Smarter. Greener. Together.

Industrial Automation Headquarters

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

Asia

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

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

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

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

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

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

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

Americas

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

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

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

EMEA

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

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

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

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

Iberia: Delta Electronics Solutions (Spain) S.L.U
Ctra. De Villaverde a Vallecas, 265 1º Dcha Ed.
Hormigueras – P.I. de Vallecas 28031 Madrid
TEL: +34(0)91 223 74 20
C/Llull, 321-329 (Edifici CINC) | 22@Barcrelona, 08019 Barcelona
Mail: Sales.IA.Iberia@deltaww.com
TEL: +34 93 303 00 60

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

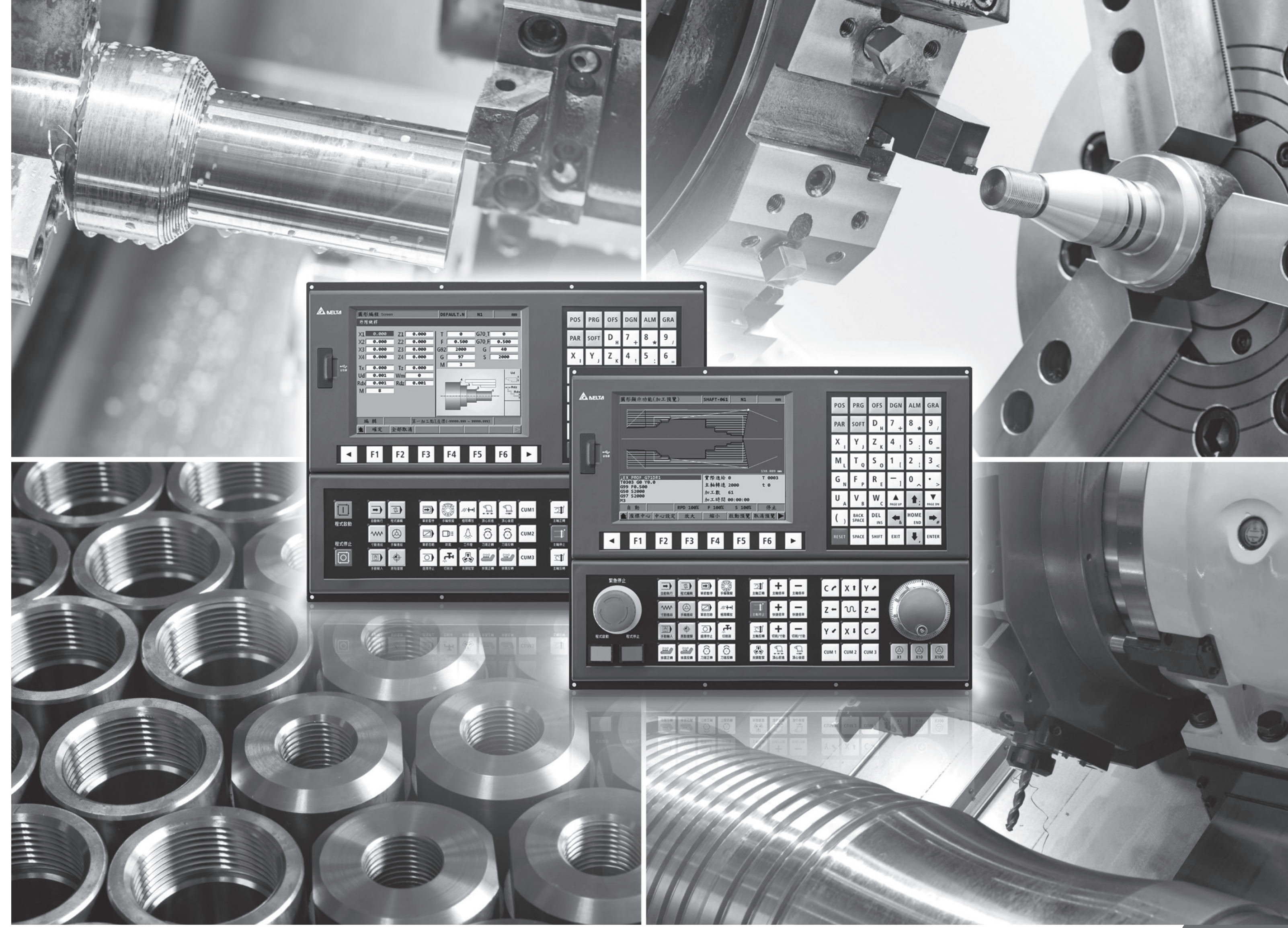
Russia: Delta Energy System LLC
Vereyskaya Plaza II, office 112 Vereyskaya str.
17 121357 Moscow Russia
Mail: Sales.IA.RU@deltaww.com
TEL: +7 495 644 3240

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

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

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

Delta CNC Lathe Machine Solution G Command Guidelines



Delta CNC Lathe Machine Solution G Command Guidelines

www.deltaww.com

Smarter. Greener. Together.

Preface

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

This manual includes:

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

NC controller product features:

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

How to use this manual:

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

DELTA technical services:

Please consult your DELTA equipment distributor or DELTA Customer Service Center if you encounter any problems.

Safety Precautions

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

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

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

DANGER	**Danger. May cause severe or fatal injuries to personnel if the instructions are not followed.**
WARNING	**Warning. May cause moderate injury to personnel, or lead to severe damage or even malfunction of the product if the instructions are not followed.**
STOP	**Absolutely prohibited activities. May cause serious damage or even malfunction of the product it the instructions are not followed.**

Installation

DANGER

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

Wiring

DANGER

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

Operation

WARNING

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

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

Maintenance and Inspection

- Do no disassemble the panel or touch the internal parts of the controller when power is on, or it may cause electric shock.
- Do not touch the ground terminal within 10 minutes after turning off the power, or the residual voltage may cause electric shock.
- Turn off the power before replacing the backup battery, and recheck the system settings afterwards.
- Do not obstruct the ventilation holes when operating the controller, or poor heat dissipation may cause malfunction.

Wiring

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

Wiring of Communication Circuit

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

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

(This page is intentionally left blank.)

Table of Contents

1 G-code List

2 G-code Description

3 M-code Description

4 Macro and Variable

(This page is intentionally left blank.)

1 G-code list

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

1

1.1 G-code list for lathe system

This lathe system G-codes can be categorized into three types, which are A, B, and C. You can use parameter P306 to switch the setting depending on your preference. Set 0 for type A, 1 for type B, or 2 for type C. G-codes in this manual are written based on the type A G-codes.

Type			Group	Function description
A	B	C		
G00	G00	G00	01	Rapid positioning
G01	G01	G01	01	Linear interpolation
G02	G02	G02	01	Clockwise (CW) circular interpolation
G03	G03	G03	01	Counterclockwise (CCW) circular interpolation
G04	G04	G04	00	Dwell time
G09	G09	G09	00	Exact stop
G10	G10	G10	00	Data setting
G11	G11	G11	00	Data cancellation
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	Return through reference point
G29	G29	G29	00	Return from reference point
G30	G30	G30	00	Return to the 2nd, 3rd, or 4th reference point
G31	G31	G31	00	Skip command
G32	G33	G33	01	Thread cutting
G34	G34	G34	01	Variable lead threading
G40	G40	G40	07	Cancel tool nose radius compensation
G41	G41	G41	07	Tool radius compensation left
G42	G42	G42	07	Tool radius compensation right
G50	G92	G92	00	Coordinate system setting / maximum spindle speed
G52	G52	G52	00	Local coordinate system setting
G53	G53	G53	00	Machine coordinate system setting
G54	G54	G54	12	1st workpiece coordinate system selection
G55	G55	G55	12	2nd workpiece coordinate system selection
G56	G56	G56	12	3rd workpiece coordinate system selection
G57	G57	G57	12	4th workpiece coordinate system selection
G58	G58	G58	12	5th workpiece coordinate system selection

Type			Group	Function description
A	B	C		
G59	G59	G59	12	6th workpiece coordinate system selection
G61	G61	G61	13	Exact stop mode (one-shot)
G64	G64	G64	13	Cutting mode
G65	G65	G65	00	One-shot macro call
G66	G66	G66	14	Continuous effect macro call
G67	G67	G67	14	Cancel continuous effect macro call
G70	G70	G72	09	Multiple type finish turning cycle
G71	G71	G73	09	Multiple type rough turning cycle
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	Face turning cycle
G80	G80	G80	09	Cancel cycle
G83	G83	G83	09	Face drilling cycle
G84	G84	G84	09	Face tapping cycle
G85	G85	G85	09	Face boring cycle
G87	G87	G87	09	Side drilling cycle
G88	G88	G88	09	Side tapping cycle
G89	G89	G89	09	Side boring cycle
--	G90	G90	03	Absolute coordinates
--	G91	G91	03	Incremental coordinates
G98	G94	G94	05	Feed per minute (mm/min)
G99	G95	G95	05	Feed per revolution (mm/rev)
G96	G96	G96	17	Constant speed surface control (m/min)
G97	G97	G97	17	Constant speed surface control cancellation (rev/min)

(This page is intentionally left blank.)

G-code Description 2

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

2.1 G-codes for lathe system

Absolute / increment command

Description: in the lathe system, you can assign absolute and incremental values for the movements. If you assign absolute values in the command, the coordinate you specify is based on the workpiece coordinate system origin. If you assign incremental values in the command, the movement is the current position plus the increments you specify. One block can have both absolute and incremental values at the same time.

Absolute command		Increment 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
C__	C axis	H__	C axis

Diameter / radius command

Description: the workpieces for machining are mainly in cylinder shapes in the lathe system, so you can set the X-axis movement with diameter or radius values. When assigning with diameter, the actual moving amount is 50% of the command value; when using radius, the actual movement is the exact command value.

Use P306 to switch between diameter and radius commands:

1. Set 0 to assign with diameter (default).
2. Set 1 to assign with radius.

2

G00: Rapid positioning

Format: G00 X/U_Y/V_Z/W_ (applicable to single-axis, double-axis, and triple-axis synchronous motion controls)

X/U_Y/V_Z/W_: end coordinates.

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

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

Assume that the maximum speed of the X, Y, and Z axes (P316) is 15 m/min:

1. When the rapid feed rate is 100%, the axes operate at the maximum speed of 15 m/min.
2. When the rapid feed rate is 50%, the axes operate at a speed of 7.5 m/min.
3. When the rapid feed rate is 25%, the axes operate at a speed of 3.75 m/min.
4. When the rapid feed rate is 0%, the axes moving speed is determined by the speed set in P315.

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

[Example]

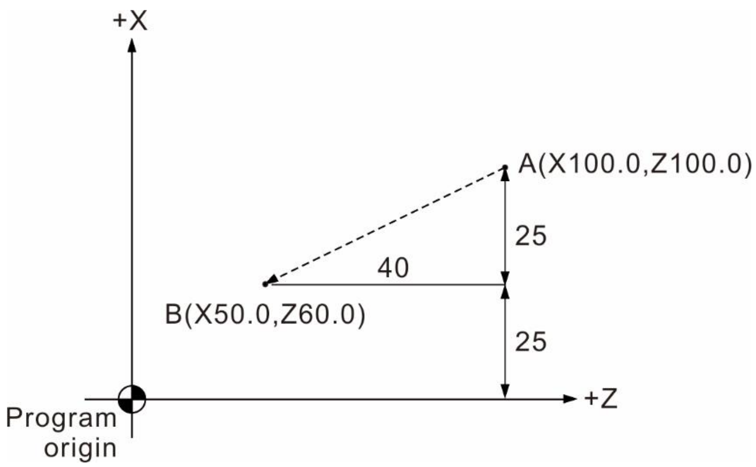
The following diagram illustrates the usage of G00. The tool moves from point A to point B in rapid traverse.

Express with absolute values:

X100. Z100.	(Point A coordinates)
G00 X50.0 Z60.0	(Point B coordinates)

Express with incremental values:

X100. Z100.	(Point A coordinates)
G00 U25.0 W-40.0	(Point B incremental distance)



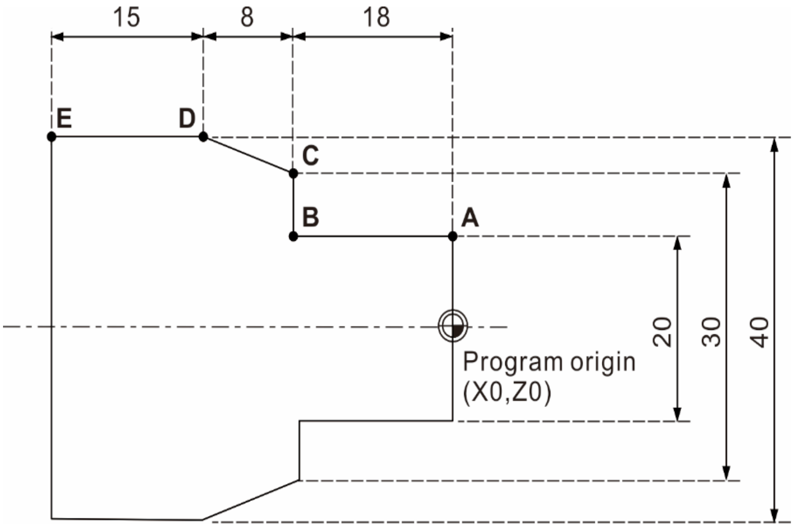
G01: Linear interpolation

Format: G01 X/U_Y/V_Z/W_F_

X/U_Y/V_Z/W_: end coordinates.
F_: cutting feed rate in the unit of mm/min or min/rev.

Description: G01 enables the cutter to make linear interpolation from the current position to the next command position at feed rate F. X-, Y-, and Z-coordinates represent the cutting end point. This command is applicable to single-axis, double-axis, or triple-axis synchronous motion control. The feed rate is set by the F parameter as well as **Rapid %** on the secondary control panel. You can switch the unit with G98 (mm/min) and G99 (min/rev).

[Example]



G98;	(Set mm/min feed mode for the spindle)
G54 X0.0 Z0.0;	(Program start point)
G00 X20.0;	(Move to point A in rapid traverse)
G01 Z-18.0 F500;	(From point A to point B)
X30.0;	(From point B to point C)
X40.0 Z-26.0;	(From point C to point D)
Z-41.0;	(From point D to point E)

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

G02/G03: Circular interpolation

Format: arcs in the X-Y plane:

G17 G02 (G03) X/U_ Y/V_ R_ F_ or

G17 G02 (G03) X/U_ Y/V_ I_ J_ F_

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

Arcs in the Z-X plane:

G18 G02 (G03) Z/W_ X/U_ R_ F_ or

G18 G02 (G03) Z/W_ X/U_ K_ I_ F_

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

Arcs in the Y-Z plane:

G19 G02 (G03) Y/V_ Z/W_ R_ F_ or

G19 G02 (G03) Y/V_ Z/W_ J_ K_ F_

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

G02: clockwise (CW) circular interpolation

G03: counterclockwise (CCW) circular interpolation

X/U, Y/V, and Z/W: end coordinates expressed with absolute / incremental values

R: arc radius (The format expressed with R is called radius format)

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

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

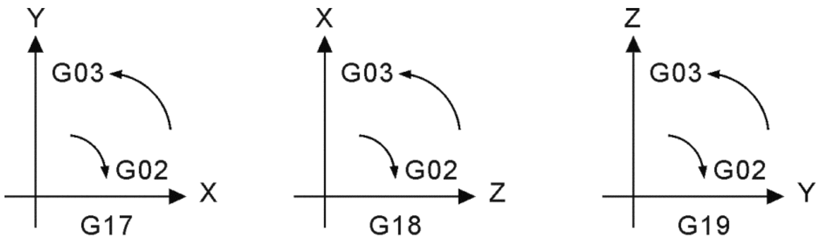
K: distance from the arc start point to the arc center point in Z-axis direction.

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

F: feed cutting rate.

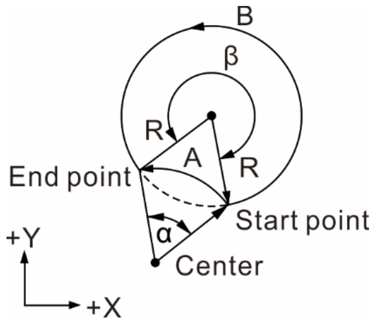
Description: G02 and G03 are circular interpolation commands. Since the workpiece is three-dimensional, the circular interpolation direction (G02 or G03) varies in different planes, as shown in the diagram below.

Definition: based on the right-handed coordinates, G02 is for clockwise direction while G03 is for counterclockwise direction.



Statement expressions with center format and radius format are as follows:

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



When β > 180°, it means R is a negative value, so arc B is generated.

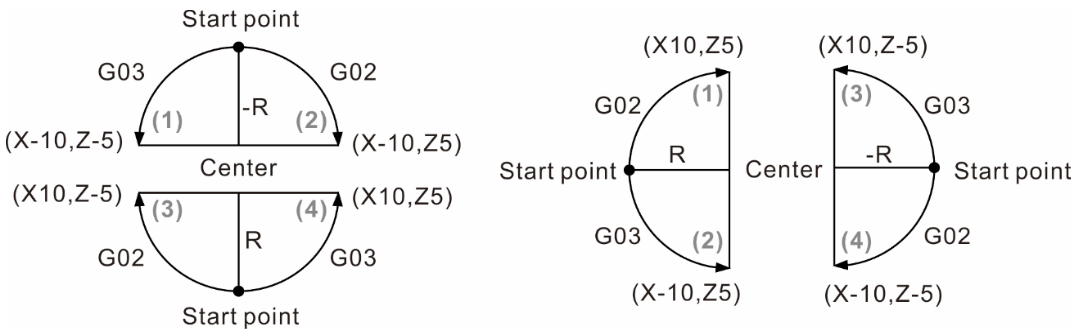
When α ≦ 180°, it means R is a positive value, so arc A is generated.

In the above diagram, assume that R = 50 mm and the end coordinates are (100.0 , 80.0), then:
Central angle > 180° (path B) G03 X100.0 Y80.0 R-50.0 F80
Central angle ≦ 180° (path A) G03 X100.0 Y80.0 R50.0 F80

2. Center format: I, J, and K are the relative distances from the arc start point to the circle center, which are the incremental values from the start point to the center in the X-, Y-, and Z-axis directions. See the figure below for description.

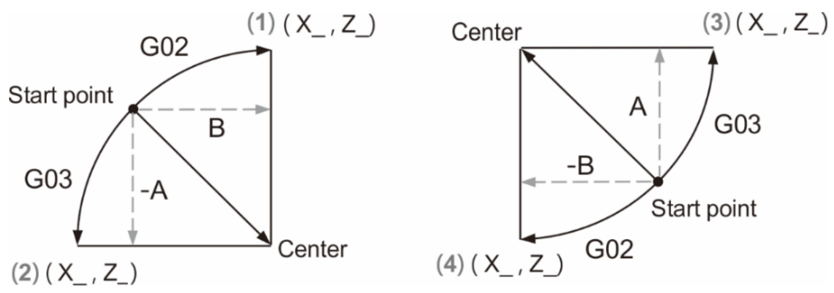
[Example]

Use G18 in the following example.



(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_

2

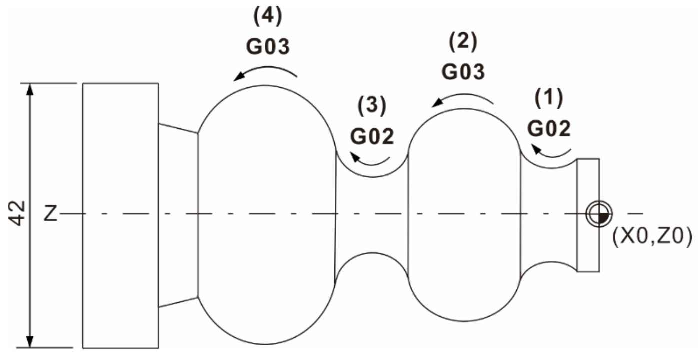


(1) G2 X_ Z_ K+B F_ (3) G3 X_ Z_ I+A F_

(2) G3 X_ Z_ I-A F_ (4) G2 X_ Z_ -B F_

[Example]

Description for using G01, G02, and G03.




```
O0003
G54 X0 Z0 S800 M3
G0 X41. Z0. 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
```

Instructions for G02/G03 circular interpolation:

1. After booting, the default machining plane is G18 (Z-X plane) in the lathe system. Therefore, if the circular interpolation is in the Z-X plane, you can omit G18.
2. If a block has parameters I, J, K, and R at the same time, only parameter R is valid.
3. I0, J0, or K0 can be omitted.
4. If the end X-, Y-, and Z-coordinates are not specified, it means the start and end points are the same, which cutting path will be a full circle. If the command is set in radius format, the tool does not move.
5. The system prompts an alarm message “Arc radius error” when it is unable to form an arc using the end coordinates and the set radius, and the deviation exceeds the value set in machining parameter P323.
6. For a linear interpolation followed by a circular interpolation, you must use G02 or G03 to switch the motion. To switch to linear interpolation, you must use G01.
7. If the circular interpolation command (G02/G03) has no R, I, J, and K parameters specified, the motion path is the same as that of G01.

G04: Dwell time

Format: G04 X_ or
G04 P_

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

Setting range:

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

[Example]

G04 X1.5

G04 P1500

When you use the above two formats, their execution results are the same, the dwell time during program execution is 1.5 seconds.

Note: you can use this command for machining a concave or drilling and this command enables the tool to stop at the bottom and separate the chips from the workpiece.

G05: Parameter group change

Format: G05 P_

2

P_: parameter group number.

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

G05P_: set 1 - 20 to enable the group function; set G05P0 to cancel the parameter group function and restore to the default settings.

PARAMETER(PARAM GROUP) | TEST | N1 | SFT

No.	Parameter Name	GROUP1	GROUP2	GROUP3	GROUP4
309	Nominal arc feed rate	10	15	20	25
312	Depth of cut in G71 and G7	15	30	45	60
314	G1 speed	5000	10000	15000	20000
316	G00 Rapid speed	30000	60000	300000	600000
320	S curve time constant	50	150	450	1500

Range: 10 ~ 50000 (mm/min)

EDIT | Ready

SAVE | DEL GRP | WRT PAR | RED PAR | SRT PAR | AVERAGE

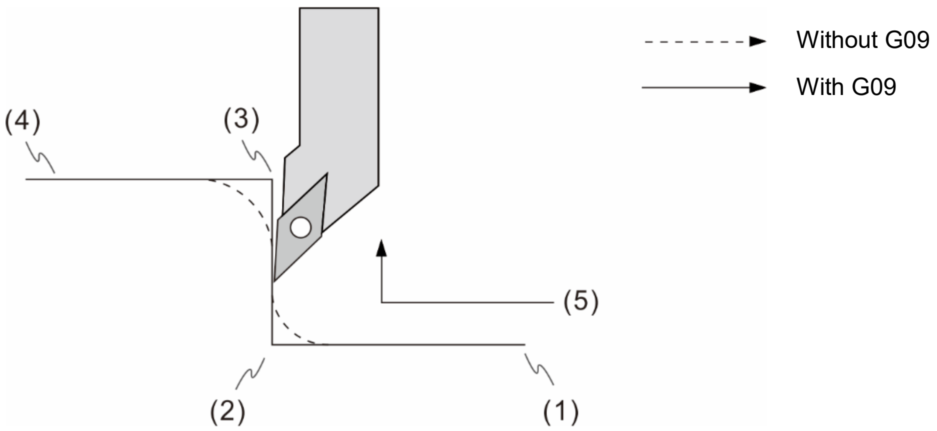
You can individually set each parameter in a group. When you move the cursor to the field, the status bar below shows its setting range. Press **Enter** and you can input the values in the row. Meanwhile, the system checks whether the range is valid; if the input value exceeds the range, the system prompts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

G09: Exact stop

Format: G09 G01 X_ Y_

Description: because the tool cuts at a constant feed rate, the cutting command execution of the next block will start before the current block execution completes. In this case, there will be a small arc generated at the corner between motion blocks. To eliminate this arc, you can use G09 to have the system confirm the tool position each time it executes a motion block. Once the tool is in the right position that is consistent with the command value, the execution for the next block starts. Therefore, there will be a minor discontinuity between blocks when G09 is used, which improves the precision at the cost of speed. This command is valid 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 (Decelerates to stop and then starts executing the next block after the position is confirmed.)

(3) G09 X50.0

(4) Z-50.0

(5) Tool feeding direction

G10/G11: Data setting / cancellation

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

G10 L10 P_X/U_Y/V_Z/W_R_Q_; set the tool length, cutter radius compensation, and tool nose type.

G10 L11 P_X/U_Y/V_Z/W_R_Q_; tool wear and compensation for cutter radius wear.

