



Mill Machine Program Manual.

匯出日期：2022-01-03

修改日期：2020-07-31

1 G Code Command Table.

● 中文版 Mandarin Version: G码指令一览表

Name definition

Description	Definition	Example
Modal G code	<p>The following two conditions can be considered as modal G codes:</p> <ol style="list-style-type: none"> 1. The G code function continues to be valid until the G code is turned off, and the axial index of the subsequent block is affected by this G code command. 2. The G code function continues to be valid before executing other G codes of the same group. The axial parameter of the subsequent block is affected by this G code command. 	<p>1. e.g. G43, G44, G49 After executing G43 and G44, if G49 is not used, the tool length compensation will continue to be effective.</p> <p>2. e.g. G00、G01</p> <p>Program Example</p> <pre>G00 X0. // execute G00 Y0. // execute G00 G01 X100. // execute G01 Y100. // execute G01</pre>
Not modal G code	<p>The G code command is valid only in a single block, and the axial argument is only affected by the block in the G code command.</p>	<p>e.g. G04</p> <p>Program Example</p> <pre>G01 X1. // execute G01 G04 X3. // execute G04 X5. // execute G01</pre>

Item	Function Name	Mod al	Group	Switch Command	Release Command	Note
G00	Linear positioning	O	Interpolation Mode	Execute other G code in the same group.	NA	
G01	Linear interpolation, cutting feed	O				
G02	Arc interpolation (clockwise)	O				
G02.4	Three points arc interpolation (clockwise)	O				
G03	Arc interpolation (Counter-clockwise)	O				

G03.4	Three points arc interpolation (Counter-clockwise)	O				
G04	Pause specified time	X		NA	NA	
G04.1	Path synchronization waiting	X				
G05	High speed and precision mode	O		NA	G05 P0	
G05.1	Path smoothing mode	O		NA	G05.1 Q0	
G06.2	NURBS Curve interpolation	O		NA	G05 P0	
G09	Stop testing	X		NA	NA	
G10	Programmable data input	X				
G10.9	Straight and radius axis programming switch	X				
G12.1	Polar Interpolation	O	Polar Interpolation Mode	NA	G13.1	
G15	Cancel polar coordinate command	O	Polar Command Mode	NA	G15	
G16	polar coordinate command	O				
G17	Set X-Y working plane	O	Working Plane Mode	Execute other G code in the same group.	NA	
G18	Set Z-X working plane	O				
G19	Set Y-Z working plane	O				
G22	Second software stroke limit	O	Stroke Check Mode	NA	G23	
G23	Cancel second software stroke limit	O				

G28	Reference point returning	X		NA	NA	
G29	Return from reference point	X				
G30	Any reference point returning	X				
G31	Skip command	X				
G31.10 / G31.11	Multi-Axis Multi-Signal Skip Function	X				
G33	Screw cutting	O	Interpolation Mode	Execute other G code in the same group.	NA	
G37	Automatic tool setting I	X		NA	NA	
G37.2	Automatic tool setting II	X				
G37.3	Automatic tool setting III	X				
G40	Cancel cutter radius compensation mode	O	Cutter Compensation Mode	NA	G40	
G41	Cutter radius left compensation	O				
G42	Cutter radius right compensation	O				
G43	Cutter length front compensation	O	Cutter Compensation Mode	NA	G49	
G43.4	Cutter point control	O				five axis
G44	Cutter length negative compensation	O				

G49	Cancel cutter length compensation	O				
G45	Tool offset (1x cutter radius in positive direction)	X		NA	NA	
G46	Tool offset (1x cutter radius in negative direction)	X				
G47	Tool offset (2x cutter radius in positive direction)	X				
G48	Tool offset (2x cutter radius in negative direction)	X				
G50	Cancel scaling mode	O	Scaling Mode	NA	G50	
G51	Scaling mode	O				
G50.1	Cancel mirror function	O		NA	G50.1	
G51.1	Mirror function	O				
G52	Local coordinate system setting	O		NA	G52 X0.0 Y0.0 Z0.0	
G52.1	Axis removal	X	Axis removal/Axis borrowing function	Execute other G code in the same group.	G52.1	
G52.2	Axis borrowing	X				
G53	Mechanical coordinate orientation	X		NA	NA	
G53.1	Tilted working plane machining tool alignment	X				five axis
G54	Working coordinate system setting	O		NA	Pr3229	
G59	Working coordinate system setting	O				

G61	Stop testing	O	Cutting Feed Control Mode	Execute other G code in the same group.	NA	
G64	Cutting mode	O				※
G65	Single macro program call	X		NA	NA	※
G66	Mode macro program call	O		NA	G67	※
G67	Cancel mode macro program cal	O				
G68	Start coordinate rotation	O	Coordinate Rotation Mode	NA	G69	
G68.2	Tilted working plane machining	O				five axis
G69	Cancel coordinate rotation	O				
G70	Inch unit processing	O	Input Dimension Mod)	Execute other G code in the same group.	NA	
G71	Metric unit processing	O				
G73	High speed jaw drilling cycle	O		Interpolation Mode	G80	
G74	Left hand tapping cycle	O				
G76	Fine pupillary cycle	O				
G80	Cancel cycle	O				
G81	Drilling mode	O		Interpolation Mode	G80	
G82	Pause drilling cycle at the bottom of the hole	O				
G83	Peck drilling cycle	O				
G84	Tapping cycle	O				
G84.48	Tapping Retract	O				
G85	Drilling cycle	O				

G86	high-speed drilling cycle	O				
G87	Fine boring cycle on the back	O				
G88	Semi-automatic fine boring cycle	O				
G89	Pause boring cycle at the bottom of hole	O				
G90	Absolute position input method	O	Input Command Mode	Execute other G code in the same group.	NA	
G91	Relative position input method	O				
G92	Absolute zero point coordinate system setting	O	Coordinate System Setting Mode	Execute other G code in the same group.	NA	
G92.1	Absolute zero point coordinate system preset	O				
G93	Anti-time feed	O	Feed Mode	Execute other G code in the same group.	NA	
G94	Feed per minute (mm/min.)	O				
G95	Feed per revolution (mm/rev.)	O				
G96	Equal surface cutting speed	O	Spindle Speed Mode	NA	G97	
G97	Cancel equal surface cutting speed	O				
G98	Return to initial point	O		Execute other G code in the same group.	NA	
G99	Return to R point	O				
G120.1	Multiple sets of processing conditions	O		NA	G121	
G134	Circumferential hole circulation	X		NA	NA	

G135	Angle straight hole cycle	X				
G136	Arc hole cycle	X				
G137.1	Checkerboard cycle	X				

※SYNTEC 900M G code adopts the international RS274D specification, and the only difference from FANUC 0M specification is G70, G71 (public, imperial) which is relative to G20, G21.



SYNTEC

2 G Code Command Description

2.1 G00 : Rapid Linear Positioning

2.1.1 Command Form

G00 [P1] X_ Y_ Z_ [F1=_];

X_ Y_ Z_: specified position.

P1: active constant speed command

F1: feedrate mm/min or inch/min

2.1.2 Description

Each axles move to appointed point without interpolation, X_ Y_ Z are the coordinate of end position, depends on G90/G91 to move absolute or increment value.

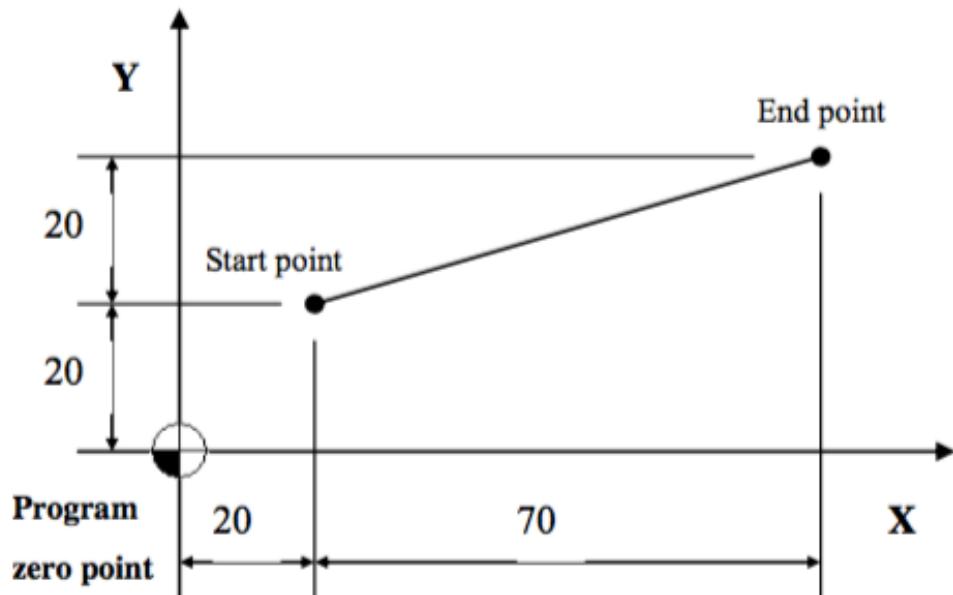
2.1.3 Notice

1. The movement mode can decide by parameter #411 (0: linear, 1: each axle move in max speed independently)
2. Constant feedrate is legally with both P1 and F1 argument are determined. If P1 argument is missing, system wouldn't refer F1 argument as moving speed. Moving speed is refer to Pr461~Pr480.
3. Pr411=0, Resultant speed is limited by F1 and Pr461~Pr480 if executing with P1, F1. Each axis speed is limited by Pr461~Pr480.
4. R18 Rapid Traverse Override is still valid with P1 F1=_.
The actually highest speed wouldn't exceed Pr461~Pr480 or F1, while the override is 0~100%.
The actually highest speed wouldn't exceed Pr461~Pr480 over five times, while the override is F0(PR3207=2, R18=1).
5. G00 command doesn't support G10 L1100 P1002 R_.
6. G70/G71 supported unit of F1 can be mm/min or inch/min.
7. Unit of F1 can be mm/min, but always remains G94. G93/G95 is invalid.
8. F1 argument support version: 10.118.12 and after.

2.1.4 Example

Example1:

PIC:



Program description:

1. first way(absolute): G90 G00 X90.0 Y40.0;
//use difference value between appointed point and zero point to do straight interpolation to appointed point.
2. second way(increment): G91 G00 X70.0 Y20.0;
// use difference value between appointed point and initial point to do straight interpolation to appointed point.

Example2:

Pr411 = 0、 Pr461 = Pr462 = 10000

```
G71
G0 P1 X500. Y500. F1=12000.
G1 X0. Y0.
M30
```

Both axis movement are identical, the moving speed should be $\sqrt{10000^2 + 10000^2} = 14142.1$ without F1=_. Hence, G0 P1 F1=12000 is applied, the limit is 12000.



2.2 G01 : Linear Interpolation, Cutting Feed

2.2.1 Command Form

G01 X_Y_Z_ F__ ;

X、Y、Z: Specified position

F : Feedrate

In G94 mode, unit is mm/min(inch/min) <- Default of Mill
In G95 mode, unit is mm/rev(inch/rev) <- Default of Lathe

2.2.2 Description

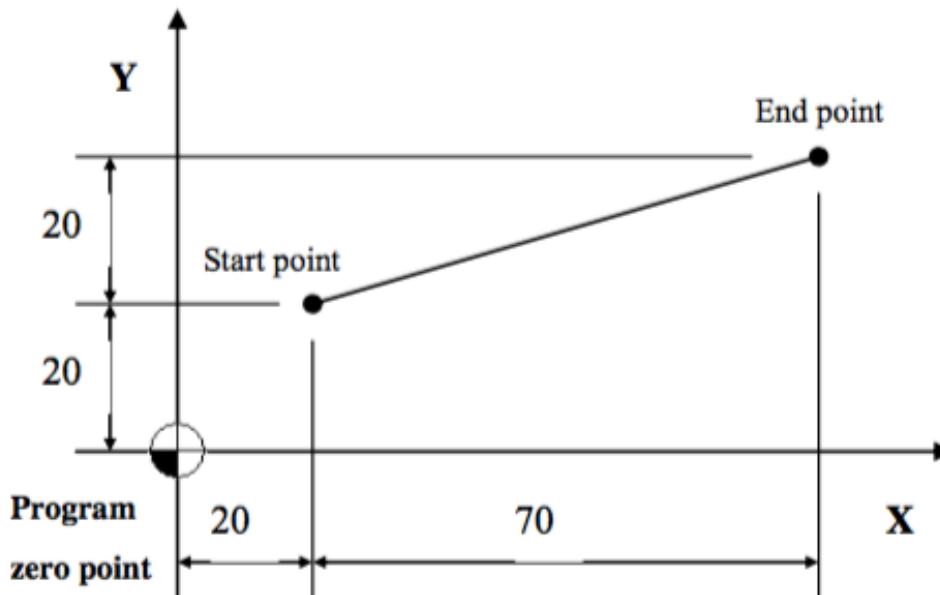
G01 executes linear interpolation, it can be used G90/G91 to decide absolute or increment mode, use feedrate defined by F to move to the specified position.

2.2.3 Notice

- The max. feedrate of G01 is defined by Pr405-maximum cutting feed rate or (PR621~PR640)-each axis maximum cutting feedrate.
- Default value of F: 1000mm/min(inch/min) for G94 mode, and 1 mm/rev(inch/rev) for G95 mode.
- Default mode G94/G95 can be set by parameter Pr3836 (reboot controller to activate setting).

2.2.4 Example 1

PIC:



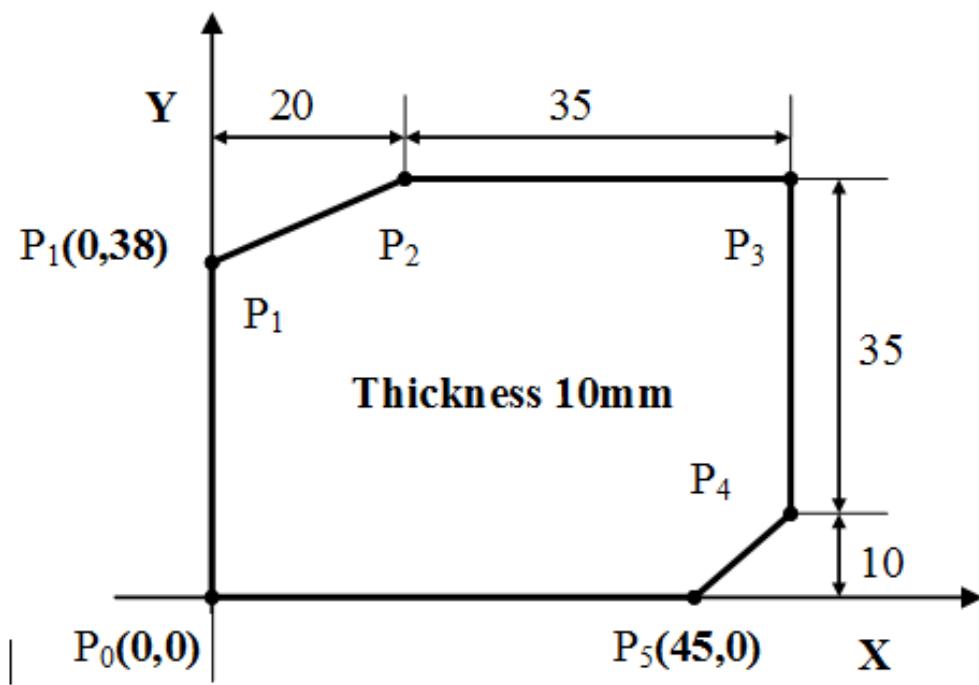
Program description:

1. Method 1 (Absolute value): G90 G01 X90.0 Y40.0 ;
//Move to the specified point with the program zero point as the relative coordinate
2. Method 2 (Increment value): G91 G01 X70.0 Y20.0 ;
// Move specified distance by the difference between the specified point and the starting point

2.2.5 Example 2

PIC:

SYNTec



Program description:

1. absolute way:

```
N001 G00 X0.0 Y0.0 Z10.0;//positioning to above of P0
N002 G90 G01 Z-10.0 F1000;
//straight interpolation to bottom of workpiece, feedrate is 1000mm/min
N003Y38.0;//P0 -> P1
N004X20.0 Y45.0;//P1-> P2
N005X55.0;//P2 -> P3
N006Y10.0;//P3 -> P4
N007X45.0 Y0.0;//P4 -> P5
N008X0.0;//P5 -> P0
N009 G00 Z10.0;//positioning back to above of P0
N010 M30;//program ends
```

2. increment way:

```
N001 G00 X0.0 Y0.0 Z10.0;//positioning to above of P0
N002 G91 G01 Z-20.0 F1000;
//straight interpolation to bottom of workpiece, feed rate is 1000mm/min
N003Y38.0;//P0 -> P1
N004X20.0 Y7.0;//P1-> P2
N005X35.0;//P2 -> P3
N006Y-35.0;//P3 -> P4
N007X-10.0 Y-10.0;//P4 -> P5
N008X-45.0;//P5 -> P0
N009 G00 Z20.0;//positioning back to above of P0
N011 M30;//program ends
```

2.3 G02/G03 : CW/CCW Circular Interpolation

2.3.1 Command Form

1. X-Y plane circular interpolation:

$$G17 \left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} X_ Y_ \left\{ \begin{array}{l} R_ \\ I_ J_ \end{array} \right\} F_ ;$$

2. Z-X plane circular interpolation:

$$G18 \left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} X_ Z_ \left\{ \begin{array}{l} R_ \\ I_ K_ \end{array} \right\} F_ ;$$

3. Y-Z plane circular interpolation:

$$G19 \left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} Y_ Z_ \left\{ \begin{array}{l} R_ \\ J_ K_ \end{array} \right\} F_ ;$$

X, Y, Z: End point position

I, J, K: the vector component of starting point of arc to the circle center.(circle center coordinate minus starting point)

R: Radius of arc

F: Feedrate

G90/G91 define absolute or increment mode

Description

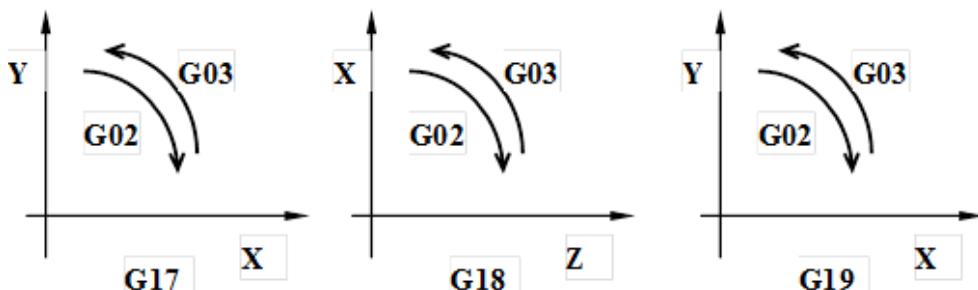
G02、G03 do circular interpolation according to appointed plane、coordinate system、size of arc and speed of interpolation, and the rotate direction decide by G02(CW)、G03(CCW). Five conditions of a circular interpolation list as below:

Setting Data		Command	Definition
1	Plane selection	G17	X-Y plane setting
		G18	X-Z plane setting
		G19	Y-Z plane setting
2	Tool Direction	G02	Clockwise direction (CW)

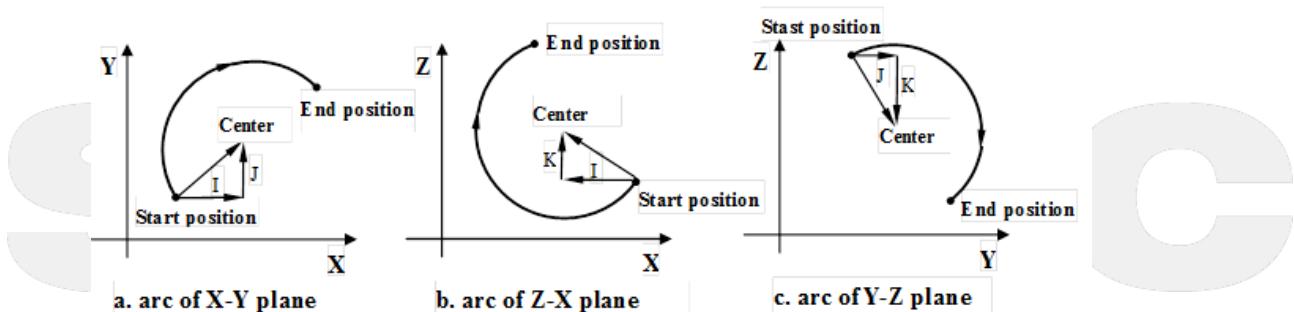
			G03	Counterclockwise direction (CCW)
3	End position	G90	Two axes of X, Y, Z	End coordinate of arc
		G91	Two axes of X, Y, Z	Vector value from start point to end point
4	Distance between start point and center of circle		Two axes of I, J, K	Vector value from start point of arc to center of circle
	Radius of arc		R	Radius of arc
5	Speed of feed (feedrate)	F		Feedrate along the arc

2.3.2 Notice

1. G02, G03 direction



2. I, J, K definition:



3. how to use R:

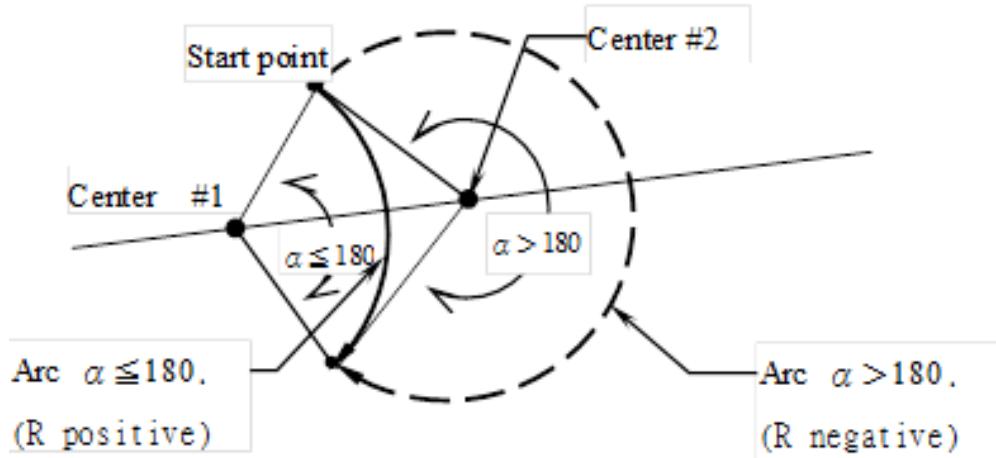
a. When $\theta \leq 180$ degree, R sets positive value.

$$\left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} X_ Y_ R25.0;$$

- b. When $180 \text{ degree} < \theta < 360 \text{ degree}$, R sets negative value.

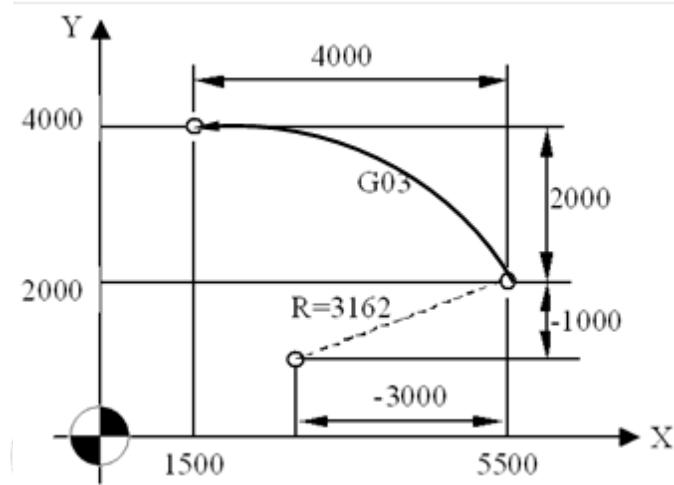
$$\left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} X_- Y_- R-25.0;$$

- c. When $\theta=360 \text{ degree}$, only use I, J, K.



2.3.3 Example

Example Program 1

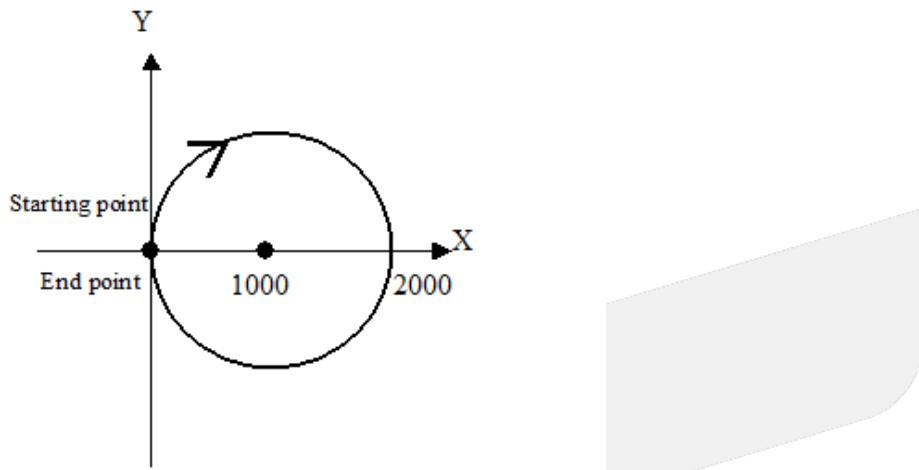


G90 G00 X5500 Y2000;//positioning to start point of arc

G17 G90 G03 X1500 Y4000 I-3000 J-1000 F200;//absolute command

(G17 G91 G03 X-4000 Y2000 I-3000 J-1000 F200;//increment command)

Example Program 2 (Full circular interpolation)



G90 G00 X0 Y0;
G02 I1000 F100; //interpolate a full circle

2.4 G02/G03: Helical Interpolation

2.4.1 Command Form

1.

$G17 \left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} X_- Y_- \left\{ \begin{array}{l} R_- \\ I_- J_- \end{array} \right\} Z_- F_- ;$

X, Y : end position of arc;
Z : end position of straight line;
R : radius of arc;
I, J : vector from the starting point to the center of the circle;
F : feedrate;

2.

$G18 \left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} X_- Z_- \left\{ \begin{array}{l} R_- \\ I_- K_- \end{array} \right\} Y_- F_- ;$

X, Z : end position of arc;
Y : end position of straight line;
R : radius of arc;
I, K : vector from the starting point to the center of the circle
F : feedrate;

3.

$G19 \left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} Y_- Z_- \left\{ \begin{array}{l} R_- \\ J_- K_- \end{array} \right\} X_- F_- ;$

Y, Z : end position of arc;

X : end position of straight line;
 R : radius of arc;
 J, K : vector from the starting point to the center of the circle;
 F : feed rate;

2.4.2 Description

When the 3rd axis is moving vertical to arc plane, G02/G03 is to be helical interpolation. The way to define arc plane of helical interpolation is the same as circular interpolation. Helical interpolation uses G code(G17/G18/G19) to define which plane to do circular interpolation.

G17 mode: The X-Y plane is a circular interpolation plane, and the Z-axis is linear interpolation axis.

G18 mode: The Z-X plane is a circular interpolation plane, and the Y-axis is linear interpolation axis.

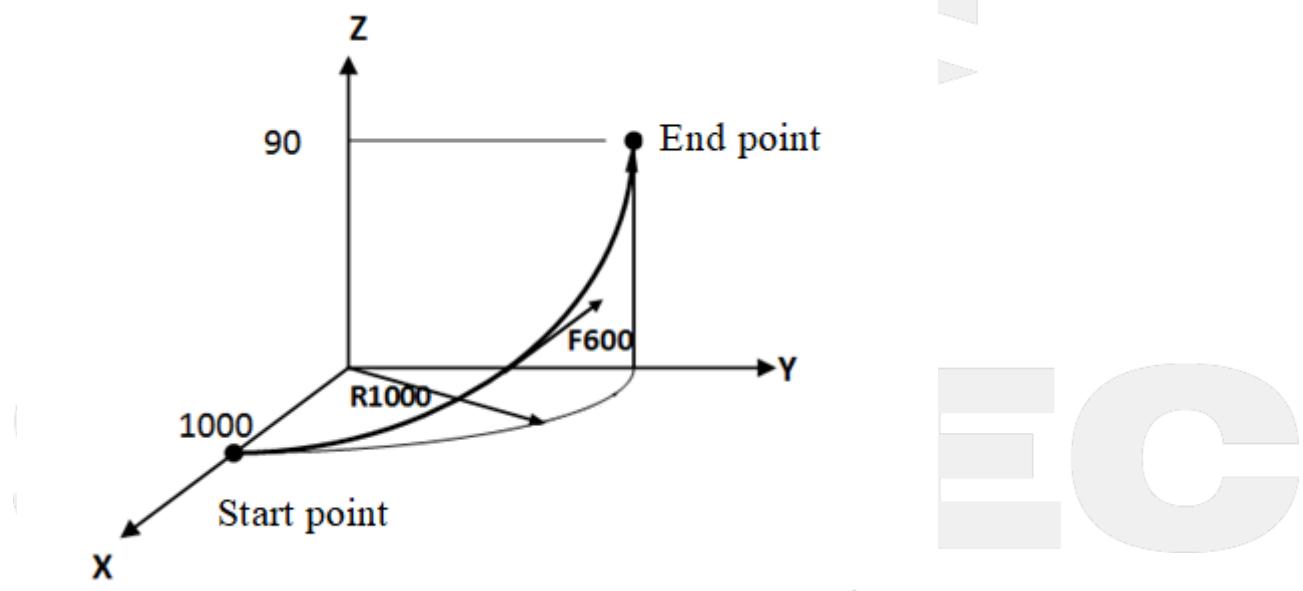
G19 mode: The Y-Z plane is a circular interpolation plane, and the X-axis is linear interpolation axis.

2.4.3 Notice

1. When G02/G03 without any R, I, J, or K argument, the block will be executed as G01.
2. When X, Y, Z, I, J, K and R argument given by G02/G03 is out of order. For example, in G17 mode, the K argument is not zero, the system would issue a [COR-006 arc end is not on the arc] alarm. This alarm can adjust the region through Pr3807.

2.4.4 Example

PIC:



Program description:

```
G17 G03 X0.0 Y1000.0 R1000.0 Z90.0 F600;
// X-Y plane arc, counterclockwise direction (CCW), Z-axis linear interpolation
// Cutting feedrate 600mm/min for spiral cutting
```

2.5 G02.4/G03.4 : Three Point Circular Interpolation

2.5.1 Command Form

$\left\{ \begin{array}{l} G02.4 \\ G03.4 \end{array} \right\} X1_ Y1_ Z1_ \alpha1_ \beta1_ F_ ; \text{ // 第一程序段(圆弧中間點)}$

$X2_ Y2_ Z2_ \alpha2_ \beta2_ ; \text{ // 第二程序段(圆弧終點)}$

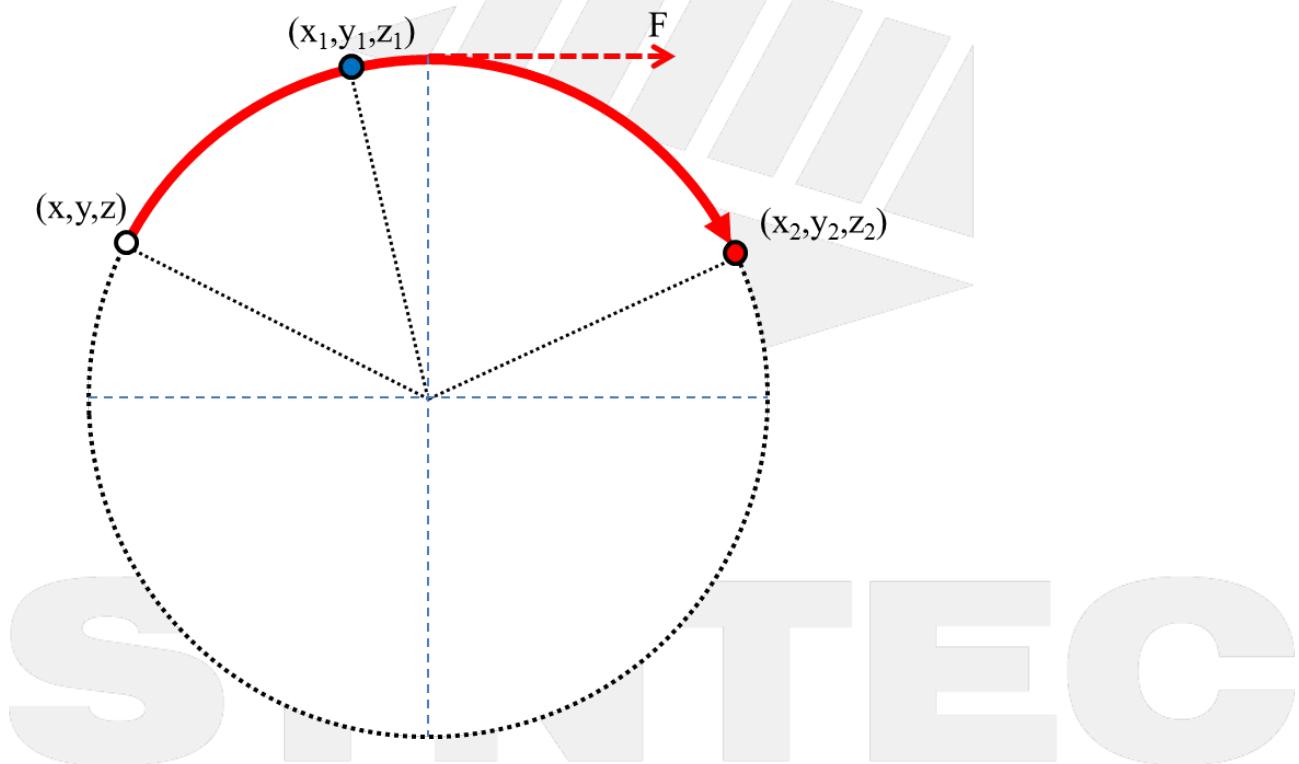
$X1_ Y1_ Z1_ \alpha1_ \beta1_$: First sequence(the mid-point of arc);

$X2_ Y2_ Z2_ \alpha2_ \beta2_$: Second sequence(the end point of arc);

F : Arc tangent Feedrate;

α, β : The axial direction other than the X, Y and Z axis can be not specified if not required.

X, Y, Z : If any of the axial specified point coordinates is omitted, it will follow the value as the previous specified point.



(x, y, z) : Start point of arc(the and point of previous block)

(x_1, y_1, z_1) : the mid-point of arc

(x_2, y_2, z_2) : the end point of arc

F : Arc tangent Feedrate

2.5.2 **Description**

G02.4/G03.4 three-point circular interpolation, in order to utilize the three points given in the space, through the geometric relationship calculation, a circular interpolation function that can be connected according to the three-point sequence is obtained. When using, there is only need to give the arc intermediate point and the end point coordinate (G02.4 or G03.4 the end point of the previous block is the starting point of the arc), also the arc tangent feedrate, a three-point arc in a space can be determined.

2.5.3 **Notice**

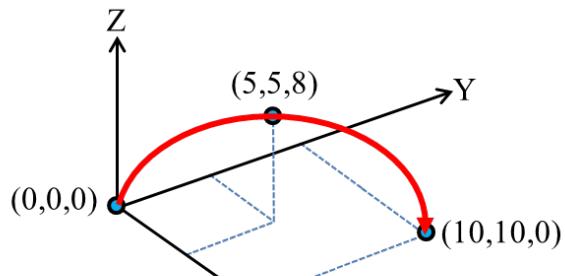
1. The valid version is 10.116.16B (inclusive) and needs to active Option19. If applying G02.4/G03.4 without Option19 is active, the COR-100 alarm would be triggered (this model does not support this G code).
2. The G02.4 command has the same action as the G03.4 command and can be replaced with each other and supports absolute/incremental commands.
3. The end point of the previous block of G02.4/G03.4 is the starting point of this arc.
4. The G02.4/G03.4 command is regarded as a group in two lines, which can be specified continuously. The end point of the previous arc will become the starting point of the next arc, but the F argument can only be in the odd line. If the total number of lines in the command is odd or the number of lines in the F-number is even, the COR-134 alarm would be triggered (G02.4/G03.4 command format error).
5. If the three points forming the arc are on the same line or if any two points coincide (for example, if the whole circle is specified, the end point will be the same), the interpolation will be performed in the linear interpolation mode (G01). The interpolation path starts from the start point to the middle point, then from the middle point to the end point.
6. When single block is executed, it would move from the start point to the end point of the arc when starting a cycle, and will not stop at the middle point of the arc.
7. When using this function, the tool radius compensation function must be turned off first, and the commands such as A, C, and R are not supported. Otherwise, the COR-133 alarm will be triggered (this command is not supported in G02.4/G03.4 interpolation mode).
8. In G02.4/G03.4 interpolation mode, G53 machine coordinate positioning is illegal; nor can the G02.4/G03.4 command be connected to the next block after G53 machine coordinate positioning. Both of the above situations will cause the arc path to go wrong.
9. If the α/β command is omitted in the first block and only specified in the second block, the unspecified α/β axis does not move from the start point to the intermediate point of the arc, but moving from the intermediate point to the end point, the α/β axis also moves to the specified position.
10. As mentioned above, if the α/β command is omitted in the second block and only specified in the first block, the α/β axis will move to the specified position when moving from the starting point of the arc to the intermediate point. When the middle point of the arc reaches the end point, the unspecified α/β axis does not move.

2.5.4 **Example**

Example 1 :

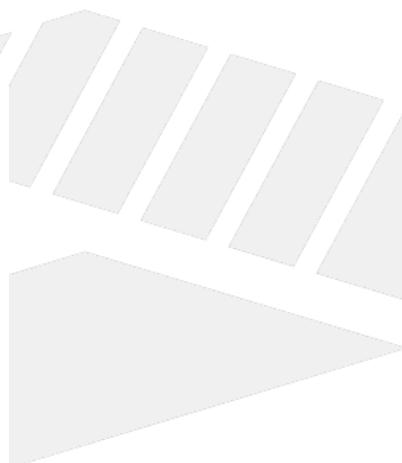
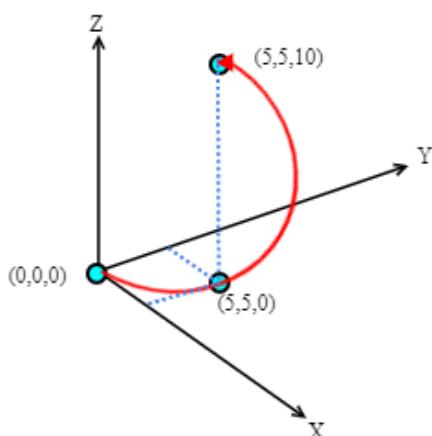
G90 G01 X0 Y0 Z0 F5000
G02.4 X5. Y5. Z8. F1000

X10. Y10. Z0.



Example 2 :

```
G90 G01 X0 Y0 Z0 F5000
G02.4 X5. Y5. F1000
Z10.
```



2.6 G02/G03 : Spiral Interpolation, Conical Interpolation

2.6.1 Command Form

```
G17 G02/G03 X_Y_I_J_L_F;
G18 G02/G03 Z_X_K_I_L_F;
G19 G02/G03 Y_Z_J_K_L_F;
X_Y_Z_ : end point position;
I_J_K_ : vector from the starting point to the center of circle;
L_ : number of turns
F : feedrate;
```

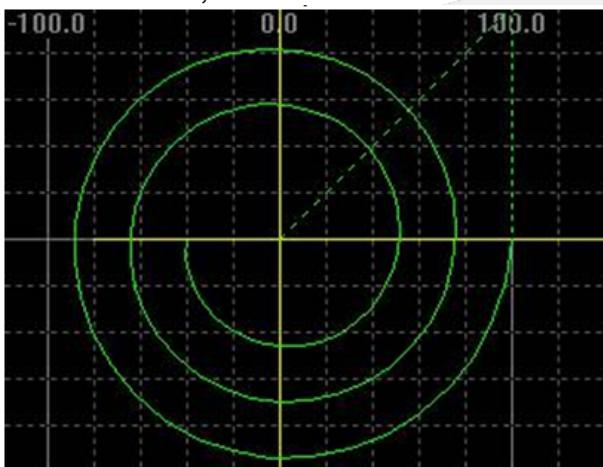
2.6.2 Description

The spiral interpolation syntax is similar to the circular interpolation G02/G03. The only difference is that there is one more argument L, turns of circle. L is an integer, and a part of the circle is regarded as one turn. When the G02/G03 command has the number of L turns, it is regarded as the Spiral Interpolation. The starting radius and the ending radius are allowed to be different, and the alarm that the arc end point is not at the arc will not be issued. For Spiral Interpolation, when the vertical axis command is added, it is conical interpolation.

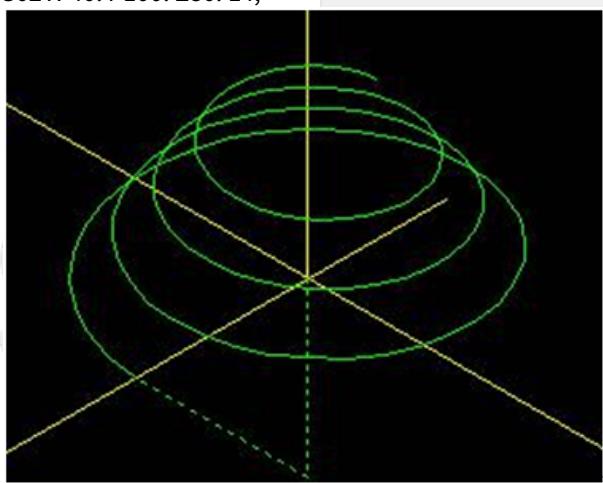
Notice that 6 series do not support this function.

2.6.3 Example

G17 G00 X100. Y0 Z0;
G02 X-40. I-100. L3;



G17 G00 X100. Y0 Z0;
G02 X-40. I-100. Z80. L4;



2.7 G04 : Dwell

2.7.1 Command Form

G04 { *X* _ *P* _ } *Q* _

X: dwell time (With decimal point in seconds; no decimal point, in milliseconds.)

P: dwell time (in milliseconds, decimal point not permitted)

Q: skip signal source, range: 101~132, corresponding to C101~C132 respectively

2.7.2 Description

Sometimes delay are required when machining(taper pits, milling corners), then we can use G04 function to pause each axis for a period of time to achieve the desired precision.

2.7.3 Precaution

- G04 command is only effective in single block.
- With Q argument, G04 can be terminated before the specified time.

Example

G04 X2500;//delay 2.5 sec

G04 X2.5;//delay 2.5 sec

G04 P2500;//delay 2.5 sec

G04 P2.5;//delay 2 millisec (decimal point not permitted)

2.8 G04.1 : Path Synchronization Waiting

2.8.1 Command Form

G04.1 P _ [*Q* _]

P : Numbering of the signal waited

Q : Enter the specified main system to synchronize, without *Q* argument means synchronizing all system paths.

Value in decimal.

Q argument format description :

1. *Q* argument assign the paths to synchronize, support up to 4 paths. The corresponding *Q* arguments for each path are in the table below:

Path ID	Q Argument(decimal number)
1	1

Path ID	Q Argument(decimal number)
2	2
3	3
4	4

2. Take 4 main system paths as an example (Pr731=4): To synchronize path 1, 2, and 4, Q argument must be a series of number 1, 2, and 4 such as Q124.
3. The Q argument number series has no order restriction, all six combinations make path 1, 2, and 4 to synchronize:
Q124, Q142, Q241, Q214, Q412, Q421
4. The synchronization is effective when command's P are equal and Q assigned path exists, even if Q number orders are different in each path's command. The following cases are equivalent Q arguments, and path 1, 2, and 4 will synchronize:
 - a. Processing program of 1st path G04.1 P2 Q124
 - b. Processing program of 2nd path G04.1 P2 Q241
 - c. Processing program of 3th path G04.1 P2 Q412

2.8.2 Description

- G04.1 is used for the need to synchronize different paths. For instance: Example 3, to use \$1 to change \$2 main spindle RPM. When \$2 is under G95 mode, use G04.1 in \$1 & \$2 to update modes in both paths to prevent incorrect RPM and feed rate in \$2.
- If there are 2 paths, G04.1 P1 [Q12] in path 1 and G04.1 P1 [Q12] in path 2 will wait until both paths are synchronized, and then execute then ext block.
- Similarly, G04.1 P2 [Q12] in path 1 and G04.1 P2 [Q12] in path 2 will wait until both paths are synchronized, and then execute then ext block. Others P values are in similar function.
- G04.1 with Q value must be used in the assigned path (no Q value as well), and P value must be use in order.

\$1	\$2	\$3
G00 X0. G04.1 P1 G01 X10. F1000 <u>G04.1 P2 Q13</u> X20. <u>G04.1 P3 Q13</u> X30. <u>G04.1 P5 Q13</u> X40. G04.1 P6 M30	G00 Y0. G04.1 P1 G01 Y10. F1000 Y15. <u>G04.1 P4 Q23</u> Y20. G04.1 P6 M99	G00 Z0. G04.1 P1 G01 Z10. F1000 <u>G04.1 P2 Q13</u> Z25. <u>G04.1 P3 Q13</u> Z40. <u>G04.1 P4 Q23</u> Z55. <u>G04.1 P5 Q13</u> Z70. G04.1 P6 M99

*Program above: G04.1 w/o Q showed twice in each path; G04.1 Q13 showed 3 times in path 1 & 3; G04.1 Q23 showed once in path 2 & 3.

- To repeat program, use M99 at the end of path 1. Notice to add same G04.1 P_ before M99 in each path to ensure all paths repeat synchronously. Ex: the program above has G04.1 P6 before the end of program.

2.8.3 **Notice**

1. Alarm "COR-137 G04.1 P arguments in wrong order" will be triggered in following cases:
 - a. Different P when no Q is used.
 - b. Different P when same Q is used.
 - c. Same P when different Q are used.
2. Alarm "COR-144 G04.1 Q argument value incorrect" will be triggered in following cases:
 - a. Q is not positive integer.
 - b. Q value assigned to a non-existing path.
 - c. Q value excludes current path. Ex: G04.1 P1 Q23 under path 1.
3. For compatibility, G04.1 with no Q argument means all paths are assigned.
4. G04.1 is not supported under non-CNC main path and all its sub-programs.
5. When waiting at G04.1, system status is "Running". Examples below (shows status of path 1):

Environment : C40 On, programs runs one block at each Cycle Start hit.

Description: \$1 and \$2 finished single block, and path 1 is at "Block Stop" status.



Description: \$1 waits for \$2, path 1 status is "Busy".



2.8.4 Example

Program 1 :

\$1	\$2
G04.1 P1; G01 X50. F2000; G04.1 P2; Z100.; G04.1 P3; X0.; G04.1 P4; Z0.; G04.1 P5; M99;	G04.1 P1; G01 X250. F3000; G04.1 P2; Z500.; G04.1 P3; X0; G04.1 P4; Z0; G04.1 P5; M99;

Program 2 :

\$1	\$2	\$3
G04.1 P1 Q12; // \$1 & \$2 sync G01 X50. F2000; Z100.; X0.; Z0.; G04.1 P2; // all path sync, and repeat. M99;	G04.1 P1 Q12; // \$1 & \$2 sync G01 X25. F3000; Z50.; X0.; Z0.; G04.1 P2; // all path sync, and repeat. M99;	G00 X10. Z10.; G04 X1.; X0. Z0.; G04 X1.; G04.1 P2; // all path sync, and repeat. M99;

SYNTEC

Program 3 : Two spindles first synchronize, and the two paths begin cutting separately. Notice the order of G04.1P_ and main spindle S argument, incorrect order will cause feed rate F of \$2 abnormal.

