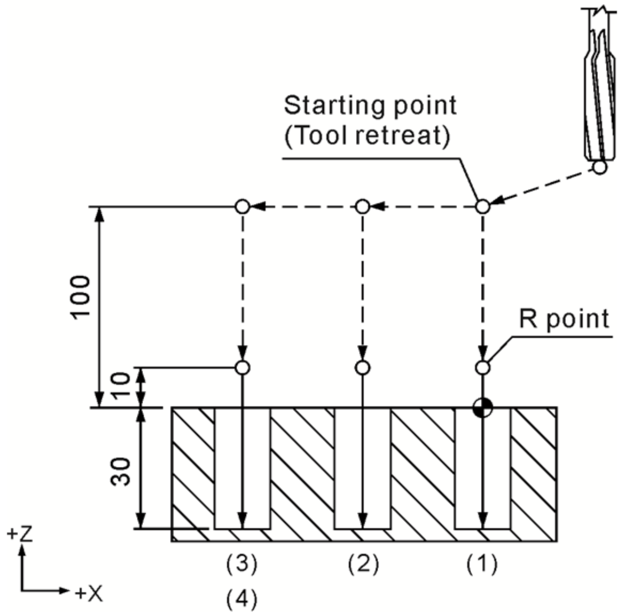




[Example]

```
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

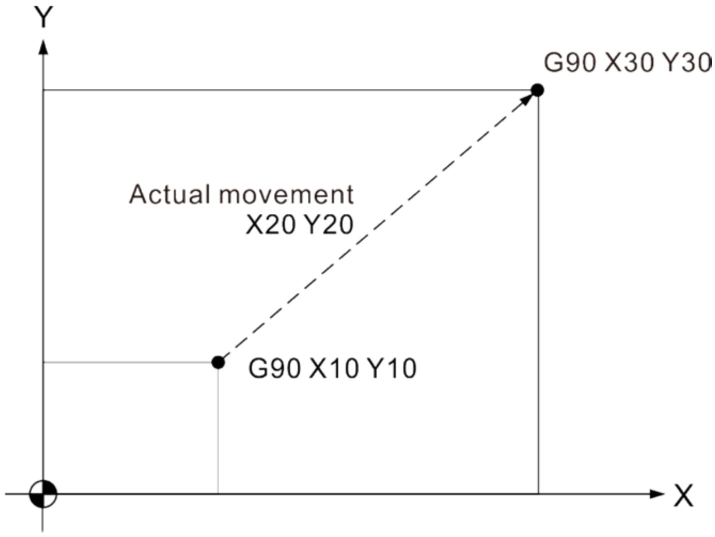
G90: Absolute coordinates command

Format: G90 X_ Y_ Z_

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

[Example]

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




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

G91: Incremental coordinates command

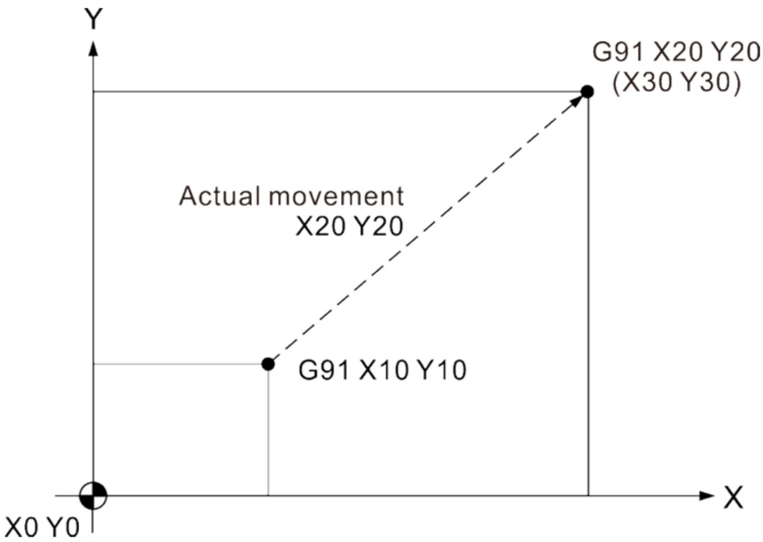
Format: G91 X_ Y_ Z_

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

2

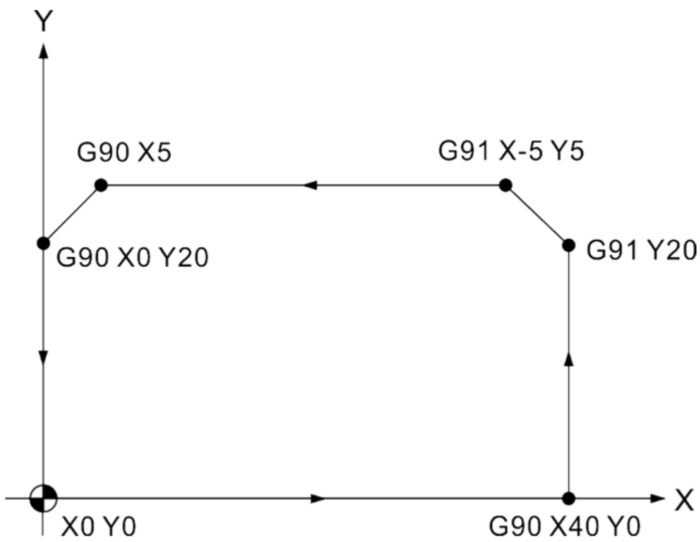
[Example]

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




```
O1001
G01 G91 X10.0 Y10.0
F500.0
X20.0 Y20.0
```

Example of using G90 and G91 commands together:




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

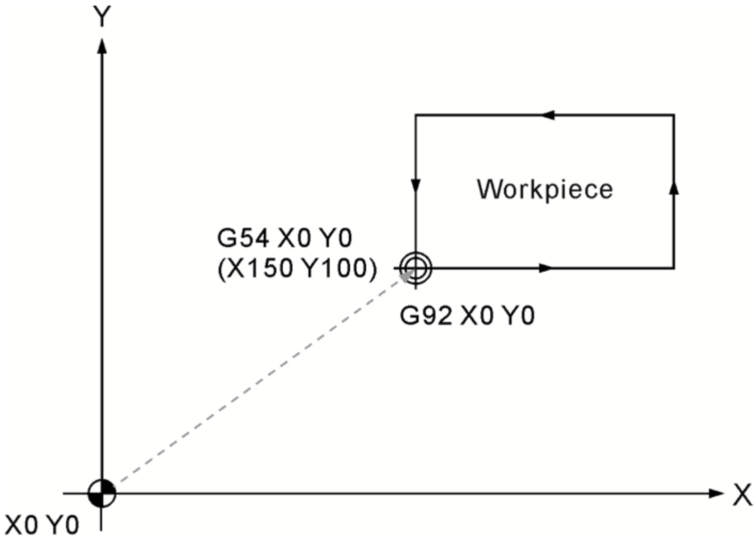
G92: Coordinate system setup command

Format: G92 X_ Y_ Z_

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

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

[Example]

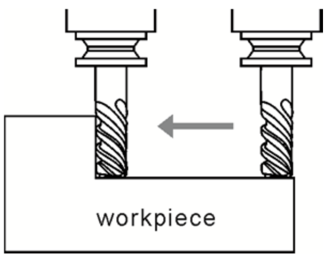


O0005
G00 G90 G54 X0.0 Y0.0
G00 G92 X0.0 Y0.0
G01 X100.0 Y0.0 F1000.0
X100.0 Y50.0
X0.0 Y50.0
X0.0 Y0.0
M30

G94: Feed rate (mm/min) setup command

Format: G94 G01 X_ Y_ Z_ F_

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



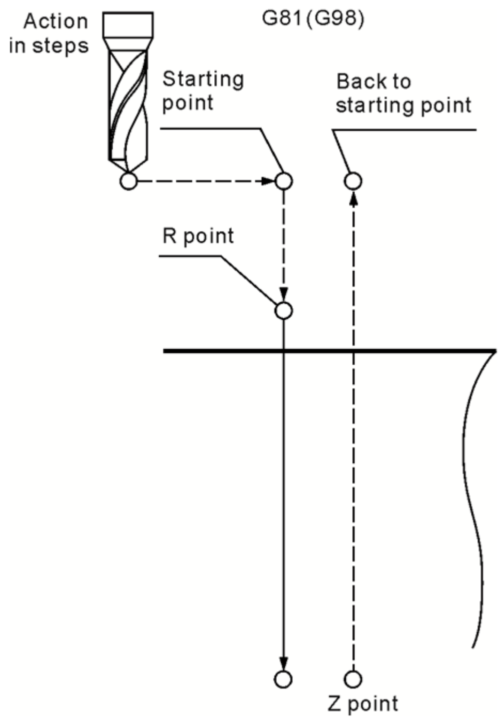
Feed rate in unit of maintenance/min or inch/min.

2

G98: Return to the initial point of the fixed cycle

Format: G98 G8_ X_ Y_ Z_ R_ F_

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

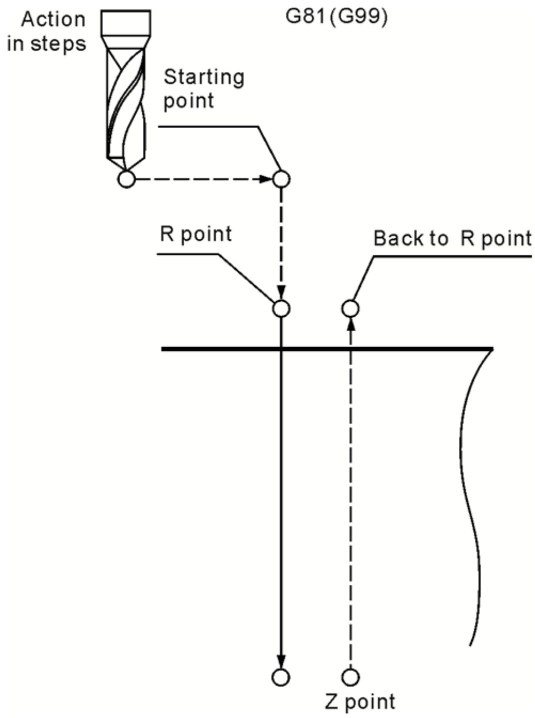


2

G99: Return to the R point of the fixed cycle

Format: G99 G8_ X_ Y_ Z_ R_ F_

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



2

2.2 G code for turning system

Absolute / Incremental command

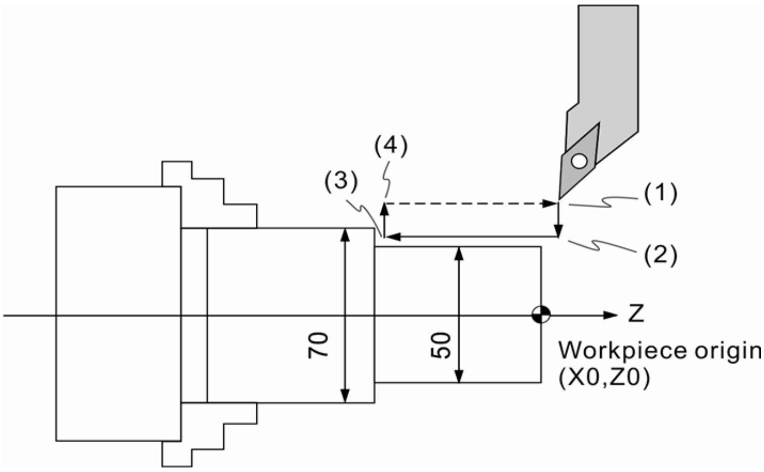
Description: The motion command for turning system can be specified by absolute and incremental values. The absolute coordinates in the command are specified based on the origin of the workpiece coordinates. When incremental command is used, the motion will be made based on the current coordinates plus the increments. Both absolute and incremental values can be set in the same block.

Absolute command		Incremental command	
Code	Corresponding axis	Code	Corresponding axis
X__	X-axis	U__	X-axis
Y__	Y-axis	V__	Y-axis
Z__	Z-axis	W__	Z-axis

Diameter / Radius command

Description: As the workpieces for turning are mostly in cylinder shape, the motion command of X-axis can be specified with diameter or radius. When setting with a diameter, the actual moving distance is 1/2 of the command value. On the other hand, if a radius is set, the actual moving distance will be equal to the command value.
The selection of diameter and radius can also be set by P306.
(1) Set 0 to set with diameter (default).
(2) Set 1 to set with radius.

[Example]
Assume the workpiece is a cylinder bar with diameter of 70 mm. After turning, the diameter has to be 50 mm. The commands specified with diameter and radius are shown as follows.



Setting with diameter:

T0303 // Select tool No.3 and compensation No.3.

M3S1000 //Spindle rotates forward at 1000 rpm.

G0 X72. Z2. // Move to point (1) in rapid traverse.

G1 X50. // Turn to point (2)), where diameter is 50 mm.

Z-40. // Turn to point (3) (-40 mm) in Z-axis direction.

X70. // Retract to point (4), where diameter is 70 mm in X-axis direction.

G0 X72. Z5. //Go to the tool change point in rapid traverse.

M5 // Spindle stop

M30 //Program end

Setting with radius:

T0303 // Select tool No.3 and compensation No.3

M3S1000 // Spindle rotates forward at 1000 rpm

G0 X36. Z2. // Move to point (1) in rapid traverse.

G1 X25. // Turn to point (2), where diameter is 50 mm.

Z-40. // Turn to point (3) (-40 mm) in Z-axis direction.

X70. // Retract to point (4), where diameter is 70 mm in X-axis direction.

G0 X36. Z5. // Go to the tool change point in rapid traverse.

M5 // Spindle stop

M30 // Program end

2

G00: Fast positioning command

Format: G00 X/U_Y/V_Z/W_(This command I applicable to three-axis, two-axis synchronous control, and single-axis control.)

X/U_Y/V_Z/W_: Coordinates of the end point

Description: The G00 command quickly moves the center of the tool to the specified coordinate position (X, Y, Z). When G00 is applied, the moving speed is not specified by value F_ in the command; instead, it is controlled by the **Rapid %** key on the secondary control panel.

If the max. moving speed of X-, Y-, and Z-axis (parameter 316) is set to 15 m/min,
(1) When the rapid feed rate is set to 100%, the moving speed will be 15 m/min.
(2) When the rapid feed rate is set to 50%, the moving speed will be 7.5 m/min.
(3) When the rapid feed rate is set to 25%, the moving speed will be 3.75 m/min.
(4) When the rapid feed rate is set to 0%, the axial moving speed will be the speed set by parameter 315.

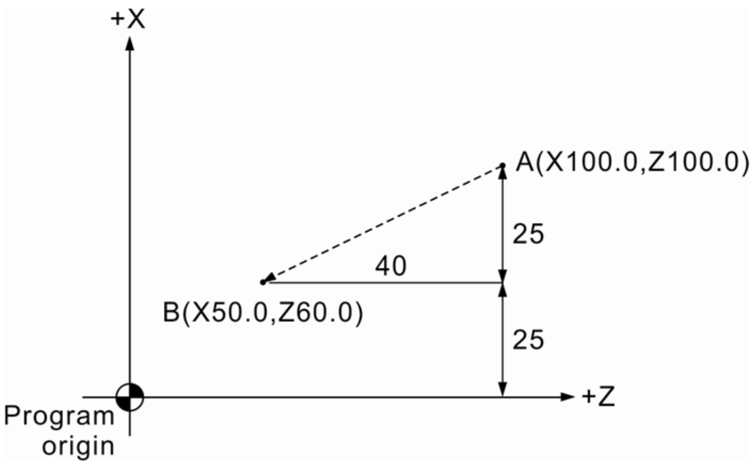
G00 is mainly for fast positioning instead of cut feeding. It can be used in the circumstances such as positioning at start cutting point from the mechanical origin in rapid traverse or application for retraction and fast positioning of Z- and X-axis after cutting.

[Example]

The following figure illustrates the usage of G00. The tool moves from A and positioned at B in rapid traverse.

Expressed with absolute value: X100. Z100. (Coordinates of point A)
G00 X50.0 Z60.0 (Coordinates of point B)

Expressed with incremental value: X100. Z100. (Coordinates of point A)
G00 U25.0 W-40.0 (Increments to point B)



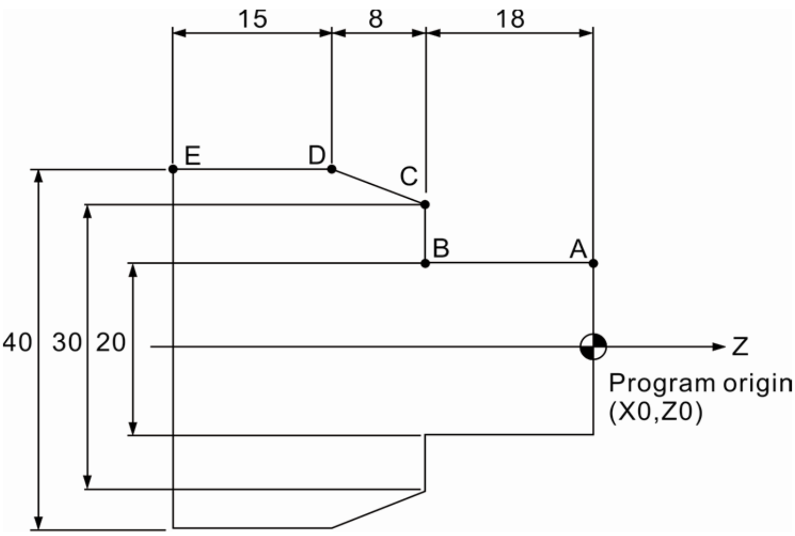
G01: Linear interpolation command

Format: G01 X/U_Y/V_Z/W_F_

X/U_Y/V_Z/W_: Coordinates of the end point
F_: Feed rate in unit of mm/min or min/rev

Description: This command enables a cutter to make linear cutting from the current position to a given position at F feed rate. X, Y, Z coordinates stands for the end point of cutting. The movement can be three-axis synchronous, two-axis synchronous or single axis movement.
The feed rate is set by the F parameter in unit of mm/min along with the **Rapid %** key of the secondary control panel. Its unit can also be changed by G98 (mm/min) and G99 (min/rev).

[Example]



G98	(Spindle feeding mode is set to “mm/min”)
G54 X0.0 Z0.0;	(Program origin)
G00 X20.0;	(Go to point A in rapid traverse)
G01 Z-18.0 F500;	(From point A to B)
X30.0;	(From point B to C)
X40.0 Z-26.0;	(From point C to D)
Z-41.0;	(From point D to E)

As the F command is continuous effective, the next block is ignored when the cutting speed is the same. See the program above.

2

G02/G03: Arc interpolation command

Format: Arcs in the X - Y plane:

G17 G02 (G03) X/U_ Y/V_ R_ F_
G17 G02 (G03) X/U_ Y/V_ I_ J_ F_
You may add parameter Z_ in the command to allow the tool to move spirally in the X - Y plane.

Arcs in the Z - X plane
G18 G02 (G03) Z/W_ X/U_ R_ F_
G18 G02 (G03) Z/W_ X/U_ K_ I_ F_
You may add parameter Y_ in the command to allow the tool to move spirally in the Z - X plane.

Arcs in the Y - Z plane
G19 G02 (G03) Y/V_ Z/W_ R_ F_
G19 G02 (G03) Y/V_ Z/W_ J_ K_ F_
You may add parameter X_ in the command to allow the tool to move spirally in the Y - Z plane.

G02: Clockwise (CW) arc interpolation.
G03: Counterclockwise (CCW) arc interpolation.

X/U, Y/V, and Z/W: Coordinates of the end point in absolute or incremental values
R: Arc radius (expressed in Radius format).

I: Distance between center and starting point in X-axis direction. That is, the increments from the starting point to center.

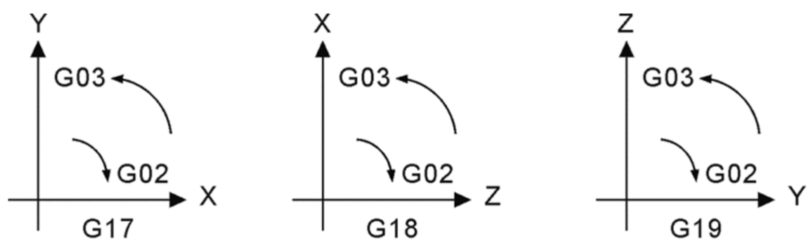
J: Distance between arc center and starting point in Y-axis direction. That is, the increments from the starting point to center.

K: Distance between arc center and starting point in Z-axis direction. That is, the increments from the starting point to center.

(The I, J, and K expression is also known as the center method.)

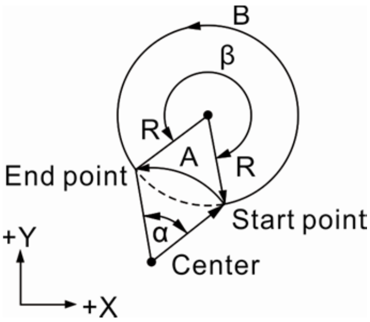
F: The cutting feed rate.

Description: G02 or G03 is the arc interpolation command. The arc interpolation direction (G02 or G03) for a three-dimensional workpiece in individual planes is shown in the figure below. G02 is for clockwise direction while G03 is for counterclockwise.



The center and radius methods are described below:

1. Radius method: R is the radius of the arc and shows in radius value. In this method, arc is made according to the start point, end point, and the arc radius. Thus, there will be two arc segments generated. See the figure below. For the positive R value, it is an arc of central angle ≤ 180° and an arc of central angle > 180° for the negative R value.



β > 180°, the B arc with negative R

α ≤ 180°, the A arc with positive R

See the above figure. Assume that R = 50 mm and coordinates of the end point is (100.0 , 80.0), then:

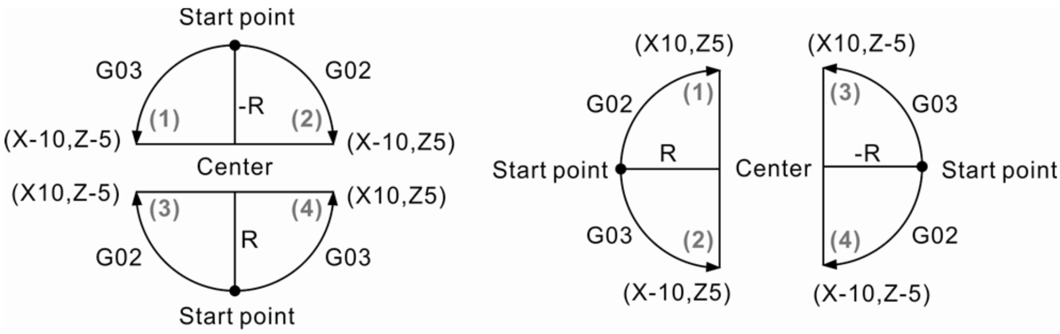
(1) β > 180°, Path B G03 X100.0 Y80.0 R-50.0 F80

(2) α ≦180°, Path A G03 X100.0 Y80.0 R50.0 F80

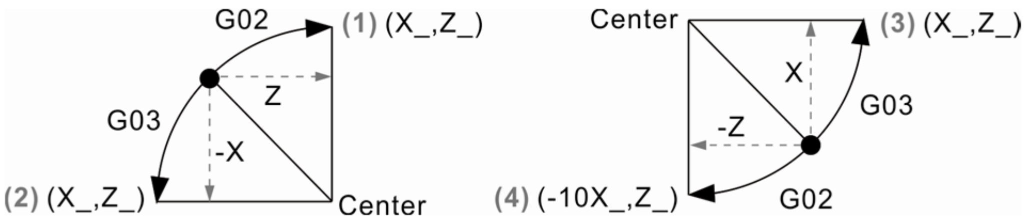
2. Center method: Parameters I, J, and K define the relative distance from the start point to the center of an arc. That is, it is the increments from the start point to the center in X, Y, and Z axis directions respectively. See the figure below for illustrations.

[Example]

Coordinate system of G18 is used in the example below.



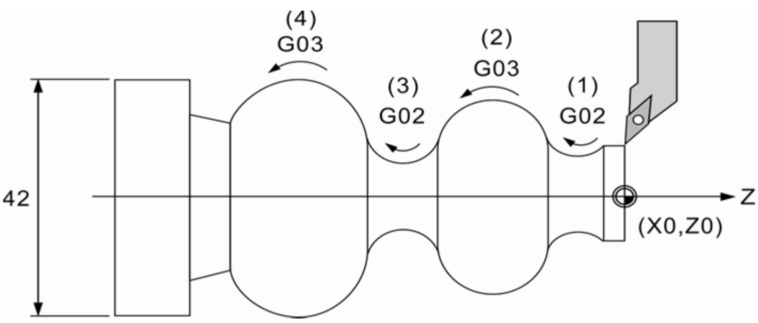
(1) G3 X-10. Z-5. I-R. F_	(1) G2 X10. Z5. KR. F_
(2) G2 X-10. Z5. I-R. F_	(2) G3 X-10. Z5. KR. F_
(3) G2 X10. Z-5. IR. F_	(3) G3 X10. Z-5. K-R. F_
(4) G3 X10. Z5. IR. F_	(4) G2 X-10. Z-5. K-R. F_



(1) G2 X_ Z_ K+Z F_
(2) G3 X_ Z_ I-X F_
(3) G3 X_ Z_ I+X F_
(4) G2 X_ Z_ IZ F_

[Example]

This example illustrates how to use G01, G02 and G03.




```
O0003
G54 X0 Y0 S800 M3
G0 X41. Y0. Z2.
G0 X20. Z0.5
G01 Z-5. F0.12
G02 Z-15. R8.      (1)
G03 Z-35. R13.     (2)
G02 Z-45. R7.5     (3)
G03 X28. Z-65. R11.   (4)
G01 X30. Z-73.
X40.2
M5
M30
```

Note on G02 and G03 arc interpolation:

(1) When machine startup, the system default cutting command is G18 (Z-X plane); thus, G18 command can be omitted if arc interpolation is to be made on Z-X plane.
(2) When I, J, K, and R values are included in the same block, value R will be referred whereas I, J, and K will be omitted.
(3) If I, J, and K values are 0, designating values to them is not necessary.
(4) When end point coordinates (X, Y, Z) are not specified, it means the start point and end point is the same; that is, the tool cuts one circle. If radius method is used, the tool will not be moving.
(5) The system prompts an alarm message “Arc radius error” when the end point does not intersect with the given radius path and the deviation exceeds the setting of machining parameter 323.