G10 L20 P_X_Y_Z_; set the extension work coordinate system.

G10 L21 P_X_Y_Z_; set the software limit coordinates.

G10 L30 P_; set the spindle position offset.

G10 L1100 R_S_; set the maximum and current speed.

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

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

G11; cancel the data settings.

Description:

1. The format, G10 L2 P_ X_ Y_ Z_, is for workpiece coordinate system data entry. When you set 0 for P_, it means you are setting the offset coordinates of the workpiece coordinate system; P1 - P6 correspond to G54 - G59 workpiece coordinate systems; and X, Y, and Z specify the position of its coordinate system origin.
2. The format, G10 L10 P_ X/U_ Y/V_ Z/W_ R_ Q_, is for setting the tool length and tool radius compensation data. P_ is the compensation number, X/U_ Y/V_ Z/W_ is the actual tool length compensation data (U, V, and W for increment input), R_ specifies the compensation for the tool radius, and Q_ sets the tool nose type. If you omit L10 when setting the tool length / radius compensation, P_ will be [10000 + compensation number], and the other commands remain unchanged.
3. The format, G10 L11 P_ X/U_ Y/V_ Z/W_ R_ Q_, sets the compensation data for tool wear and tool radius wear. P_ sets the compensation number, X/U_ Y/V_ Z/W_ sets the actual tool wear compensation data for each axis (U, V, and W for increment input), R_ sets the tool radius wear compensation, and Q_ sets the tool nose type. If you omit L11 when using this command, P_ is the compensation number, and the other commands remain unchanged.
4. In G10 L20, you can input P1- P64 for the P value to set the corresponding extension workpiece coordinate systems.
5. G10 L21 P_: P1 sets the first set of the positive software limit;
   P2 sets the first set of the negative software limit;
   P3 sets the second set of the positive software limit;
   P4 sets the second set of the negative software limit.
6. G10 L30 P_ sets the spindle positioning offset. P_ sets the offset angle in the unit of 0.01 degree; and G11 L30 cancels the spindle positioning offset setting (resets to Pr405 settings).

7. In G10 L1100 R_ S_, R_ sets the maximum rotation speed and S_ is the current rotation speed setting. When you select DMCNET for the spindle, you can use this command to control the analog output.
8. The command G10 L3100 P_ triggers the status M-code without stopping the motion; G10 L4100 P_ cancels the status M-code without stopping the motion; and P2080 - P2111 (P_ value) correspond to M2080 - M2111.

Data entry type

L command format	Argument format	Data type description
L2	P_X_Y_Z_	Data entry for the workpiece coordinate system P: 0 is the offset coordinate; 1 - 6 correspond to G54 - G59 work coordinates
L10	P_X_Y_Z_R_Q_	Data entry for the tool length and tool radius compensation P: 1 - 64 correspond to 1 - 64 tool length compensation data R: cutter radius compensation value Q: tool nose type setting
L11	P_ X_Y_Z_R_Q_	Data entry for the tool wear and tool radius wear compensation P: 1 - 64 correspond to 1 - 64 tool wear data R: tool radius wear compensation value Q: tool nose type setting
L20	P_X_Y_Z_	Data entry for the workpiece coordinate system P: 1 - 64 correspond to the 1st - 64th set of workpiece coordinates
L21	P_X_Y_Z_	Data entry for the software limit coordinates P: 1 is the first set of positive software limit; 2 is the first set of negative software limit; 3 is the second set of positive software limit; 4 is the second set of negative software limit.
L30	P_	P_: spindle positioning offset (0.01 degree)
L1100	R_S_	R: maximum rotation speed setting (rpm) S: current rotation speed setting (rpm) When you select DMCNET for the spindle, you can use this command to control the analog output.
L3100	P_	P: 2080 - 2111 represent M2080 - M2111.
L4100	P_	P: 2080 - 2111 represent M2080 - M2111.

2

[Example]

G10 L10 P1 X-50. W20. R2. Q3

or

G10 P10001 X-50. W20. R2. Q3

In the program above, it sets the tool No.1. The compensation for the X-axis tool length is -50.0; the compensation for the Z-axis tool length is the original value plus 20.0; the tool radius is 2 and the tool nose type is 3.

Note:

1. G10 is a one-shot command which functions only within the block where it is specified. Compensation amounts of the offset coordinates and the workpiece coordinate systems both refer to the origin of the machine coordinate system. To cancel the data entry, you can execute G11.
2. When you execute L2 or L20 to change the coordinates, it functions only in the block that includes it. When you use L10 - L13 to change the tool compensation data, you must execute the compensation command (G41/G42) and specify the compensation data number to update the compensation value.

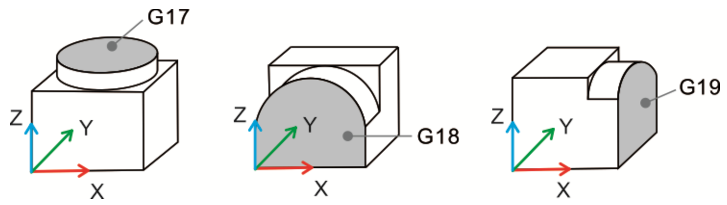
G17/G18/G19: Plane designation

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

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

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

Description: this command is for switching among the planes. If it is a triple-axis synchronous motion, setting this command is not required. G17 - G19 can only be used in the condition that allows linear interpolation, arc interpolation, or tool compensation. The lathe system's default plane after boot up is G18. In this case, you do not need to set G18 additionally when selecting Z-X plane for machining.



G21/G20: Metric / inch input

Format: G21 or G20

G21: metric unit settings

G20: inch unit settings

Description: you can use this command to specify the unit in metric or inch. G21 and G20 are only applicable to linear axes and do not affect the rotation angles of the rotation axes. You must input this command before the program starts running; changing the metric / inch setting is not allowed during program execution. This command changes the numeric units relevant to the system, such as the cutting feed rate (F value), coordinates command values, workpiece coordinate offset, tool compensation amount, and moving distance. G21 and G20 commands are continuously effective; once you have specified the system unit at the beginning of the program, the program refers to this setting and uses metric or inch as the unit. You cannot use both G21 and G20 in the same program.

2

G28: Return through reference point

Format: G28 X_ Y_ Z_ or

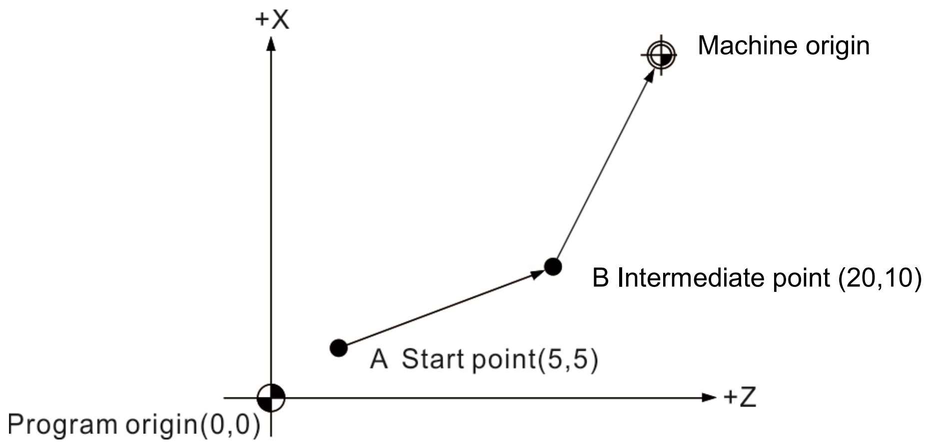
G28 U_ V_ W_

X_ Y_ Z_: intermediate point coordinates.

Description: G28 command can have the tool pass through the intermediate point and return to the machine origin in rapid traverse (G00).

The format X_Y_Z_ refers to the intermediate point coordinates. The unspecified axes will not pass through the intermediate point to return to the origin. If you have set the tool radius compensation (G41 or G42), you need to cancel the setting; otherwise, when the system is executing G28, the tool radius compensation and its compensation distance setting are temporarily canceled when the tool goes to the intermediate point and then returns to the machine origin without compensation. Then, the tool radius compensation function resumes at the next motion block. When G28 is in execution, the tool length compensation function remains effective when the tool reaches the reference point. Next, the tool returns to the machine origin without the tool length compensation. Then, the tool length compensation resumes in the next motion block.

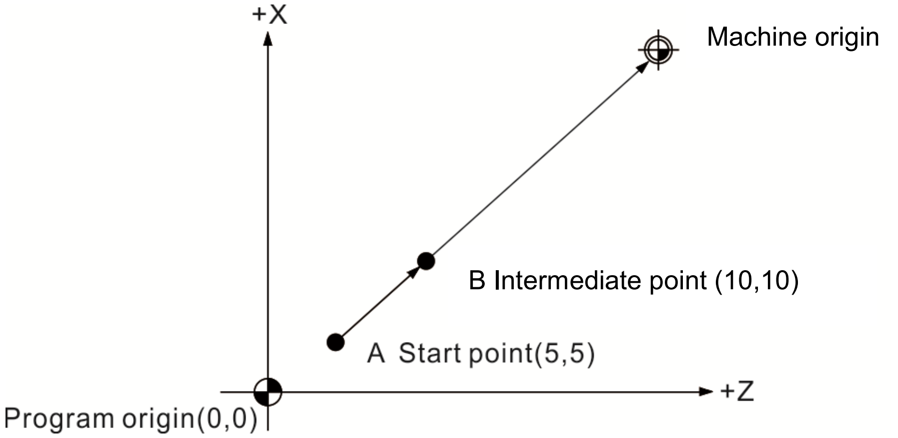
[Example]



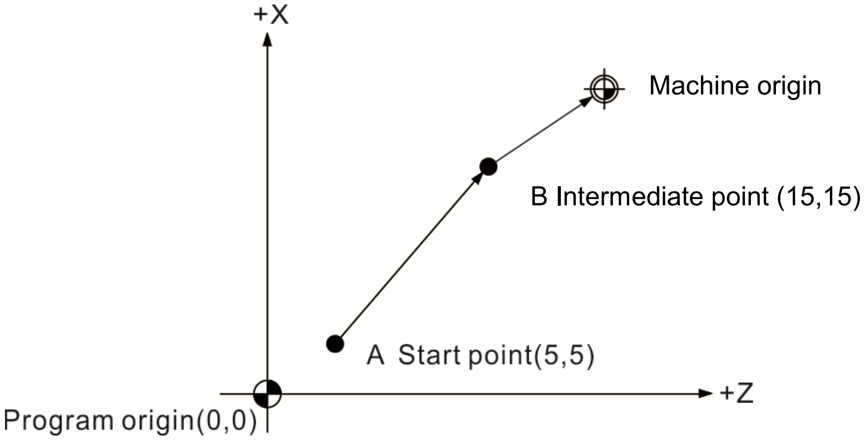
G28 X20. Z10. (Go through the intermediate point B and return to the machine origin.)

[Example]

When G28 is in execution, the increment / absolute status setting will affect the process of returning to the machine origin. See the figure below.



G28 X10.0 Z10.0



G28 U10.0 W10.0

2

G29: Return from reference point

Format: G29 X_ Y_ Z_ or

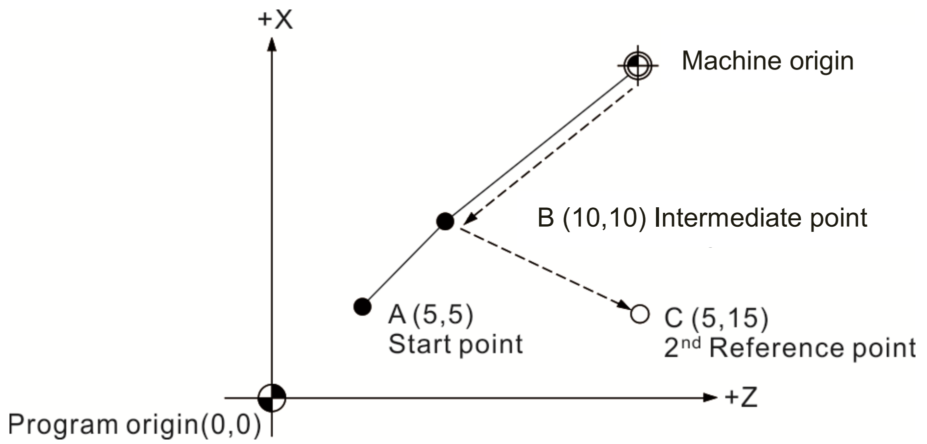
G29 U_ V_ W_

X_ Y_ Z_: end of the motion in the block.

Description: G29 can have the tool move from the machine origin or any point, pass through the intermediate point, and then go to the specified point in the block. X_Y_Z_ represents the motion end coordinates. G29 and G28 must be used together, so the tool moves to the intermediate point designated by G28 and then moves to the position specified in G29 without calculating the actual moving distance from the intermediate point to the machine origin.

If you execute G29 solely without G28 the intermediate point setting, the system will display the alarm message and stop the motion.

[Example]



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

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

Format: G30 P2 X_ Y_ Z_ or

G30 P3 X_ Y_ Z_ or

G30 P4 X_ Y_ Z_

P_: selection of the 2nd, 3rd, and 4th reference points.

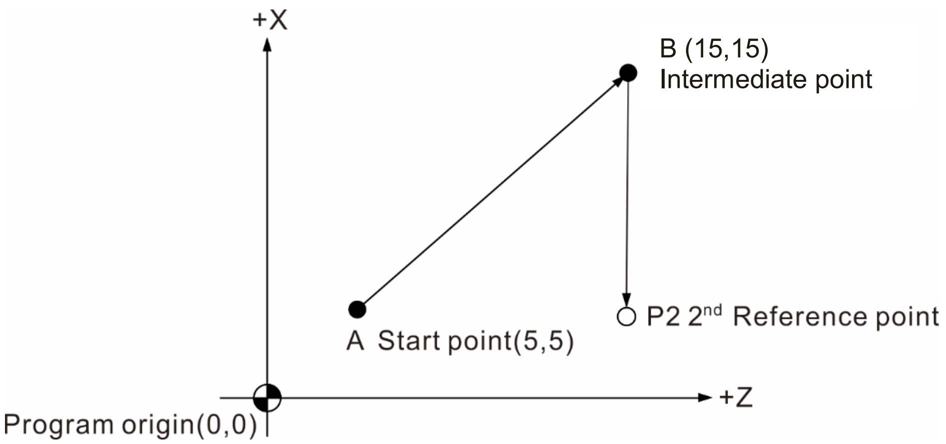
X_ Y_ Z_: the intermediate point coordinates.

Description: P2, P3, and P4 commands correspond to the 2nd, 3rd, and 4th reference points respectively, which you can set with the homing parameters P607, P608, and P609. When you select the 2nd reference point, you can omit P2 in the command format.

X_Y_Z_ represents the intermediate point coordinates. The tool passes through the specified intermediate point and then returns to the 2nd, 3rd, or 4th reference point. To specify the 2nd, 3rd, and 4th reference point coordinates, you need set the homing parameters.

G30 is mostly used for tool changing. When the command is set with absolute values and the motion block G30 Z0.0 is executed, the Z axis returns to the reference point (Z0.0) and then moves to the 2nd reference point to complete the designated motion.

You must cancel the tool compensation setting (using G40) before executing G28 and G30. If you execute G30 or G28, the block including this command will cancel the tool radius compensation and tool length compensation. After homing to the reference point is complete, the tool length compensation and tool radius compensation resume in the next motion block.



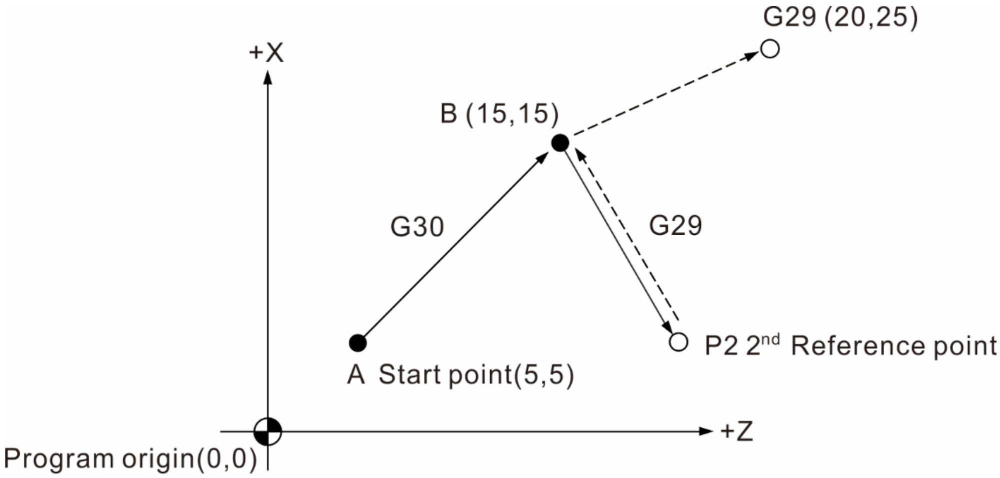
G30 P2 X15. Z15.

Move from point A to point B, and then to P2 the 2nd reference point.

As shown in the figure above, when you execute G30 with absolute values specified, the Z axis first returns to the intermediate point and then goes to P2 to complete the homing procedure.

2

[Example 1]

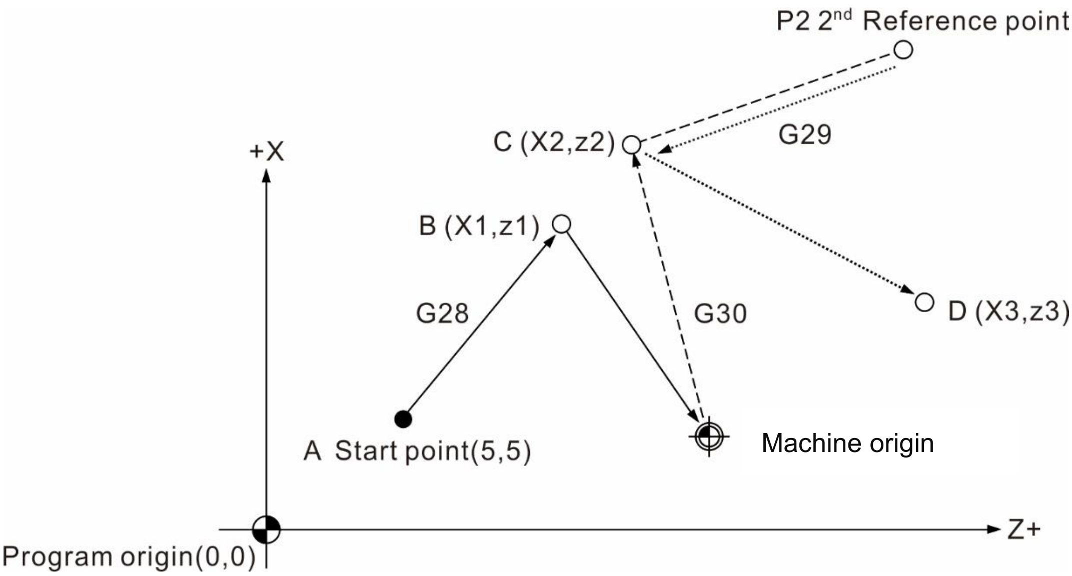


Program example:

G30 P2 X15.0 Z15.0

G29 X20.0 Z25.0

[Example 2]



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 command

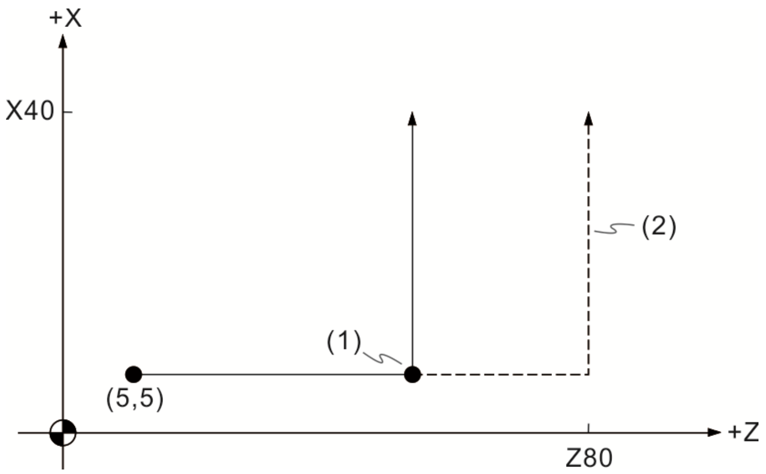
Format: G31 X_ Y_ Z_ F_P_

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

Follow the instructions before using G31 Skip command:

1. You can enable the G31 high speed input 1 or 2 with the Pr46 setting.
2. If you do not assign the P_ value in G31, the system refers to the setting of Pr307 instead.
   G31 selection range: 0 - 3;
   0: no selection;
   1: triggered by HSI 1;
   2: triggered by HSI 2;
   3: triggered by either HSI 1 or HSI 2
3. If you assign the P_ value in G31, the system will not refer to the setting of Pr307.
   The P_ value determines which HSI to trigger. Set P1 to trigger with HSI 1, set P2 to trigger with HSI 2, or set P3 to trigger by either HSI 1 or HSI 2.

[Example1]

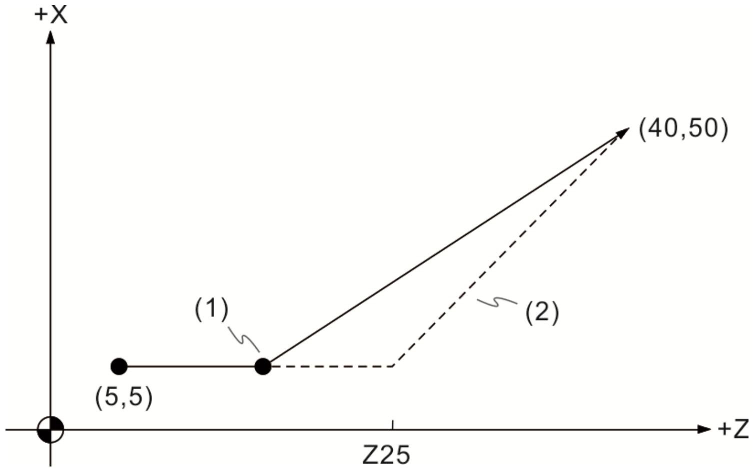


G00 X5. Z5.
G01 G31 Z80.0 F0.5
X40.0

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

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

[Example 2]



G00 X5. Z5

G01 G31 Z25.0 F0.5

X40.0 Z50.0

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

If there is no skip signal input in the process, the actual path is shown as the dotted line in the figure above. If you input a skip signal, the actual path is shown as the solid line; the tool skips the current point as soon as the signal is input and starts executing motions of the next block.

G32: Thread cutting

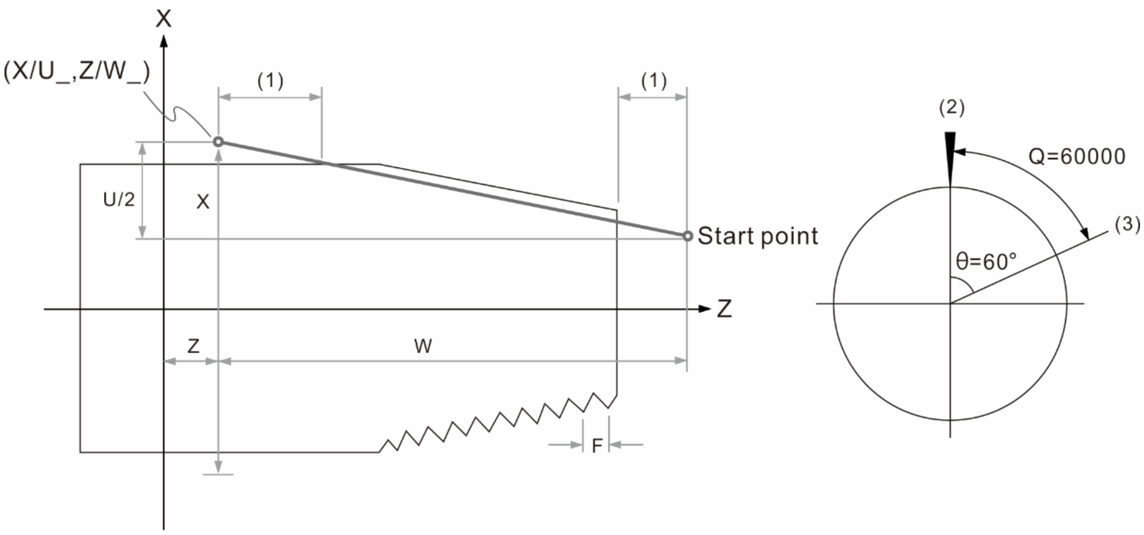
Format: G32 X/U_ Z/W_ F_ Q_

X/U_ Z/W_: end coordinates for threading.