\$1	\$2
<pre> G04.1 P1 // sync with \$2 M03 S30 G114.1 R0 // enable spindle synchronize G04.1 P2 // sync with \$2 G01 U10. 1. U-10. G04.1 P3 // sync with \$2, change spindle RPM after \$2 finishes M03 S60 // spindle syncing, spindle 1= 60RPM G04.1 P4 // sync with \$2 G04.1 P5 // sync with \$2, disable spindle synchronize after \$2 finishes G113 // disable spindle synchronize M05 G04.1 P6 // sync with \$2, keep \$1 from executing M30 while \$2 not finishes. </pre>	<pre> G04.1 P1 // sync with \$1, prevent M99 to restart M13 S15 G04.1 P2 // sync with \$1, spindle 2 sync to 30 RPM. G01 U10. F2. // G95 mode, feed rate = 30*2 = 60 mm/ min U-10. G04.1 P3 // sync with \$1 G04.1 P4 // sync with \$1, spindle 2 sync to 60 RPM. G01 U10. // G95 mode, feed rate = 60*2 = 120 mm/ min U-10. G04.1 P5 // sync with \$1 M15 G04.1 P6 // sync with \$1 M99 </pre>

2.9 G05.1 : Path Smoothing

2.9.1 Command Form

G5.1 { Q1
Q2 } E_ : enable smoothing function

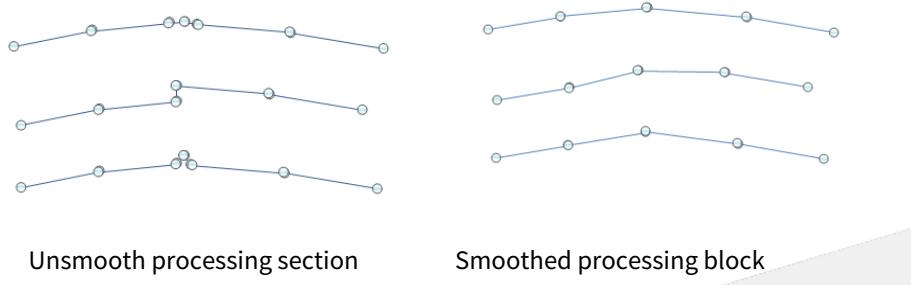
G5.1 Q0 : Disable smoothing function

Q : Enable/Disable the smoothing function. The enable function can be divided into two modes: Q1/Q2.

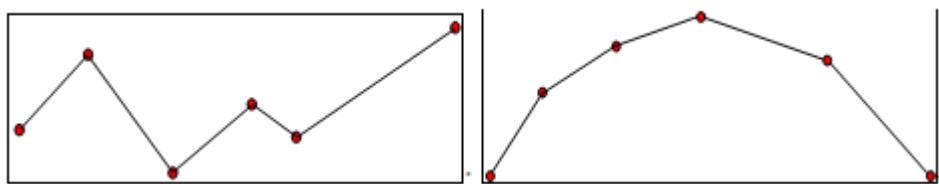
E : The maximum allowable path error while smoothing. Unit is IU(Metric mm/Imperial inch)

2.9.2 Description

According to the E_ argument of G05.1, the processing path is modified and smoothed to shorten the processing time and improve the stability. The larger the E argument is, the greater the difference between the smoothed trajectory and the original path is.



1. Q1 Applicable situation: As shown in the left-down figure, the NC path is more rugged and uneven.
2. Q2 Applicable situation: As shown in the right-down figure, the NC path has the same curvature, but the density of dots is relatively sparse. Generally, the machining accuracy of CAD/CAM is set lower.



Notice

1. This feature only supports Mill model CE systems, the effective version starts at 10.114.56.
2. The upper limit of the E_ argument is 1.0, and the unit is 1mm/1inch. When the argument is greater than 1.0, the internal order is 1.0. (10.116.56B and before 10.118.12J version, the upper limit of E argument is 0.1mm)
3. The command is incomplete (for example, Q or E is missing) or the command is incorrect (for example, the Q value is wrong, or the E value is less than 0 or equal to), then system issues [COR-107 G5.1/G05 command format error] alarm.
4. In the G61/G63 mode, enable the smoothing function (G5.1) is prohibited, otherwise system issues [COR-106 G5.1 is prohibited to do path smoothing in G61 or G63 mode] alarm.
5. As mentioned above, when G61/G63 is enabled in the G5.1 mode, the system will stop the smoothing function until jumps out of the G61/G63 mode then enable again.
6. The smooth motion is valid only for the G01 command between G5.1 Q1/Q2 E_to G5.1 Q0.
7. If the tool length correction command (such as G43) or the coordinate definition command (such as G54) is issued in the G5.1 mode, also the next command is G01, then smoothing action is not performed, and the subsequent G01 command returns to the smoothing action.
8. If UVW is set to XYZ axis increment command (Pr3809=1), this function is prohibited.
9. This feature doesn't support Multi-Axis Multi-Signal Skip Function(G31.10/G31.11).

2.9.3 Example

N001 G05.1 Q1 E0.01//enable path smoothing function, allowable error: 10um

N002 G90 G01 F2000

N003 X-0.002 Y-0.001//the following commands perform path smoothing function

N004 X-0.003 Y-0.003

N005 X-0.004 Y-0.005

N006 X-0.005 Y-0.007

N007 X-0.007 Y-0.008

N008 X-0.008 Y-0.009

N009 X-0.011 Y-0.010

N010 X-0.013 Y-0.012

```

N011 X-0.014 Y-0.013
N012 X-0.015 Y-0.015
N013 X-0.016 Y-0.018
N014 G05.1 Q0//disable path smoothing function
N015 M30//program ends

```

```

N001 G05.1 Q2 E0.01//enable path smoothing function, allowable error: 10um
N002 G91 G01 F2000
N003 X-0.002 Y-0.001//the following commands perform path smoothing function
N004 X-0.001 Y-0.002
N005 X-0.001 Y-0.002
N006 X-0.001 Y-0.002
N007 X-0.002 Y-0.001
N008 X-0.001 Y-0.001
N009 X-0.003 Y-0.001
N010 X-0.002 Y-0.002
N011 X-0.001 Y-0.001
N012 X-0.001 Y-0.002
N013 X-0.001 Y-0.003
N014 G05.1 Q0//disable path smoothing function
N015 M30//program ends

```

2.10 G05 : High-Precision Contour Control Function (XYZ Geometry Axes)

2.10.1 Command Form

Enable G05 high-precision contour control mode :
G05P10000
G05P10000 E_
G05E_

Disable G05 high-precision contour control mode :
G05 P0
G05

P : Specifies to enable or disable the G05 high-precision contour control mode. The P value is set 10000 to enable and 0 for disable. If the E argument is set, the P argument default is 10000; if E argument and the P argument are not set, the P argument is set 0.

E : According to this error value, controller adjusts the parameters or contour automatically, in units of metric mm, imperial inch.

2.10.2 Description

1. G05 High-Precision contour Control Mode (HPCC) consists of two cores :
 - a. Curve fitting : According to the allowable geometric axis error value Pr407, the curve is fitted to achieve an ideal continuous smooth contour.
 - b. Parameter optimization : Adjust the core internal parameters automatically according to the allowable error value E or the machining error tolerance TOL, and compensate the accuracy so that the final machining result is close to the initial tolerance value.
2. Related parameters
 - a. Pr407 : Curve fitting error (um), this parameter is used for curve fitting.

- b. TOL : Machining error tolerance (um), this parameter is used for parameter optimization. If E value is not set when the function is enabled, then refer to this parameter (parameter setting: parameter setting → high speed high precision parameter → quick parameter setting).

3. Function enable description

The following table shows how to set G code to turn on HPCC. The conditions follows :

Command	Pr3808(SPA)	Smooth Level	Enable Function
G05 P10000	0	0~9	Curve fitting
	1~5	0	
G05 P10000	1~5	1~9	Curve fitting Parameter optimization(Refer to TOL)
G05 P10000 E_	1~5	0~9	Curve fitting
G05 E_			Parameter optimization(Refer to E value)

The following table shows how to set Pr3802 to turn on the HPCC. It does not need to program G05. The HPCC function is preset when processing. The conditions follows :

Pr3802	Pr3808(SPA)	Smooth Level	Enable Function
2	0	0~9	Curve fitting
	1~5	0	
2	1~5	1~9	Curve fitting Parameter optimization

Note 1 : In the later version of 10.116.36 (inclusive), the core uses the new version of SPA2.0 (ZPEC), which does not support parameter optimization. The reason can be referred to "Note 2".

The version between 10.116.6 (inclusive) and 10.116.35 (inclusive), the core uses the original SPA to support parameter optimization.

Note 2 : Since the accuracy control of SPA2.0 (ZPEC) is greatly improved, it is not necessary to support parameter optimization to get high precision.

Note 3 : E_ and parameter optimization valid versions end before 10.116.54A (inclusive).

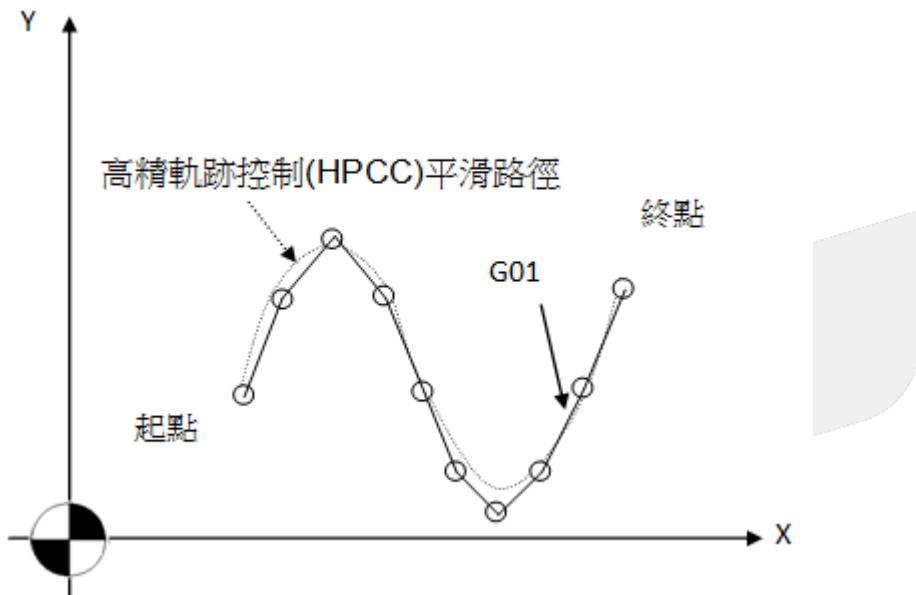
2.10.3 **Notice**

1. G05 mode is only available on the CE version of the milling machine. high-precision contour control mode (HPCC) is the software function option 11. Please purchase it from OEM first if needed; the parameter optimization function needs to enabled the SPA function (Pr3808 is not 0, it is recommended to set 5).
2. The value of the E argument must be a positive value. If the argument is too small, the processing will be too slow or pause continued.

3. When parameter optimization is performed, Pr401 and Pr408 are used as the limit of the parameters, and the final acceleration and arc speed do not exceed the default parameters.
4. When the lathe system or Pr3809 is set to 1 (UVW is the XYZ axis incremental command in the machining program), high-precision contour control mode cannot enable.
5. High-precision contour control function only valid with radius axis (Pr281~=0).
6. In high-precision contour control mode, the exact stop (G09/G61) will be invalid.
7. In high-precision contour control mode, program G61/G63 will stop high-precision contour control function, and automatically restart high-precision contour control till jumping out of the G61/G63 mode.
8. In the G61/G63 mode, it is prohibited to enable high-precision contour control function, otherwise the system will issue an alarm.
9. The P argument can only be 10000 or 0. If the argument is wrong, the system will issue an [COR-107 G5.1/G05 command format error] alarm.
10. High-precision contour control mode supports pause, emergency stop, reset, and feedrate override switching, but does not support single-segment stop (C40/M00) and reverse handwheel simulation.
 - a. If single block stop C40 is needed, enable C40 before processing of disable HPCC mode.
 - b. It does not support single block stop C40. It means that it cannot be guaranteed to stop at the same position when high-precision contour control mode is disable. Because the block may be smoothed, stop at the smoothed position after C40.
11. High-precision contour control mode supports post-acceleration, deceleration, SPA function, and mechanism compensation functions. However, single-block stop prohibition (#1502) and feedrate override change prohibition (#1504) are not supported.
12. In high-precision contour control mode, the man-machine line number will display the actual leading position.
13. In high-precision contour control mode, up to three geometry axes are supported, and the rotary axis is not supported. To support the rotary axis, refer to the G05 high-precision contour control function (including rotary axis) page, the program specifications and software version restrictions.
14. In high-precision contour control mode, the axis inhibit function (R603) checks the X, Y, and Z geometry axes simultaneously.
15. In high-precision contour control mode, the inverse time feed function (G93) is not supported.
16. Graphic simulation is not supported in high-precision contour control mode.
17. In high-precision contour control mode, Remaining distance display is not supported, because in this mode, the distance from the end point of the curve is displayed instead of the end point of the single block (because the end point of the single block may be smoothed).
18. In high-precision contour control mode, the interrupt signal (C49) of the interrupted subprogram call function is not supported.
19. 10.116.54B, 10.118.1 and later versions, the quick parameter setting function has been removed, and Multi-condition Machining are used instead. The processing monitoring page also removes the input box for setting the smoothing level.



2.10.4 Example



N001 G0 X3. Y4. Z0.
 N002 G05 P10000 // Enable high precision contour control mode. Interpolation smoothing curve.
 N003 G01 X3.8 Y6.1 F5000.
 N004 X4.6 Y7.
 N005 X5.4 Y6.1
 N006 X6.1 Y4.
 N007 X6.9 Y1.9
 N008 X7.7 Y1.
 N009 X8.5 Y1.9
 N010 X9.3 Y4.
 N011 X10. Y6.1
 N012 G05 P0 // Disable high precision contour control mode
 N013 M30

2.11 G05 : High-Precision Contour Control Mode (Rotary Axis Included)

2.11.1 Command Form :

Enable G05 high-precision contour control mode (HPCC) :
 G05 P10000 X0 Y0 Z0 (α_-) (β_-)

Disable G05 high-precision contour control mode (HPCC) :
 G05 P0
 G05

P : Specifies that enable or disable the G05 high-precision contour control mode (HPCC). The P value is set 10000 for enable and 0 for disable .

X Y Z : Specifies the axis name of the geometric axis to be smoothed. The argument must be 0.

(α_-) (β_-) : Specifies the axis name of the rotary axis to be smoothed. The argument is tolerance. It must be positive (>0) and the unit of argument refers to "description".

2.11.2 **Description**

1. G05 high-precision contour control mode (HPCC)
 - a. Curve fitting : According to the allowable geometric axis error value Pr407 and the allowable error value (α_-) (β_-) of the given rotary axis, the curve is fitted to achieve a desired continuous smooth contour.
2. Related parameters
 - a. Pr407 : Curve fitting error (um), this parameter is used for curve fitting.
 - b. Pr17、Pr3241 : It is used to set the unit of (α_-) (β_-) argument. The following table shows common cases. The rest of the settings can be referred from parameters link.

Parameter	Pr17 = 2, Pr3241 = 0	
Program	G05 P10000 X0 Y0 A0.1 B0.3 ... G05 P0	G05 P10000 X0 Y0 A100 B300 ... G05 P0

SYNTec

Description	Enable HPCC function and set the axis to X, Y, A, and B axes to be smoothed. The smoothing tolerance of the A axis is 0.1 deg. The smoothing tolerance of the B axis is 0.3 deg.	Enable HPCC function and set the axis to X, Y, A, and B axes to be smoothed. The smoothing tolerance of the A axis is 0.1 deg. The smoothing tolerance of the B axis is 0.3 deg.
--------------------	--	--

2.11.3 **Notice**

1. G05 mode is only available on the CE version of the milling machine. high-precision contour control mode (HPCC) is the software function option 11. Please purchase it from OEM first if needed.
2. The version after 10.116.36 (inclusive), high-precision contour control mode supports command specification G05 P10000 X0 Y0 Z0 (α_-) (β_-). It can set 1~3 geometric axes and 0~2 rotation axes to be smooth axial direction, and support up to three geometry axes, two rotation axes meanwhile.
3. When the lathe system or Pr3809 is set to 1 (UVW is the XYZ axis incremental command in the machining program), high-precision contour control mode cannot enable.
4. high-precision contour control function only valid with radius axis (Pr281~=0).
5. In high-precision contour control mode, the exact stop (G09/G61) will be invalid.
6. In high-precision contour control mode, program G61/G63 will stop high-precision contour control function, and automatically restart high-precision contour control till jumping out of the G61/G63 mode.
7. In the G61/G63 mode, it is prohibited to enable high-precision contour control function, otherwise the system will issue an alarm.
8. In high-precision contour control mode, when RTCP mode (G43.4)/STCP mode (G43.4 L1) is enabled, the HPCC mode would disable automatically. After jump out RTCP/STCP mode (G49), high-precision contour control would enable automatically.

9. In RTCP mode (G43.4)/STCP mode (G43.4 L1), HPCC function is prohibited, otherwise the system will issue an alarm [COR-140 tool nose control mode disable G05].
10. The P argument can only be 10000 or 0. If the argument is wrong, the system will issue a [COR-107 G5.1/G05 command format error] alarm.
11. When only program G05 P10000 to enable high-precision contour control mode (HPCC) without setting smooth axis, the default smooth axis is X, Y, and Z(equivalent to G05 P10000 X0 Y0 Z0).
12. Enable command G05 P10000 X0 Y0 Z0 α _ β _, α , β represent rotary axis. The specified tolerance unit of the rotary axis will be affected by Pr17 and Pr3241. In the following cases, the system will issue an [COR-107 G5.1/G05 Instruction format error] alarm:
 - a. If more than five axes are specified.
 - b. If geometry axis argument is not 0.
 - c. If rotation axis argument is 0.
 - d. If geometry axis axis is not set, but the axis of the rotation axis is set.
 - e. If more than two axes of rotation are specified.
 - f. If any axial argument is negative.

In particular, X, Y, and Z are not recommended to be set as rotary axis type, and the α and β axis are not recommended to be set as linear axis. It also means that this function suggests that the number of linear axis is at most three and the number of rotating axes is at most two.

13. high-precision contour control mode supports pause, emergency stop, reset, and feedrate override switching, but does not support single-segment stop (C40/M00) and reverse handwheel simulation.
 - a. If single block stop C40 is needed, enable C40 before processing of disable HPCC mode.
 - b. It does not support single block stop C40. It means that it cannot be guaranteed to stop at the same position when high-precision contour control mode is disable. Because the block may be smoothed, stop at the smoothed position after C40.
14. High-precision contour control mode supports post-acceleration, deceleration, SPA function, and mechanism compensation functions. However, single-block stop prohibition (#1502) and feedrate override change prohibition (#1504) are not supported.
15. In high-precision contour control mode, the man-machine line number will display the actual leading position.
16. In high-precision contour control mode, if there is a non-smooth axial command or a single block with zero geometric axis movement, high-precision contour control function will be temporarily disable and no alarm will be issued.
17. In high-precision contour control mode, the axis inhibit function (R603) would checks the X, Y, and Z geometry axes and α , β two rotary axes simultaneously.
18. In high-precision contour control mode, the inverse time feed function (G93) is not supported.
19. Graphic simulation is not supported in high-precision contour control mode.
20. In high-precision contour control mode, Remaining distance display is not supported, because in this mode, the distance from the end point of the curve is displayed instead of the end point of the single block (because the end point of the single block may be smoothed).
21. In high-precision contour control mode, the interrupt signal (C49) of the interrupted subprogram call function is not supported.
22. The HPCC function of the linear axis and the rotary axis is automatically disable when Reset.

2.11.4 Example

N001 G0 X3. Y4. Z0. B0.

N002 G05 P10000 X0 Z0 B0.002 // Enable high-precision contour control mode (HPCC), and the smoothing axis are X, Z, and B. B axis tolerance is 0.002 degree (in the case of Pr17=2, Pr3241=0), and the smoothing curve is interpolated.

N003 G01 X3.8 Z6.1 B3.4 F5000

N004 X4.6 Z7.

N005 X5.4 B6.1

```

N006 X6.1 Z4. B3.6
N007 X6.9 Z1.9
N008 X7.7 B1.
N009 Z8.5 B1.9
N010 Z9.3 B4.
N011 X10. Z6.1 B1.
N012 G05 P0    //Disable high precision contour control mode (HPCC).
N013 M30

```

2.12 G06.2 : NURBS Curve Interpolation

2.12.1 Command Form

G05 P10000; // Enable high speed & high precision interpolation

:

G06.2 P_K_X_Y_Z_R_F__;//NURBS curve interpolation

K_X_Y_Z_R_;

K_X_Y_Z_R_;

K_X_Y_Z_R_;

K_;

K_;

K_;

K_;

:

G05 P0;// Disable high speed & high precision interpolation

P : Order of NURBS curve (2 ~ 4) , default value is 4 if argument is null.

K : NURBS node value of curve

X_ Y_ Z : NURBS control-point coordinates

R : NURBS curve weight (0.001 ~ 1000) , default value is 1.0 if argument is null.

F : The maximum feedrate of NURBS curve (mm/min), default value is that of previous curve if argument is null.

2.12.2 Description

G06.2 executes NURBS curve interpolation according to the program. G90/G91 determines whether absolute or incremental mode is used. The cutting feedrate of NURBS curve interpolation is set by “F” argument . (This function only provided in CE system)

2.12.3 Notice

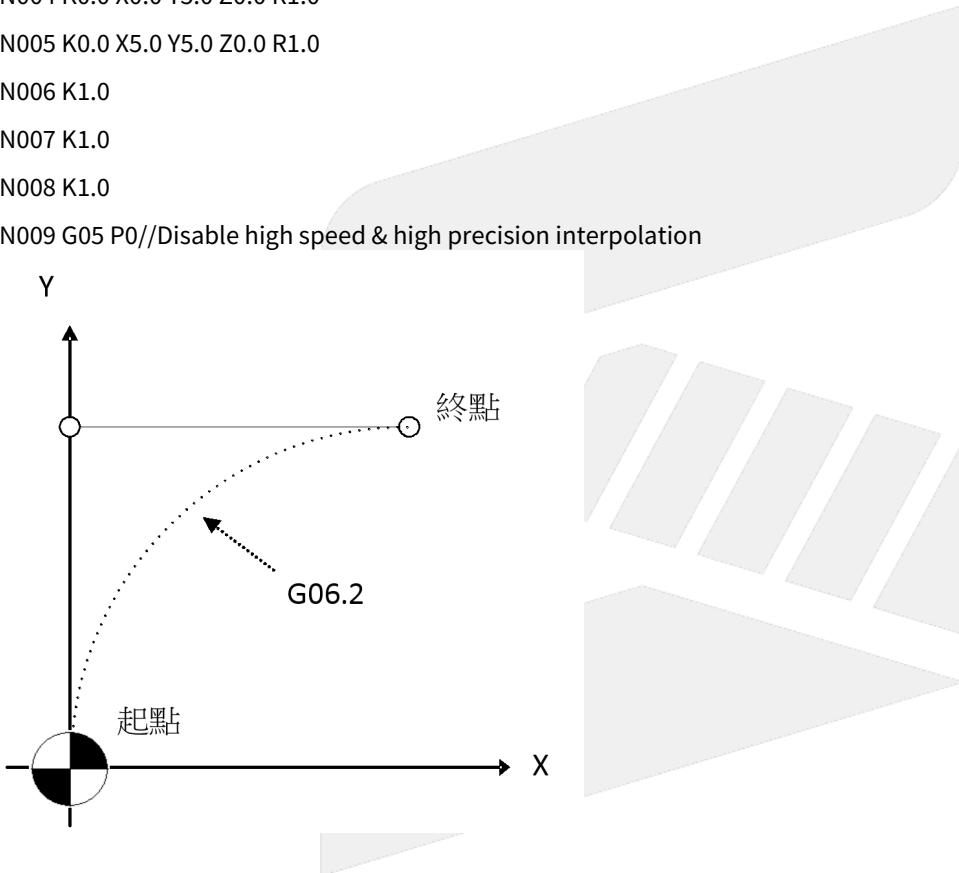
Single block execution and handwheel negative simulation are not supported.

2.12.4 Example

```

N001 G0 X0.0 Y0.0 Z0.0
N002 G05 P10000//Enable high speed & high precision mode
N003 G06.2 P3 K0.0 X0.0 Y0.0 Z0.0 R1.0 F5000.
//execute NURBS curve interpolation
N004 K0.0 X0.0 Y5.0 Z0.0 R1.0
N005 K0.0 X5.0 Y5.0 Z0.0 R1.0
N006 K1.0
N007 K1.0
N008 K1.0
N009 G05 P0//Disable high speed & high precision interpolation

```



2.13 G09/G61 : Exact Stop

2.13.1 Command Form

```

G09 X__ Y__ Z__ ;
G61 ;

```

X, Y, Z: position of exact stop

2.13.2 Description

while cutting the corner, because of the tool fast moving or servo delays, tool can not cut the exact shape of corner. But, when high precision rectangular is needed, G09 or G61 is able to make it. It slow down the tool while approaching to the corner, when reach to the specified position (in CNC parameter range), it will run the next block.

G09 exact stop only effected in one block which has G09; G61 exact stop effected each cutting command (G01~G03) after G61, till automatic corner feed percentage (G62) or tapping mode (G63) or cutting mode (G64) is specified.

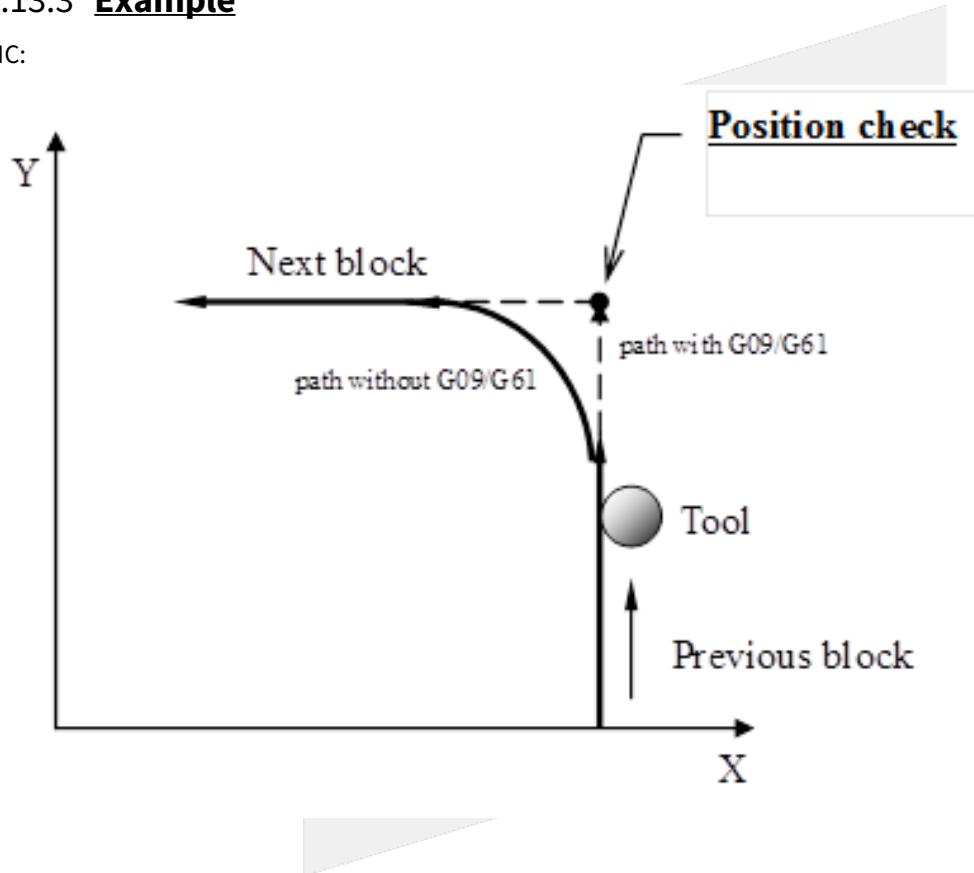
Note :

G01 checking window : Pr 421-440

G00 checking window : Pr 481-500

2.13.3 Example

PIC:



2.14 G10 : Programmable Data Input

2.14.1 Command Form

$$G10 \left\{ \begin{matrix} L10 \\ L11 \\ L12 \\ L13 \end{matrix} \right\} P_- R_- ;$$

L10 : Apply with tool length geometric

L11 : Apply with tool length wear

L12 : Apply with tool diameter geometric

L13 : Apply with tool diameter wear

P: tool NO. ;

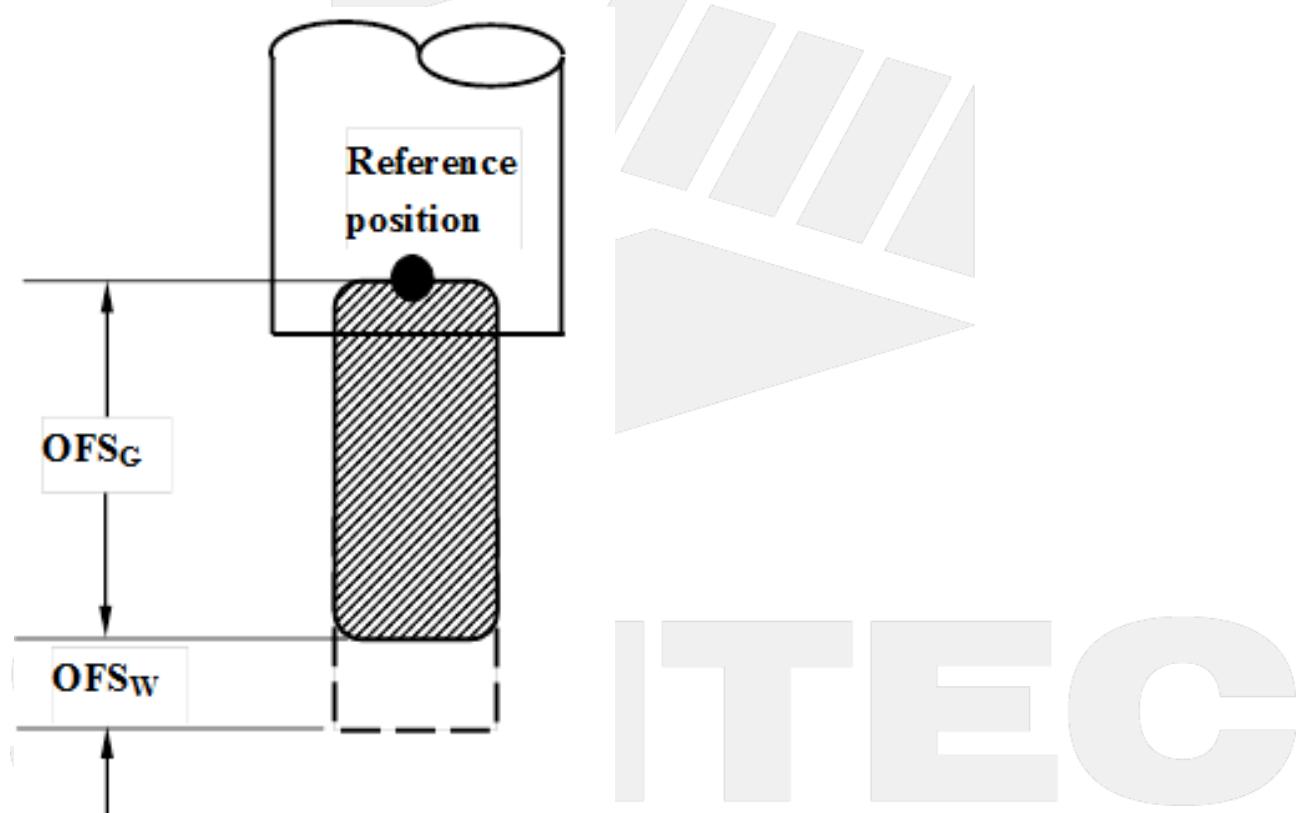
R: compensation value(data of tool length or tool diameter) ;

2.14.2 **Description**

1. G10 command can directly use to enter tool compensation value by program command.
2. In absolute mode (G90), compensation of G10 is new value; in increment mode (G91), compensation of G10 is the sum of the current value and the new compensation value.
3. This syntax is only available for Pr3816=0/1, input a single-axis compensation value.
4. PLC axis control components (PLC Axis) are not supported.

2.14.3 **Example**

PIC:



2.15 G10.9 : Diameter/Radius Axis Program Switching

2.15.1 Command Format

G10.9 X_ Y_ Z_ Diameter/radius axis programming switching
 X_ Y_ Z : Assign the specific axis to program in diameter/radius axis
 0 : Program in radius axis
 1 : Program in diameter axis

2.15.2 Description

Users can give G10.9 command in the program to specify all the axial commands in diameter or radius axis afterwards.

2.15.3 Notice

1. Diameter/radius axis program switching(G10.9) is valid after version 10.1118.8.
2. Please program G10.9 X_ Y_ Z_ command in a line and only, without other commands.
3. With G10.9 command given, it'll return to the Pr281~Pr300 diameter/radius axis setup after reboot or reset.
4. The diameter/radius axis program switching(G10.9) setup is effective to the assigned axis controlled by each path. Which means, if X axis belongs to 2 paths, \$1 and \$2, and it's defined as a radius axis by the parameter; when \$1 is given command G10.9 X1, the X axis will also be applied with diameter axis programming when \$2 executing command with X axis.
5. When the axis is switched to radius axis programming from diameter axis, the block movements will be doubled. Please make sure if the block movements is correct to avoid tool interference or crashes.
6. The system will deactivate the tool compensation temporary when executing diameter/radius program switching (G10.9), return after new movements.
7. This function only affects the linear axis(Pr221~Pr240 axis type set 0)
8. In polar coordinate interpolation mode(G12.1), diameter/radius axis program switching command(G10.9) is illegal, alarm COR-325 would be issued.
9. If given G10.9 X_ first then activates the polar coordinate interpolation function(G12.1), the X axis will follow the setup value of Pr4020 program under polar coordinate interpolation. After deactivated the polar coordinate interpolation function(G13.1), the X axis will return to the initial parameter setup value. Please assign G10.9 X_ again, if needs to return the G10.9 setup value.
10. If giving G10.9 command without any axis assigned or assigned with the value besides 0 and 1, alarm COR-326 would be issued.
11. Tool length/cutter radius compensation, offset and workpiece coordinate offset is determined by the PR28x setup value when ready, switching G10.9 during machining won't change the actual machine compensation amount.

2.15.4 Example

1. Set Y axis as a radius axis with parameters.

```
T0101;
G90 G00 Y0.;      //move to the orientation point, coordinate 0.0.
G01 Y5.;          //Y axis actually moves 5mm, coordinate 5.0.
G10.9 Y1;         //Y axis switched to diameter axis programming, coordinate 10.0.
Y20.;             //Y axis actually moves 5mm, coordinate 20.0.
M30;              //after Reset, Y axis switched to radius axis, coordinate 10.0.
```
2. Set X axis as a diameter axis and Z axis as radius axis with parameters.

```
T0101;
```

```

G90 G00 X0.; //move to the orientation point, coordinate 0.0.
X10. Z10. //X axis actually moves 5mm, Z axis actually moves 10mm, coordinate (10.0,10.0).
G10.9 X0 Z1; //X axis switched to radius axis programming; Z axis switched to diameter axis
programming, coordinate (5.0,20.0).
X40. Z40. //X axis actually moves 35mm, Z axis actually moves 10mm, coordinate (40.0,40.0).
M30; //after Reset, X axis switched to diameter axis, Z axis switched to radius axis, diameter
(80.0,20.0).

```

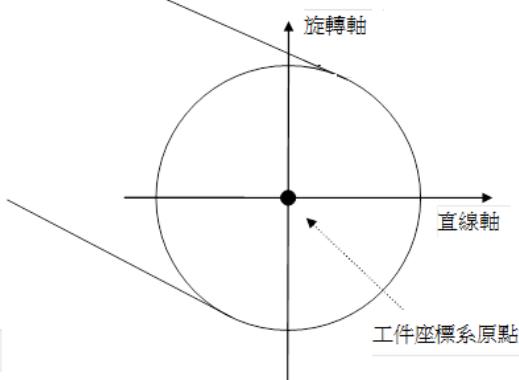
2.16 G12.1/G13.1 : Activate/Deactivate Polar Coordinate Interpolation

Command Form

G12.1 X C; // enable polar coordinate interpolation function ;
... // (linear or circular interpolation command in Cartesian coordinate system,
... // Cartesian coordinate system is composed of linear axes and rotary axes)
G13.1; // disable polar coordinate interpolation function
X : The X offset from the rotation center to program zero.
C : The C offset from the rotation center to program zero.

2.16.1 **Description**

1. The polar coordinate interpolation function is to transform the contour control command in Cartesian coordinate system to a linear axis exercise (tool movements) and a rotary axis exercise (workpiece movements).
2. Polar coordinate interpolation plane. Enable the polar coordinate interpolation function with G12.1 and choose a polar coordinate interpolation plane (shown as below). The polar coordinate interpolation will be completed on the chosen plane.



3. After G12.1, the absolute coordinate of C axis shows negative C axis offset; the absolute coordinate of X is affected by its diameter axis setup, below are the details :
 - a. X as a radius axis and applies the radius axis programming in polar coordinate interpolation. The absolute coordinate of X shows the location after minus the X axis offset.
G0 X50. C90. // absolute coordinate X = 50, C = 90
G12.1 X10. C5. // absolute coordinate X = 50 - 10 = 40, C = 0 - 5 = -5
G13.1 // absolute coordinate X = 50, C = 90
 - b. X as a diameter axis, it'll apply the radius axis programming in the polar coordinate interpolation. The absolute coordinate of X axis will be the location of original coordinate divided by 2 and minus the X axis offset.
G0 X50. C90. // absolute coordinate X = 50, C = 90

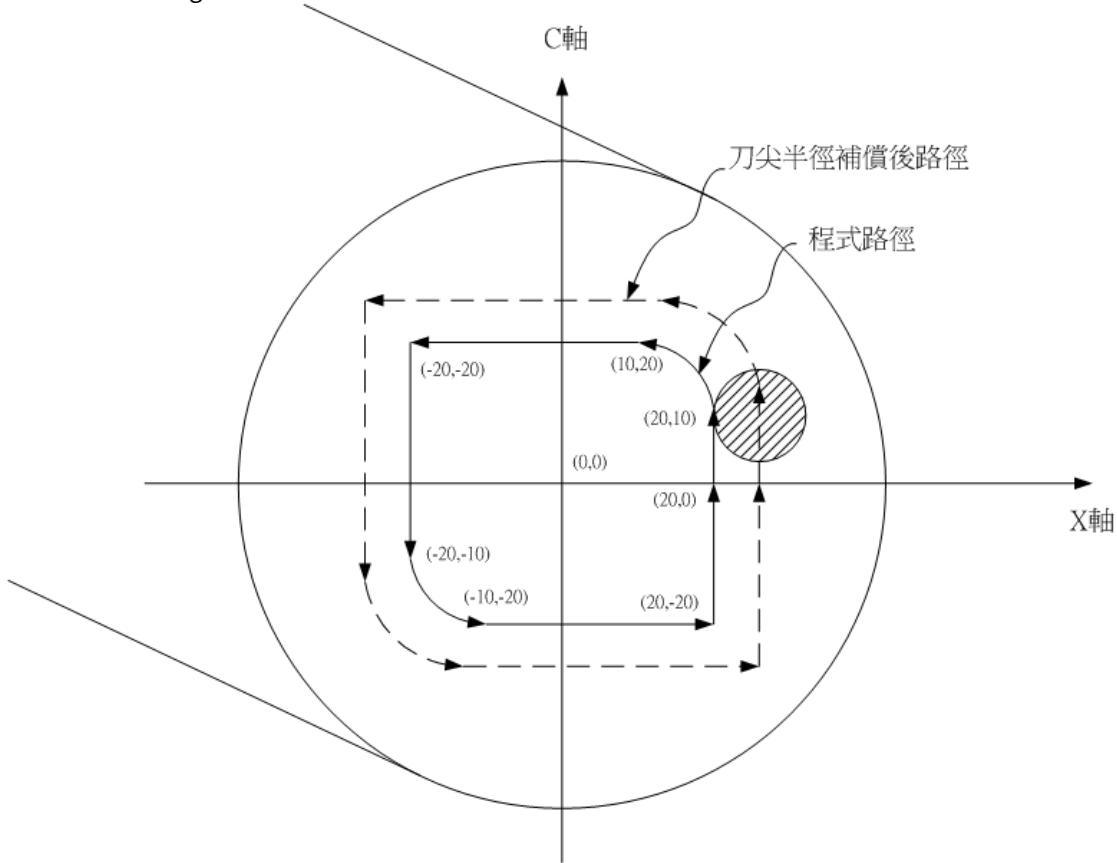
G12.1 X10. C5. // absolute coordinate X = 50/2 - 10 = 15, C = 0 - 5 = -5
 G13.1 // absolute coordinate X = 50, C = 90

2.16.2 Notice

1. The offset argument is valid from version 10.116.11.
2. The polar coordinate interpolation function will be disable after reboot or system reset.
3. G codes below are available in polar coordinate interpolation :
 - G01 linear interpolation
 - G02/G03 circular interpolation (IJR argument is the same as normal syntax)
 - G04 dwell
 - G40/G41/G42 cutter radius compensation
 - G65/G66/G67 call macro
4. After enable polar coordinate interpolation function, the plane selection(G17/G18/G19)(G17, if not customized) will be disabled and specified to G12.1 working plane. After the polar coordinate interpolation system is disable or reset, the system will return to the original working plane specified before the polar coordinate interpolation function is enable.
5. After enable polar coordinate interpolation function, it's unable to change the coordinate system (G50/G52/G53/G54~G59).
6. The polar coordinate interpolation function can't be enable or disable with cutter radius compensation function(G41/G42), it's only applicable when cutter radius compensation function is disable(G40).
7. After enable polar interpolation system, if needed to enable the cutter radius compensation function (G41/G42), please program an extra tool carrying block with 0 movement to ensure the contour correctness.
8. In polar coordinate interpolation mode, it's unable to select the preview mode (PR3815=1) for cutter radius compensation.
9. Program Restart : Do not restart the program section of G12.1., or it might lead to contour error.
10. After switched to polar coordinate, the actions will be planned with hypothetical 0 degree at the instant C axis direction. Therefore, please execute the C axis orientation before executing G12.1 to make sure the continuing feeding angle is the same. (example below)
11. Currently not supporting the function with Z axis not activated, or it might lead to G02/G03 contour error.
12. The polar interpolation function can't be applied with five-axis cutter radius compensation(G43.4/G43.5).
13. In polar coordinate interpolation mode (G12.1), do not apply diameter/radius axis programming switch command (G10.9), or alarm COR-325 will be issued.
14. If given G10.9 X_ command first then enable the polar coordinate interpolation function (G12.1), the X axis will be using radius axis programming. After deactivated the polar coordinate interpolation function (G13.1), the X axis will return to the initial setup value, if wanted to return to the G10.9 setup value X, please specified G10.9 X_ again.
15. If the tool starts moving from the mechanism center of C axis, the moving way might be separated into 2 stages: first the C axis orientation then the X axis movements; it's a proper action, not asynchronous between each axes. The cause of the 2-stage action is the non-linear mechanisms transform of polar coordinate interpolation, a singular point exists at the mechanism center of C axis and the moving performance will be special at the point.
16. This feature doesn't support Multi-Axis Multi-Signal Skip Function(G31.10/G31.11).