(6) For arc interpolation following a linear interpolation, the G command must convert to command G02 or G03 and G01 for straight cutting.
(7) When R, I, J, and K are not specified in the arc interpolation command (G02 and G03), the motion path will be the same as that of G01.

G04: Dwell time

Format: G04 X_
Or G04 P_

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

Command value:

Range of pause time (set by X value)		Range of pause time (set by P value)	
Range of valid values	Unit	Range of valid values	Unit
0.001 ~ 99999.999	Seconds	1 ~ 99999999	0.001 second

[Example]

G04 X1.5

G04 P1500

The two formats in the example both show that the program dwells for 1.5 seconds.

Note: This command can be used for drilling and for machining a concave-shaped contour. With this command, the cutting tool is able to stop at the bottom and separate the chips from the workpiece.

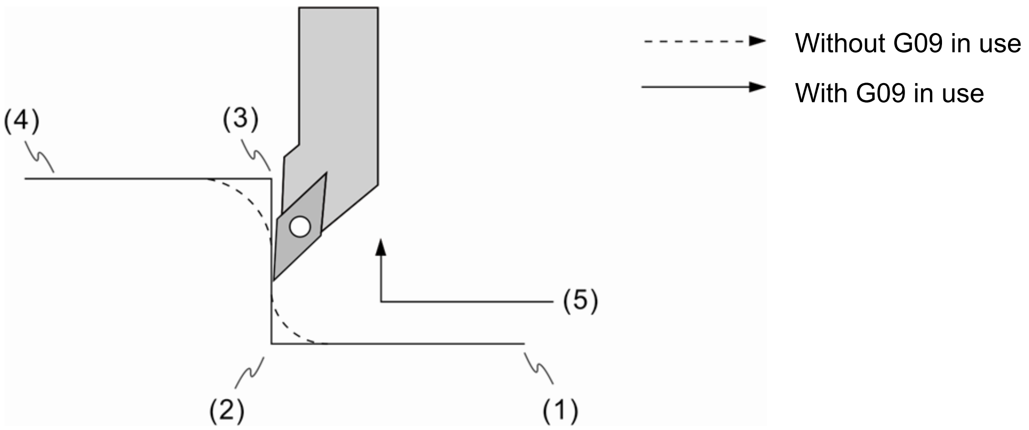
2

G09: Exact stop command

Format: G09 G01 X_ Y_

Description: When cutting, the feed rate is constant; that is, the cutting command in the next block will be executed before the execution of the current block is complete. In this case, this mechanism will create a subtle arc at the corner. To eliminate this round corner with G09, the system will confirm the tool position every time it executes a motion block. Once the tool is in the right position which is consistent with the command value, the next block can be executed. With command G09 used, there will be minor discontinuity between blocks; this improves precision at the cost of speed. This command is only active for cutting commands (G01, G02, and G03) of a single block.

[Example]



(1) G0 X0.0 Z0.0
(2) G09 G01 Z-50. F0.2 (The tool decelerates to stop. Then, the system confirms the tool position and executes the next block.)
(3) G09 X50.0
(4) Z-50.0
(5) Tool movement direction

G10/G11: Data entry setup and cancel

2

Format: G10 L2 P_ X_ Y_ Z_
G10 L10 P_X/U_Y/V_Z/W_R_Q_(Tool length, tool radius compensation and tool nose setting)
G10 L11 P_X/U_Y/V_Z/W_R_Q_ (Tool wear and radius wear compensation)
G10 L20 P_X_Y_Z_ (Entry of extension workpiece coordinate system)
G10 L21 P_X_Y_Z_ (Software limit coordinates setting)
G10 L30 P_ (Spindle positioning offset setting)
G11 (Data entry cancelation)

Description: Command format G10 L2 P_ X_ Y_ Z_ is used for data entry of the workpiece coordinate system. When P is set to 0, it stands for the offset coordinates of the workpiece coordinate system. P1 ~ P6 stands for the workpiece coordinate system of G54 ~ G59. X, Y, and Z values are for setting the coordinate origin. P value of L20 command can set the extension workpiece coordinate system paired with P1 ~ P64.

If command format G10 L10 P_ X/U_ Y/V_ Z/W_ R_ Q_ is used, it sets the tool length and tool radius compensation.
P_value specifies the compensation number. X/U_ Y/V_ Z/W_ sets the actual tool length compensation data of each axis (U, V, and W is for specifying incremental values). R_ specifies the tool radius compensation value and Q_ for tool nose setting.
If L10 is ignored in the tool length and tool radius compensation setting, the format will be "G10 P10005 X10. Y10. Z10. R2. Q1".

If command format G10 L11 P_ X/U_ Y/V_ Z/W_ R_ Q_ is used, it sets the tool wear and radius wear compensation.
P_value specifies the compensation number. X/U_ Y/V_ Z/W_ sets the actual tool wear compensation data of each axis (U, V, and W is for specifying incremental values). R_ specifies the radius wear compensation value and Q_ for tool nose setting. If L11 is ignored in the tool wear and radius wear compensation setting, the value of P_ will be the compensation number. And other settings remain the same.

G10 L21 P_: P1 sets the 1st positive software limit;
P2 sets the 1st negative software limit;
P3 sets the 2nd positive software limit;
P4 sets the 2nd negative software limit.

G10 L30 P_ sets the offset amount of spindle positioning. P_ value specifies the offset angle in unit of 0.01 degree. For example, when G10 L30 P1000 is set, it means the spindle offsets 10 degree (1000*0.01). Value of G10 can be input in both absolute and incremental form for data setting.

Data entry type

L command Format	Other argument format	Description about data type
L2	P_X_Y_Z_	Data entry format of workpiece coordinate system P: 0 is the offset coordinates. 1 ~ 6 set the G54 ~ G59 workpiece coordinates.
L10	P_X_Y_Z_R_Q_	Data entry format of tool length and tool radius compensation P: 1 ~ 64 are paired with No. 1 ~ 64 of the tool length compensation data. R: Value of tool radius wear compensation Q: Tool nose type setting
L11	P_ X_Y_Z_R_Q_	Data entry format of tool wear and tool radius wear compensation P: 1 ~ 64 are paired with No. 1 ~ 64 of the tool wear data. R: Value of tool radius wear compensation. Q: Tool nose type setting.
L20	P_X_Y_Z_	Data entry format of extension workpiece coordinate system P: 1 ~ 64 are paired with No. 1~ 64 of the extension workpiece coordinates system.
L21	P_X_Y_Z_	Data entry format of software limit coordinates P: Set to 1 to set the 1st positive software limit; Set to 2 to set the 1st negative software limit; Set to 3 to set the 2nd positive software limit; Set to 4 to set the 2nd negative software limit.
L30	P_	Spindle positioning offset setting in unit of 0.01 degree.

[Example]

G10 L10 P1 X-50. W20. R2. Q3
Or G10 P10001 X-50. W20. R2. Q3
Setting of the above program is as follows:
Tool length compensation value of No.1 on axis X is -50.0;
Tool length compensation of Z-axis = [original compensation value] + 20.0;
Tool radius is 2;
Tool nose type is 3.

Example of absolute / Incremental command:

G10 L11 P1 X-10. Z-10. R1.
The values are input in absolute form: -10.0 is input in the tool wear data field of No.1 on axis X; 10.0 is input in the tool wear data field of Z-axis; 1.0 is input in the data field of tool radius wear.

G10 L11 P1 U-10. W-10. R1.
The values are input in incremental form: The tool wear current value of No.1 on axis X plus -10.0; The tool wear current value of Z-axis plus -10.0; Tool radius wear current value plus 1.0.

Note:

(1) The G10 command is non-continuous so it is effective in a single command block. The compensation values of the offset coordinates and workpiece coordinates system are given relative to the origin of the mechanical coordinates system. You can execute command G11 to cancel data entry settings.

(2) During program execution, the command coordinates data changed by L2 or L20 command takes effects in next motion block. The tool compensation data changed by commands L10 ~ L13 take effect only after compensation commands (G41/G42 or G43/G44) are executed with compensation data number (D or tool No.) again.

2

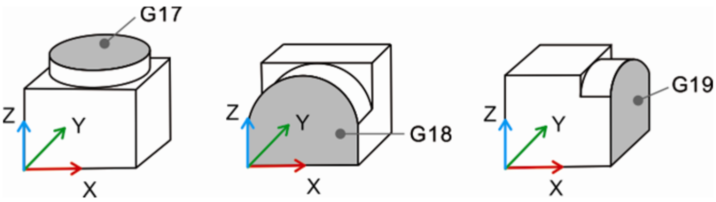
G17/G18/G19: Plane designation command

Format: X - Y plane G17 {G01~G03} X_ Y_{I_ J_ or R_}F_

Z - X plane G18 {G01~G03} Z_ X_{K_ I_ or R_}F_

Y - Z plane G19 {G01~G03} Y_ Z_{J_ K_ or R_}F_

Description: G17, G18, and G19 are for selecting different planes for cutting. If it is for the application of three-axis synchronous motion, this command is not required. G17 ~ G19 commands set up the plane for straight interpolation, arc interpolation, or tool compensation. The default command of the turning system after power on is G18. Therefore, you don't have to set G18 again if Z-X plane is to be applied.



G21/G20: Metric / Inch input command

Format: G21 or G20

G21: Set the metric system
G20: Set the inch system

Description: This command sets the metric or inch unit of measure for the system. Command G21 and G20 are applicable to the linear axes but not the rotation axes. It must be executed at the beginning of a program. Please note that metric and inch setting cannot be changed during command execution. All relevant values set in the system will be referring to this unit setting, such as the F value for setting the cutting feed rate, coordinates command, offset of the tool workpiece coordinate system, tool compensation, and moving distance. G21/G20 are continuous effective commands so the system will apply the unit you set at the beginning. In addition, G21 and G20 cannot be used in one program at the same time.

2

G28: Homing through reference point

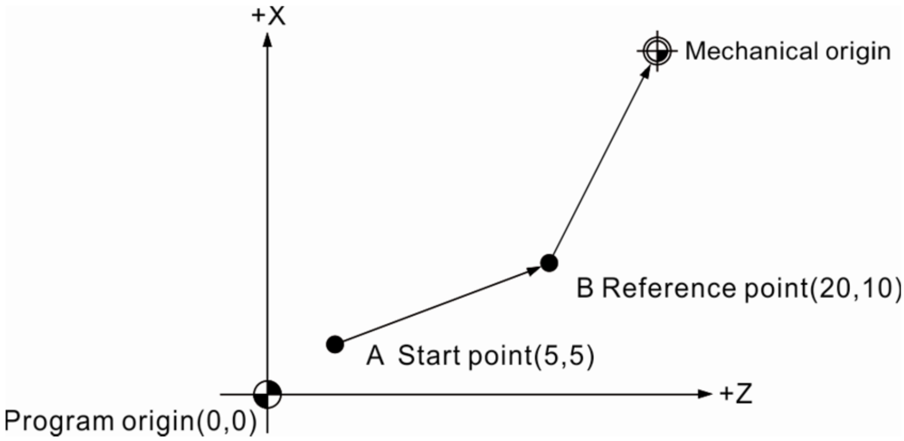
Format: G28 X_ Y_ Z_
Or G28 U_ V_ W_

X_ Y_ Z_: Coordinates of the reference point

Description: This command instructs the tool to fast move (G00) from the reference point given by the command to the mechanical origin. With this command, the tool can go to the specified reference point and then return to the mechanical origin in rapid traverse.

The X_ Y_ Z_ of the format represents the coordinates of the reference point. The undesignated axis will not pass the reference point and return to the origin. When tool radius compensation is applied (G41 and G42), please cancel it in advance. Otherwise, when G28 is executed, this compensation function will be temporarily canceled when the tool goes to the reference point and return to the mechanical origin. That is, the tool will go to the mechanical origin in the state without radius compensation. Then, the compensation will resume automatically in the next motion block. When G28 is executed, tool length compensation is still effective when the tool moves to the reference point but will be canceled when returning to the mechanical origin. Then, tool length compensation will resume automatically in the next motion block.

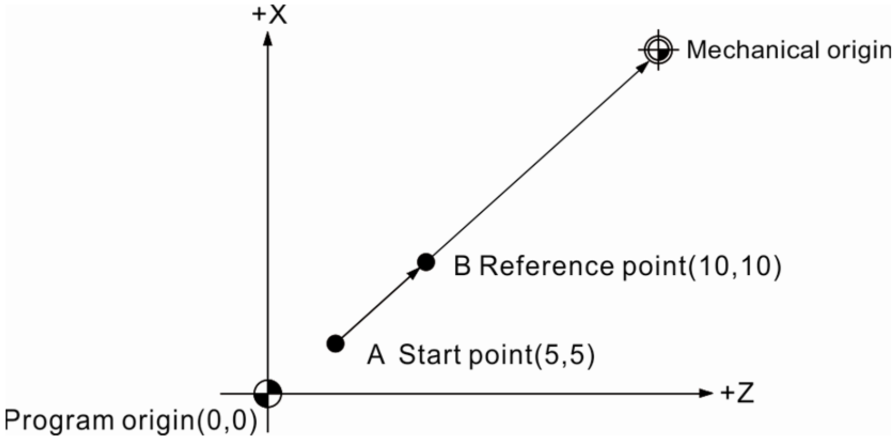
[Example]



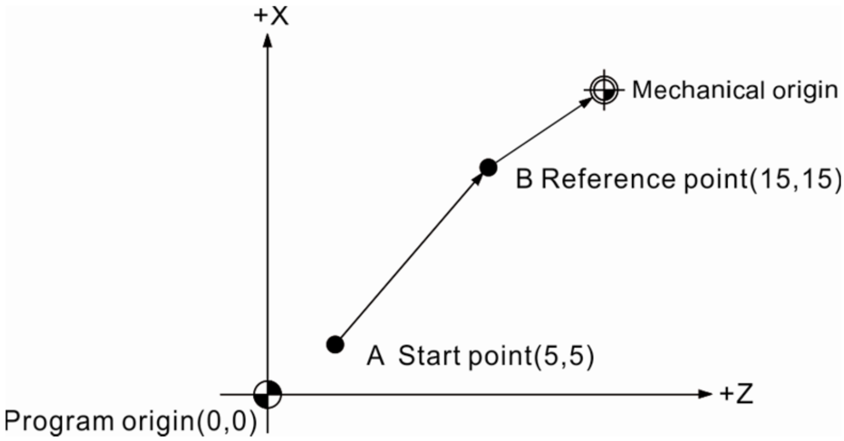
G28 X20. Z10. (Pass the reference point B and then go to the mechanical origin)

[Example]

When G28 is executed, the incremental/absolute command setting will change the path of returning to the mechanical origin. See the figure below.



G28 X10.0 Z10.0



G28 U10.0 W10.0

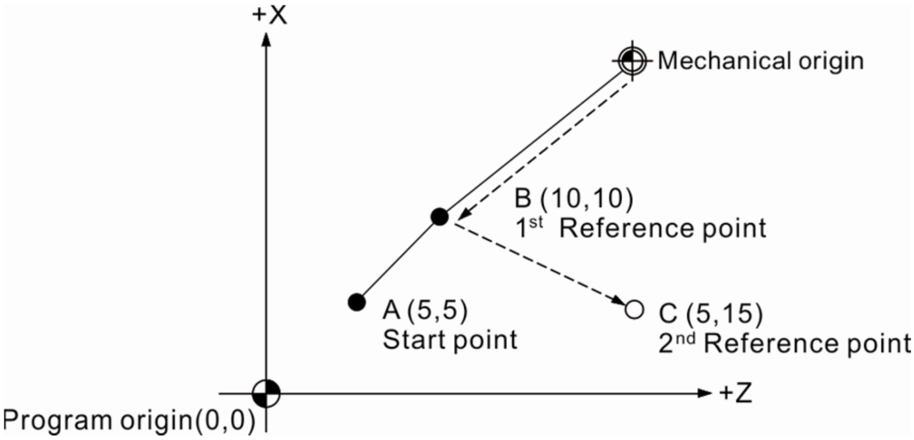
2

G29: Homing through start point

Format: G29 X_ Y_ Z_
Or G29 U_ V_ W_
X_ Y_ Z_: The final motion position of current block

Description: When G29 is applied, the tool moves from the mechanical origin or any point and pass the reference point and then go to the point specified by the block. X_Y_Z_ is the end point of G29 motion block. Please note that G29 and G28 have to be used together; that is, the tool will move to the reference point set by G28 and then go to the point specified by G29. In this case, you don't have to calculate the actual moving distance from reference point to the mechanical origin.
If G29 is executed independently without G28 that sets the reference point, an alarm will occur and the tool stops moving.

[Example]




```
G0 X5. Z5.        (Move to point A)
G28 X10. Z10.     (Move from point A to B and then go to the mechanical origin)
G29 X5. Z15.      (Move from mechanical origin to point B and go to C)
```

G30: Homing to the second, third, and fourth 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, and 4th reference point

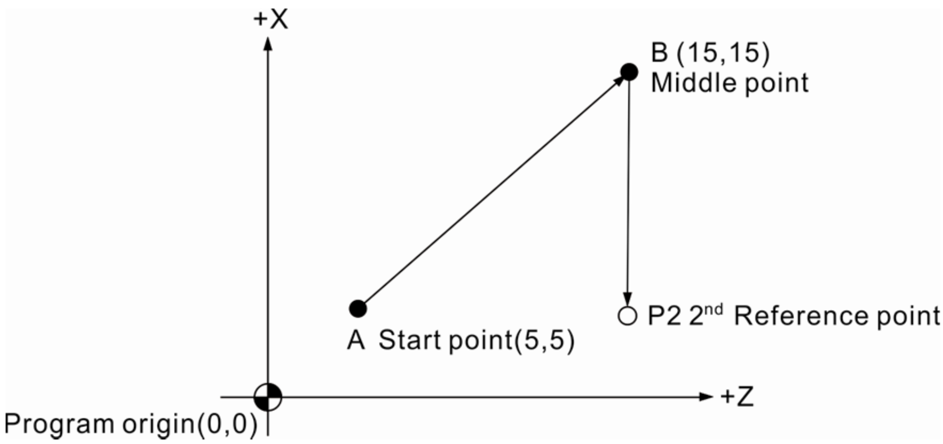
X_ Y_ Z_ : Middle point

Description: P2, P3, and P4 in the command are paired with the 2nd, 3rd, and 4th reference point. They can be set by the home parameters 607, 608, and 609. Please note that when the 2nd reference point is selected, P2 can be ignored.

X_Y_Z_ represents the middle point's coordinates. The tool will move from the current poisition to the middle point and then return to the 2nd, 3rd, and 4th reference point. And coordinate data of these reference points have to be set by home parameters.

G30 is usually used for tool change. When it is set as absolute command, take G30 Z0.0 for example, Z-axis will return to the middle point (Z0.0) and then move to the 2nd reference point and the command is complete.

Please cancel the tool compensation (execute G40) when using G28 and G30. When G30 or G28 is executed, the tool compensation and tool length compensation in the current block will be canceled. The compensation function will resume in the next block after homing to reference point.



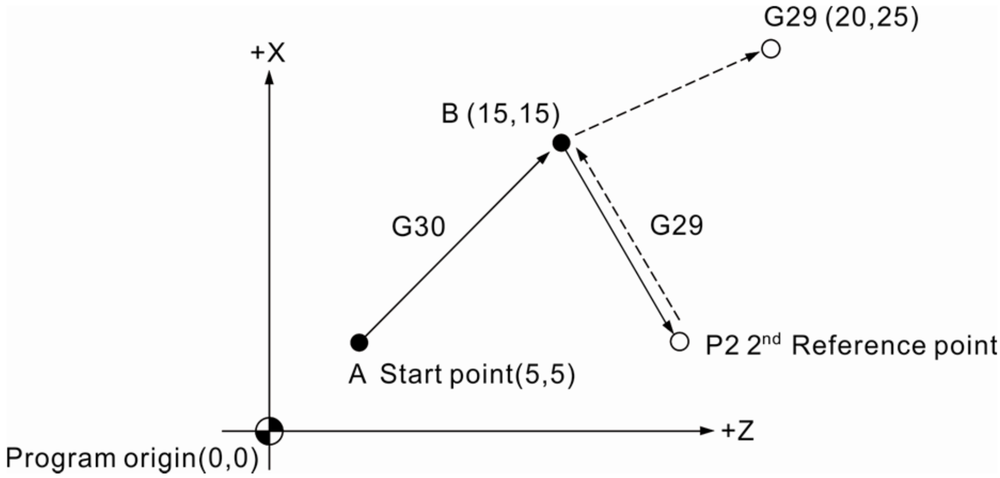
G30 P2 X15. Z15.

Move from point A to B and then to P2 (2nd reference point)

See the figure above. G30 is specified with an absolute value. Thus, Z-axis returns to the middle point first and then goes to P2. Then, homing to the 2nd reference point is complete.

2

[Example 1]

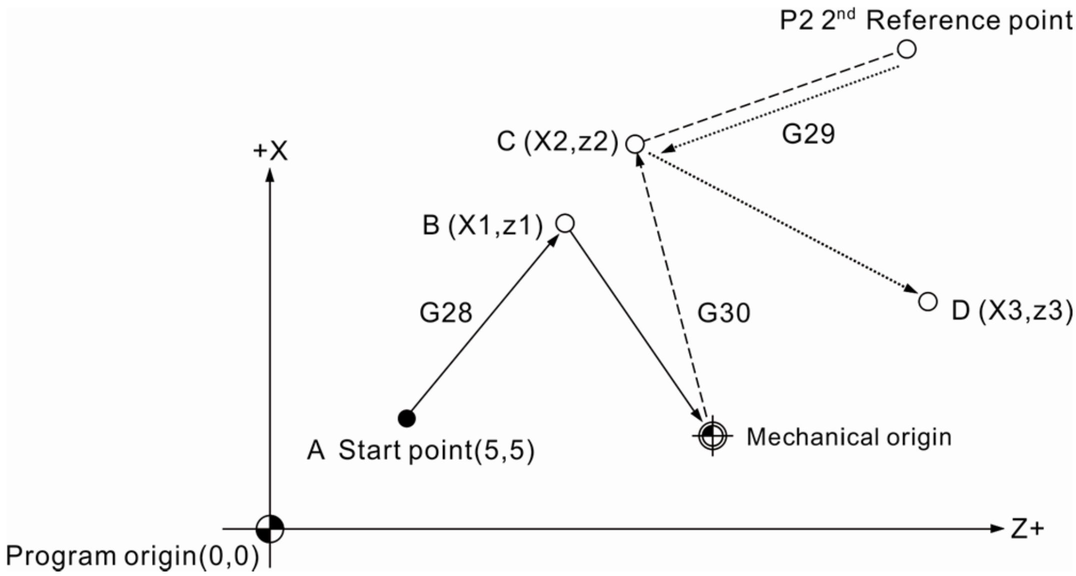


Program example:

G30 P2 X15.0 Z15.0

G29 X20.0 Z25.0

[Example 1]



Program example:

G28 X<u>x1</u> Z<u>z1</u>

G30 P2 X<u>x2</u> Z<u>z2</u>

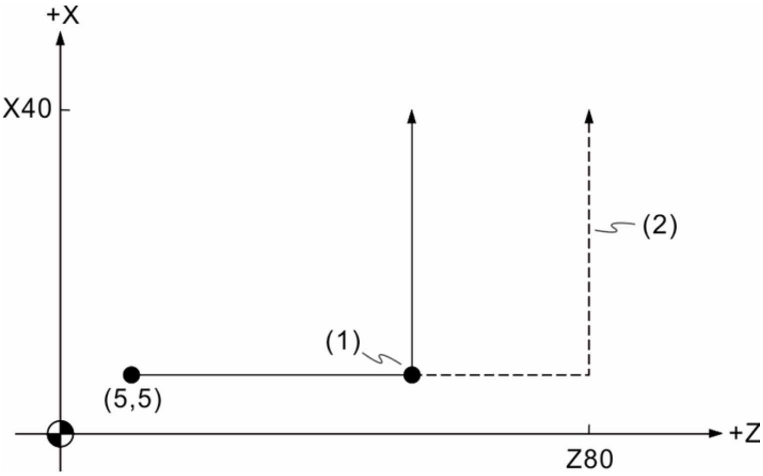
G29 X<u>x3</u> Z<u>z3</u>

G31: Skip function command

Format:G31 X_ Y_ Z_ F_

Description: During linear motion, G31 stops running the motion path immediately via external skipping signal and the next block will be executed. This command is only valid in single block. G31 cannot be executed when tool radius compensation (G41/G42) is functioning. Thus, please cancel tool compensation (G40) before using this command.

[Example 1]



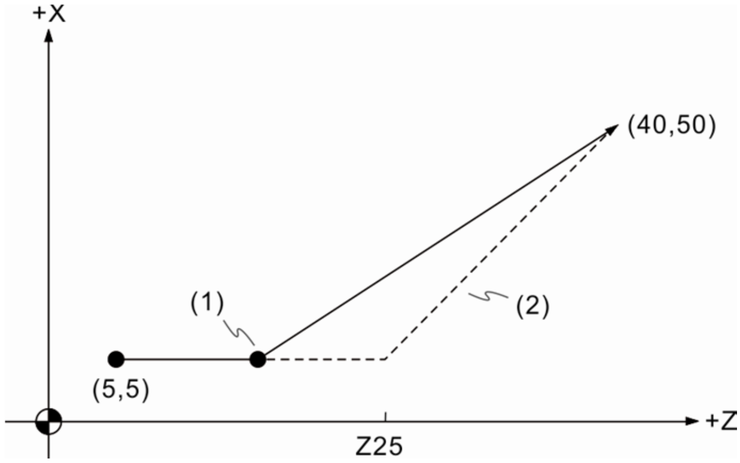
G00 X5. Z5.
G01 G31 Z80.0 F0.5
X40.0

(1) Skip signal; (2) Original programmed path

The dashed line presents the motion path when no skip signal is input. On the other hand, the solid line shows the path with skip signal input; the system will immediately stop executing the current block once the skip signal is input and start executing the next block.