F_: lead; the linear distance of one thread rotation.

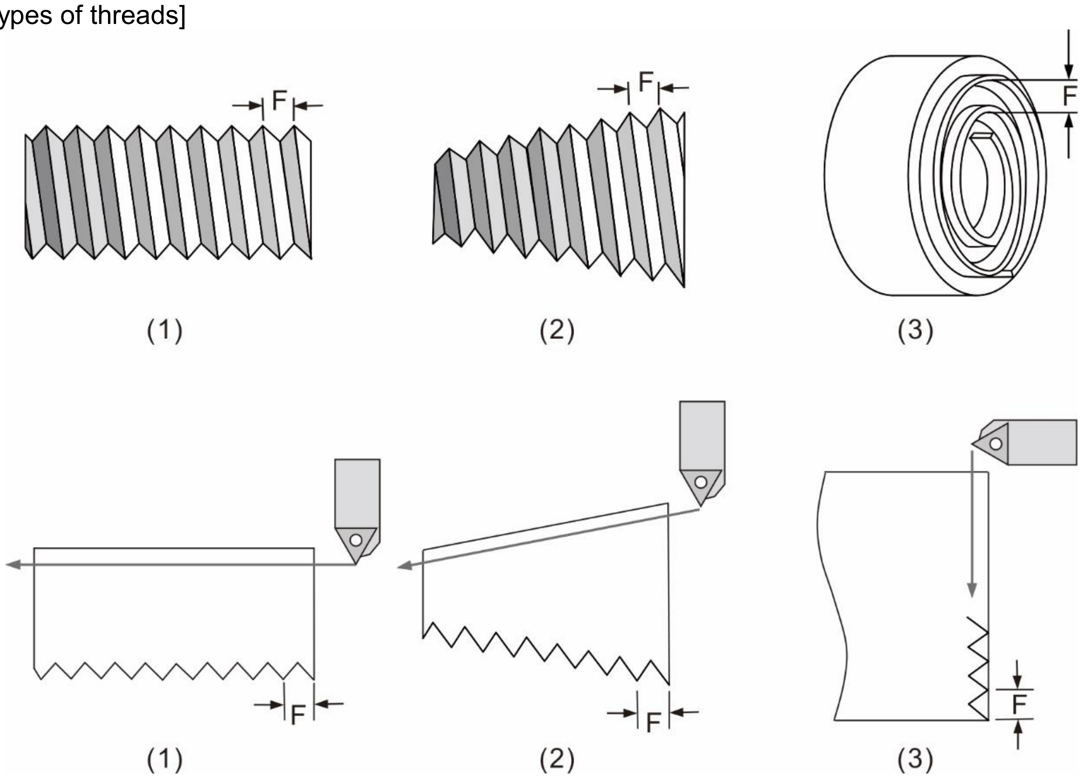
Q_: start angle of the thread in the unit of 0.001. The default is 0.

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



(1) Incorrect leads; (2) Spindle Z-pulse; (3) The point for entering the thread

[Types of threads]



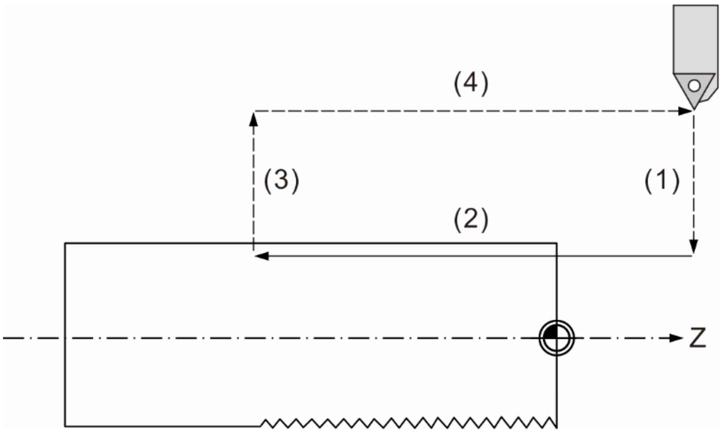
(1) Straight thread; (2) Tapered thread; (3) Scroll thread (face)

Note:

1. You must execute G32 Thread cutting when the spindle speed is fixed.
2. When threading, you cannot manually adjust the spindle feed rate; the spindle speed keeps at 100%.
3. If you press **Feed Hold** during threading, the threading motion does not stop immediately; instead, it stops at the end of the next block that has no threading command.
4. Pressing the **RESET** key can stop the threading operation immediately but causing damage to the thread.
5. If the spindle speed is 3,000 rpm and the pitch (F) is 1.5, the Z axis threading feed rate is 4,500 mm/min. If the alarm B01D Spindle overspeed occurs, it means the feed axis speed exceeds the maximum and you need to lower the spindle speed.
6. The lag of servo system might produce incorrect leads at the start and end points when threading. To avoid affecting the screw thread function, the specified threading length has to be longer than the actual required length.

2

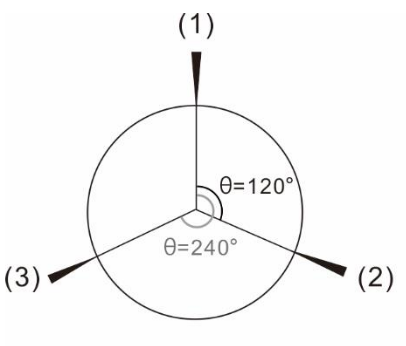
[Example]

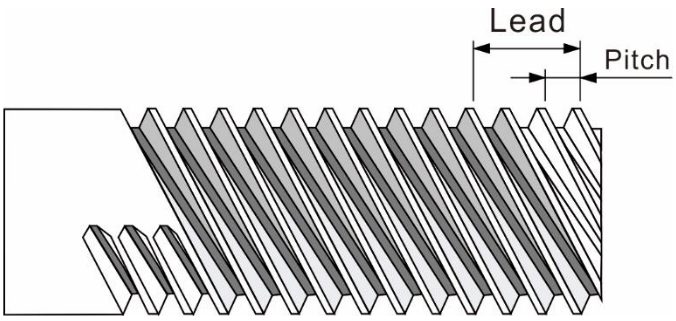


T0202	
M3 S1000	
G0 X40.	
Z15.	
X17.45	(1) Height for the first thread cutting
G32 Z-30. F1.5	(2) Threading
G0 X40.	(3) X axis retraction using G0 command
Z15.	(4) Z axis retraction using G0 command
X17.20	(1) Height for the second thread cutting
G32 Z-30. F1.5	(2)
G0 X40.	(3)
Z15.	(4)
X17.00	(1) Height for the third thread cutting
G32 Z-30. F1.5	(2)
G0 X40.	(3)
Z15.	(4)
X16.85	(1) Height for the forth thread cutting
G32 Z-30. F1.5	(2)
G0 X40.	(3)
Z15.	(4)
X16.8	(1) Height for the fifth thread cutting
G32 Z-30. F1.5	(2)
G0 X40.	(3)
Z15.	(4)
M5	
M30	

[Example] Multi-start threads

L (lead) = n (number of threads) x pitch





Main program

T0202

M3 S1000

G0 X45.

Z10.

G66 P3300 A0   Point to enter the thread (1)

X17.45

X17.20

X17.00

G67

G66 P3300 A120000   Point to enter the thread (2)

X17.45

X17.20

X17.00

G67

G66 P3300 A240000   Point to enter the thread (3)

X17.45

X17.20

X17.00

G67

G0 X45.

Z10.

M30

Subprogram

O3300

G32 Z-30. F3 Q#1 (Substitute A_ into Q#1, which is the thread offset angle)

G0 X45.

Z10.

M99

G34: Variable lead threading

Format: G34 X/U_ Z/W_ F_K±_

X/U_ Z/W_: end coordinates for threading

K_: the increments per threading; a negative value represents the amount of threads to be reduced.

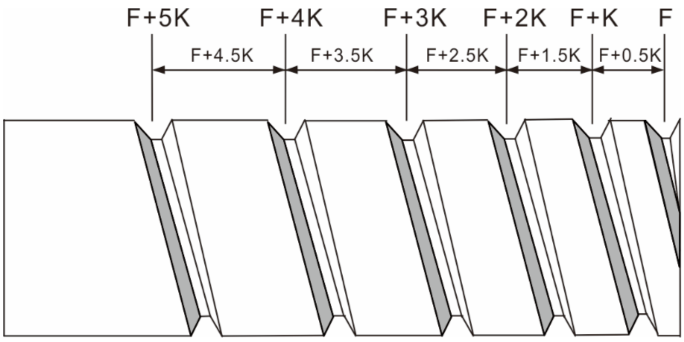
F_: the pitch between each thread.

Q_: start angle of the thread in the unit of 0.001. The default is 0.

Description: when G34 is applied, the system refers to the increment and decrement of lead specified in G34 to perform variable lead threading.

Lead: linear distance of one thread rotation.

Pitch: distance between two adjacent threads.



Lead speed

Lead distance: $d = Vo\ t + \frac{1}{2}at^2$

Vo: initial pitch of screw thread (F)

t: number of screw threads

a: increment in pitch per thread rotation (K)

[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):

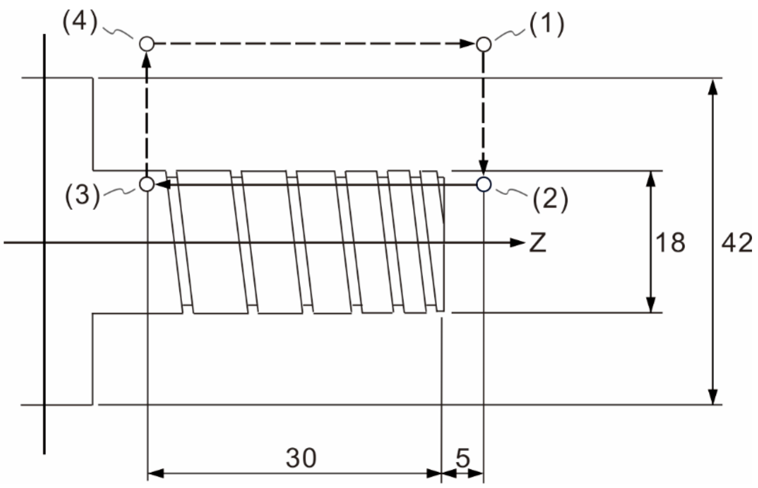
$(2F + 2K) - (F + 0.5K) = F + 1.5K$

Note:

1. You must execute G34 thread cutting when the spindle speed is fixed.
2. When threading, you cannot manually adjust the spindle feed rate; the spindle speed keeps at 100%.
3. If you press **Feed Hold** during threading, the threading motion does not stop immediately; instead, it stops at the end of the next block that has no threading command.
4. Pressing the **RESET** key can stop the threading operation immediately but causing damage to the thread.
5. If the spindle speed is 3,000 rpm and the pitch (F) is 1.5, the Z axis threading feed rate is 4,500 mm/min. If the alarm B01D Spindle overspeed occurs, it means the feed axis speed exceeds the maximum and you need to lower the spindle speed.

6. The lag of servo system might produce incorrect leads at the start and end points when threading. To avoid affecting the screw thread function, the specified threading length has to be longer than the actual required length.

[Example]



Main program

T0101 // Select tool No.1

M03 S600 // Spindle rotates forward at 600 rpm

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

G66 P0034 L1 // Macro call: execute subprogram 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 the macro call command

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

M5 // Spindle stop

M30 // Program end

Subprogram

O0034

G34 Z-30. K0.5 F1 // (3) Execute G34 and thread to Z-30 with the increment pitch of 0.5 mm per spindle revolution and default pitch of 1 mm.

X50. // (4) X axis retraction

Z5. // Return to thread start point

M99 // Return to main program

2

G40: Cancel tool nose radius compensation

Format: G40

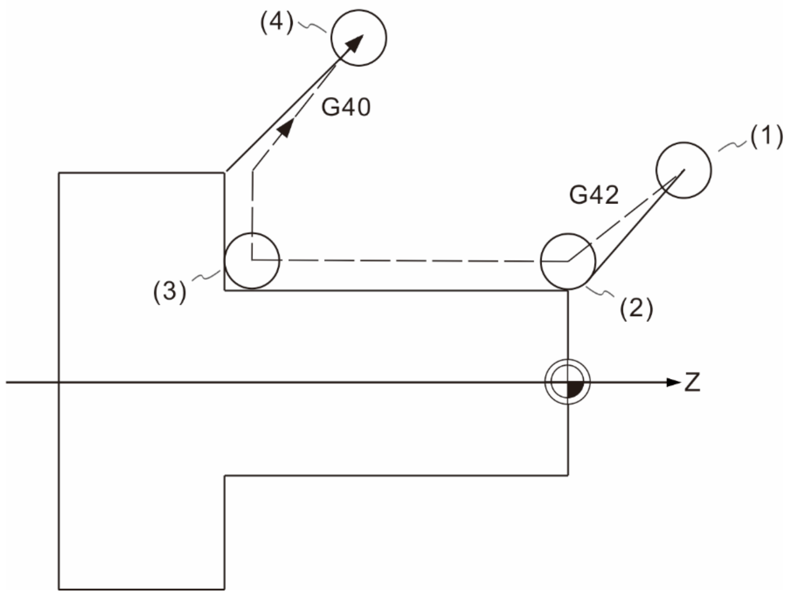
or

G40 X_ Z_

Description: if the tool path does not need tool nose radius compensation, you can use G40 command to cancel the compensation path. The compensation command is a status command, so it continues to function unless you cancel it.

When executing the homing command, the tool radius compensation function is temporarily canceled while the tool is returning to the reference point. Then, the compensation function resumes in the next motion block. Please note that the tool radius compensation cancellation is not applicable to an arc motion path.

[Example]



(1)	G0 X40. Z20.	(Start point)
(2)	G41 G1 X20. Z0. F0.25	(Enable tool compensation)
(3)	Z-30.	
(4)	X40.	
(5)	G40 G0 X60. Z-20.	(End point of tool compensation)

G41/G42: Tool nose radius compensation left / right

Format: G00 G41 P_ or

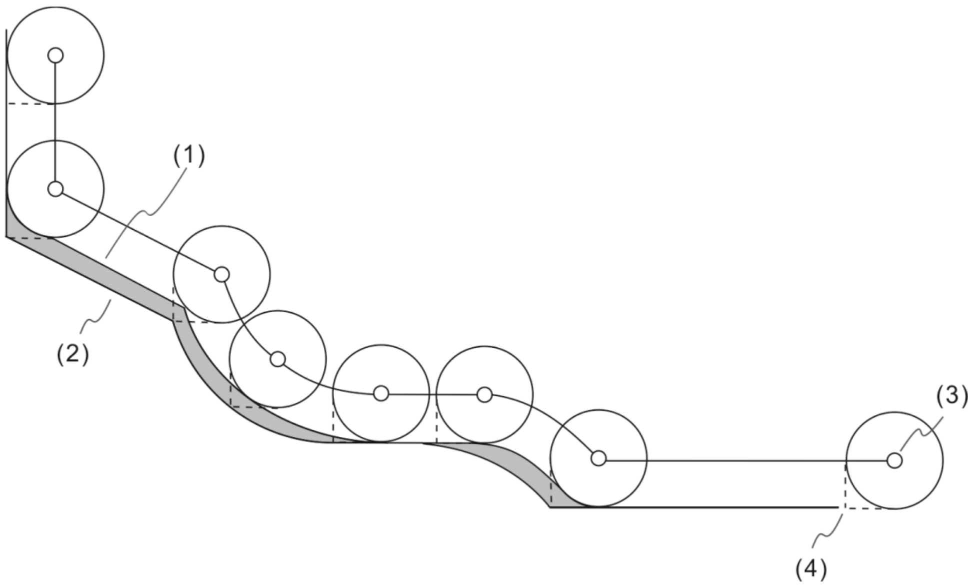
G00 G42 P_

G41: tool radius compensation left

G42: tool radius compensation right

P_: coordinates of the target point.

Description: generally, there is a deviation between the actual contour and the programmed contour when turning in an arc or a diagonal path. It is because the tool nose is usually arc-shaped and the coordinates specified in the program are referring to the hypothetical tool nose position. In this case, G41/G42 can compensate the tool nose radius error based on the settings of the tool radius, tool nose type, and compensation left / right, and can automatically calculate the compensation amount.

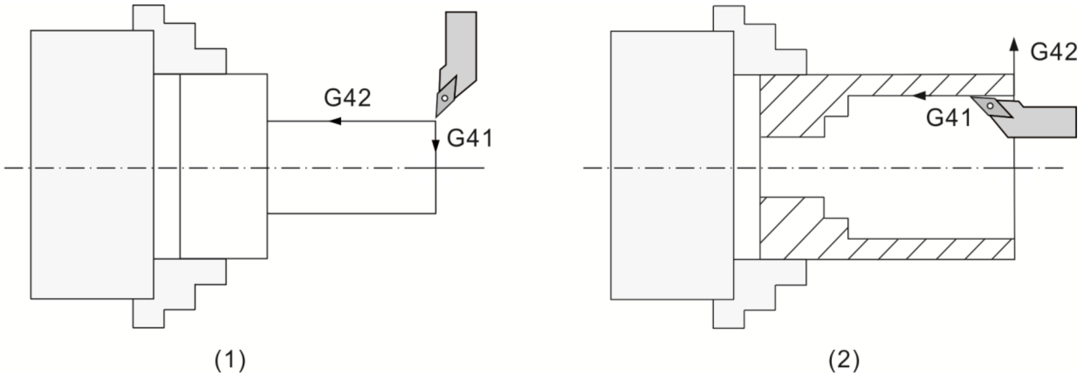


(1) Actual machining contour; (2) Machining contour set in the program; (3) Tool nose center; (4) Hypothetical tool nose position

When using the tool nose radius compensation, you must specify the positive compensation tool number, TXXXX.

2

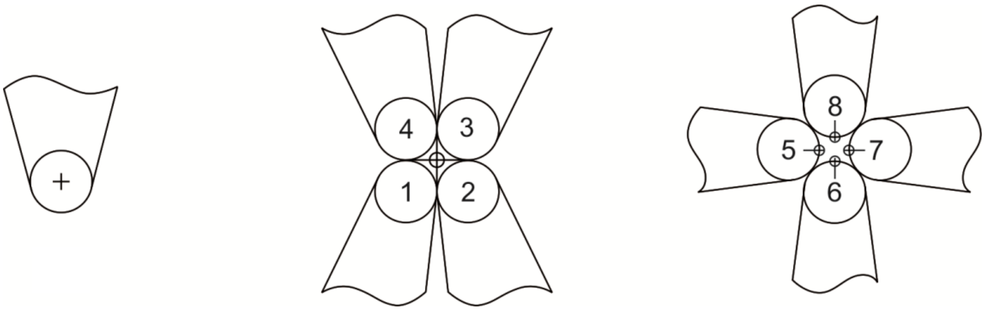
[Compensation setting of the actual machining]



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

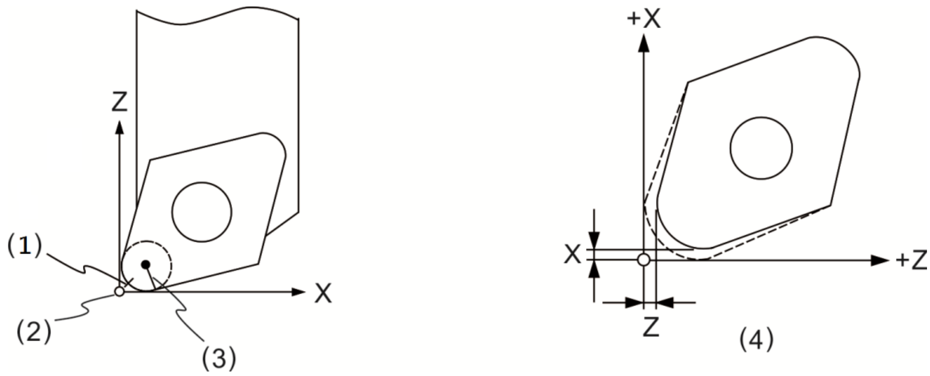
Tool nose type setting: in general, the tool nose is arc-shaped and the tool nose position varies with the tool types, as shown in the figures below. Input the corresponding tool nose type number in the type field of the OFS group tool register.

[Tool nose types]



(1) Tool nose No.0 and No.9 (2) Tool nose No. 1 - 4 (3) Tool nose No. 5 - 8

[Tool radius compensation and compensation for tool radius wear]



(1) Actual tool nose position; (2) Hypothetical tool nose position when calibration;

(3) R value of tool nose radius compensation; (4) Tool nose wear compensation

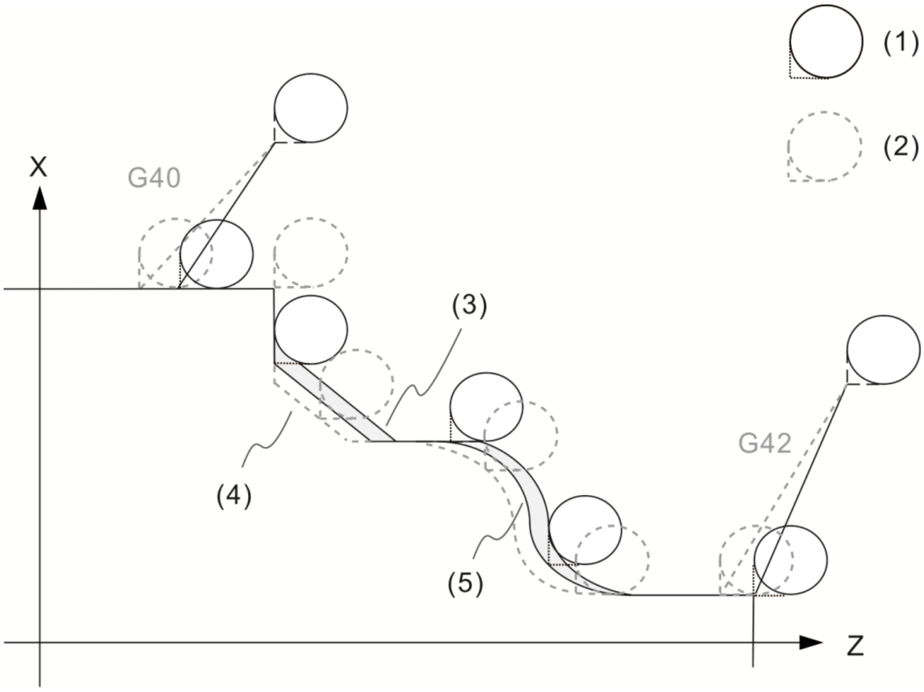
Notes for tool radius compensation:

1. This compensation command can be used with G00 or G01 in the same block. However, it has to be a motion block (tool radius compensation enabled) in order to have the compensation function work.
2. This compensation command cannot be used in the block that includes G02 and G03. To use the compensation function for an arc path, you have to first set the tool radius compensation function for the linear motion path. When the compensation is active, canceling the tool radius compensation in the arc path is not allowed.
3. During program editing, please specify the tool radius compensation number (e.g., T0111 and T0212). Each tool radius compensation number corresponds to a number in the compensation data table.
4. If there is a change in the signs (+, -) of the compensation value, the compensation direction specified in G41 and G42 will change accordingly. For example, when you assign a positive value in G41, the compensation direction is left; when you assign a negative value, the compensation direction becomes right. Likewise, when you assign a positive value in G42, the compensation direction is right; when you assign a negative value, the compensation direction is left.
5. If the 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 setting so the compensation resumes in the next motion block.
6. When the tool radius compensation is executed and the tool completes the programmed path, you must execute G40 to cancel the compensation function. The best timing to use G40 to cancel the tool radius compensation is after the tool has disengaged from the workpiece.