2.16.3 Example

1. Without offset argument - radius axis

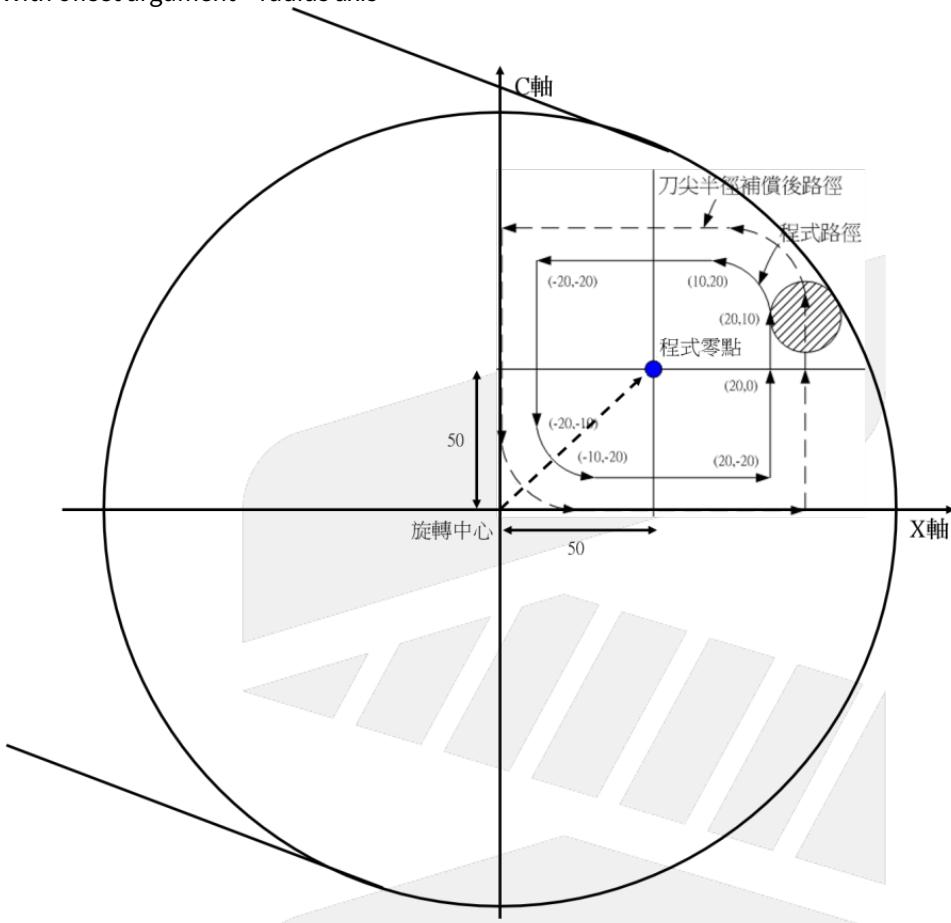


```

T0101;
G00 X110. C0. Z0.; // move to the orientation point
G40 G94;
G12.1; // enable the polar coordinate interpolation
//programming with the X-C plane of Cartesian coordinate
G42 X55. ;// add an extra block with o movement
G01 X20. F100.;
C10. ;
G03 X10. C20. R10. ;
G01 X-20. ;
C-10. ;
G03 X-10. C-20. R10. ;
G01 X20. ;
C0. ;
G40 X55. ;
G13.1; // disable the polar coordinate interpolation
M30

```

2. With offset argument – radius axis



```

T0101
G00 X110. C0. Z0.; // move to orientation point
G40 G94;
G12.1 X50. C50.;// enable the polar coordinate interpolation, eccentric coordinate(50, 50)
//programming with the X-C plane of Cartesian coordinate
G42 X55.;// add a block with 0 movement
G01 X20. F100.;
C10.;
G03 X10. C20. R10.;
G01 X-20.;
C-10.;
G03 X-10. C-20. R10.;
G01 X20.;
C0;
G40 X55.;
G13.1;// disable the coordinate interpolation
M30

```

3. Specified random axis + cutter radius compensation

For G12.1, X axis is the default linear axis and C axis is the default rotary axis. Due to different machine configurations, sometimes it might need to change the linear axis to Y axis, for the situation, please apply [G10 : Programmable Data Input](#) to generate a customized G12.1.

```
G10 L1301 X_C_R_;
X_linear axis ID
C_rotary axis ID
R_1 : Enable / 0 : Disable
```

Example (customized G012001, Y as linear axis & C as rotary axis) :

```
%@MACRO
IF (#1012<>40) THEN
    ALARM( 17 );
END_IF;
IF (#1018=96) THEN
    ALARM( 18 );
END_IF;

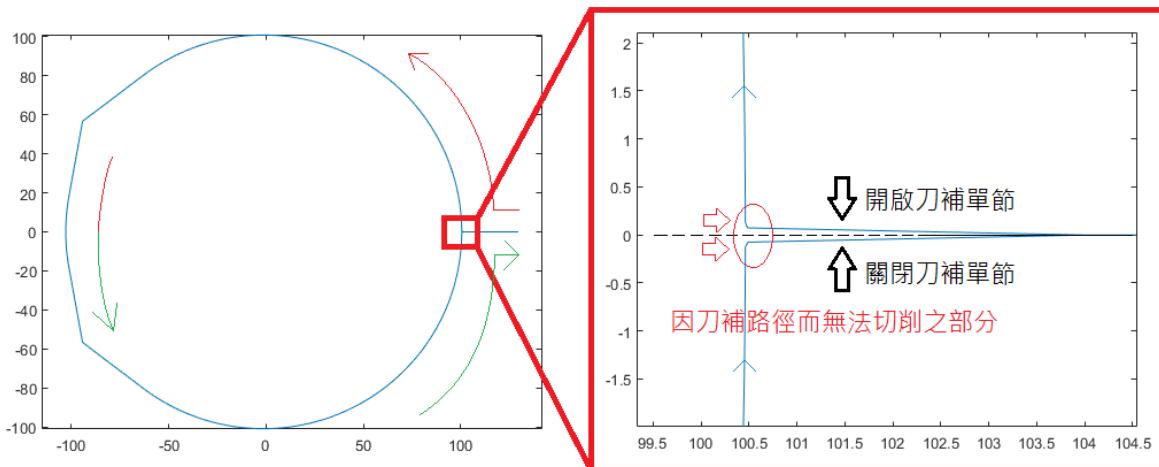
#30:=AXID(Y); // get Y axis ID
#31:=AXID(C); // get C axis ID
// get Y axis diameter/radius programming before G12.1 enable
#34 := ROUND( POW( 2, #30 ) );
#35 := #1814 AND #34;

IF (#25 = #0) THEN
    #25 := 0;
END_IF;
IF (#3 = #0) THEN
    #3 := 0;
END_IF;
IF ((#30=#0) OR (#30<=0) OR (#31=#0) OR (#31<=0)) THEN
    ALARM( 19 );
    M99;
END_IF;

// State Backup
#32:=#1004;
#2048:=#1002;
#2049:=#1008;
// When applying arc commands, cutter radius compensation or polar coordinate commands, please set the
// cutting plane with G17, G18, G19.
G91 G19 Y0 C0;
G94;
G90 G10 L1301 X#30 C#31 I#25 J#3 R1; // Enable polar coordinate interpolation mode and assign to Y-C
IF ( #35 = 0 ) THEN
    Y( #1412 - #25 ) C-#3;
ELSE
    Y( #1412 / 2.0 - #25 ) C-#3;
END_IF;

WAIT();
G#32;
M99;
```

When enable tool compensation under polar coordinate interpolation, some area near the tool entry point might be skipped if set the cutting contour directly. (please refer to the picture below)



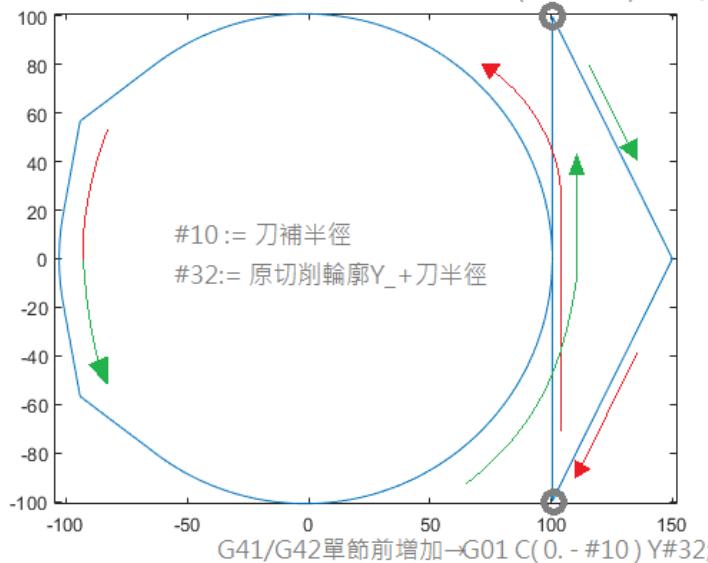
Suggests to design the tool carrying block in the machining file so the tool path can cut in along with the contour. Assume that both the start and finish position commands of the cutting contour are C0., then the simplest organization will be :

Add one moving block before G41/G42 and after G40 respectively to make approaching the **cutting contour** after **cutter radius compensation**. The contour of C axis should move at least the distance of cutter radius according to the contour direction (the direction is decided by the original contour direction).

Please refer to the example below for further details, Y-C polar coordinate + cutter radius compensation (tool carrying block included)

Example (using customized G12.1 + cutter radius compensation)

G40單節後增加→G01 C(0. + #10) Y#32;



%@MACRO // the orange parts can be replaced by the blue MACRO codes

G90 G00X0.Y150. C0. Z0. F1500 ;

M03 S3000;

M08;

G40;

G90 G10 L12 P13 R99.86; // #10 := 99.86; cutter radius 99.86

// G90 G10 L12 P13 R#10; set the cutter radius of tool No.13

G12.1;

// enable customized Y-C polar coordinate interpolation

G90G01 F1600;

G01 C-99.86 Y100.462; // #32 := 0.602 + #10; cutting contour + cutter radius, 99.86 + 0.602 = 100.462

// #G01 C(0. - #10) Y#32; add a tool carrying block, Y_ moves to 100.462 and C moves the distance of cutter radius

G42 D13 C0. Y0.602;

// cutting contour

// =====

G03 C0.464 Y0.383 R0.602;

G03 C0.536 Y-2.952 R2.69;

G03 C-0.536 Y-2.952 R3.0;

G03 C-0.464 Y0.383 R2.69;

G03 C0. Y0.602 R0.602;

// =====

G40;

G01 C99.86 Y100.462; // G01 C(0. + #10) Y#32; add a tool retracting block, C_ moves the distance of cutter radius

// the tool retracting block is still affected by the tool compensation specifications, should be written before G13.1

// =====

G90 G01 C0. Y150.; // back to orientation point

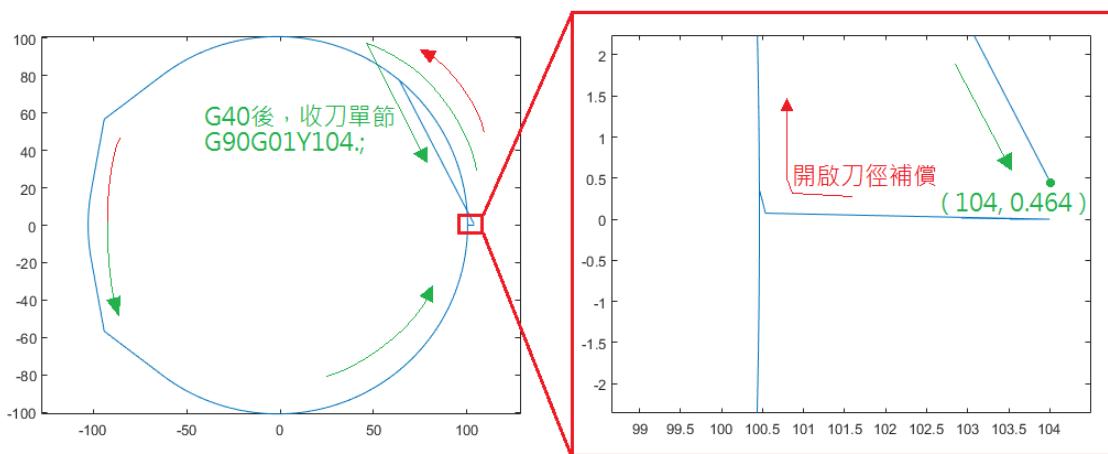
G13.1; // deactivate polar coordinate interpolation

M30;

※ Note : During the tool compensation process, please not to give action commands with working plane declaration, it might trigger a temporary pause of tool compensation action and leads to the wrong moving path.

4. Overcut caused by tool retracting path

For the problem mentioned in point 3 that some area near the tool entry point might be skipped, if wants to move the path to C≠0 with planning repeated paths, please be careful with the path of Y-C during the tool retract. In the example, the command G90 G01 Y104 is given after G40.. For instinct, the contour will be moved to 104. along Y axis, but since it's the tool retracting block and no command about C is given, C will be moving according to the command given by the previous block (G03 C0.464 Y0.383 R0.602;), and the position after disable the tool compensation will be at (104.0, 0.464). From the picture below, it's able to see the overcut caused by tool retracting path moving across the cutting contour. If wants the path to move out along Y direction, the block should be placed before G40. But please note that it's also involved with the turning of tool compensation path so the tool compensation might be functioning improperly (excessive tool radius alarm in this example). Please refer to point 3 for path modification methods.



```
%@MACRO
G90 G01X0.Y104.C0.Z0.F1500 ;
M03 S3000;
M08;
G40;

G90 G10 L12 P13 R99.86; // set up the tool radius of tool No.13
G12.1;

G90G01 F1600;
G42D13 C0. Y0.602 ;
// interpolation contour
//=====
G03 C0.464 Y0.383 R0.602;
G03 C0.536 Y-2.952 R2.69;
G03 C-0.536 Y-2.952 R3.0;
G03 C-0.464 Y0.383 R2.69;
G03 C0. Y0.602 R0.602;
G03 C0.464 Y0.383 R0.602; // repeated contour, cutting the skipped parts due to cutter radius compensation.
//=====
G40; // disable tool compensation
G90 G01 Y104; // tool retracting block, coordinate of C not specified
// retracted to (104.0, 0.464) and caused the overcut
//=====
G13.1; // disable polar coordinate interpolation
M30;
```

2.17 G15/G16 : Polar Coordinates Command Mode

2.17.1 Command Form

G16; //Start polar coordinate mode

G __ X __ Y __

:

:

G15; //Cancel polar coordinate command

} //Polar coordinate command

X: polar coordinate radius

Y: polar coordinate angle(“+” for CW, “-” for CCW)

2.17.2 Description

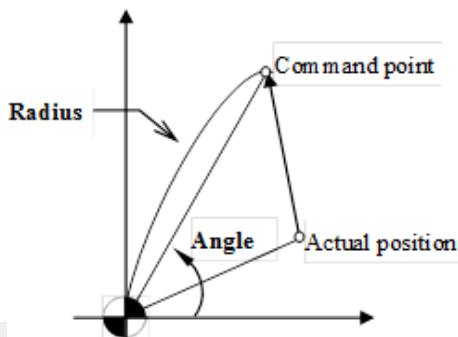
Enable polar coordinate mode in first line, G16 for polar coordinate command enable, G15 for polar coordinate command disable, it can use polar coordinate mode to enter position(radius and angle), G90/G91 can specify in it.

First argument is radius, second argument is angle.

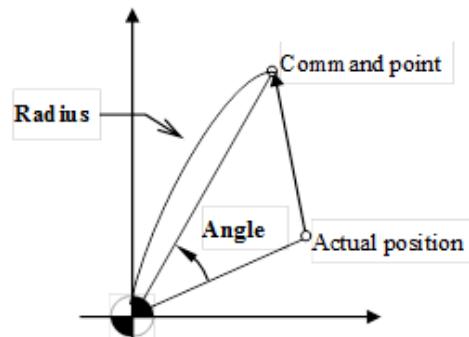
Absolute or increment is specified by G90 or G91, G90 is absolute, G91 is increment, in absolute mode, the increase of radius or angle from origin point; in increment mode, angle or radius total from the last radius or angle.

2.17.3 Notice

- when polar coordinate zero point is the same as working coordinate system

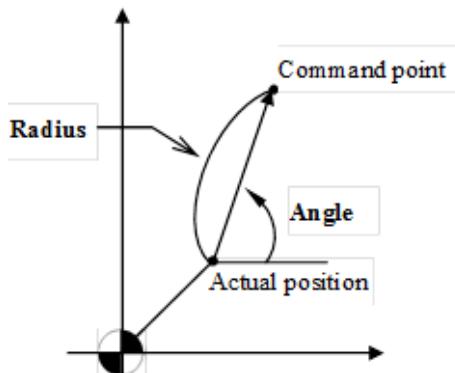


a. When angle is specified with an absolute command

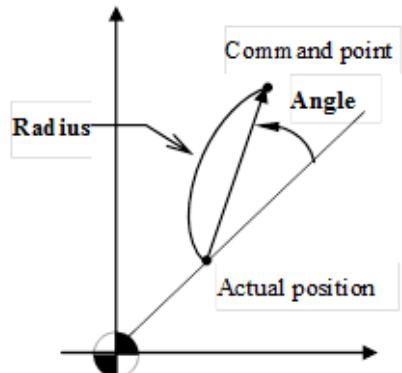


b. when angle is specified with an increment command

2. when polar coordinate zero point is in normal position



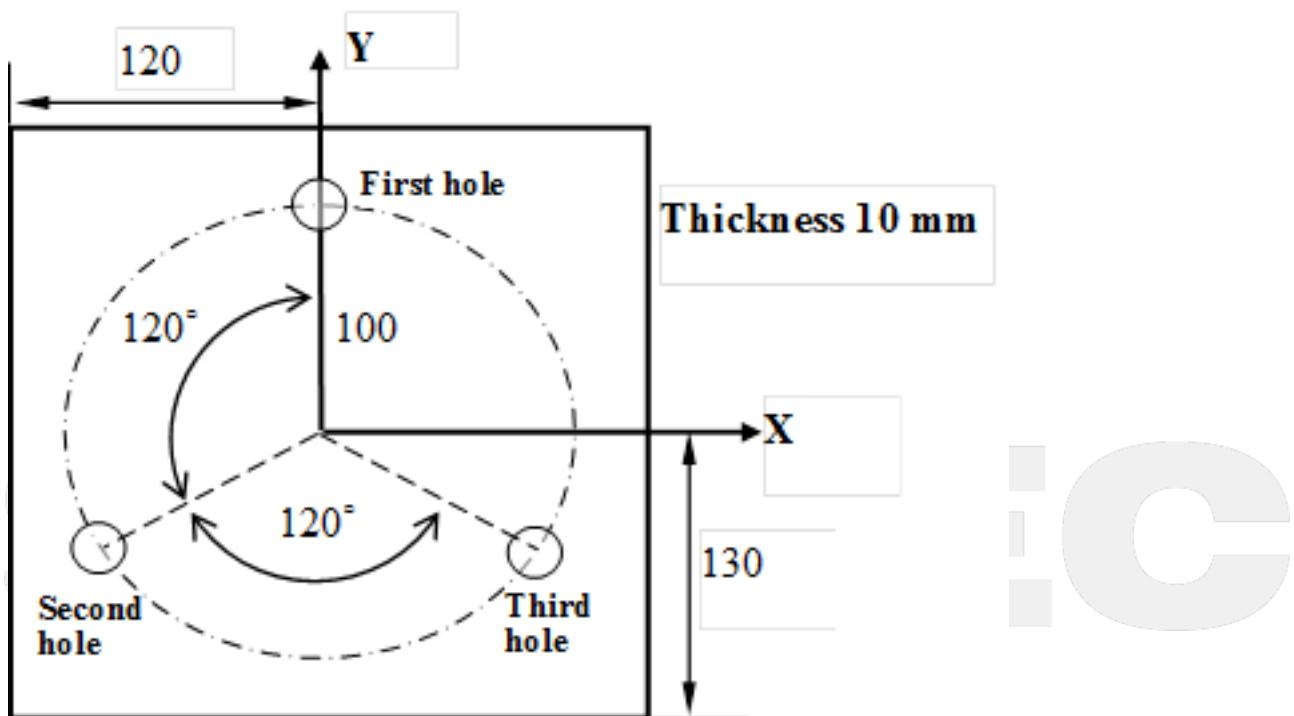
a. When angle is specified
with an absolute command



b. when angle is specified with
an increment command

2.17.4 Example

PIC:



1. Absolute command:
N001 T1 S1000 M03 ;
//NO.1 tool(diameter r10 mm drill), spindle 1000rpm (CW)
N002 G17 G90 G16 ;

```
//X-Y plane, absolute mode, enable polar coordinate mode
N003 G99 G81 Z-12.0 R2.0 F600 K0 ;
//do drilling cycle, depth 12mm, feedrate 600mm/min, back to R point when finish
N004 X100.0 Y90.0 ;
//specified a distance 100mm, angle 90 degree(first hole)
N005 Y210.0 ;
//specified a distance 100mm and angle 210 degree, from the origin point(second hole)
N006 Y330.0 ;
//specified a distance 100mm and angle 330 degree, from the origin point(third hole)
N007 G15 G80 M05 ;
//polar coordinate mode disable, cycle cancel, spindle stop
N008 M30 ; //program ends
```

2. Increment command:

```
N001 T1 S1000 M03 ;
// NO.1 tool(diameter 10 mm drill), spindle 1000rpm (CW)
N002 G17 G90 G16 ;
// X-Y plane, absolute mode, enable polar coordinate mode
N003 G99 G81 Z-12.0 R2.0 F600 K0 ;
// do drilling cycle, depth 12mm, feedrate 600mm/min, back to R point when finish
N004 X100.0 Y90.0 ;
//specified a distance 100mm, angle 90 degree(first hole)
N005 G91 Y120.0 K2 ;
//increment command, angle totals 120 degree from last point (second hole)
N006 Y120.0 ;
//increment command, angle totals 120 degree from last point (third hole)
N007 G15 G80 M05 ;
// polar coordinate mode disable, cycle cancel, spindle stop
N008 M30 ; //program ends
```

2.18 G17/G18/G19 : Plane Selection

2.18.1 Command Form

G17; X-Y plane selection

G18; Z-X plane selection

G19; Y-Z plane selection

2.18.2 Description

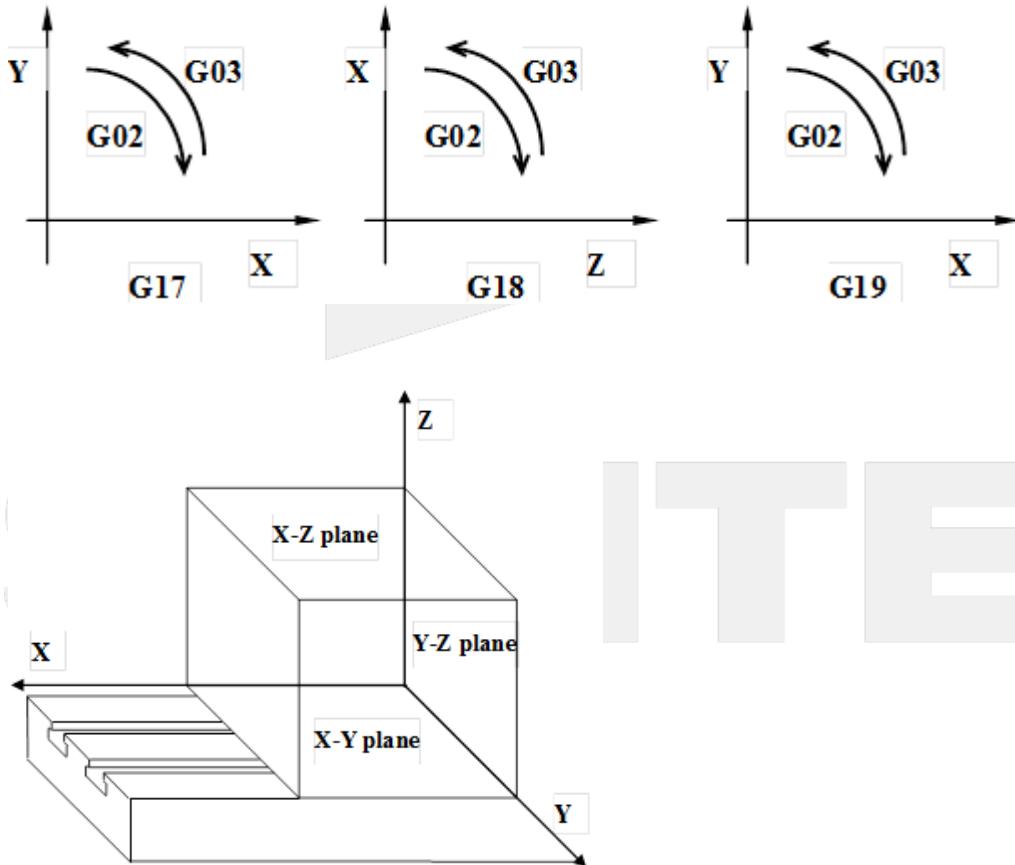
1. When using the arc interpolation, tool radius compensation command or polar coordinate command, the cutting plane must be set with G17, G18, G19 to inform the controller of the machining plane (default is G17).
2. In the X, Y, and Z directions on the cutting plane, the actual corresponding axis is called the geometry axis. The selection rules for the geometry axis are as follows :
 - a. The controller divides the axis into three categories based on the axis name.
 - i. X category : X, X1~X99, U, U1~U99, A, A1~A99.
 - ii. Y category : Y, Y1~Y99, V, V1~V99, B, B1~B99.
 - iii. Z category : Z, Z1~Z99, W, W1~W99, C, C1~C99.

- b. The axial direction of the X category is eligible to be selected as the axis in the X direction; the axial direction of the Y category is eligible to be selected as the axis in the Y direction; the axial direction of the Z category is eligible to be selected as the axis in the Z direction.
- c. In the same category, if there are multiple axes, the order described above will be referred to, and the one former, the more preferred.
- d. If there is no corresponding axis declared in a certain category, then the axis with the smallest axis ID is selected as the geometry axis of the category from the axial direction that is not selected as the geometry axis.
- e. If the number of axes declared by the system is less than three axes, there will be no geometry axis selected for a certain category. In this case, the arc interpolation, tool radius compensation command or polar coordinate command will be limited in use.
 - i. Taking a two-axis lathe (Z, X-axis) as an example, only the G18 work plane can be used.
- f. An axis would not selected as two category of geometry axes at the same time.

2.18.3 **Notice**

1. When using G17 \ G18 \ G19 for machining plane switching, if the axial command is added in the same block, in addition to changing the working plane geometry axis, the movement command will also be generated. Please aware of the action of the machine to avoid danger.

2.18.4 **Example**



2.18.5 Example Description

The known controller parameters are set as follows :

Pr21、22、23、24、25=[1, 2, 3, 4, 5]

Pr321、322、323、324、325=[101, 100, 800, 302, 301]

Therefore, there are five axes in the system named X1, X, V, Z2, and Z1 (axis ID is from small to large)

According to the above rules, the three geometric axes that make up the spatial geometric coordinates are: X, V, Z1.

Example 1 :

G17;

G91 G02 X5. R20. F2000;// After the G17 is specified, the arc interpolation will be displayed on the plane composed of the X and V axes.

Example 2 :

G18;

G91 G02 X5. R20. F2000;// After the G18 is specified, the arc interpolation will be displayed on the plane composed of the Z1 and X axes.

2.18.6 Appendix

附录

2.18.7 Description

1. If the user adds an axial command behind the G17/G18/G19 block, indicating that the user wants to specify the axis as the geometry axis, the priority order of the specified geometry axis is as follows (high to low) :
 - a. In the axial command, the axes of the axis category are X, Y, Z, U, V, and W, which are preferentially according to the axis category. The axis ID is from small to large to the geometric axis (independent of the order of declaration).
 - b. The axial direction of all declared axes is from small to large to the geometric axis required by the working plane (Ex: G17 requires X, Y; G18 requires Z, X).
 - c. The remaining axes in the system are assigned to the geometry axes according to preset selection rules. (not appearing in the axial direction behind the G17/G18/G19 block)

2.18.8 Notice

1. Not Support in 21GA-E or 6GA-E after 10.118.34(included).
2. if Pr3957 set 1, it won't take H command as C geometry axis.

2.18.9 Example Description

The known controller parameters are set as follows :

Pr21、22、23、24、25=[1, 2, 3, 4, 5]

Pr321、322、323、324、325=[100, 200, 302, 301, 303]

Therefore, there are five axial axes named X, Y, Z2, Z1, and Z3, and the axis ID is from small to large.

Example 1 :

```
G17 G01 Y1. Z2 = 10. Z1 = 20. ;
```

The axial command contains three axes of Y, Z2 and Z1.

1. The axis that appears after the G17 block will be preferentially designated as the geometry axis: the Y axis belonging to the Y category is designated as the Y geometry axis; the Z2 axis belonging to the Z category and having the smallest axis ID is designated as the Z geometry axis.
2. G17 requires an X \ Y geometry axis, but there is no X class axis behind the G17 block. Therefore, the Z1 axis with the smallest axis ID is designated as the X geometry axis, and the X axis declared by Pr21 is designated as the geometry axis.

Example 2 :

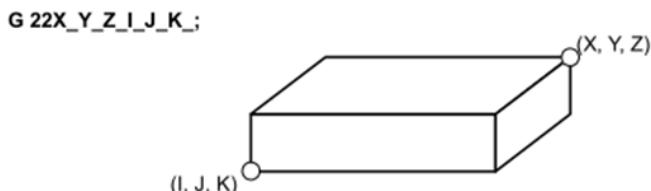
```
G18 G01 Z2 = 10. Z1 = 20. ;
```

The axial command contains two axes Z2 and Z1

1. The axis that appears after the G18 block will be preferentially designated as the geometry axis: the Z2 axis that belongs to the Z category and has the smallest axis ID is designated as the Z geometry axis.
2. The G18 requires a Z \ X geometry axis, but there is no X class axis behind the G18 block. Therefore, the Z1 axis with the smallest axis ID is designated as the X geometry axis.
3. Since the axis that appears behind the G18 block has been specified, the Y axis declared by Pr22 will be designated as the Y geometry axis.

2.19 G22/G23 : Second Software Stroke Limit

2.19.1 Command Form



```
G22X_Y_Z_I_J_K_;  
G22 X_Y_Z_I_J_K_ ; // X_Y_Z_ : positive limit of machine coordinate  
// I_J_K_ : negative limit of machine coordinate  
G23 // disable software stroke limit protection
```

2.19.2 Description

1. This is the new function of 10.116.x, Second Software Stroke Limit is renamed as Third Software Stroke Limit.
2. G22 is able to dynamic activate the second software stroke limit protection and modify the protection region of XYZ axes in the program.
3. There are 3 sets of G22 arguments: X-I, Y-J, Z-K, each set corresponds to the positive and negative limit value of XYZ axes.
4. G23 is used to disable the second software stroke limit protection function.
5. The second software stroke limit value can be set with parameters 2501~2540.

6. It's able to decide the protection region to be the inner or outer side of the setup range via Pr2542.
7. It's able to decide the default state after power On to be G22 or G23 with Pr3838.
8. If there are arguments after G22, the stroke protection region will be set with the machine coordinate assigned by the arguments, but the setup won't modify the parameter values. The chart below listed the protection reference of different command forms:

Program Command	X	Y	Z	Other Axes
G22	Parameter	Parameter	Parameter	Parameter
G22 X_	COR-109 G22 command error, activation failed			
G22 X_I_	Command	Parameter	Parameter	Parameter
G22 X_Y_Z_I_J_K_	Command	Command	Command	Parameter

9. The setup values of a argument set (X & I, Y & J, Z & K) can be opposite, the protection region will be the same. For example: G22 X100. I100. has the same protection region as G22 X200. I100..
10. If the subtraction value of a argument set is 0, the protection won't be activated even the parameters are set. For example: G22X0. I0. means X axis protection won't be activated; G22X10. I0. means the protection region is set X0. ~ X10.. It's the same with the other axes.
11. When giving command G22 to main program (\$1), only the protection of axes of 1st path will be activated, the axes of 2nd path won't be affected, and vice versa.
12. If an axis belongs to multiple paths at the same time, the G22 and G23 commands will only affect the limit protection of this axis under this path.
13. Giving command G22 X_Y_Z_I_J_K_ to the second program, the range will be declared to axes X2, Y2, Z2.

2.19.3 Notice

1. Pressing Reset won't deactivate the G22 protection state, it can only be disable by G23.
2. The protection activating/deactivating function is only valid in the next block of G22/G23 command.
3. The positive limit should be always bigger than the negative limit for the second software stroke protection(Pr2501~), or the protection won't be provided.
4. Please apply versions after 10.116.0
P.S. Please refer to 软体行程极限应用手册 for more details.

2.20 G28 : Return to Reference Point

2.20.1 Command Form

G28 X_Y_Z_ ;

X, Y, Z: mid-point position (absolute value in G90 mode, increment value in G91 mode)

2.20.2 Description

G28 is used to return to reference point or return to origin point. To avoid tool crack, it will move(G00) to the safe mid-point assigned by users first then return to machine origin point.

Note

1. The command is usually applied to auto-tool changing. Therefore, please deactivate the tool compensation function before running G28 for safety. Also, please note that the XYZ arguments are corresponded to the program coordinate.
2. If the axis type (parameter 221~236) is set to be rotary axis, please refer to Pr221~236 "Axial type" for setups.

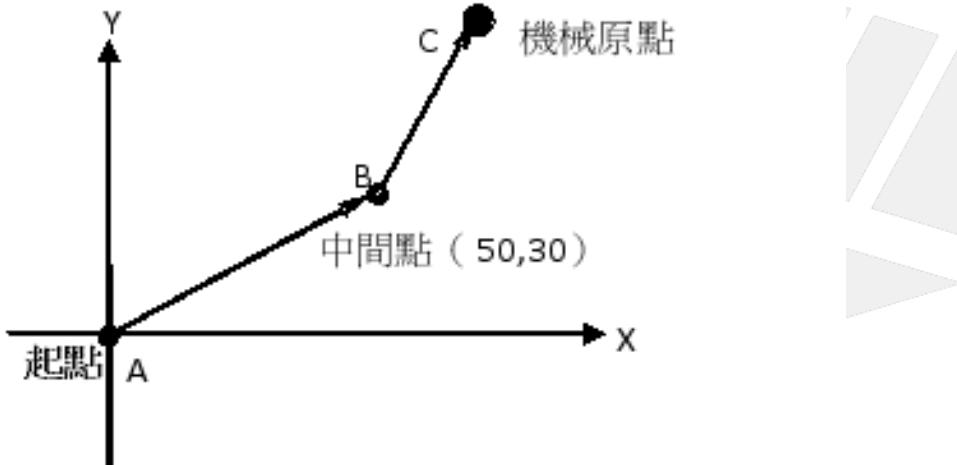
2.20.3 Limitations

1. G28 function only supports axes named X, Y, Z, U, V, W, A, B, C, X1, Y1, Z1, X2, Y2, Z2, the return action won't be executed if axes named with other names.
2. If apply G28 function to axes with other names (ex: X3, X4), G0028 can be customized. Add Xn, Yn or Zn axial command processes.

Example

Problem 1:

G90 G28 X50.0 Y30.0; //A->B->C, mid-point(50,30)



Problem 2:

G28 X0; //only X axis return to reference point
 G28 Y0; //only Y axis return to reference point
 G28 Z0; //only Z axis return to reference point

2.21 G29 : Return From Reference Point

2.21.1 Command Form

G29 X_Y_Z_;

X, Y, Z: specified point ; (absolute value in G90 mode, increment value in G91 mode)

2.21.2 **Description**

G29 command can make the tool rapidly moves from reference point through mid-point to the specified point base on G28 applied. Notice that G29 can't be applied alone because G29 does not specify the mid-point, it refers to the mid-point from G28. Therefore, G28 must be executed first before executing G29

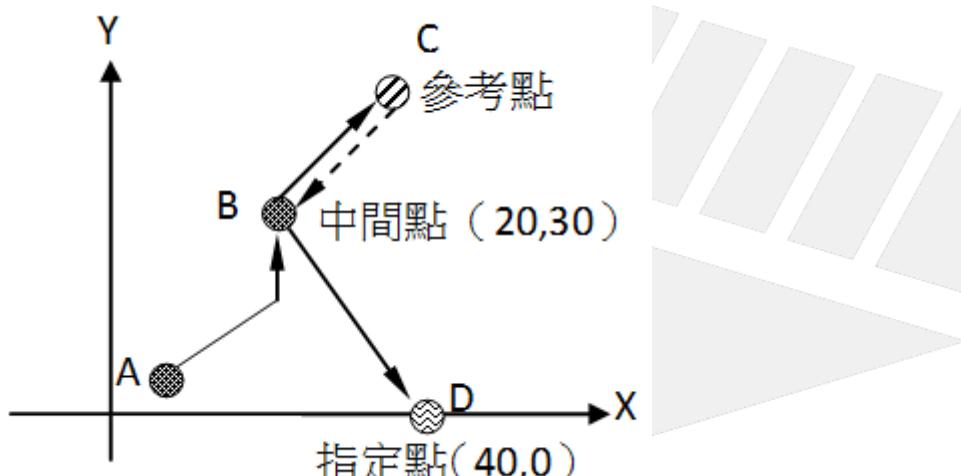
Under the absolute command (G90), the command content is the absolute coordinate; under G91, it is the increment distance from mid-point to specified point.

Limitations

1. G29 function only supports the axes named with X, Y, Z, U, V, W, A, B, C, X1, Y1, Z1, X2, Y2, Z2, the return action won't be executed if axes named with other names.
2. If apply G29 function to axes with other names (ex: X3, X4), G0029 can be customize. Add Xn, Yn or Zn axial command processes.

Example

PIC:



1. Absolute command:

```
N001 G90 G28 X20.0 Y30.0;
//A>B>C, mid-point(20,30), in absolute command mode
N002 M06;//change tool
N003 G29 X40.0 Y0.0;
//C>B>D, the command content is the absolute coordinate
```

2. Increment command:

```
N001 G91 G28 X20.0 Y30.0;
//A>B>C, mid-point(20,30), in increment command mode
N002 M06;//change tool
N003 G29 X20.0 Y-30.0;
//C>B>D, the command content is the increment value from mid-point to the target point
```

2.22 G30 : Return From Specified Reference Point

2.22.1 Command Form

G30 Pn X_Y_Z_;

X, Y, Z: mid-point coordinates; (absolute value under G90, increment value under G91)

Pn: Specified reference point (Pr2801 ~ Pr2860)

P1: Machine zero point ;

P2: Second reference point ;

P_: default is P2 if omitted

2.22.2 Description

G30 Return From Specified Reference Point command, usually being applied when the position of auto-tool changing is not the same with machine zero point. The action moves from current point to the user-defined safe mid-point with G00, then return to the specified reference point (Pr2801~Pr2860).

2.22.3 Notice

The command is usually applied in auto-tool change, for safety concerns, please deactivate the tool compensation function before executing G30. Also, please note that the XYZ arguments are corresponded to program coordinate but the final location (reference point) is corresponded to mechanical coordinate.

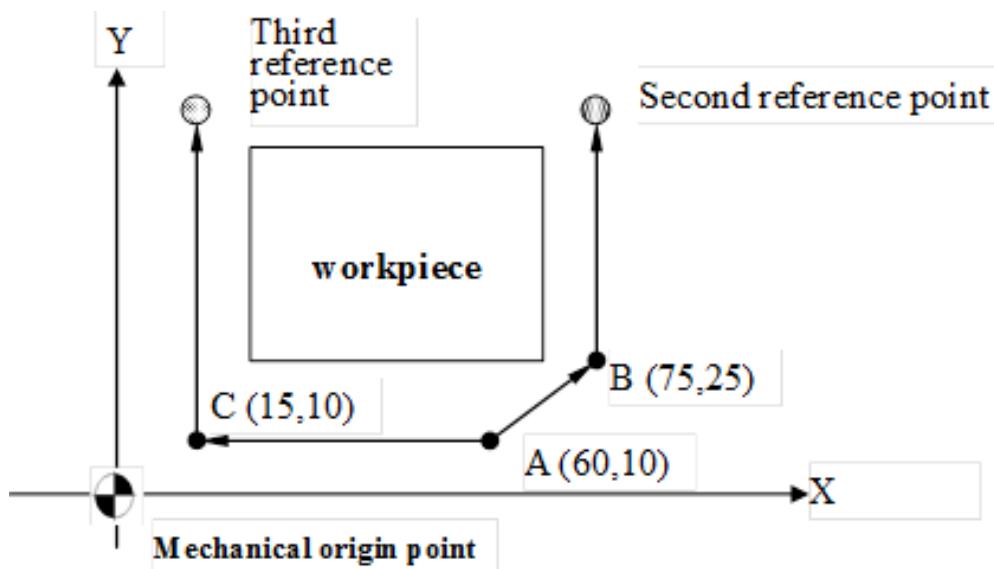
2.22.4 Limitations

1. G30 function only supports axes named with X, Y, Z, U, V, W, A, B, C, X1, Y1, Z1, X2, Y2, Z2, the return action won't be executed if axes named with other names.
2. If apply G30 function to axes with other names (ex: X3, X4), G0030 can be customized. Add Xn, Yn or Zn axial command processes.

2.22.5 Example

PIC:





Program description: presume the tool is at point A(60,10)

1. moving to second reference point
G30 P2 X75.0 Y25.0 ; //A>B> 2nd reference point
2. moving to third reference point
G30 P3 X15.0 Y10.0 ; //A>C> 3rd reference point

2.23 G31 : Skip Function

2.23.1 Command Form

G31 X_ Y_ Z_ F_ Q_ P_;

X, Y, Z: Specified Point

F: Feedrate

Q: Skip source reference

P: Deceleration time (ms)

2.23.2 Description

1. If Q argument is not specified, corresponds to C62.
2. The signal source of **Q101~Q132** is C-bit, corresponds to **C101~C132** respectively.
3. The signal source of **Q201~Q218** is the **external signal (EXT) source** of serial drivers, it detect the signal of the 1st ~ 18th axis respectively. The supporting serial drivers are listed below.

Supporting Serial Drivers	External Signal (EXT) Source
M2	EXT1
M3	EXT1

Supporting Serial Drivers	External Signal (EXT) Source
RTEX	EXT1

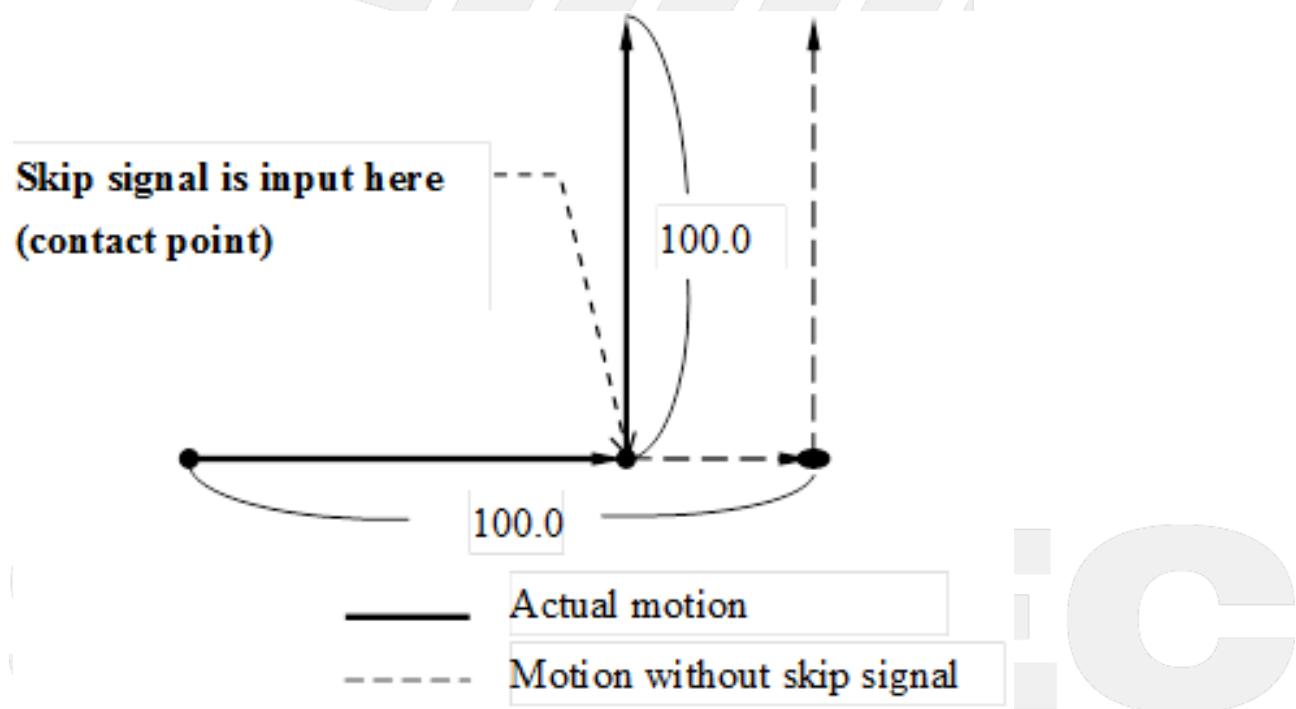
Note: **Syntec M2 doesn't support**

4. When P argument is not specified, deceleration function missing, the command will be suspended.
5. When specified **P0**, deceleration function is missing, the command will be suspended.
6. when P argument is specified, the deceleration according to the planned deceleration time. The program will stop at the block if deceleration time given is not enough.
7. If the final point location of G31 is too close to the skip signal triggering location, it might occasionally lead to the situations that the PLC scanned the C-bit signal after G31 is finished. In the situations, although the signal is triggered, but it's too late for G31 to execute the skip. Take the tool measuring action as example, when the situation happened, it's better to move the G31 block deeper to avoid the situation.
8. The P argument should be bigger than 0 or an integer, or alarm COR-64 will issued.

Note : If the signal source is C62, C101~ or external signal (EXT) source of serial drivers, please contact the machine manufacturer for wiring connection.

2.23.3 Example

Example 1: Incremental command(G91)

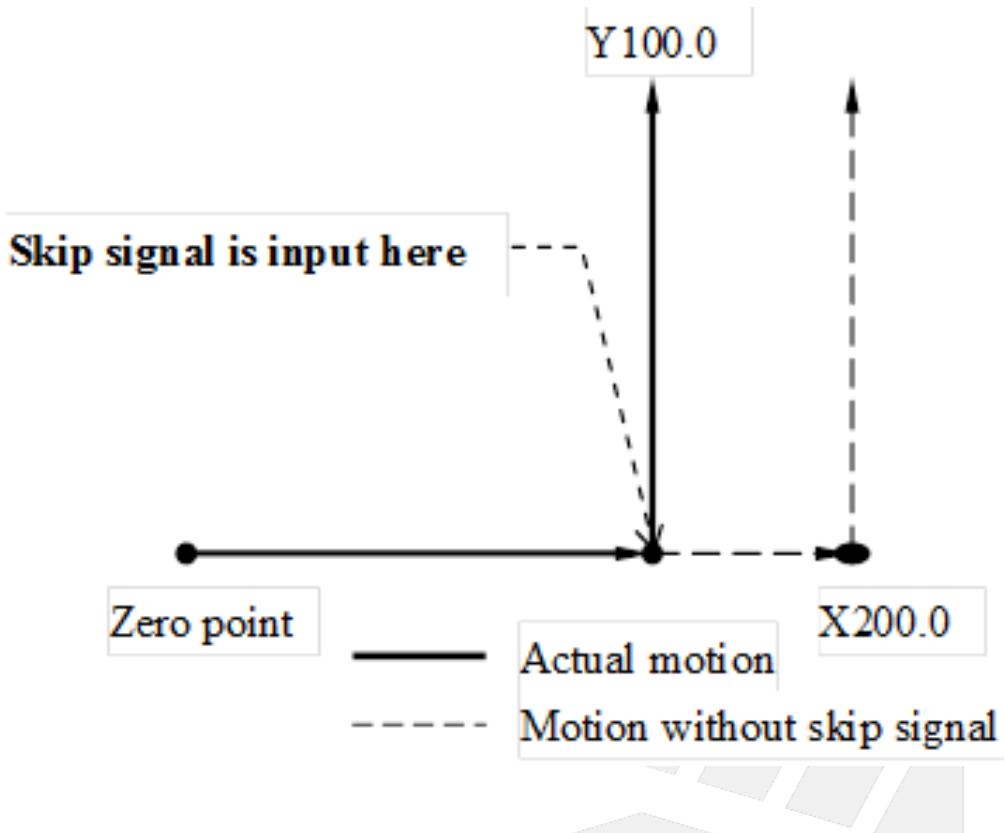


Program description:

N001 G31 G91 X100.0 F100; //original contour until run into impede

N002 Y100.0; //not waiting for the completion of former block, take the contact point as relative coordinate and change the contour to specified position

Example 2: absolute command for single axis(G90)

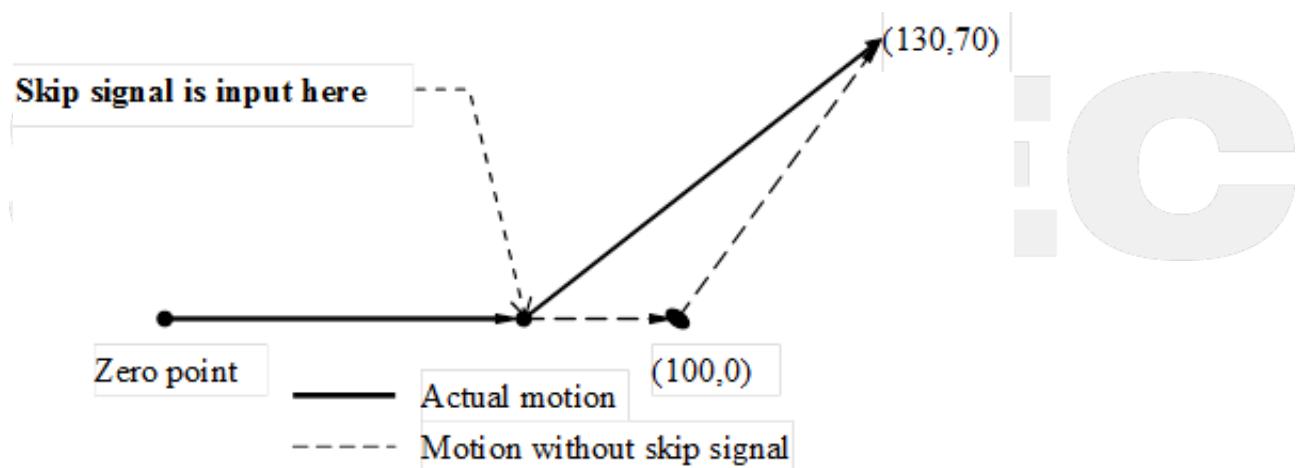


Program description:

```
N001 G31 G90 X200.0 F100; //original contour until run into impede
```

```
N002 Y100.0; //not waiting for the completion of former block, take the zero point as relative coordinate and change the contour to specified position
```

Example 3: absolute command for 2 axes(G90)



Program description:

N001 G31 G90 X100.0 F1000; //original contour until run into impede

N002 X130.0 Y70.0; //not waiting for the completion of former block, take the origin point as relative coordinate and change the contour to specified position

2.24 G31.10/G31.11 : Multi-Axis Multi-Signal Skip Function

2.24.1 Command Form

G31.10 X_ Y_ Z_ F_ Q_ P_

Set multi-axis multi-signal skip function

X, Y, Z: Specified Position

F: Feedrate

Q: Skip source reference

P: Deceleration time (ms)

G31.11

Execute multi-axis multi-signal skip function

2.24.2 Description

1. A multi-axis multi-signal skip function is composed of one G31.10 at least and one G31.11. Set first then execute.
Both commands should be used in a multi-axis multi-signal skip function, and it is not allowed to insert other commands in between.
 - a. Set multi-axis multi-signal skip function(G31.10) :
 - i. Supports up to six G31.10 commands in a set of multi-axis multi-signal skip function. Skip source and deceleration time can be different in each G31.10 command.
 - ii. In each G31.10, every axis specified in this command will move, stop and skip at the same time.
 - iii. If the specified axis is the virtual axis set by G10 L801, all settings of F, P and Q will be applied to corresponding axis.
 - b. Execute multi-axis multi-signal skip function(G31.11) :
Execute multi-axis multi-signal skip function with previous settings of G31.10.
When the skip signal is triggered, the axis corresponding to the skip source skips.
2. The unit of position can be mm or inch, depends on which unit setup is used(G70/G71).
3. Feedrate F :
 - a. Previous feedrate will be considered while F argument is not specified.
 - b. unit :
 - mm/min(inch/min) in G94.
 - mm/rev(inch/rev) in G95.
4. Skip source reference Q :
 - a. C62 is the default skip source while Q argument is not specified
 - b. The signal source of **Q101~Q132** is C-bit, corresponds to **C101~C132** respectively.