2

[Example 2]



G00 X5. Z5
G01 G31 Z25.0 F0.5
X40.0 Z50.0

(1) Skip signal; (2) Original programmed path

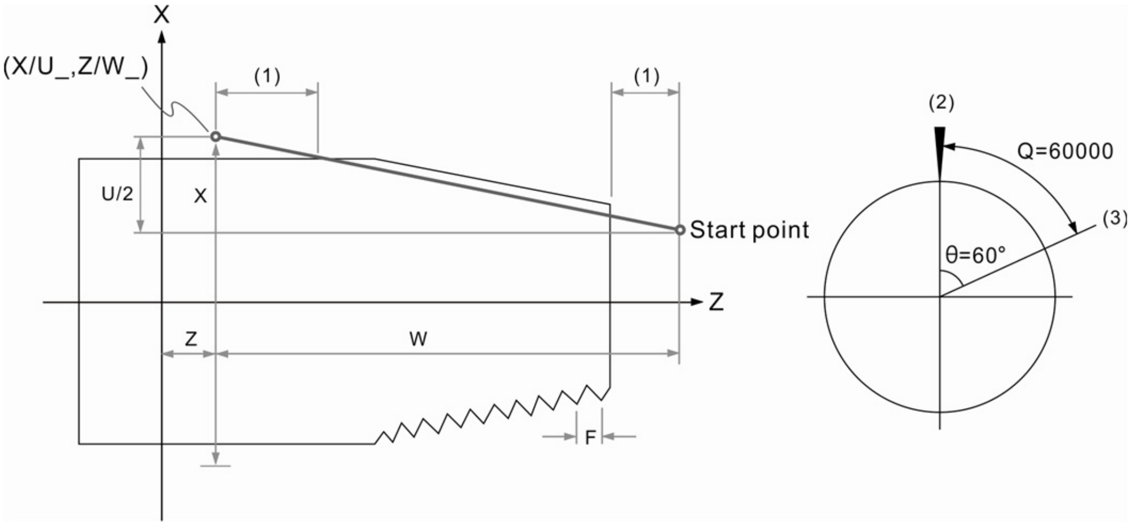
If skip signal is not input, the actual path will be the one shown in dashed line. While skip signal is input, the tool will then make the motion specified in the next block immediately as shown with the path of solid line.

G32: Thread cutting

Format: G32 X/U_ Z/W_ F_ Q_

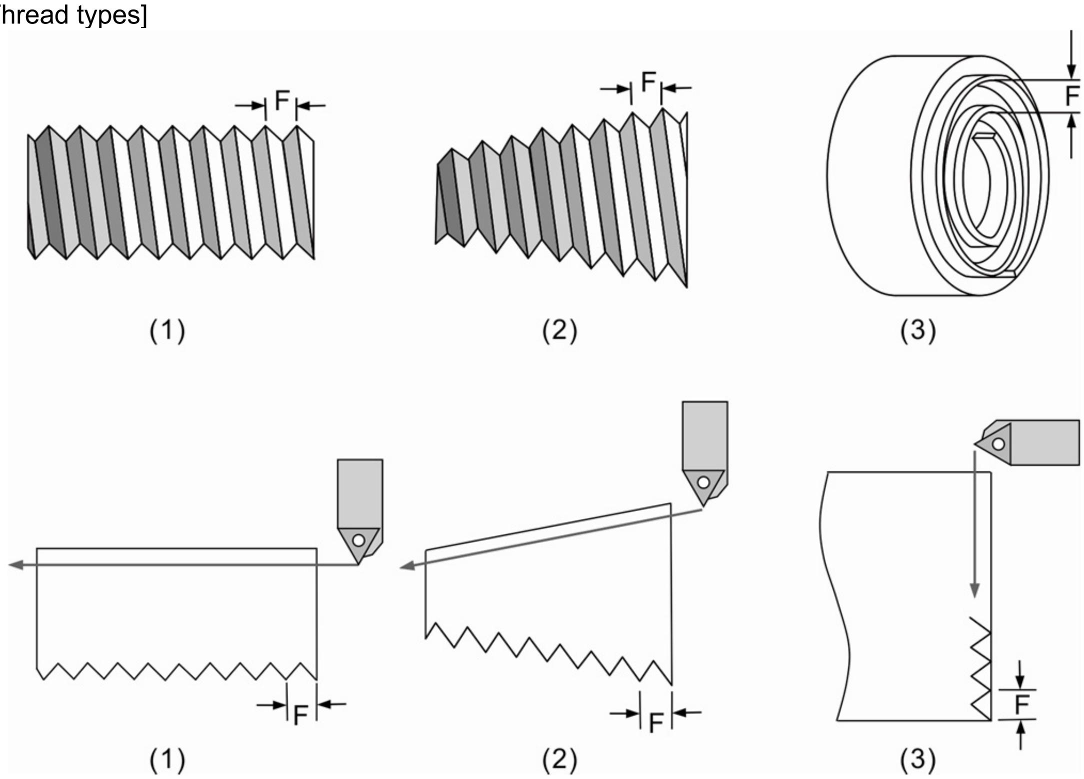
X/U_ Z/W_: Coordinates of the threading end point.
F_: Thread lead; the linear distance per spindle revolution.
Q_: Start offset angle of threading in unit of 0.001 degree. The degree is 0 when value is not specified.

Description: G32 is a thread cutting command for machining applications of straight threading, tapered threading and scroll threading.



(1) Invalid screw thread；(2) Z pulse of the spindle；(3) Threading engaged point

[Thread types]

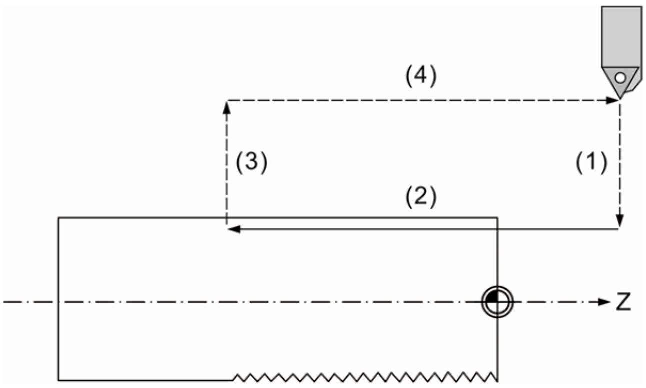


(1) Straight thread; (2) Tapered screw; (3) Scroll thread

2

Note:
(1) G32 has to be used with constant spindle speed.
(2) While threading, spindle feed rate cannot be adjusted manually; it keeps at 100% of the speed.
(3) When pressing the **Cycle Stop** key during threading, the threading will not stop immediately but at the end of the next block which has no threading command.
(4) Pressing the **RESET** key can immediately stop the threading. However, this will cause screw damage.
(5) Assume that spindle speed is 3000 rpm with pitch (F value) set to 1.5, Z-axis feed rate will be 4500 mm/min when threading. If alarm [B01D Spindle speed is too high] occurs, it means the speed has exceeded the max. feed rate and lowering the speed is required.
(6) The following deviation of the servo system will make an invalid screw thread at the start and end point when threading. In this case, the specified threading length has to be longer than the actual required length so that the screw function will not be affected.

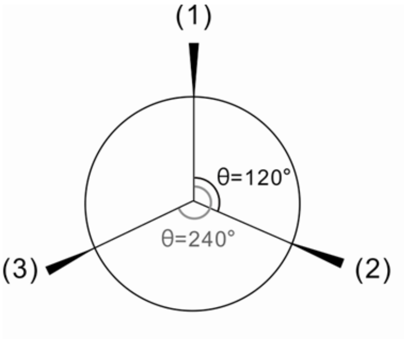
[Example]

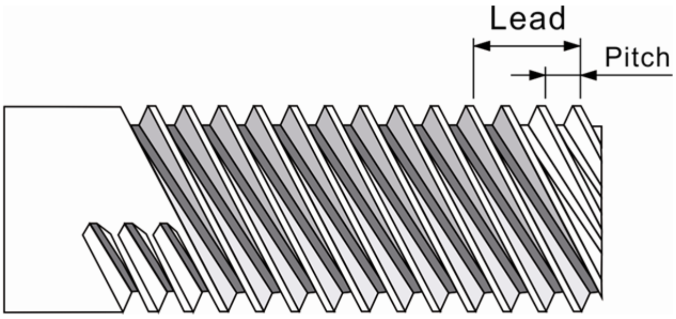



```
T0202
M3 S1000
G0 X40.
Z15.
X17.45    (1) Height of the 1st threading
G32 Z-30. F1.5    (2) Threading
G0 X40.    (3) Retracts in X-axis
               direction with G0
Z15.    (4) Retracts in Z-axis direction
           with G0
X17.20    (1) Height of the 2nd threading
G32 Z-30. F1.5    (2)
G0 X40.    (3)
Z15.    (4)
X17.00    (1) Height of the 3rd threading
G32 Z-30. F1.5    (2)
G0 X40.    (3)
Z15.    (4)
X16.85    (1) Height of the 4th threading
G32 Z-30. F1.5    (2)
G0 X40.    (3)
Z15.    (4)
X16.8    (1) Height of the 5th threading
G32 Z-30. F1.5    (2)
G0 X40.    (3)
Z15.    (4)
M5
M30
```

[Example] Multiple thread screw machining

L (Lead) = n (Number of screw threads) x pitch





Main program

T0202
M3 S1000
G0 X45.
Z10.
G66 P3300 A0 Engaged point (1)
X17.45
X17.20
X17.00
G67
G66 P3300 A120000 Engaged point (2)
X17.45
X17.20
X17.00
G67
G66 P3300 A240000 Engaged point (3)
X17.45
X17.20
X17.00
G67
G0 X45.
Z10.
M30

Subroutine

O3300
G32 Z-30. F3 Q#1 (#1 is brought in the subroutine by A_, which is the threading offset angle.)
G0 X45.
Z10.
M99

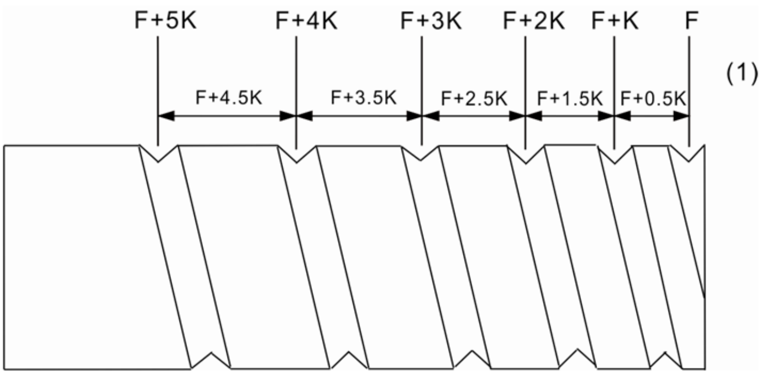
2

G34: Variable lead threading

Format: G34 X/U_ Z/W_ F_K±_

X/U_ Z/W_: Coordinates of the threading end point
K_: The increment and decrement of pitch per spindle revolution. Negative value indicates the decrement of pitch .
F_: The initial pitch
Q_: Lead offset angle at the start point in unit of 0.001 degree. The default is 0 degree.

Description: When G34 is applied, the system will refer to the increment and decrement of lead specified in G34 to perform variable lead threading.
Lead: The linear distance of per spindle revolution
Pitch: The distance between two adjacent screws



(1) Lead: $d = \text{Vo}\,t + \frac{1}{2}at^2$

Vo: The initial pitch of screw thread (F)

T: Number of screw threads

A: The increment in pitch per spindle revolution (K)

(1) The speed changes in each screw threading lead

[Example]

Lead of the 1st spindle revolution $d = F*1 + \frac{1}{2}*K*1^2 = F + 0.5K$
Lead of the 1st and 2nd spindle revolution $d = F*2 + \frac{1}{2}*K*2^2 = 2F + 2K$

(Lead of the 1st and 2nd spindle revolution) – (Lead of the 1st spindle revolution) = (Lead of the 2nd spindle revolution):

$(2F + 2K) - (F + 0.5K) = F + 1.5K$

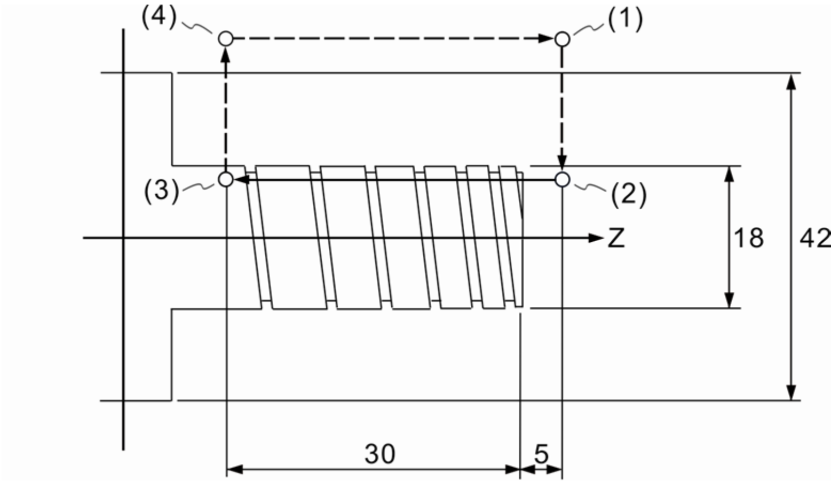
Note:
(1) G34 has to be used with constant spindle speed.

(2) While threading, spindle feed rate cannot be adjusted manually; it keeps at 100% of the speed.

(3) When pressing the **Cycle Stop** key during threading, the threading will not stop immediately but at the end of the next block which has no threading command.

(4) Pressing the **RESET** key can immediately stop the threading. However, this will cause screw thread damage.
(5) Assume that spindle speed is 3000 rpm with pitch (F value) set to 1.5, Z-axis feed rate will be 4500 mm/min when threading. If alarm [B01D Spindle speed is too high] occurs, it means the speed has exceeded the max. feed rate and lowering the speed is required.
(6) The following deviation of the servo system will make an invalid screw thread at the start and end point when threading. In this case, the specified threading length has to be longer than the actual required length so that the screw function will not be affected.

[Example]



Main program

T0101 // Select tool No.1

M03 S600 // Spindle rotates forward at 600 rpm

G0 X50. Z5. // (1) Move to the enaged point in rapid traverse

G66 P0034 L1 //Macro call: execute subroutine O0034 once

X17.65 // (2) Threading depth

X17.45 // (2) Threading depth

X17.25 // (2) Threading depth

X17.05 // (2) Threading depth

G67 //End of macro call command

G0 X50. Z5. // Retract to the safety point in rapid traverse.

M5 //Spindle stop

M30 //Program end

Subroutine

O0034

G34 Z-30. K0.5 F1 // (3) Execute command G34. Perform threading operation to Z-30 and the pitch increases 0.5 mm per spinlde revolution. The initial pitch is 1 mm.

G0 X50. // (4) Retract in X-axis direction.

Z5. //Return to the engaged point.

M99 //Return to the main program.

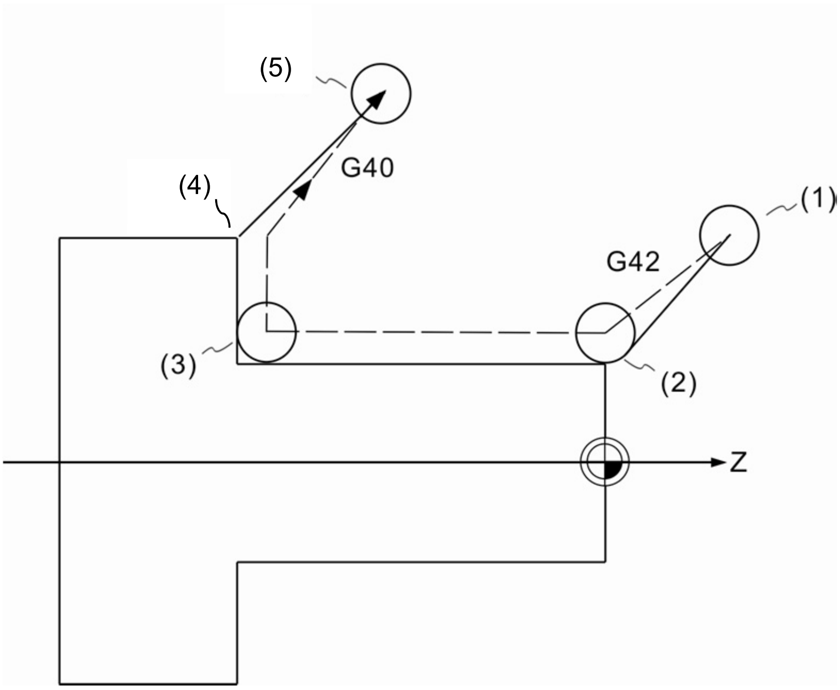
2

G40: Cancel right/left compensation of tool nose radius

Format: G40
Or
G40 X_ Z_

Description: When tool nose radius compensation for the tool path is not required, use G40 to cancel the compensation. However, if the compensation is not canceled, this functinon will be continous effetive as G40 is a status command.
When homing procedure is executed and the tool returns to the reference point, the tool compensation will temperarily be canceled and resume at the next motion block. In addition, compensation cancelation is not available for arc motion path.

[Example]



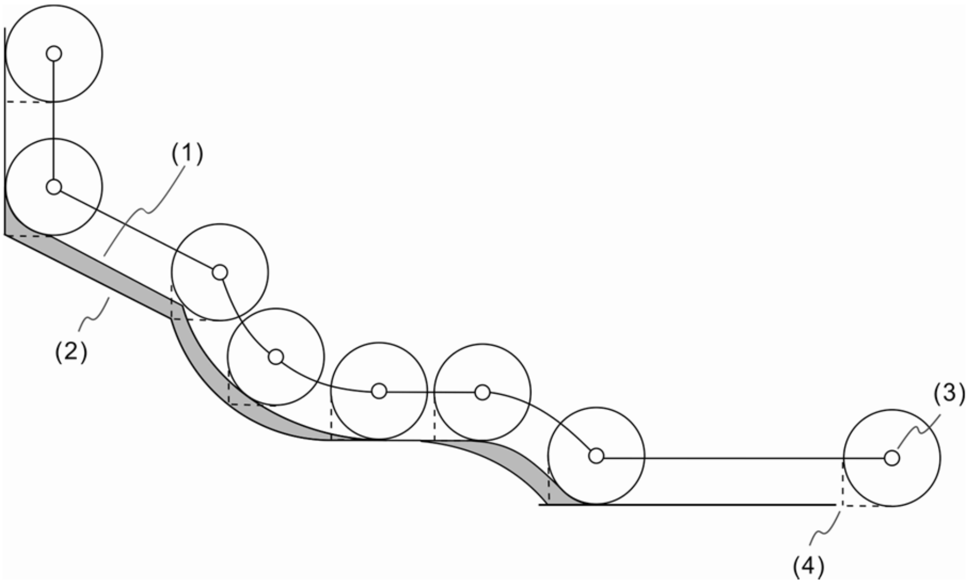
(1) G0 X40. Z20. (Start point)
(2) G41 G1 X20. Z0. F0.25 (Tool compensation enabled)
(3) Z-30.
(4) X40.
(5) G40 G0 X60. Z-20. (Move to this point and compensation ends)

G41/G42: Left / Right compensation of tool nose radius

Format: G00 G41 P_
G00 G42 P_

G41: Tool radius leftward compensation
G42: Tool radius rightward compensation
P_: Coordinates to be moved to

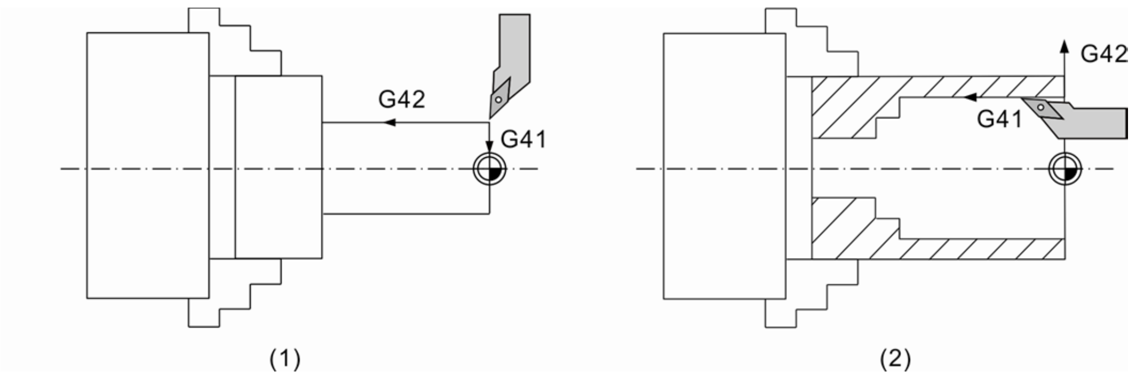
Description: Generally, there will be a deviation between the actual contour and the programmed contour. This is because the tool nose is usually in arc shape, and the coordinates specified in the program are referring to the hypothetical tool nose position. When G41 or G42 is applied, compensation for tool nose radius will be done based on the setting of tool radius, tool nose type, and right/left compensation and auto calculate the compensation amount.



(1) Acutal machining contour ; (2) The machining contour set by the program; (3) Tool nose center; (4) Hypothetical tool nose position

Specifying compensation tool number TXXXX is required before applying tool nose radius compensation.

[Compensation setting of the acutal machining]

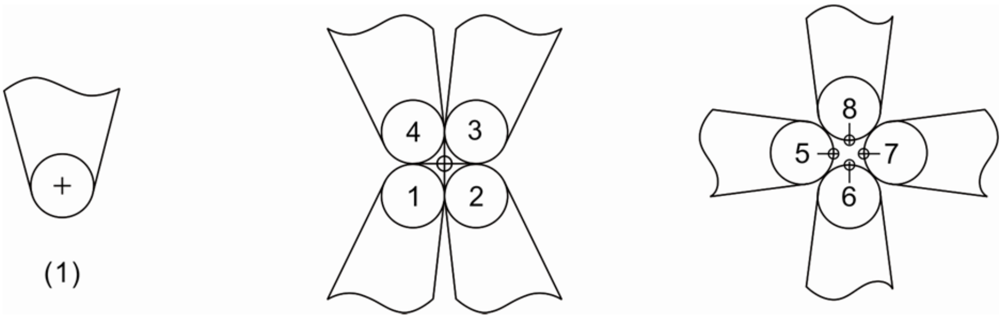


(1) Machining outer diameter and face; (2) Machining inner diameter

2

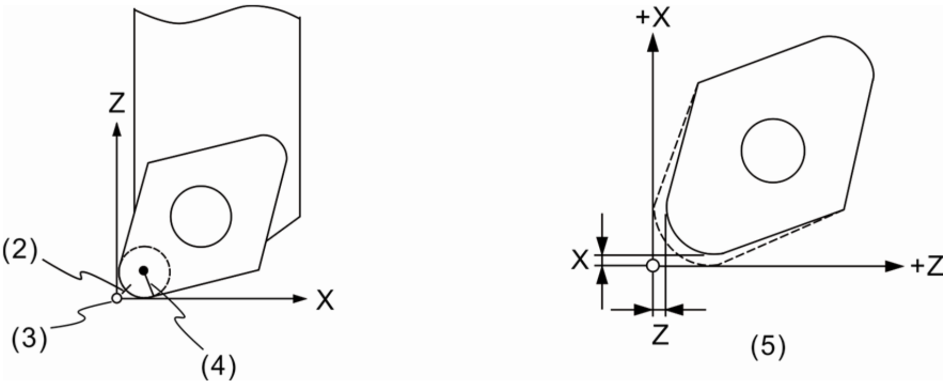
Tool nose type setting: In general, the tool nose is in arc shape and the tool nose position varies with the tool type. See the figure below. Please input the tool nose type number based on the tool shape in the type field of tool register in OFS group.

[Tool nose types]



(1) Tool nose No. 0

[Tool radius compensation and compensation for tool radius wear]



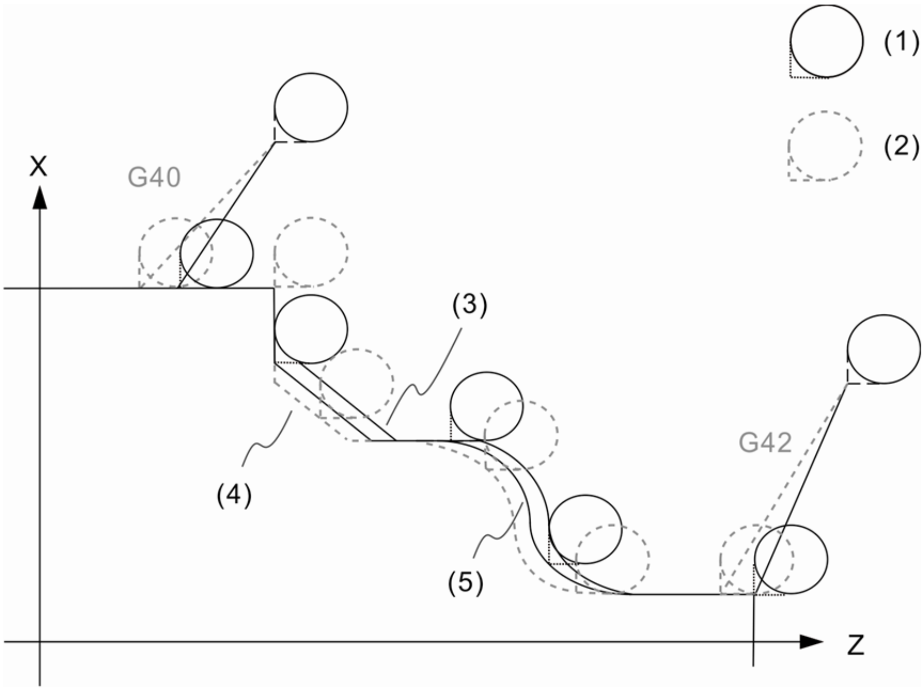
(2) Actual tool nose position；(3) The assuming tool nose position when calibration；(4) R value of tool nose radius compensation；(5) Compensation for wear in tool nose

Notes for tool radius compensation:

(1) Compensation command can be used with G00 and G01 in the same block. But it has to be a motion block (Tool radius compensation enabled) in order to have the function work.
(2) Compensation command cannot be used in the block with G02 and G03. To apply compensation function to an arc path, you have to set tool radius compensation for linear motion path in advance. When the compensation is active, cancelling is not allowed in the arc path.
(3) During program editing, please specify the tool radius compensation number (e.g.,T0111 and T0212). Each tool radius compensation number will correspond to a number in compensation data table. The compensation value is input via the number data of tool register in OFS group screen.
(4) When compensation value changes its sign, the compensation direction specified in G41 and G42 will be changed. For example, when positive value is given in G41, the compensation direction is left; when negative value is given, the compensation direction becomes right.
(5) When tool radius compensation function (continuous effective) is active and G28 or G29 is executed, the compensation will be temporarily canceled. However, the system will reserve this state so the compensation will resume in the next motion block.