2

[Example 1]

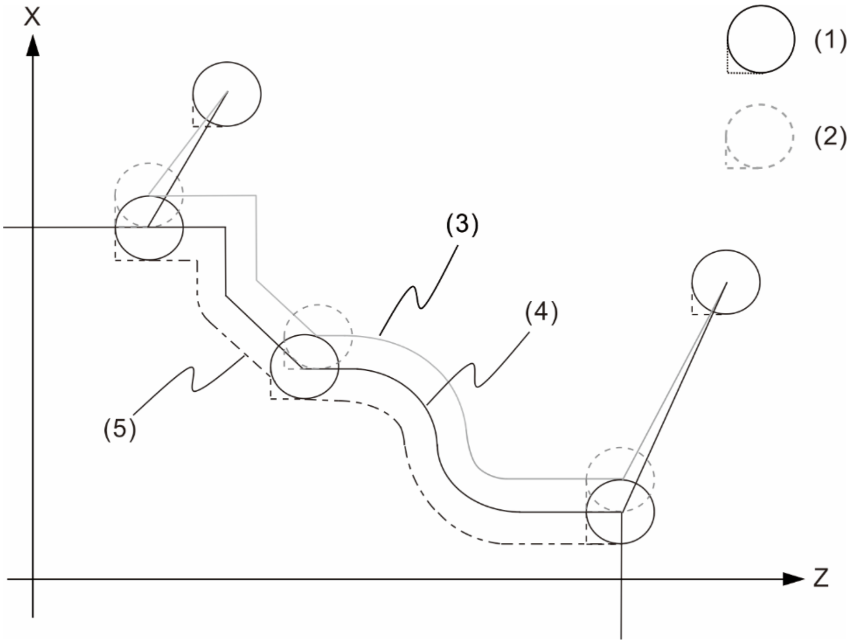
When using tool nose type 3, the machining condition with compensation is as follows.



(1) Program tool path
(2) Compensation tool path
(3) Machining contour without tool compensation;
(4) Tool nose path after compensation;
(5) Machining contour after compensation

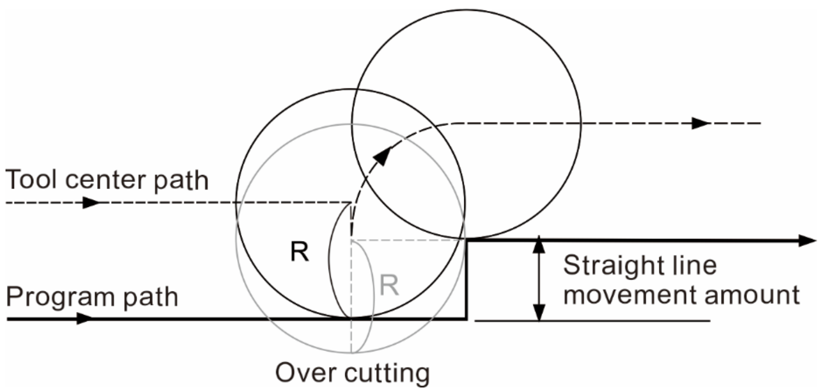
[Example 2]

When using tool nose type 0 or 9, the machining condition with compensation is as follows.

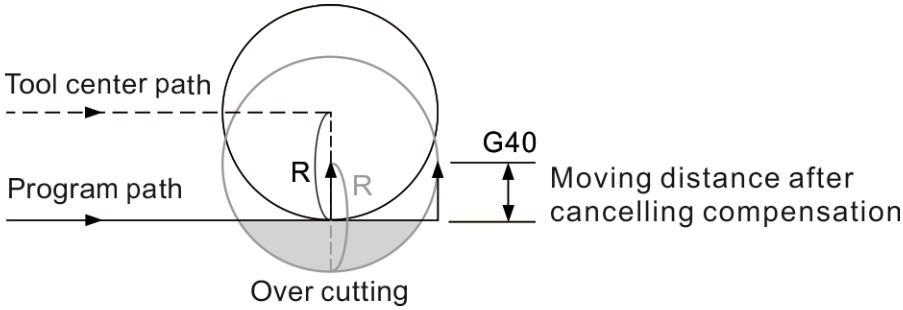


(1) Program tool path
(2) Compensation tool path
(3) Tool center path after compensation
(4) Tool center path without compensation and machining contour with compensation
(5) Machining contour when tool compensation is disabled

When compensation is enabled, the linear moving amount and inner arc interpolation radius must be ≧ the tool nose radius. Otherwise, there will be interference with the compensation vector thus causing excessive cutting. When this issue occurs, the controller will stop running and display the alarm message, as shown in the figure below.



The canceled moving amount of the tool radius compensation must be ≧ the tool radius. Otherwise, there will be interference with the cutting path thus causing excessive cutting. When this issue occurs, the controller will stop running and display the alarm message, as shown in the following figure.



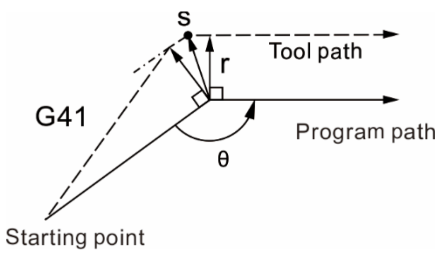
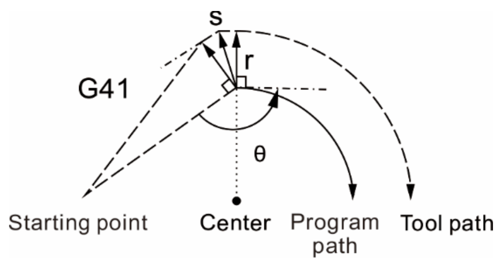
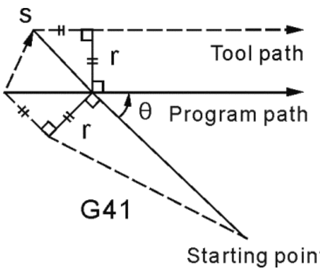
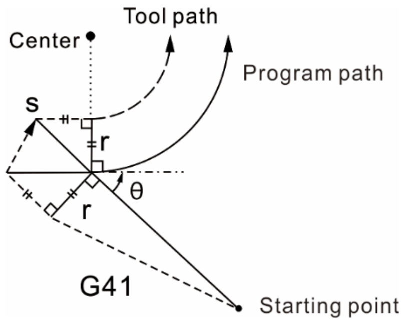
Canceled compensation distance (G40) < Tool nose radius R

The tool radius compensation is not operable in the following conditions:

When you execute tool compensation for the motion block following G40.

Compensation path: compensate for both the start and end points of the tool path.

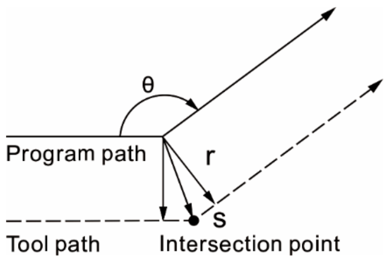
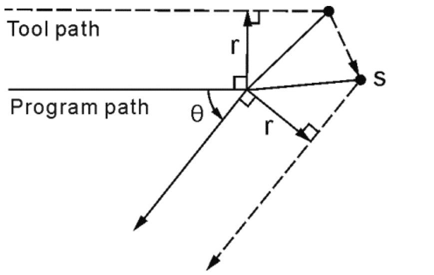
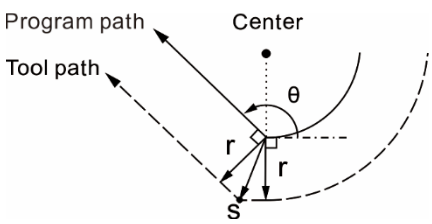
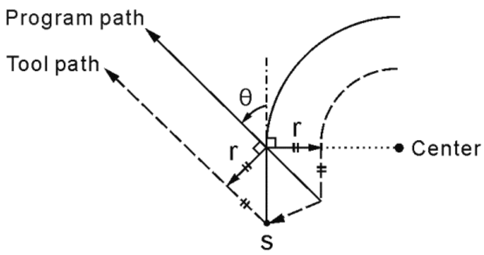
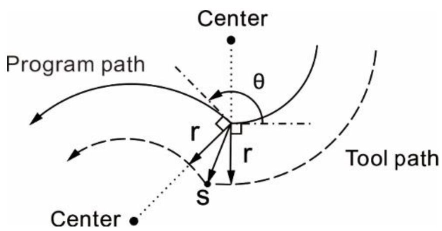
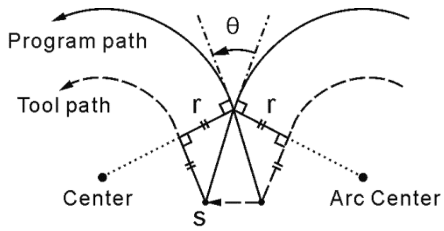
The compensation motion diagram is shown as below.

<table>
<tr><th>Line to line</th><th>Line to arc</th></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

Types of tool radius compensation path:

For the compensation path, you must consider the included angle θ (180° > θ > 90°, 0 < θ < 90°) formed between each block.

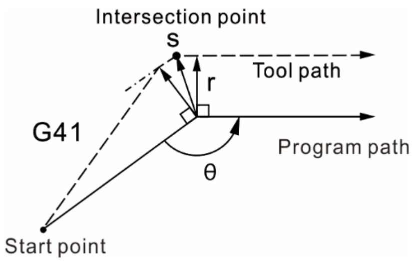
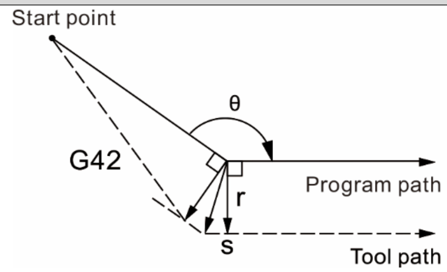
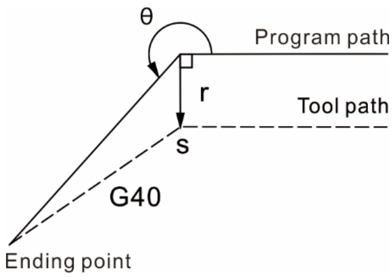
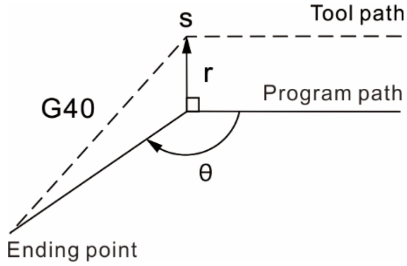
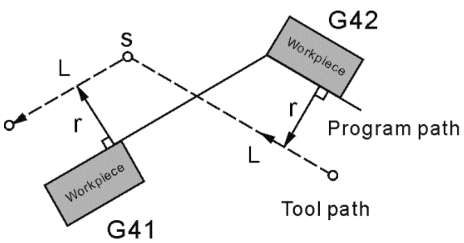
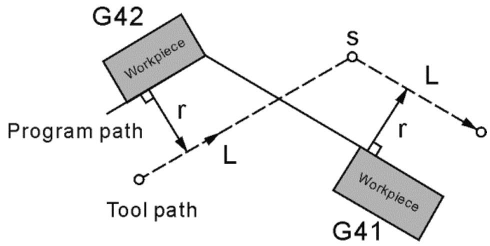
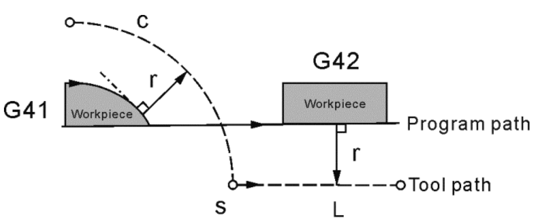
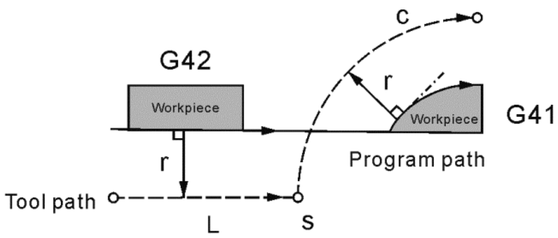
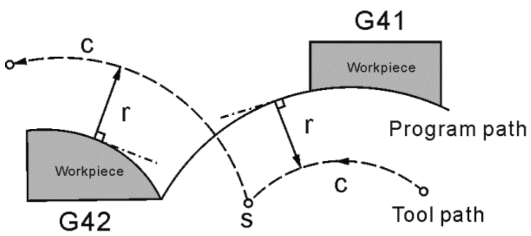
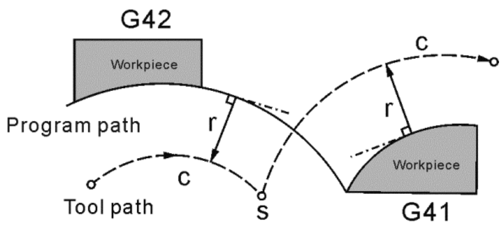
1. If 180° > θ > 90°, the tool radius motion path is inward-shaped.
2. If 0 < θ < 90°, the tool radius motion path is outward-shaped.

Inward angle	Outward angle
	
Inward angle - arc to line	**Outward angle - arc to line**
	
Inward angle - arc to arc	**Outward angle - arc to arc**
	

2

Compensation path switch:

1. The motion path without compensation transits to the tool center motion path with compensation.
2. When the compensation is in execution, the compensation continues to function for the motion path; if you use G40 to cancel the compensation path or directly change the compensation direction, the motion paths are shown as follows.

<table>
<tr><th>G40 > G41</th><th>G40 > G42</th></tr>
<tr><td>
</td><td>
</td></tr>
<tr><th>G41 > G40</th><th>G42 > G40</th></tr>
<tr><td>
</td><td>
</td></tr>
<tr><th>G41 > G42</th><th>G42 > G41</th></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

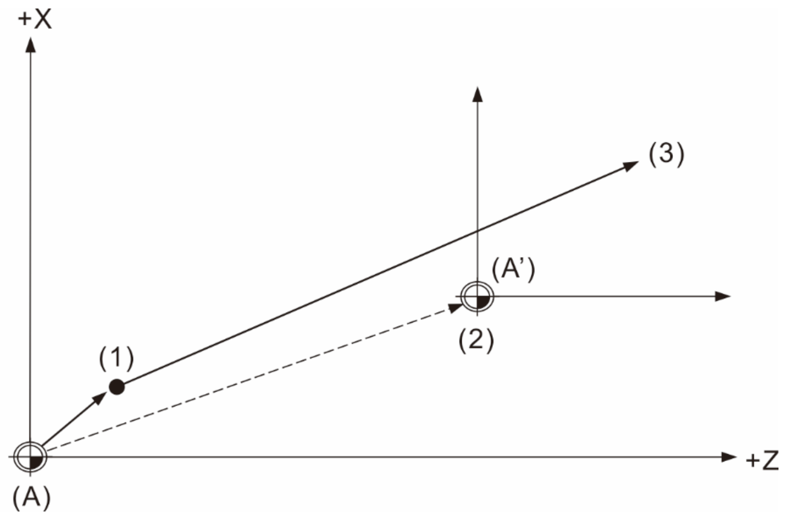
G52: Local coordinate system setting

Format: G52 X_ Y_ Z_

X_ Y_ Z_: local coordinates system origin.

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

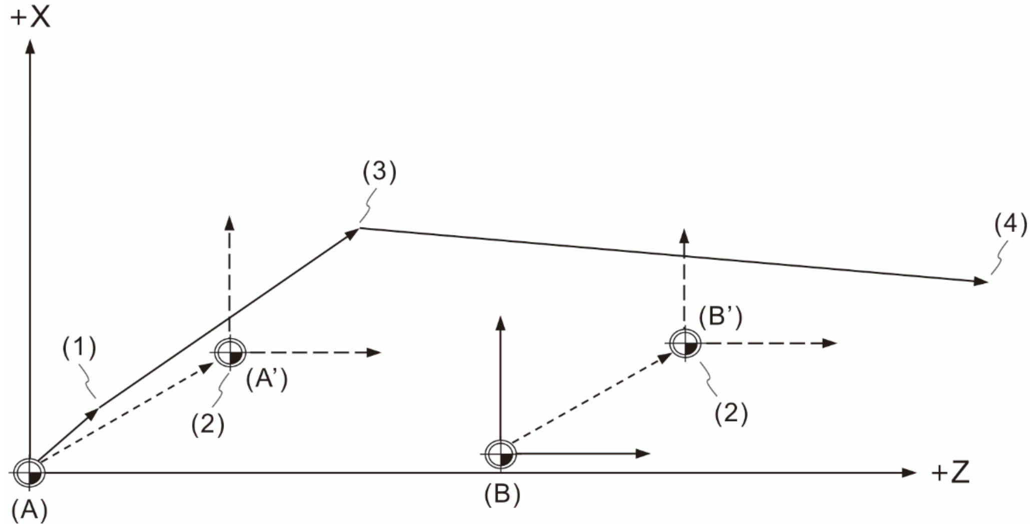
[Example1]



(A) G54 coordinate system; (A') new G54 coordinate system

G54 X10. Z10.;	Move from the origin to point (1)
G52 X20.0 Z40.0;	G54 coordinate system origin shifts to point (2)
G00 X20.0 Z20.0;	Move to point (3) in the new G54 coordinate system

[Example 2]



(A) G54 coordinate system; (A') new G54 coordinate system;

(B) G56 coordinate system; (B') new G56 coordinate system

G54 G00 X5. Z10.;	Move from G54 coordinate system origin to point (1)
G52 X10. Z20.;	G54 coordinate system origin shifts to point (2)
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 effective and the current workpiece coordinate system converts to another coordinate system, the shift setting of G52 is also effective after the conversion.
2. To cancel the local coordinate setting, set 0 for X, Y, Z in G52 (format: G52 X0 Y0 Z0).

G53: Machine coordinate system setting

Format: G53 X_ Y_ Z_

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

Description: coordinates (X, Y, Z) specify the actual end point in the machine coordinate system set in the program. Machine suppliers usually use this command to set the tool change position, which is given based on the machine coordinates. You must set this command in the absolute format; G53 with increment format will not be executed. Command G53 is a one-shot G command and is valid for single block. After machine booting and before using G53 to set the coordinate system, 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 are canceled automatically. The tool radius compensation resumes at the next motion block while the tool length compensation will function unless you set it again.

Note:

1. Command G53 functions only when specified in the absolute format. G53 set with incremental values will not be executed; however, the status command such as G00/G01 in the same block will remain effective and continue to function in the next block.
2. If the block contains G53 and includes an axial command, the axis moves to the specified position; otherwise, the axis does not move.
3. When one block contains both commands G53 and G28, the command read later is effective. If G53 is effective, the movement is made based on the machine coordinates. If G28 is effective, the movement is made based on the absolute coordinates.

[Example]

Example 1:

G53U150.W-150.	(It is in the increment format so this block is omitted.)

Example 2:

G53X50.Z-50.	(Move to the actual machine coordinates X50. Z50.)

Example 3:

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

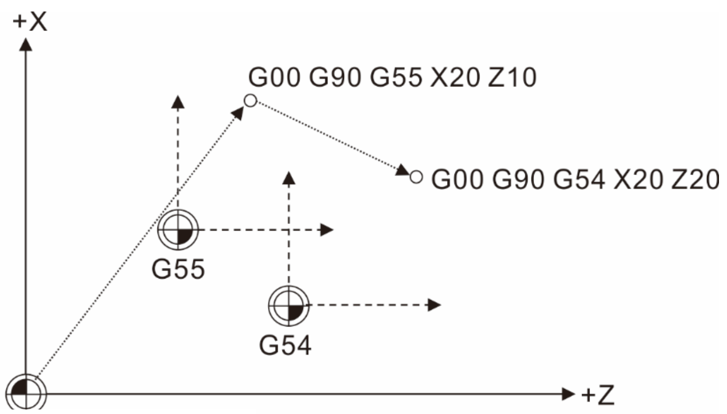
2

G54 - G59: Workpiece coordinate system selection

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

[Example]



Machine origin (0, 0)

G00 G55 X20 Z10
G00 G54 X20 Z20

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

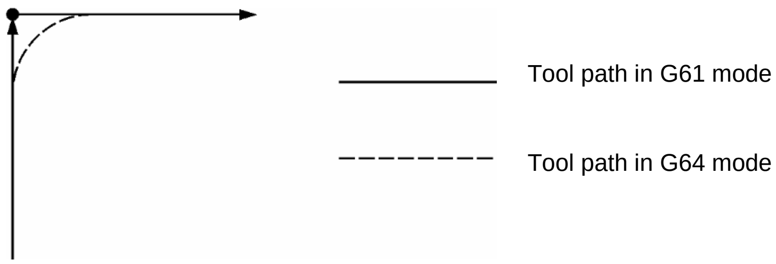
G61: Exact stop mode (one-shot)

Format: G61

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

Note: the machine default is G64.

[Example]



G61 G91 G01 X100. F200.	(Exact positioning)
Z100.	(Exact positioning)
G64	(Exact positioning stops)

G64: Cutting mode

Format: G64

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

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

1. The block contains a G00 command (positioning in rapid traverse)
2. The block contains a G09 command (exact stop)
3. The next block has no motion commands

G65: Macro call (one-shot)

Format: G65 P_ L_I

P_: program number.

L_: number of repetitions.

I_: independent variable.

Description: you can use G65 to call a macro program. Macro programs are used for variable operations, MLC data input / output, control, making statement and discrepancy, and enable the system to perform calculations and measurements.

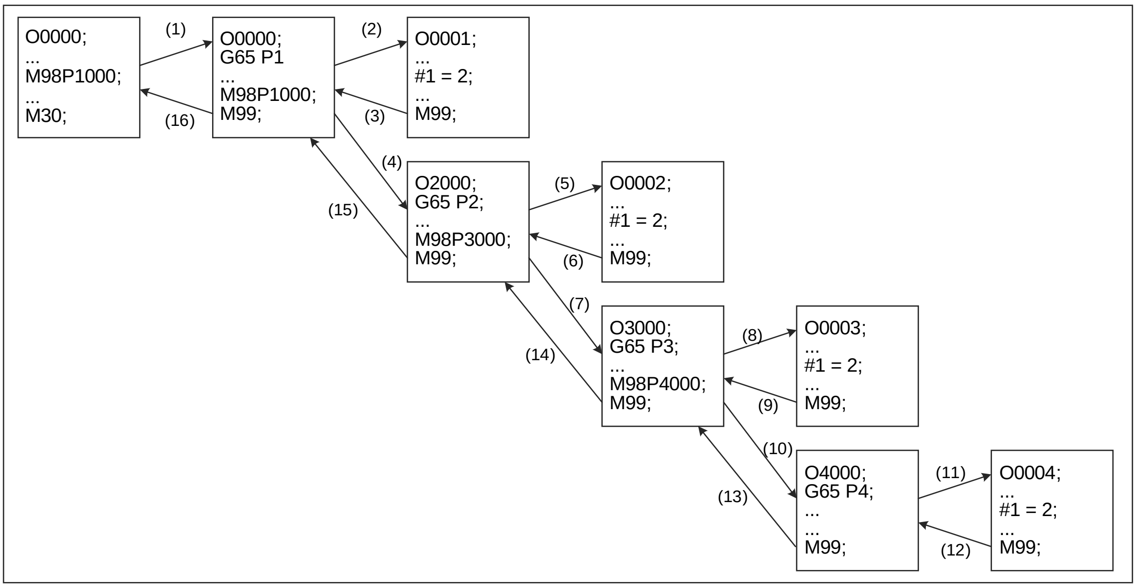
[Example]

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

After the macro program is executed, the program returns to the block following G65 command in the main program, which means the block following G65 is executed. See the example above, A5.0 represents variable #1, which value 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

[Program illustration]



Command G65/G66 can nest macros up to 8 layers. When using with M98 the subprogram call, the maximum layer it can call remains 8.

2

G66/G67: Continuous effect macro call / cancellation

Format: G66 P_ L_I or

G67

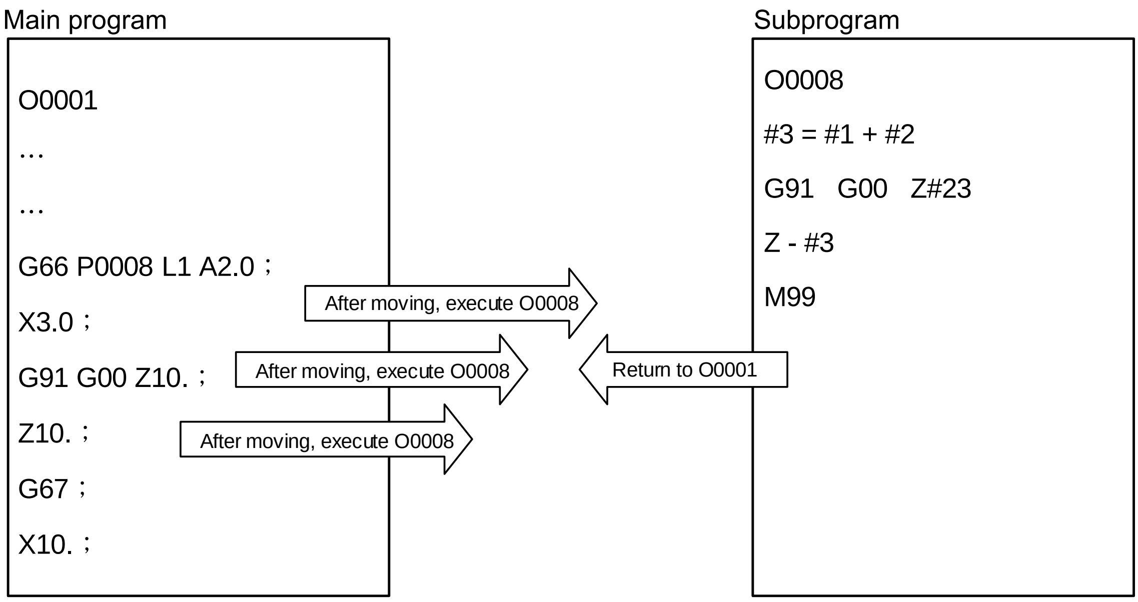
P_: program number.