- c. The signal source of **Q201~Q218** is the **external signal (EXT) source** of serial drivers, it detect the signal of the 1st ~ 18th axis respectively. The supporting serial drivers are listed below.

Supporting Serial Drivers	External Signal (EXT) Source
M2	EXT1
M3	EXT1
RTEX	EXT1

Note: Syntec M2 doesn't support.

5. Deceleration time P :

- a. When P argument is not specified or P0 is given, there is no deceleration and the command will be suspended directly.
- b. When P argument is given, deceleration time will be considered in deceleration planning. Axis reaches the position specified by G31.10 when deceleration time is so long that the brake distance exceeds the destination of this block.

2.24.3 **Notice**

1. #1361~#1378, #1441~#1458, #1608 are set to 0 when CNC just powered on, RESET, system executes G31 or G31.11 or G28.1 again.
To avoid showing the previous escape location before skip signal comes in and causes misjudgment.
2. If the final point location of G31.10 & G31.11 is too close to the skip signal triggering location, it might occasionally lead to the situations that the PLC scanned the C-bit signal after G31.11 is finished. In the situations, although the signal is triggered, but it's too late for G31.11 to execute the skip. Take the tool measuring action as example, when the situation happened, it's better to move the G31.10 & G31.11 block deeper to avoid the situation.
3. The P argument should be bigger than 0 or an integer, or alarm COR-064 will issued.
4. Alarm COR-362 issues when only G31.10 or G31.11 command is given, or inserting other commands between G31.10 and G31.11.
5. Alarm COR-362 issues when the same axis is used in different G31.10.
6. Alarm COR-362 issues when G31.10 is given more than 6 times in a multi-axis multi-signal skip function.
7. The functions below are not supported in multi-axis multi-signal skip function :
 - a. G5.1 (Path Smoothing)
 - b. G12.1/G13.1 (Activate/Deactivate Polar Coordinate Interpolation)
 - c. G15/G16 (Polar Coordinates Command Mode)
 - d. G40/G41/G42 (Cutter Radius Compensation)
 - e. G43.4/G43.5 (Rotate Tool Center Point (RTCP) Type1 & 2)
 - f. G10 L16 (Virtual circle radius)
8. While executing multi-axis multi-signal skip function, F(command) displays the latest F command given by NC file, F(actual) displays the value of current resultant velocity. Therefore, F(actual) may be greater than F(command). The block feedrate command of G31.11 while interpolation can be obtained by using variable K62.

example

sample code

```

G90 G71
G31.10 Z1=10. F300. Q101 P100 // set Z1 goes to 10 by F300, the skip source
is C101, deceleration time is 100 ms
G31.10 Z2=20. F400. Q102 P100 // set Z2 goes to 20 by F400, the skip source
is C102, deceleration time is 100 ms
G31.11 // execute multi-axis multi-signal skip
function
M30

```

When Z1 and Z2 reach F300 and F400 respectively, the display result :

F(command) : 400 mm/min

F(actual) : 500 mm/min

The block feedrate command of G31.11 while interpolation can be obtained by variable K62.

The actual compound feedrate and axis feedrate can be obtained by R700 and R701~718 :

- R700 : Actual compound feedrate command, unit: LIU/min.
- R701~718 : Velocity of each axis, Servo On mode: according to command value; Servo Off mode: according to feedback value, unit : BLU/min.

R700 : 500 IU/min = 500000 LIU/min

R701 : 300 IU/min = 300000 BLU/min (if Z1 is the first axis)

R702 : 400 IU/min = 400000 BLU/min (if Z2 is the second axis)

9. Valid version : after 10.118.40G, 10.118.44 (included).

2.24.4 Example

Example 1: each axis moves and skips with the settings of G31.10.(the feedrate of all G31.10 refers the previous one)

example 1

```

G90
F100.
G31.10 Z1=10. Q101 P100 // set Z1 to move to 10 by F100, skip source is C101, set
deceleration time to 100 ms
G31.10 Z2=20. Q102 P100 // set Z2 to move to 20 by F100, skip source is C102, set
deceleration time to 100 ms
G31.11 // execute multi-axis multi-signal skip function
M30

```

The value of #1608 when all skip source are triggered.

There are axes: X, Y, Z1, Z2, Z3, Z4, the corresponding Pr21~ and Pr321~ are listed below.

When all skip source are triggered, the value of each bit of #1608 is :

bit 0 : 0. (supports G31 only)

bit 1~18 :

Command	G31.10 & G31.11					
Axis corresponding axis card port number(Pr21~)	Pr21	Pr22	Pr23	Pr24	Pr25	Pr26
Axis name(Pr321~)	X	Y	Z ₁	Z ₂	Z ₃	Z ₄
bit	1	2	3	4	5	6
value	0	0	1	1	0	0

Therefore, the value of #1608 is $2^3 + 2^4 = 24$.

Example 2: each axis moves and skips with the settings of G31.10.(the second feedrate of G31.10 refers the one of the first G31.10)

example 2

```

G90
G31.10 Z1=10. F100. Q101 P100    // set Z1 to move to 10 by F100, skip source is C101,
set deceleration time to 100 ms
G31.10 Z2=20. Q102 P100          // set Z2 to move to 20 by F100, skip source is C102,
set deceleration time to 100 ms
G31.11                           // execute multi-axis multi-signal skip function
M30

```

The value of #1608 when all skip source are triggered.

There are axes: X, Y, Z1, Z2, Z3, Z4, the corresponding Pr21~ and Pr321~ are listed below.

When all skip source are triggered, the value of each bit of #1608 is $2^3 + 2^4 = 24$.

Command	G31.10 & G31.11					
Axis corresponding axis card port number(Pr21~)	Pr21	Pr22	Pr23	Pr24	Pr25	Pr26
Axis name(Pr321~)	X	Y	Z ₁	Z ₂	Z ₃	Z ₄

Example 3: use multi-axis multi-signal skip function with virtual axis setting

When using G10 L801, all axes are considered as the same axis. Therefore, all axes apply the same skip function setting.

example 3

```

G90
G10 L801 P300 Q0      // cancel virtual axis Z
G10 L801 P300 Q301    // virtual axis Z corresponding to Z1
G10 L801 P300 Q302    // virtual axis Z corresponding to Z2
G10 L801 P300 Q303    // virtual axis Z corresponding to Z3

G31.10 Z10. F100 Q101  // set virtual axis Z to move to 10 by F100. i.e., Z1, Z2 and
Z3 move to 10 by resultant velocity F100
                                // skip source is C101 and there is no deceleration time
G31.11                      // execute multi-axis multi-signal skip function
M30

```

The value of #1608 when all skip source are triggered.

There are axes: X, Y, Z1, Z2, Z3, Z4, the corresponding Pr21~ and Pr321~ are listed below.

When all skip source are triggered, the value of each bit of #1608 is $2^3+2^4+2^5=56$.

Command	G31.10 & G31.11					
Axis corresponding axis card port number(Pr21~)	Pr21	Pr22	Pr23	Pr24	Pr25	Pr26
Axis name(Pr321~)	X	Y	Z ₁	Z ₂	Z ₃	Z ₄

Example 4: Z1 and Z2 move simultaneously, Z3 moves independently.

example 4

```

G90
G31.10 Z1=10. Z2=20. F100 Q101  // Z1 and Z2 move to 10 and 20 respectively, the
resultant velocity is 100
                                // skip source is C101 and there is no deceleration
time
G31.10 Z3=30. Q103            // set Z3 to move to 30 by F100, skip source is C101
and there is no deceleration time
G31.11                      // execute multi-axis multi-signal skip function
M30

```

The value of #1608 when all skip source are triggered.

There are axes: X, Y, Z1, Z2, Z3, Z4, the corresponding Pr21~ and Pr321~ are listed below.

When all skip source are triggered, the value of each bit of #1608 is $2^3+2^4+2^5=56$.

Command	G31.10 & G31.11					
Axis corresponding axis card port number(Pr21~)	Pr21	Pr22	Pr23	Pr24	Pr25	Pr26
Axis name(Pr321~)	X	Y	Z ₁	Z ₂	Z ₃	Z ₄

2.25 G33 : Thread Cutting

2.25.1 Command Form

G33 Z_ F_ ;

Z:

Absolute command (G90), coordinate of Z axis for cutting end;

Incremental command (G91), for axial moving distance of the cutting thread;

F: the length of thread lead (0.01mm);

2.25.2 Description

When the spindle starts rotation, the command controls the tool moving up and down along the Z axis repeatedly to finish Thread Cutting. Since the spindle inertial lag occurs at the beginning and the end of cutting, the process should be extended a little bit. And the limitation of spindle rotation speed during Thread Cutting :

$$1 \leq \text{spindle speed}(R) \leq \frac{\text{Max feedrate}}{\text{thread lead}}$$

R: spindle rotation speed(rpm)

Thread lead(F): mm or inch

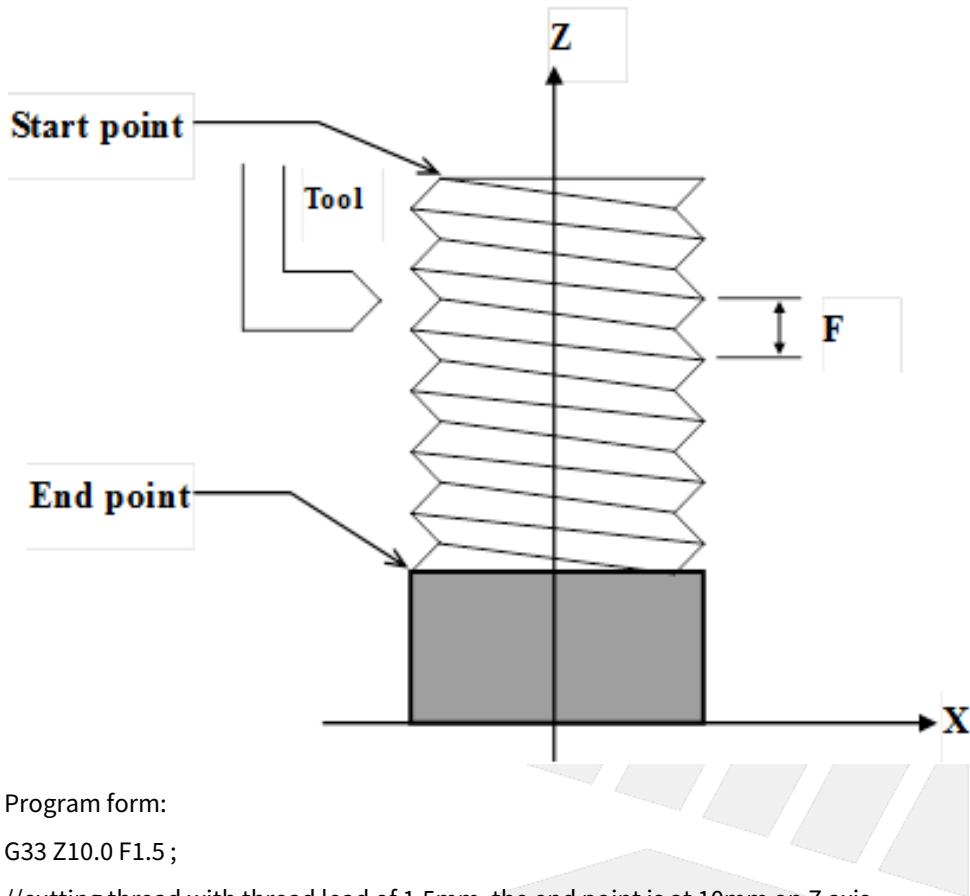
Feedrate: mm/min or inch/min

2.25.3 Note

1. The maximum feedrate can be set by Pr405.
2. Acceleration time of thread cutting can be set by Pr409.

2.25.4 Example

PIC:



2.26 G37 : Automatic Tool Presetting and Measuring I

2.26.1 Command Form

G37 Z_ [R_] [D_] [F_] [P_] ;

Z: Measuring position. Absolute command in program coordinate.

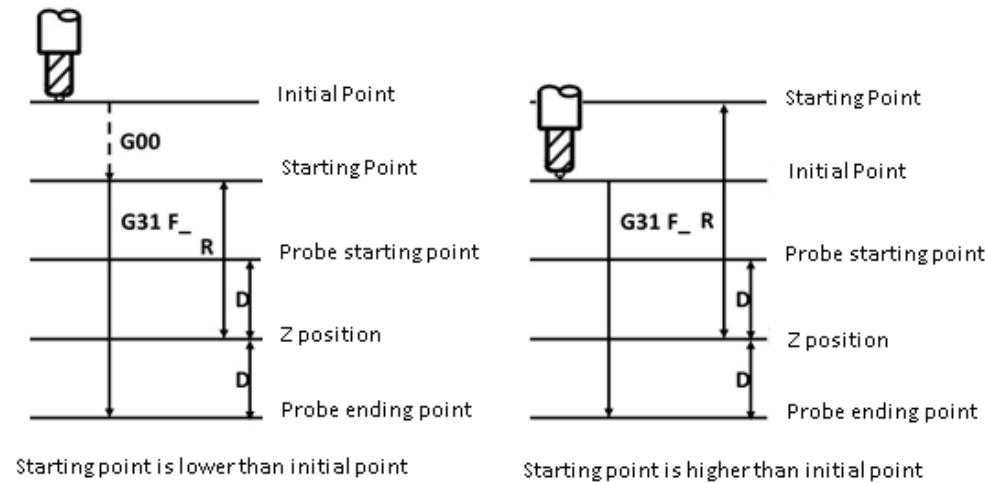
R : Measuring distance, incremental value respective to Z position. The system default value will be decided by Pr4055, if not specified.

D : Allowed overtravel distance for signal triggering, incremental value respective to Z position. The system default value will be decided by Pr4056, if not specified.

F : Measuring speed. The system default value will be decided by Pr4057, if not specified.

P : Reference point P. The system default value will be decided by Pr4058, if not specified. If Pr4058 is not 1~4, the returning action of G30 won't be executed.

2.26.2 Description



Tool presetting and measuring steps :

1. Each axis returns to reference point P with G30. (won't be executed if P is not specified)
2. Z axis moves to the starting point of measurement with G00.
3. Z axis moves to the probe starting point of allowed overtravel distance with G31 F_
4. Z axis moves to end point of allowed overtravel distance with G31 F_, and begins the tool length measurement.
5. After the tool length measurement is completed, the result will be filled in to the assigned tool number on tool length table automatically.

2.26.3 Note

1. Available version starts from:
-SUPER/10s/20s : 10.116.10C
-11s/21s : 2.2.3
2. Please install the tool presetting and measuring machine before executing the function. After the installation, it's recommended to set the XY machine coordinate to the machine coordinate of reference point P (Pr2801~2860). Thus, for future G37 Pn commands in the program, the tool will automatically move to the correct location before presetting and measuring.
3. The interpolation override within the measuring distance is defined as 100% during the tool presetting and measuring process.
4. The MPG offset of Z axis will be reset after the tool presetting and measuring is completed.
5. [MAR-330_Z min. coordinate set error alarm!] would issued if Z argument is not specified.
6. [MAR-333_Z start point error alarm!] would issued if the current location is lower than Z (measuring location)
7. [MAR-334_without issue H code before G code tool length measurement] would issued if the tool number is not specified by H before the tool measurement.
8. [MAR-335_measuring signal triggered beyond allowed distance] would issued if the tool measuring signal is triggered while Z axis is moving to the starting point of allowed overtravel distance with F.
9. [MAR-336_measuring location setup error, measuring signal not triggered] would issued if the tool measuring signal is not triggered after Z axis reached the end point of allowed overtravel distance with F.

2.26.4 Program form:

```

G30 P2 X75.0 Y25.0; // return to reference point 2
M06 T1;           // change to Tool No.1
H1;               // define the tool length compensation H1 after measurement
G37 Z-150;        // move Z axis to -150. and start the tool presetting and measuring
M06 T2;           // change to Tool No.2
H2;               // define the tool length compensation H2 after measurement
G37 Z-150;        // move Z axis to -150. and start the tool presetting and measuring
H3;               // define the tool length compensation H3 after measurement
G37 P3 Z-200.;   // return to reference point 3, move Z axis to -200. and start
                  // the tool presetting and measuring

```

2.27 G37.2 : Automatic Tool Presetting and Measuring II

2.27.1 Command Form

G37.2 K_ [Z_] [R_] [F_] [P_] [Q_]; (suitable for single tool with multiple workpieces; the tool length will be compensated to the specified coordinate system)

K : The coordinate system to be compensated by an amount of the tool length

0	External Offset
1~6	G54P1(G54)~G54P6(G59) Offset of workpiece coordinate system
7~15	G54P7(G59.1)~G54P15(G59.9) Offset of workpiece coordinate system
16~100	G54P16~G54P100 Offset of workpiece coordinate system

Z: Measuring position. Absolute command in program coordinate.

R : Measuring distance, incremental value respective to Z position. The system default value will be decided by Pr4055, if not specified.

F : Measuring speed. The system default value will be decided by Pr4057, if not specified.

P : Reference point P. The system default value will be decided by Pr4058, if not specified. If Pr4058 is not 1~4, the returning action of G30 won't be executed.

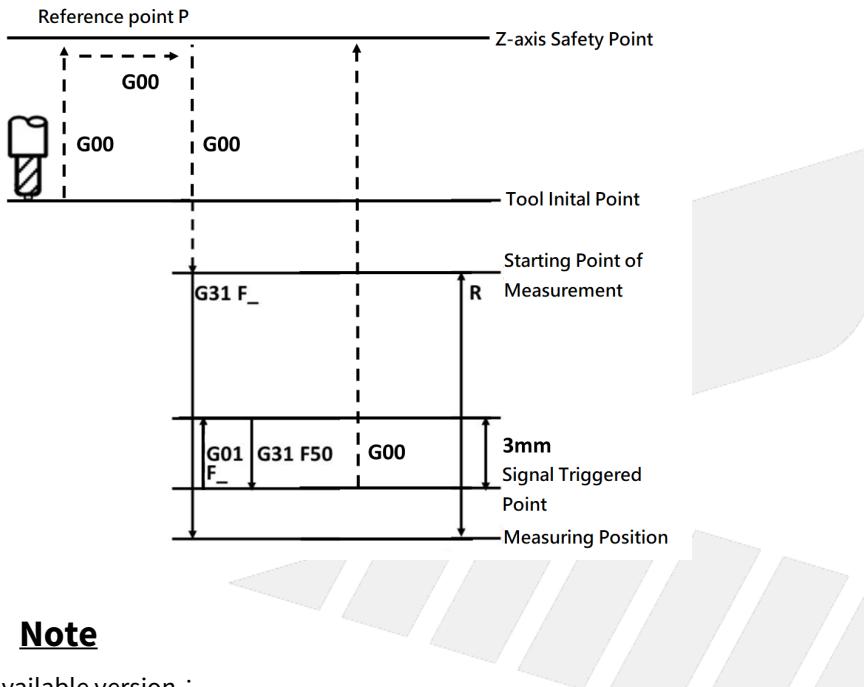
Q : Safe point: If it's not specified, the system default safety point is the machine zero point.

2.27.2 Description

Tool presetting and measuring process :

1. Z axis being pulled back to the safety point or mechanical zero point in mode G53.
2. Each axis returns to the XY axis location of reference point P in mode G30 (won't be executed if P argument is not specified).
3. M90 activate the tool air blast function.
4. Z axis moves to the starting point of measurement with G00 speed.
5. M91 deactivate the tool air blast function.
6. Z axis moves to the measuring location with G31 F_ speed and starts the measurement.
7. If the measuring signal is triggered in step 6, Z axis will retract 3mm with G01 speed.

8. Z axis moves to the measuring location with G31 F50 speed and executes the measurement.
9. After the tool length measurement is completed, the result will be filled in to the assigned tool number on tool length table automatically.
10. Z axis being pulled back to the safety point or mechanical zero point in mode G53.



2.27.3 Note

1. Available version :
After version 10.116.16P, 10.116.24J (included)
2. Please install the tool presetting and measuring machine before executing the function. After the installation, it's recommended to set the XY machine coordinate to the machine coordinate of reference point P (Pr2801~2860). Thus, for future G37.2 Pn commands in the program, the tool will automatically move to the correct location before presetting and measuring.
3. Before applying G37.2 to single tool with multiple workpieces, please fill the Z axis drop difference between each workpieces into the workpiece coordinate system(G54~G54P100) manually before executing the tool presetting and measuring function.
4. For G31 measuring action in tool presetting and measuring, the override ratio is defined as 100%.
5. The MPG offset of Z axis will be reset 0 after the tool presetting and measuring is completed.
6. If the current tool number is 0, alarm [MAR-330_T code error, wrong tool number] will be issued.
7. Before executing the tool measurement, please specify the workpiece coordinate system with K. If not specified, alarm [MAR-333_tool measuring, workpiece coordinate K code range error] will be issued.
8. If the tool measuring signal is not triggered after Z axis reached Z point(measuring location) with G31 F_, alarm [MAR-336_measuring location setup error, measuring signal not triggered] will be issued.

2.27.4 Program Form

```

G30 P2 X75.0 Y25.0 // return the tool to reference point 2
M06 T1 // change to tool No.1
G37.2 Z-300. K2 Q0. R30. F100 // fill the Z axis offset value into second offset machine coordinate G54P2 (G55)
G30 P3 X150.0 Y25.0 // return the tool to reference point 3
G37.2 Z-250. K3 Q0. R30. F100 // fill the Z axis offset value into third offset machine coordinate G54P3 (G56)
M30

```

2.28 G37.3 : Automatic Tool Presetting and Measuring III

2.28.1 Command Form

G37.3 [Z_] [H_] [R_] [F_] [P_] [Q_] [T_]; (suitable for multiple tools with multiple workpieces, the tool length will be compensated to the assigned tool length table)

Z : Measuring position: Absolute command in machine coordinate. The system default value will be decided by Pr4059 if it's not specified.

H : Tool length compensation number.

R : Measuring distance, incremental value respective to Z position. The system default value will be decided by Pr4055, if not specified.

F : Measuring speed. The system default value will be decided by Pr4057, if not specified.

P : Reference point P. The system default value will be decided by Pr4058, if not specified. If Pr4058 is not 1~4, the returning action of G30 won't be executed.

Q : Safe point: If it's not specified, the system default safety point is the machine zero point.

T : Tool number.

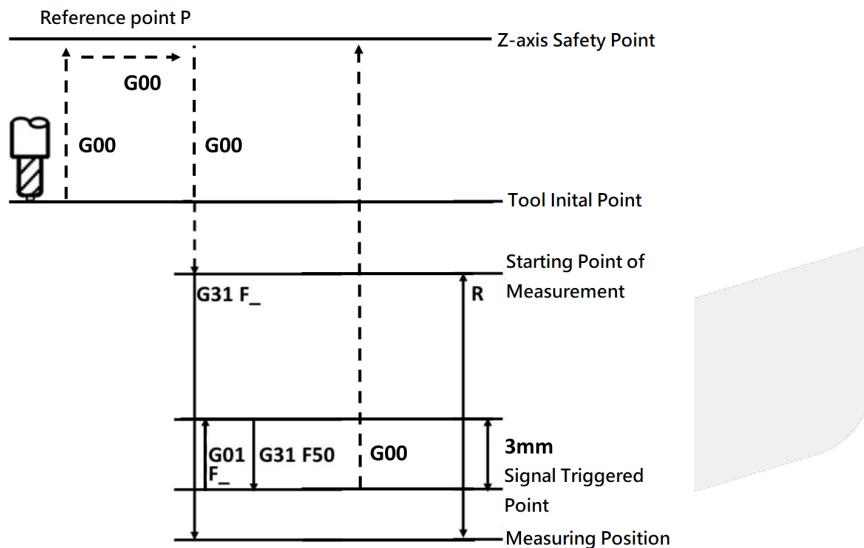
2.28.2 Description

Tool presetting and measuring process:

1. Z axis being pulled back to the safe point or mechanical zero point in mode G53.
2. Each axis returns to the XY axis position of reference point P in mode G30 (won't be executed if P argument is not specified).
3. M90 activate the tool air blast function.
4. Z axis moves to the starting point of measurement with G00 speed.
5. M91 deactivate the tool air blast function.
6. Z axis moves to the measuring position with G31 F_ speed and starts the measurement.
7. If the measuring signal is triggered in step 6, Z axis will retract 3mm with G01 speed.
8. Z axis moves to the measuring location with G31 F50 speed and executes the measurement.
9. After the tool length measurement is completed, the result will be filled in to the assigned tool number on tool length table automatically.



10. Z axis being pulled back to the safety point or mechanical zero point in mode G53



2.28.3 Note

- Available version :
After 10.116.16P、10.116.24J (included)
- Please install the tool presetting and measuring machine before executing the function. After the installation, it's recommended to set the XY machine coordinate to the machine coordinate of reference point P (Pr2801~2860). Thus, for future G37.3 Pn commands in the program, the tool will automatically move to the correct location before presetting and measuring.
- Before applying G37.3 to single tool with multiple workpieces, please fill the Z axis drop difference between each workpieces into the workpiece coordinate system(G54~G54P100) manually before executing the tool presetting and measuring function.
- For G31 measuring action in tool presetting and measuring, the override ratio is defined as 100%.
- The MPG offset of Z axis will be reset 0 after the tool presetting and measuring is completed.
- If the current tool number is 0, alarm [MAR-330_T code error, wrong tool number] will be sent.
- Before executing the tool measurement, please specify the workpiece coordinate system with K. If not specified, alarm [MAR-333_tool measuring, workpiece coordinate K code range error] will be issued.
- If the tool measuring signal is not triggered after Z axis reached Z point(measuring location) with G31 F_, alarm [MAR-336_measuring location setup error, measuring signal not triggered] will be issued.

Program Form

```

G30 P2 X75.0 Y25.0 // return the tool to reference point 2
G37.3 T1 H1 P2 Z-150.
// change to tool No.1
// return the tool to reference point 2
// move Z axis to -150. and start the tool presetting and measuring
// define tool length compensation H1 after measurement
G37.3 T2 H2 P2 Z-150.
// change to tool No.2
// return the tool to reference point 2
// move Z axis to -150. and start the tool presetting and measuring
// define tool length compensation H2 after measurement
// move Z axis to -150. and start the tool presetting and measuring

```

```

G30 P3 X150.0 Y25.0 // return the tool to reference point 3
G37.3 T3 H3 P3 Z-200.
// change to tool No.3
// return the tool to reference point 3
// move Z axis to -200. and start the tool presetting and measuring
// define tool length compensation H3 after measurement

```

2.29 G40/G41/G42 : Cutter Radius Compensation

2.29.1 Command Form

$\left\{ \begin{array}{l} G41 \\ G42 \end{array} \right\} X_ Y_ D_;$

G40;

G41: Left cutter compensation

G42: Right cutter compensation

G40: Cancel cutter compensation

X, Y: End coordinate of each axis

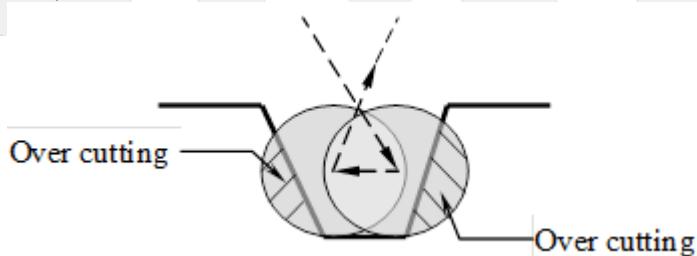
D: ID for specifying the cutter compensation number.

2.29.2 Description

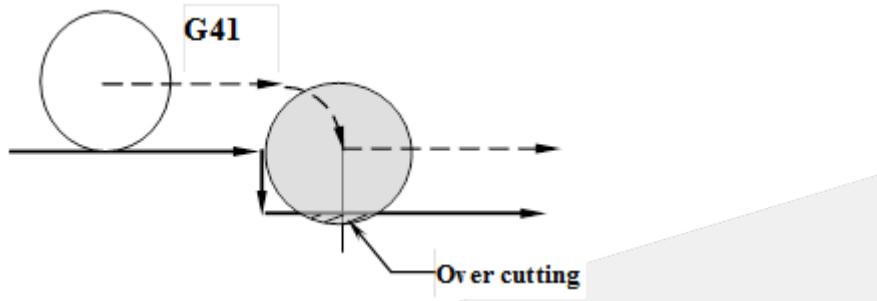
In general, if a machining program is executed with the tool center moving along the designed workpiece contour, there will be a cutter radius size overcut on every machining contour. The function of cutter radius compensation command is to give an offset with the length of cutter radius between the programmed contour and the actual moving contour, so the workpiece can match the original design after machining. Therefore, users don't need to be bothered by the calculations considering cutter radius anymore, it's able to get the correct end product by editing the machining program according to the designed size with this compensation function applied.

2.29.3 Precautions

1. The effective block of setup (G41/G42) and cancel (G40) of the cutter radius compensation function is defined by Pr3815 Tool radius compensation mode.
2. The effective block should have displacement on the working plane(G17, G18, G19), and could not be G02, G03 commands.
3. During the grooving process, if the groove width is smaller than twice the cutter radius, alarm [COR-074_excessive cutter radius, path overcut] will be issued due to the overcut. (Only check when Pr3819 is set 1)



4. When machining a workpiece in ladder shape, if the height of ladder is smaller than the cutter radius, alarm [COR-074_excessive cutter radius, path overcut will be issued due to the overcut. (Only check when Pr3819 is set 1)]

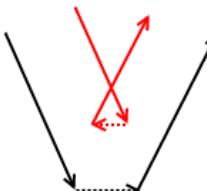
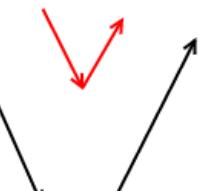


5. The G10 L12 and G10 L13 commands cannot be used in the same block as the D code commands.
 6. Tool radius compensation relies on the motion block after the G40 block to end the compensation path. If the next line of G40 is not a motion block(i.e. other non-motion blocks in front of the motion block, such as tool change T code, plane switching G17/G18/G19...etc.), the end of compensation path will not be as expected.
 7. This feature doesn't support Skip Function(G31) and Multi-Axis Multi-Signal Skip Function(G31.10/G31.11).

2.29.4 Related Setting

Parameters:

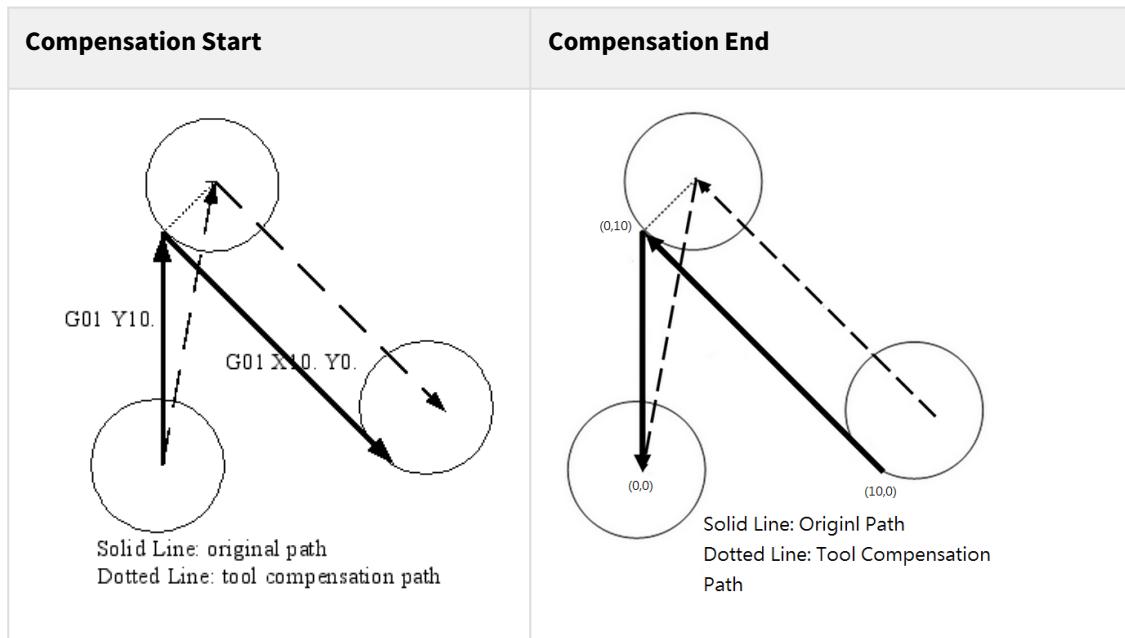
- Pr3814 Rapid traverse compensation mode (Setting tool compensation under rapid traverse command)
 - Specification Diagram:

Black: Original path	Red: Compensation Path		
Solid Line: G01	Dotted Line: G00		
Movement Spe cification	Pr3814 = 0	Pr3814 = 1	Pr3814 = 2
Undercutting Check	 <p>G00 does no overcutting check.</p>	 <p>Pr3819 = 1: Trigger alarm</p>	<p>The same overcut check specification as Pr3814 = 1.</p>

Movement Specification	Pr3814 = 0	Pr3814 = 1	Pr3814 = 2
Compensation Path	 <p>G00's tool compensation path is the same as G01.</p>	The same tool compensation path as as Pr3814 = 0.	 <ul style="list-style-type: none"> i. G01(G02/G03) followed by G00, will move to the end point of tool compensation of previous block. (Not reference G00) ii. G00 followed by G00, will cancel tool compensation iii. G00 followed by G01(G02/G03), will move to start point of tool compensation of the next block. (Not reference G00)

- Pr3815 Tool radius compensation mode (Setting the tool radius compensation initial and end block)
 - 2: Type A (normal mode)

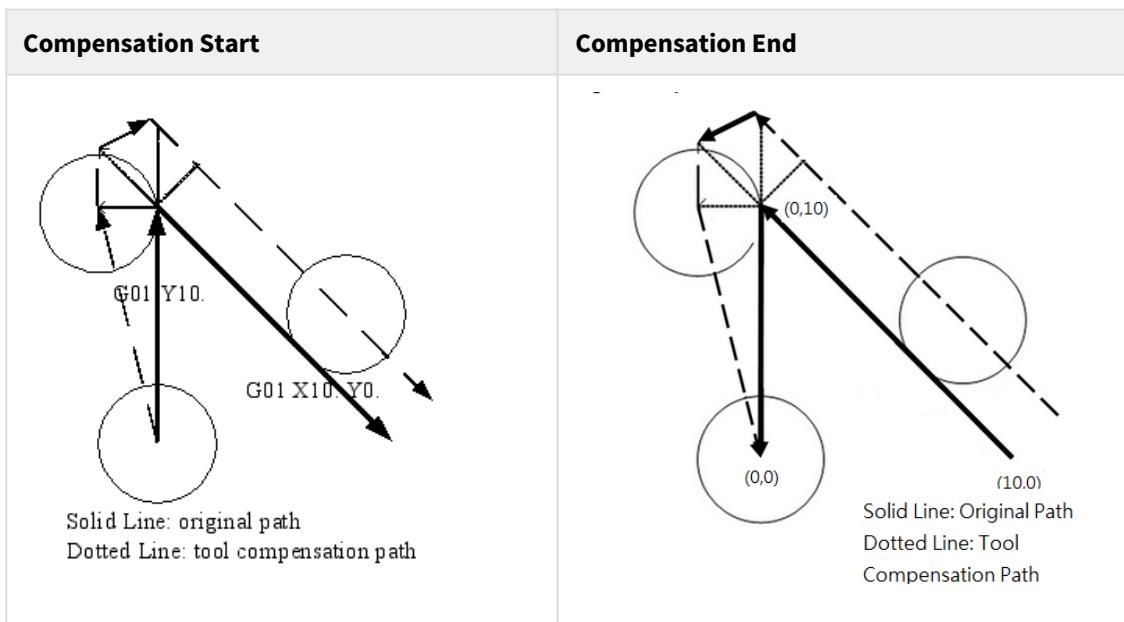
Compensation Start	Compensation End
G00 X0. Y0. Z0. G41 D1 // D = 2.5mm G01 Y10. // Startup block. G01 X10. Y0. // Compensation start at the beginning of this block.	G00 X10. Y0. Z0. G40 G01 X0 Y10. // Compensation end at the ending of this block. G01 X0. Y0. // Cancellation block.
Compensation start: The beginning of the second block The first block moves to the compensation start point of the second block.	Compensation stops: The end of the second to last block Tool compensation enabled until the end of the second to last block, then cancel compensation and move to the end of last block.



- 0: Type B (normal mode)

Compensation Start	Compensation End
<p>G00 X0. Y0. Z0. G41 D1 // D = 2.5mm G01 Y10. // Startup block. Compensation start at the end of this block. G01 X10. Y0.</p> <p>Compensation start: The end of the first block Directly move to the end point of the first block, then tool compensation is enabled afterward.</p>	<p>G00 X10. Y0. Z0. G40 G01 X0 Y10. G01 X0. Y0. // Cancellation block. Compensation end at the beginning of this block.</p> <p>Compensation stop: The beginning of the last block Tool compensation enabled until the beginning of the last block, then cancel compensation and move to the end of last block.</p>

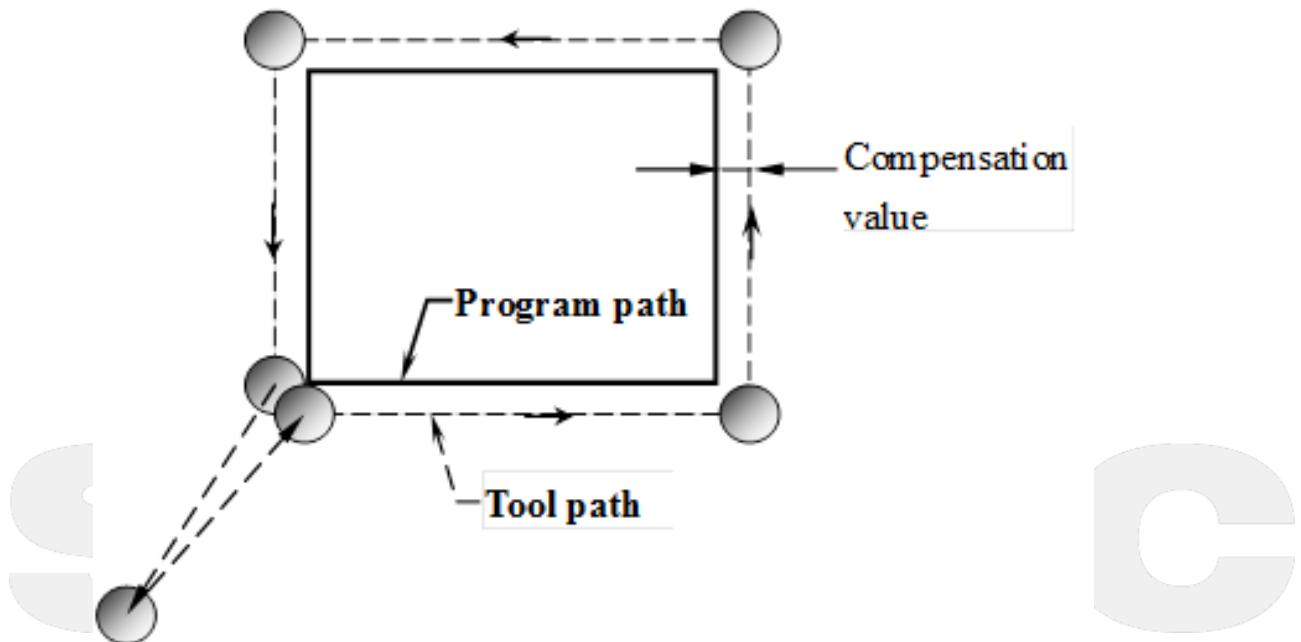
SYNTEC



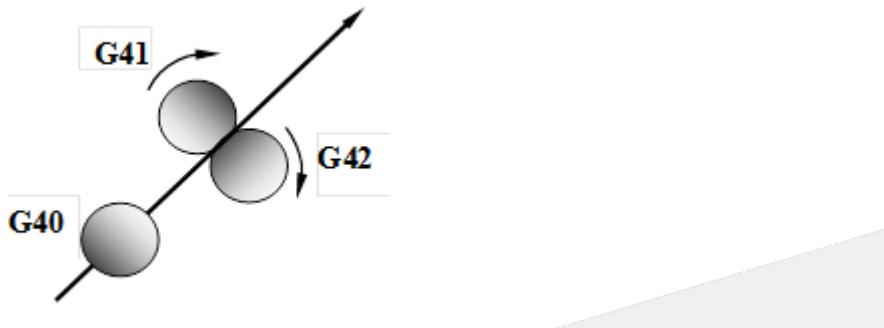
- Pr3819 Overcut check type (Set the overcut check logic after tool radius compensation)

2.29.5 Example

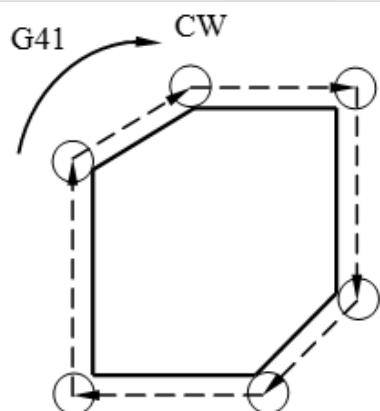
1. Cutter radius compensation:



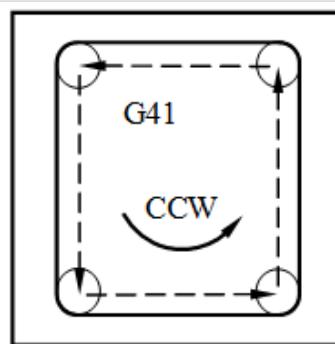
2. Direction of cutter radius compensation:



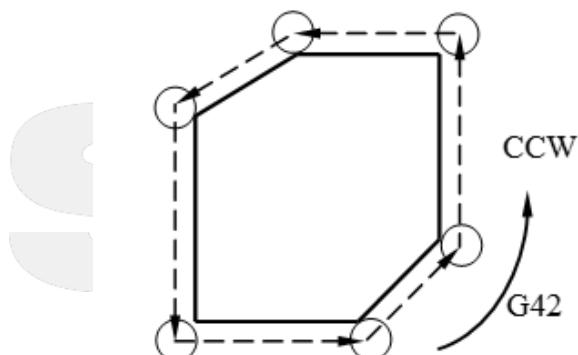
Compensation value	Positive	Negative
G41	Left Compensation	Right Compensation
G42	Right Compensation	Left Compensation



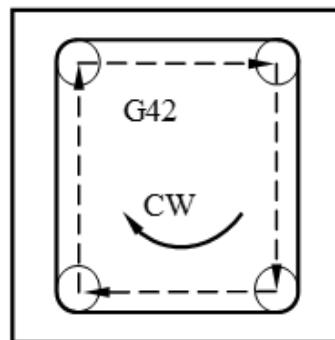
a. G41-outline cut (CW)



b. G41-inline cut (CCW)



c. G42-outline cut (CCW)

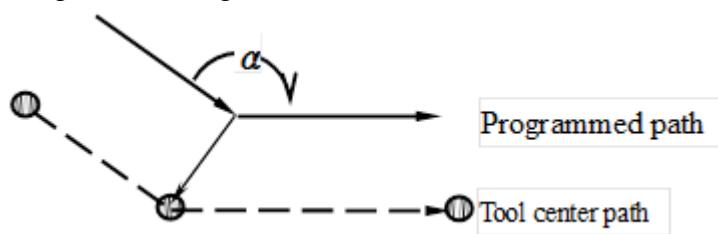


d. G42-inline cut (CW)

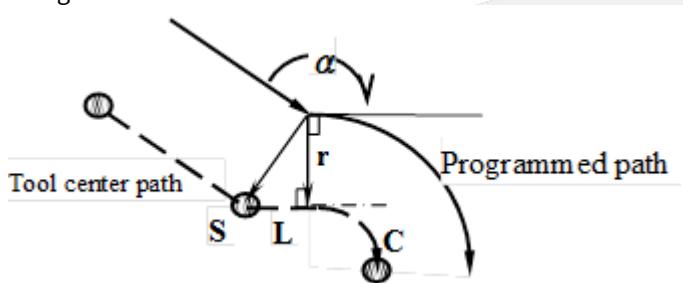
3. Cutter radius compensation on corners:

a. Corner: $90^\circ \leq \alpha < 180^\circ$

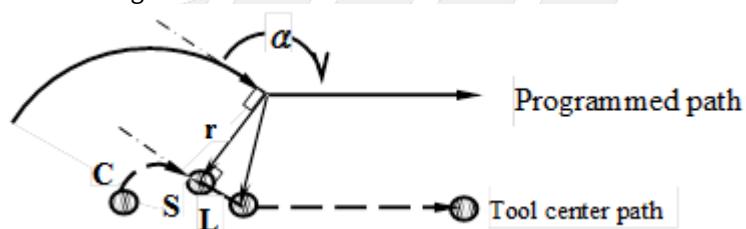
- straight line -> straight line



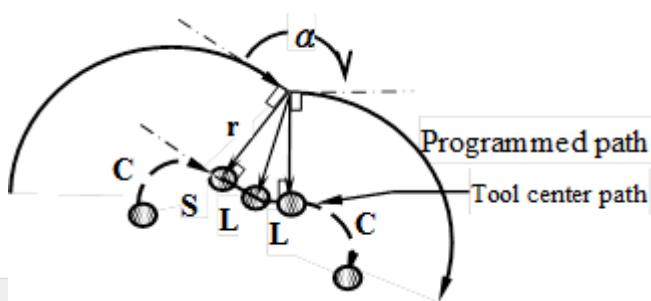
- Straight line -> Arc



- Arc -> Straight line

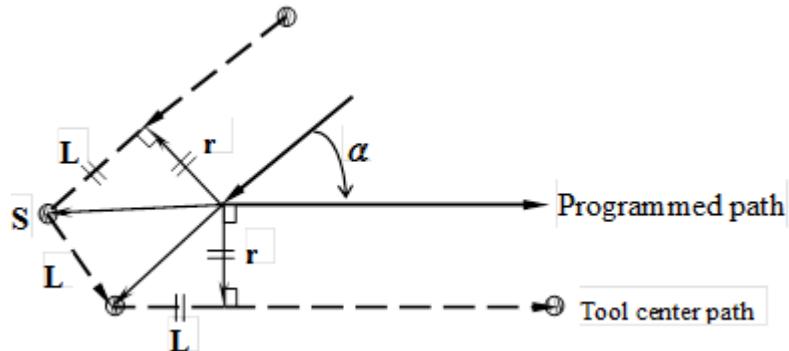


- Arc -> Arc

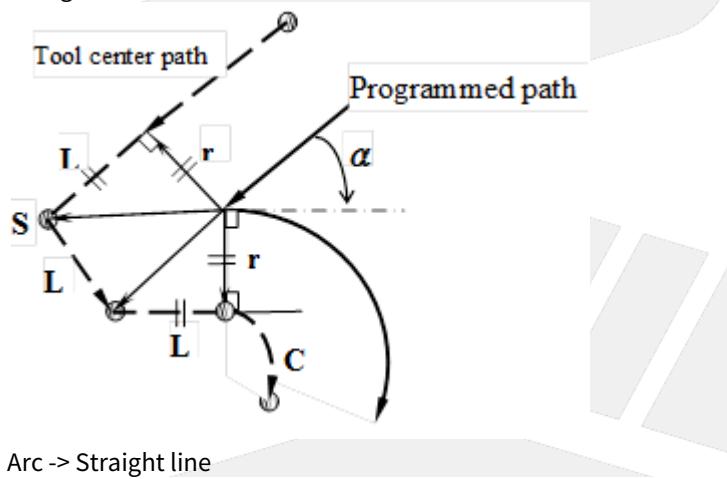


b. Corner: $90^\circ \leq \alpha < 180^\circ$

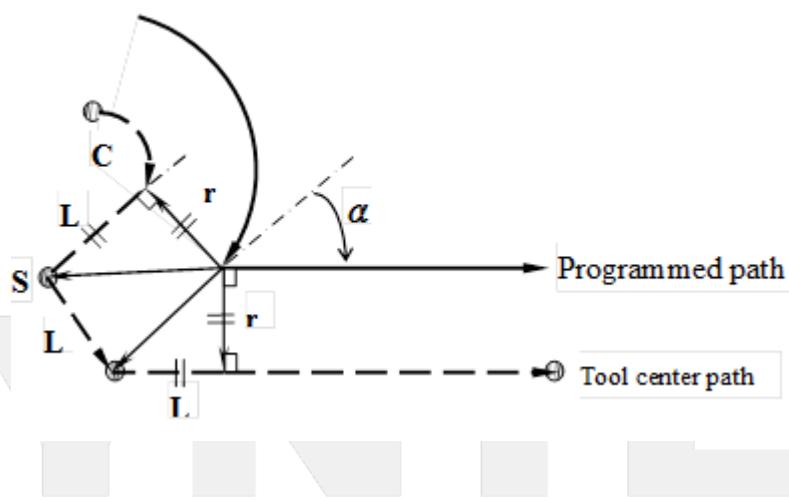
- Straight line -> Straight line



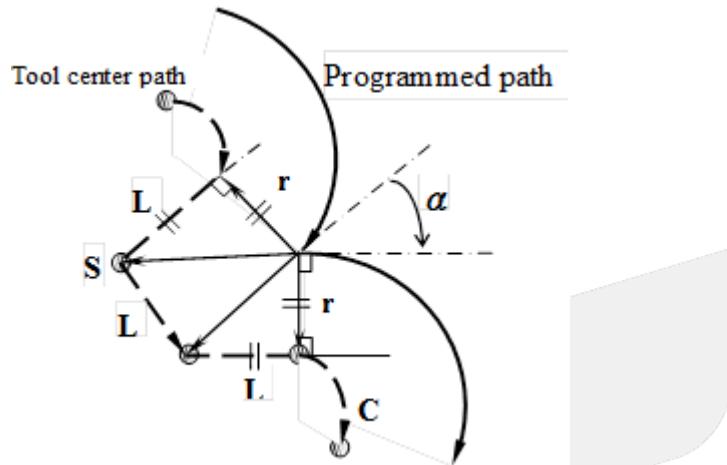
- Straight line -> Arc



- Arc -> Straight line



- Arc -> Arc



4. Cutter radius compensation composition

If cutter radius changes during compensation, the new cutter radius valid in next block. The contour will be compensated gradually during current block. Arc block would change to spiral interpolation.