(6) When a program path is completed with the tool radius compensation, please execute G40 to cancel the compensation. That is, the best timing for using G40 is when the tool is removed from the workpiece.

[Example 1]

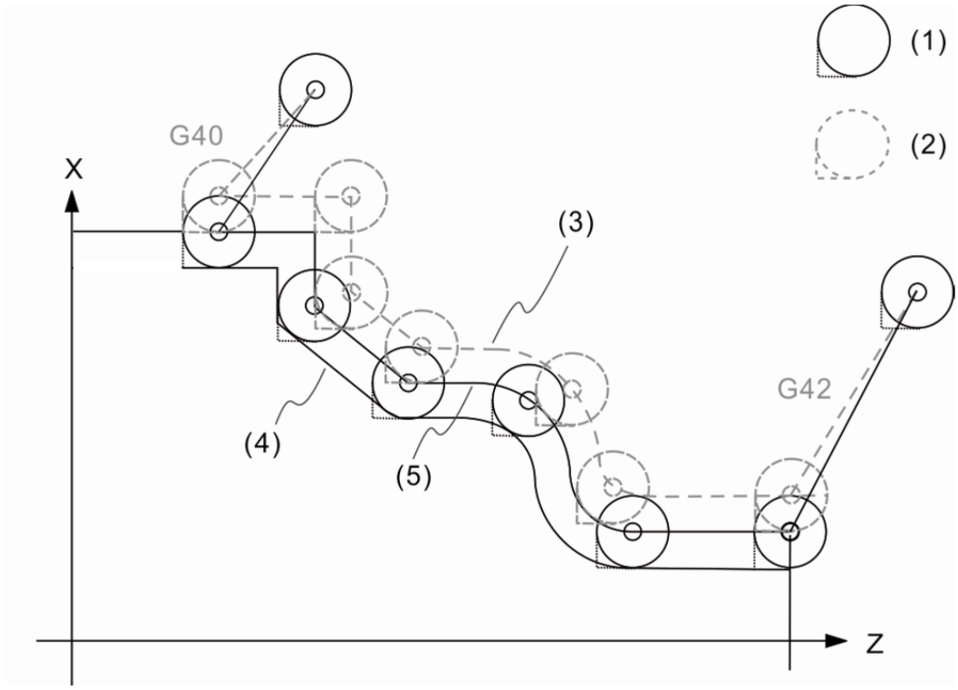
Below is the machining illustration when compensation function is enabled with the use of tool nose type 3.



(1) Programmed tool trail; (2) Tool compensation trail; (3) The machining contour without tool compensation; (4) Tool nose path after compensation; (5) The machining contour after compensation

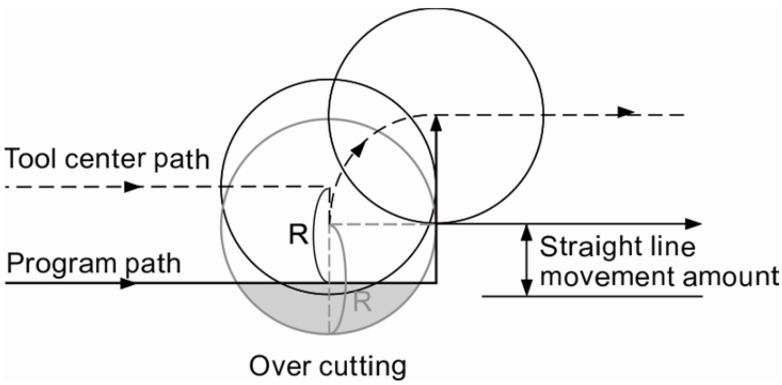
[Example 2]

Below is the machining illustration when compensation function is enabled with the use of tool nose type 0 or 9.

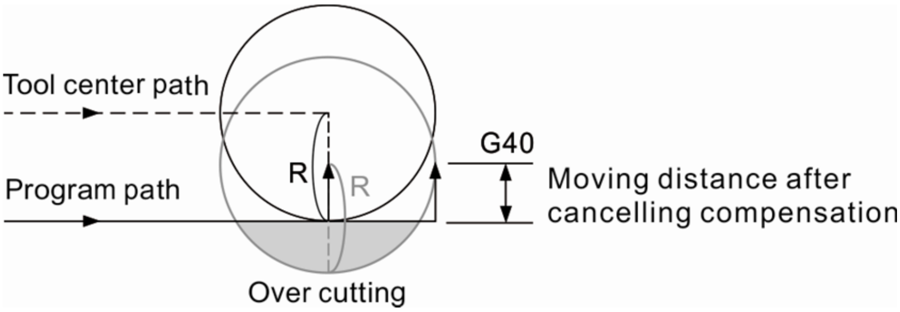


(1) Programmed tool trail; (2) Tool compensation trail; (3) Tool center path after compensation; (4) Machining contour when compensation disabled; (5) The tool center path when compensation disabled & the machining contour after compensation enabled.

During the compensation, the linear moving distance and inner arc radius for cutting should ≧the tool radius; otherwise, over cutting may occur due to compensation vector interference. In a case like this, the controller stops operation and prompts alarm messages.

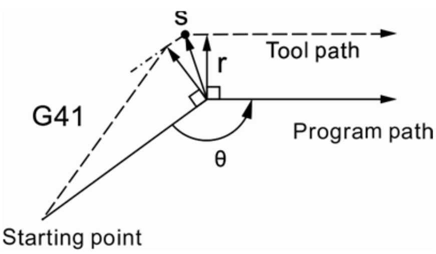
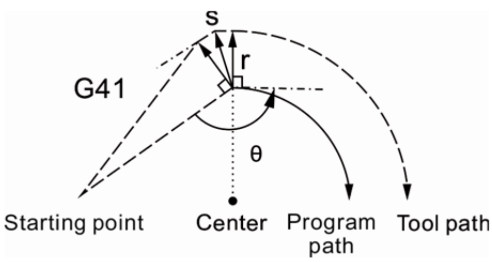
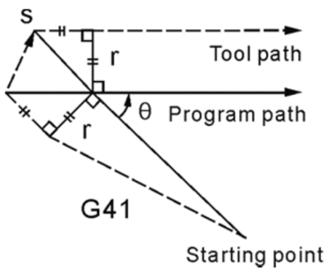
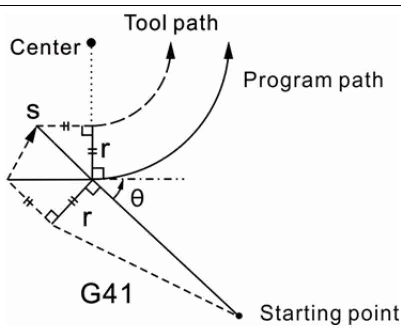


The moving distance after cancelling the compensation should ≧ tool radius. When it is smaller than compensation vector, the cutting path will be interfered and over cutting may occur. When this happens, the tool stops moving and an alarm message pops up. See the figure below for illustration.



Moving distance after cancelling compensation (G40) < Tool nose radius R

(1) The tool radius compensation will be canceled in the motion block that follows G40. Compensation path: Compensation is done at the start and end point. See the figure below.

Line to arc	Line to arc
	
	

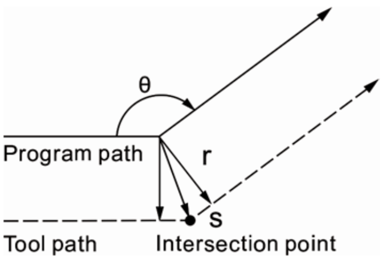
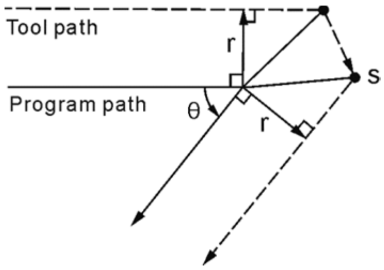
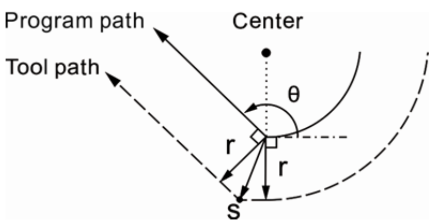
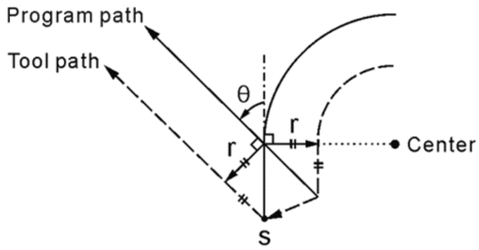
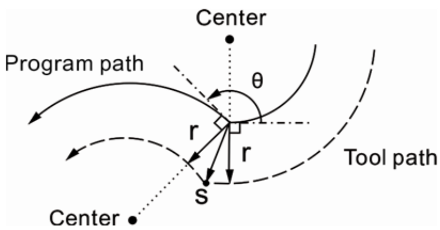
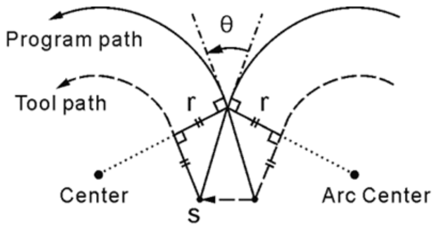
2

Path type of tool radius compensation:

Regarding the compensation path, the angles (180° > θ> 90°, 0 < θ < 90°) generated at the intersection point (when transiting between two blocks) will determine the path type.

(1) When the angle generated by two blocks is within the range of 90°~ 180° (180° > θ > 90°), the tool moves in an inward curved angle path.

(2) When the angle generated by two blocks is within the range of 0° and 90° (0 < θ < 90°), the tool moves in an outward curved angle path.

Inward curved angle	Outward curved angle
	
Inward curved angle: Arc to line	**Outward curved angle: Arc to line**
	
Inward curved angle: Arc to arc	**Outward curved angle: Arc to arc**
	

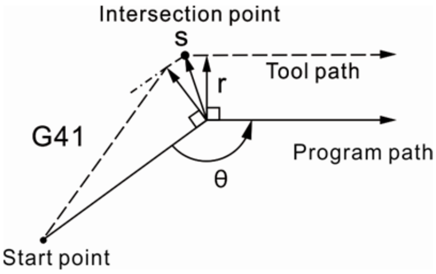
Compensation path switch:

(1) When switching to the path with compensation, the motion track of the tool center is as shown in the figure below (G40 >G41; G40 > G42).

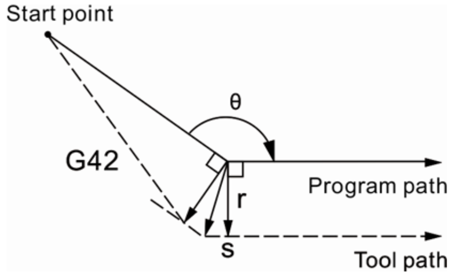
(2) During compensation, the motion track remains active. When compensation is cancelled (G40) or switched to compensation direction is changed as elaborated in the figure of G41 > G40 and G42 >G40, the motion track will be as illustrated in the figures of G41 > G42 and G42 > G41 below.

2

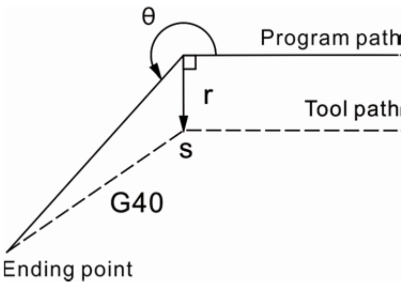
G40 > G41



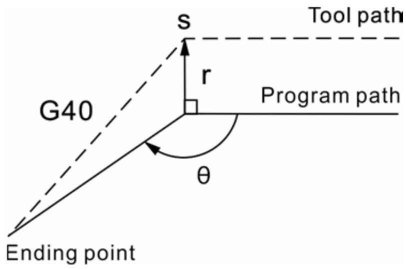
G40 > G42



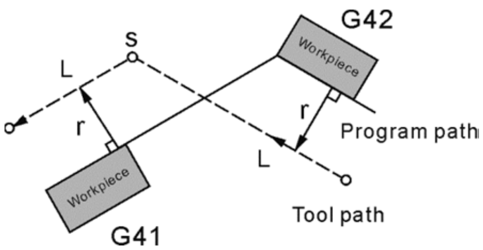
G41 > G40



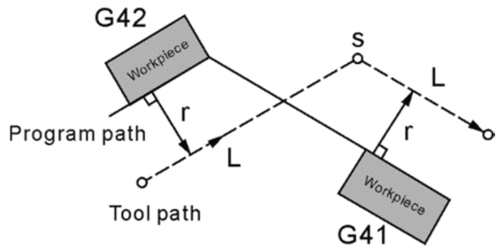
G42 > G40



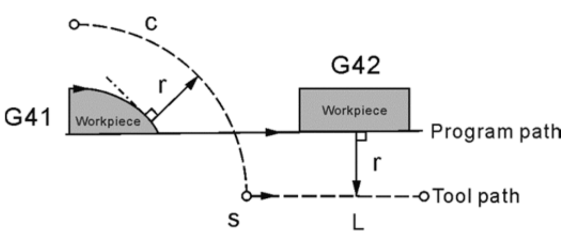
G41 > G42



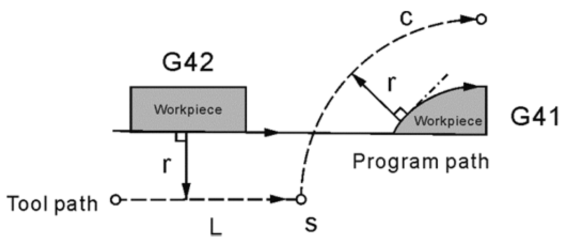
G42 > G41

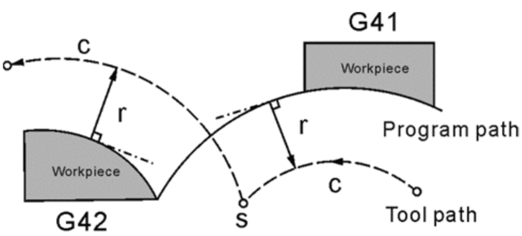


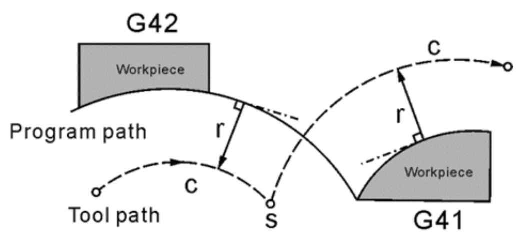
G41 > G42



G42 > G41







2

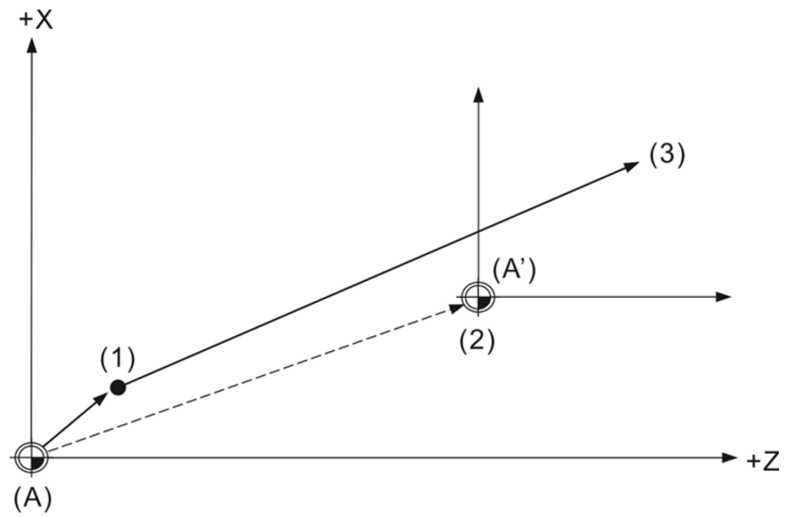
G52: Local coordinate system setup command

Format: G52 X_ Y_ Z_

X_ Y_ Z_: Origin of the local coordinates system

Description: During program editing, you can designate another sub-coordinate system based on the workpiece coordinates for specifying the path. And this assigned sub-coordinate system is called 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). Please note that G52 is active only when it is specified with absolute values instead of incremental values. Command G52 specified as zero cancels the local coordinates system setup.

[Example1]



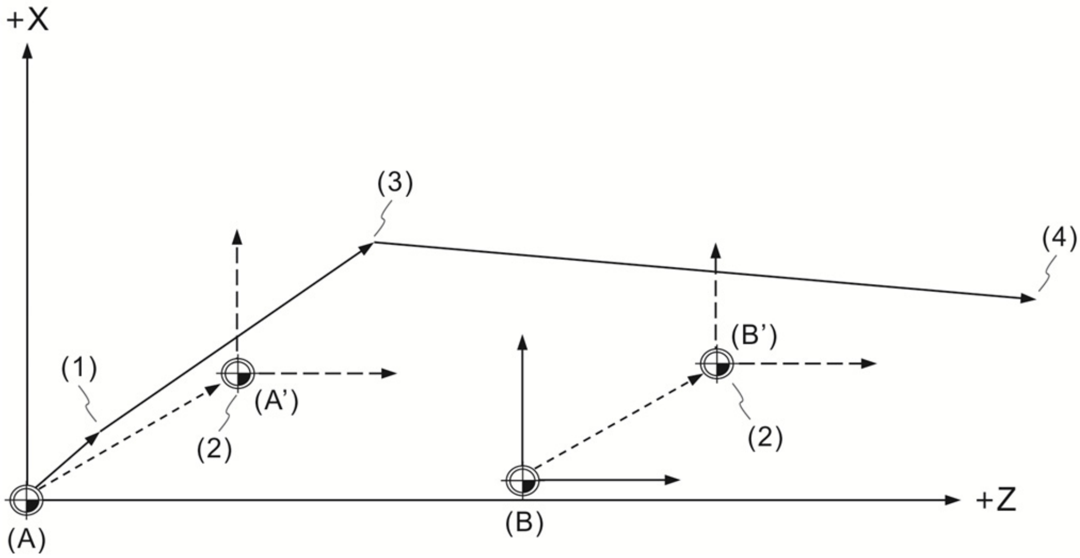
(A) Coordinate system of G54; (A') New coordinate system of G54

G54 X10. Z10. (Move from the origin to point (1))

G52 X20.0 Z40.0 (Origin of G54 coordinate system shifts to point (2))

G00 X20.0 Z20.0 (Move to point (3) in the new G54 coordinate system)

[Example 2]



(A) Coordinate system of G54; (A') New coordinate system of G54; (B) Coordinate system of G56; (B') New coordinate system of G56

G54 G00 X5. Z10. (Move from origin of G54 coordinate system to point (1))
G52 X10. Z20. (The tool moves to point (2) with increments of G52)
G00 X20. Z20. (Move to point (3) in the new G54 coordinate system)
G56 G00 X10. Z40. (Move to point (4) in the new G56 coordinate system)

Note:

(1) When G52 command is active and the current workpiece coordinate system is changed to another one, the shift set by G52 will be effective.

(2) To cancel the local coordinate setting, specify coordinate value Z, Y, Z as 0 in G52; That is, G52 X0 Y0 Z0.

G53: Mechanical coordinate system setup command

Format: G53 X_ Y_ Z_

X_ Y_ Z_: Actual arriving position of mechanical coordinates

Description: Coordinates (X, Y, Z) is the actual end point in mechanical coordinates system specified by the program. Machine suppliers use this command to set up the tool change position, which is given in the mechanical coordinates. The command format must be specified as absolute; G53 with increment format will not be executed. Command G53 is a non-continuous effective G command and is valid for single block. Before using G53 to complete the coordinate setting, please complete the homing procedure in auto or manual mode.
When G53 is executed, the tool moves in rapid traverse (G00) and both tool radius compensation and tool length compensation will be cancelled automatically. The former resumes at the next motion block while the latter has to be set again in order to be active.

Note:
(1) Command G53 functions only when specified with absolute values. Specified with incremental values G53 will be ignored. However, when G00/G01 is used, their settings will be effective and change motion specified in the next block.
(2) For the block containing G53 without motion specified (axial command), the tool does not move.
(3) If both commands G53 and G28 are set in the same block, the one read later becomes active. When command G53 is active, the system will refer to the mechanical coordinates. If command G28 is active, then the system will refer to the absolute coordinates.

[Example]

Ex 1:
G53U150.W-150. (It is specified with incremental values so this block is ignored.)

Ex 2:
G53X50.Z-50. (Move to the actual mechanical coordinates X50. Z50.)

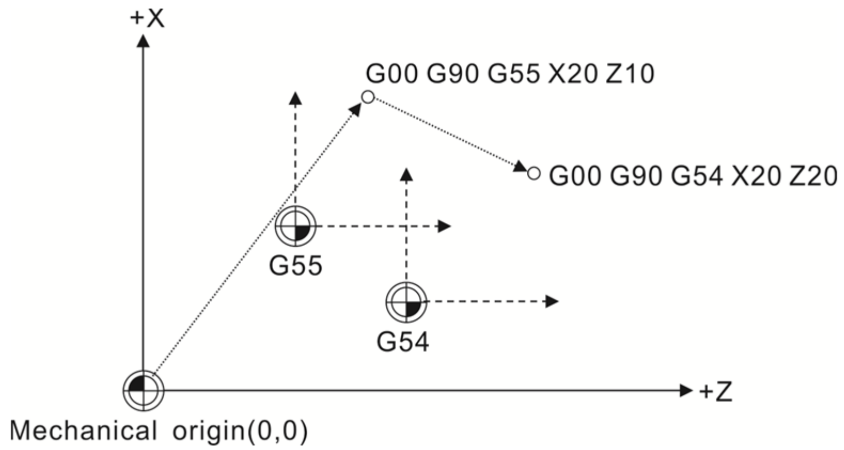
Ex 3:
G1G53X100.Z-100.F1000 (Execute this block in rapid traverse G00)
X50.Y50. (The motion status is changed in this block (Straight cutting at G01F1000.))

G54~G59: Workpiece coordinate system selection command

Format: G54 X_ Y_ Z_
Or G55 X_ Y_ Z_
Or G56 X_ Y_ Z_
Or G57 X_ Y_ Z_
Or G58 X_ Y_ Z_
Or G59 X_ Y_ Z_

Description: Users can assign any one of the 6 coordinate system (G54 ~ G59) as the workpiece coordinate system. To create a workpiece coordinate system, you can firstly move the tool from the mechanical origin to the programmed origin (X, Y). Then, input this position data in OFS group > Workpiece coordinates system setup (G54 ~ G59). Next, use the workpiece coordinate ID to set the workpiece origin.
In addition, the system also provides 64 extension workpiece coordinate systems for selection. You can designate one by setting P_ value in G54, which setting range is 1 ~ 64. For example, if G54 P10 X_ Y_ Z_ is set, it means the 10th workpiece coordinates system of expanded ones is used.

[Example]



G00 G55 X20 Z10
G00 G54 X20 Z20

With the selection for workpiece coordinate system, you can easily calculate and design the programming path and create multiple coordinate systems on the working platform for switching among programs. As shown in the figure above, when coordinate origin is changed, you can edit the workpiece coordinate data instead of creating another program to have the machining carried out.

2

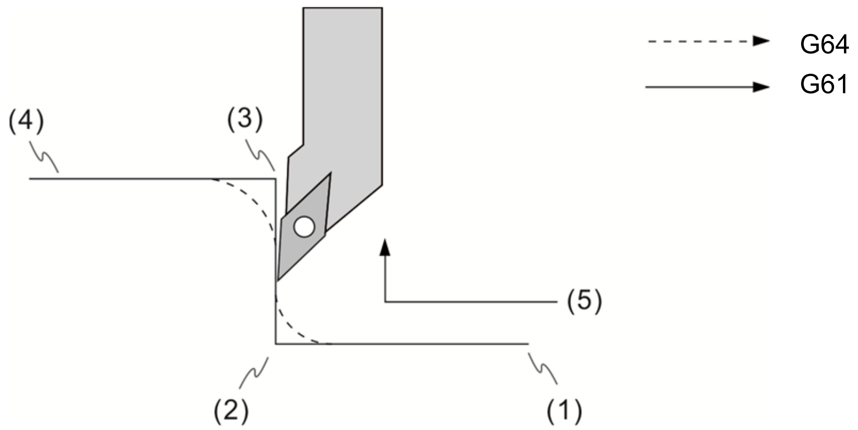
G61: Exact stop mode

Format: G61;

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

Note: The machine default is G64.

[Example]



(1) G61 G00 X0.0 Z0.0 (Assign G61)
(2) G01 Z-50. F0.2 (G61 is continuous effective)
(3) X50. (G61 is continuous effective)
(4) W-50. (G61 is continuous effective)
(5) Feeding direction

G64: Cutting mode command

Format: G64

Description: When G64 is applied 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. With this command, the NC tool path becomes smoother when machining. Command G64 differs from G61 in that it enables the tool to cut at a constant feed rate and does not decelerate to stop between motion blocks.

However, deceleration to full stop for inspection will take place in the following circumstances even when G64 is in use:

(1) The system executes the block that contains G00 Fast positioning command.
(2) The system executes the block that contains G09 Exact stop command.
(3) The system executes the block followed by the block without a motion command.

G65: Non-continuous effect macro calling command

Format: G65 P_ L_I

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

Description: Command G65 calls a macro program. Macro program is applicable to commands such as operations, MLC interface data input/output, control, making statement and discrepancy. Thus, the system is able to do calculation and measurement.

Macro program makes a subroutine with different types of control functions such as variables, calculation commands, and control commands. In the main program, a macro is active only when it is called. This command works the same way as M98 does except that it is non-continuous.