L_: number of repetitions.

I_: independent variable.

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

[Example]



G71: Multiple type rough turning cycle

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). You can use machining parameter 312 to specify the default value.

Re: retraction amount (input radius value only), which default can be specified with 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.

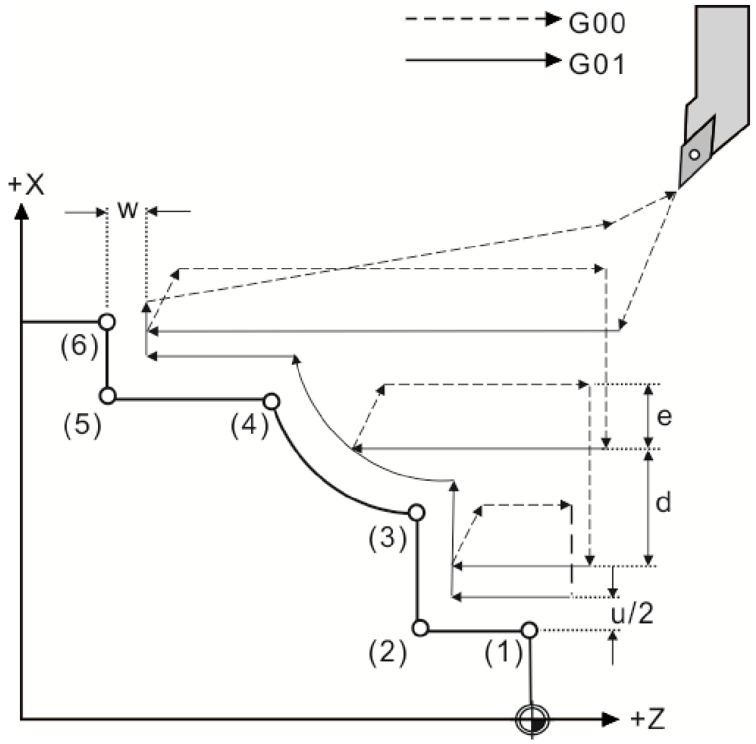
T_: tool number.

S_: spindle speed.

Description: when G71 is executed, the system first reads the end dimension. Next, the system automatically calculates the outer diameter turning path of the workpiece by referring to the parameter settings and then starts the rough turning cycle.

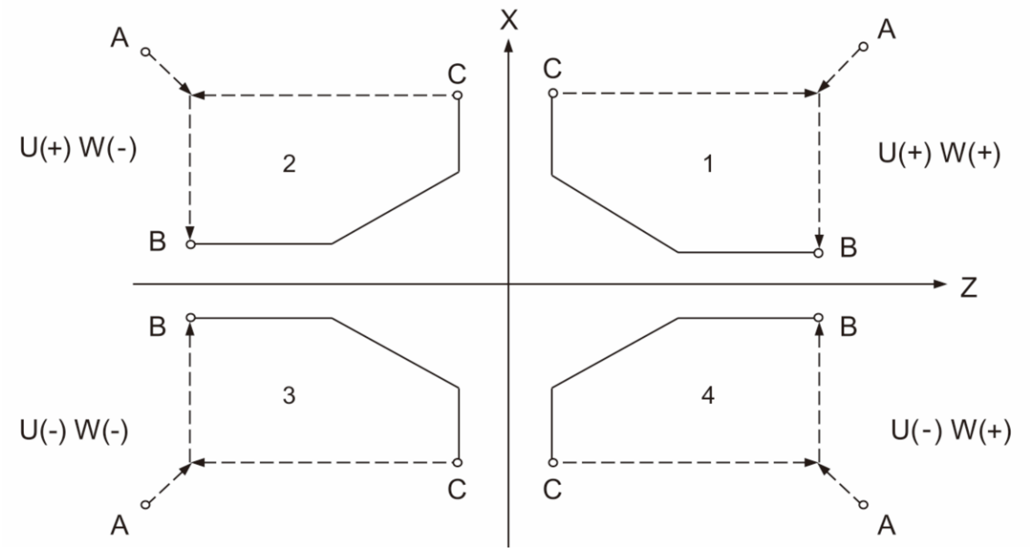
2

[Example]




```
G54 X40.0 Z5.0
M03 S1000
G71 U2. R3.
G71 P50 Q60 U0.2 W0.0 F0.2 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)
N60 X40.                         (6)
M5
M30
```

[Workpiece position and cutting direction]



The figure above shows the workpiece position and the cutting direction corresponding to the positive / negative U and W values; 1 - 4 represent 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. You can use G71 rough turning cycle to perform turning for the concave-shaped workpieces.
2. The P_ and Q_ values specifying the finish cutting sequence cannot be repeated in the program.
3. In the finish turning program, the blocks without movement commands or N, F, S, M, or T command will be omitted.
4. If the command does not specify the cutting depth and retraction amount, the system automatically refers to the parameter settings.
5. The tool compensation does not function 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 one-shot 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 (set as radius), which default value can be specified with machining parameter 312.

Re: retraction amount (set as radius), which default value can be specified with 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).

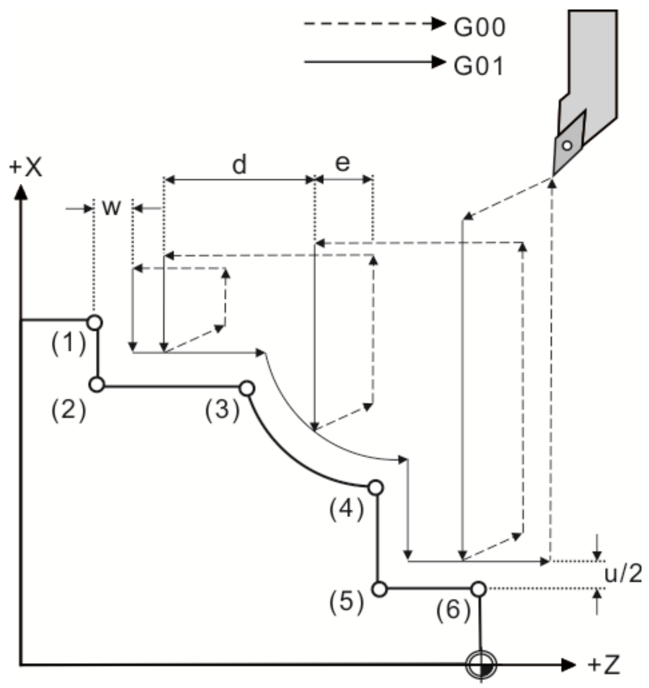
F_: feed rate.

T_: tool number.

S_: spindle speed.

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

[Example]

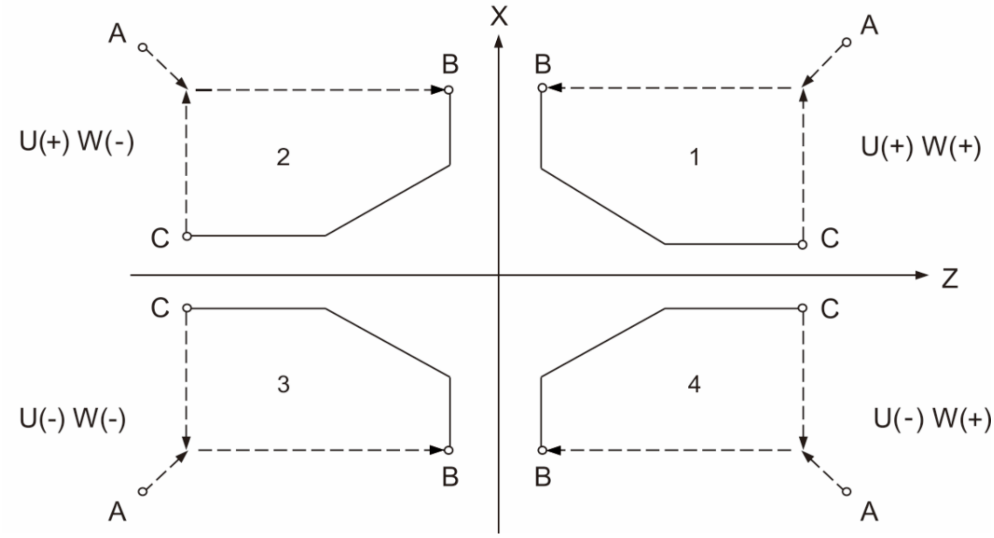


Machining program:

```
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
```

2

[Workpiece position and cutting direction]



The figure above shows the workpiece position and the cutting direction with respect to the positive / negative U and W values; 1 - 4 represent 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 without movement commands or N, F, S, M, or T command will be omitted.
3. When the command does not specify the cutting depth and retraction amount, the system will refer to the parameter setting automatically.
4. The tool compensation is not operable 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 one-shot command.

G73: Multiple type pattern repeating cycle

Format: G73 Ui Wk Rd;

G73 P_ Q_ Uu Ww F_ S_ T_;

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

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

Rd: number 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).

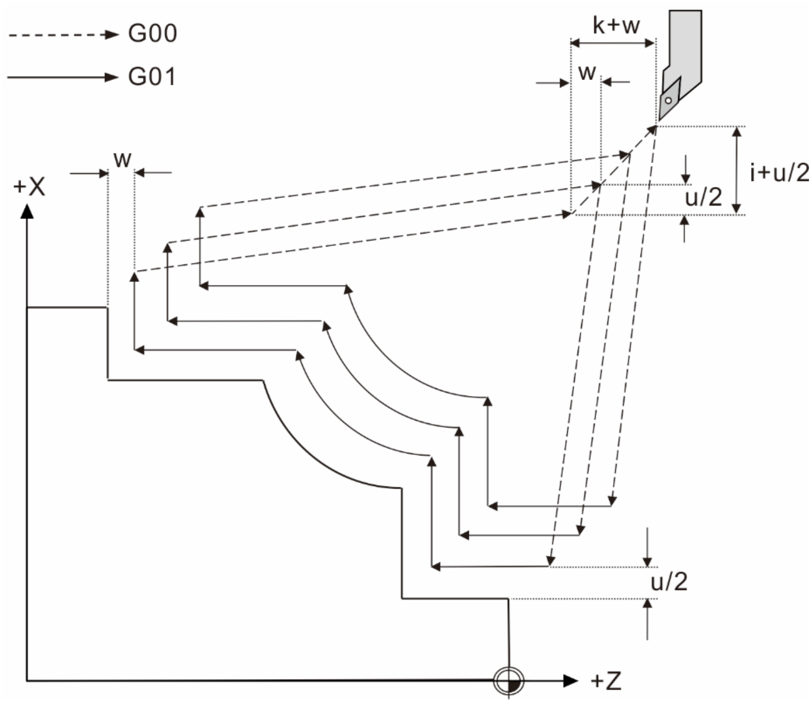
F_: feed rate.

S_: spindle speed.

T_: tool number.

Description: when G73 pattern repeating cycle is used, the system first reads the end dimensions and refers to the parameter setting to automatically calculate the turning path for the workpiece before starting the pattern repeating cycle.

[Contour machining example]



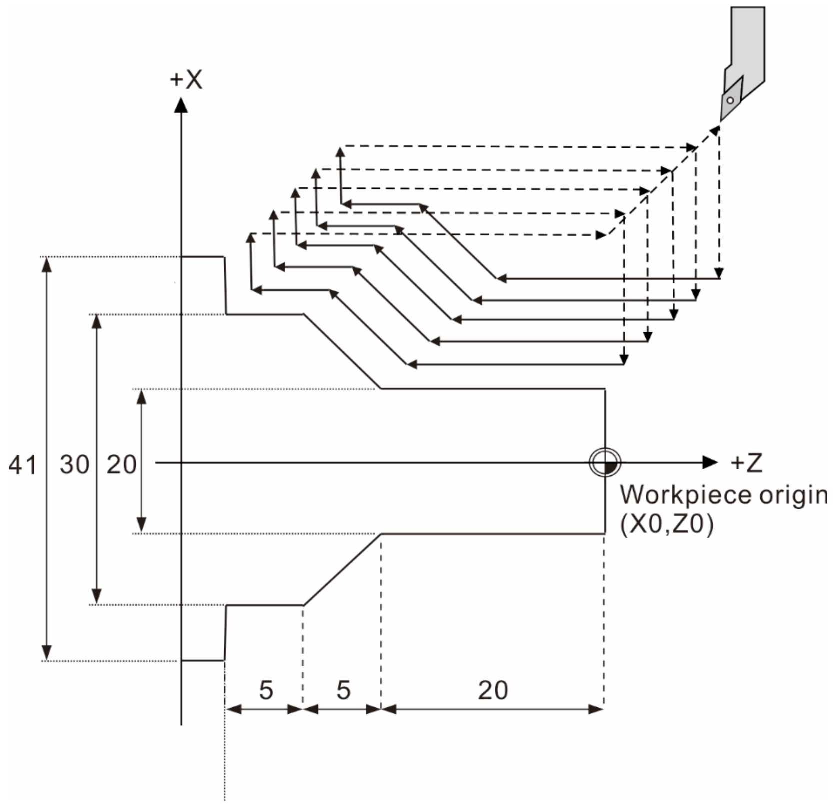
The cutting amount per time: [the total cutting amount of each axis] ÷ [value of cutting times (d-1)]

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, or casting). If the workpiece is a complete bar, using G73 may result in tool or workpiece damage because of the excessive turning amount.