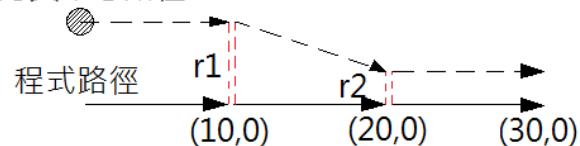
- Straight line -> Straight line

.....

```
G01 X10.0  
G10 L12 P1 R(r2)  
G41 D1 X20.0  
X30.0
```

.....

刀具中心路徑

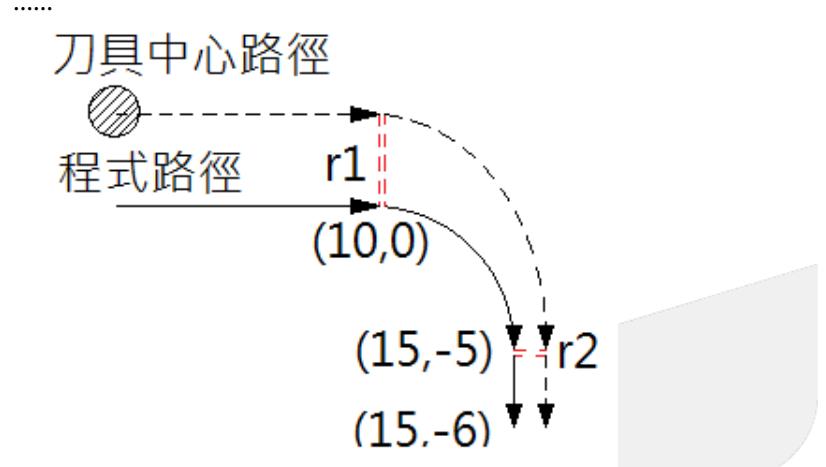


- Straight line -> Arc (spiral)

.....

```
G01 X10.0  
G10 L12 P1 R(r2)  
G41 D1 G02 X15.0 Y-5.0 R5.0  
G01 Y-6.0
```

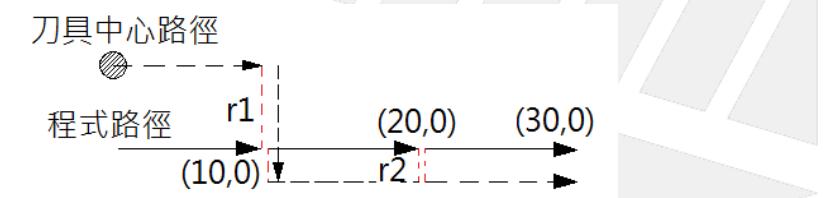
SYNTEC



Note, if G41/G42 changes meanwhile, the new cutter radius valid at initial point of block.

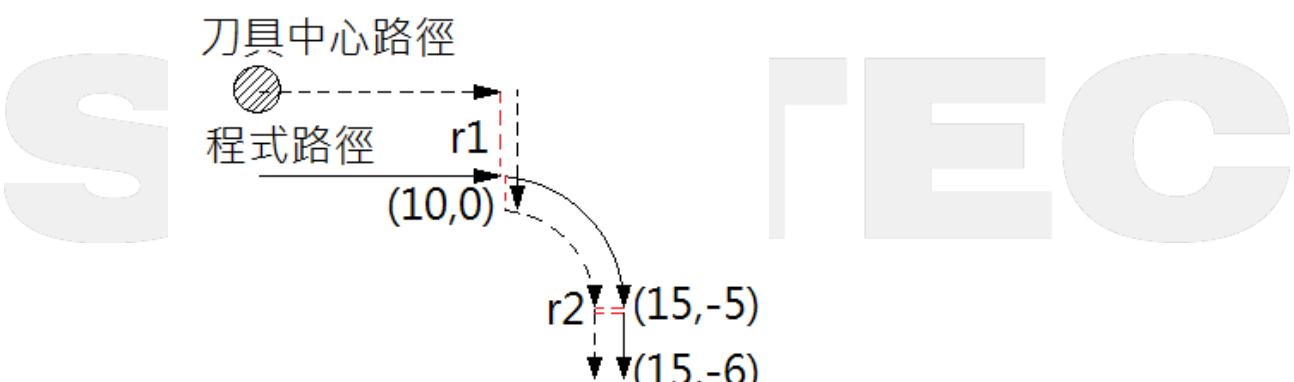
- Straight line -> Straight line

```
..... // G41
G01 X10.0
G10 L12 P1 R(r2)
G42 D1 X20.0
X30.0
```

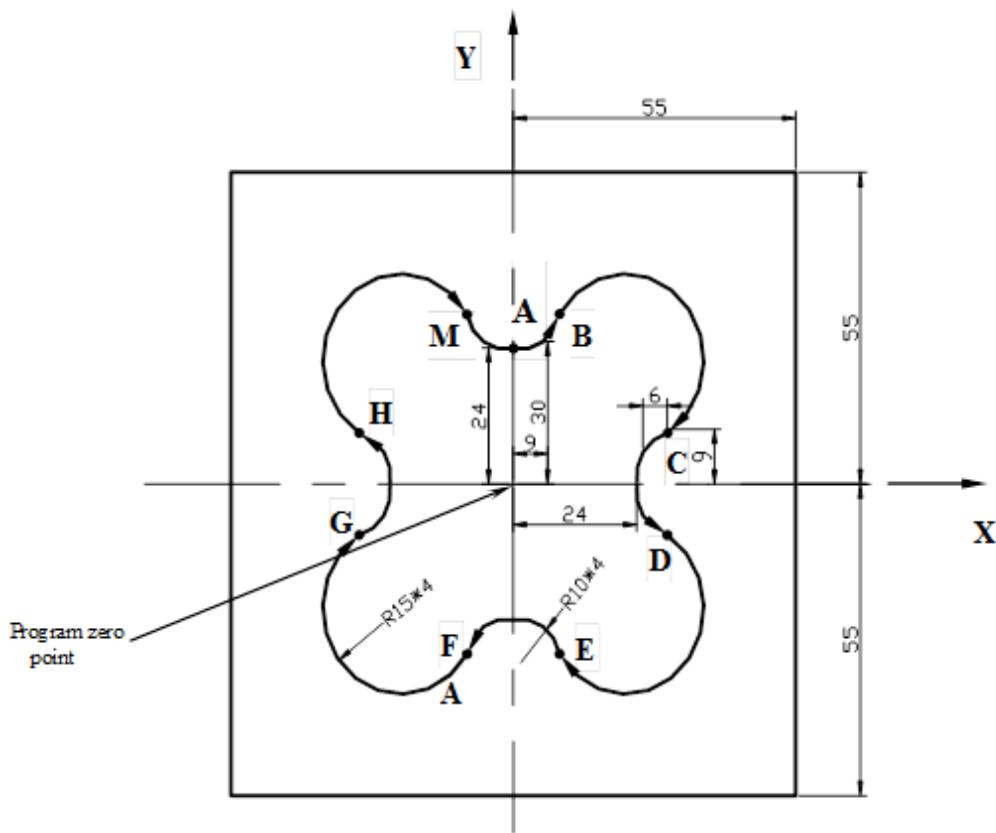


- Straight line -> Arc

```
..... // G41
G01 X10.0
G10 L12 P1 R(r2)
G42 D1 G02 X15.0 Y-5.0 R5.0
G01 Y-6.0
```



Program example



Program description:

N001 T1 S1000 M03; //tool No.1 (diameter 10mm end mill), spindle 1000rpm (CW)
 N002 G00 X0.0 Y0.0 Z10.0; //rapid orientation to above program zero point
 N003 M08; //open cutting fluid
 N004 G90 G01 Z-10.0 F600; //linear cutting to the bottom of notch, feedrate 600mm/min
 N005 G42 Y24.0 D01; //cutter left compensation, program zero point -> **A**
 N006 G03 X9.0 Y30.0 R10.0; //**A->B** CCW arc cutting
 N007 G02 X30.0 Y9.0 R15.0; //**B->C** CW arc cutting
 N008 G03 X30.0 Y-9.0 R10.0; //**C->D** CCW arc cutting
 N009 G02 X9.0 Y-30.0 R15.0; //**D->E** CW arc cutting
 N010 G03 X-9.0 Y-30.0 R10.0; //**E->F** CCW arc cutting
 N011 G02 X-30.0 Y-9.0 R15.0; //**F->G** CW arc cutting
 N012 G03 X-30.0 Y9.0 R10.0; //**G->H** CCW arc cutting
 N013 G02 X-9.0 Y30.0 R15.0; //**H->M** CW arc cutting
 N014 G03 X0.0 Y24.0 R10.0; //**M->A** CCW arc cutting
 N015 G00 Z10.0; //pull up Z axis and return to the start point
 N016 G40 X0.0 Y0.0 ; //deactivate cutter compensation and return to start point
 N017 M09; //cutting fluid OFF

N018 M05; //spindle stop

N019 M30; //program end

2.30 G43.4 : Rotate Tool Center Point (RTCP) Type1

2.30.1 Command Form

G43.4 H_;
G49;

G43.4 : Enable Rotate Tool Center Point Type1 (RTCP Type1)

G49 : Disable Rotate Tool Center Point Type1

H : Tool number

The applying method of RTCP is the same as tool length compensation (G43), only needs to give the G43.4 command before the machining starts. After giving the command to assign the tool number, it'll be able to apply RTCP with the specified tool length.

2.30.2 Description

RTCP is the Rotate Tool Center Point function. For general machines, moving commands from the controller are given to the tool holder or the spindle nose; after activating the RTCP function, the moving commands will be controlling the tool center point. The RTCP function is the specified function of five-axis machining centers.

There are 2 machining contour in the picture below, the orange one is the contour of general machining state, the controller controls the contour of spindle nose, so there will be a one tool length gap between the machining contour and workpiece surface; the blue one is the contour of tool center point after activated RTCP, which is able to directly edit the machining program with workpiece surface coordinates.

The SYNTEC logo is displayed prominently at the bottom of the page. It consists of the word "SYNTEC" in a bold, sans-serif font. The letters are light gray and appear to be embossed or slightly recessed into a white background. The letter "T" has a small vertical bar extending downwards from its middle, and the letter "E" has a small horizontal bar extending to the right from its middle.

The change of tool length and the mechanism differences between machines can all be ignored by using this kind of machining programs and the programs can also be applied more efficiently.



2.30.3 Note

1. G41, G42 cutter radius compensation functions can't be applied together.
2. G43, G44, G43.4 tool length compensation functions can't be applied together.
3. The tool length should be set positive.
4. Before applying G53, G28, G29, G30, please remember to disable RTCP Type1 with G49 to avoid abnormal machine actions.
5. In RTCP Type1 mode, if activates the HPCC function with G05 P10000, alarm [COR-140 G05 is banned in RTCP mode] will be issued.
6. Do not apply with the polar axis interpolation function (G12.1).
7. This feature doesn't support Multi-Axis Multi-Signal Skip Function(G31.10/G31.11).

2.30.4 Program Example

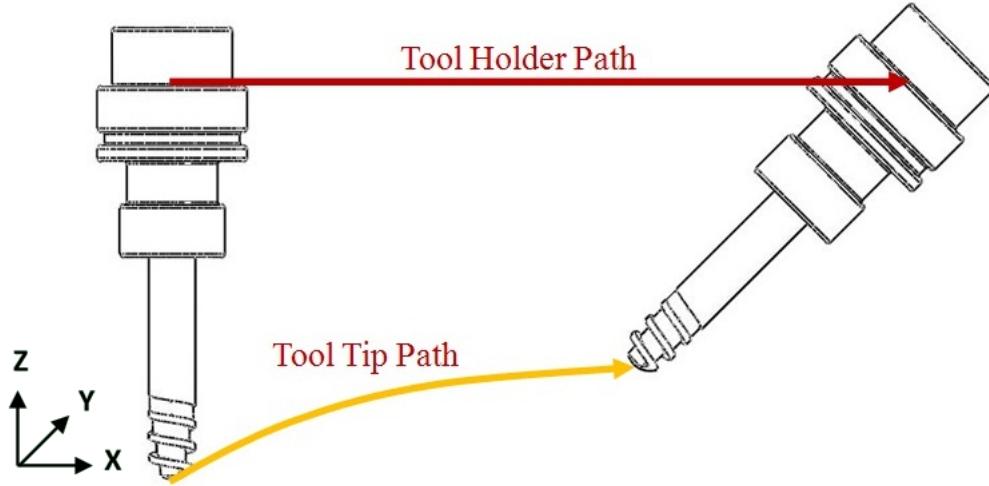
RTCP Type1 disable :

G00 X0. Y0. Z0. B0. C0.

G01 X50. Y0. Z0. B45. C0.

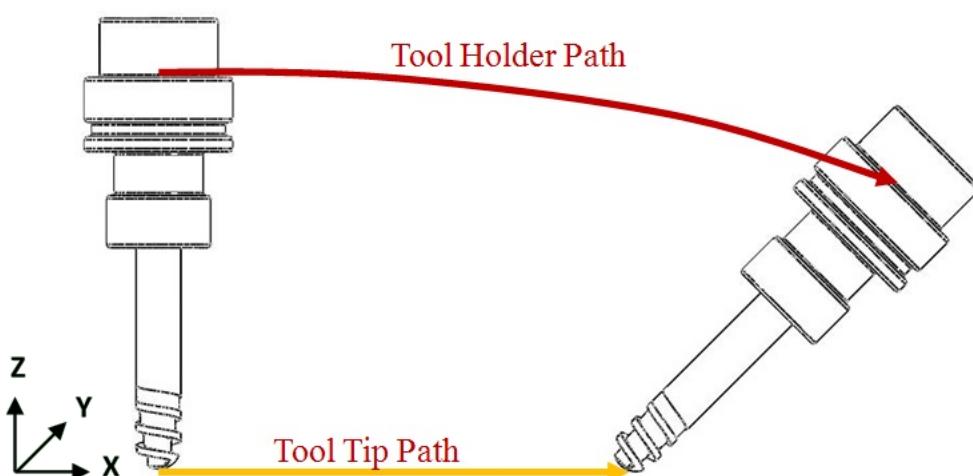
SYNTEC

Machine motion :



RTCP Type1 enable :

G43.4 H1
G00 X0. Y0. Z0. B0. C0.
G01 X50. Y0. Z0. B-45. C0.
Machine motion :



Before enable RTCP Type1, the motion of linear and rotary axes are independent; after activating RTCP Type1, the linear motion commands of tool center point will take the priority and the rotary axis will be rotating with the tool center point.

2.31 G43.5 : Rotate Tool Center Point (RTCP) Type2

2.31.1 Command Form

G43.5 H_;
X_Y_Z_I_J_K_;
G49;

G43.5 : Enable Rotate Tool Center Point Type2 (RTCP Type2) ;

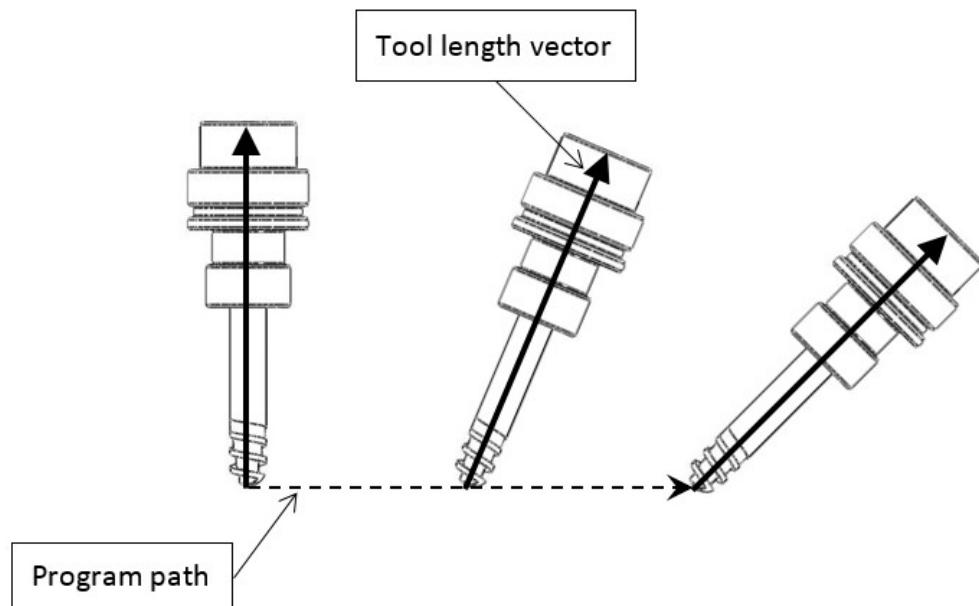
G49 : Disable Rotate Tool Center Point Type2 ;

H : Tool number

X_Y_Z_ : The coordinate of tool center point moving block on the program coordinate system.

I_J_K_ : The tool axis direction of the moving block on the program coordinate system (please refer to the picture below for definitions of tool axis direction)

The applying method of RTCP is the same as tool length compensation (G43), only needs to give the G43.5 command before the machining. After giving the command to assign the tool number, it'll be able to apply RTCP with the specified tool length.



2.31.2 Description

RTCP is the Rotational Tool Center Point function. For general machines, moving commands from the controller are given to the tool holder or the spindle nose; after activating the RTCP function, the moving commands will be controlling the tool center point. The RTCP function is the specified function of five-axis machining centers. There are 2 machining paths in the picture below, the orange one is the path of general machining state, the controller controls the path of spindle nose, so there will be a one tool length gap between the machining path and workpiece surface; the blue one is the path of tool center point after activated RTCP, which is able to directly edit the machining program with workpiece surface coordinates.

The change of tool length and the mechanism differences between machines can all be ignored by using this kind of machining programs and the programs can also be applied more efficiently.



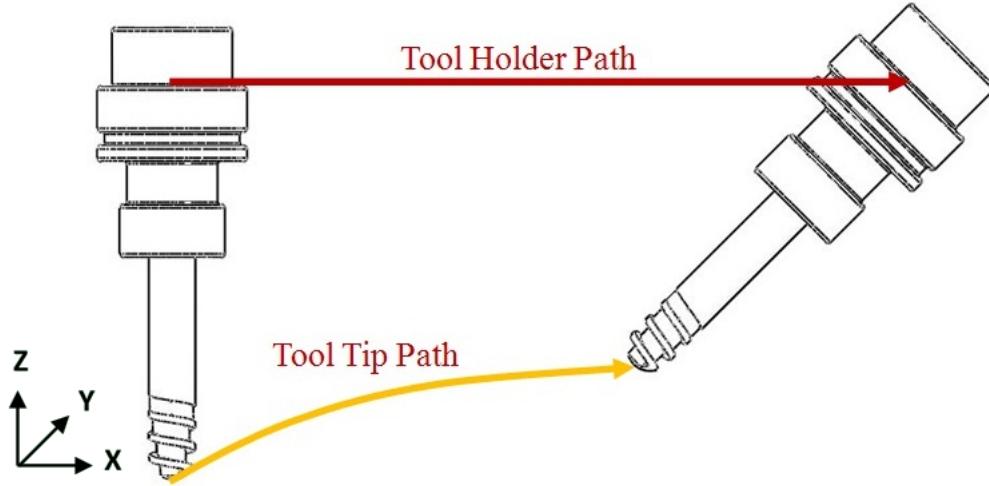
2.31.3 **Note**

1. G41, G42 cutter radius compensation functions can't be applied together.
2. G43, G44, G43.4 tool length compensation functions can't be applied together.
3. G91 increment command can't be applied together.
4. The tool length should be set positive.
5. Before applying G53, G28, G29, G30, please remember to disable RTCP Type2 with G49 to avoid abnormal machine actions.
6. In RTCP Type2 mode, if Enable the HPCC function with G05 P10000, alarm [COR-140 G05 is banned in RTCP mode] will be issued.
7. In RTCP Type2 mode, if executes the moving command of 1st/2nd rotary axis, alarm [COR-158 1st/2nd rotary axis command can't be executed in G43.5 mode] will be issued.
8. If omitted some of arguments I, J, K, the argument will be taken as 0; if omitted all the arguments, it'll be taken as the same tool direction as the previous block.
9. Does not support STCP function (Smooth Tool Center Point).
10. Do not apply with the polar axis interpolation function (G12.1).
11. This feature doesn't support Multi-Axis Multi-Signal Skip Function(G31.10/G31.11).

2.31.4 **Program Example**

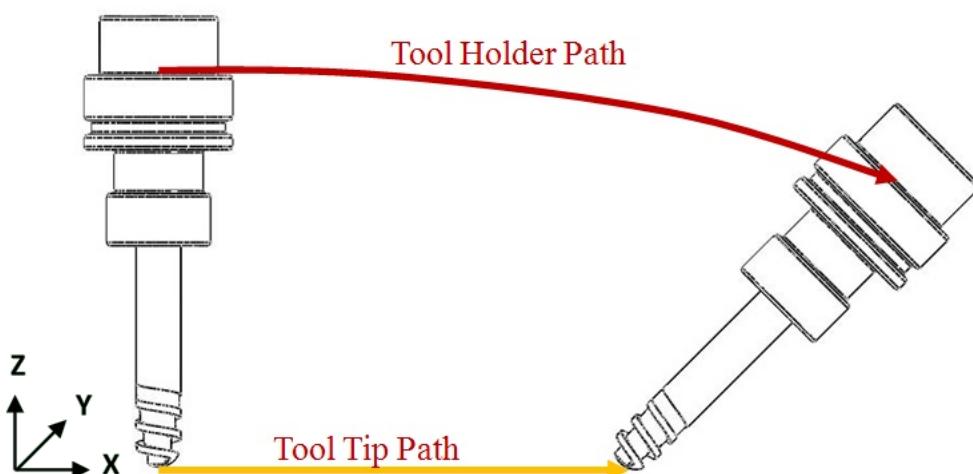
RTCP Type2 disable :
 G00 X0. Y0. Z0. B0. C0.
 G01 X50. Y0. Z0. B45. C0.

Machine motion



RTCP Type2 enable :

G43.5 H1
 G00 X0. Y0. Z0. I0. J0. K1.
 G01 X50. Y0. Z0. I1. J0. K1.
 Machine motion :



Before activating RTCP Type2, the motion of linear and rotary axes are independent; after activating RTCP Type2, the linear motion commands of tool center point will take the priority and the rotary axis will be rotating with the tool center point.

2.32 G43/G44/G49 : Tool Length Compensation

2.32.1 Command Form

$$\left\{ \begin{array}{l} G43 \\ G44 \end{array} \right\} Z_- H_-;$$

G49;

G43: Compensation along positive direction;

G44: Compensation along negative direction;

G49: Compensation cancel;

Z: Z axis end point coordinate;

H: Tool number;

2.32.2 **Description**

- When machining with milling machine or cutting center, many tools are used but each of them has different tool lengths, which makes the distance between tool point and workpiece is not always the same.
- The difference between the tool length before and after tool change leads to the Z axis error. The tool length compensation function is to compensate in Z axis direction and modify the tool length error.

- **First way:**

Move the tool down till the workpiece surface manually, take the distance as tool length compensation value and enter the value of each tool to the tool setup in operation interface. Set the tool number with H value of program command.

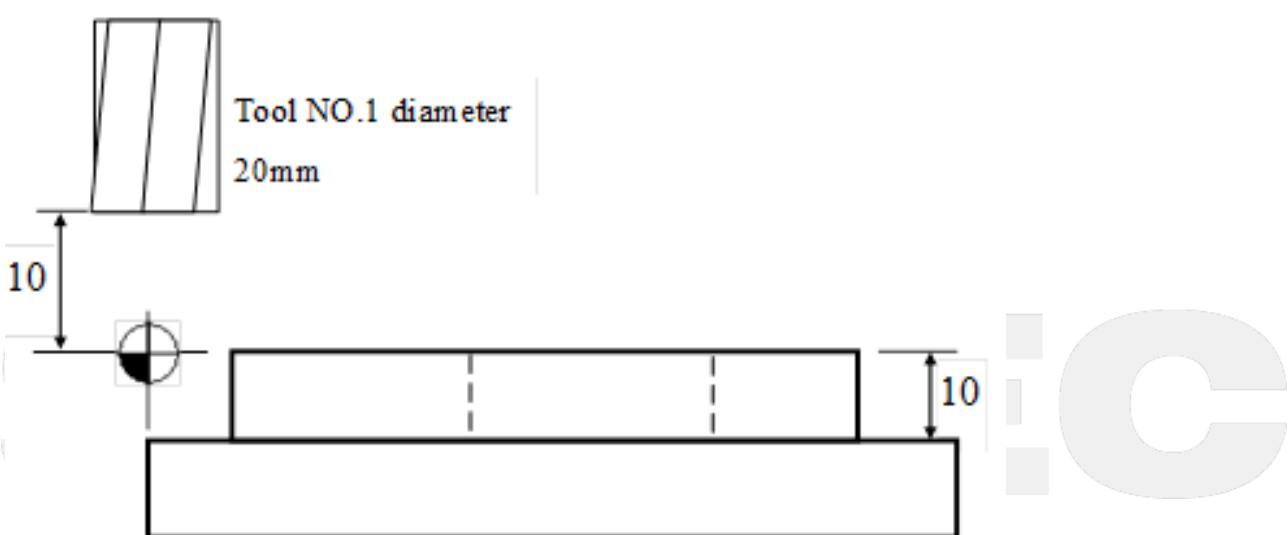
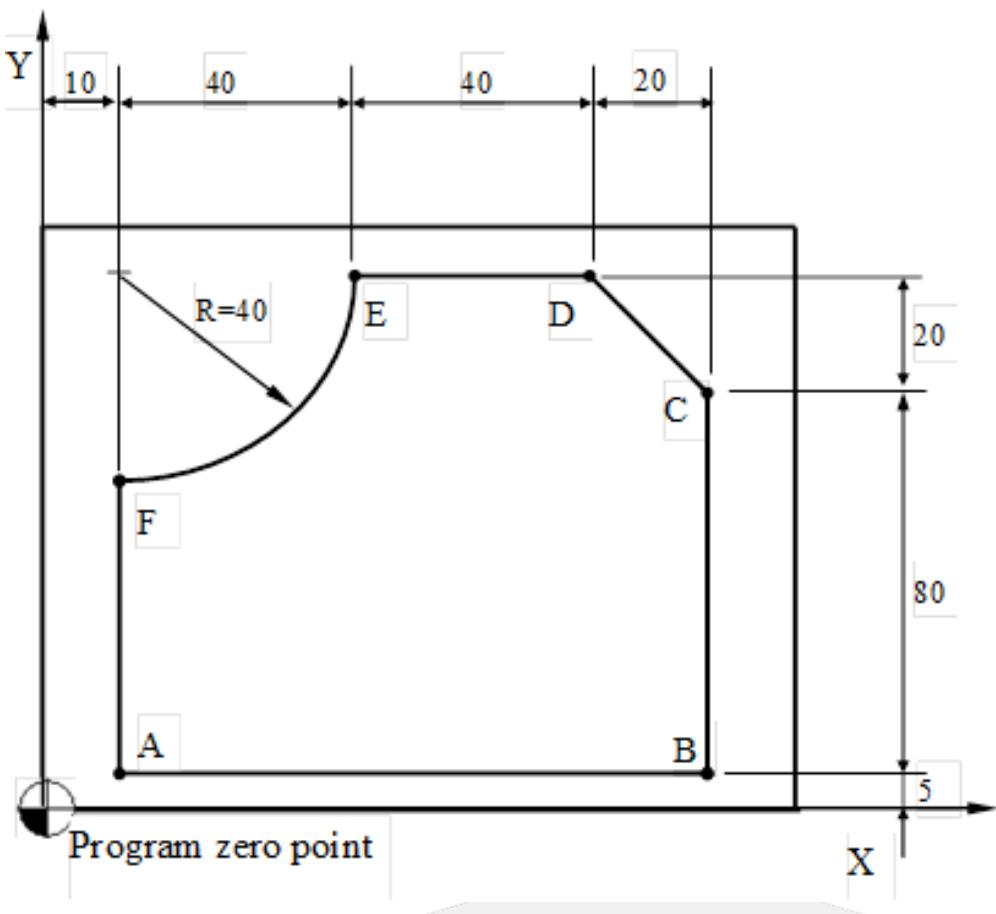
- **Second way:**

Choose a tool as the basis, enter the controller operation interface and run the tool length adjustments in the set G54 working coordinate. For all the tools used afterwards, the subtraction between the tool and basis tool will be turned into the tool length compensation value.

	Positive	Negative
G43	Positive Direction	Negative Direction
G44	Negative Direction	Positive Direction

- If axial addresses are specified together with G43, G44 or G49 command, the tool length compensation will be executed first, followed by moving of the tool.

Program Example



Program description:

T1 S1000 M03 ; //use tool No.1 (diameter 20mm end mill), spindle 1000rpm(CW)

```

G42 D01 ; //right cutter radius compensation (D01=10)
G00 X10.0 Y5.0 Z15.0 ; //rapid orientation to above point A
G43 H01 ; //positive tool length compensation (H01=-10)
G01 Z-10.0 ; //linear cutting to the bottom of point A
X110.0 ; //A->B
Y85.0 ; //B->C
X90.0 Y105.0 ; //C->D
X50.0 ; //D->E
G02 X10.0 Y65.0 R40.0 ; //E->F
G01 Y5.0 ; //F->A
G00 Z15.0 ; //rapid tool retract to above point A
G40 G49 ; //cancel compensation
M05 ; //spindle stop
M30 ; //program end

```

2.33 G45/G46/G47/G48 : Tool Offset

2.33.1 Command Form

$$\left\{ \begin{array}{l} G45 \\ G46 \\ G47 \\ G48 \end{array} \right\} X_ Y_ Z_ D_;$$

G45 : 1x cutter radius compensation in positive contour direction
 G46 : 1x cutter radius compensation in negative contour direction
 G47 : 2x cutter radius compensation in positive contour direction
 G48 : 2x cutter radius compensation in negative contour direction
 X, Y, Z : End point coordinate of each axis
 D : The set compensation ID of cutter radius compensation

2.33.2 Description

For some application situations, users need to inner/outer offset the cutting path to adjust the size of workpiece without changing the program content. Therefore, G45/G46 provides 1x cutter radius compensation in positive/negative direction; G47/G48 provides 2x cutter radius compensation in positive/negative direction. The offset will be applied to each axis when the moving commands are given to multiple axes at the same time.

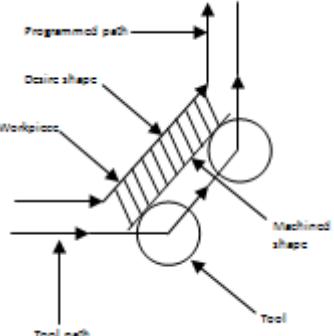
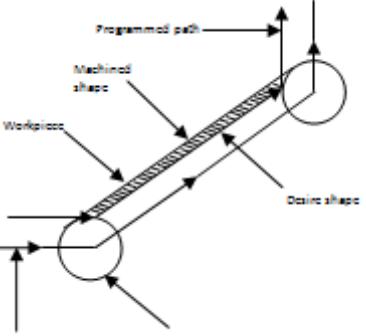
2.33.3 Note

1. This G code is only valid in single block, won't affect the following moving blocks if it's not specified.

2. The judgment of G45 positive direction/G46 negative direction is according to the start and end point of the moving path. If the path starts at point A and ends at point B ($A \neq B$), the $A \rightarrow B$ direction will be the positive direction and the $B \rightarrow A$ direction will be the negative direction.
3. When the axial argument of moving command is 0 and the coordinate is in G91 mode, the compensation will be executed according to the set mode (G45/G46/G47/G48). Take G91G45Y0 command as example, the compensation will be executed along Y+ (G45 = 1x cutter radius compensation in positive direction). If the coordinate is in G90 mode then there will be no movements.

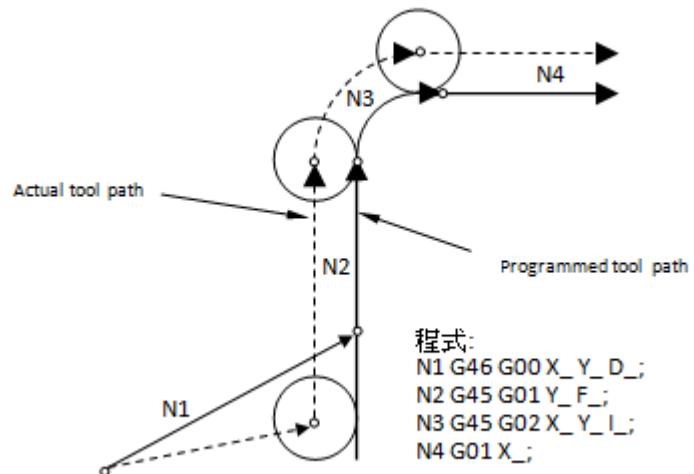
Mode	G45Y0D1	G45Y-0D1	G46Y0D1	G46Y-0D1
G91	+10.	-10.	-10.	10.
G90	No Compensation	No Compensation	No Compensation	No Compensation

4. Do not apply with G41/G42 at the same time, alarm [COR-082_G41/G42 not allowed with G45~G48 at the same time] will be issued.
5. When applying G45~G48 to more than 2 axes for synchronous interpolation, it might lead to overcut or undercut. Please apply G41/G42 cutter radius compensation to solve the problem.

Undercut	Overtcut
 <p>G01 G45 Xx₁ Dd₁; Xx₂ Yy₂; G45 Yy₃;</p>	 <p>G01 Xx₁; G45 Xx₂ Yy₂ Dd₂; Yy₃;</p>

6. When interpolation with G02/G03 circular interpolation, the application of G45~G48 must comply with the 2 rules below:
 - a. Can only apply to arc of 90/270 degrees, or alarm [COR-081_G02/G03 can only be 90/270 degrees for G45~G48] will be issued.

b. The circular interpolation command can only specified the center of circle with I, J, K.



2.33.4 Example

Figure	Explanation
↔	cutter radius compensation
→	program path
—→	actual moving path

Actual moving path

G45 1x cutter radius compensation in positive direction :

positive compensation value	negative compensation value
加工程序 → ↔	加工程序 →
實際路徑 —→	實際路徑 —→ ↔

G46 1x cutter radius compensation in negative direction :

positive compensation value	negative compensation value

加工程序 →

實際路徑 ↔

加工程序 →↔

實際路徑 →

G47 2x cutter radius compensation in positive direction :

positive compensation value

加工程序 →→↔×

實際路徑 →→

negative compensation value

加工程序 →

實際路徑 ↔↔×

G48 2x cutter radius compensation in negative direction :

positive compensation value

加工程序 →

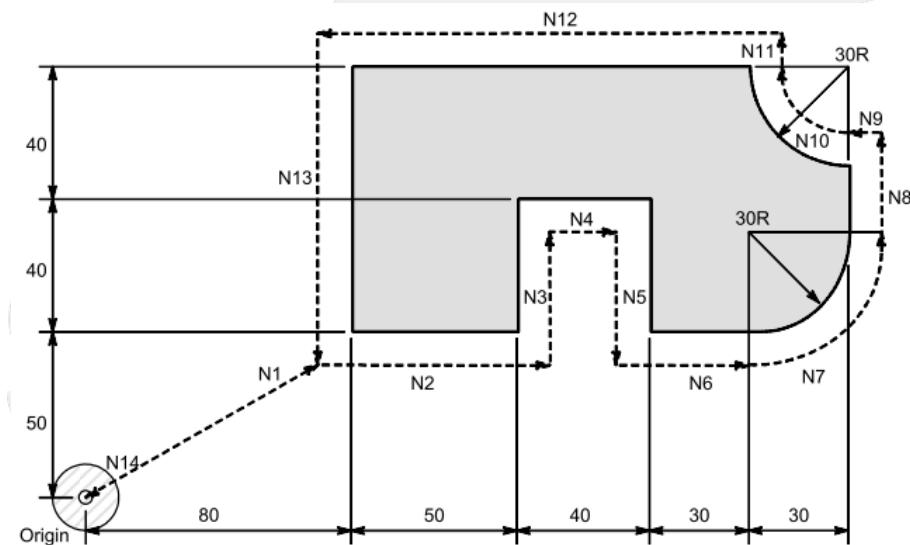
實際路徑 ↔↔×

negative compensation value

加工程序 →→↔×

實際路徑 →→

2.33.5 Program Example



Program description :

Line No.	Program Content	Note

N001	G91 G46 G00 X80.0 Y50.0 D01;	// X/Y 2 axes compensate in the negative direction relative to moving direction with D1 setup value. The N1 coordinate relative to zero point after moving is (80-D1, 50-D1)
N002	G47 G01 X50.0 F120.0;	// X axis compensates in the positive direction relative to moving direction with twice the D1 setup value. The distance X axis moved is (50 + D1 x 2);
N003	Y40.0;	// Y axis moves up to 40.0 in G01 speed with out any compensation since the G47 in previous line won't be inherited.
N004	G48 X40.0;	// X axis compensates in the positive direction relative to moving direction with twice the D1 setup value. The X axis movement in this line is (40 – D1 x 2).
N005	Y-40.0;	// Y axis moves down to 40.0 in the speed of G01 without any compensation since the G47 in previous line won;t be inherited.
N006	G45 X30.0;	// X axis compensates in the positive direction relative to moving direction with D1 setup value. The X axis movement in this line is (30 + D1).
N007	G45 G03 X30.0 Y30.0 J30.0;	// For the arc interpolation, X/Y axis execute the outer offset of the arc with D1 setup value by extending the radius of arc. Therefore, the arc radius of actual tool moving path is (30 + D1).
N008	G45 G01 Y20.0;	// Y axis compensates in the positive direction relative to moving direction with D1 setup value. The Y axis movement in this line is (20 + D1).
N009	G46 X0;	// X axis compensates in the negative position of coordinate with D1 setup value. Note that the X axis argument here is 0, thus the X axis movement in this line is (-D1).
N010	G46 G02 X-30.0 Y30.0 J30.0;	// For the arc interpolation, X/Y axis execute the inner offset of the arc with D1 setup value by reducing the radius of arc. Thus the arc radius of actual tool interpolation contour is (30 - D1).

N011	G45 G01 Y0;	// Y axis compensates in the positive direction of coordinate with D1 setup value. Note that the Y axis argument here is 0, thus the Y axis movement in this line is (D1).
N012	G47 X-120.0;	// X axis compensates along the positive direction relative to the X axis moving direction with twice the D1 setup value. The X axis movement in this line is (- (120 + 2 x D1)).
N013	G47 Y-80.0;	// Y axis compensates along the positive direction relative to the Y axis moving direction with twice the D1 setup value. The Y axis movement in this line is (- (80 + 2 x D1)).
N014	G46 G00 X80.0 Y-50.0;	// X/Y axes compensates in the negative direction relative to the moving direction with D1 setup value. Stops at the zero point after N14 is completed.

2.34 G51/G50 : Scaling

2.34.1 Command Form

$X_- Y_- Z_- \left\{ \begin{array}{l} I_- J_- K_- \\ P_- \end{array} \right.$

X, Y, Z: Coordinate of scaling center and the assigned scaling axis

I, J, K: Scaling factor(applies when the scaling factor of each axis are different)

P: Scaling factor(applies when the scaling factor of each axis are the same)

G50: Disable scaling function

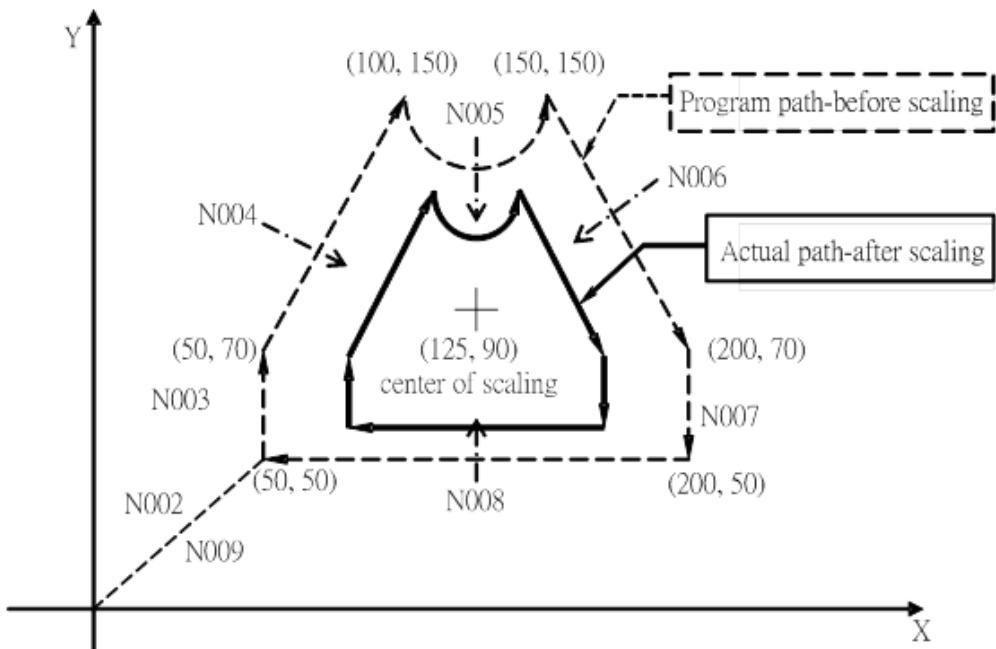
2.34.2 Description

G51 command can enlarge or reduce the cutting contour with setup value.

2.34.3 Note

1. Do not apply this G code with G51.1.
2. While applying this function to arc interpolation(G02,G03), if the setup scaling factor of each axis is different, the largest one will be set as the magnification value of arc radius.

2.34.4 Program Example



Program description:

```

N002 G51 X125.0 Y90.0 P0.5; //decide the scaling center X125,Y90, scaling factor 0.5 and apply to steps N003~N009
N003 G00 X50.0 Y50.0; // rapid orientation
N004 G01 Y70.0 F1000; // linear interpolation, feedrate 1000mm/min
N005 X100.0 Y150.0;
N006 G03 X150.0 I25.0; // arc interpolation, radius 25mm ;
N007 G01 X200.0 Y70.0; // linear interpolation
N008 Y50.0;
N009 X50.0;
N010 G00 X0.0 Y0.0; // fast return
N010 G50; // disable scaling function
N011 M30; // program ends

```

2.35 G51.1/G50.1 : Programmable Mirror Image

2.35.1 Command Form

G51.1 X__ Y__ Z__;
 G50.1; cancel programmable mirror image
 X, Y, Z: mirror point (axis) coordinate

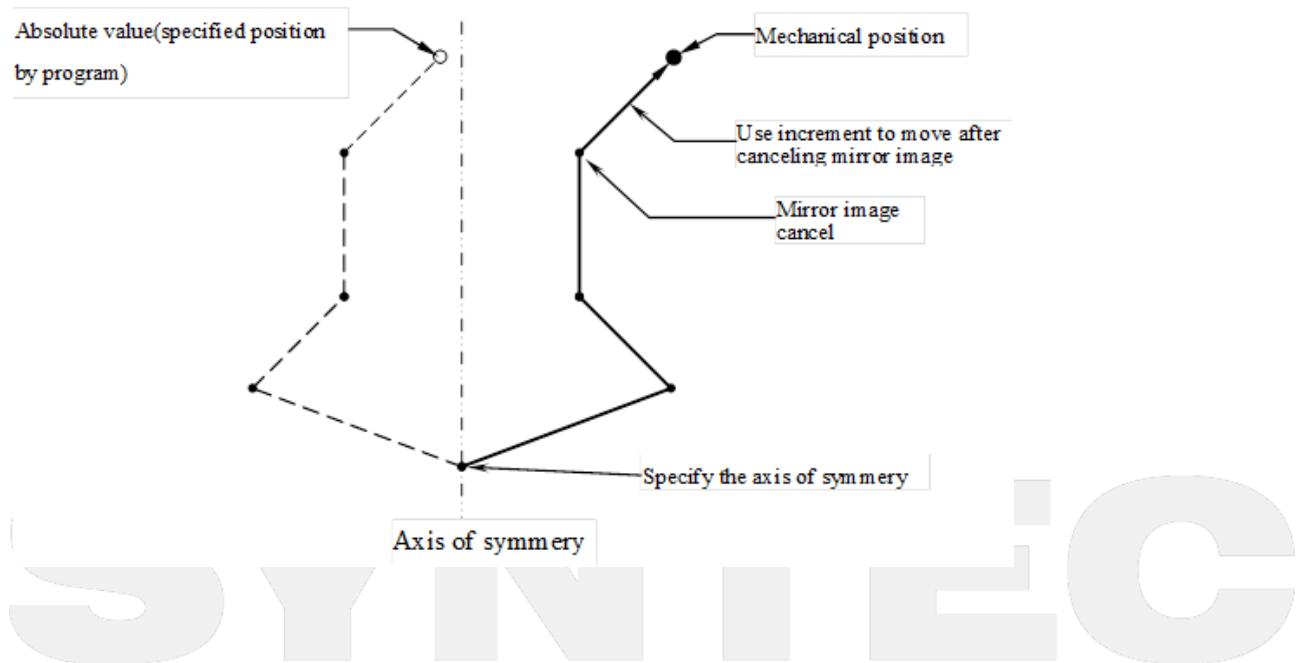
2.35.2 **Description**

When cutting symmetry shapes, only one of the left side or the right side program is needed for this function to process the cutting of the other side. G51.1 command specifies the effective mirror axis and center coordinate (absolute/increment value).