[Example]

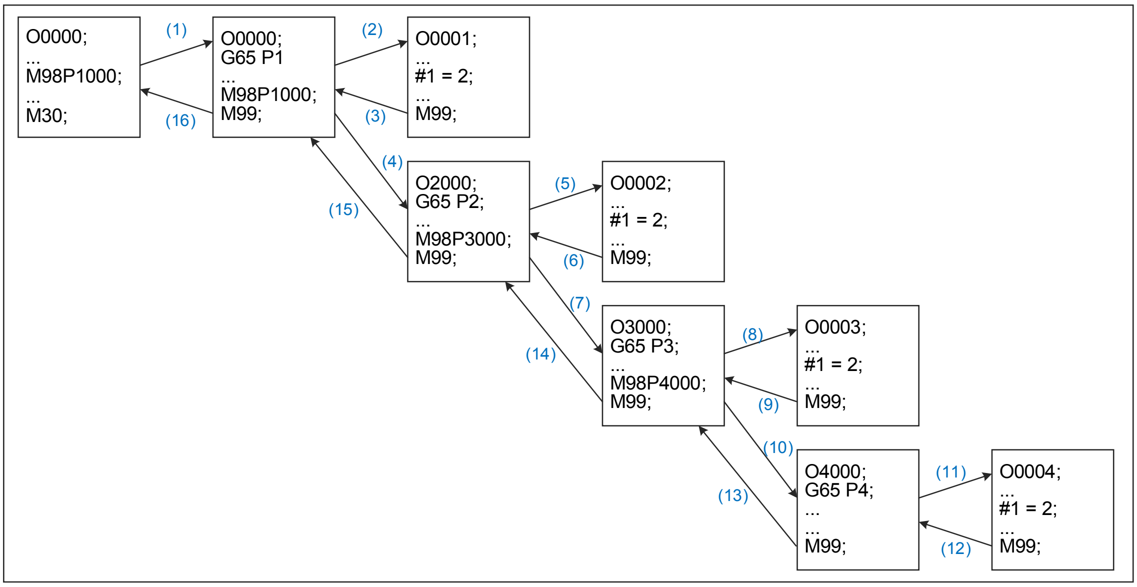
Main program	Subroutine
N0010	O1010
N0020 G65 P1010 L2 A5.0 B3.0	N1020 #3=#1+#2
N0030	N1030
N0040	N1040
N0050	N1050 M99

2

After the macro program is executed, the program will return to the block following G65 command in the main program. That is, the block that follows G65 will carry on to be executed. See the example above. A5.0 represents that the value of local variable #1 is 5.0. . Please see the table below for corresponding position and variable.

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

[Program Illustration]



Command G65/G66 can nest macros up to 8 layers. When used together with subroutine calling command M98, the program nest layer number remains eight.

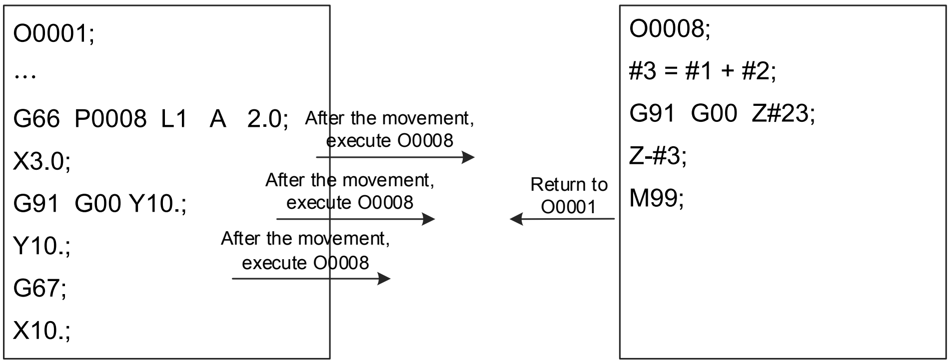
G66/G67: Continuous effect macro calling command / Cancelling command

Format: G66 P_ L_I
Or G67

P_: Program code
L_: Number of repetitions
I_ : Value of independent variable

Description: Command G66 functions the same as G65 except that instead of being active for a single block, the former keeps on calling the macro in later statements until it is canceled by command G67.

[Example]



G71: Multiple type rough turning cycle (Outer diameter)

Format: G71 Ud Re;

G71 P_ Q_ Uu Ww F_ S_ T_ ;

Ud: Roughing depth per move in X-axis direction (users can only input radius value); use machining parameter 312 to specify the default value.

Re: Retraction amount (users can only input radius value), which default value can be specified by machining parameter 313.

P_: Start block number for contour finish turning.

Q_: End block number for contour finish turning.

Uu: Finish allowance of X-axis (diameter/radius)

Ww: Finish allowance of Z-axis (diameter/radius)

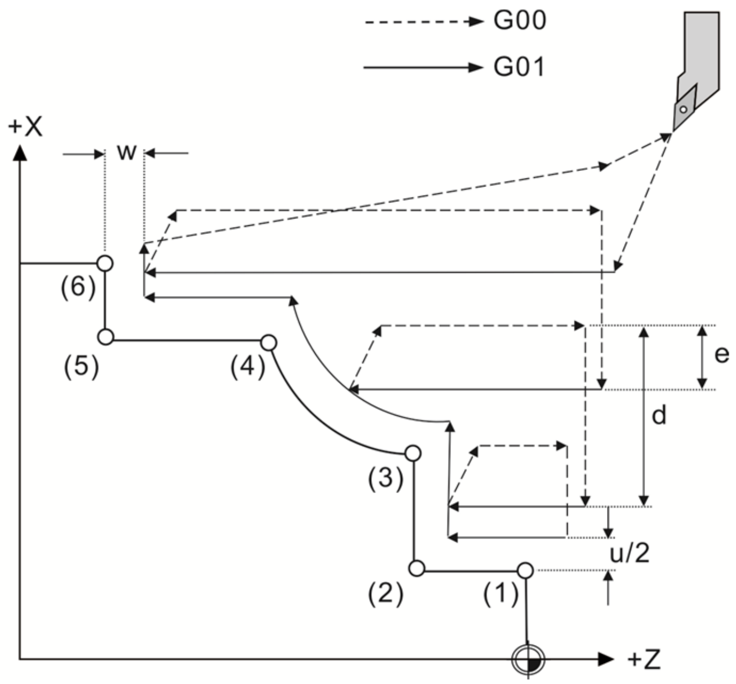
F_: Feed rate

T_: Tool number

S_: Spindle speed

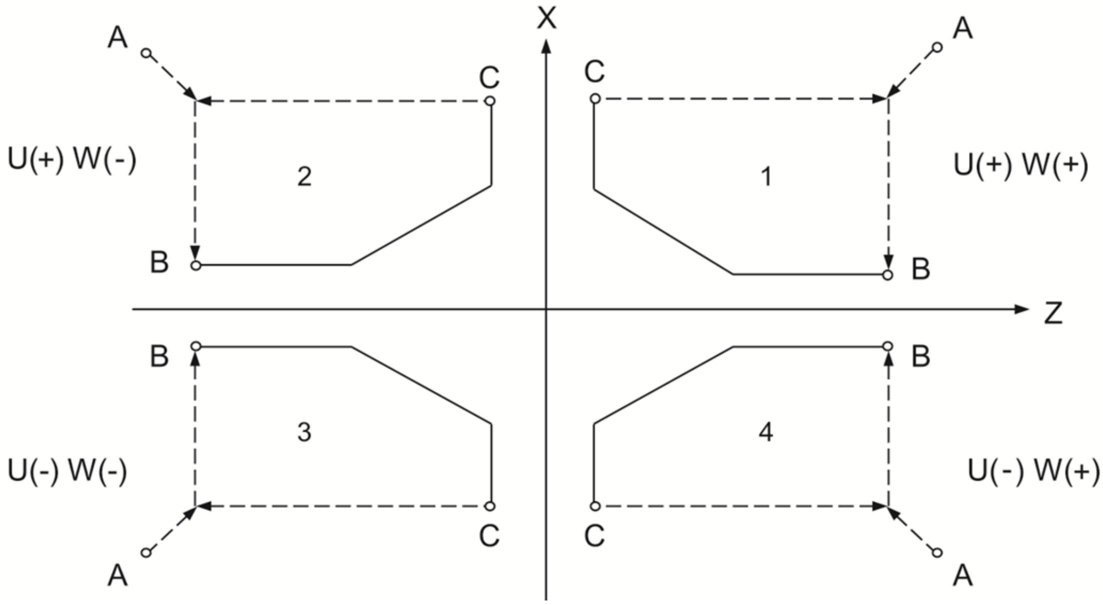
Description: When G71 is executed, the system will read the end dimension first. Next, it automatically calculates the rough turning path for the outer diameter of workpiece by referring to the parameter setting and starts rough turning.

[Example]




```
S1000
N50 G01 X5.0 Z0.0 F0.15   (1)
Z-5.   (2)
X10.   (3)
G02 X20. Z-15. R10.   (4)
G01 Z-20.   (5)
G54 X40.0 Z5.0
M03 S1000
G71 U2. R3.
G71 P50 Q60 U0.2 W0.0 F0.2
N60 X40.   (6)
M5
M30
```

[Workpiece position and cutting direction]



The figure above shows the workpiece position and the cutting direction with respect to positive/negative U and W; 1 ~ 4 stands for the four quadrants respectively.

A: Start point of the cycle
B: Start point of the cutting path
C: End point of the cutting path

Note:
(1) G71 rough turning cycle function can be used to perform turning for the concave-shaped workpiece.
(2) Do not repeatedly assign the P_ and Q_ values that specify the finish cutting sequence in the program.
(3) The blocks that indicates no movement and without N, F, S, M, and T values specified will be ignored.
(4) When the command that does not specify the cutting depth and retraction amount, the system will refer to the parameter setting automatically.
(5) Tool compensation is not available when G71 rough turning cycle is used.
(6) An alarm will occur if the system does not read the P_ and Q_ values that specify the contour finish cutting sequence in G71.
(7) G71 is a non-continuous effective command.

G72: Multiple type rough facing cycle

Format: G72 Wd Re;

G72 P_ Q_ Uu Ww F_ S_ T_ ;

Wd: The cutting depth per move in Z-axis direction (specified by radius); use machining parameter 312 to specify the default value.

Re: Retraction amount (set as radius), which default value can be specified by machining parameter 313.

P_: Start block number for contour finish turning.

Q_: End block number for contour finish turning.

Uu: Finish allowance of X-axis (set as diameter/radius)

Ww: Finish allowance of Z-axis (set as diameter/radius)

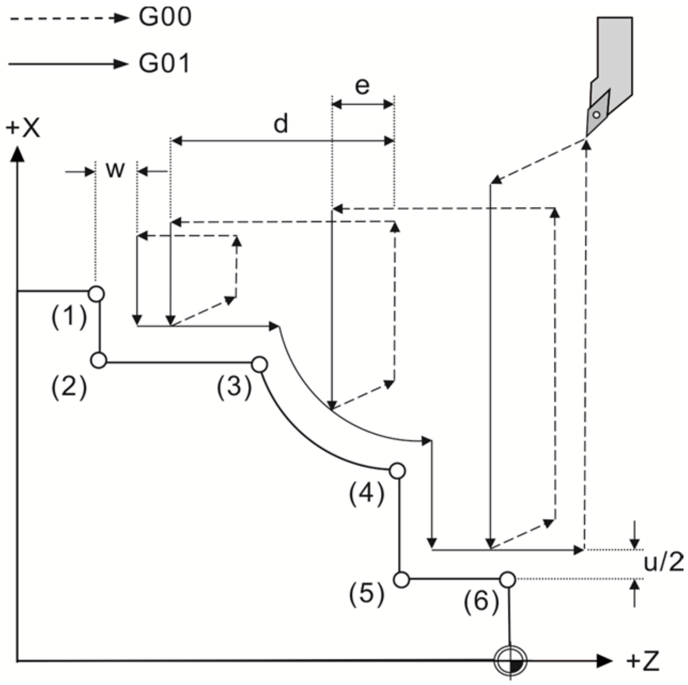
F_: Feed rate

T_: Tool number

S_: Spindle speed

Description: When G72 rough facing cycle is used, the system will read the end dimension first. Then, it automatically calculates facing path for the workpiece by referring to the parameter setting and starts the rough facing cycle.

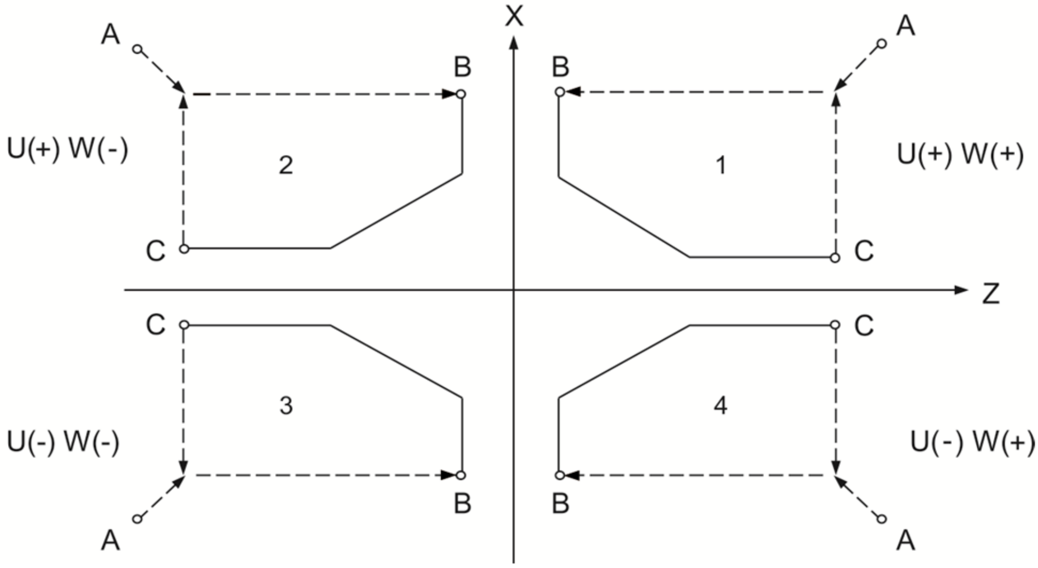
[Example]



Machining program:

G54 X40.0 Z5.0
M03 S1000
G01 X45. Z5. F0.2
G72 W2. R3.
G72 P50 Q60 U0.2 W0.0 F0.15
N50 G01 X40. Z-20. (1)
X30. (2)
Z-15. (3)
G03 X20. Z-5. R10. (4)
G01 X5. (5)
N60 Z0. (6)
M5
M30

[Workpiece position and cutting direction]



The figure above shows the workpiece position and the cutting direction with respect to positive/negative U and W; 1 ~ 4 stands for the four quadrants respectively.

A: Start point of the cycle
B: Start point of the cutting path
C: End point of the cutting path

Note:

(1) Do not repeatedly assign the P_ and Q_ values that specify the finish cutting sequence in the program.
(2) The blocks that indicates no movement and without N, F, S, M, and T values specified will be ignored.
(3) When the command does not specify the cutting depth and retraction amount, the system will refer to the parameter setting automatically.
(4) Tool compensation is not available when G72 rough facing cycle is used.
(5) An alarm will occur if the system does not read the P_ and Q_ values that specify the contour finish cutting sequence in G72.
(6) G72 is a non-continuous effective command.

G73: Multiple type pattern repeating cycle

Format: G73 Ui Wk Rd;
G73 P_ Q_ Uu Ww F_ S_ T_;

Ui: The total cutting amount in X-axis direction (set as radius), which default value can be specified by machining parameter 345.

Wk: The total cutting amount in Z-axis direction (set as radius), which default value can be specified by machining parameter 346.

Rd: Times of cutting, which default value can be specified by machining parameter 347.

P_: Start block number for contour finish turning.

Q_: End block number for contour finish turning.

Uu: Finish allowance of X-axis and the direction of allowance (set as diameter/radius)

Ww: Finish allowance of Z-axis and the direction of allowance (set as diameter/radius)

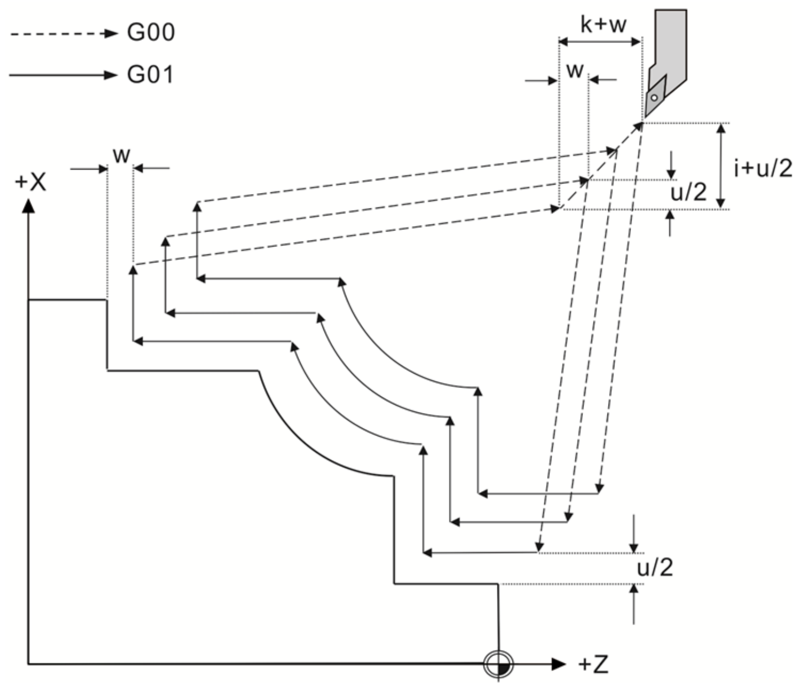
F_: Feed rate

S_: Spindle speed

T_: Tool number

Description: When G73 pattern repeating cycle is used, the system will read the end dimensions first. Then, it automatically calculates the turning path for the workpiece by referring to the parameter setting and starts the pattern repeating cycle.

[Contour machining example]

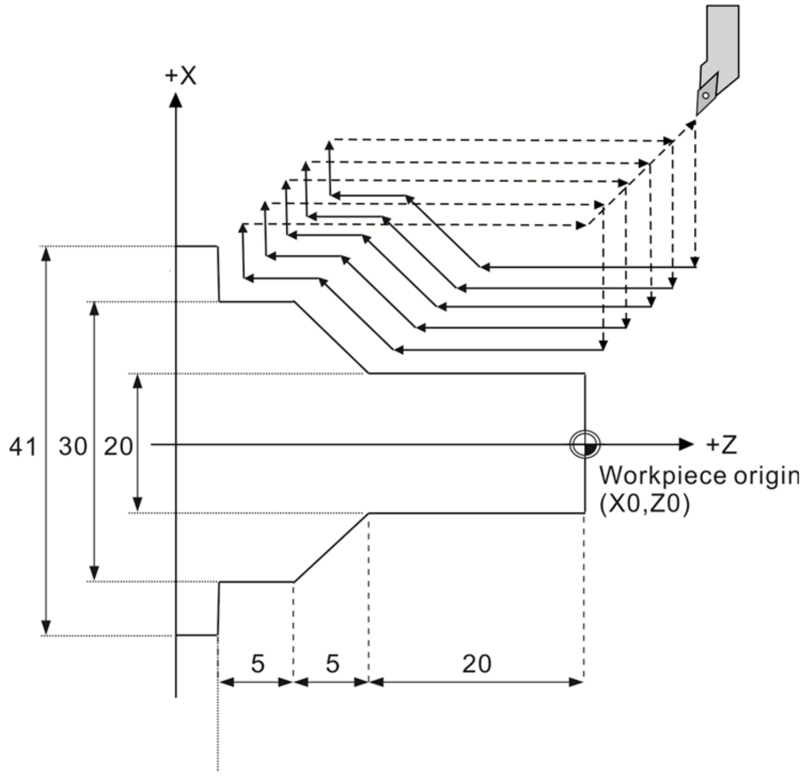


The cutting amount per time: [the total cutting amount of each axis]÷ [value of cutting time (d-1)]

X-axis direction: $\frac{i}{(d-1)}$, Z-axis direction: $\frac{k}{(d-1)}$

Note: G73 cycle command is suitable for the workpiece that has been machined (such as roughing, forging, casting). If the workpiece is a complete bar, using G73 may result in tool or workpiece damage as turning amount is excessive.

[Example]



In this program, a bar with diameter 42 mm is used for machining.

```
O0007
M3 S1600
T3
G0 X41. Z2.
G73 U10. W10. R5.
G73 P50 Q60 U0.4 W0.2 F0.25
N50 G0 X20.
G1 Z-20. F0.12
X30. Z-25.
W-5.
N60 X41.
G0 X50. Z10.
M5
M30
```

2

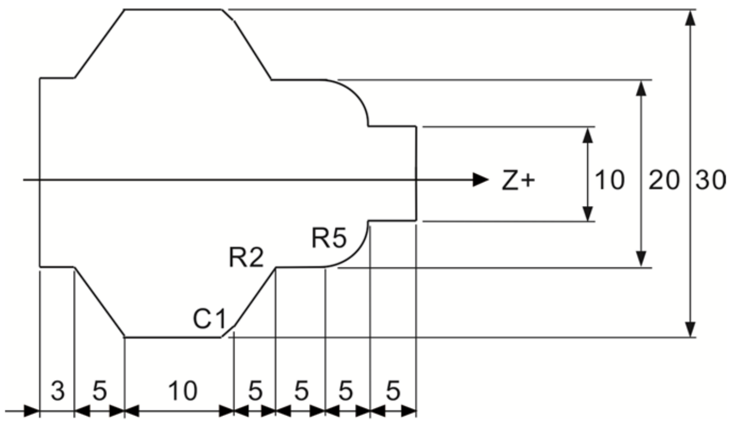
G70: Multiple type finish turning cycle

Format: G70 P_ Q_;

P_: Start block number for contour finish turning.
Q_: End block number for contour finish turning.

Description: G70 can be applied for finish turning to meet the end dimension requirement after rough turning cycles (G71, G72, and G73) are executed.

[Contour machining example]




```
M3 S1000
T3
G0 X41. Z2.
G71 U2. R3. (Rough turning)
G71 P50 Q60 U0.2 W0.2 F0.25
N50 G0 X10. Z0.5
G1 Z-5. F0.12
G03X20. W-5. R5.
G01Z-15.,R2.
U10.Z-20.,C1.
Z-30.
X20.Z-35.
W-3.
N60 X40.2
G70 P50 Q60 (Finish turning)
M5
M30
```

Note:

Once the G70 finish turning is complete, the tool returns to the start point in rapid traverse and the blocks that follow G70 will continue to be executed.

[Example]

After all rough turning cycles are complete, all G70 finish turning cycles are executed.

```
G71 ...
G71 P10 Q20 ...
N10
  …
  ...
N20
  ...
G71 ...
G71 P30 Q40 ...
N30 ...
  ...
  ...
N40 ...
  ...
G70 P10 Q20
G70 P30 Q40
```

2

G74: Multiple type face pecking cycle

Format: G74 Re;

G74 X/U_ Z/W_ P△i Q△K R△d F_;

Re: Retraction amount of Z-axis, which default value can be specified by machining parameter 348.

X/U__: Coordinates of cutting end on X-axis/ Incremental distance of X-axis

Z/W__: Coordinates of cutting end on Z-axis/ Incremental distance of Z-axis

P△i: The tool feeding amount of X-axis per cycle. This value can only be set with radius. When it is an integer, the unit will be 0.001 mm.

Q△k: The peck turning amount of Z-axis per time. When it is an integer, the unit will be 0.001 mm.

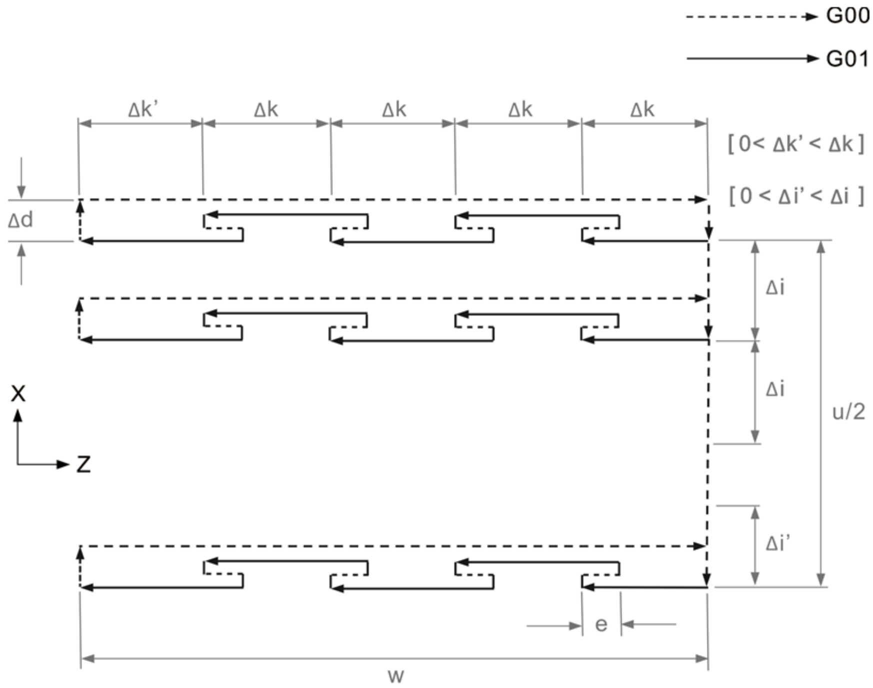
R△d: The escape distance in X-axis direction at the bottom of cut.

F_: Feed rate

Description: G74 is mainly used in the applications of face grooving. When G74 is applied, the facing cycle will be executed based on the specified values in the command, including coordinates for turning end, cutting amount, tool offset, escape distance at the bottom of cut.

After completing the turning for amount △k in Z-axis direction each time, the tool retracts for amount e. This cycle carries on and stops when reaching the target on Z-axis (bottom). Then, the tool escapes for amount △d and returns to the start point of Z-axis in rapid traverse. Next, the tool moves for amount △i and repeats the motion mentioned above. Finally, it stops when reaching the target on X-axis. See the following figure for motion assigned by G74:

Note: Tool nose compensation is not available when G74 is used.



G75: Multiple type axial pecking cycle

Format: G75 Re;

G75 X/U_ Z/W_ P△i Q△K R△d F_;

Re: The retraction amount of X-axis after each peck turning. This value can only be set with radius. The default value can be specified by machining parameter 348.

X/U__: Coordinates of cutting end in X-axis direction/ Incremental distance of X-axis

Z/W__: Coordinates of cutting end in Z-axis/ Incremental distance of Z-axis

P△i: The peck turning amount per time. This value can only be set with radius. When it is an integer, the unit will be 0.001 mm.