2

[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
```

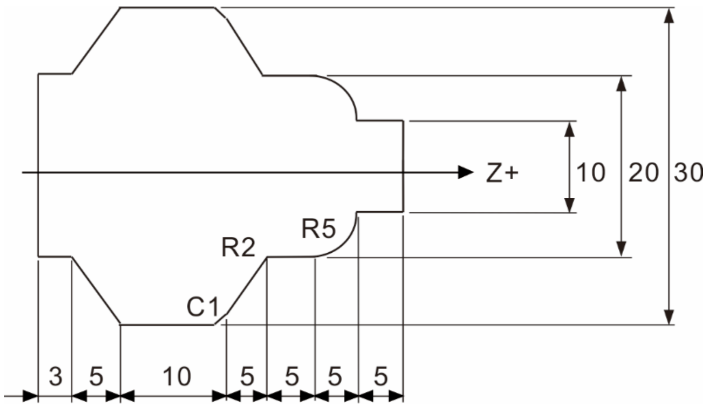
G70: Multiple type finish turning cycle

Format: G70 P_ Q_;

P_: start block number for contour finish turning.
Q_: end block number for contour finish turning.

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

[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 be executed.

2

[Example]

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

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

G74: Multiple type face pecking cycle

Format: G74 Re;

G74 X/U_ Z/W_ PΔi QΔK RΔd F_;

Re: Z axis retraction amount, which you can set its default with machining parameter 348.

X/U__: X-coordinate of the cutting end point / incremental distance of X axis.

Z/W__: Z-coordinate of the cutting end point / incremental distance of Z axis.

PΔi: the tool feeding amount of X axis per cycle; you can only specify the value in radius. When it is an integer, the unit is 0.001 mm.

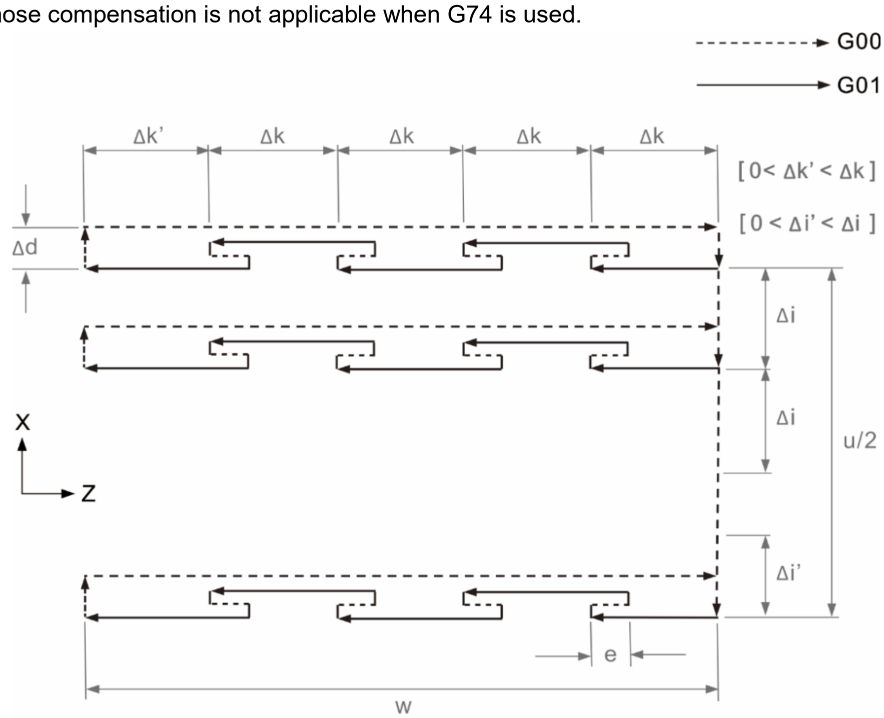
QΔk: the peck turning amount of Z-axis per time. When it is an integer, the unit is 0.001 mm.

RΔd: the escape distance in X-axis direction at the bottom of cut.

F__: feed rate.

Description: G74 is mainly used in the face grooving machining. When G74 is used, the system performs the cycle on the face based on the specified turning end coordinates, cutting amount, tool offset, and escape distance at the bottom of cut.
After completing the turning for amount Δk 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 Δd and returns to the start point of Z axis in rapid traverse. Next, the tool moves for amount Δi in X-axis direction 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 applicable 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. You can only specify the value in radius and set the default with machining parameter 348.

X/U__: X-coordinate of the cutting end point / incremental distance of X axis.

Z/W__: Z-coordinate of the cutting end point / incremental distance of Z axis.

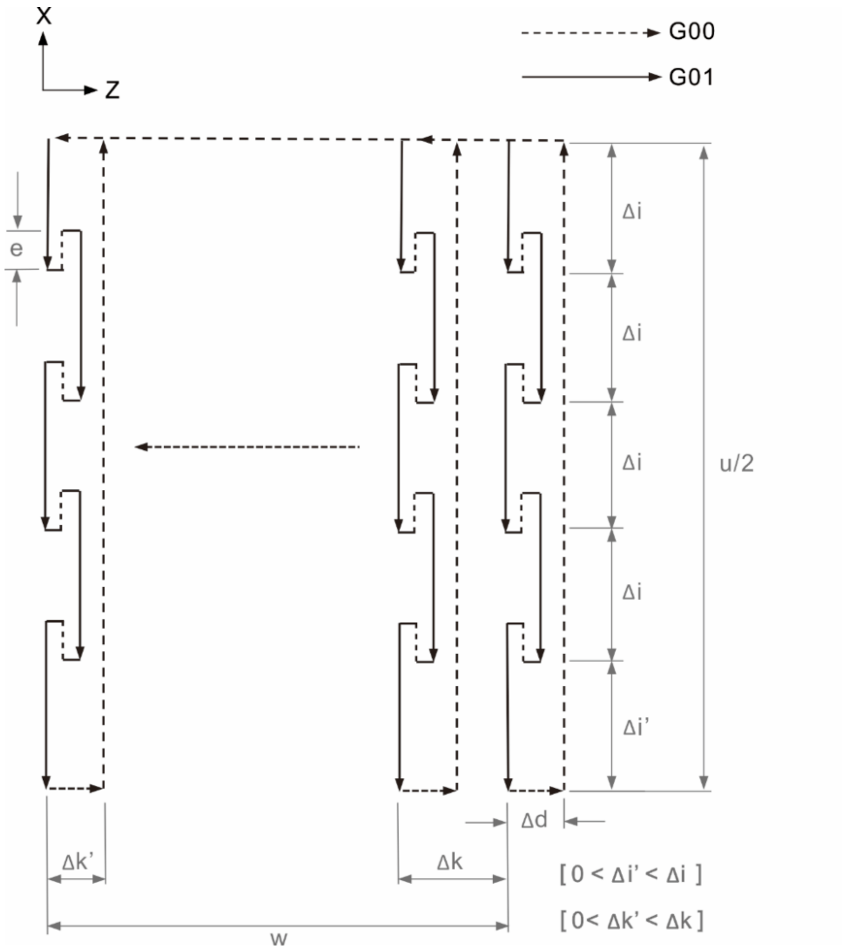
P△i: the peck turning amount of X axis per time; you can only specify the value in radius. When it is an integer, the unit is 0.001 mm.

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

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

F__: feed rate.

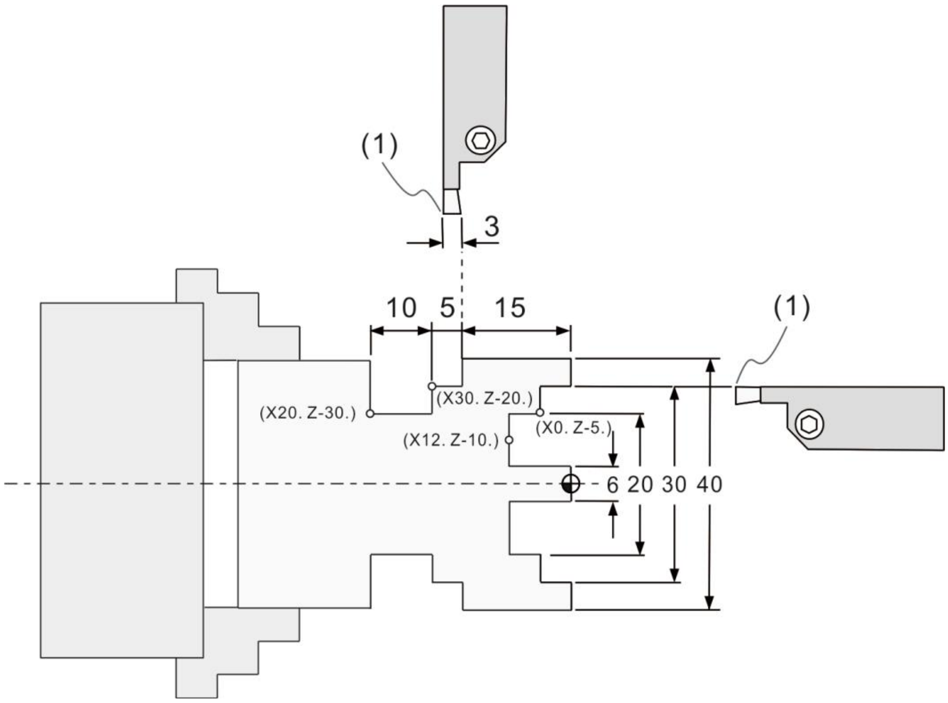
Description: G75 is mainly used for grooving in axial direction. When G75 is used, the system performs auto axial pecking cycle based on the specified turning end coordinates, cutting amount, tool offset, and escape distance at bottom of cut. After completing the turning for amount △i, the tool retracts for amount e. This turning cycle carries on and stops when reaching the target on X axis (bottom). Then, the tool escapes for amount △d and returns to the start point of X axis in rapid traverse. Next, the tool moves for amount △k in Z-axis direction and repeats the motion mentioned above. Finally, it stops when reaching the target on Z-axis. See the following figure for the motion assigned by G75:



Note: tool nose compensation is not applicable when G75 is used.

2

[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    (Start point of G74 cycle)
G0Z5.
X50.
T0505
G0 Z-18.    (Start point of G75 cycle)
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 P<u>mra</u> Q<u>△dmin</u> R_;

G76 X/U_ Z/W_ R<u>i</u> P<u>k</u> Q<u>△d</u> F_;

P<u>mra</u>: m represents the times of finish turning (1 - 99) and you can define the default value with machining parameter 381; r represents the chamfering amount (0 - 99). Assume that L is the threading lead, the chamfering setting is 0.1*r*L, and you can set the default value with machining parameter 380. a represents the tool nose angle (threading angle), which you can specify the default by machining parameter 382 (with available options of 0°, 29°, 30°, 55°, 60°, and 80°).

For example, when you set P011160, it means the number of the finish turning time is 1, chamfering amount is 1.1 L (L is the threading lead), and tool nose angle is 60°.

Q<u>△dmin</u>: minimum cutting depth. When you input an integer, its unit is 0.001 mm and you can set the default value with machining parameter 383.

R_: finish allowance specified with radius. You can set the default with machining parameter 439.

X/U_: X-coordinate of the threading end point / incremental threading distance of X axis.

Z/W_: Z-coordinate of the threading end point / incremental threading distance of Z axis.

R<u>i</u>: the difference when [Start screw thread radius] minus [End screw thread radius] (set as radius value).

P<u>k</u>: thread depth (set as radius value) When it is an integer, the unit is 0.001 mm.

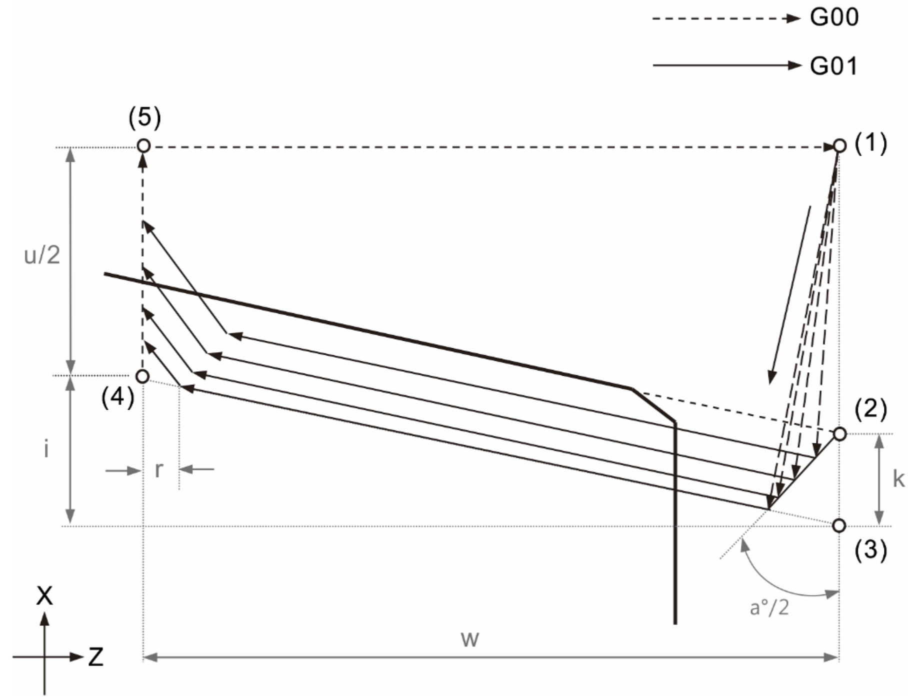
Q<u>△d</u>: depth of the first cut (set as radius value) When it is an integer, the unit is 0.001 mm.

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

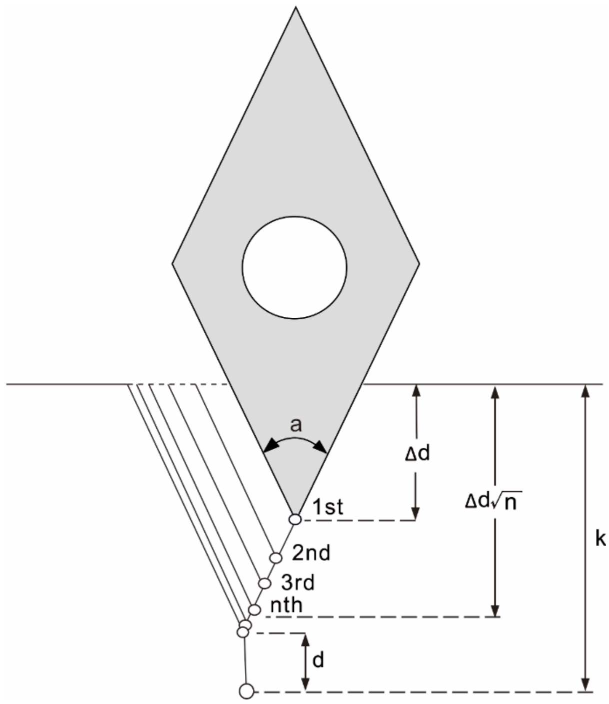
Description: when you set the cutting end coordinate in G76, the system performs the thread turning cycle according to the specified turning times with the fixed amount.

2

[Thread turning]



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



Note:

1. You must execute G76 Thread turning cycle when the spindle speed is fixed.
2. When G76 is executed, spindle speed keeps at 100%.

G90: Axial turning cycle

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

X/U_: X-coordinate of the cutting end point / incremental distance of X axis (set as diameter / radius).

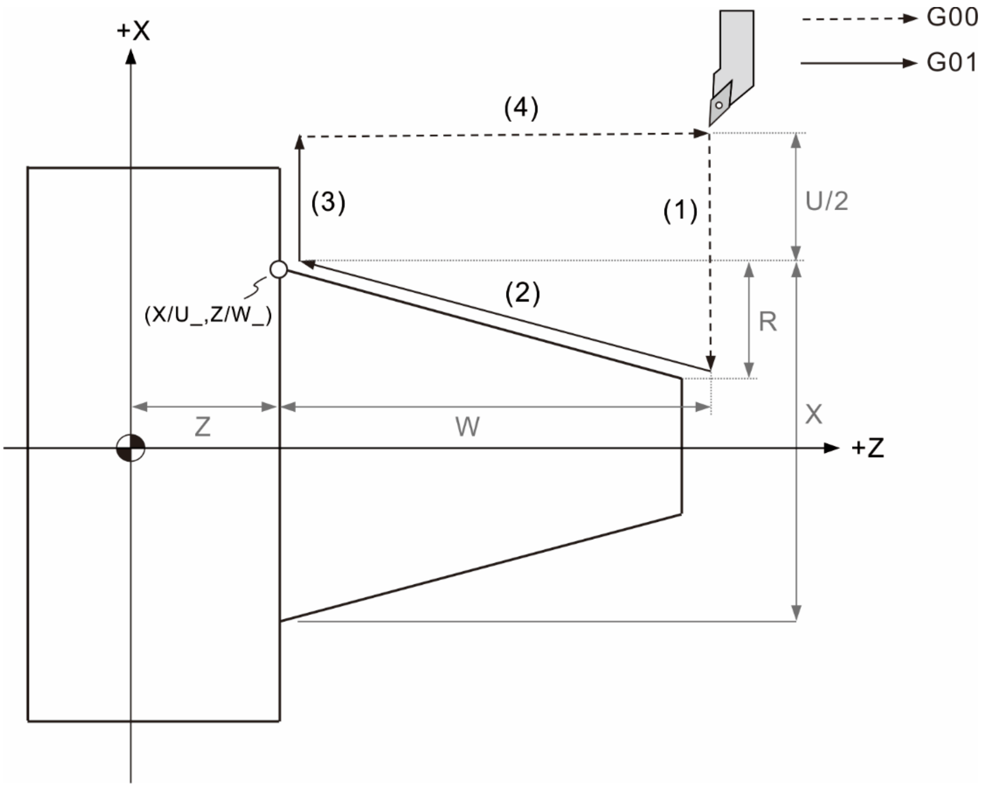
Z/W_: Z-coordinate of the cutting end point / incremental distance of Z axis.

R_: taper amount (set as radius value with positive / negative sign). When axial linear turning, R_ value is omitted.

F_: feed rate.

Description: when you use G90, the system performs a complete outer diameter (taper) turning cycle.

[Contour machining example]



G90 motion analysis:

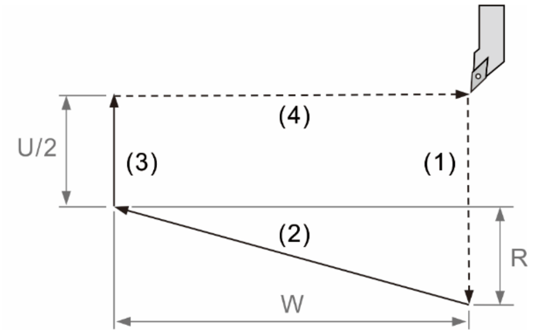
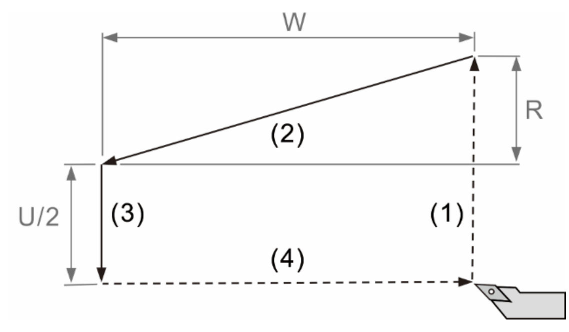
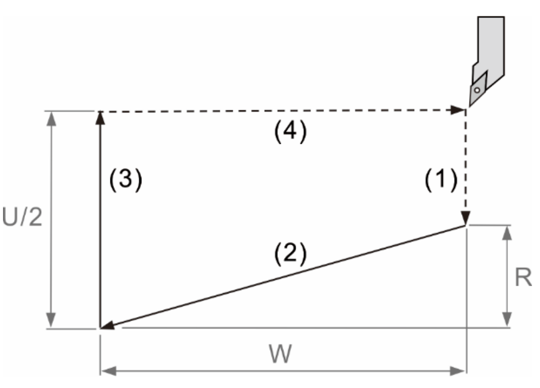
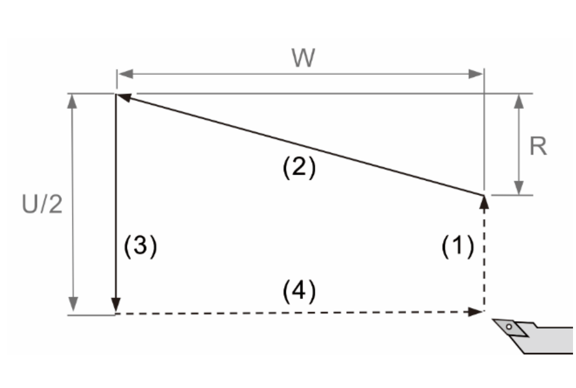
(1) Move from the start point to the assigned coordinates of X-axis according to the given taper moving distance / direction in rapid traverse.

(2) Move to the assigned Z-axis and X-axis coordinates.

(3) Move to the start point on X axis.

(4) Return to the start point in rapid traverse.

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

Outer diameter turning	Inner diameter turning
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 you use the G90 axial turning cycle, its setting is continuously effective. To cancel or quit the G90 command, you can use G92, G94, G80 (cycle cancel), or the Group 1 commands (G00, G01, G02, and G03).

G92: Threading cycle

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

X/U_: X-coordinate of the cutting end point / incremental distance of X axis (set as diameter / radius).

Z/W_: Z-coordinate of the cutting end point / incremental distance of Z axis.

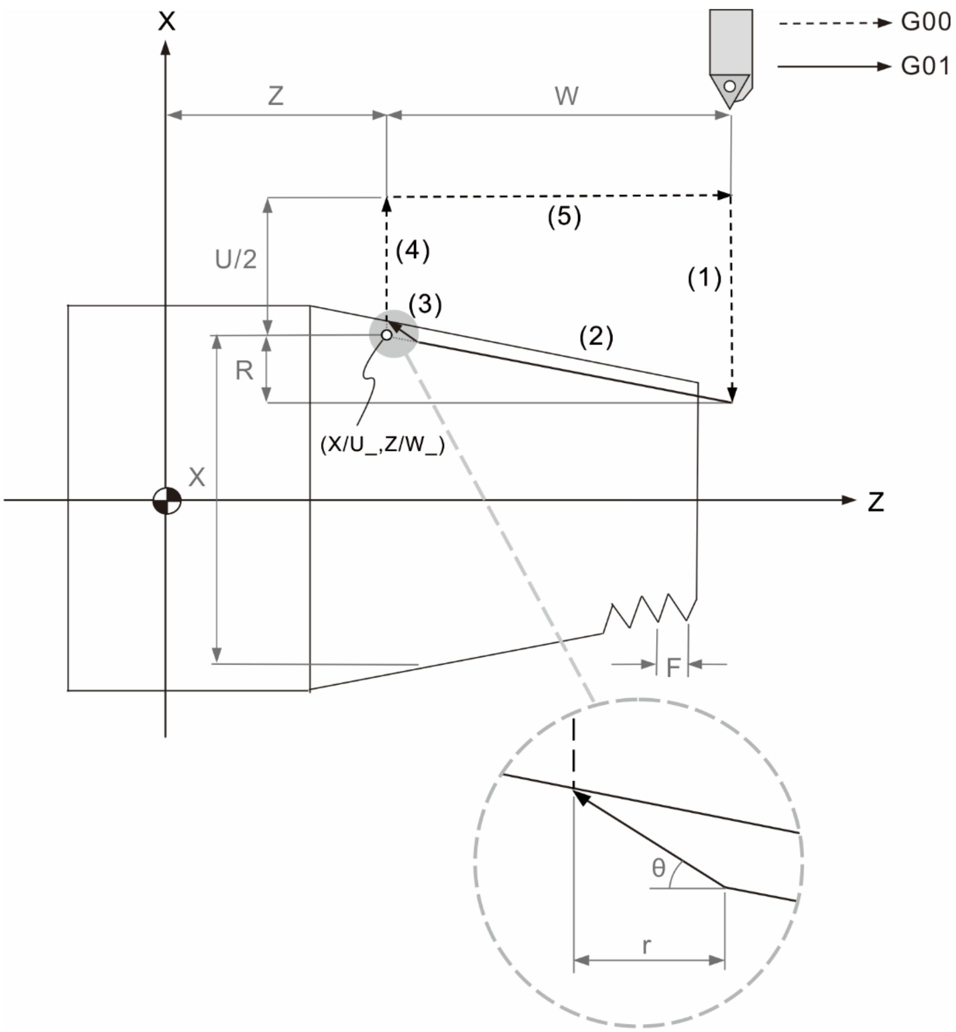
R_: taper amount (set as radius value with positive / negative sign). When axial linear turning, R_ value is omitted.

F_: thread lead.

Q_: offset angle for threading start. If you input an integer, the unit is 0.001 degree.

Description: when you use G92, the system performs a complete outer diameter (taper) threading cycle once.

[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 X-axis coordinate according to the given taper amount in rapid traverse.

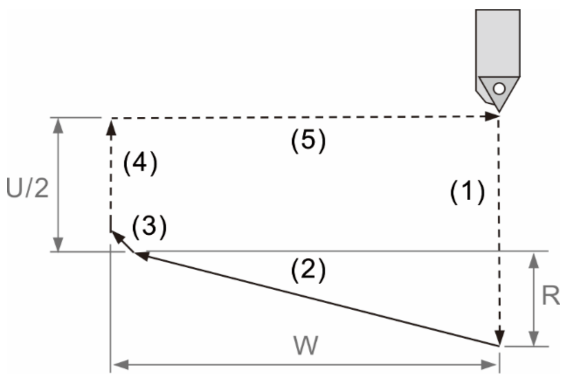
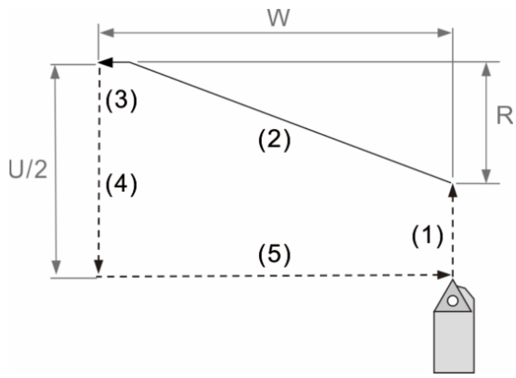
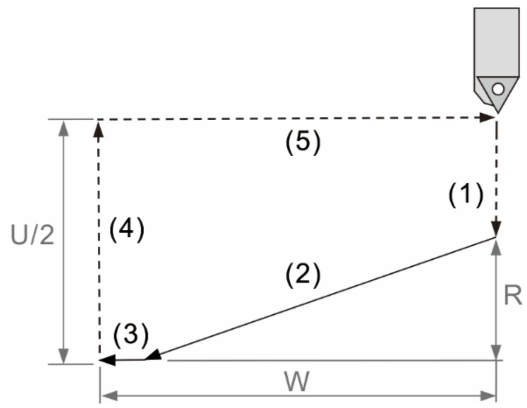
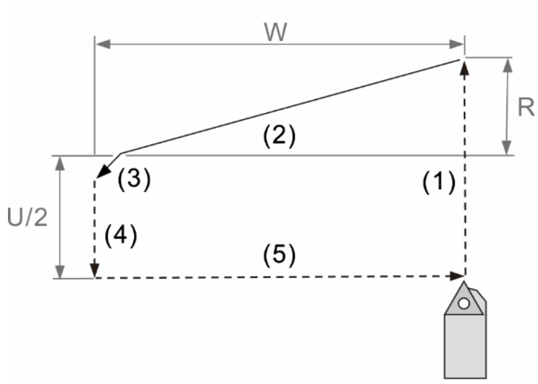
(2) Perform thread turning to the specified Z-axis and X-axis coordinates.

(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.

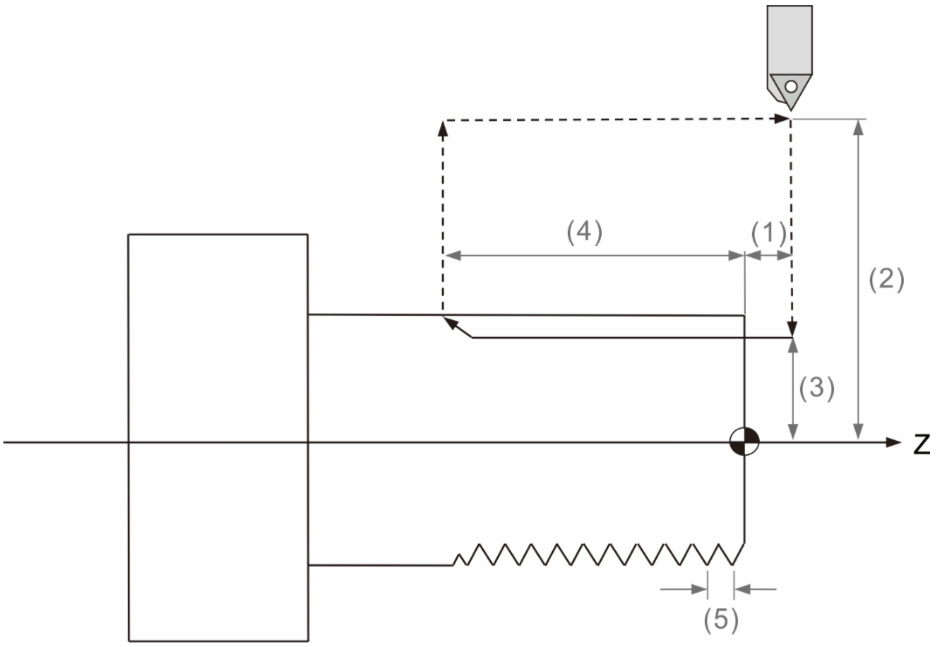
The turning directions corresponding to symbol U, W, and R:

<table>
<tr><th>Outer diameter turning</th><th>Inner diameter turning</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 you use the G92 threading cycle, its setting is continuously effective. To cancel or quit the G92 command, you can use G90, G94, G80 (cycle cancel), or the Group 1 commands (G00, G01, G02, G03).
2. You can set the default chamfer angle for thread turning by machining parameter 349 and set the default chamfer length by machining parameter 380.
3. Please refer to the note for G32 thread cutting command when performing thread turning.

[Example]



O0010	
T0202;	Select tool No.2 and tool compensation No.2
M03 S2000;	Spindle rotates forward at 2,000 rpm
G0 Z5.;	(1) Z-axis threading start point
X30.;	(2) X-axis retraction distance
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 to a safe distance
Z10.;	The tool retracts to a safe distance
M30;	Program end

G94: Face turning cycle

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

X/U_: X-coordinate of the cutting end point / incremental distance of X axis (set as diameter / radius).

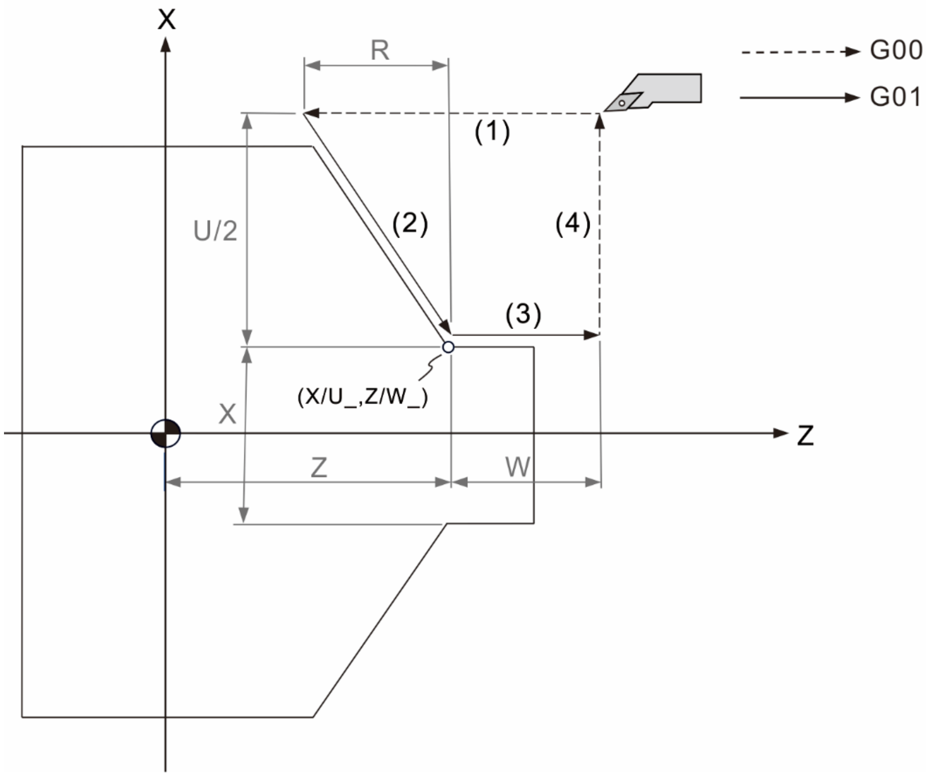
Z/W_: Z-coordinate of the cutting end point / incremental distance of Z axis.

R_: taper amount (set as radius value with positive / negative sign). When straight cutting, R_ value will be omitted.

F_: feed rate.

Description: when you use G94, the system performs a complete face (taper) turning cycle.

[Contour machining example]



G94 motion analysis:

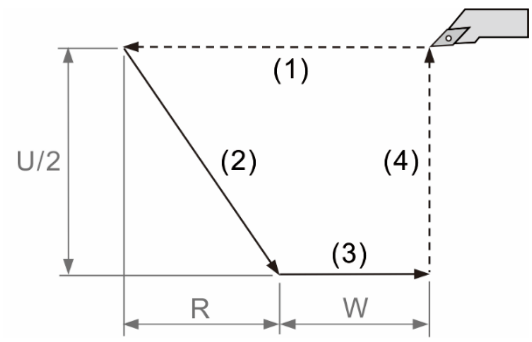
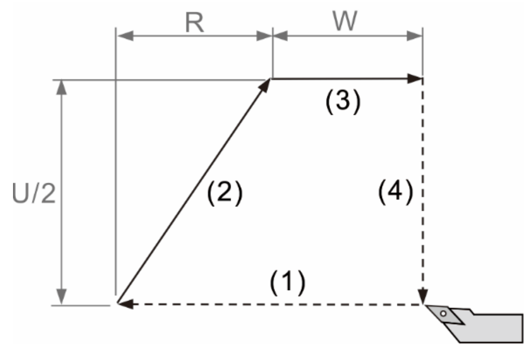
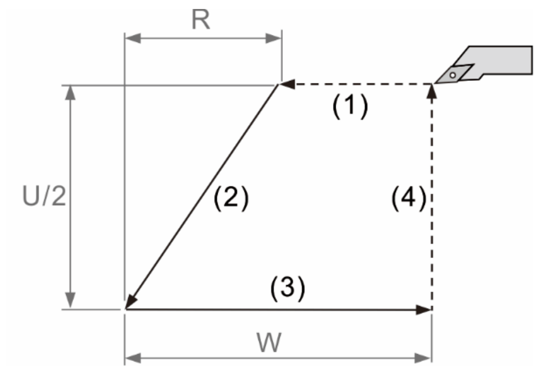
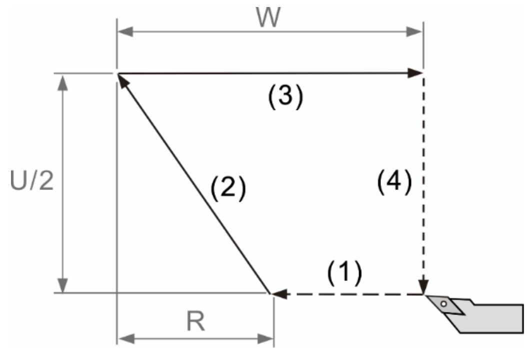
(1) Move from the start point to the assigned Z-axis coordinate according to the given taper amount in rapid traverse.

(2) Move to the assigned Z-axis and X-axis coordinates.

(3) Move to the start point on Z axis.

(4) Return to the start point in rapid traverse.

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

<table>
<tr><th>Outer diameter turning</th><th>Inner diameter turning</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 figures represent the sequence of the tool movement.

Note: when you use the G94 face turning cycle, its setting is continuously effective. To cancel or quit the G94 command, you can use G90, G92, G80 (Cancel cycle), or the Group 1 commands (G00, G01, G02, and G03).

2

G80: Cancel cycle

Format: G80;

Description: this command is for canceling the G90, G92, and G94 cycles. It can also cancel the drilling cycles (G83 and G87), tapping cycles (G84 and G88), and boring cycles (G85 and G89). In addition, you can use Group 1 commands, such as G00, G01, G02, and G03 to cancel the cycles.

G83: Face drilling cycle

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

X(U)_C_: hole position. (In non-turning / milling mode, if the X-coordinate of the face drilling point is not the 0 point of the workpiece coordinates, an alarm showing "B6A5: Turning drilling and tapping cmd error" will occur.)

Z(W)_: bottom of the hole.

R_: the reference point set as an incremental value, which the tool moves toward the opposite direction of the workpiece.

Q_: depth of cut for each cutting feed.

P_: dwell time at the bottom of the hole in the unit of ms (input integers only).

F_: feed rate.

K_: number of repetitions for machining, which is 1 by default.

L_: machining cycle mode selection; this parameter functions the same as parameter 326 (cycle setting). When there is an L argument, the system will first refer to L; otherwise, it refers to parameter 326.

M_: M-code call, which is usually used for clamping / unclamping control of the C axis. Action flow of M-code call: when the tool moves to the start point, the system executes the M-code (e.g., M80: C-axis clamping). Then, when the tool completes drilling and retracts to point R and before pausing at the top of the hole, the system executes M + 1 code (e.g., M81: C-axis unclamping).
Parameter 513: dwell time at the top of the hole. When the cycle command is completed, the tool pauses at point R for the duration (unit: second) set in parameter 513.

Description: each time the tool drills by the distance (depth) of Q, it retracts by the distance set in parameter 324 in rapid traverse or retracts to point R according to the setting of parameter 326. This cycle carries on and stops when the tool drills to depth Z.

Cycle parameter 326:

If bit 2 - 3 are 0, the tool does not perform peck drilling; it drills directly to the bottom.

If bit 2 - 3 are 1, the tool performs deep peck drilling, which feeding amount is Q and retracts to point R.

If bit 2 - 3 are 2, the tool performs general peck drilling, which feeding amount is Q and the escape amount is set in parameter 324.

G84: Face tapping cycle

Format: G84 X(U)_ C_ Z(W)_ R_ Q_ P_ F_ K_L_M_;

X(U)_ C_: tapping position. (In non-turning / milling mode, if the X-coordinate of the face tapping point is not the 0 point of the workpiece coordinates, an alarm showing "B6A5: Turning drilling and tapping cmd error" will occur.)

Z(W)_: bottom of the hole

R_: the reference point set as an incremental value, which is the distance from the start point to the bottom of the hole in Z-axis direction.

P_: dwell time at bottom of the hole in the unit of ms (input integers only).

F_: Feed rate

K_: number of repeated machining time, which is 1 by default.

L_: machining cycle mode selection; this parameter functions the same as parameter 326 (cycle setting). When there is an L argument, the system will first refer to L; otherwise, it refers to parameter 326.

M_: M-code call, which is usually used for clamping / unclamping control of the C axis. Action flow of M-code call: when the tool moves to the start point, the system executes the M-code (e.g., M80: C-axis clamping). Then, when the tool completes tapping and retracts to point R and before pausing at the top of the hole, the system executes M + 1 code (e.g., M81: C-axis unclamping).
Parameter 513: dwell time at the top of the hole. When the cycle command is completed, the tool pauses at point R for the duration (unit: second) set in parameter 513.

Description: the spindle first rotates one cycle to look for the Z pulse and execute positioning once the Z pulse is found. Then, the Z axis moves to the reference point R in rapid traverse. Next, the spindle rotates and taps to the specified bottom position, and then retracts to R in reverse direction, and finally returns to the initial point in rapid traverse.

Cycle parameter 326:

If bit 2 - 3 are 0, the tool does not perform peck tapping; it taps directly to the bottom.

If bit 2 - 3 are 1, the tool performs deep peck tapping, which feeding amount is Q and retracts to point R.

If bit 2 - 3 are 2, the tool performs general peck tapping, which feeding amount is Q and the escape amount is set in parameter 324.

G85: Face boring cycle

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

X(U)_ C_: boring position. (In non-turning / milling mode, if the X-coordinate of the face boring point is not the 0 point of the workpiece coordinates, an alarm showing "B6A5: Turning drilling and tapping cmd error" will occur.)

Z(W)_: bottom of the hole.

R_: the reference point set as an incremental value, which is the distance from the start point to the bottom of the hole in Z-axis direction.

P_: dwell time at the bottom of the hole in the unit of ms (input integers only).

F_: Feed rate.

K_: number of repetition for machining, which is 1 by default.

M_: M-code call, which is usually used for clamping / unclamping control of the C axis.

Action flow of M-code call: when the tool moves to the start point, the system executes the M-code (e.g., M80: C-axis clamping). Then, when the tool completes boring and retracts to point R and before pausing at the top of the hole, the system executes M + 1 code (e.g., M81: C-axis unclamping).

Parameter 513: dwell time at the top of the hole. When the cycle command is completed, the tool pauses at point R for the duration (unit: second) set in parameter 513.

Description: G85 is usually used for machining with reamers or boring bars for the applications that require higher precision in hole diameters. Before starting boring, the tool goes to point R in rapid traverse. Then, the tool cuts to point Z with the set feed rate F and retracts to R with the same speed. Finally, the tool returns to the start point with G00 fast positioning command.

G87: Side drilling cycle

Format: G87 Z(W)_ C_ X(U)_ R_ Q_ P_ F_ K_L_M_;