1. If there is only one axis specified for the mirror image on the assigned plane, the rotation direction and compensation direction for functions such as circular/tool length compensation and coordinate rotation will reverse.
2. Since the function is applied in local coordinates, the mirror center changes when the counter is reset or the working coordinate is changed.
3. In mirror image function, the G28, G30 command actions before mid-point are effective, but the actions from mid-point to origin won't be mirrored.
4. The G29 command in mirror image function is valid for the mirroring from mid-point.

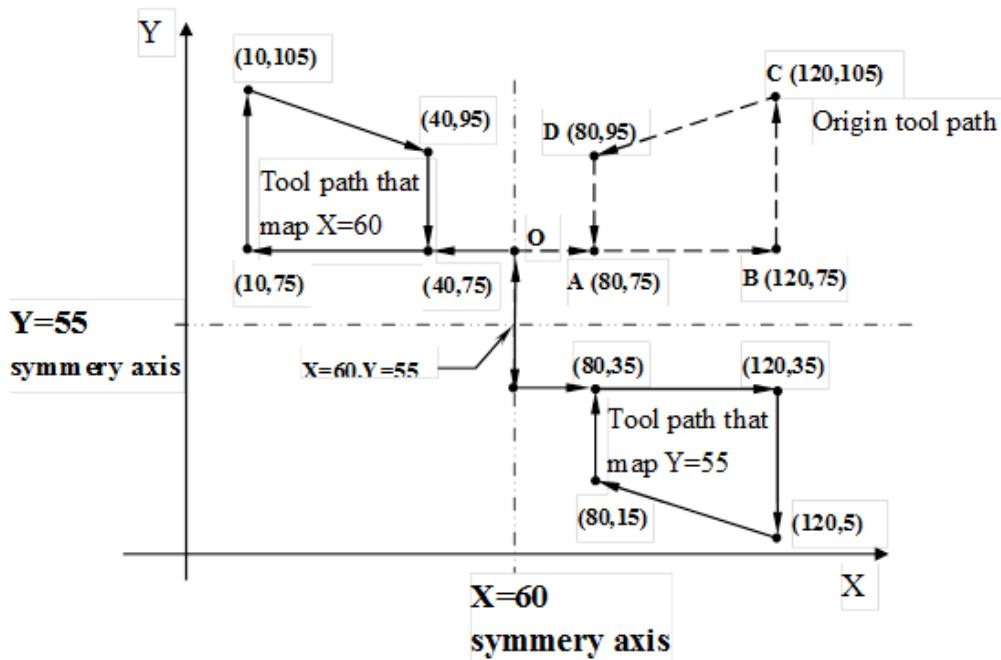
2.35.3 **Note**

1. Cancel the mirror image function outside the center point will make the absolute value not able to match with the machine location, shown in the picture below (the state will last till the program executes G90, G28 or G30 command).
2. If the center point is specified again in unmovable state of absolute value, the center point might be specified to an unpredictable location.
3. Please run the orientation command after the mirror image function is cancelled.
4. Do not apply this code with G51.



2.35.4 **Program Example**

Example 1:



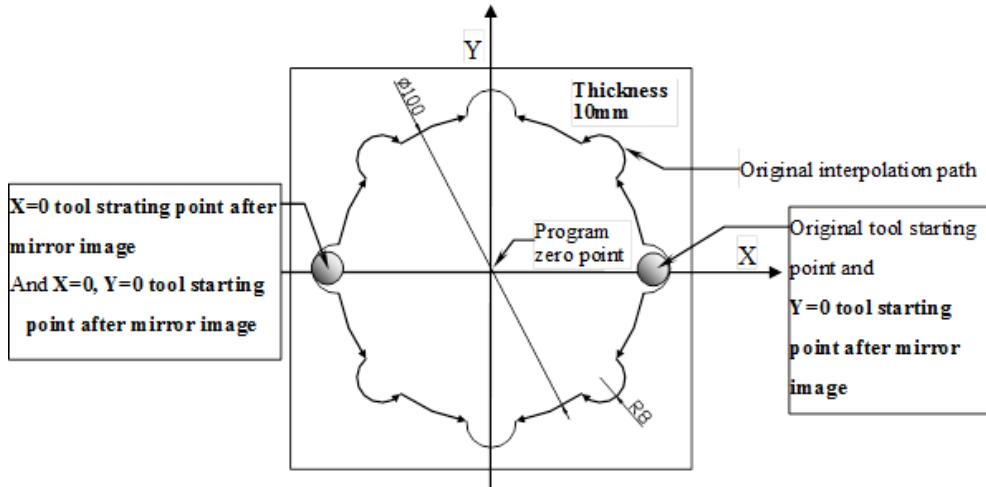
Program description:

```

N001 T1 S1000 M03; // apply tool NO. 1, 1000rpm(CW)
N002 M98 H100; // execute the sub-program
N003 G51.1 X60.0; // execute programmable mirror image with symmetry axis X=60
N004 M98 H100; // execute the sub-program
N005 G50.1; // cancel programmable mirror image function
N006 G51.1 Y55.0; // execute programmable mirror image with symmetry axis Y=55
N007 M98 H100; // execute the sub-program
N008 G50.1; // cancel the programmable mirror image function
N009 M05; // spindle stop
N010 M30; // end program
N100; // sub-program list
G00 X60.0 Y55.0; // rapid orientation to the specified point
G01 Y75.0; // linear cutting to point O
X80.0; // O->A
X120.0; // A->B
Y105.0; // B->C
X80.0 Y95.0; // C->D
Y75.0; // D->A
M99; // end sub-program

```

Example 2:



Program description: process a flower shaped trough

```

N001 T1 S1000 M03; // tool No.1 (diameter 10mm), 1000rpm(CW)
N002 G41 D01; // set up the left cutter radius compensation of tool No.1(D01 = 5)
N003 M98 H100; // execute the sub-program
N004 G42 D01; // set up the right cutter radius compensation of tool No.1
N005 G51.1 X0.0; // execute the mirror image function with symmetry axis X=0
N006 M98 H100; // execute the sub-program
N007 G50.1; // cancel the mirror image function
N008 G41 D01; // set up the left cutter radius compensation of tool No.1
N009 G51.1 X0.0 Y0.0; // execute the mirror image function with symmetry point X=0, Y=0
N010 M98 H100; // execute the sub-program
N011 G50.1; // cancel the mirror image function
N012 G42 D01; // set up the right cutter radius compensation of tool No.1
N013 G51.1 Y0.0; // execute the mirror image function with symmetry axis Y=0
N014 M98 H100; // execute the sub-program
N015 G50.1; // cancel the mirror image function
N016 G40; // cancel cutter compensation
N017 M05; // spindle stop
N018 M30; // end program

```

Sub-program

```

N100; //sub-program code
G00 X58.0 Y0.0 Z10.0; // rapid orientation to above the starting position
G01 Z-10.0; //linear cutting to bottom of trough

```

```

G03 X49.36 Y7.9744 R8.0; // CCW arc cutting, radius 8mm
G03 X40.5415 Y29.2641 R50.0; // CCW arc cutting, radius 50mm
G03 X29.2641 Y40.5415 R8.0; // CCW arc cutting, radius 8mm
G03 X7.9744 Y49.36 R50.0; // CCW arc cutting, radius 50mm
G03 X0.0 Y58.0 R8.0; // CCW arc cutting, radius 8mm
G00 Z10.0; // rapid tool retract to above the end point
M99 ;// end sub-program and continue the main program

```

2.36 G52 : Local Coordinate System Setup

2.36.1 Command Form

G52 X__ Y__ Z__ ;

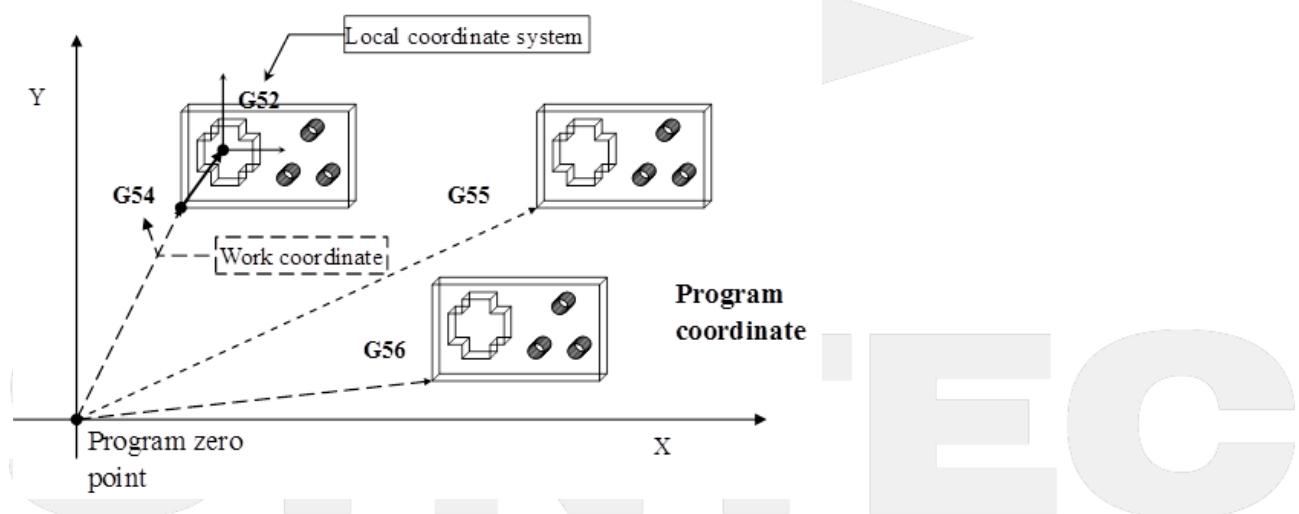
X, Y, Z: coordinate value setup

2.36.2 Description

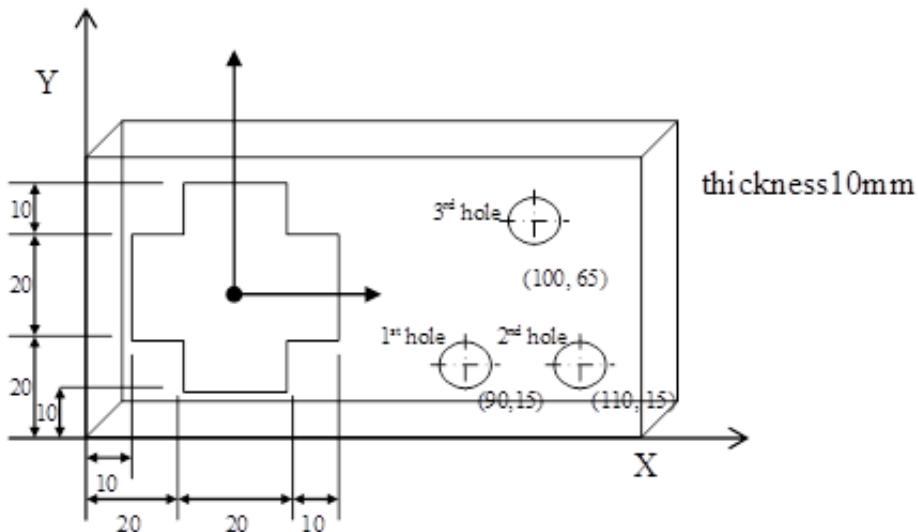
When a working coordinate system (G54~G59) is specified, if another sub-coordinate system is required for the machining workpiece, the sub-coordinate system is the local coordinate system.

G52 X0.0 Y0.0 Z0.0: cancel the local coordinate system

2.36.3 Coordinate System



2.36.4 Program Example



Program description:

```

N001 T1 S1000 M03; // tool No.1 (diameter 10mm), spindle 1000rpm (CW)
N002 G54 X0.0 Y0.0 Z0.0; // specify the working coordinate system (G54)
N003 G00 X90.0 Y15.0 Z10.0; // rapid orientation to above the specified position
N004 G43 H01; // activate tool length compensation (tool No.1)
N005 G99 G81 Z-15.0 R2.0 F1000; // execute the drilling cycle, stop at point R when return, feedrate 1000mm/min,
drill hole NO.1
N006 X110.0; // drill hole NO.2
N007 X100.0 Y65.0; // drill hole NO.3
N008 G80; // cancel cycle
N009 M05; // spindle stop
N010 G28 X0.0 Y0.0 Z10.0; // return to reference point, middle point X0.0, Y0.0, Z10.0
N011 T2 M06 S1000 M03; // execute the tool change (tool No.2, diameter 10mm), spindle rotation start after tool
exchange, 1000rpm (CW)
N012 G52 X30.0 Y30.0 Z0.0; // specify the zero point of local coordinate to X30.0, Y30.0, Z0.0 on the coordinate
system (G54) (geometry center of workpiece)
N013 G00 X0.0 Y0.0 Z10.0; // rapid orientation to X0.0, Y0.0, Z10.0 on local coordinate (above the hole)
N014 G01 Z-12.0; // linear interpolation to bottom of the hole
N015 G17 G41 D02; // activate left cutter radius compensation (tool No.2)
N016 G91 X20.0; // specify the interpolation movements to be executed with increment value
N017 Y10.0;
N018 X-10.0;
N019 Y10.0;

```

```

N020 X-20.0;
N021 Y-10.0;
N022 X-10.0;
N023 Y-20.0;
N024 X10.0;
N025 Y-10.0;
N026 X20.0;
N027 Y10.0;
N028 X10.0;
N029 Y10.0;
N030 G90 G00 Z10.0; // specify the rapid orientation to be executed with absolute value (tool retract)
N031 G52 X0.0 Y0.0 Z0.0; // cancel the local coordinate system
N032 G40 M05; // deactivate the compensation, spindle stop
N033 M30; // program end

```

2.37 G52.1/G52.2 : Axis Removal/Axis Borrowing Function

2.37.1 Instruction

G52.1 P_ Q_ R_

P_ Q_ R_ : Remove the axis name corresponding to the axis, the range: 100~999, the axis name refers to the last three codes of **Pr321~Pr340 Axis name**.

G52.2 P_ Q_ R_ [I_] [J_] [K_]

P_ Q_ R_ : Borrow the axis name corresponding to the axis, the range: 100~999, the axis name refers to the last three codes of **Pr321~Pr340 Axis name**.

I_ : Waiting for response setting, range: 0~2, if not set, the default value is zero.

0 : Wait at this block of NC program until the axes to be borrowed are all successfully borrowed, then execute the next block of NC program;

1 : If all the specified axes cannot be successfully borrowed, an alarm COR-364 Axis borrowing function failed to borrow will be issued;

- If the J argument is set, the user can judge whether the borrowing is successful by using the # value specified by the J argument in MACRO.
- If the K argument is set, before successfully borrowing all the specified axes, the axis group will wait for the time specified by the K argument in this block of NC program. An alarm will be issued when it cannot be successfully borrowed after the time has elapsed.

2 : If there is a axis cannot be borrowed successfully, then do not borrow any axis in this command and no alarm will be issued. Continue to execute the next block of NC program.

- If the J argument is set, the user can judge whether the borrowing is successful by using the # value specified by the J argument in MACRO.

- If the K argument is set, before successfully borrowing all the specified axes, the axis group will wait for the time specified by the K argument in this block of NC program. The next NC block will be executed when it cannot be successfully borrowed after the time has elapsed.

J_ : The # variable that stores the borrowing result information, the range: 27~400 (corresponding to #27~#400), if it is not set, the borrowing result will not be returned to any variable.

The meaning of the borrowing result information is:

- 0: The borrowing of any specified axis in this block failed;
1: The borrowing of all specified axes in this block is successful.

K_: Waiting response delay time (with a decimal point, in seconds; without a decimal point, in milliseconds. Use range: 0.001 to 9999.999 seconds).

2.37.2 Description

When multiple axis groups need to use one axis in turn, the axis can be set as a roaming axis. By using G52.1, G52.2 command to switch the control of roaming axis among axis groups, that ensures the axis will not be controlled by two axis groups at the same time. The above operations also ensure the position synchronization between axis groups.

Setting reference for roaming axis: **Pr742 *The rules of the axis group shared axis**

2.37.3 Precautions

- About arguments:
 - P, Q, R:
 - A row of G52.1 and G52.2 can be removed or borrowed from at least one and at most three axes, corresponding to the three arguments of P, Q and R respectively.
 - If G52.1, G52.2 do not set P, Q, R arguments, an alarm COR-363 Invalid axis removal/axis borrowing function will be issued.
 - If the axis corresponding to the P, Q, R arguments cannot be found, an alarm **COR-363 Invalid axis removal/axis borrowing function** will be issued.
 - It is not supported to remove or borrow axis through virtual axis name. Virtual axis reference: G10L800, G10L801.
 - I :
 - The argument I determines the waiting for response of the three axes P, Q and R at the same time.
 - J :
 - After the borrowing instruction ends, the borrowing result will be returned to a # variable, and the variable number is determined by the J argument.
 - For example: G52.2 P100 Q200 R300 I2 J27. Because the third axis cannot be borrowed, no axis is borrowed in a row of G52.2. At the end of this instruction, #27 is filled with 0.
 - If the argument I is set to 0 or not set, storage of the borrowing result is not supported, and the system will ignore the J argument setting.
 - K :
 - If the argument I is set to 0 or not set, the waiting response delay time is not supported, and the system will ignore the K parameter setting.
 - If the argument P, Q, R, I, J are not integers, an alarm COR-146 Single block argument type error will be issued.
 - If the argument is an integer but exceeds the settable range, an alarm **COR-363 Invalid axis removal/axis borrowing function** will be issued.

- Axis borrowing or removal is only applicable to roaming axes, and an alarm **COR-363 Invalid axis removal/axis borrowing function** will be issued for borrowing or removing command under general axes.
- Before the axis borrowing or removing, the coordinate will decelerate to zero first, and then the axis borrowing or removing command will be processed.
- When the argument I is set to 0 for borrowing, the multiple axis groups may wait for the roaming axis that is borrowing by the other coordinate, which may cause the system to be stuck in waiting for borrowing.
 - Solution: Modify the NC program to make sure the other coordinate remove roaming axis first.
- If the axis to be removed is not borrowed by the coordinate, an alarm **COR-363 Invalid axis removal/axis borrowing function** will be issued.
if the axis to be borrowed has been borrowed by the coordinate, an alarm **COR-363 Invalid axis removal/axis borrowing function** will be issued.
- The number of axes that can be borrowed by a axis group is limited to the total number of roaming axes belongs to the axis group. Multiple G52.2 command can be issued to borrow all roaming axes.
- For non-CNC main system axis groups, by writing G52.1, G52.2 commands in the sub-program specified by the main program number (ex: R532) of each axis group to achieve the roaming axes removal or borrowing.
- For the PLC Rn sub-program components, by writing G52.1 and G52.2 commands in the designated processing program to achieve the roaming axes removal or borrowing.
- The non-linear kinematic transform related axes are not allow to set as roaming axes. The following conditions will issue an alarm **COR-363 Invalid axis removal/axis borrowing function**. Include:
 - RTCP(G43.4, G43.5)
 - Tilted working plane machining(G68.2, G68.3 + G53.1, G53.3, G53.6)
 - Polar Coordinates Interpolation(G12.1)
 - General 2D Kinematic Transform(Special Machine)
- It is forbidden to use the break point return in the path with the roaming axis command, because it may move to the unborrowed roaming axis, otherwise an alarm COR-365 Issue a movement command to the unborrowed roaming axis will be issued.
- Before the axis group or PLC Rn subprogram component borrows the roaming axis successfully, it is not allowed to issue a command to the axis, otherwise an alarm COR-365 Issue a movement command to the unborrowed roaming axis will be issued.
- Supported version: 10.118.42R, 10.118.48C, 10.118.50 and later versions.

2.37.4 **Example**

Example 1: Multi-axis groups execute the same axis in turn

Pr321~Pr324 Axial name = {101, 200, 300, 102}

(There are four axes X1, Y, Z, X2 in the system)

Pr701~Pr704 Axial belonging axis group = {1, 3, 3, 2}

Pr742 The rules of the axis group shared axis = 1

(Y, Z axes are roaming axes, the first and second axis groups can borrow Y, Z axes)

\$1	\$2
<pre>// Initial state: owned axis X1. G04.1 P1; G52.2 P200 Q300 I0; // Borrow Y, Z axes. G04.1 P2; G52.1 P200 Q300; // Remove Y, Z axes. G04.1 P3; #400 := -1; G52.2 P200 Q300 I2 J400 K5.; // Borrow Y, Z axis, wait for 5 seconds to continue to execute the next block, #400 is written as 0. #399 := -1; G52.2 Q300 I2 J399; // Borrow Z axis successfully, #399 is written as 1. G04.1 P4; // Ensure that \$1 M30 will not be executed too early. M30;</pre>	<pre>// Initial state: owned axis X2. G04.1 P1; // Synchronize with \$1, avoid \$2 M99 back to head of the program and continue to execute. G04.1 P2; // Make sure to borrow Y, Z axis from \$1 first. G52.2 P200 I1 K5.; // Borrow Y axis(may not be borrowed at first, wait 5 seconds to borrow successfully). G04.1 P3; // Ensure that \$2 will borrow the Y axis successfully. G04.1 P4; M99;</pre>

Example 2: Determine the follow-up processing path based on whether the borrowing is successful or not

Pr321~Pr322 Axial name = {100, 200}

(There are X, Y axes in the system)

Pr701~Pr702 Axial belonging axis group = {3, 3}

Pr742 The rules of the axis group shared axis = 1

(X, Y axes are roaming axes, the first and second axis group can borrow X, Y axes)

```
%@MACRO
G10 L1000 P6000 R0;      // R6000=0 cycle state flag(busy)
#27:=#0;                  // Clear the # value of the return value
G52.2 P100 Q200 I2 J27;   // If both X and Y axes can be borrowed, then two axes will be borrowed.
                           // if any one of the X or Y axis cannot be borrowed successfully, then neither axis will be
borrowed. The borrowing result will be returned to #27.
IF (#27 = 0 ) THEN       // Use #27 to determine the next machining NC program
  G10 L1000 P6000 R999;  // R6000=999 cycle state flag(fail), trigger PLC to change machining NC program
file(not including X, Y axes machining)
  M30;
END_IF
//... Including X, Y axes machining...//
G52.1 P100 Q200;         // Remove X, Y axes
G10 L1000 P6000 R1;      // R6000=0 cycle state flag(finish)
M30;
```

2.38 G53 : Machine Coordinate Orientation

2.38.1 Command Form

G53 [P1] X____ Y____ Z____ [F1=_];

X, Y, Z: specified position.

P1: active constant speed command

F1: feedrate mm/min or inch/min

2.38.2 Description

The machine zero point is a fixed point set by the machine manufacturer while building the CNC machine, it's a fixed and unchangeable coordinate system. When G53 coordinate command is specified, the tool will move to the specified position on machine coordinate, when it returns to the machine zero point (0,0,0), the point is the zero point of machine coordinate system.

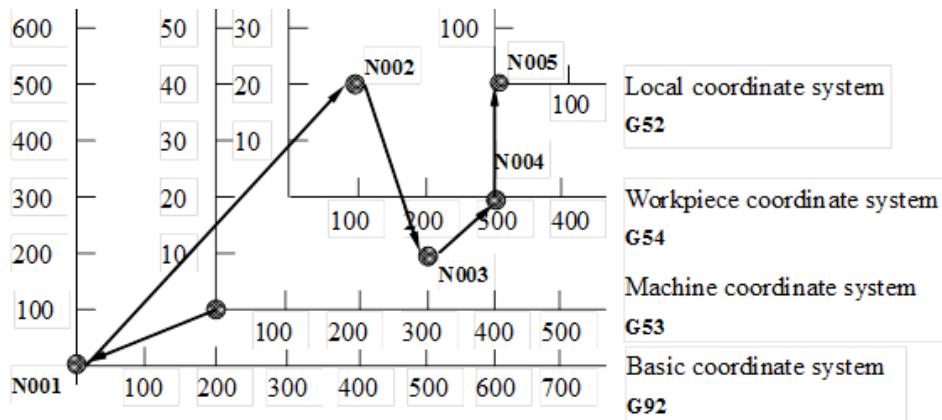
2.38.3 Note

1. G53 only affects in the specified block (returns to program coordinate system if only the coordinate arguments are given in the next block);
2. G53 only works in the absolute mode (G90), it'll simply run the increments in increment mode(G91);
3. If Pr3809 set 1 and G53 is specified together with the UVW command it won't be taken as the XYZ axis inc. command.
4. Before specifying G53, please cancel the related cutter radius compensation, tool length compensation or position compensation;
5. Before setting up the coordinate system with G53, please build the coordinate system manually according to the reference point return.
6. If the axis type(Pr221~236) is set to be rotary axis, please refer to "Syntec CNC Parameter" Pr221~236 "Axis type".
7. G53 default feedrate is feedrate of G00.
8. Constant feedrate is legally with both P1 and F1 argument are determined. If P1 argument is missing, system wouldn't refer F1 argument as moving speed.
9. G70/G71 supported unit of F1 can be mm/min or inch/min.
10. Unit of F1 can be mm/min, but always remains G94. G93/G95 is invalid.
11. F1 argument support version: 10.118.41P, 10.118.48A, 10.118.49 and after.



2.38.4 Program Example

Example1:



Program description:

```
N001 G92 X-200.0 Y-100.0; // specify the basic coordinate system
N002 G54 G90 X100.0 Y200.0; // move to the specified position on workpiece coordinate system
N003 G53 X300.0 Y100.0; // move to the specified position on machine coordinate system
N004 X300.0 Y0; // since G53 only affects in the specified block, this block continues the G54 command and moves to
the specified position on workpiece coordinate system
N005 G52 X300.0 Y200.0; // set local coordinate system to the specified position on workpiece coordinate system
N006 X0.0 Y0.0;
```

Example2:

```
G71;
G53 X100. Y100.; // G53 feedrate is feedrate of G00
G53 P1 X50. Y50. F1=1000.; // G53 feedrate is F1=1000
G01 X0. Y0.;
M30;
```

2.39 G53.1 : Tool Alignment for Tilted Working Plane Machining

2.39.1 Command Form

G68.2 X_ Y_ Z_ I_ J_ K_;
G53.1 [P_];

G68.2 : enable the tilted working plane coordinate system;
G53.1 : tool alignment function;

P: select the moving direction of rotary axis,

0 : the 1st rotary axis (Master axis) moves along with the shortest contour;(default value)

1 : the 1st rotary axis rotates towards positive direction;

2 : the 1st rotary axis rotates towards negative direction;

Before execute the machining process, please apply G53.1 or G53.6 after G68.2 so the tool can be aligned to the tilted working plane coordinate system.

2.39.2 Description

After the tilted working plane coordinate system is set, G53.1 is required for the tool alignment. The G53.1 command is attached to G68.2, they must be applied at the same time.

2.39.3 Note

1. Do not specified G53.1 before G68.2.
2. Please apply positive tool length (G43 after G53.1)
3. After G43 is executed, the program coordinate would represent the position of tool center point. User should apply G49 after tilted working plane machining to cancel tool center point control.
4. The P argument will be 0 (default value) if it's not specified.
5. If specified arguments that are not P0, P1, P2, alarm [COR-149 Tilted working plane machining tool alignment P argument over range] will be issued.
6. When the argument is set 0, the system will search for the shortest moving contour for 1st rotary axis (Master axis) first. If the target angle or the route to it is out of range (Pr3009~), the other target angle will be selected instead; if both target angles or routes to them are out of range(Pr3009~), alarm [COR-153 No solution for the tool direction] will be issued.
7. When the argument is set 1 or 2, if the target angle or the route to it is out of range(Pr3009~), alarm [COR-153 No solution for the tool direction] will be issued.
8. For rotary axis definitions corresponding to different mechanisms, please refer to 1.3 旋转轴定义 and 1.4 参数说明。

	0(default)	1	2
Spindle/Table/ Mixed	1st rotary axis (Master axis)/shortest path	1st rotary axis (Master axis)/positive	1st rotary axis (Master axis)/negative

2.39.4 Program Example

Take the program below as example, below explains the basic actions of tilted working plane coordinate

```

N1 G90 G54 G01 X0 Y0 Z50. F1000 ;
N2 G68.2 X100. Y100. Z50. I30. J15. K20. ;
N3 G01 X0 Y0 Z50. F1000 ;
N4 G53.1 ;
N5 G43 H1 ;
N6 G01 X0 Y0 Z0 ;

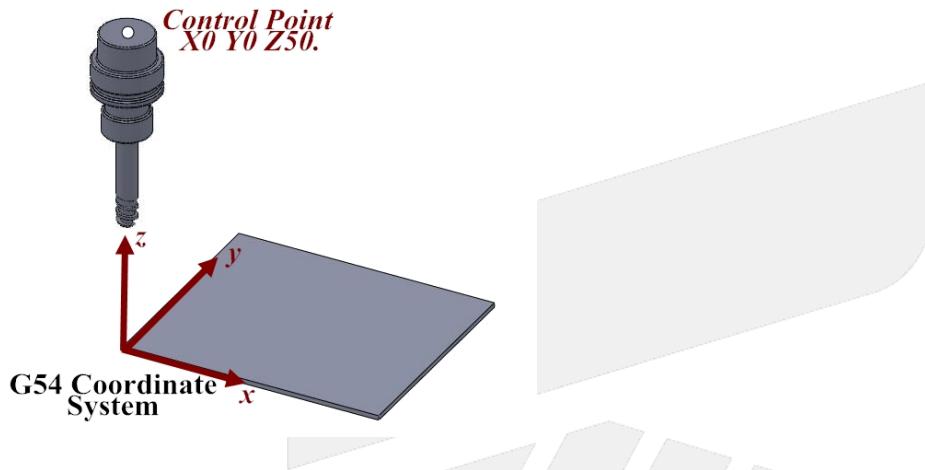
... // tilted working plane machining

N98 G49 ;
N99 G69 ;
N100 G01 X0. Y0. Z50. ;

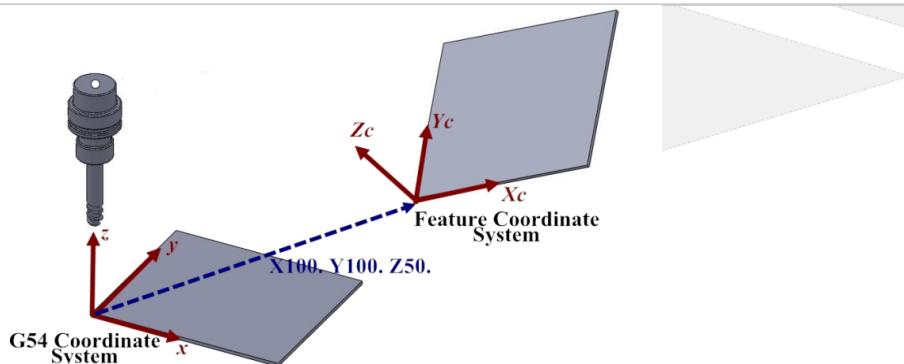
```

The example will be explain line by line below:

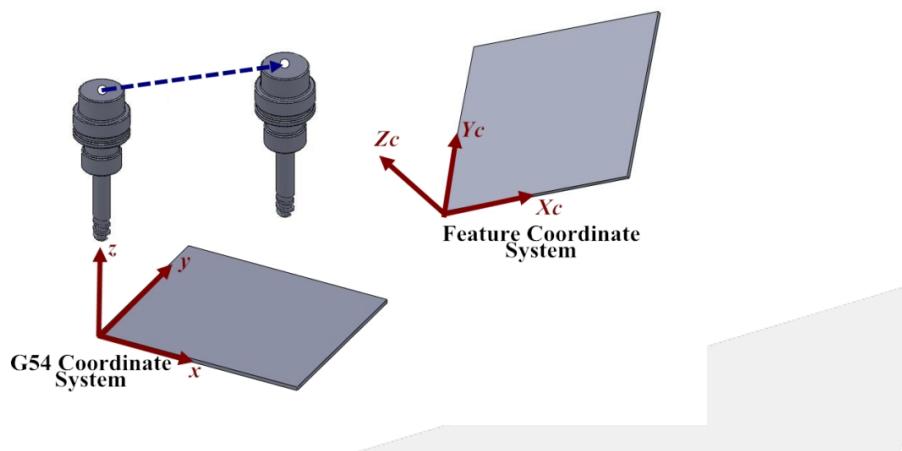
```
N1 G90 G54 G01 X0 Y0 Z50. F1000;
// interpolation to Z50 on G54 coordinate system in the speed at F1000 speed
```



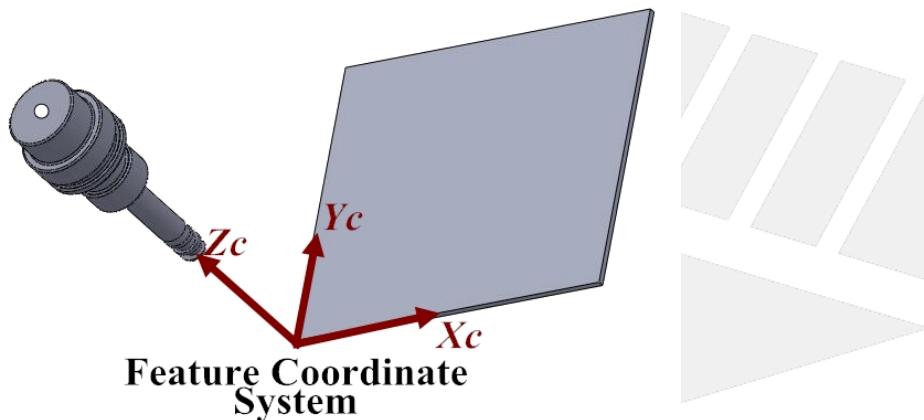
```
N2 G68.2 X100. Y100. Z50. I30. J15. K20. ;
// specify X100. Y100. Z50. on G54 coordinate system as the zero of tilted working plane coordinate system, and set the Euler angle I30. J15. K20. The program coordinate system will change to the tilted working plane coordinate system after command G68.2 is executed.
```



```
N3 G01 X0 Y0 Z50. F1000;
// interpolation to Z50. of tilted working plane coordinate system at F1000 speed,
but the tool direction remains the same
```

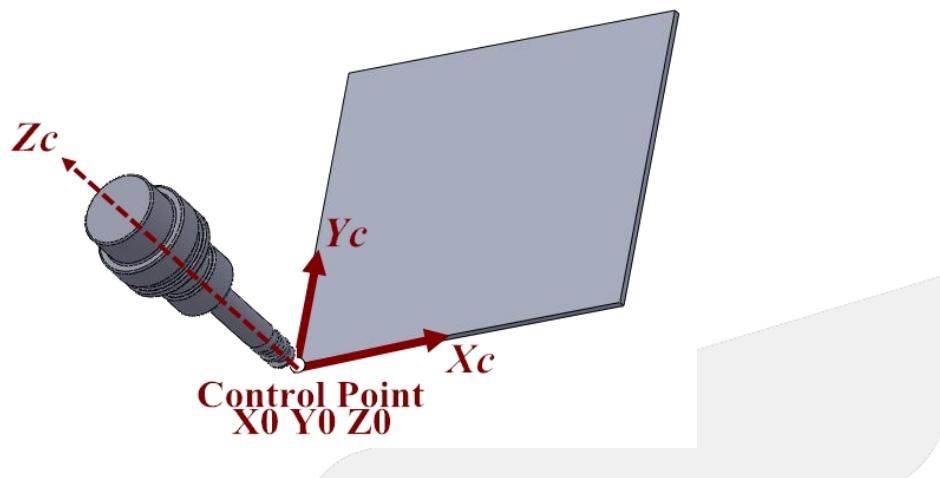


```
N4 G53.1;
// the tool direction automatically aligns to the Z axis of tilted working plane
coordinate system
```

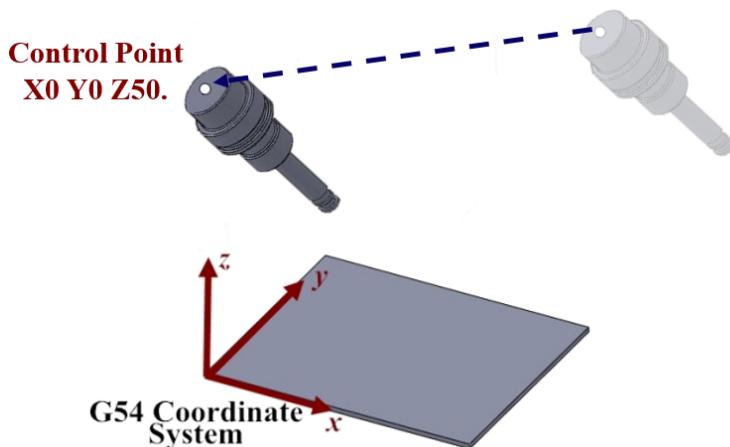


```
N5 G43 H1;
// tool length compensation, the control point changes to the position of tool center
point.

N6 G01 X0 Y0 Z0;
// interpolation to the X0 Y0 Z0 position of tilted working plane coordinate system.
```



```
N98 G49 ;
// cancel tool center point control
N99 G69 ;
// cancel tilted working plane coordinate system
N100 G01 X0. Y0. Z50. ;
// interpolation to X0. Y0. Z50. of G54 coordinate system
```



2.39.5 Appendix

Singular point situation

When a tool vector can be achieved by "multiple position" of one rotary axis, this rotary axis is said to be in a **singular point**.

After the working plane coordinate tilting(G68.2, G68.3), there's a chance that a rotary axis is in a singular point.

Rotary axis in singular point will be fixed at its current program position after a tool alignment. And the new working plane coordinate will be based on this fixed rotary position, too.

⚠ Because of singular points, one tool vector can map to multiple rotary angle answers, depending on your current rotary position.

This might cause different results of absolute position from one machine position under different mechanical chain settings.

EX1:

Assume a five-axis milling machine with +Z direction has work piece coordinate setting and position as follow:

G54P1(G54)		Machine		Absolute		Fig.
X	0.000	X	0.000	X	0.000	
Y	0.000	Y	0.000	Y	0.000	
Z	0.000	Z	0.000	Z	0.000	
A	0.000	A	0.000	A	0.000	
C	0.000	C	0.000	C	0.000	

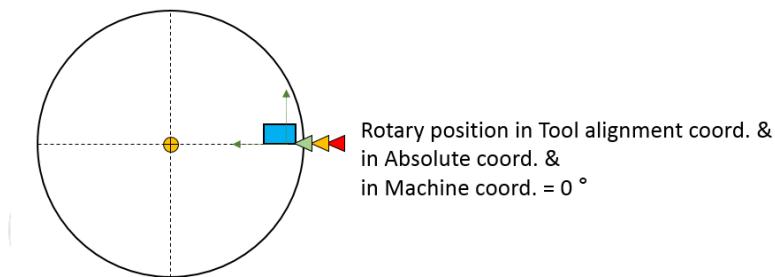
Now, execute commands below:

1	G68.2 X0. Y0. Z0. I0. J0. K0.
2	G53.1

According to working plane coordinate tilting at L1, tool direction should be +Z still.

To make tool direction +Z, axis A should be fixed at 0.000 degree, whereas axis C can be at ANY angle. That is, axis C is in a singular point.

Since axis C is in a singular point, it will be fixed at 0.000 degree(in absolute coordinate).



EX2:

Assume a same situation like EX1, except the work piece coordinate offset of C axis is 10.000 degree:

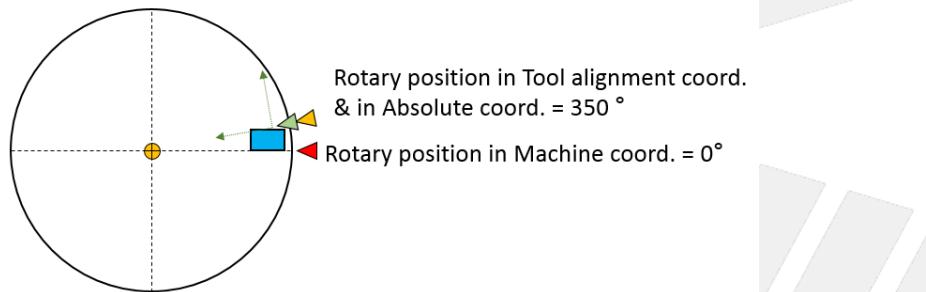
G54P1(G54)		Machine		Absolute		Fig.
X	0.000	X	0.000	X	0.000	
Y	0.000	Y	0.000	Y	0.000	

G54P1(G54)		Machine		Absolute		Fig.
Z	0.000	Z	0.000	Z	0.000	
A	0.000	A	0.000	A	0.000	
C	10.000	C	0.000	C	350.000	

Execute the same commands as EX1:

1	G68.2 X0. Y0. Z0. I0. J0. K0.
2	G53.1

C axis is in a singular point; thus, it will be fixed at 350.000 degree(in absolute coordinate).



2.40 G53.3 : Tool Alignment for Tilted Working Plane Machining(Five-Axis Simultaneous Machining)

2.40.1 Command Form

G68.2 X_ Y_ Z_ I_ J_ K_;
G53.3 [X_] [Y_] [Z_] [H_] [P_];

G68.2 : enable the tilted working plane coordinate system;
G53.3 : tool alignment and position function;

X_ Y_ Z_: specified position.

H: Tool number;

P: select the moving direction of rotary axis,

0 : the 1st rotary axis (Master axis) moves along with the shortest contour;(default value)

1 : the 1st rotary axis rotates towards positive direction;

2 : the 1st rotary axis rotates towards negative direction;

Before execute the machining process, please apply G53.1, G53.3, or G53.6 after G68.2 so the tool can be aligned to the tilted working plane coordinate system.

2.40.2 Description

Applying G53.3 after the tilted working plane coordinate system is set will leads to the following simultaneous actions:

1. Activate tool compensation along positive tool direction. The tool compensation number is equal to G53.3's H argument.
2. Align the tool with the tilted working plane.
3. Rapid travel to the specified position which is specified by G53.3's XYZ arguments.

The G53.3 command is attached to G68.2, so they must be applied at the same time.

2.40.3 Note

1. Do not specified G53.3 before G68.2.
2. After G53.3 is executed, the program coordinate would represent the position of tool center point. User should apply G49 after tilted working plane machining to cancel tool center point control.
3. The P argument will be 0 (default value) if it's not specified.
4. If specified P arguments that are not P0, P1, P2, alarm [COR-149 Tilted working plane machining tool alignment P argument over range] will be issued.
5. When the argument is set 0, the system will search for the shortest moving contour for 1st rotary axis (Master axis) first. If the target angle or the route to it is out of range (Pr3009~), the other target angle will be selected instead; if both target angles or routes to them are out of range(Pr3009~), alarm [COR-153 No solution for the tool direction] will be issued.
6. When the argument is set 1 or 2, if the target angle or the route to it is out of range(Pr3009~), alarm [COR-153 No solution for the tool direction] will be issued.
7. For rotary axis definitions corresponding to different mechanisms, please refer to 1.3 旋转轴定义 and 1.4 参数说明。

	0(default)	1	2
Spindle/Table/ Mixed	1st rotary axis (Master axis)/shortest path	1st rotary axis (Master axis)/positive	1st rotary axis (Master axis)/negative

2.40.4 Program Example

Take the program below as example, below explains the basic actions of tilted working plane coordinate

```

N1 G90 G54 G01 X0 Y0 Z50. F1000 ;
N2 G68.2 X100. Y100. Z50. I30. J15. K20. ;
N3 G01 X0 Y0 Z50. F1000 ;
N4 G53.3 X0 Y0 Z0 H1 ;

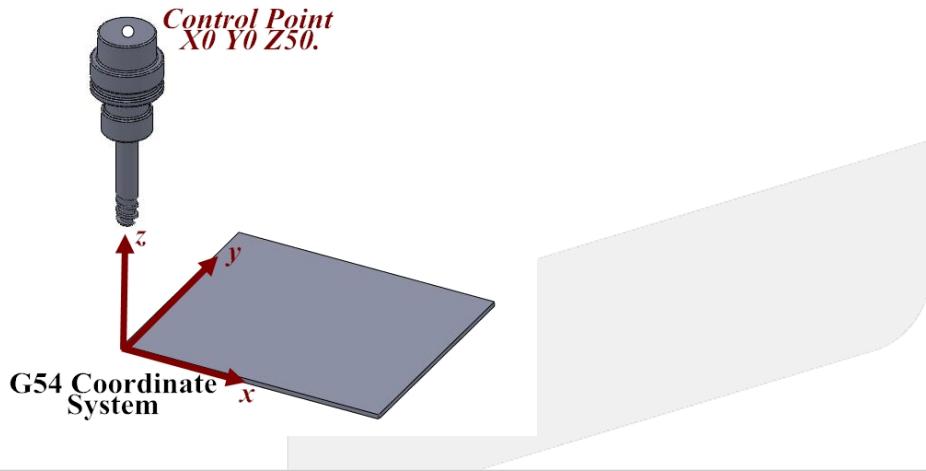
... // tilted working plane machining

N98 G49 ;
N99 G69 ;
N100 G01 X0. Y0. Z50. ;

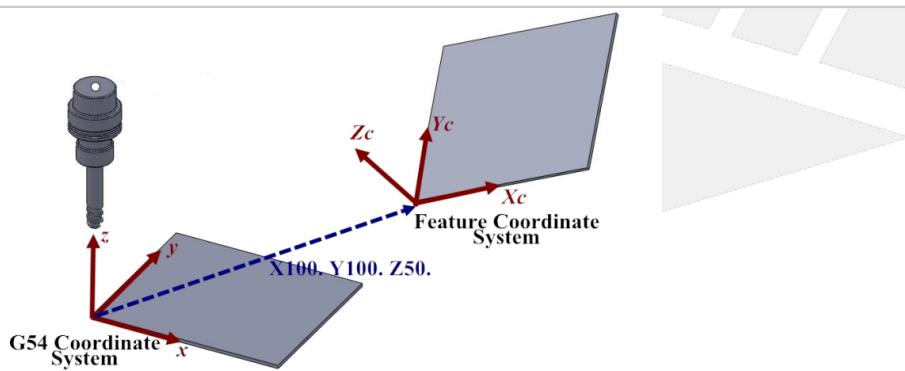
```

The example will be explain line by line below:

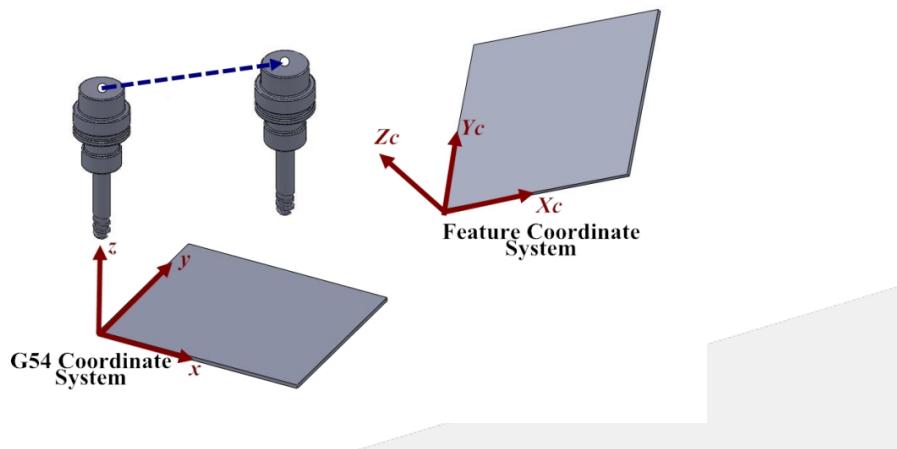
```
N1 G90 G54 G01 X0 Y0 Z50. F1000;
// interpolation to Z50 on G54 coordinate system in the speed at F1000 speed
```



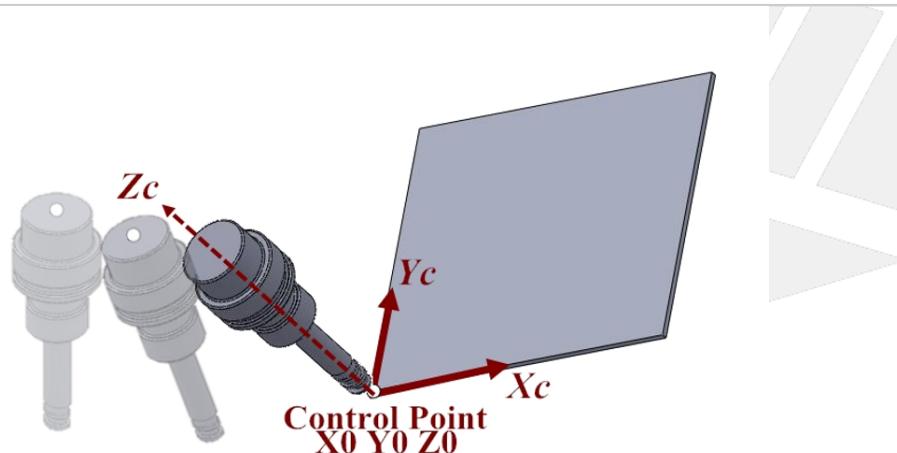
```
N2 G68.2 X100. Y100. Z50. I30. J15. K20. ;
// specify X100. Y100. Z50. on G54 coordinate system as the zero of tilted working
plane coordinate system, and set the Euler angle I30. J15. K20. The program
coordinate system will change to the tilted working plane coordinate system after
command G68.2 is executed.
```



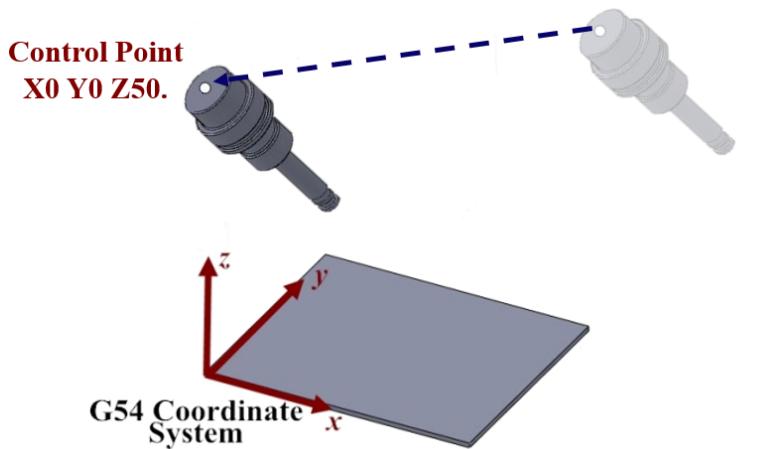
```
N3 G01 X0 Y0 Z50. F1000;
// interpolation to Z50. of tilted working plane coordinate system at F1000 speed,
but the tool direction remains the same
```



```
N4 G53.3 X0 Y0 Z0 H1;
// tool length compensation, the control point changes to the position of tool center
point.
// the tool direction automatically aligns to the Z axis of tilted working plane
coordinate system
// interpolation to the X0 Y0 Z0 position of tilted working plane coordinate system.
```



```
N98 G49 ;
// cancel tool center point control
N99 G69 ;
// cancel tilted working plane coordinate system
N100 G01 X0. Y0. Z50. ;
// interpolation to X0. Y0. Z50. of G54 coordinate system
```



2.40.5 Appendix

Singular point situation

When a tool vector can be achieved by "multiple position" of one rotary axis, this rotary axis is said to be in a **singular point**.