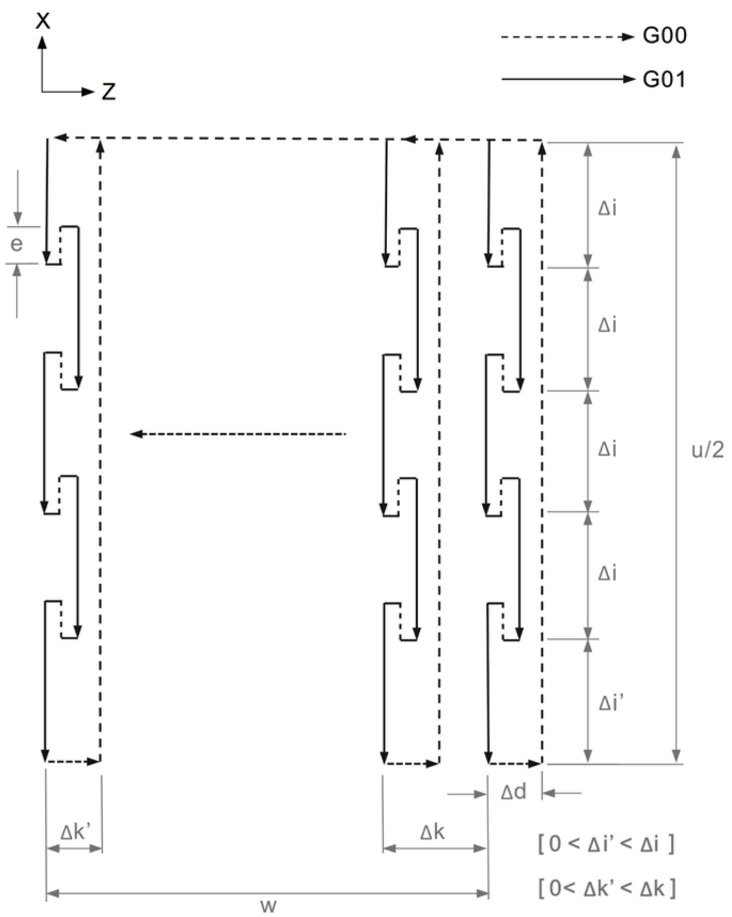
Q△k: The tool feeding amount of Z-axis per cycle. When it is an integer, the unit will be 0.001 mm.

R△d: The escape distance in Z-axis direction at the bottom of cut.

F__: Feed rate

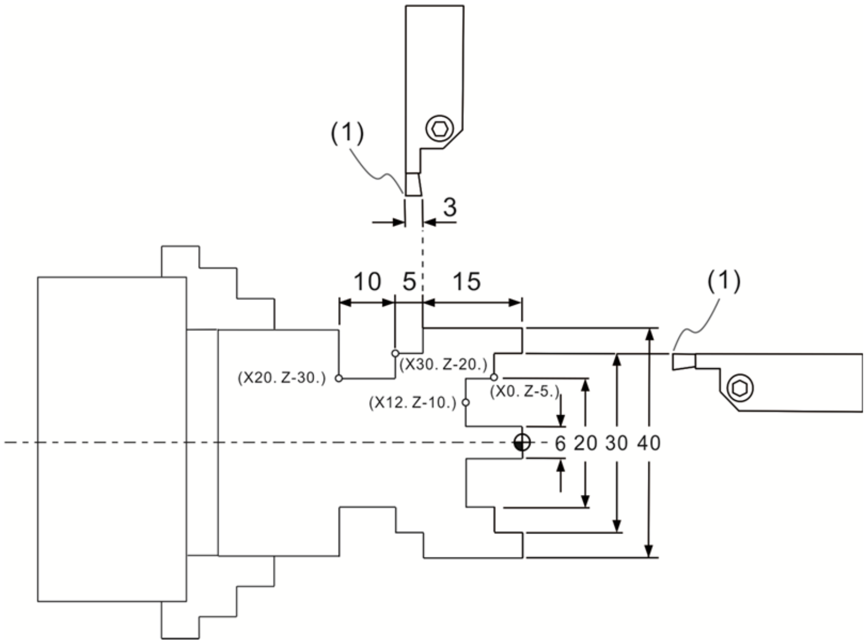
Description: G75 is mainly used for grooving in axial direction. When G75 is applied, the system will perform auto axial pecking cycle based on the value specified in G75, including the set coordinates for turning end, cutting amount, tool offset, and escape distance at bottom of cut.

After completing the turning for amount △i, the tool retracts for distance e. This motion cycle carries on and stops when reaching the target (bottom) on X-axis, and the tool escapes for amount △d. Then, the tool returns to start point of X-axis in rapid traverse. And the tool moves for amount △k in Z-axis direction and carry on making the motions mentioned above. Finally, this cycle stops at the target on Z-coordinates. See the following figure for motion assigned by G75:



Note:
Tool nose compensation is not available when G75 is applied.

[Example]



(1) Tool nose position

T0404
M3 S2000
G0X30.Z5. (Start point of G74 cycle)
G74 R1.
G74 X20. Z-5. P3000 Q5000 R0. F0.3
G0X20.Z5.
G74 R1.
G74 X12. Z-10. P3000 Q5000 R0. F0.3 (Calculating the tool width is required)
G0Z5.
X50.
T0505
G0 Z-18. (Start point of G75 cycle; calculating the tool width is required)
G75 R1.
G75 X30. Z-20. P5000 Q3000 R0. F0.3
G0Z-20.
G75 R1.
G75 X20.Z-30.P5000 Q3000R0. F0.3
G0 X50.
Z5.
M5
M30

G76: Multiple type thread turning cycle

Format: G76 Pmra Q△dmin R_;
G76 X/U_ Z/W_ Ri Pk Q△d F_;

Pmra: m stands for the times of finish turning (1 ~ 99), which default value can be specified by machining parameter 381; r stands for the chamfering amount (0 ~ 99). Assume that L is the threading lead, the chamfering setting is 0.1*r*L and its default value can be set by machining parameter 380. a stands for the tool nose angle (threading angle). Its default value can be specified by machining parameter 382 (with options of 0°, 29°, 30°, 55°, 60°, and 80°).
For example, when P011160 is set, it means the finish turning time is 1, chamfering amount is 1.1 L (L is the threading lead), and tool nose angle is 60°.

Q△dmin: Minimum cutting depth. When an integral is input, the unit is 0.001 mm and its default value can be specified by machining parameter 383.

R_: Finish allowance, which is specified in radius value and its default value can be set by machining parameter 439.

X/U_: Coordinates of the threading end point in X-axis direction/Incremental threading distance of X-axis.

Z/W_: Coordinates of the threading end point in Z-axis direction/Incremental threading distance of Z-axis.

Ri: The difference when [Start screw thread radius] minus [End screw thread radius] (set as radius value)

Pk: Thread depth (set as radius value). When an integer is input, the unit will be 0.001 mm.

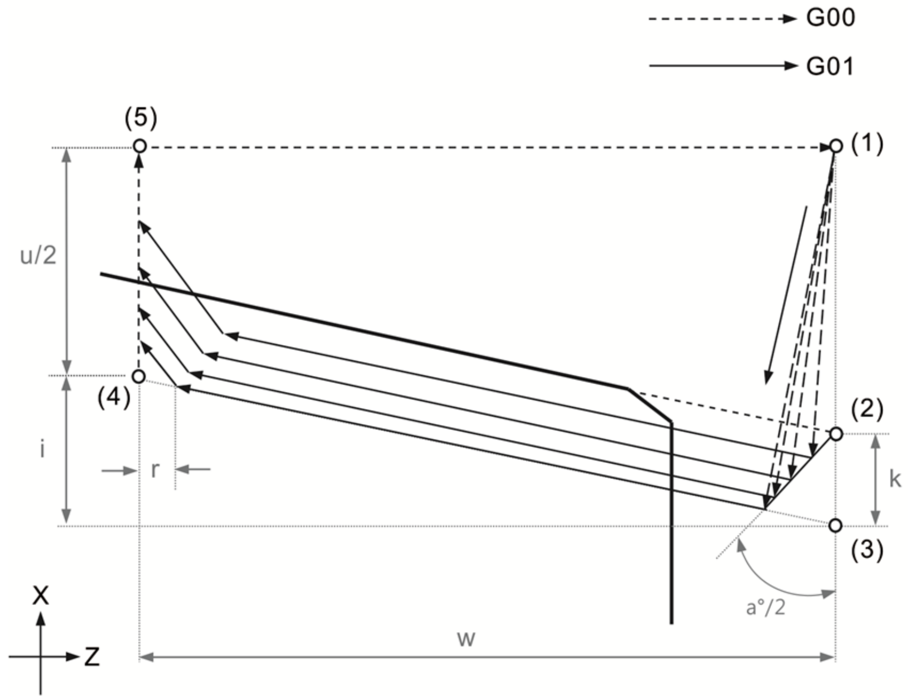
Q△d: Depth of the first cut (set as radius value). When an integer is input, the unit will be 0.001 mm.

F__: Thread lead; the linear distance of one thread rotation.

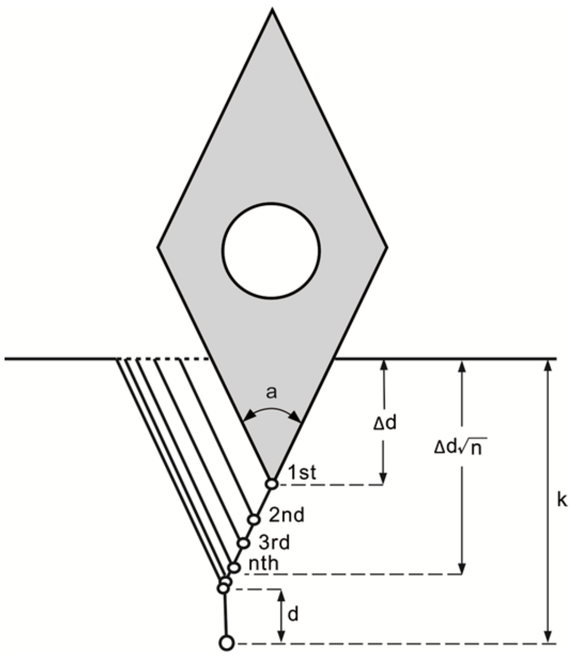
Description: When the coordinates of cutting end is given in G76, the system will perform thread turning cycle according to the turning times specified.

2

[Thread turning]



(1) ~ (5) in the figure represent the tool movement sequence.



Note:
(1) G76 Thread turning cycle has to be executed with constant spindle speed.
(2) When G76 is executed, spindle speed keeps at 100%.

G90: Axial turning cycle

Format: G90 X/U_ Z/W_ R_ F_;

X/U_: Coordinates of the cutting end in X-axis direction/ Incremental distance of X-axis (set as diameter/radius)

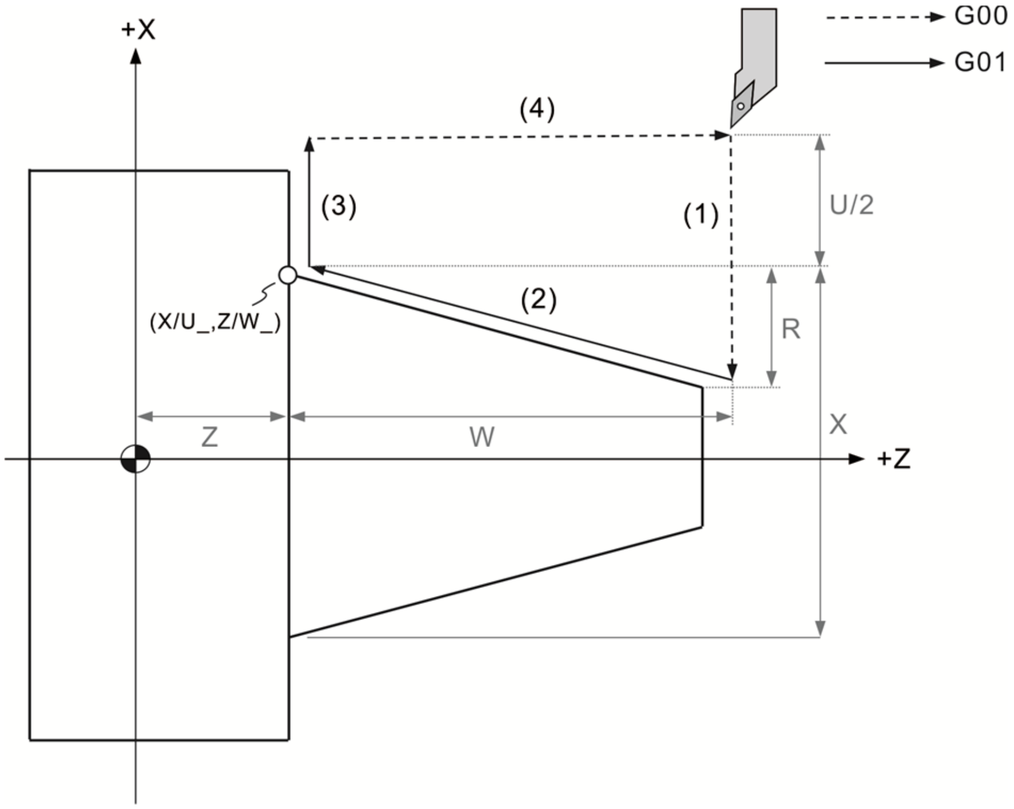
Z/W_: Coordinates of the cutting end in Z-axis direction/ Incremental distance of Z-axis

R_: Distance and direction of taper (set as radius with positive/negative sign). When axial linear turning, R_ value will be ignored.

F_: Feed rate

Description: When G90 is applied, the system will perform a complete outer diameter (taper) turning cycle.

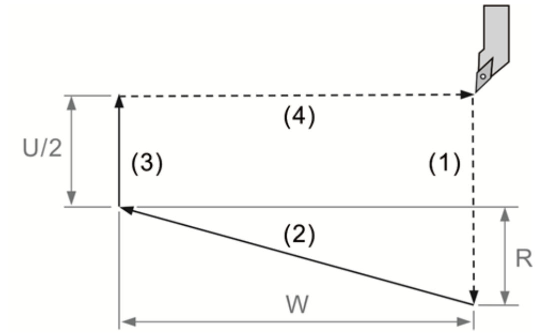
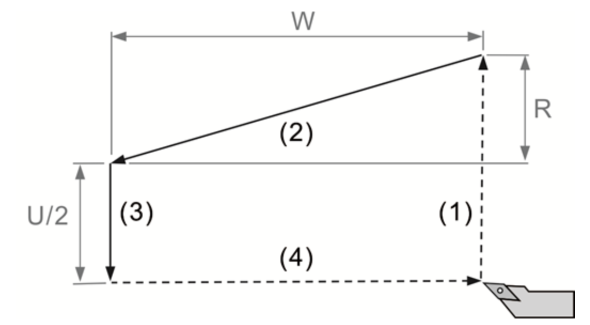
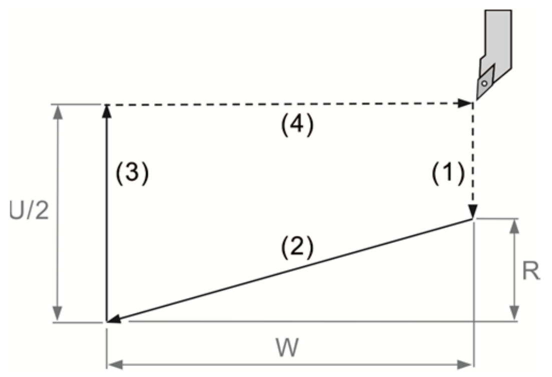
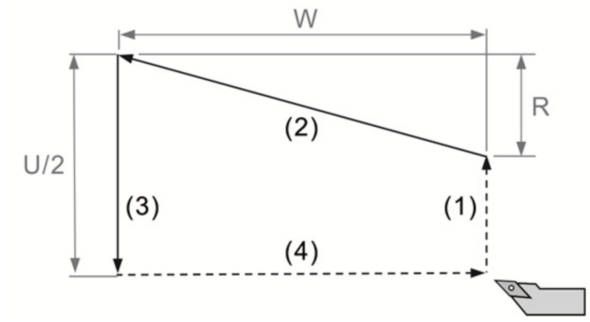
[Contour machining example]



G90 motion analysis:

(1) Move from the start point to the assigned coordinates of X-axis according to the given taper distance/direction in rapid traverse.
(2) Straight cut to the assigned coordinates of Z-axis and X-axis.
(3) Straight cut to the start point of the X-axis.
(4) Return to the start point in rapid traverse.

The turning directions correspond to symbol U, W, and R:

Outer diameter machining	Inter diameter machining
1. U < 0, W < 0, R < 0	3. U > 0, W < 0, R < 0 at |R|≤|U/2|
	
2. U < 0, W < 0, R > 0 at |R|≤|U/2|	4. U > 0, W < 0, R > 0
	

Note:
When G90 axial turning cycle is applied, its setting is continuous effective. To cancel the G90 command, you can use G92, G94, G80 (cycle cancel), or other commands in Group 1 (G00, G01, G02, and G03).

G92: Threading cycle

Format: G92 X/U_ Z/W_ R_ F_ Q_;

X/U_: Coordinates of the cutting end in X-axis direction/ Incremental distance of X-axis (set as diameter/radius)

Z/W_: Coordinates of the cutting end in Z-axis direction/ Incremental distance of Z-axis

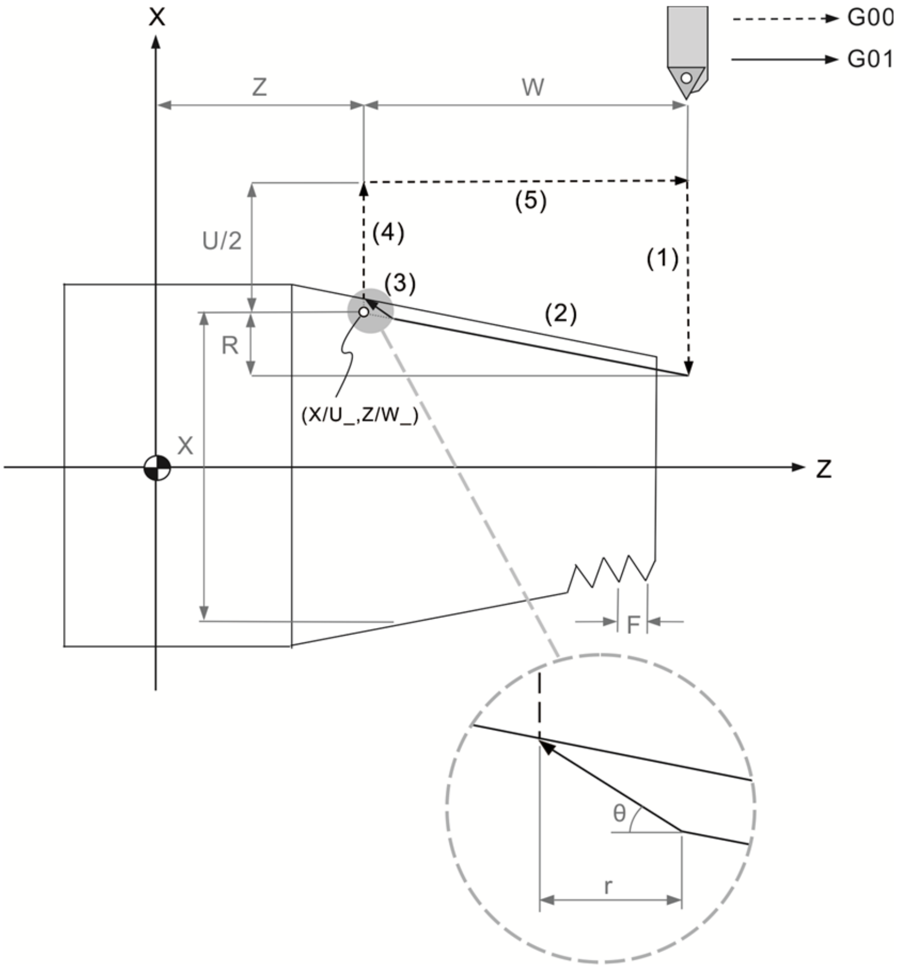
R_: Distance and direction of taper (set as radius value with positive/negative sign) When performing straight cutting, R_ value will be ignored.

F_: Thread lead

Q_: Offset angle for threading start. When an integral is input, the unit is 0.001 degree.

Description: When G92 is applied, the system will perform a complete outer diameter (taper) threading cycle.

[Contour machining example]



ϴ is the chamfering angle for thread turning, which can be specified by machining parameter 349.

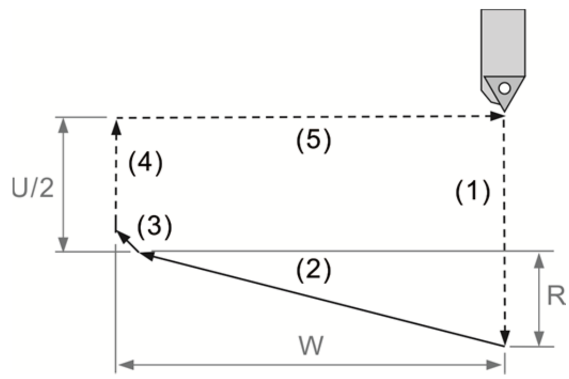
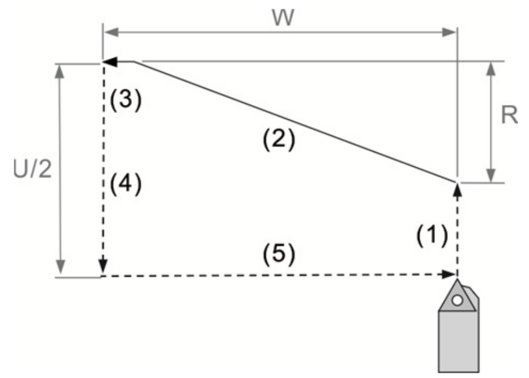
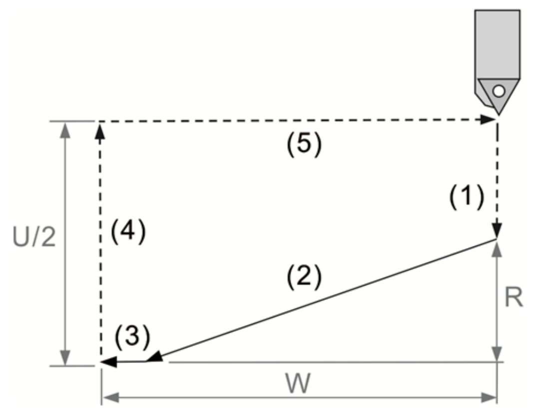
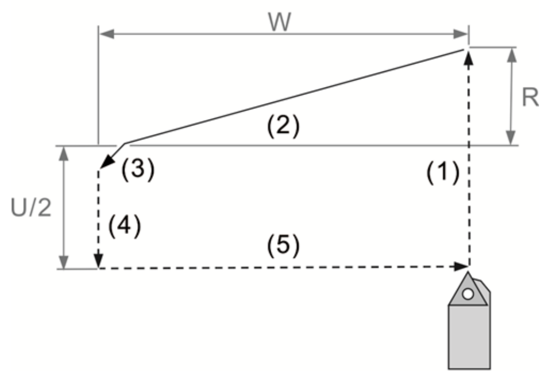
r is the chamfering length, which can be specified by machining parameter 380.

2

G92 motion analysis:

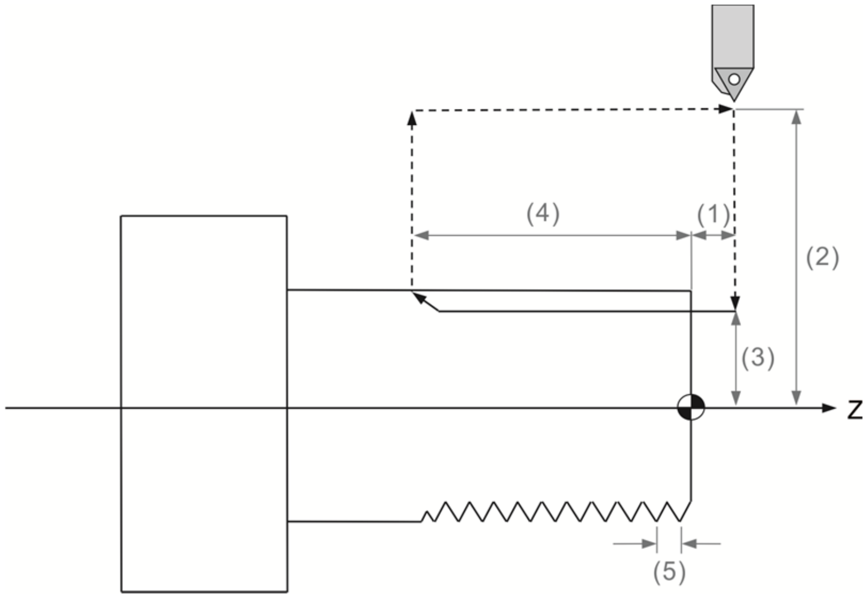
(1) Move from the start point to the assigned coordinates of X-axis according to the given taper distance/direction in rapid traverse.
(2) Make thread turning to the specified coordinates of the Z-axis and X-axis.
(3) Execute chamfering command.
(4) Move to the start point on X-axis in rapid traverse.
(5) Return to the start point in rapid traverse.

The turning directions corresponding to symbol U, W, and R:

<table>
<tr><th>Outer diameter machining</th><th>Inter diameter machining</th></tr>
<tr><td>1. U < 0, W < 0, R < 0</td><td>3. U > 0, W < 0, R < 0 at |R|≤|U/2|</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>2. U < 0, W < 0, R > 0 at |R|≤|U/2|</td><td>4. U > 0, W < 0, R > 0</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

Note:
(1) When G92 is applied, its setting is continuous effective. To cancel G92 command, you can use G90, G94, G80 (cycle cancel) or commands in Group 1 (G00, G01, G02, G03).
(2) The default chamfer angle for thread turning can be set by machining parameter 349; the default chamfer length can be set by machining parameter 380.
(3) Refer to the note of G33 thread turning command when applying thread turning.