Z(W)_ C_: drilling position.

X(U)_: bottom of the hole.

R_: the reference point set as an incremental value, which the tool moves toward the direction of the bottom of the hole.

Q_: depth of peck drilling for each cutting feed. When Q_ value is an integer, the unit is 0.001 mm.

P_: dwell time at the bottom of the hole in the unit of 0.001 ms (input integers only).

F_: feed rate.

K_: number of repetitions for machining, which is 1 by default.

L_: machining cycle mode selection; this parameter functions the same as parameter 326 (cycle setting). When there is an L argument, the system will first refer to L; otherwise, it refers to parameter 326.

M_: M-code call, which is usually used for clamping / unclamping control of the C axis. Action flow of M-code call: when the tool moves to the start point, the system executes the M-code (e.g., M80: C-axis clamping). Then, when the tool completes drilling and retracts to point R and before pausing at the top of the hole, the system executes M + 1 code (e.g., M81: C-axis unclamping).
Parameter 513: dwell time at the top of the hole. When the cycle command is completed, the tool pauses at point R for the duration (unit: second) set in parameter 513.

Description: the tool performs peck drilling for distance (depth) Q_ and retracts by the distance set in parameter 324 or retracts to point R based on the setting of parameter 326 in rapid traverse. This cycle continues until the tool performs machining to the depth of Z.

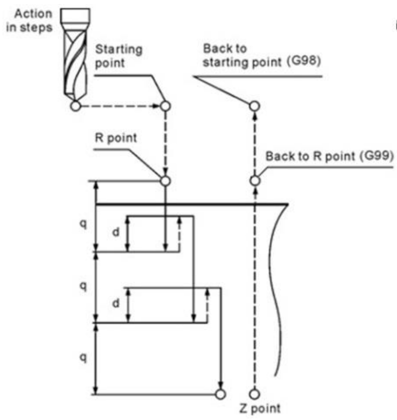
Cycle parameter 326:

If bit 2 - 3 are 0, the tool does not perform peck drilling; it drills directly to the bottom.

If bit 2 - 3 are 1, the tool performs deep peck drilling, which feeding amount is Q and retracts to point R.

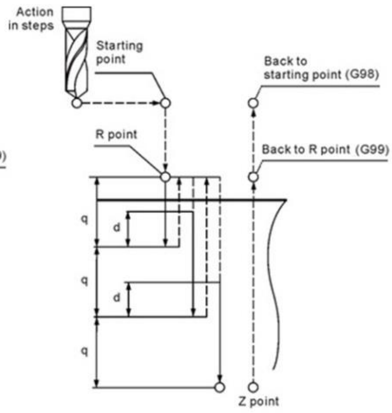
If bit 2 - 3 are 2, the tool performs general peck drilling, which feeding amount is Q and the escape amount is set in parameter 324.

[Example] G83 / G87



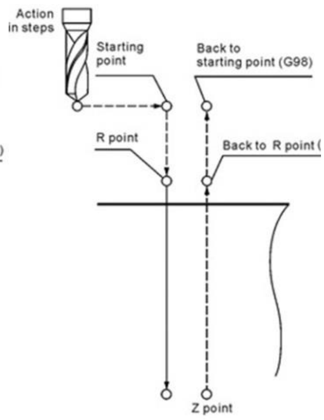
(1) Par326: bit2 - bit3 = 2

G87 X_ Z_ R_ Q_ P_ F_



(2) Par326: bit2 - bit3 = 1

G87 X_ Z_ R_ Q_ F_



(3) Par326: bit2 - bit3 = 0

G87 X_ Z_ F_

G88: Side tapping cycle

Format: G88 Z(W)_ C_ X(U)_ R_ Q_ P_ F_ K_L_M_;

Z(W)_ C_: tapping position.

X(U)_: bottom of the hole.

R_: the reference point set as an incremental value, which the tool moves toward the bottom of the hole.

Q_: depth of tapping for each cutting feed. When Q_ value is an integer, the unit is 0.001 mm.

P_: dwell time at the bottom of the hole in the unit of 0.001 ms (input integers only).

F_: feed rate.

K_: number of repetitions for machining, which is 1 by default.

L_: machining cycle mode selection; this parameter functions the same as parameter 326 (cycle setting). When there is an L argument, the system will first refer to L; otherwise, it refers to parameter 326.

M_: M-code call, which is usually used for clamping / unclamping control of the C axis. Action flow of M-code call: when the tool moves to the start point, the system executes the M-code (e.g., M80: C-axis clamping). Then, when the tool completes tapping and retracts to point R and before pausing at the top of the hole, the system executes M + 1 code (e.g., M81: C-axis unclamping).

Parameter 513: dwell time at the top of the hole. When the cycle command is completed, the tool pauses at point R for the duration (unit: second) set in parameter 513.

Description: the spindle first rotates one cycle to look for the Z pulse and perform positioning once the Z pulse is found. Next, the Z axis moves to the reference position R in rapid traverse. Then, the spindle rotates and carries on tapping to the bottom of the hole, reverses and retracts to point R, and finally moves to the start point in rapid traverse.

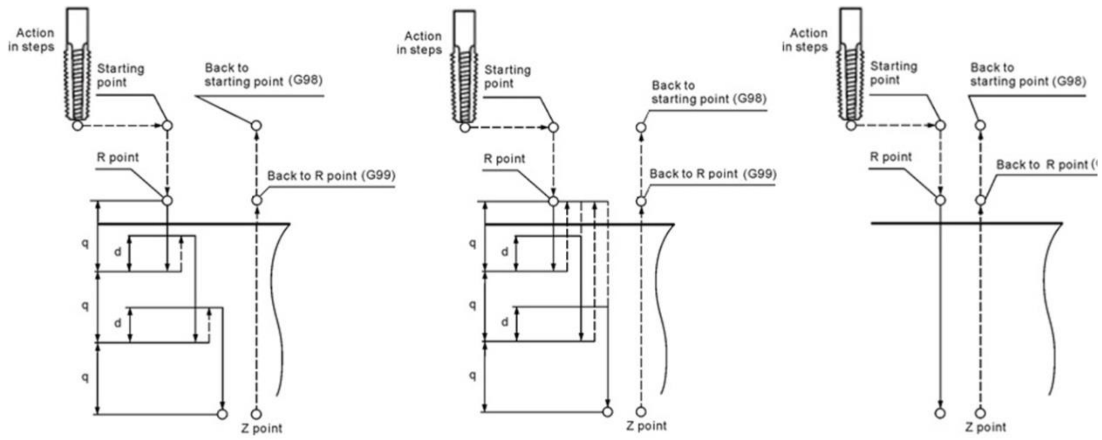
Cycle parameter 326:

If bit 2 - 3 are 0, the tool does not perform peck tapping; it taps directly to the bottom.

If bit 2 - 3 are 1, the tool performs deep peck tapping, which feeding amount is Q and retracts to point R.

If bit 2 - 3 are 2, the tool performs general peck tapping, which feeding amount is Q and the escape amount is set in parameter 324.

[Example] G84 / G88



(1) Par326: bit 2 - bit 3 = 2
G88 X_ Z_ R_ Q_ P_ F_

(2) Par326: bit 1 - bit 3 = 2
G88 X_ Z_ R_ Q_ F_

(3) Par326: bit 0 - bit 3 = 2
G88 X_ Z_ F_

2

G89: Side boring cycle

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

Z(W)_C_: boring position.

X(U)_: bottom of the hole.

R_: the reference point set as an incremental value, which is the distance from the start point to the bottom of the hole in Z-axis direction.

P_: dwell time at the bottom of the hole in the unit of ms (input integers only).

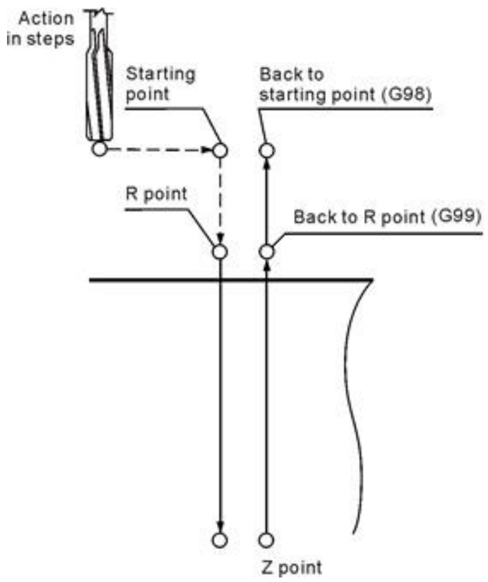
F_: feed rate.

K_: number of repetitions for machining, which is 1 by default.

M_: M-code call, which is usually used for clamping / unclamping control of the C-axis. Action flow of M-code call: when the tool moves to the start point, the system executes the M-code (e.g., M80: C-axis clamping). Then, when the tool completes boring and retracts to point R and before pausing at the top of the hole, the system executes M + 1 code (e.g., M81: C-axis unclamping).

Description: G89 is usually used for reaming or boring applications that require higher precision in hole radius of the workpiece to be machined. The tool first moves to point R in rapid traverse and cuts to the depth specified with the Z value at feeding speed F. Then, the tool retracts to point R at the same speed and finally returns to the start point using command G00.

[Example] G85 / G89



X_ Z_ (fast positioning)

G89 X_ Z_ R_ P_ F_

G90/G91: Absolute / incremental coordinates

Format: G90 X_ Y_ Z_

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

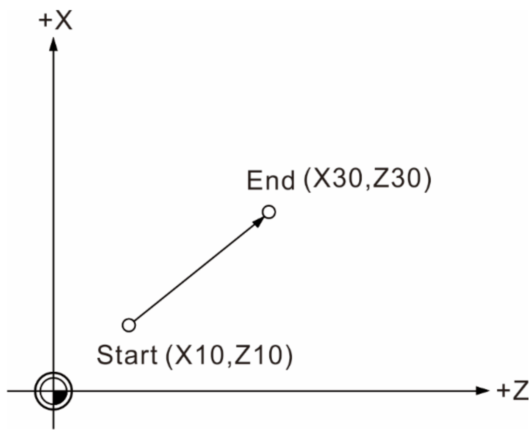
Format: G91 X_ Y_ Z_

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

Note:

1. The function of G90 for type A lathe system is “Axial turning cycle”.
2. G91 (incremental command) is not applicable for the type A lathe system; you need to use the format U_, V_, W_ to input the incremental value.

[Example] command type B



Absolute command: G90 X30. Z30.

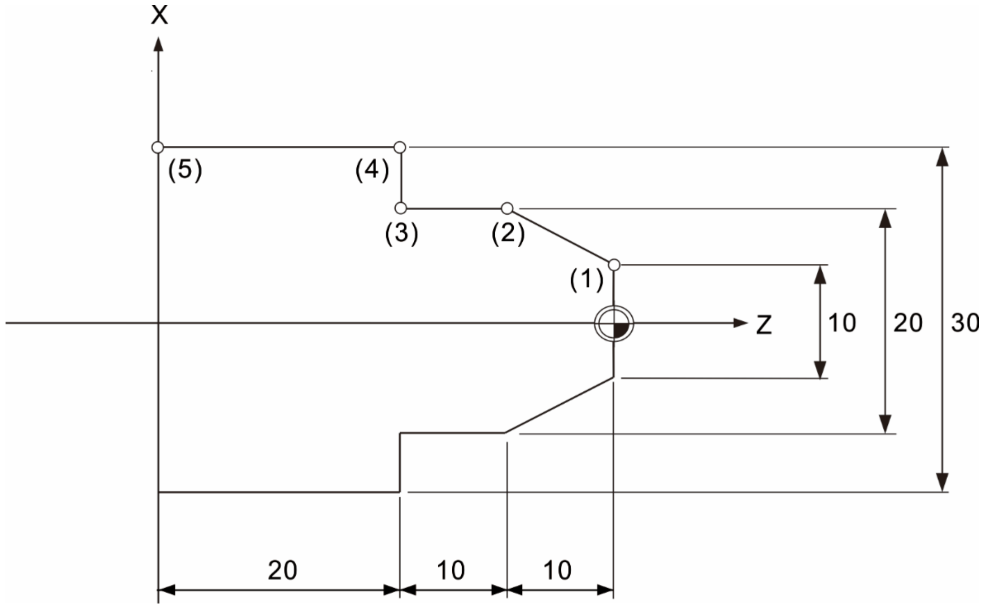
Incremental command: G91 X20. Z20.

G90 + incremental command: G90 U20. W20.

G91 + incremental command: G91 U20. W20.

2

[Example of using both G90 and G91]



G98;	Feed-per-minute mode
G00 G90 G54 X0 Y0;	Move to G54 (X0, Z0) in rapid traverse
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 position in rapid traverse

G50: Coordinate system setting / maximum spindle speed

Format: G50 X_ Y_ Z_ (coordinate system setting)

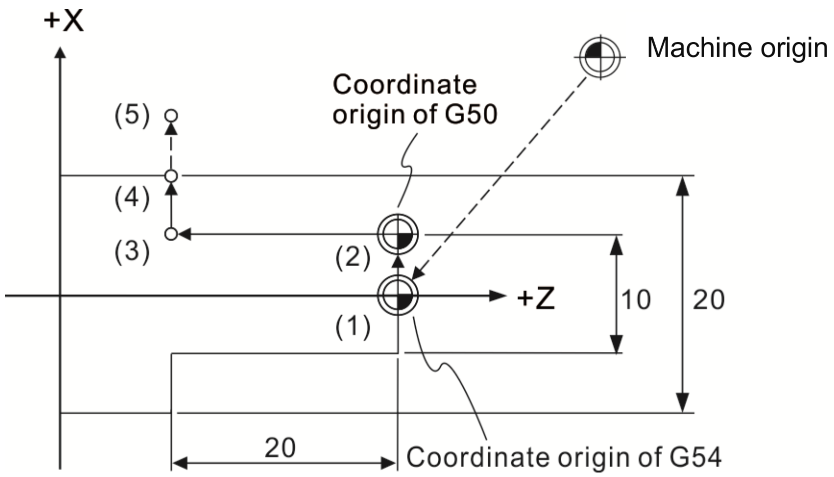
G50 S_ (maximum spindle speed)

Description: G50 X0 Y0 Z0 command can set the current tool position as the absolute coordinate system origin. The absolute commands in this program will refer to this origin when calculating the position. In G50 X_ Y_ Z_, if you have specified the value for X, Y, or Z, the absolute coordinates and the current position display will update according to the G50 command. G50 S_ sets the maximum spindle speed for retaining the precision and performance of the machine.

Note:

1. G50 coordinate system setting command continues to function unless M02/M30 is executed (program end command).
2. Pressing the **RESET** key can cancel the state set in the G50 coordinate system setting.

[Example]



M3 S1500;	Spindle rotates forward at 1,500 rpm
G0 G54 X0. Z0.;	Move to the G54 coordinate system origin (1) in rapid traverse
G0 X10.;	Move to point (2) in rapid traverse
G50 S1000;	The maximum speed limit is 1,000 rpm; the speed reduces to 1,000 rpm
G50 X0. Z0.;	The workpiece origin (2) specified in G50
G1 W-20. F0.25;	Move to point (3)
U10.;	Move to point (4)
G0 X30.;	Move to the safety point (5) in rapid traverse
M5 ;	Spindle stop
M30 ;	Program end; the absolute coordinate system specified in G92 transits to G54 coordinate system

G98: Feed per minute (mm/min)

Format: G98 G01 X_ Y_ Z_ F_

Description: the unit for G98 Feed per minute is mm/min. This means the tool performs turning each minute at speed F_. You can execute G98 command with motion blocks at the same time as well as using G98 independently in one block. This command is continuous effective.

G99: Feed per revolution (mm/rev)

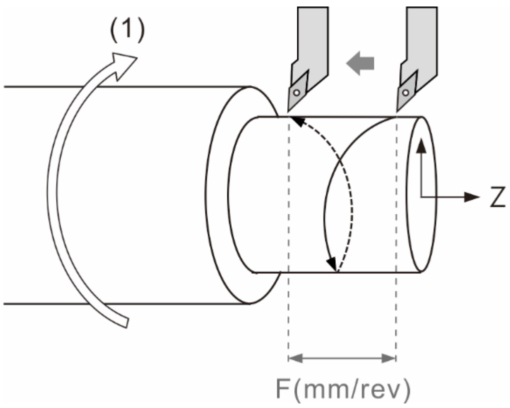
Format: G99 G01 X_ Y_ Z_ F_

Description: the unit for G99 Feed per revolution is mm/rev. This means the tool performs turning at speed F_ per spindle revolution. You can execute G99 command with motion blocks at the same time as well as using G99 independently in one block. This command is continuous effective and its format is set based on the tool feeding calculation for the lathe system.

[Example]

M3 S1000 (Spindle rotates forward at 1,000 rpm)

G99 G01 Z-20. F0.35 (Feed 0.35 mm per rotation)



(1) Spindle rotation direction

Actual turning feed speed: 1000 rev/min * 0.35 mm/rev = 350 mm/min

G96: Constant speed surface control

Format: G96 S_ (m/min)

Description: the S_value set in G96 is the cutting speed in the unit of m/min. If you execute G96 S200, the current tool cutting speed is 200 m/min. When the surface cutting speed is fixed, the spindle speed varies based on the command diameter coordinate (X-axis command); the spindle speed is faster when the machining diameter is smaller while the spindle speed is slower when the machining diameter is larger.

You can use this command to keep the consistency in workpiece surfacing.

G97: Cancel constant speed surface control

Format: G97 S_ (rev/min)

Description: the S_value set in G97 is the cutting speed in the unit of rev/min. If you execute G97 S2000, the spindle speed keeps at 2,000 rev/min.

You can use this command for the applications that require a fixed spindle speed when threading, drilling, and tapping.

Chamfer / corner rounding function

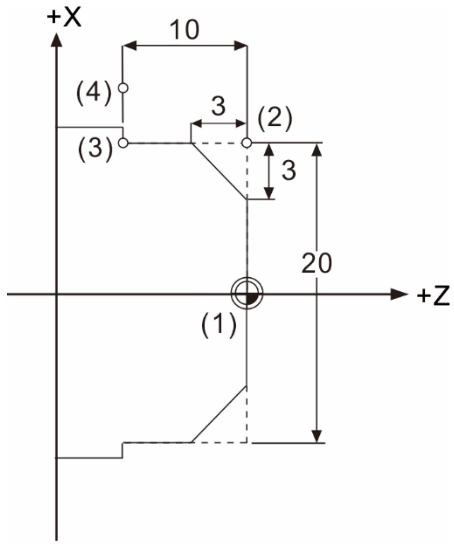
2

Format for chamfering: G01/G02/G03 X/U_ Z/W_ ,C_

G01/G02/G03 X/U_ Z/W_

Chamfer a length of C_ at the intersection between two motion blocks.

[Chamfer example 1]



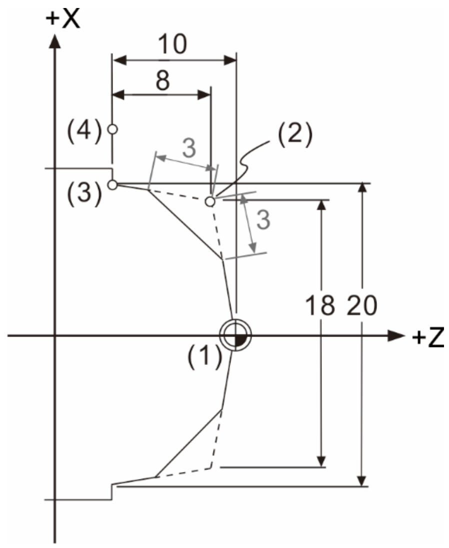
G1X0.0Z0.0F0.1 (1)

X20.,C3. (2) Hypothetical intersection

W-10. (3)

X30. (4)

[Chamfer example 2]



G1X0.0Z0.0F0.1 (1)

X18.W-2.,C3. (2) Hypothetical intersection

X20.W-8. (3)

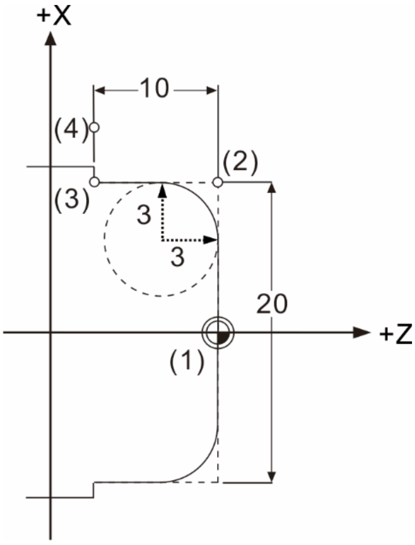
X30. (4)

Format for corner rounding: G01/G02/G03 X/U_ Z/W_ ,R_

G01/G02/G03 X/U_ Z/W_

Perform corner rounding with the set radius R_ at the intersection between two motion blocks.

[Corner rounding example]



G1X0.0Z0.0F0.1 (1)

X20.,R3. (2) Hypothetical intersection point

W-10. (3)

X30. (4)

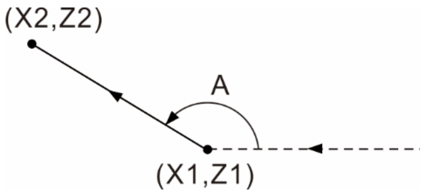
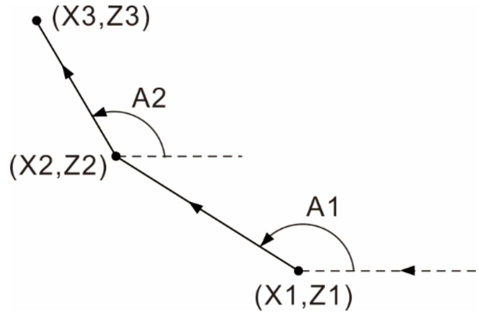
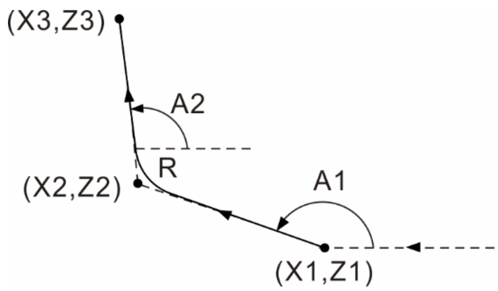
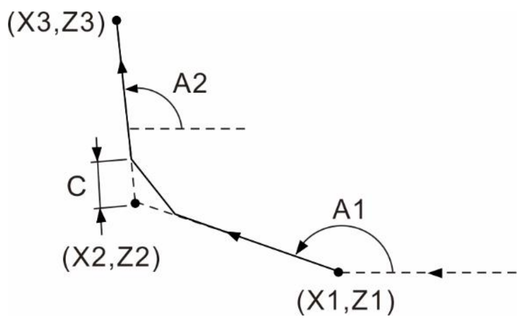
Note: chamfering and corner rounding commands must be followed by a motion block so the chamfering and corner rounding data can be calculated.

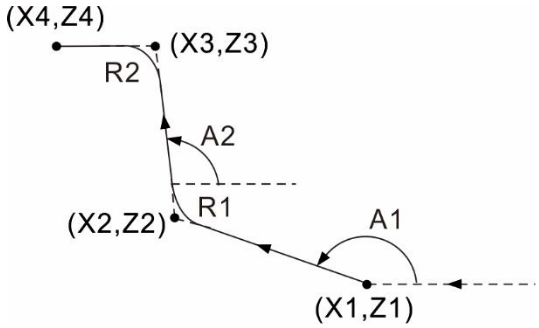
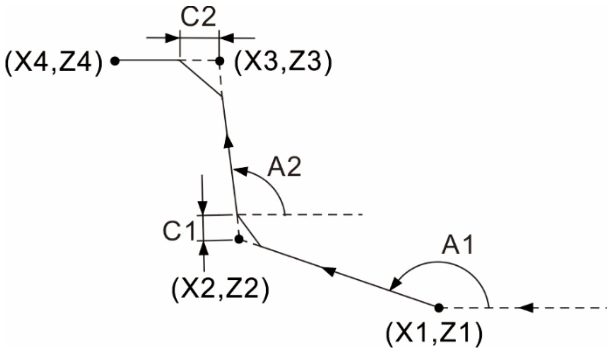
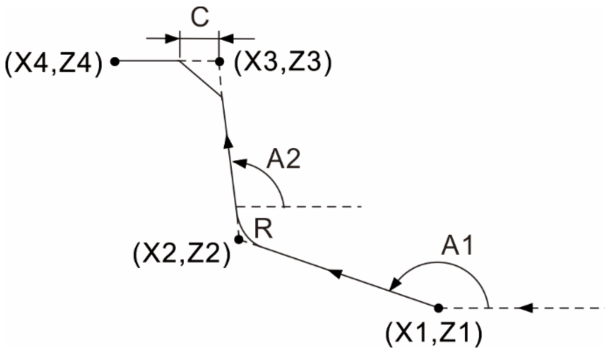
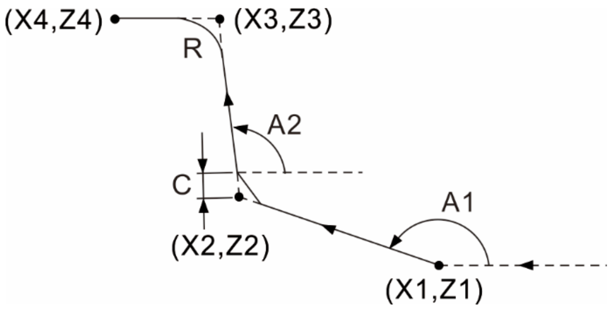
Linear angle command

Format: G01 X _ (Z_) ,A_

X_ (Z_) ,A_

Set 1 to 2 end coordinates and the degree data in two consecutive blocks, and the controller can calculate the corresponding coordinate data of the path, as shown in the table below.

Item	Command	Motion
1.	X X1 Z Z1 X X2 Z Z2,A A	
2	X X1 Z Z1 ,A A1 X X3 Z Z3,A A2	
3.	X X1 Z Z1 X X2 Z Z2,R R X X3 Z Z3 or use X X1 Z Z1 ,A A1,R R X X3 Z Z3,A A2	
4.	X X1 Z Z1 X X2 Z Z2,C C X X3 Z Z3 or use X X1 Z Z1 ,A A1,C C X X3 Z Z3,A A2	

Item	Command	Motion
5.	X X1 Z Z1 X X2 Z Z2,R R1 X X3 Z Z3,R R2; X X4 Z Z4 or use X X1 Z Z1 ,A A1,R R1 X X3 Z Z3,A A2,R R2 X X4 Z Z4	
6.	X X1 Z Z1 X X2 Z Z2,C C1 X X3 Z Z3,C C2; X X4 Z Z4 or use X X1 Z Z1 ,A A1,C C1 X X3 Z Z3,A A2,C C2 X X4 Z Z4	
7.	X X1 Z Z1 X X2 Z Z2,RR X X3 Z Z3,CC; X X4 Z Z4 or use X X1 Z Z1 ,A A1,RR X X3 Z Z3,A A2,CC X X4 Z Z4	
8.	X X1 Z Z1 X X2 Z Z2,CC X X3 Z Z3,RR X X4 Z Z4 or use X X1 Z Z1 ,A A1,CC X X3 Z Z3,A A2,RR X X4 Z Z4	

(This page is intentionally left blank.)

M-code Description 3

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

3

3.1 M-code Description

The M-code format is a capital M followed by numeric digits, which range is 0 - 65534.

The controller has some system-defined M-codes that do not require MLC coding to run and are usually used for program control. The table below is the commonly used M-codes. Except the system-defined codes, you need to use the MLC to specify the M-code functions.

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

M00: Program stop (non-optional)

3

Format: M00

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

M01: Program stop (optional)

Format: M01

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

M02: End of program

Format: M02

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

M30: End of program with return to program start position

Format: M30

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

M98: Subprogram call

Format: M98 P_ L_

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

P_: subprogram code; L_: number of call for the subprogram

[Example]

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

M99: Return from subprogram

Format: M99

Description: after M98 is used to run the subprogram, to return to the main program, you need to use M99 to have the cursor return to the block following the block that has called the subprogram and then carry on the execution in the main program.

[Example]

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

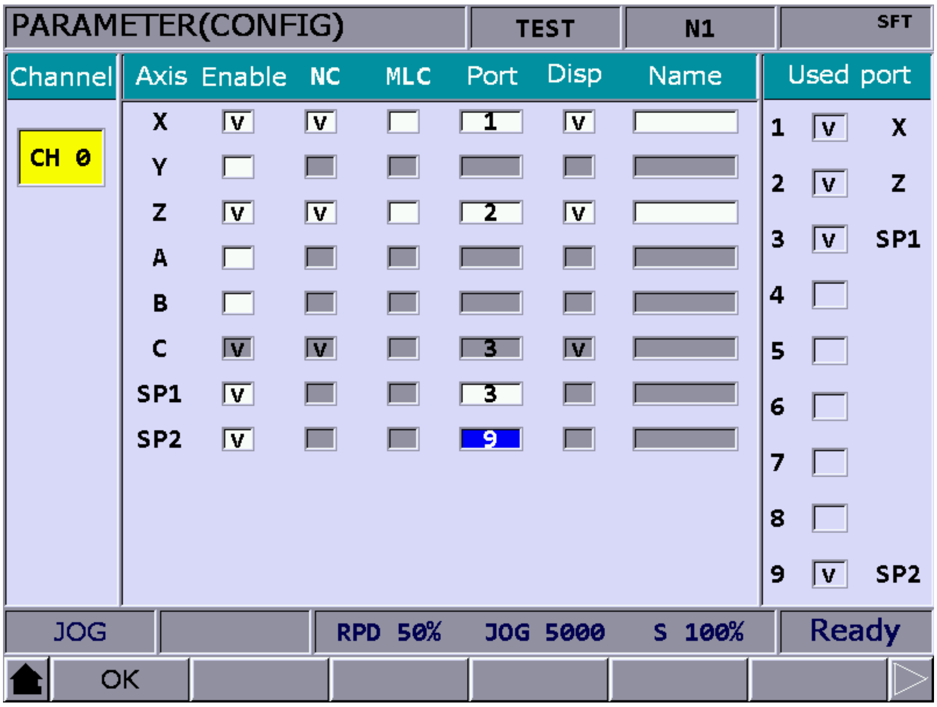
3.2 Spindle and C-axis switching

3.2.1 Description for Spindle and C-axis switching

Description: the function of switching between the Spindle and C-axis modes are used in the lathe / milling machining applications. Some functions are operable only in C-axis mode, such as face and side drilling / tapping, polar coordinate interpolation, and cylindrical coordinate interpolation.

Parameter settings:

1. C-axis mode setting: go to Parameter (CONFIG) > parameter 308 Channel auxiliary settings > set 0 (select C-axis mode) for lathe system.
2. To switch from Spindle to C-axis mode, you need to set the corresponding M-code setting for parameter 358.
3. To switch from C-axis to Spindle mode, you need to set the corresponding M-code setting for parameter 359.
4. The channel setting for the feed axis and spindle is shown in the figure below.



M-codes for switching between C-axis and Spindle modes:

1. MLC ▶ NC (The MLC writes the values to the register and the NC reads the values in the register.)

M-code	Function	Status
M1126	C-axis mode	On
	Spindle mode	Off

Function description:

- When the G-code program runs the M-code for switching from the Spindle mode to C-axis mode (M-code setting in parameter 358), the MLC sets M1126 to on to notify the NC to switch from Spindle mode to C-axis mode.
- When the G-code program runs the M-code for switching from the C-axis mode to Spindle mode (M-code setting in parameter 359), the MLC sets M1126 to off to notify the NC to switch from C-axis mode to Spindle mode.
- Spindle mode is the default when booting (M1126 = off).

2. NC ▶ MLC (The NC writes the values to the register and the MLC reads the values in the register.)

M-code	Function	Status
M2239	The NC system completes mode switching and is now in C-axis mode.	On
	The NC system completes mode switching and is now in Spindle mode.	Off

Function description:

- When the NC is in Spindle mode and it executes the M-code (switch from Spindle to C-axis mode) and receives the On signal of M1126, the NC starts switching the mode from Spindle to C-axis. Once the switching is complete, M2239 is turned on and the NC is in C-axis mode.
- When the NC is in C-axis mode and it executes the M-code (switch from C-axis to Spindle mode) and receives the Off signal of M1126, the NC starts switching the mode from C-axis to Spindle. Once the switching is complete, M2239 is turned off and the NC is in Spindle mode.
- Spindle mode is the default when booting (M2239 = off).

[Example]

The parameter setting is as follows.

Parameter No.	Description	Value
358	The M-code for switching from the Spindle to C-axis mode in the lathe system.	66
359	The M-code for switching from the C-axis to Spindle mode in the lathe system.	77

G-code example:

```
M03 S1000
G00 X50. Z10.
G98 F1000
G01 Z-20.
M66     (Switch from Spindle mode to C-axis mode)
C0.
C90.
C180.
M77     (Switch from C-axis mode to Spindle mode)
M30
```

3.2.2 Notes for Spindle and C-axis mode switching

1. You can use M1126 to switch between C-axis and Spindle modes in all operation modes except the Auto mode which you must use the M-codes to switch.
2. When switching from the Spindle to C-axis mode, if the first spindle has not passed through the Z pulse, the system automatically sends M29 the positioning command. If the first spindle is running, then the system sends M05 the spindle stop command.
3. In all operation modes, the C axis has to be static when the system switches to the Spindle mode. If the C axis is operating and M1126 is on, the mode switching starts only after the C axis stops.
4. In all operation modes, when the system switches from C-axis to Spindle mode and the second spindle (dynamic tool turret) is operating, the system sends M05 the spindle stop command to the second spindle and then starts the mode switching.

(This page is intentionally left blank.)

Macro and Variable 4

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

4.1 Variables

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

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

4.1.1 Arguments and local variables

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

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

4.1.2 Systems variables

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

■ G-code group data

No.	Function	Read	Write
#2000 - #2019	G-code group	★	
#2020	F-code, the NC machining speed F	★	
#2023	T-code, the NC tool number T	★	
#2024	S-code, the NC spindle speed S	★	

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

No.	Function
#2000	G04, G09, G10, G11
#2001	Interpolation mode: G00, G01, G02, and G03
#2002	Plane selection: G17, G18, and G19
#2003	Absolute / increment designation: G90 and G91
#2005	Feed rate setting: G94 and G95
#2006	Inch / metric setting: G21 and G20
#2007	Tool nose radius compensation: G40, G41, and G42
#2009	Cycle: G70, G71, G72, G73, G74, G75, G76, G77, G78, G79, G80, G83, G84, G85, G86, G87, G88, and G89
#2010	Homing position: G98 and G99
#2012	Workpiece coordinates: G54, G55, G56, G57, G58, and G59
#2013	Cutting mode: G61 and G64
#2014	Macro call: G66 and G67
#2017	Surface cutting speed setting: G96 and G97

- Position related data

You can use variables #2100 - #2217 to read the following coordinates (read-only).

Position data / Axis	Machine coordinates	Relative coordinates	Machine coordinates when G31 skip command is triggered	Absolute coordinates of breakpoint search line
	Absolute coordinates	Block end coordinates	Absolute coordinates when G31 skip command is triggered	Offset between machine coordinates of breakpoint search line and current machine coordinates
X axis	#2100	#2180	#2148	#2196
	#2116	#2132	#2164	#2212
Y axis	#2101	#2181	#2149	#2197
	#2117	#2133	#2165	#2213
Z axis	#2102	#2182	#2150	#2198
	#2118	#2134	#2166	#2214
A axis	#2103	#2183	#2151	#2199
	#2119	#2135	#2167	#2215
B axis	#2104	#2184	#2152	#2200
	#2120	#2136	#2168	#2216
C axis	#2105	#2185	#2153	#2201
	#2121	#2137	#2169	#2217

- Workpiece coordinate data

  You can use variables #3000 - #3646 to read the offset coordinates and workpiece coordinates (read-only).

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

- Others

No.	Function	Read	Write
#2300	The variable corresponding to single block I when an arc command is executed.	★	
#2301	The variable corresponding to single block J when an arc command is executed.	★	
#2302	The variable corresponding to single block K when an arc command is executed.	★	
#2303	Timer starts after system power on	★	
#2304	Tool number of Spindle 1 (dual tool magazine)	★	
#2305	Tool number of Spindle 2 (dual tool magazine)	★	
#5000 - #5013	Breakpoint search function: recently used M-codes (14 sets), #5000 - #5013 from the latest to the oldest	★	
#5014 - #5015	Breakpoint search function: recently used T-codes (2 sets), #5014 - #5015 from the latest to the oldest	★	
#5016	Breakpoint search function: the last used S-code	★	
#6000	System macro alarm with the value range of 1 - 1000. The messages are edited with the Screen Editor.		★
#6001 - #6064	X-axis tool length	★	★
#6201 - #6264	Y-axis tool length	★	★
#6401 - #6464	Z-axis tool length	★	★
#6601 - #6664	X-axis tool wear	★	★
#6801 - #6864	Y-axis tool wear	★	★
#7001 - #7064	Z-axis tool wear	★	★
#7201 - #7264	Tool nose radius	★	★

No.	Function	Read	Write
#7401 - #7464	Tool nose wear	★	★
#7601 - #7664	Tool nose type	★	★
#8600	Program timer in the unit of ms	★	★

4.1.3 Macro interface input / output

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

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

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

■ MLC word output; read the MLC signal status (MLC > NC)

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

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

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

■ MLC word input; write the MLC signal status (NC > MLC)

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

4.2 Variable syntax

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

(1) The allowable range for the local variables:
    # i: the ith variable (when $1 \leqq i \leqq 50$)

(2) Define the variable number with formula:
    When calculating #[A], the range of A must be $1 \leqq A \leqq$ System's maximum variable number. The value of A must be a positive value and no smaller than 0.

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

(3) Place a symbol before a variable

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

(4) Using variable for definition

#20 = 10 (#20 equals 10)

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

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

(5) Conditional expression

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

4.3 Operation commands

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

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

4.4 Control flow

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

```
WHILE [statement]
 ⁞
ENDW
```

Example:

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

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

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

- Branch conditions

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

`IF [Statement]  GOTO N`	(Execute the IF statement, conditionally jump to statement N for execution.)
`GOTO N`	(When GOTO N is used independently, unconditionally jump to statement N.)

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

```
      ∫
N10 #12=#10
#13=#11+2;
IF[#2=1]GOTO200;
#12=#10-#3;
#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 searches from the start of the program. An alarm will occur if the target number is not found. If the same sequence number repeats in the program, the system will execute the block of code that is first found.

- Types of conditional statements:

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

Example:

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

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

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

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

Macro function	G-code number	Note
O9010	0 - 1000	Set the G-code number to 0 if not using the macro call function.
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	

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

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

Macro function	M-code number	Note
O9020	0 - 1000	Set the M-code number to 0 if not using the macro call function.
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	

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

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

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

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

The variable definition is as follows.

No.	Description	Read	Write
#1 - #50	Local variables	★	★
#51 - #250	Global variables	★	★
#1601 - #1800	Maintain variables (non-volatile)	★	★
#10001 - #10450	Extension variables (non-volatile)	★	★
#1801 - #1832	MLC logic output points. MLC > NC macro input points: M1024 - M1055 (32 points in total)	★	
#1833 - #1848	MLC data output points. MLC > NC macro input points: D1024 - D1039 (16 points in total)	★	
#1864 - #1895	MLC logic input points. NC > MLC macro output points: M2080 - M2111 (32 points in total)		★
#1896 - #1911	MLC data input points. NC > MLC macro output points: D1336 - D1351 (16 points in total)		★
#2000 - #2019	G-code group	★	
#2020	F-code	★	
#2023	T-code	★	
#2024	S-code	★	
#2100 - #2105	Machine coordinates of X - C axes	★	
#2116 - #2121	Absolute coordinates of X - C axes	★	
#2132 - #2137	End coordinates of X - C axes in single block	★	
#2148 - #2153	Machine coordinates of X - C axes when G31 skip command is triggered	★	
#2164 - #2169	Absolute coordinates of X - C axes when G31 skip command is triggered	★	
#2180 - #2185	Relative coordinates of X - C axes	★	
#2196 - #2201	Absolute coordinates of breakpoint search line (X – C axes)	★	
#2212 - #2217	Offset between machine coordinates of breakpoint search line and current machine coordinates (X – C axes)	★	
#2300	Single block I (for arc command)	★	
#2301	Single block J (for arc command)	★	
#2302	Single block K (for arc command)	★	
#2303	Timer starts after system power on	★	
#2304	Tool number of Spindle 1 (dual tool magazine)	★	
#2305	Tool number of Spindle 2 (dual tool magazine)	★	
#2500	Write the tool number for tool magazine 1		★
#2501	Write the tool number for tool magazine 2		★
#3000	X-axis offset coordinate	★	

No.	Description	Read	Write
#3001 - #3006	X-axis workpiece coordinate G54 - G59	★	
#3128	Y-axis offset coordinate	★	
#3129 - #3134	Y-axis workpiece coordinate G54 - G59	★	
#3256	Z-axis offset coordinate	★	
#3257 - #3262	Z-axis workpiece coordinate G54 - G59	★	
#3384	A-axis offset coordinate	★	
#3385 - #3390	A-axis workpiece coordinate G54 - G59	★	
#3512	B-axis offset coordinate	★	
#3513 - #3518	B-axis workpiece coordinate G54 - G59	★	
#3640	C-axis offset coordinate	★	
#3641 - #3646	C-axis workpiece coordinate G54 - G59	★	
#5000 - #5013	Breakpoint search function: recently used M-codes (14 sets), #5000 - #5013 from the latest to the oldest	★	
#5014 and #5015	Breakpoint search function: recently used T-codes (2 sets), #5014 - #5015 from the latest to the oldest	★	
#5016	Breakpoint search function: the last used S-code	★	
#6000	System macro alarm		★
#6001 - #6064	X-axis tool length	★	★
#6201 - #6264	Y-axis tool length	★	★
#6401 - #6464	Z-axis tool length	★	★
#6601 - #6664	X-axis tool wear	★	★
#6801 - #6864	Y-axis tool wear	★	★
#7001 - #7064	Z-axis tool wear	★	★
#7201 - #7264	Tool nose radius	★	★
#7401 - #7464	Tool nose wear	★	★
#7601 - #7664	Tool nose type	★	★
#8600	Program timer in the unit of ms	★	★

Revision History

Release date	Version	Chapter	Revision contents
December, 2018	V1.0 (First edition)		

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

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

(This page is intentionally left blank.)