After the working plane coordinate tilting(G68.2, G68.3), there's a chance that a rotary axis is in a singular point.

Rotary axis in singular point will be fixed at its current program position after a tool alignment. And the new working plane coordinate will be based on this fixed rotary position, too.

- ⚠** Because of singular points, one tool vector can map to multiple rotary angle answers, depending on your current rotary position.
This might cause different results of absolute position from one machine position under different mechanical chain settings.

EX1:

Assume a five-axis milling machine with +Z direction has work piece coordinate setting and position as follow:

G54P1(G54)		Machine		Absolute		Fig.
X	0.000	X	0.000	X	0.000	
Y	0.000	Y	0.000	Y	0.000	
Z	0.000	Z	0.000	Z	0.000	
A	0.000	A	0.000	A	0.000	
C	0.000	C	0.000	C	0.000	



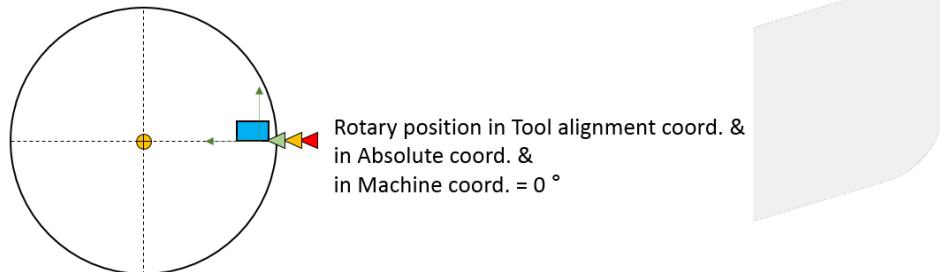
Now, execute commands below:

1	G68.2 X0. Y0. Z0. I0. J0. K0.
2	G53.3

According to working plane coordinate tilting at L1, tool direction should be +Z still.

To make tool direction +Z, axis A should be fixed at 0.000 degree, whereas axis C can be at ANY angle. That is, axis C is in a singular point.

Since axis C is in a singular point, it will be fixed at 0.000 degree(in absolute coordinate).



EX2:

Assume a same situation like EX1, except the work piece coordinate offset of C axis is 10.000 degree:

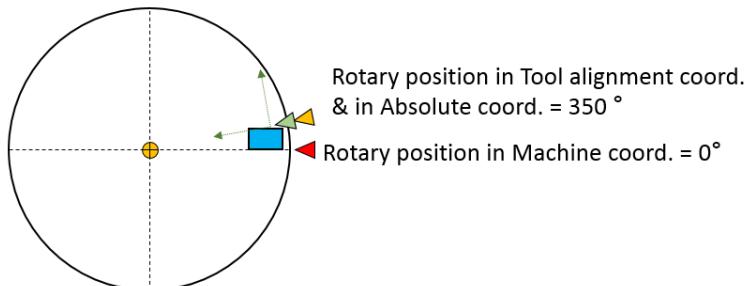
G54P1(G54)		Machine		Absolute		Fig.
X	0.000	X	0.000	X	0.000	
Y	0.000	Y	0.000	Y	0.000	
Z	0.000	Z	0.000	Z	0.000	
A	0.000	A	0.000	A	0.000	
C	10.000	C	0.000	C	350.000	



Execute the same commands as EX1:

1	G68.2 X0. Y0. Z0. I0. J0. K0.
2	G53.3

C axis is in a singular point; thus, it will be fixed at 350.000 degree(in absolute coordinate).



2.41 G53.6 : Tool Alignment for Tilted Working Plane Machining (RTCP/Rotation Center Control)

2.41.1 Command Form

G68.2 X_Y_Z_I_J_K_;
G53.6 [H_] [P_] [R_];

G68.2 : enable the tilted working plane coordinate system;
G53.6 : tool alignment function (RTCP/Rotation center control);

H : tool number, if H code is not specified, the former tool number will be applied; if no former tool number can be found (H0), alarm [MAR-407 G53.6 tool number can't be 0] will be issued.

P: select the moving direction of rotary axis,

0 : the 1st rotary axis (Master axis) moves along with the shortest contour;(default value)

1 : the 1st rotary axis rotates towards positive direction;

2 : the 1st rotary axis rotates towards negative direction;

R : distance from tool center point to rotation center;

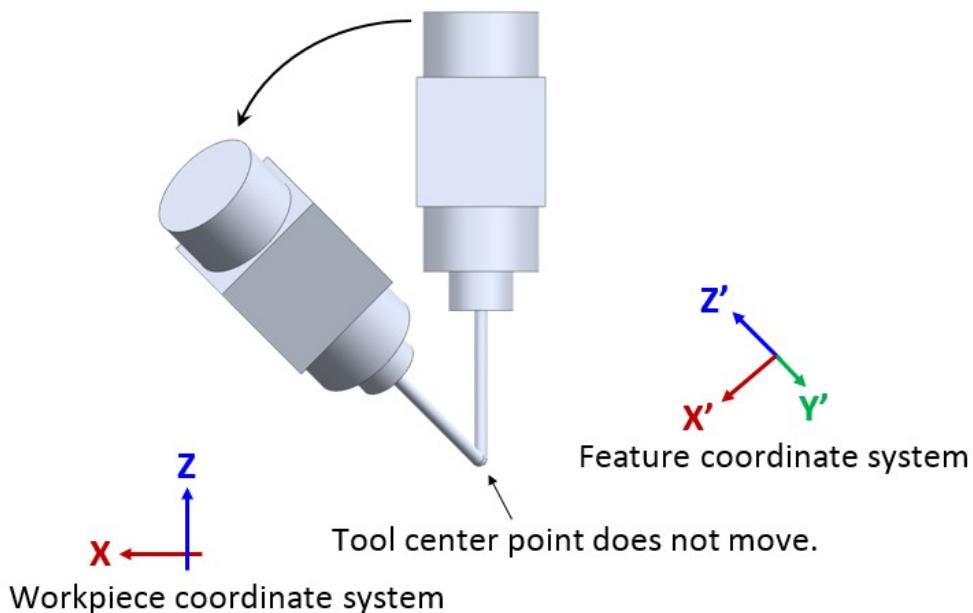
Before execute the machining process, please apply G53.1 or G53.6 after G68.2 so the tool can be aligned to the tilted working plane coordinate system.

2.41.2 Description

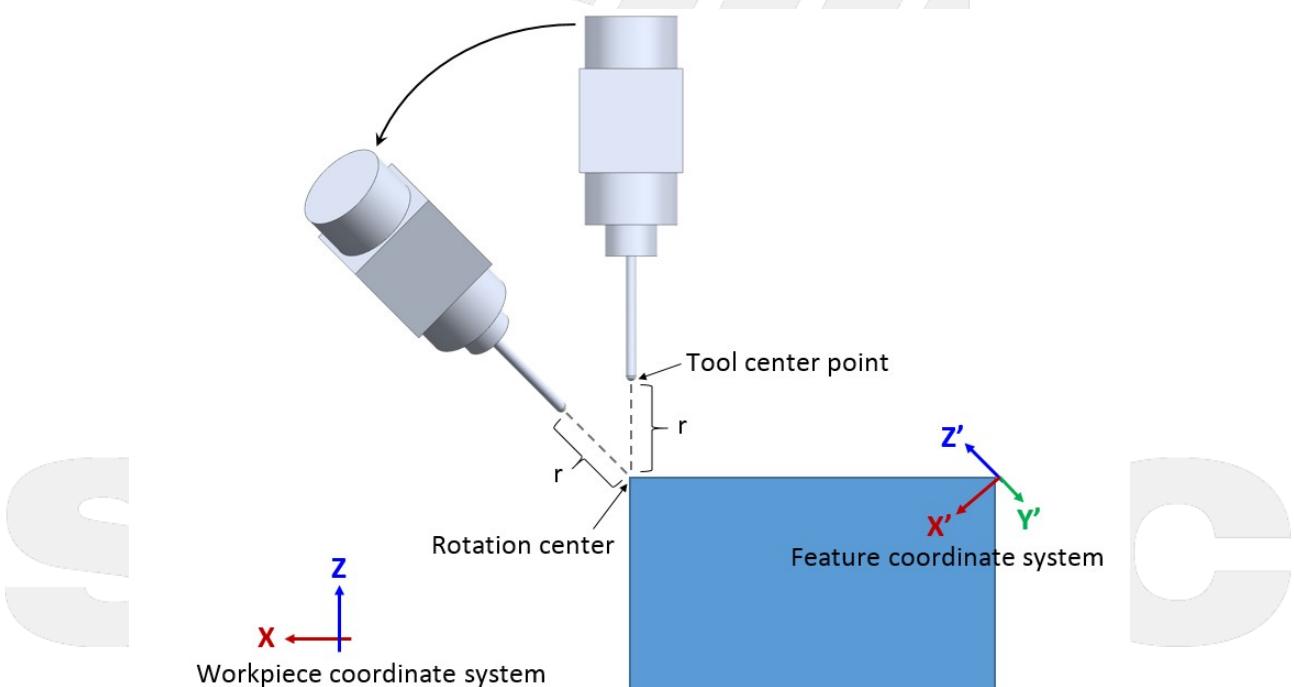
1. After the tilted working plane coordinate system is set, G53.6 is required for the tool alignment. The G53.6 command is attached to G68.2, they must be applied at the same time.
2. Both G53.6 and G53.1 can control the tool direction and make it perpendicular to the tilted working plane, though G53.6 will maintain the distance between tool center point and rotation center while moving the tool. The distance can be specified with G53.6 (R argument); the pictures below shows the difference between with and without R argument:

- **Without R argument** : The tool center point won't change while moving the rotary axis.

The logo consists of the word "SYNTEC" in a bold, sans-serif font. The letters are light gray and have a slight shadow effect, giving them a 3D appearance. The letters are arranged in a single row: S, Y, N, T, E, C. The letter 'Y' is slightly taller than the others, and the 'E' has a distinctive vertical cutout on its left side.



- With R argument (Rr) : The rotation center extended from the tool center point with a distance r won't change while moving the axis.



2.41.3 Note

- Do not specify G53.6 before G68.2.
- Please apply positive tool length (G53.6 is able to assign the tool number with H code)

3. G53.6 executes the tool rotation with the way of RTCP, the moving commands afterwards will also be executed with the position of tool center point. User should apply G49 after tilted working plane machining to cancel tool center point control.
4. Do not specified the G41, G42 cutter radius compensation function before G53.6, or alarm 【MAR-G53.6 must be activated in G40 mode】will be issued.
5. If G53.6 is executed without H argument and the current tool number is 0, alarm 【MAR-407 G53.6 chosen tool number can't be 0】will be issued.
6. The P argument will be 0 (default value) if it's not specified.
7. If enter arguments that are not P0, P1, P2, alarm 【COR-149 Tilted working plane machining tool alignment P argument over range】will be issued.
8. When the argument is set 0, the system will search for the shortest moving contour for 1st rotary axis (Master axis) first . If the target angle or the route to it is out of range (Pr3009~), the other target angle will be selected instead; if both target angles or routes to them are out of range(Pr3009~), alarm 【COR-153 No solution for the tool direction】will be issued.
9. When the argument is set 1 or 2, if the target angle or the route to it is out of range(Pr3009~), alarm 【COR-153 No solution for the tool direction】will be issued.
10. For rotary axis definitions corresponding to different mechanisms, please refer to 1.3 旋转轴定义 and 1.4 参数说明。

	0(default)	1	2
Spindle/Table/ Mixed	1st rotary axis (Master axis)/shortest path	1st rotary axis (Master axis)/positive	1st rotary axis (Master axis)/negative

2.41.4 Program Example

Take the program below as example, below explains the basic actions of tilted working plane coordinate

```

N1 G90 G54 G01 X0 Y0 Z50. F1000 ;
N2 G68.2 X100. Y100. Z50. I30. J15. K20. ;
N3 G53.6 H1 ;
N4 G01 X0 Y0 Z0 ;

... // tilted working plane machining

N98 G49 ;
N99 G69 ;
N100 G01 X0. Y0. Z50. ;

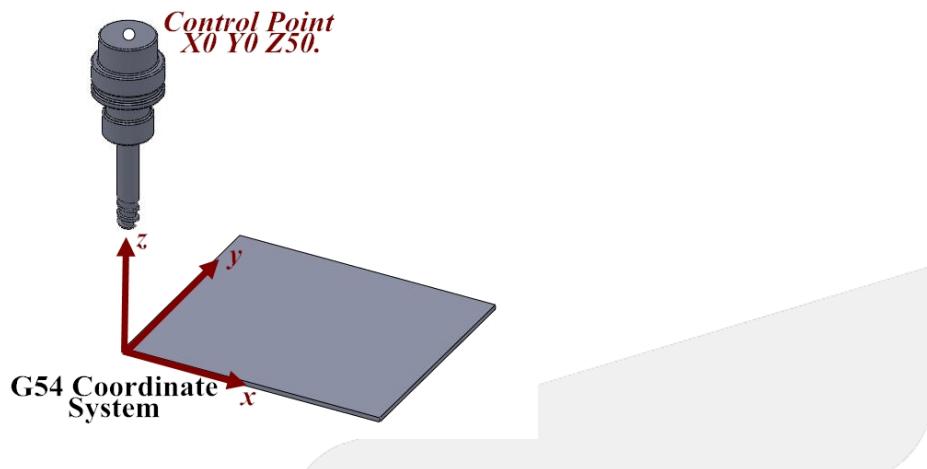
```

The example will be explain line by line below:

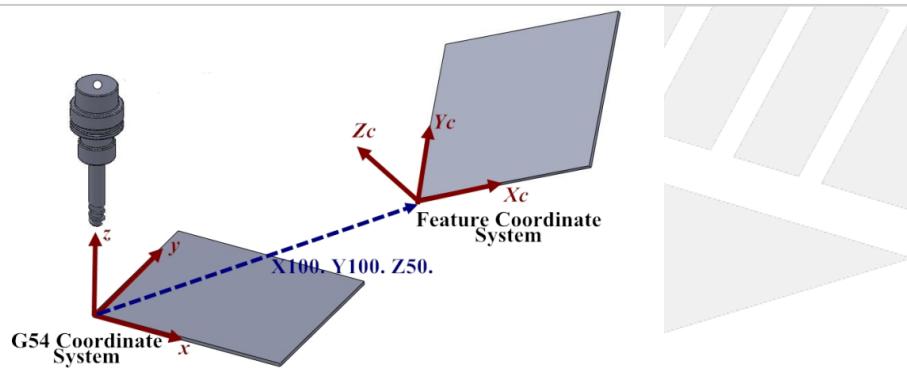
```

N1 G90 G54 G01 X0 Y0 Z50. F1000;
//interpolationto Z50 of G54 coordinate system in the speed of F1000

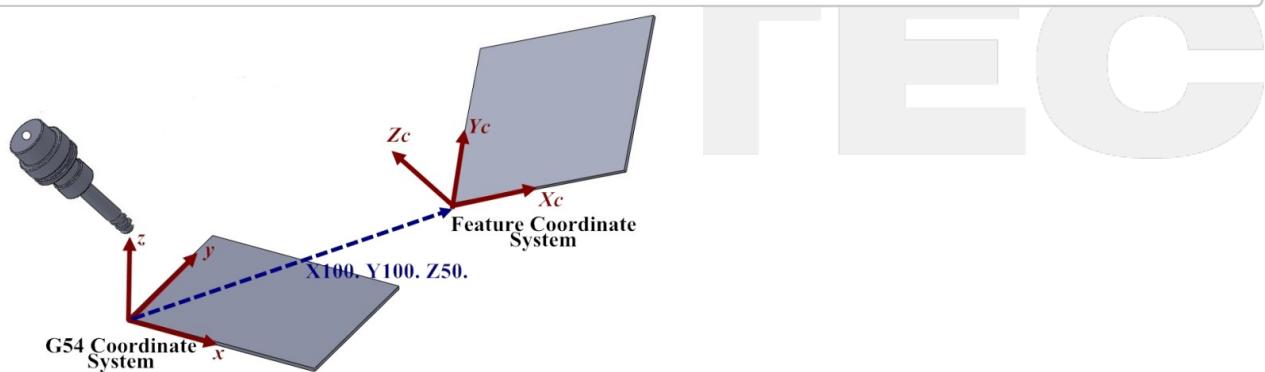
```



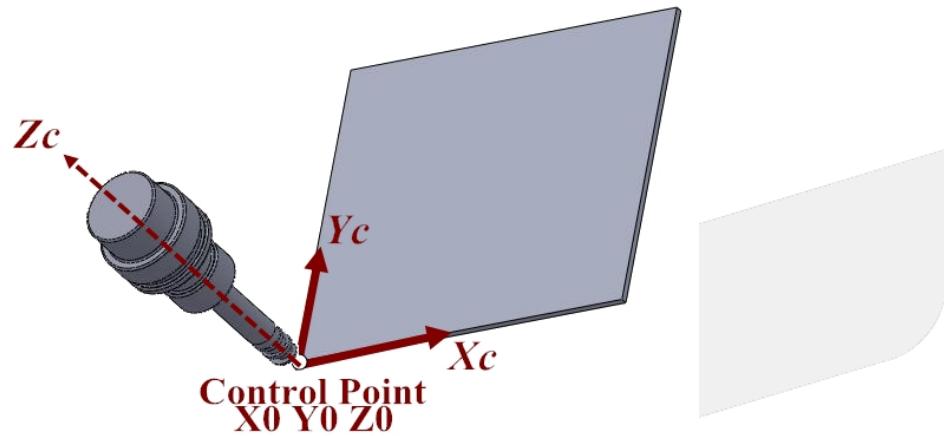
```
N2 G68.2 X100. Y100. Z50. I30. J15. K20.;  
// specify X100. Y100. Z50. on G54 coordinate system as the origin of tilted working  
plane coordinate system, and set the Euler angle I30. J15. K20.  
// The program coordinate system changes to the tilted working plane coordinate  
system after command G68.2 is executed.
```



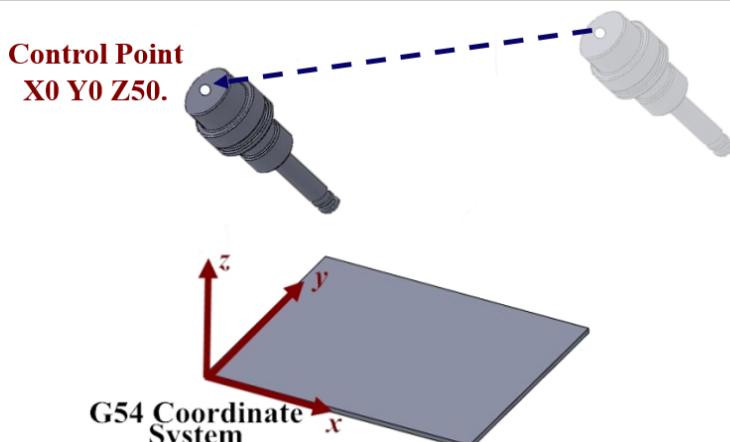
```
N3 G53.6;  
// the tool direction automatically aligns to the Z axis of tilted working plane  
coordinate system
```



```
N4 G01 X0 Y0 Z0;  
// interpolation to the X0 Y0 Z0 position of tilted working plane coordinate system
```

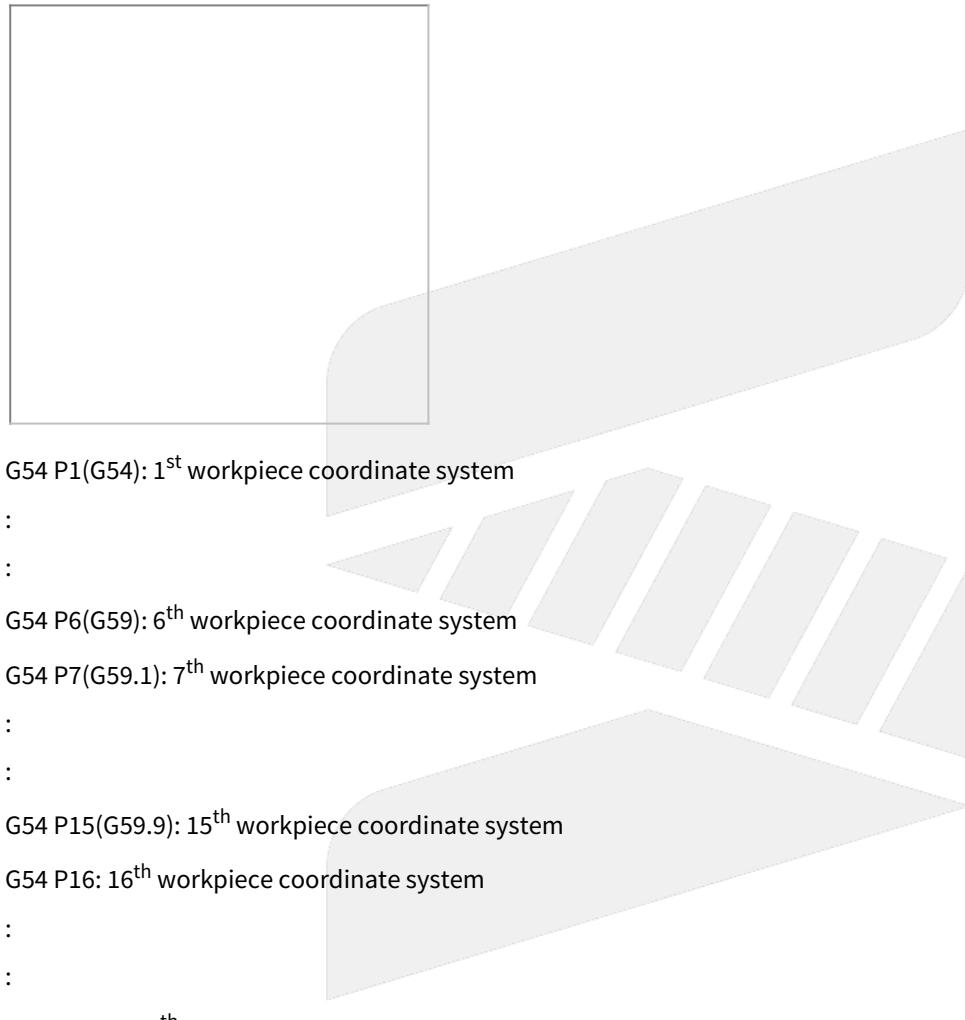


```
N98 G49 ;  
// cancel tool center point control  
N99 G69 ;  
// cancel tilted working plane coordinate system  
N100 G01 X0. Y0. Z50.;  
// interpolation to X0. Y0. Z50. of G54 coordinate system
```



2.42 G54...G59.9 : Workpiece Coordinate System Setup

2.42.1 Command Form



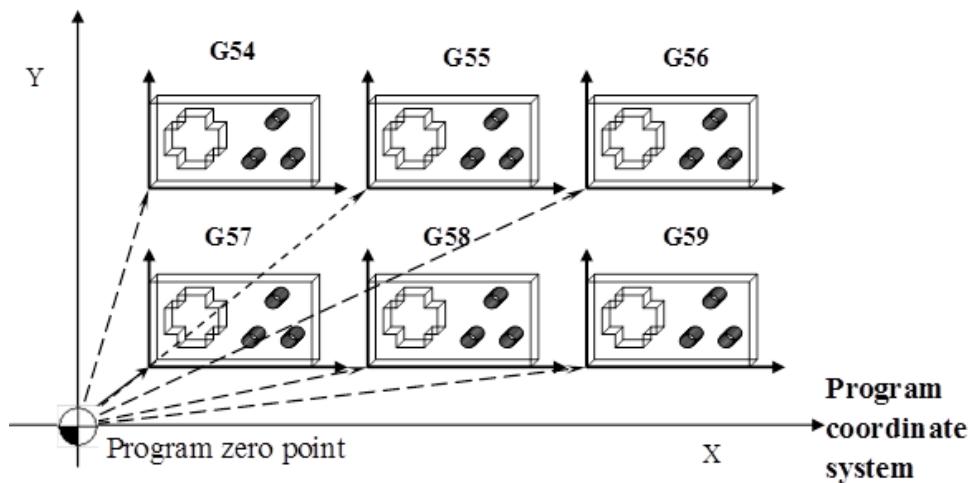
X, Y, Z: move to the specified position on the set workpiece coordinate system ;

2.42.2 Description:

If there are multiple workpieces on the machine platform when operating. By applying workpiece coordinate systems, it's able to define their position in the machine coordinate system through 100 different coordinate system: G54~G59, G59.1~G59.9, G54 P16~G54 P100. Therefore, the machining process can be executed one by one.

The workpiece coordinate system can be set by Pr3229, 0: activate; 1 deactivate.

2.42.3 Example



2.43 G61/G62/G63/G64 : Cutting Mode

2.43.1 Command Form

```
G61; // exact stop examination mode
G62; // curved surface cutting mode
G63; // tapping mode
G64; // curved surface cutting mode
```

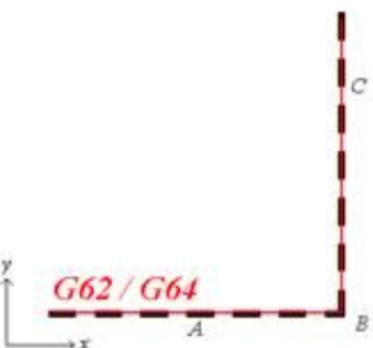
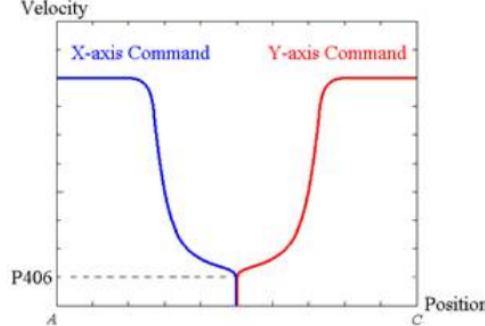
2.43.2 Description

The differences between each mode are listed below, the default mode is G64 cutting mode. After execute a certain mode, it'll be effective till another mode is assigned.

Command name	G code	Valid region	Description
Exact stop	G09	Only effective in block with G09.	When tool decelerates at the end of contour, The precision error occurs at the corner when the tool direction turns. G09 is used to control the precision error.
Exact stop mode	G61	Enable after execute G61, disable till G62, G63 or G64	The tool decelerates at the end of cutting contour, when it arrives at the end point, a feedback signal is sent to make sure the position is in the set range. The next contour will be executed after confirmation.

Command name	G code	Valid region	Description
Curved Surface Cutting Mode	G62	Enable after execute G62, disable til G61, G63 or G64	Applicable to curved surface cutting. The tool does not decelerate at the end of contour (refer to the speed command curve shown below) and continues to execute next contour. Able to carry the P argument and select a high speed high accuracy parameter. (Note 2)
Tapping mode	G63	Enable after execute G63, disable til G61, G62 or G64	Applicable to tapping. The synchronous between spindle and feeding axis is determined by the ratio of spindle rotation speed S and feedrate F. During tapping, feed overrode and feed hold cannot be adjusted.
Cutting mode	G64	Enable after execute G64, disable til G61, G62 or G63	Applicable to curved surface cutting. The tool does not decelerate at the end of path (refer to the speed command curve shown below) and continues to execute next path. Able to carry the P argument and select a high speed high accuracy parameter. (Note 2)

Figure : The action of G62/G64 while cutting over the corner.

G Code	Cutting Path	Speed Command Curve
G62/G64		

Explanation :

G62/G64 corner speed control mode will slow the speed down to the corner speed with Pr406 while interpolation over the corner, so there will be no command contour error at the corner. For the machining requires repeating cutting process such as mold machining, this mode provides better corner precision and reappearance. For the corner, the vibration caused by the JERK of speed command can be improved by Pr404, set Pr404 to 10~20 can make effective improvements.

2.43.3 **Note**

1. G62 / G64 mode are more suitable for mold machining.
2. G62 Pn/G64 Pn, n = 0 ~ 5, it's able to choose a high speed high accuracy parameter
3. For multiple high speed high accuracy parameter sets, the command later one will overwrite the former one. Only the last set of parameter will be reserved after reset, but it'll return to the default parameter (P0) after reboot.

2.44 G65 : Call Single Macro

2.44.1 **Command Form**

G65 P_L_;

P: number of the calling program;

L: repeating times;

2.44.2 **Description**

After calling macro, the program specified by P will be called and executed, L specifies the repeating times. But it is only effective in the block with G65.

Example

G65 P10 L20 X10.0 A10.0 Q10.0;

// Call and execute sub-program O0010 20 times continuously and assign the X, A, Q value into the sub-program.

// Therefore, it's able to do the calculations with #24, #1, #17 in the sub-program.

// The arguments are not limited to be X, Y, Z, it only needs to follow the macro writing rules.

2.45 G66/G67 : Call Modal Macro

2.45.1 **Command Form**

G66 P_L_;// macro calling

G67;// cancel macro calling

P: number of the calling program;

L: repeating times;

2.45.2 **Description**

After the macro command G66 is called, the sub-program specified by P will be called and executed, and L specifies the repeating times of G66. If there is moving instruction in the block, the G66 block will be executed again after the moving block is completed. The mode won't stop till being cancelled by G67 (system first calculates the number of blocks with moving instruction inside between G66 and G67, and does the required repeating times while executing the G66 block)

Program Example

```

N001 G91
N002 G66 P10 L2 X10.0 Y10.0; // repeat the calling of sub-program O0010 twice and assign value X0.0 Y10.0 to
execute
N003 X20.0; // move X axis to the position of 20.0 and call G66 P10 L2 X10.0 Y10.0
N004 Y20.0; // move Y axis to the position of 20.0 and call G66 P10 L2 X10.0 Y10.0
N005 G67 // cancel the macro calling mode

```

2.46 G68/G69 : Coordinate Rotation

2.46.1 Command Form

(G17) G68 X_ Y_ R_;
(G18) G68 Z_ X_ R_;
(G19) G68 Y_ Z_ R_;
X_, Y_, Z_: absolute coordinate of rotation center
R_: rotation angle
G69; // cancel coordinate rotation

2.46.2 Description

After the coordinate rotation starts, all movement commands will rotate with the rotation center, so the geometric figure will be rotated by an angle. The rotation center is only effective in absolute command, if all the commands are increment value, the actual rotation center will be the starting point of the contour.

2.46.3 Program Example

program 1:

```

G54 X0 Y0 F3000.;

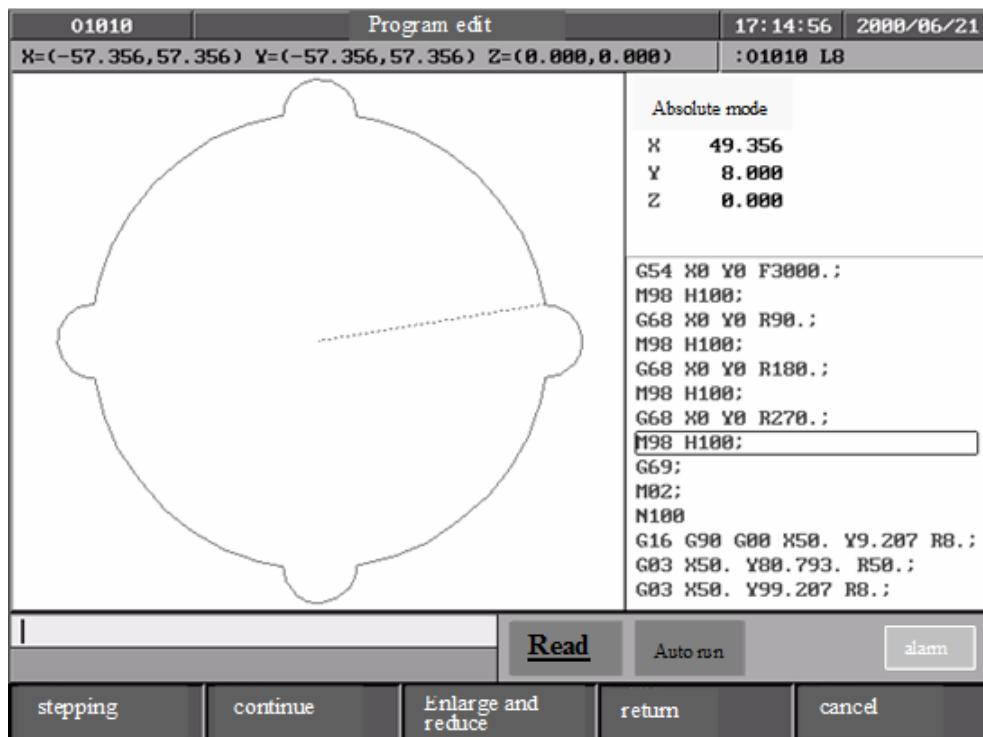
G16; // enable polar coordinates
      // orientate to the starting point
G90 G00 X50. Y9.207 R8.; // first machining process
M98 H100; // second machining process
G68 X0 Y0 R90.; // the coordinate rotates by 90 degrees
M98 H100; // third machining process
G68 X0 Y0 R180.; // the coordinate rotates by 180 degrees
M98 H100; // fourth machining process
G68 X0 Y0 R270.; // the coordinate rotates by 270 degrees
M98 H100; // disable coordinate rotation
G15; // disable polar coordinate
M02; // main program end

```

```

N100          // contour sub-program start
G90 G01 X50. Y9.207 R8.;
G03 X50. Y80.793. R50.;
G03 X50. Y99.207 R8.;
M99;          // contour sub-program return

```



program 2:

```

G54 X0 Y0 F3000.;

G16;          // enable polar coordinate system

G90 G00 X50. Y9.207 R8.;      // orientate to the starting point

M98 H100;        // first machining process
G68 X0 Y0 R45.;    // the coordinate rotates by 45 degrees
M98 H100;        // second machining process
G68 X0 Y0 R90.;    // the coordinate rotates by 90 degrees
M98 H100;        // third machining process
G68 X0 Y0 R135.;   // the coordinate rotates by 135 degrees
M98 H100;        // fourth machining process

```

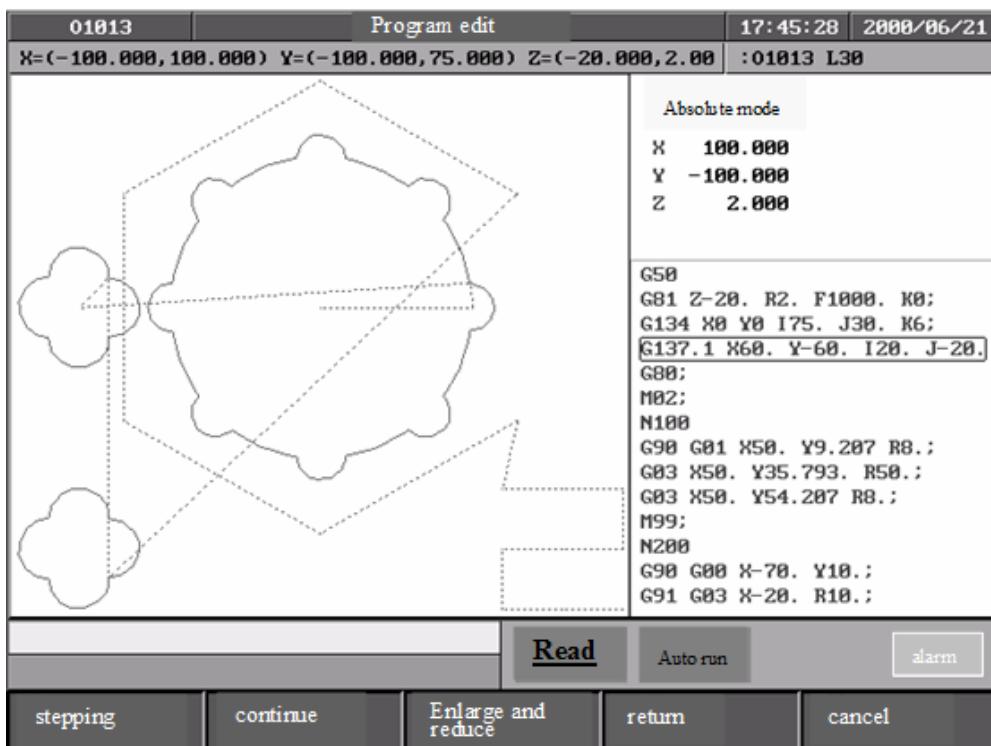
```

G68 X0 Y0 R180.; // the coordinate rotates by 180 degrees
M98 H100; // fifth machining process
G68 X0 Y0 R225.; // the coordinate rotates by 225 degrees
M98 H100; // sixth machining process
G68 X0 Y0 R270.; // the coordinate rotates by 270 degrees
M98 H100; // seventh machining process
G68 X0 Y0 R315.; // the coordinate rotates by 315 degrees
M98 H100; // eighth machining process
G69; // disable coordinate rotation
G15; // disable polar coordinate system
G00 X-80. Y0.
M98 H200; // process first “flower”
G51.1 Y-40.; // symmetry axis Y-40.
M98 H200; // process second “flower”
G50; // cancel mirror image
G90 G81 Z-20. R2. F1000. K0; // start G81 drilling cycle
G134 X0 Y0 I75. J30. K6; // circumference hole cycle
G137.1 X60. Y-60. I20. J-20. P3 K3; // chess type hole cycle
G80; // cancel drilling cycle
M02; // main program end
N100 // cdontour sub-program
G90 G01 X50. Y9.207;
G03 X50. Y35.793 R50.; // sub-program return
G03 X50. Y54.207 R8.;
M99; // sub-program start (flower)
N200 // sub-program start (flower)
G90 G00 X-70. Y10.;

G91 G03 X-20. R10.; // sub-program return(flower)
G03 Y-20. R10.;
G03 X20. R10.;
G03 Y20. R10.;

M99; // sub-program return(flower)

```



2.47 G68.2 : Tilted Working Plane Machining

Command Form

G68.2 X_ Y_ Z_ I_ J_ K_ ;

G69 ;

G68.2 : enable the tilted working plane coordinate function;

G69 : disable the tilted working plane coordinate function;

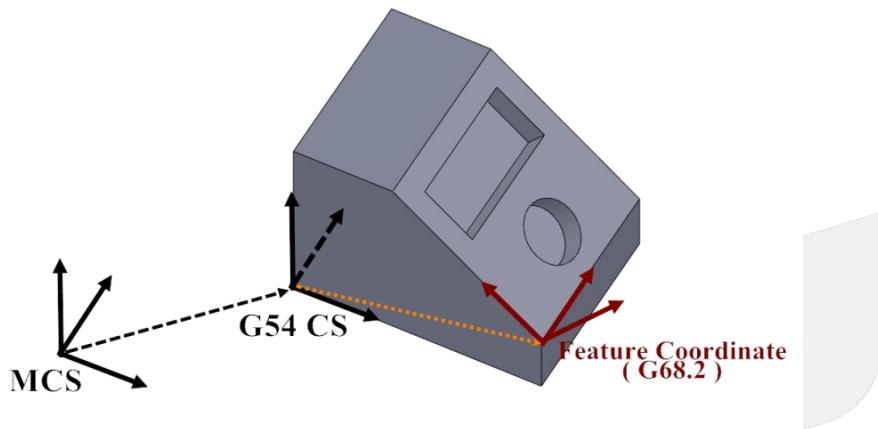
X_ Y_ Z_ : zero point of tilted working plane coordinate (relative to the zero point of G54 coordinate system);

I_ J_ K_ : Euler angle of tilted working plane coordinate;

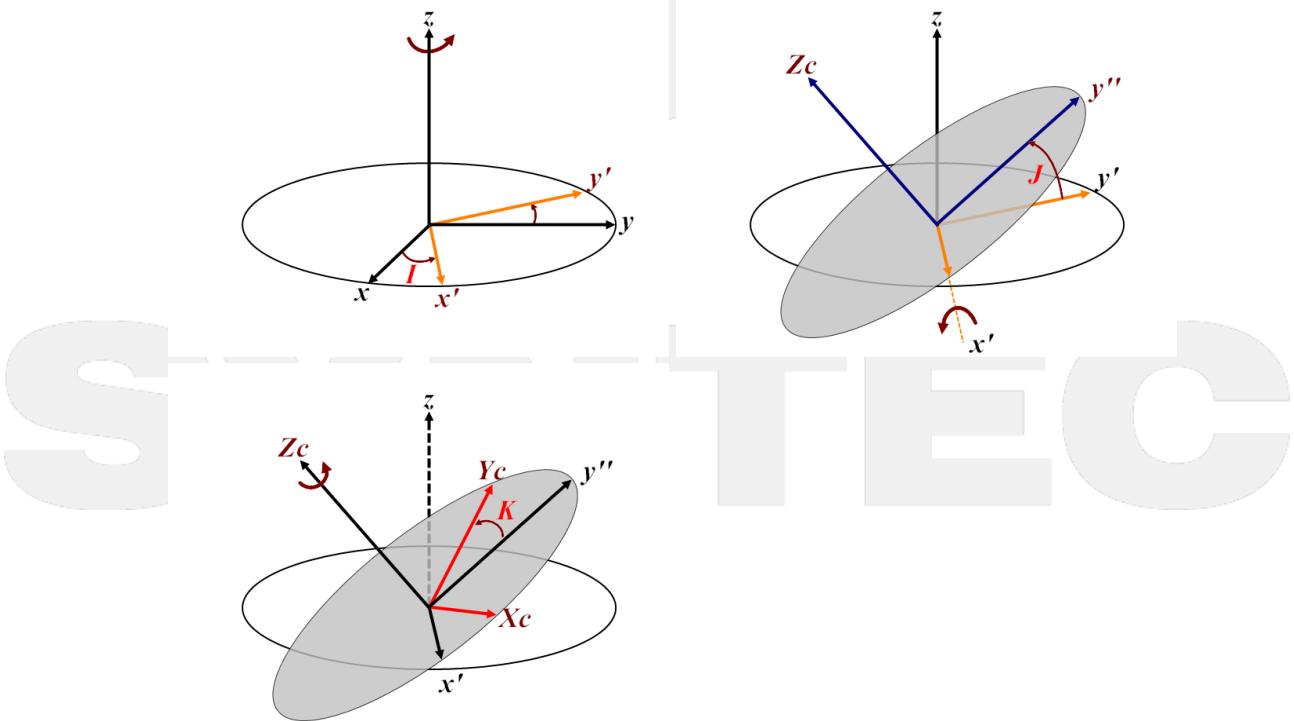
2.47.1 Description

1. Tilted Working Plane Machining, also called Feature Coordinate, is able to rotate the program coordinates to any specified angle and execute the machining process with the normal 3-axis machining program.

2. The zero point of tilted working plane coordinate is set related to the G54 coordinate system, shown as the orange dotted line in the picture below; and the tilting angle is set by the Euler angle.



3. For the zero point setup of tilted working plane coordinate, which are XYZ arguments after G68.2, only needs to specified the program zero point and the distance of each axis from the G54 coordinate system zero point.
4. The Euler angle is required for the setup of tilted working plane coordinate angle, the Euler angle has a clear definition. The IJK arguments after G68.2 represent for 3 rotating angle, respectively Z axis-X axis- Z axis in order.
- First, rotate the original XYZ coordinate system around Z axis by a certain angle, we get a new coordinate system X'Y'Z', the rotated angle is defined as angle I;
 - Then rotate the original X'Y'Z' coordinate system around X' axis by another certain angle, we get a new coordinate system X''Y''Z'', the rotated angle is defined as angle J,
 - Last, rotate the original X''Y''Z'' coordinate system around Z'' axis by another certain angle, we get a new coordinate system XcYcZc, the rotated angle is defined as angle K,
 - XcYcZc is the final tilted working plane coordinate system.