[Example]



O0010

T0202 (Select tool No.2 and tool compensation No.2)

M03 S2000 (Spindle rotates forward at 2000 rpm)

G0 Z5. ((1) Threading start point of Z-axis)

X30. ((2) Retraction distance of X-axis)

G92 X17.65 Z-25.0 F1.5 ((3) Threading depth:17.65 mm; (4) length: 25 mm; (5) Pitch: 1.5 mm)

X17.45 (Threading depth)

X17.25

X17.05

X16.85

X16.65

X16.45

X16.25

X16.05

X15.9

M5 (Spindle stop)

G0X50. (The tool retracts and keeps a safe distance)

Z10. (The tool retracts and keeps a safe distance)

M30 (Program end)

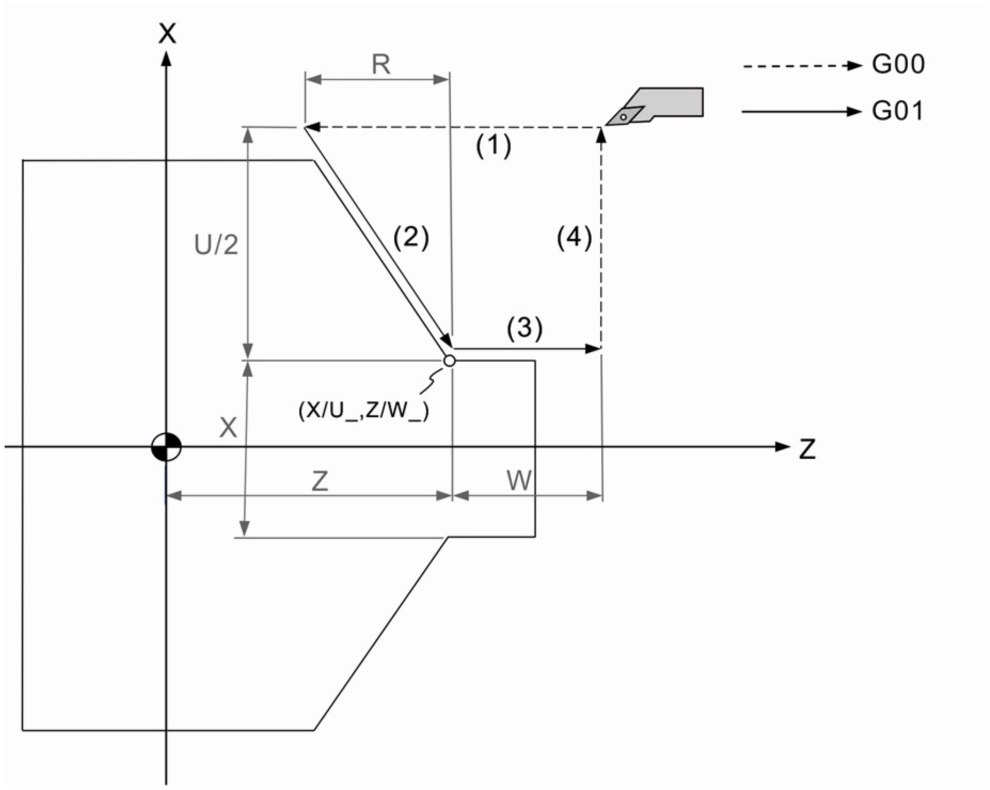
G94: End face turning cycle

Format: G94 X/U_ Z/W_ R_ F_;

X/U_: Coordinates of cutting end point in X-axis direction/ Incremental distance of X-axis (set as diameter/radius)
Z/W_: Coordinates of cutting end point in Z-axis direction/ Incremental distance of Z-axis (set as diameter/radius)
R_: Taper amount (as radius value with positive/negative sign). In straight cutting, R__ will be ignored.
F_: Feed rate

Description: When G94 is applied, the system will perform a complete straight cutting (tapering) cycle of end facing.

[Contour machining example]

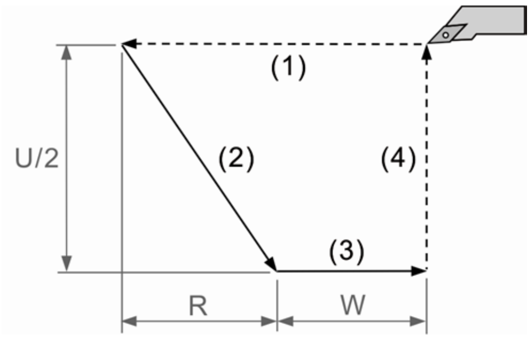
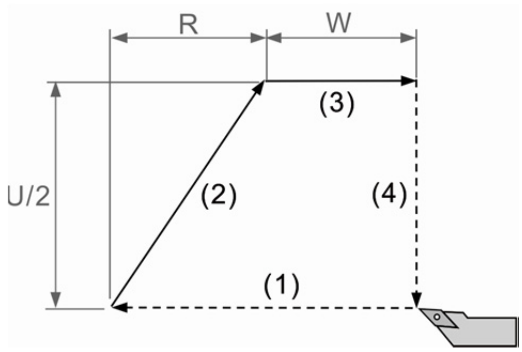
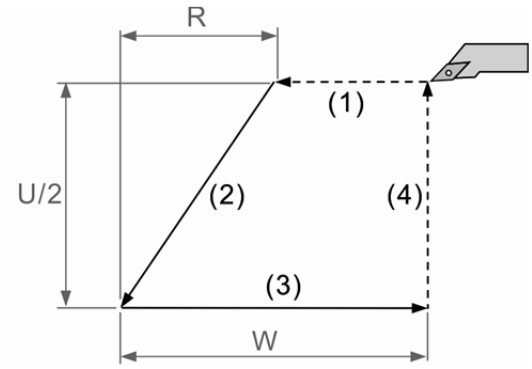
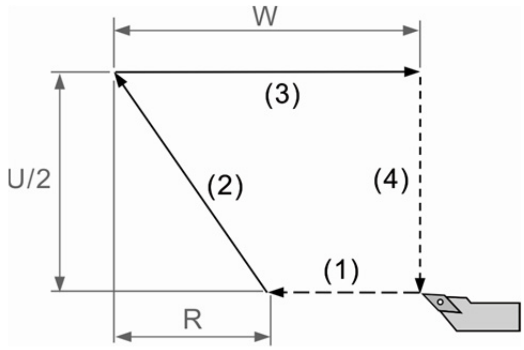


G94 motion analysis:

(1) The tool moves from the start point and go to [the specified coordinates of Z-axis + R the tapering amount] in rapid traverse.
(2) Perform straight cuts all the way to the specified coordinates of Z-axis and X-axis.
(3) Perform straight cuts to the specified start point of Z-axis.
(4) Return to the start point in rapid traverse.

2

The directions corresponding to symbol U, W, and R for turning:

<table>
<tr><th>Outer diameter machining</th><th>Inter diameter machining</th></tr>
<tr><td>1. U < 0, W < 0, R < 0</td><td>3. U > 0, W < 0, R < 0 at |R| ≤ |W|</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>2. U < 0, W < 0, R > 0 at |R| ≤ |W|</td><td>4. U > 0, W < 0, R > 0</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

(1) ~ (4) in the figure represent the tool movement sequence.

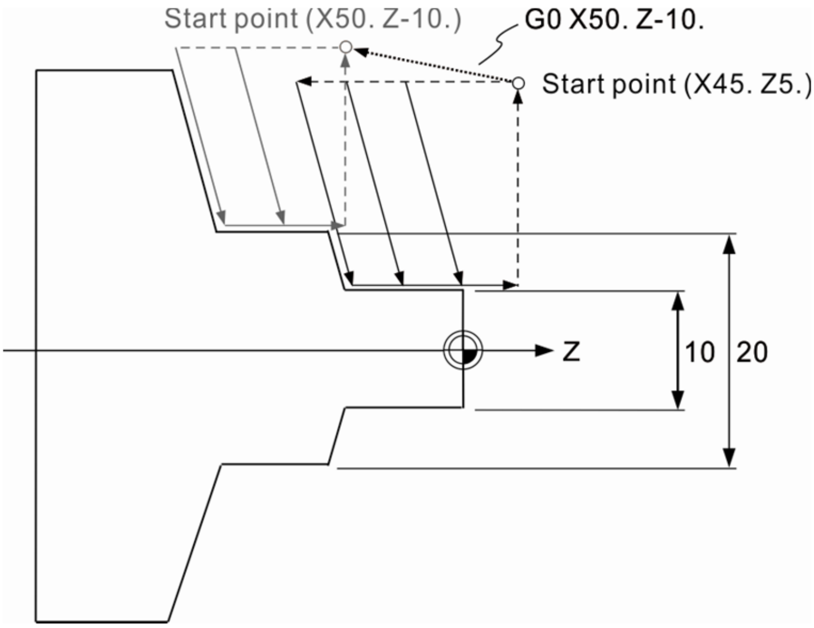
Note:
G94 end face turning cycle is a continuous effect command. To cancel G94 command, you can apply G90, G92, G80 (Cycle cancel) or commands (G00, G01, G02, and G03) in Group 1.

G80: Cycle cancel

Format: G80;

Description: This command is used for canceling G90, G92, and G94 cycles.

[Example]



G98

G0 X45. Z5. (Start point of the cycle)

G94 X10. Z0. R-5. F500

Z-5.

Z-10.

G0 X50. Z-10. (Start point of the cycle- G0 can be used to cancel the canned cycles G90, G92, and G94)

G94 X20. Z-15. R-5. F500

Z-20.

G80 (Cancel the canned cycle)

M5

M30

2

G83: Face peck drilling cycle

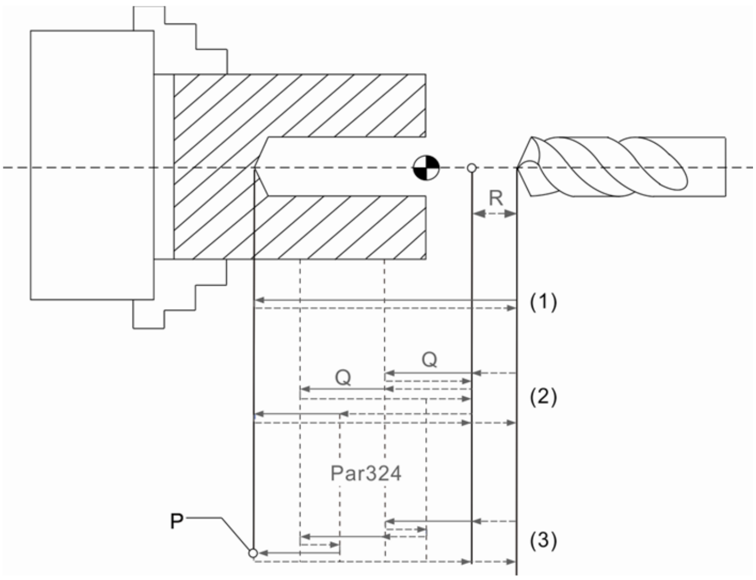
Format: G83 X(U)_ Z(W)_ R_ Q_ P_ F_ K_;

X(U)_: Drilling position; It is the 0 point of X-axis during face drilling;
Note: If the coordinates of X-axis is not the 0 point of workpiece coordinate system, an alarm showing “B6A5 Drilling tapping error” will occur.
Z(W)_: Bottom of the hole
R_: The reference point, which moves toward the opposite direction of the workpiece This is set as an incremental value.
Q_: Feeding distance per time (per peck)
P_: Dwell time at bottom of the hole. The unit is ms. Note: This value has to be integral.
F_: Feed rate
K_: Number of repeat machining time. The default value is 1.

Description: Each time the tool drills the distance (depth) Q, it retracts the distance set by parameter 324 in rapid traverse or retracts to point R according to the setting of P326. This cycle will carry on and stops when the tool drills to Z.

Cycle parameter 326:
If bit 2 ~ 3 are set to 0, the tool does not perform peck drilling; it drills directly to the bottom.
If bit 2 ~ 3 are set to 1, the tool performs deep peck drilling, which feeding amount is Q and retracts to point R.
If bit 2 ~ 3 are set to 2, the tool performs general peck drilling, which feeding amount is Q and the escape amount is set by parameter 324.

[Example]



(1) Parameter 326:
bit2~bit3 = 0
X_ Z_ (Go to the drilling position)
G83 X_ Z_ F_

(2) Par326: bit2~bit3=1
X_ Z_
G83 X_ Z_ R_ Q_ F_

(3) Par326: bit2~bit3=2
X_ Z_
G83 X_ Z_ R_ Q_ P_ F_

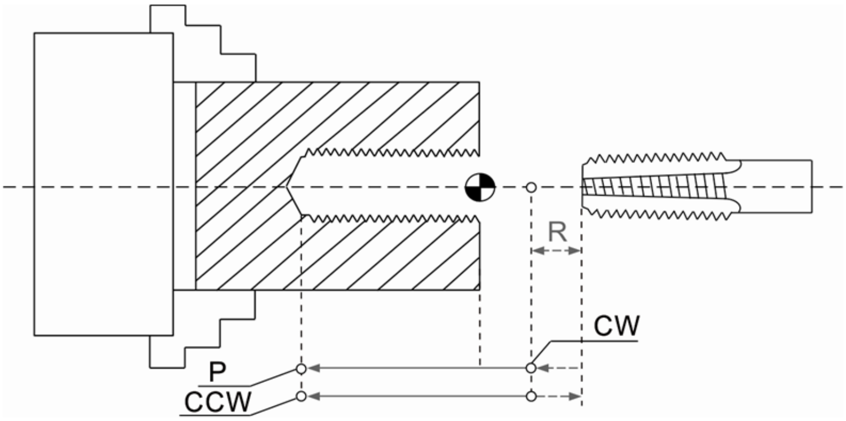
G84: Face tapping cycle

Format: G84 X(U)_ Z(W)_ R_ P_ F_ K_ ;

X(U)_: Tapping point; This is the 0 point of X-axis during face tapping.
Note: If the coordinates of X-axis is not the 0 point of workpiece coordinates, an alarm showing “B6A5: Drilling tapping error” will occur.
Z(W)_: Bottom of the hole
R_: The reference point, which moves toward the opposite direction of the workpiece This is set as an incremental value.
P_: Dwell time at bottom of the hole. The unit is ms. Note: This value has to be integral.
F_: Feed rate
K_: Number of the repeat machining time. The default value is 1.

Description: The spindle will rotate once to look for the Z pulse and execute positioning once Z pulse is found. Then, Z-axis will move to reference point R in rapid traverse. Next, the spindle rotates and starts tapping to the specified position (bottom) and then retracts to R in reverse direction and returns to the initial point in rapid traverse.

[Example]



X_ Z_ (Fast positioning)
G84 X_ Z_ R_ P_ F_

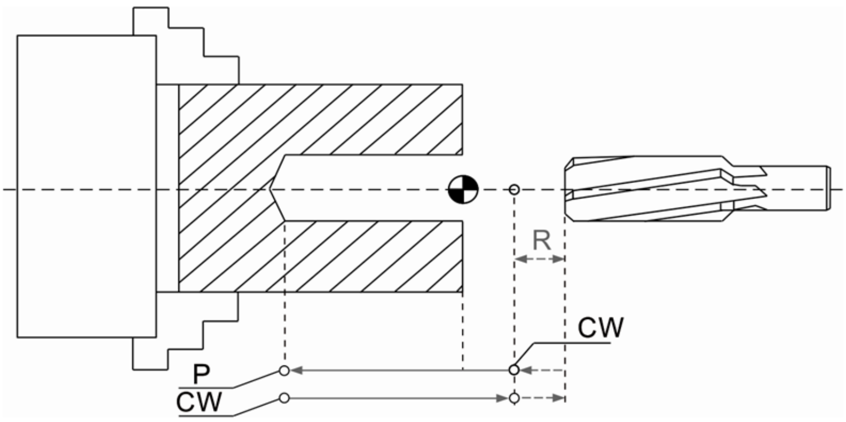
G85: Face boring cycle

Format: G85 X(U)_ Z(W)_ R_ P_ F_ K_;

X(U)_: Boring position; This is the 0 point of X-axis during facing.
Note: If the coordinates of X-axis is not the 0 point of workpiece coordinates, an alarm showing "B6A5: Drilling tapping error" will occur.
Z(W)_: Bottom of the hole
R_: The reference point, which moves toward the opposite direction of the workpiece This is set as an incremental value.
P_: Dwell time at bottom of the hole. The unit is ms. Note: This value has to be integral.
F_: Feed rate
K_: Number of the repeat machining time. The default value is 1.

Description: G85 is usually used for machining with reamers or boring bars in the applications with higher precision requirement for hole diameters. Before starts boring, the tool will go to point R in rapid traverse. Then, the tool will cut to point Z with the setting feed rate F and retract to R with the same speed. Finally, the tool returns to the initial point with G00 fast positioning command.

[Example]



X_ Z_ (Fast positioning)
G84 X_ Z_ R_ P_ F_

G90: Absolute coordinate system command

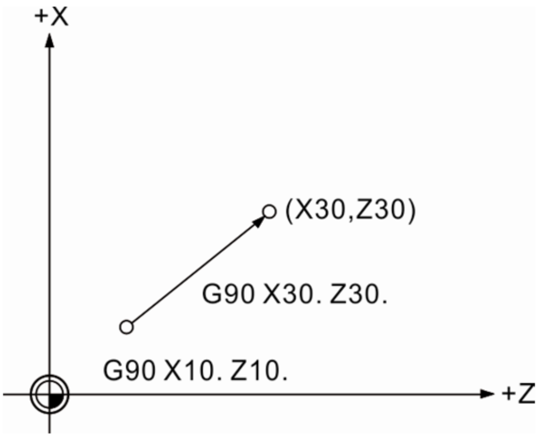
Format: G90 X_ Y_ Z_

Description: Command G90 is a status command of continuous effect. When this command is applied, all axial motion commands and rotation command values will be executed in absolute form. That is, the tool moves relative to a reference point of the origin in workpiece coordinate system. Each axial motion command following G90 moves an actual distance relative to the origin of the workpiece coordinates.

Note: G90 of type A turning system is "Axial turning cycle"

[Example]

When moving from the first point (X10, Y10) to the second point (X30, Y30), the tool moves the incremental distance of X20 and Z20. The corresponding mechanical coordinates is (X30, Y30). See the figure below.



G98
G00 G90 X10.0 Z10.0
X30.0 Z30.0

2

G91: Incremental coordinate system command

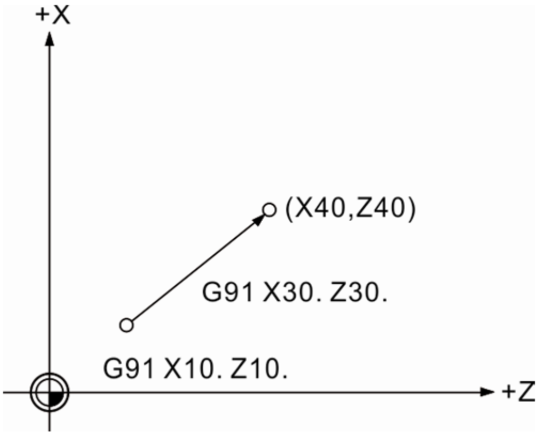
Format: G91 X_ Y_ Z_

Description: When increment command G91 is applied, all axial movements and coordinate angle of a single motion block will be executed in incremental form. The axis will move or rotate from the current position to the specified position according to the set incremental values. The G91 command is a status command that cancels command G90 once it is activated.

Note: This G code is not available in type A turning system.

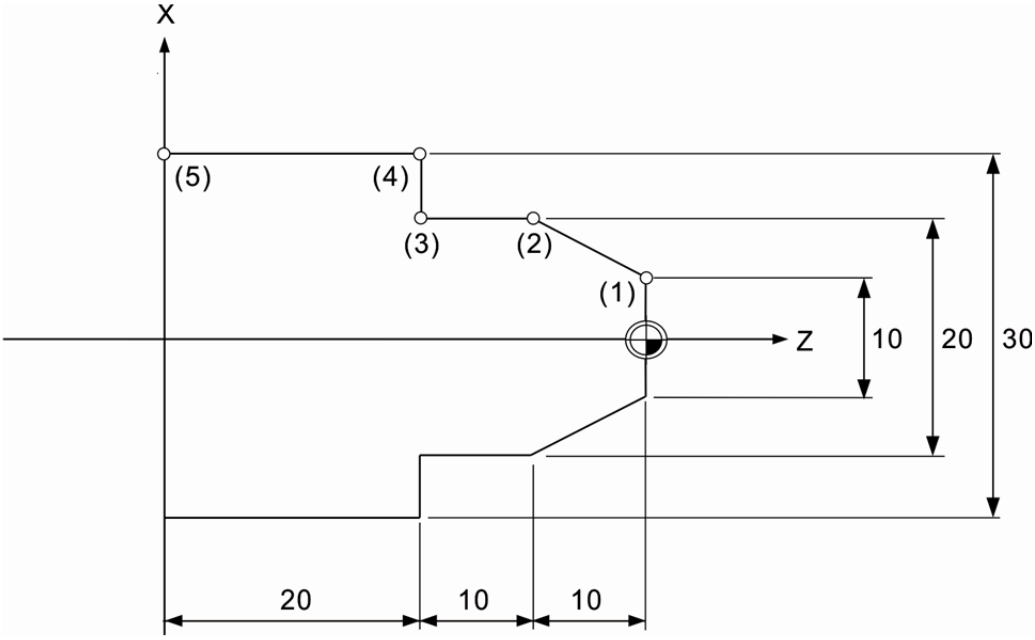
[Example]

When moving from the first point (X10, Z10) to the second point (X20, Z20), the tool moves the incremental distance of X20 and Z20. The corresponding mechanical coordinates is (X30, Y30). See the figure below.



G98
G01 G91 X10.0 Y10.0 F500.0
X30.0 Y30.0

[Example: G90 and G91 are used together]



G98 (Feed by minute)

G00 G90 G54 X0 Y0 (Move to G54 in rapid traverse (X0 , Z0))

G01 X10. F500 (Move to point (1))

X20. Z-10. (Move to point (2))

G91 Z-10.0 (Move to point (3))

X10.0 (Move to point(4))

G90 Z-40.0 (Move to point (5))

G00 X50. Z5. (Move to the safety point in rapid traverse)

2

G50: Coordinate system setting / Max. spindle speed

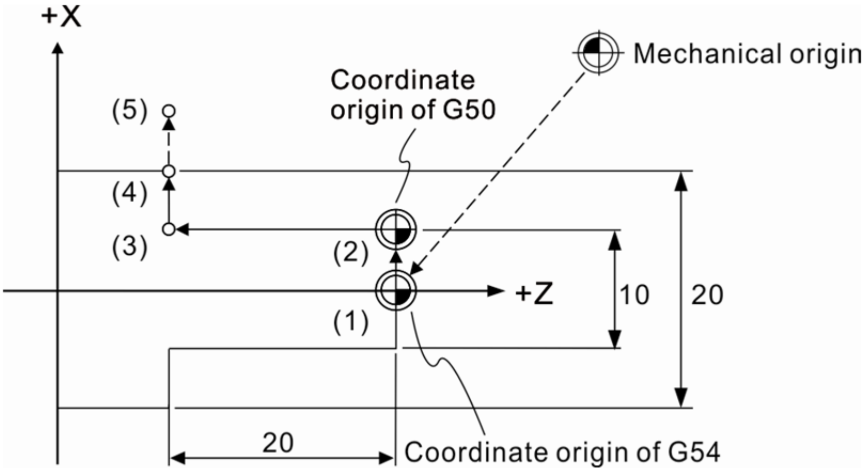
Format: G50 X_ Y_ Z_ (Coordinate system setting)

G50 S_ (Max. Spindle speed)

Description: If G50 X0 Y0 Z0 is set, the current tool position will be regarded as the absolute coordinates 0. This 0 point will be referred by all the absolute command in the program. In G50 X_ Y_ Z_, if X and Y values are set, the absolute coordinates and the current position will be updated according to G50 command. S_ value in G50 sets the maximum spindle speed for maintaining the precision and performance.

Note:
(1) The setting of G50 will be effective unless it is canceled by the program end command (M02/M30).
(2) Press the **RESET** key will cancel the setting of G50.

[Example]



M3 S1500 (Spindle rotates forward at 1500 rpm)

G0 G54 X0. Z0. (Move to G54 coordinates origin (1) in rapid traverse)

G0 X10. (Move to point (2) in rapid traverse)

G50 S1000 (The max. speed limit is 1000 rpm; the speed slows downs to 1000 rpm)

G50 X0. Z0. (The workpiece origin (2) specified by G92)

G1 W-20. F0.25 (Move to point (3))

U10. (Move to point (4))

G0 X30. (Move to safety point (5) in rapid traverse)

M5 (Spindle stop)

M30 (Program end; the absolute coordinates is changed from G92 to G54.)

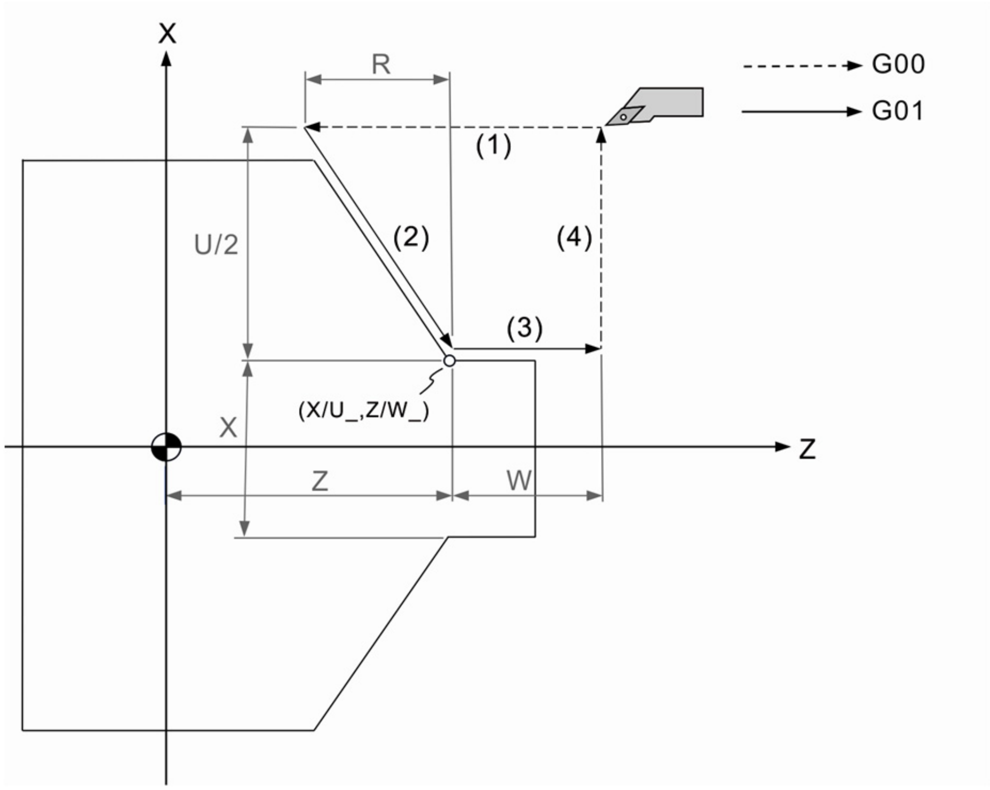
G94: End face turning cycle

Format: G94 X/U_ Z/W_ R_ F_;

X/U_: Coordinates of cutting end point in X-axis direction/ Incremental distance of X-axis (set as diameter/radius)
Z/W_: Coordinates of cutting end point in Z-axis direction/ Incremental distance of Z-axis (set as diameter/radius)
R_: Taper amount (as radius value with positive/negative sign). In straight cutting, R__ will be ignored.
F_: Feed rate

Description: When G94 is applied, the system will perform a complete straight cutting (tapering) cycle of end facing.

[Contour machining example]

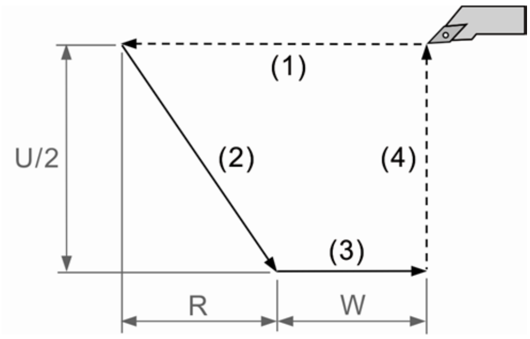
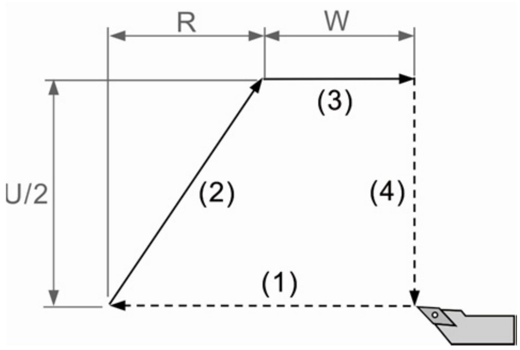
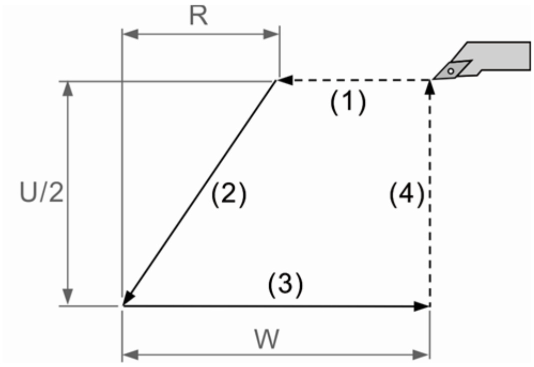
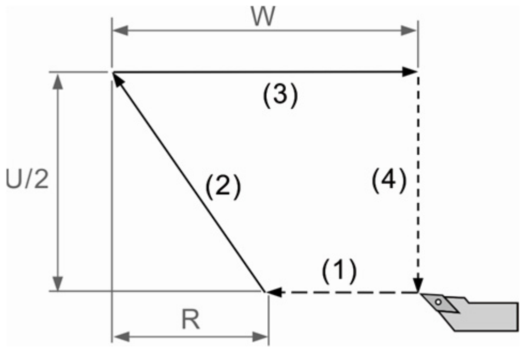


G94 motion analysis:

(1) The tool moves from the start point and go to [the specified coordinates of Z-axis + R the tapering amount] in rapid traverse.
(2) Perform straight cuts all the way to the specified coordinates of Z-axis and X-axis.
(3) Perform straight cuts to the specified start point of Z-axis.
(4) Return to the start point in rapid traverse.

2

The directions corresponding to symbol U, W, and R for turning:

<table>
<tr><th>Outer diameter machining</th><th>Inter diameter machining</th></tr>
<tr><td>1. U < 0, W < 0, R < 0</td><td>3. U > 0, W < 0, R < 0 at |R| ≤ |W|</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>2. U < 0, W < 0, R > 0 at |R| ≤ |W|</td><td>4. U > 0, W < 0, R > 0</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

(1) ~ (4) in the figure represent the tool movement sequence.

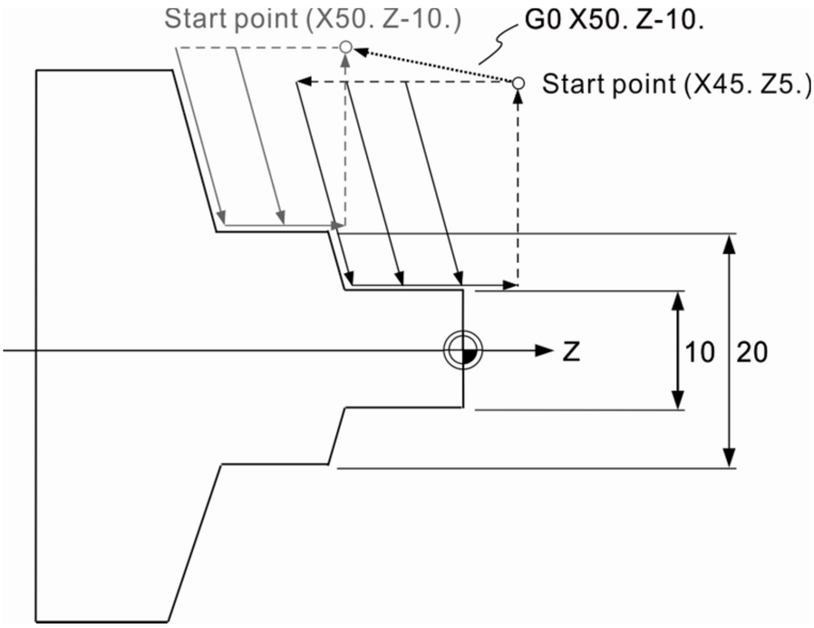
Note:
G94 end face turning cycle is a continuous effect command. To cancel G94 command, you can apply G90, G92, G80 (Cycle cancel) or commands (G00, G01, G02, and G03) in Group 1.

G80: Cycle cancel

Format: G80;

Description: This command is used for canceling G90, G92, and G94 cycles.

[Example]




```
G98
G0 X45. Z5.    (Start point of the cycle)
G94 X10. Z0. R-5. F500
Z-5.
Z-10.
G0 X50. Z-10.    (Start point of the cycle- G0 can be used to cancel the canned
                  cycles G90, G92, and G94)
G94 X20. Z-15. R-5. F500
Z-20.
G80    (Cancel the canned cycle)
M5
M30
```

G83: Face peck drilling cycle

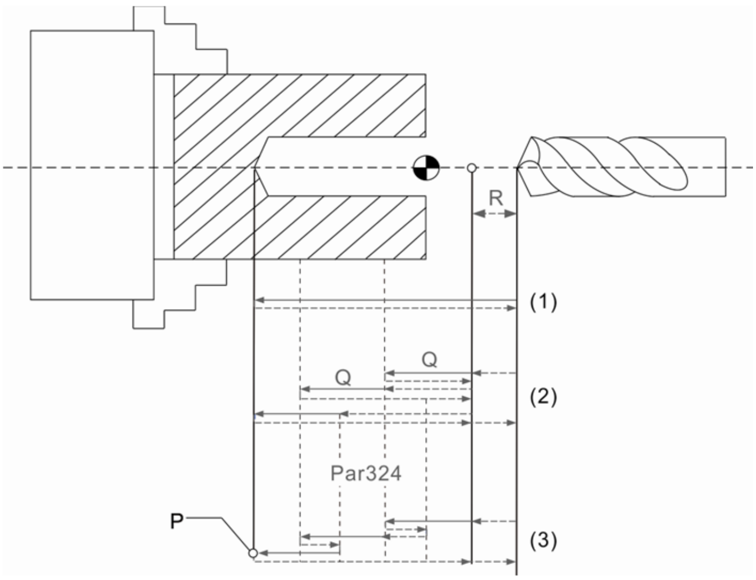
Format: G83 X(U)_ Z(W)_ R_ Q_ P_ F_ K_;

- X(U)_: Drilling position; It is the 0 point of X-axis during face drilling;
  Note: If the coordinates of X-axis is not the 0 point of workpiece coordinate system, an alarm showing “B6A5 Drilling tapping error” will occur.
- Z(W)_: Bottom of the hole
- R_: The reference point, which moves toward the opposite direction of the workpiece This is set as an incremental value.
- Q_: Feeding distance per time (per peck)
- P_: Dwell time at bottom of the hole. The unit is ms. Note: This value has to be integral.
- F_: Feed rate
- K_: Number of repeat machining time. The default value is 1.

Description: Each time the tool drills the distance (depth) Q, it retracts the distance set by parameter 324 in rapid traverse or retracts to point R according to the setting of P326. This cycle will carry on and stops when the tool drills to Z.

Cycle parameter 326:
If bit 2 ~ 3 are set to 0, the tool does not perform peck drilling; it drills directly to the bottom.
If bit 2 ~ 3 are set to 1, the tool performs deep peck drilling, which feeding amount is Q and retracts to point R.
If bit 2 ~ 3 are set to 2, the tool performs general peck drilling, which feeding amount is Q and the escape amount is set by parameter 324.

[Example]



(1) Parameter 326:
bit2~bit3 = 0
X_ Z_ (Go to the drilling position)
G83 X_ Z_ F_

(2) Par326: bit2~bit3=1
X_ Z_
G83 X_ Z_ R_ Q_ F_

(3) Par326: bit2~bit3=2
X_ Z_
G83 X_ Z_ R_ Q_ P_ F_

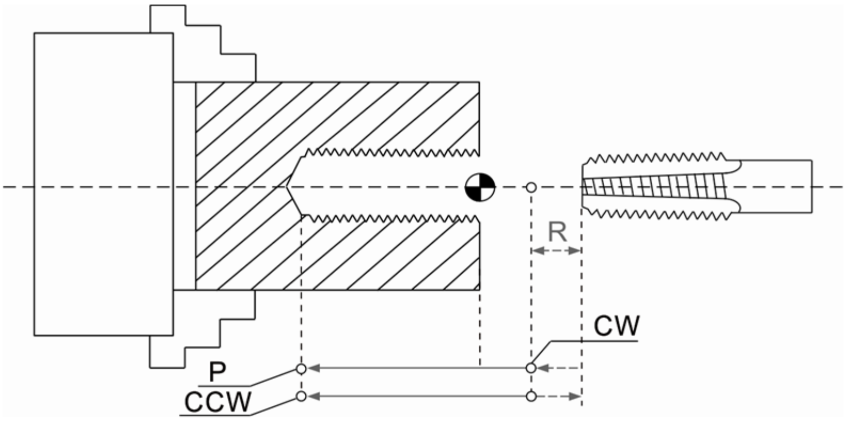
G84: Face tapping cycle

Format: G84 X(U)_ Z(W)_ R_ P_ F_ K_ ;

X(U)_: Tapping point; This is the 0 point of X-axis during face tapping.
Note: If the coordinates of X-axis is not the 0 point of workpiece coordinates, an alarm showing “B6A5: Drilling tapping error” will occur.
Z(W)_: Bottom of the hole
R_: The reference point, which moves toward the opposite direction of the workpiece This is set as an incremental value.
P_: Dwell time at bottom of the hole. The unit is ms. Note: This value has to be integral.
F_: Feed rate
K_: Number of the repeat machining time. The default value is 1.

Description: The spindle will rotate once to look for the Z pulse and execute positioning once Z pulse is found. Then, Z-axis will move to reference point R in rapid traverse. Next, the spindle rotates and starts tapping to the specified position (bottom) and then retracts to R in reverse direction and returns to the initial point in rapid traverse.

[Example]



X_ Z_ (Fast positioning)
G84 X_ Z_ R_ P_ F_

2

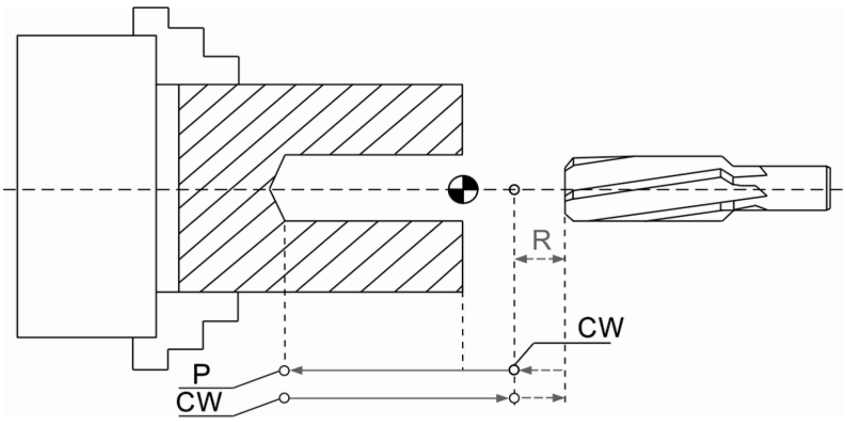
G85: Face boring cycle

Format: G85 X(U)_ Z(W)_ R_ P_ F_ K_;

X(U)_: Boring position; This is the 0 point of X-axis during facing.
Note: If the coordinates of X-axis is not the 0 point of workpiece coordinates, an alarm showing "B6A5: Drilling tapping error" will occur.
Z(W)_: Bottom of the hole
R_: The reference point, which moves toward the opposite direction of the workpiece This is set as an incremental value.
P_: Dwell time at bottom of the hole. The unit is ms. Note: This value has to be integral.
F_: Feed rate
K_: Number of the repeat machining time. The default value is 1.

Description: G85 is usually used for machining with reamers or boring bars in the applications with higher precision requirement for hole diameters. Before starts boring, the tool will go to point R in rapid traverse. Then, the tool will cut to point Z with the setting feed rate F and retract to R with the same speed. Finally, the tool returns to the initial point with G00 fast positioning command.

[Example]



X_ Z_ (Fast positioning)
G84 X_ Z_ R_ P_ F_

M Code Description 3

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

3.1 M code description ········ 3-2

3

3.1 M code description

The M code format is a capital letter M suffixed with three digits. . Some system-defined M codes can be executed for program control without any MLC coding. The table below is the commonly used M codes. The functions of an M code are defined in the MLC except those pre-defined by the system.

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

M00: Program stop (non-optional)

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

M01: Program stop (optional)

Format：M01

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

M02: End of program

Format: M02

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

M30: End of program, return to start

Format: M30

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

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

M98: Subroutine call

Format: M98 P_ L_

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

[Example]

Main program	Subroutine
N0010	O1010
N0020 M98 P1010	N1020
N0030	N1030
N0040	N1040
N0050	N1050
N0060	N1060
N0070	N1070
N0080	N1080
N0090	N1090
N0100	N1100 M99

M99: Return from subroutine

Format: M99

Description: While M98 command can run a subroutine from the main program, M99 returns the cursor to the start of the program and continues execution. Thus, after executing to the block of M99, it returns to the next block called by the subroutine of the main program to continue the execution.

[Example]

Main program	Subroutine
N0010	O1010
N0020 M98 P1010	N1020
N0030	N1030
N0040	N1040
N0050	N1050
N0060	N1060
N0070	N1070
N0080	N1080
N0090	N1090
N0100	N1100 M99

4 Macro and Variables

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

4.1 Variables ···· 4-2
4.1.1 Arguments and local variables ···· 4-3
4.1.2 Systems variables ···· 4-3
4.1.3 Macro interface input and output ···· 4-6
4.2 Variable syntax ···· 4-8
4.3 Operation Commands ···· 4-9
4.4 Control flow ···· 4-10
4.5 Macro call ···· 4-12

4.1 Variables

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

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

4.1.1 Arguments and local variables

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

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

4.1.2 Systems variables

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

G code group message:

Variable	Function	Read	Write
#2000 ~ 2019	G code group	★	
#2020	F code, the F value of NC machining speed.	★	
#2021	H code, the H value of NC tool length compensation number.	★	
#2022	D code, the D value of NC tool diameter compensation number.	★	
#2023	T code, the T value of NC tool number.	★	
#2024	S code, the S value of NC spindle rotation speed.	★	

Various mode messages during program execution (for read only)

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

Position relevant messages:

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

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

Workpiece coordinate messages:

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

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

Other messages

Variable	Function	Read	Write
#2300	Single node **I**, for arc command	★	
#2301	Single node **J**, for arc command	★	
#2302	Single node **K**, for arc command	★	
#2303	Timing after system power on	★	
#2304	Tool number at spindle 1 (dual tool magazine)	★	
#2305	Tool number at spindle 2 (dual tool magazine)	★	
#2500	Tool number fetched from tool magazine 1		★
#2501	Tool number fetched from tool magazine 2		★
#5000 ~ #5013	Log of latest used **M** codes (14 sets)	★	
#5014 ~ #5015	Log of latest used **T** codes (2 sets)	★	
#5016	Search the latest **S** code again	★	
#6000	System macro warnings in range of 1~1000. Message can be edited by the Screen Editor.		★
#6001~	Tool length	★	★
#6501~	Tool radius	★	★
#7001~	Wear in length	★	★
#7501~	Wear in radius	★	★
#8001~	Tool life span	★	★
#8600~	Program timing in unit of ms	★	★

4

4.1.3 Macro interface input and output

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

MLC Bit output, read MLC message status (MLC > NC)

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

MLC Word output, read MLC message status (MLC > NC)

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

MLC Bit input, write MLC message status (NC > MLC)

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

MLC Word input, write MLC message status (NC > MLC)

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

4.2 Variable syntax

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

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

i: the i^{th} variable (for 1<= i <=50)

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

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

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

(3) Symbol precedes a variable

-#<variable number>	Pre-conditions
Z-#20 is the same as Z-10.1	#20=10.1
#20 is the same as G1	#20=1 (When G#20 is used, it writes 1 via the program)

(4) Define with variables

#20 = 10

#20 = #5

#20 = #5 + #2

(5) Conditional expressions

IF[#20==1]

4.3 Operation commands

You can subject variables to a variety of calculations and use the result as a value for another variable, as a combination of several variables or as an alternative to other variables.
#i, #j, and #k can be replaced by constants.

<table>
<tr><th>Command</th><th>Symbol</th><th>Usage</th><th>Definition</th></tr>
<tr><td rowspan="6">Elementary arithmetic</td><td>+</td><td>#i = #j + #k</td><td>Addition</td></tr>
<tr><td>-</td><td>#i = #j - #k</td><td>Subtraction</td></tr>
<tr><td>*</td><td>#i = #j * #k</td><td>Multiplication</td></tr>
<tr><td>/</td><td>#i = #j / #k</td><td>Division</td></tr>
<tr><td>=</td><td>#i = #j</td><td>Substitution</td></tr>
<tr><td>[ ]</td><td>#i = #j * [#p + #q]</td><td>Parenthesis</td></tr>
<tr><td rowspan="15">Function</td><td>SIN</td><td>#i = SIN [#k]</td><td>Sine</td></tr>
<tr><td>ASIN</td><td>#i = ASIN [#k]</td><td>Arcsine</td></tr>
<tr><td>COS</td><td>#i = COS [#k]</td><td>Cosine</td></tr>
<tr><td>ACOS</td><td>#i = ACOS [#k]</td><td>Arccosine</td></tr>
<tr><td>TAN</td><td>#i = TAN [#k]</td><td>Tangent</td></tr>
<tr><td>ATAN</td><td>#i = ATAN [#k]</td><td>Arctangent</td></tr>
<tr><td>ATAN2</td><td>#i = ATAN2 [#m, #n]</td><td>Arctangent; the angle is adjacent side #m and diagonal side #n</td></tr>
<tr><td>ABS</td><td>#i = ABS [#k]</td><td>Absolute value</td></tr>
<tr><td>FIX</td><td>#i = FIX [#k]</td><td>Round down</td></tr>
<tr><td>FUP</td><td>#i = FUP [#k]</td><td>Round up</td></tr>
<tr><td>ROUND</td><td>#i = ROUND[#k]</td><td>Round off</td></tr>
<tr><td>SQRT</td><td>#i = SQRT [#k]</td><td>Squared value</td></tr>
<tr><td>POW</td><td>#i = POW [#m, #n]</td><td>#m to the power of #n</td></tr>
<tr><td>BIT</td><td>#i = BIT [#m, #n]</td><td>The value of the #n bit of a binary #m</td></tr>
<tr><td rowspan="4">Logic operator</td><td>&</td><td>#i = #j & #k</td><td>AND logic</td></tr>
<tr><td>|</td><td>#i = #j | #k</td><td>OR logic</td></tr>
<tr><td>Λ</td><td>#I = #j^#k</td><td>XOR logic</td></tr>
<tr><td>!</td><td>#i = ! #j</td><td>NOT logic</td></tr>
<tr><td rowspan="3">Constants</td><td>PI</td><td>PI = π</td><td>Pi</td></tr>
<tr><td>TRUE</td><td>TRUE = 1</td><td>Return value 1 when the IF statement is true</td></tr>
<tr><td>FALSE</td><td>FALSE=0</td><td>Return value 0 when the IF statement is true</td></tr>
</table>

4

4.4 Control flow

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

```
 WHILE [statement]

  ⦙
ENDW
```

Example:

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

     WHILE[#60>=20  (Enters the second cycle for execution when #60 is greater than or
                     equal to 20.)
           #60=#60-2.
     ENDW            (The second cycle ends when ENDW is encountered.)
           #80=#80+15.
           #50=#50-0.05

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

Branch conditions

When the IF statement is true, the program branches (GOTO) to statement number N for execution. Otherwise, the program executes the next node. See the example description below:

IF [Statement] GOTO N (When the IF statement is true, the program branches (GOTO) to statement number N for execution.)

GOTO N It jumps to N un-conditionally when being used independently.)

N from GOTON has to be in the same program. If not, an alarm will occur.

```
      ∫
N10 #12=#10
#13=#11+2;
IF[#2=1]GOTO200;
#12=#10-#3;
#13=#11-#4;
N200 X#12 Z#13;
#5=#5+2;
              ∫
```

When #2 = 1, branch to N200.

Note:

When searching the sequence number in the branch, the system will search from the start of the program. If the target number is not found, an alarm will occur. If more than one target number exist, the one that is found first in the block will be executed.

Types of conditional expressions:

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

Example:

#100 = 1.234;	(Assign value 1.234 to #100)
#100 = #101;	(Assign value of #100 to #101)
#100 = [[#101+#102]/2.0];	(Assign value of #100 to be the average of #101 and #102)
#100 = #102+2.;	(Assign value of #100 to be the sum of #102 and 2)
#100 = SIN[#102];	
X-#100	(Assign the negative value of #100 to X coordinate)
G1X#100Y#101;	(Assign the value of #100 to X coordinate and #101 to Y coordinate)
G1X[#100];	(Assign the value of #100 to X coordinate)
G1X[#100+#101];	(Assign the sum of values of #100 and #101 to X coordinate)
G2X[#100*SIN[#102]];	(Assign the multiplication of value of #100 by SIN#102 to X coordinate)
G1Z#100F#102S#103;	(Assign the value of #100 to Z coordinate, #102 to F, and #103 to S)

4

4.5 Macro call

(1) A Macro calls G code

Macro function	G code number	Remarks
O9010	0 ~ 1000	When macro call function is not applied, G code is set to 0.
O9011	0 ~ 1000	
O9012	0 ~ 1000	
O9013	0 ~ 1000	
O9014	0 ~ 1000	
O9015	0 ~ 1000	
O9016	0 ~ 1000	
O9017	0 ~ 1000	
O9018	0 ~ 1000	
O9019	0 ~ 1000	

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

(2) A Macro calls M code

Macro function	M code number	Remarks
O9020	0 ~ 1000	When macro call function is not applied, M code is set to 0.
O9021	0 ~ 1000	
O9022	0 ~ 1000	
O9023	0 ~ 1000	
O9024	0 ~ 1000	
O9025	0 ~ 1000	
O9026	0 ~ 1000	
O9027	0 ~ 1000	
O9028	0 ~ 1000	
O9029	0 ~ 1000	

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

(3) A Macro calls T code

Macro function	T code number	Remarks
O9000	0: disabled Else: enabled	When macro call function is not applied, T code is set to 0. T code will be defined as local variable # 20

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

Variable types are listed as follows:

No.	Description	Read	Write
#1~50	Local variables	★	★
#51~250	Global variables	★	★
#1601~1800	Maintain variables (non-volatile)	★	★
#1801~1832	MLC logic output: MLC > NC at macro input point M1024 (32 points in total)	★	
#1833~1848	MLC data output: MLC > NC at macro input point D1024 (16 points in total)	★	
#1864~1895	MLC logic input: NC > MLC at macro output point M2080 (32 points in total)		★
#1896~1911	MLC data input: NC > MLC at macro output point D1336 (16 points in total)		★
#2000~2019	G code group	★	
#2020	F code	★	
#2021	H code	★	
#2022	D code	★	
#2023	T code	★	
#2024	S code	★	
#2100~2102	Mechanical coordinates of X- ~ Z-axis	★	
#2116~2118	Absolute coordinates of X- ~ Z-axis	★	
#2132~2134	Coordinates of the cutting end on X- ~ Z-axis in single block	★	
#2148~2150	The mechanical coordinates of X- ~ Z-axis when G31 skip signal is input	★	
#2164~2166	The absolute coordinates of X- ~ Z-axis when G31 skip signal is input	★	
#2180~2182	Relative coordinates of X- ~ Z-axis	★	
#2196~2198	The absolute coordinates of X- ~ Z-axis after resuming	★	
#2212~2214	The relative coordinates of X- ~ Z-axis after resuming	★	
#2300	Single block I	★	
#2301	Single block J	★	
#2302	Single block K	★	
#2303	Spindle tool number	★	
#2304	Timing after system power on	★	
#3000	Offset amount of X-axis	★	
#3001~	X-axis workpiece coordinates of G54~G59	★	
#3128	Offset amount of Y-axis	★	
#3129~	Y-axis workpiece coordinates of G54~G59	★	
#3256	Offset amount of Z-axis	★	

No.	Description	Read	Write
#3257~	Z-axis workpiece coordinates of G54~G59	★	
#5000~#5013	Latest used M codes (14 sets)	★	
#5014~#5015	Latest used T codes (2 sets)	★	
#5016	Search the latest S code again	★	
#6000	System macro alarm		★
#6001~	Tool length	★	★
#6501~	Tool radius	★	★
#7001~	Wear in length	★	★
#7501~	Wear in radius	★	★
#8001~	Tool life span	★	★

Revision History

Release date	Version	Chapter	Revision Contents
September, 2016	V1.0	N/A	First Edition

For other relevant information about NC series command guidelines, refer to the manual below:

(1) NC Series user manual for operation and maintenance

(2) NC Series MLC Application Manual

(This page is intentionally left blank.)