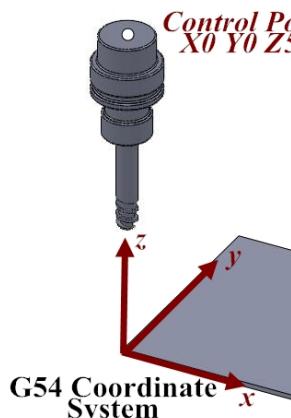
5. Related Parameters : Pr3014 Tilted working plane coordinate system retaining state
- 0 : Not retaining the tilted working plane coordinate set by G68.2 after reset or power off/on.
 - 1 : Retain the tilted working plane coordinate set by G68.2 after reset but clear the coordinate system after reboot.
 - 2 : Always retain the tilted working plane coordinate set by G68.2 after reset or reboot.

2.47.2 **Note**

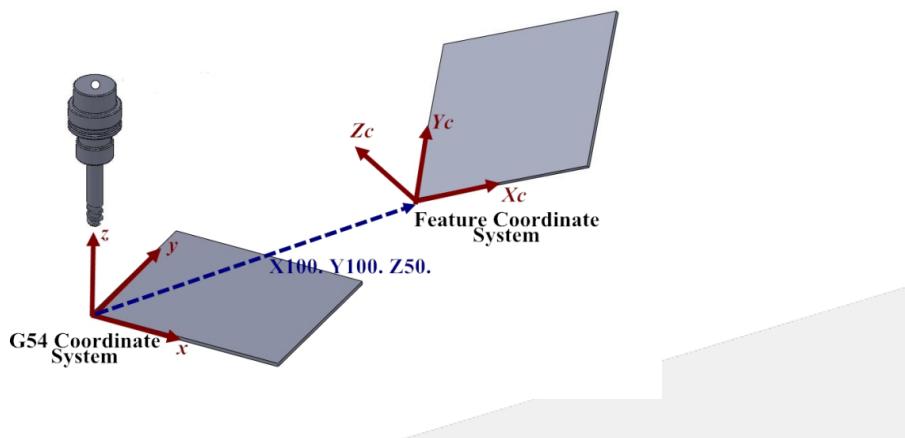
1. Do not specified command G53.1 before G68.2.
2. Please apply positive tool length (G43 after G53.1)

2.47.3 **Program Example**

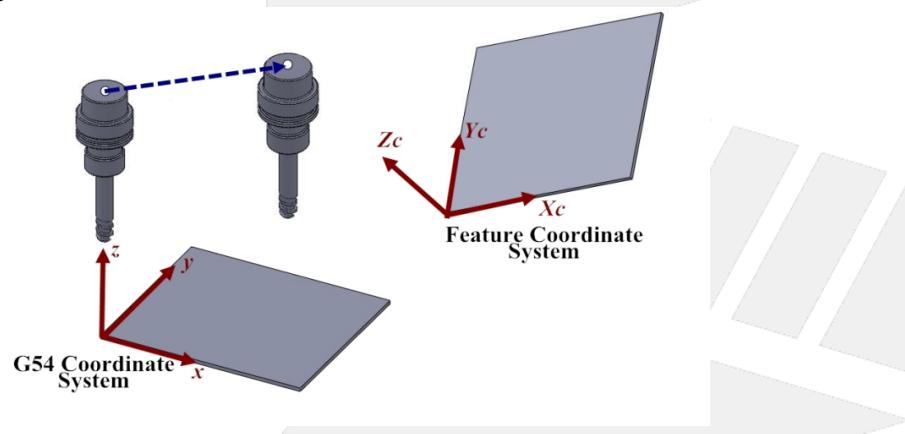
```
N1 G90 G54 G01 X0 Y0 Z50. F1000;
N2 G68.2 X100. Y100. Z50. I30. J15. K20. ;
N3 G01 X0 Y0 Z50. F1000;
N4 G53.1;
N5 G43 H1;
N6 G01 X0 Y0 Z0;
N1 G90 G54 G01 X0 Y0 Z50. F1000;
// interpolation to Z50 on G54 coordinate system at F1000 speed
```



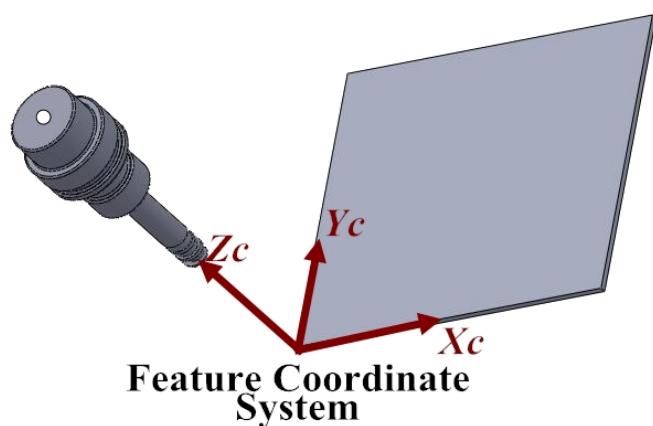
```
N2 G68.2 X100. Y100. Z50. I30. J15. K20. ;
// specify X100. Y100. Z50. on G54 coordinate system as the zero point of tilted working plane coordinate, and set
the Euler angle I30. J15. K20.. The program coordinate system will change to the tilted working plane coordinate
after G68.2 is executed.
```



N3 G01 X0 Y0 Z50. F1000;
 // interpolation to Z50. of tilted working plane coordinate at F1000 feedrate, but the tool direction remains the same



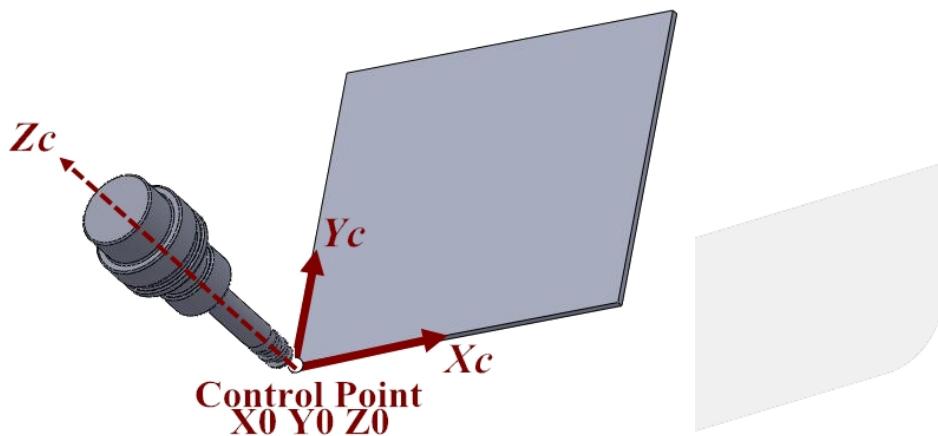
N4 G53.1;
 // the tool direction automatically aligns to the Z axis of tilted working plane coordinate system



TEC

N5 G43 H1;
 // tool length compensation, the controlling point changes to the position of tool center point

N6 G01 X0 Y0 Z0;
 // interpolation to the X0 Y0 Z0 position of tilted working plane coordinate.



2.48 G70/G71 : Imperial/SI Unit Setup

2.48.1 Command Form

G70; Imperial Unit Setup

G71; SI Unit Setup

2.48.2 Description

The origin offset value of workpiece coordinate, tool data, system parameter, and reference point will still be correct after switching the system unit. The system will deal with the unit transformation automatically.

After changing the unit system, the operation units below will be changed:

- the coordinate, unit of speed
- increment JOG unit
- MPG JOG unit

2.48.3 Note

The rotary axis has no imperial unit, therefore when executing the synchronous moving command of linear axis and rotary axis, the command value of linear axis is divided by 25.4 but the command value of rotary axis will remain the same. Thus, for the synthesis speed, the percentage of rotary axis will rise massively and make the linear axis speed decreases rapidly, please be extra careful with the situation.

2.49 G73 : High Speed Peck Drill Cycle

2.49.1 Command Form

G73 X_ Y_ Z_ R_ Q_ F_ K_;

X(U) or Y(V): hole position (absolute/increment, please set parameter 3809 is 1 when using increment value)

Z:

G91: the distance from the initial point to Z point (directional)

G90: program coordinate of Z point

R:

G91: the distance from the initial point to R point (directional)

G90: program coordinate of R point

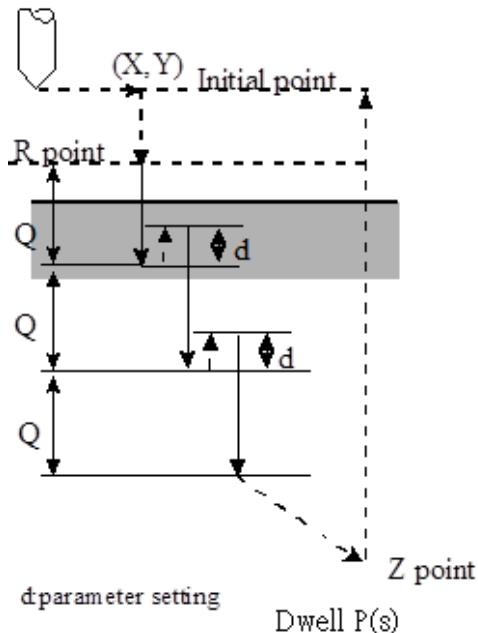
Q: depth of each feed (increment and positive, the minus will be ignored)

F: feedrate

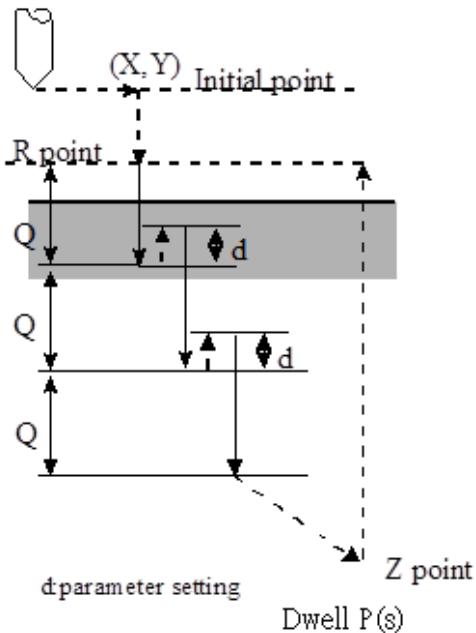
K: repeating times (repeating the moving and drilling, G91 increment input is effective)

Absolute/Increment coordinate of X, Y, Z, R can be specified by G90/G91.

G98



G99



2.49.2 Description

The process of the action is shown below:

1. Start the machining process, move the tool to specified (X,Y) with G00.
2. Move the tool to the specified R point with G00.
3. Downwards a distance Q relative to the current drilling depth with G01.
4. Lift up a retract depth d with G01 (Pr4002).
5. Repeat drilling above till reaches the bottom Z point.
6. Lift up to initial point(G98) or R point with G00 (G99).

2.49.3 Note

1. Please start spindle rotation with M code before applying G73.
2. If M Code and G73 are specified in the same block ,the M Code will only be executed once in the block.

3. When specified to repeat K times, M Code will only be executed at first cycle, it won't be executed for others.
4. G73 is a module G Code, effective after being executed. If an (X,Y) coordinate is the only argument given in the next line ,the controller will start the tapping at (X,Y).
5. G73 can be cancelled by G80, or being cancelled automatically when the program executes G00, G01, G02, G03 or other G code cycles.
6. Before changing the drilling axis, G73 cycle cycle must be cancelled first.
7. If no axis (X, Y, Z) moving command is included in the block, the drilling action won't be executed.
8. The argument specified by Q and R will only be set when executing the block with drilling action, it won't be set in the blocks with no drilling actions.
9. G codes G00, G01, G02, G03 can't be specified in the same block with G73, or the G73 cycle command will be cancelled.
10. In G73 cycle, the tool radius compensation (G41/G42/G40) will be ignored.
11. Arguments X,Y,Z,A,B,C,U,V,W are recognized as axis command when the parameter 3809 is 0.
12. The increment command for drilling axis is ignored when the increment and absolute command for drilling axis are commanded simultaneously.

2.49.4 Program Example

```

N001 F1000. S500;
N002 M03; // start the drill, CW rotation
N003 G90;
N004 G00 X0. Y0. Z10.; // move to the initial point
N005 G17;
N006 G90 G99; //specify the coordinate of R point, Z point and hole NO.1, cutting depth 2.0
N007 G73 X5. Y5. Z-10. R-5. Q2.;
N008 X15.; // hole NO.2
N009 Y15.; // hole NO.3
N010 G98 X5.; // hole NO.4, return to the initial point
N011 X10. Y10. Z-20.; // hole NO.5, set new Z point as -20
N012 G80;
N013 M05; // stop the tool
N014 M30;

```

2.50 G74 : Left Hand Tapping Cycle

2.50.1 Command Form

G74 X_ Y_ Z_ R_ P_ Q_ (F_ or E_) K_ I_ J_ ;

X(U) or Y(V): hole position (absolute/increment, please set parameter 3809 is 1 when using increment value)

Z:

G91: the distance from the initial point to Z point (directional)

G90: program coordinate of Z point

R:

G91: the distance from the initial point to R point (directional)

G90: program coordinate of R point

P: dwelling time; (sec)

Q: depth of each feed (increment and positive, the minus will be ignored)

F: feedrate

E: number of threads per inch (once F and E argument are both specified, then E argument would be ignored), valid version : after 10.116.16B、10.116.18、10.117.19.

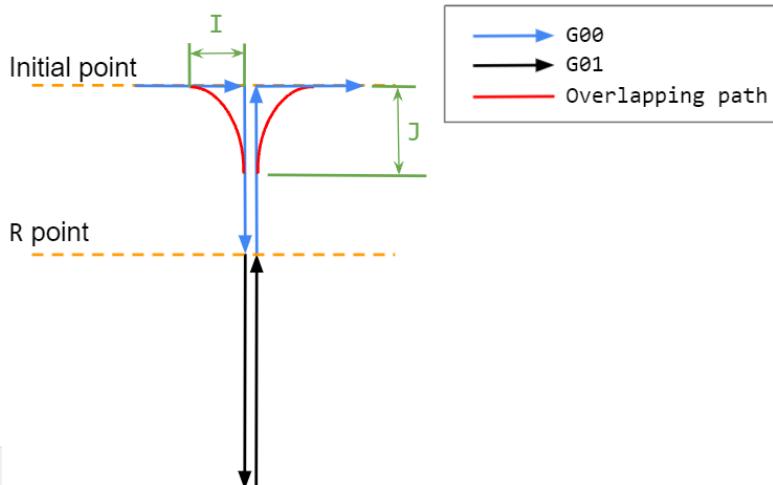
K: repeating times (repeating the moving and tapping, G91 increment input is effective)

Absolute/Increment coordinate of X, Y, Z, R can be specified by G90/G91.

I : Overlapping distance of the positioning axis (After the previous tapping finishes, it is the overlapping distance from G00 positioning to the current position.). It is valid when Pr4008(setting for drilling/tapping mode) value is 1. Unit: LIU.

J : Overlapping distance of the tapping axis (When the tapping finishes, it is the overlapping distance from exit the hole along the tapping axis to the next G00.). It is valid when Pr4008(setting for drilling/tapping mode) value is 1. Unit: LIU.

1. ~10.118.47: I, J parameters are not supported.
2. 10.118.48A~10.118.48D, 10.118.48~10.118.50: I is the overlapping distance of the tapping axis; J is the overlapping distance of the positioning axis.
3. 10.118.48E, 10.118.51 or above: I is the overlapping distance of the positioning axis; J is the overlapping distance of the tapping axis.



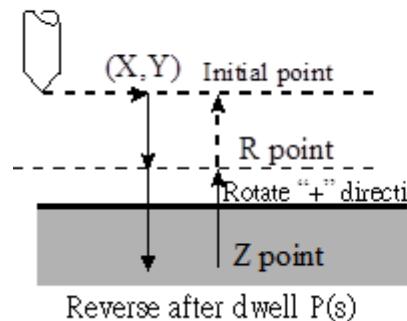
2.50.2 Description

1. The process of G74
 - a. When G74 starts, the tool moves to the point of command (X, Y) by G00
 - b. move down to point R by G00
 - c. start tapping
 - d. move up to the initial point (G98) or R point (G99)
2. The action of overlapping
 - a. When the program has two consecutive blocks of G74 or the consecutive blocks of G74 and G00, the second block will start when the first block remains a certain distance from the finish. The distance is called "overlapping distance", see I & J in the figure above.

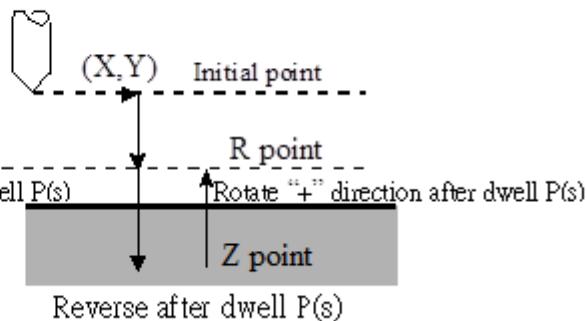
- b. Overlapping is suitable for the continuous tapping cycle. It does not need I/J argument in every line of the program but 'overlaps' by a constant distance. In Program example 2, the overlapping distance of the positioning axis and the tapping axis of each tapping command is 2 and 3 respectively. And the Overlapping distance will not reset to zero until the disable command G80 is executed.
- c. In order to enable overlapping, set Pr4008 to 1, otherwise the I and J parameters have no effect; If Pr4008 is set to 1 but the I and J arguments are not given, the overlapping still won't be executed.
- d. Overlapping will only be valid above R point. Remark: if G74 only moves up to R point(G99), overlapping won't be executed.

TYPE I : Without Q argument

G98



G99

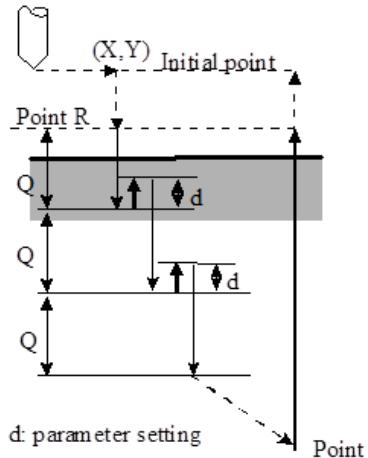


1. Start the machining process, move the tool to specified (X,Y) with G00.
2. Move the tool to the specified R point with G00.
3. Execute spindle orientation (action can be skipped if Pr4007=0).
4. Tapping till the bottom Z point with G01.
5. Dwell for P seconds and reverse the tool.
6. Lift up to R point with G01.
7. Dwell for P seconds and reverse the tool.
8. Return to the initial point (G98) or the R point (G99) with G00.

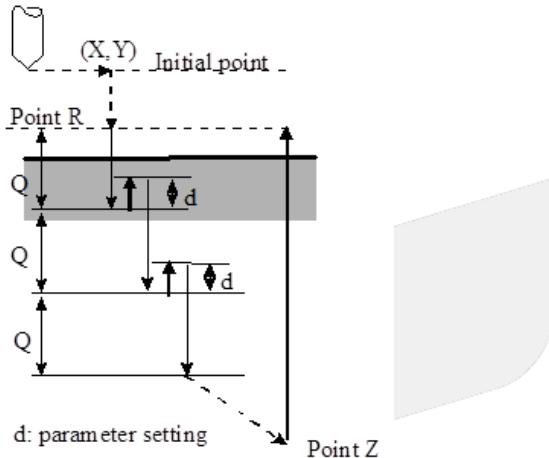
SYNTEC

TYPE II : Rapid Peck Tapping (Pr4001= 1)

G98



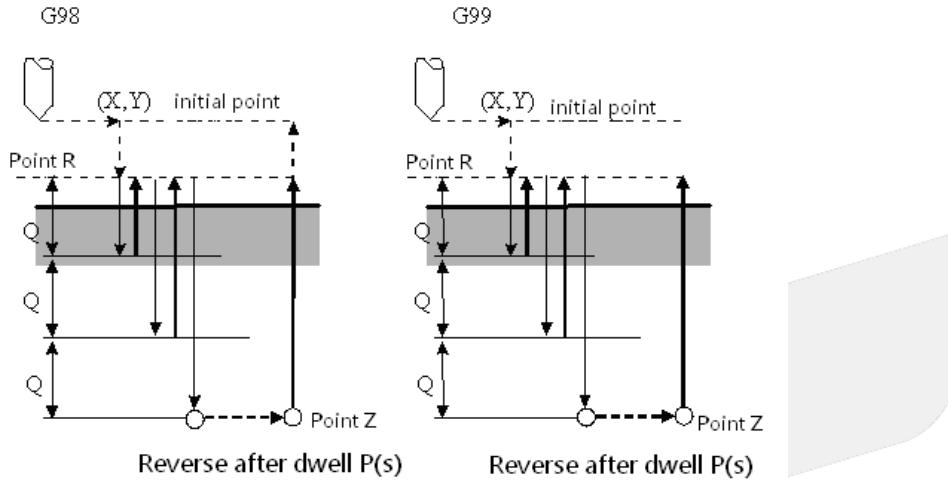
G99



1. Start the machining process, move the tool to specified (X,Y) with G00.
2. Move the tool to the specified R point with G00.
3. Execute spindle orientation (action skippable if Pr4007=0).
4. Tapping downwards for a distance of Q from the current tapping depth with G01.
5. Dwell for P seconds and reverse the tool, lift up a retract depth d with G01 (Pr4004).
6. Dwell for P seconds and reverse the tool, tapping downwards again a distance Q relative to the current tapping depth with G01.
7. Dwell for P seconds and reverse the tool, lift up a retract depth d with G01 (Pr4004).
8. Repeat tapping above till reaches the bottom Z point.
9. Dwell for P seconds and reverse the tool.
10. Lift up to R point with G01 (G99).
11. Dwell for P seconds and reverse the tool.
12. Lift up to the initial point with G00 (G98).

SYNTEC

TYPE III : Normal Peck Tapping (Pr4001=0)



1. Start the machining process, move the tool to specified (X,Y) with G00.
2. Move the tool to the specified R point with G00.
3. Execute spindle orientation (action skippable if Pr4007=0).
4. Tapping downwards for a distance of Q from the current tapping depth with G01.
5. Dwell for P seconds and reverse the tool, lift up R point with G01.
6. Dwell for P seconds and reverse the tool, tapping downwards again a distance Q relative to the current tapping depth with G01.
7. Dwell for P seconds and reverse the tool, lift up R point with G01.
8. Repeat tapping above till reaches the bottom Z point.
9. Dwell for P seconds and reverse the tool.
10. Lift up to R point with G01 (G99).
11. Dwell for P seconds and reverse the tool.
12. Lift up to the initial point with G00 (G98).

Tapping Pitch/Machining Speed Calculation

G94 : Machining Speed (F mm/min) = Spindle Speed (S r.p.m) * Pitch (P mm/rev)

G95 : Machining Speed (F mm/rev) = Pitch (P mm/rev)

G74 : During the process, machining speed F and spindle speed S is independent of the override switch (Fixed at 100%)

2.50.3 Note

1. Please start spindle rotation with M code before applying G74.
2. If M Code and G74 are specified in the same block, the M Code will only be executed once in the block.
3. When specified to repeat K times, M Code will only be executed at first cycle, it won't be executed for others.
4. G74 is a module G Code, effective after being executed. If an (X,Y) coordinate is the only argument given in the next line, the controller will start the tapping at (X,Y).
5. G74 can be cancelled by G80, or being cancelled automatically when the program executes G00, G01, G02, G03 or other G code cycles.
6. If pressed the pause or reset button during the tapping process, the current hole tapping action will be completed then stops at R point.
7. The angle of the spindle orientation before tapping can be specified by the spindle origin offset value (Pr1771~Pr1780).

8. G codes G00, G01, G02, G03 can't be specified in the same block with G74, or the G74 cycle command will be cancelled.
9. In G74 cycle, the tool radius compensation (G41/G42/G40) will be ignored.
10. The spindle orientation function before tapping is valid from version 10.116.14 and is only provided for serial spindles.
11. Before changing the machining spindle (R791~), please cancel the cycle with G80 to avoid unexpected machining action if the current spindle is in tapping state.
12. Base the model support Rapid tapping, while processing G74 without Q and P argument and spindle is serial or Pr1791=3, then rapid tapping is launched,
13. Arguments X,Y,Z,A,B,C,U,V,W are recognized as axis command when the parameter 3809 is 0.
14. The increment command for tapping axis is ignored when the increment and absolute command for tapping axis are commanded simultaneously.

2.50.4 Program Example

Example 1:

```

N001 F1000. S500;
N002 G90;
N003 G00 X0. Y0. Z10.; // move to the initial point
N004 G17;
N005 M04; // start reverse the tool.
N006 G90 G99; // specify the coordinate of R point, Z point and hole No.1, dwelling time 2 seconds
N007 G74 X5. Y5. Z-10. R-5. P2.;
N008 X15.; // hole No.2
N009 Y15.; // hole No.3
N010 G98 X5.; // hole No.4, return to the initial point
N011 X10. Y10. Z-20.; // hole No.5, set new Z point as -20.
N012 G80;
N013 M05; // stop the tool
N014 M30;

```

Example 2:

```

N001 F1000. S500;
N002 G90;
N003 G00 X0. Y0. Z10.; // move to initial point
N004 G17;
N005 M04; // start the spindle turning counterclockwise
N006 G90 G98; // specify the coordinate of R point, Z point and hole No.1.
N007 G74 X5. Y5. Z-10. R-5 I2. J3.; // set the overlapping distance of the positioning axis(I) and the tapping axis(J) to 2 and 3 respectively, the overlap section only exists above R point.
N008 X15.; // hole No.2
N009 Y15.; // hole No.3
N010 G99 X5.; // hole No.4, return to R point
N011 Y5.; // hole No.5, overlapping won't be executed
N013 G80;

```

N014 M05; // stop the tool
 N015 M30;

2.51 G76 : Fine Boring Cycle

2.51.1 Command Form

G76 X_Y_Z_R_Q_P_F_K_ ;

X(U) or Y(V): hole position (absolute/increment, please set parameter 3809 is 1 when using increment value)

Z:

G91: the distance from the initial point to Z point (directional)

G90: program coordinate of Z point

R:

G91: the distance from the initial point to R point (directional)

G90: program coordinate of R point

P: dwelling time; (sec)

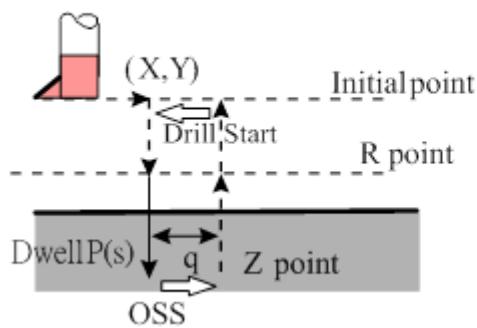
Q: depth of each feed (increment and positive, the minus will be ignored)

F: feedrate

K: repeating times (repeating the moving and drilling, G91 increment input is effective)

Absolute/Increment coordinate of X, Y, Z, R can be specified by G90/G91.

G98



G99

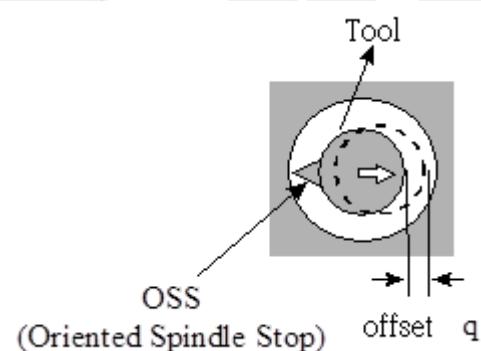
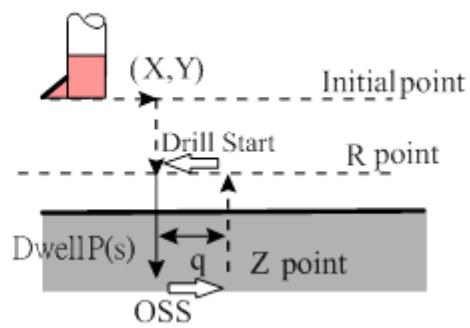


figure : Oriented Spindle Stop(OSS)

2.51.2 **Description**

1. Start the machining process, move the tool to specified (X,Y) with G00.
2. Move the tool to the specified R point with G00.(without spindle orientation)
3. Move the tool to the Z point at the bottom of the hole with G01 and dwell P seconds, then stop drilling with spindle orientation.
4. Shift a distance of Q.
5. Return to the initial point (G98) or the R point (G99) with G00.
6. Shift a distance of Q in reverse direction
7. Drill rotation starts.

2.51.3 **※ Alarm**

Q is a Modal Value required in G76 cycle, it must be specified carefully since its also applied in G76/G83/G87.

- The OSS(Oriented Spindle Stop) direction is decided by Pr4020

Parameter 4020	OSS Direction
0	+X
1	-X
2	+Y
3	-Y
4	+Z
5	-Z

2.51.4 **Note**

1. Please start spindle rotation with M code before applying G76.
2. If M Code and G76 are specified in the same block ,the M Code will only be executed once in the block.
3. When specified to repeat K times, M Code will only be executed at first cycle, it won't be executed for others.
4. G76 is a module G Code, effective after being executed. If an (X,Y) coordinate is the only argument given in the next line ,the controller will start the tapping at (X,Y).
5. G76 can be cancelled by G80, or being cancelled automatically when the program executes G00, G01, G02, G03 or other G code cycles.
6. Before changing the drilling axis, G76 cycle command must be cancelled first.
7. If no axis (X, Y, Z) moving command is included in the block, the drilling action won't be executed.
8. The Q value should be positive, it'll still be defined as a positive value even it's set negative (absolute value).
9. The argument specified by Q and R will only be set when executing the block with drilling action, it won't be set in the blocks with no drilling actions.

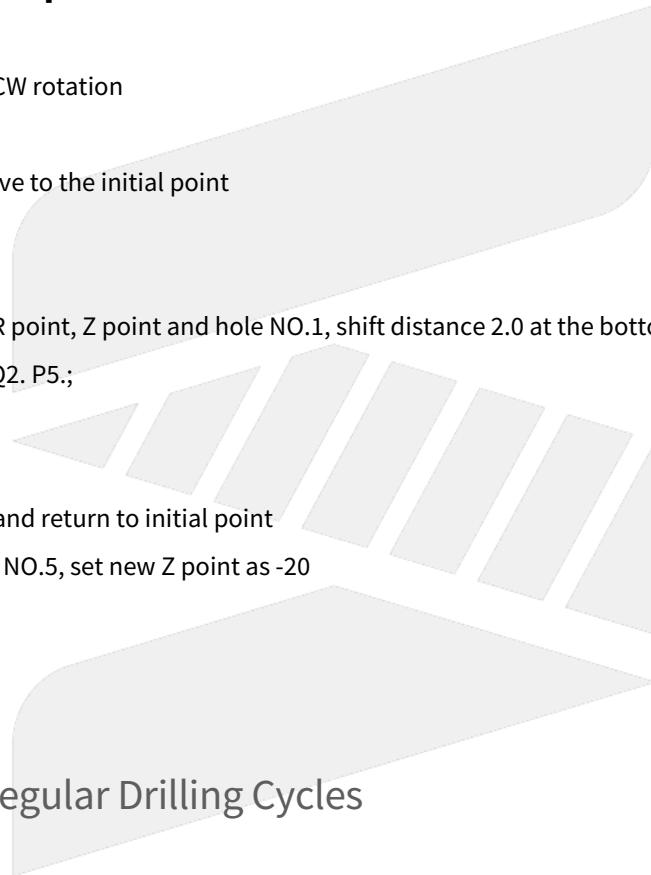
10. G codes G00, G01, G02, G03 can't be specified in the same block with G76, or the G76 cycle command will be cancelled.
11. In G76 cycle, the tool radius compensation (G41/G42/G40) will be ignored.
12. Arguments X,Y,Z,A,B,C,U,V,W are recognized as axis command when the parameter 3809 is 0.
13. The increment command for boring axis is ignored when the increment and absolute command for boring axis are commanded simultaneously.

2.51.5 Program Example

```

N001 F1000. S500;
N002 M03; // start the drill, CW rotation
N003 G90;
N004 G00 X0. Y0. Z10.; // move to the initial point
N005 G17;
N006 G90 G99;
//specify the coordinate of R point, Z point and hole NO.1, shift distance 2.0 at the bottom, dwelling time 5 seconds
N007 G76 X5. Y5. Z-10. R-5. Q2. P5.;
N008 X15.; // hole NO.2
N009 Y15.; // hole NO.3
N010 G98 X5.; // hole NO.4, and return to initial point
N011 X10. Y10. Z-20.; // hole NO.5, set new Z point as -20
N012 G80;
N013 M05; // stop the tool
N014 M30;

```



2.52 G80-G89 : Regular Drilling Cycles

Description

Regular Drilling Cycles simplifies the drilling process from multiple blocks into a specified G code in one block.
Regular Drilling Cycles:

Code	Name	Drilling Direction	Action at the bottom of the hole	Break Away Action	Usage
G80					Cancel the Cycle
G81	Normal Drilling (Pr40 08 = 0)	Z	Drill CCW Rotation	Rapid Feeding	Drilling Cycle

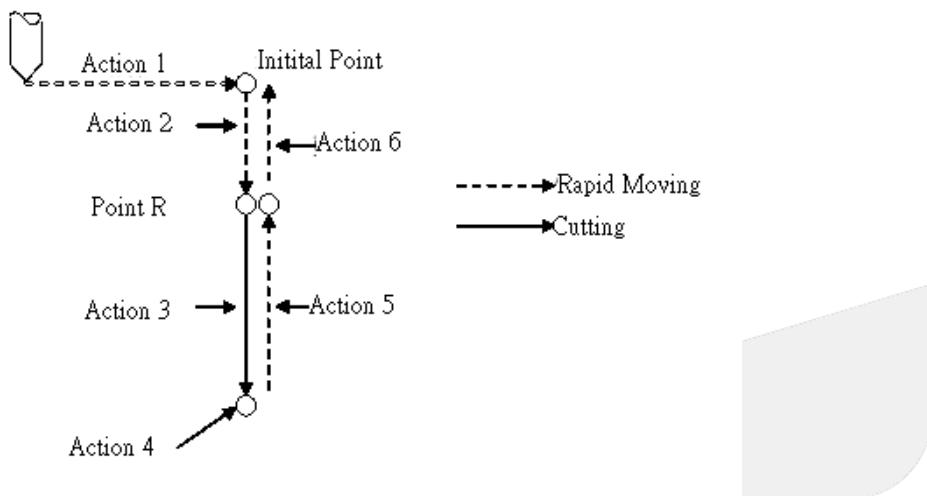
	Rapid Drilling (Pr400 8 = 1)	Z	Drill CCW Rotation	Rapid Feeding (unspecified)/ Cutting Feed (F2)	Drilling Cycle
G82	Drilling Cycle with Dwelling at Hole Bottom	Z	Pause	Rapid Feeding	Drilling Cycle
G83	Peck Drilling	Z	Drill CCW Rotation	Rapid Feeding	Drilling Cycle
G84	Tapping Cycle	Z	Pause	Cutting Feed -> Rapid Feeding	Tapping Cycle
G85	Drilling Cycle	Z	Drill CCW Rotation	Cutting Feed -> Rapid Feeding	Drilling Cycle
G86	Rapid Drilling Cycle	Z	Drill CCW Rotation	Rapid Feeding	Drilling Cycle
G87	Fine Boring Cycle	Z	Drill CCW Rotation	Cutting Feed -> Rapid Feeding	Boring Cycle
G88	Semiautomatic Fine Boring Cycle	Z	Drill CCW Rotation	Cutting Feed -> Rapid Feeding	Boring Cycle
G89	Boring Cycle with Dwelling at Hole Bottom	Z	Pause	Cutting Feed -> Rapid Feeding	Boring Cycle

Note 1 : Execute drill CCW rotation with M04.

Note 2 : With/Without the Q argument will decide whether the G83/G87 action is **continuous feed** or **intermittent feed**.

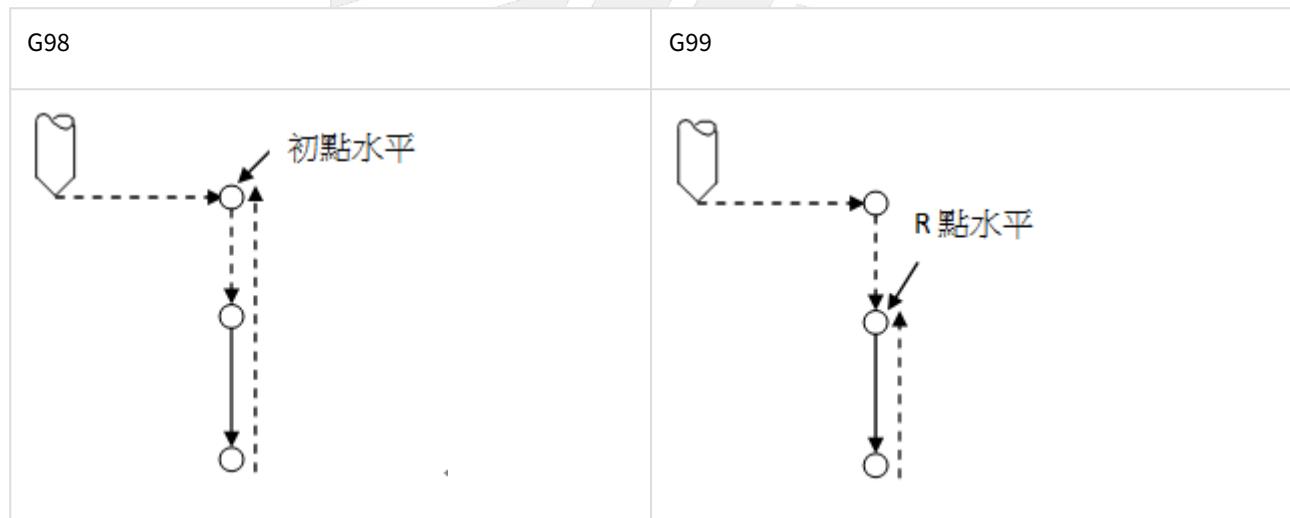
In general, the drilling cycle is composed of the 6 continuous actions below :

1. Rapid orientate to the specified point on X, Y axes.
2. Rapid moving to point R.
3. Drilling process.
4. Actions at the bottom of the hole.
5. Leave and move to point R.
6. Rapid moving to the initial point.



For the return action, specified the tool to return to R point or initial point with G98/G99. (Please refer to the picture below)

The initial point won't change even the drilling action is executed in the way of G99. If the last return position is the initial point, the starting point will be the initial point; if it's R point, then will be the R point.



2.53 G80.2 Disable high precision advanced chopping

2.53.1 **Command Form**

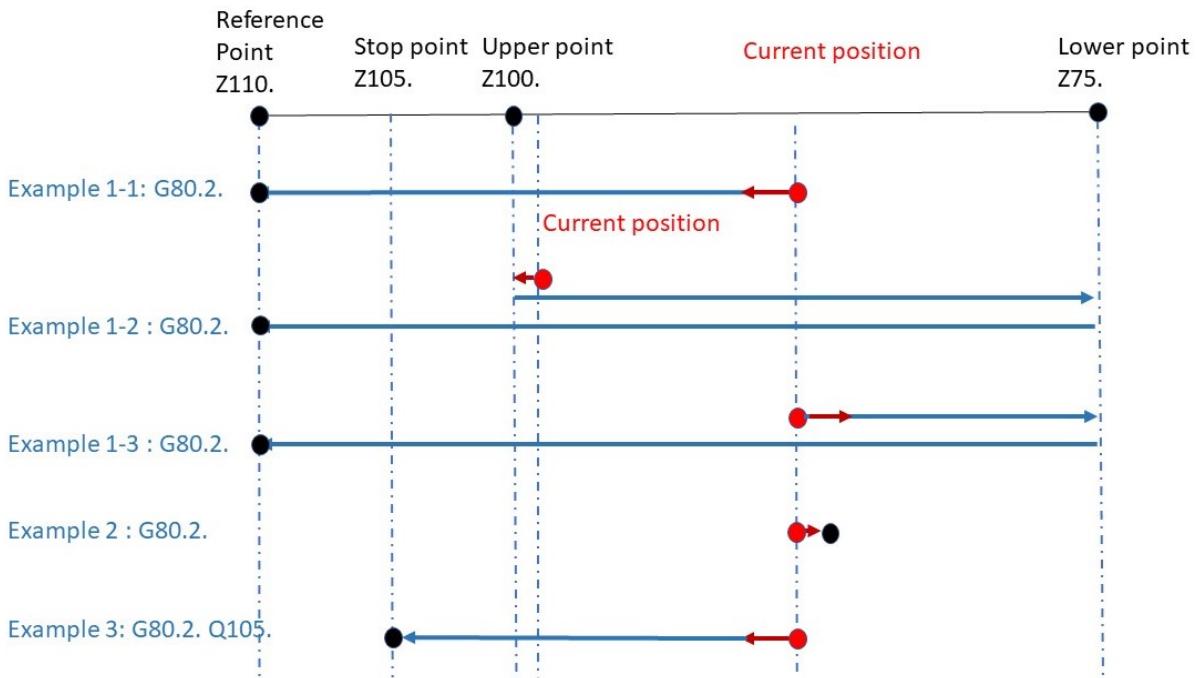
G80.2 [Q_] : disable high precision advanced chopping

Q : assign stop position (absolute command, machine coordinates, unit : IU), if the Q is undetermined, the stop position will follow Pr3957 setting.

2.53.2 **Description**

1. Pr3957 set Reference point as stop position:
 - a. If the stop position and chopping moving axis with the same direction :

- i. if the acceleration/deceleration distance is sufficient then chopping axis will move to stop position directly.(Example 1-1)
 - ii. If the acceleration/deceleration distance is insufficient chopping axis will move to boundary point and then move to stop position.(Example 1-2)
 - b. If the direction of stop position is different from chopping moving direction. chopping axis will move to boundary point and then move to stop position.(Example 1-3)
2. the stop feedrate is also using the advanced chopping and the range is from 0% to 150% or section 1~15. (Please refer to Pr3207 C/S interface version number for further descriptions of feeding ratio)
 3. The movement parameter of acceleration/deceleration section plannings is specified by the parameter of moving axis Pr541~, Pr621~ and Pr641~.



2.53.3 Program Example

Example 1-1~Example 1-3 :

```
// Pr17 = 2, 1BLU = 0.001mm
// Pr3957 = 0
@120000 :=100000;           // set R20000 as 100mm
@120001 :=75000;            // set R20001 as 75mm
G90;
G00 X0. Y0.;

G81.2 P1 Q20000 R110. F3000; // the machine platform arrives at point R at G00 speed then moves back and forth between the upper and lower vertex

G01 Y50.;                  // the chopping axis will stop at Reference point.

G80.2;
```

Example 2 :

```
// Pr17 = 2 為例, 1BLU = 0.001mm
// Pr3957 = 1
@120000 :=100000;           // set R20000 as 100mm
@120001 :=75000;            // set R20001 as 75mm
G90;
G00 X0. Y0.;

G81.2 P1 Q20000 R110. F3000; // the machine platform arrives at point R at G00 speed then moves back and forth between the upper and lower vertex
G01 Y50.;

G80.2;                      // the chopping axis will stop immediately.

M30;
```

Example 3 :

```
// Pr17 = 2 為例, 1BLU = 0.001mm
// Pr3957 = 1
@120000 :=100000;           // set R20000 as 100mm
@120001 :=75000;            // set R20001 as 75mm
G90;
G00 X0. Y0.;

G81.2 P1 Q20000 R110. F3000; // the machine platform arrives at point R at G00 speed then moves back and forth between the upper and lower vertex
G01 Y50.;

G80.2 Q105.;                // the chopping axis will stop at X105.

M30;
```

2.54 G81 : Drilling Cycle

2.54.1 Command Form

G81 X_Y_Z_R_F_F2=_K_Q_D_

X(U) or Y(V): hole position (absolute/increment, please set parameter 3809 is 1 when using increment value)

Z:

G91: the distance from the R point to Z point (directional)

G90: program coordinate of Z point

R:

G91: the distance from the initial point to R point (directional)

G90: program coordinate of R point

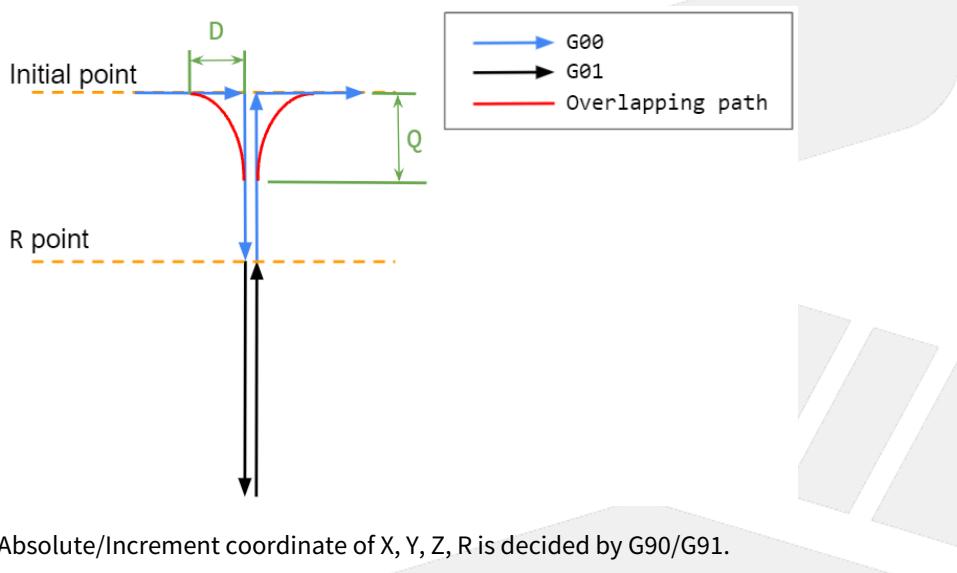
F: feedrate

F2=: Lifting rate (when Pr4008 is set 0 or the drilling mode is switch to normal drilling mode, F2 command is invalid)

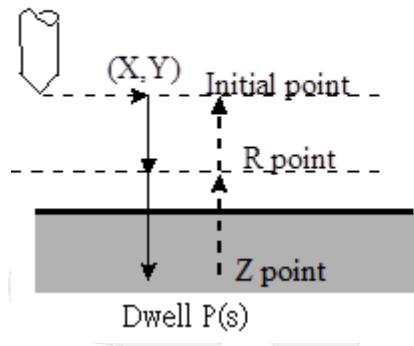
K: repeating times (repeating the moving and drilling actions, G91 increment input is effective)

Q: Overlapping distance of the drilling axis (When the drilling finishes, it is the overlapping distance from exit the hole along the drilling axis to the next G00.). It is valid when Pr4008(setting for drilling/tapping mode) value is 1. Unit: LIU.

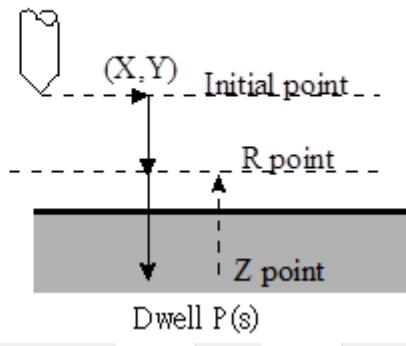
D: Overlapping distance of the positioning axis (After the previous drilling finishes, it is the overlapping distance from G00 positioning to the current position.). It is valid when Pr4008(setting for drilling/tapping mode) value is 1. Unit: LIU.



G98



G99



2.54.2 Description

1. The process of G81
 - a. Start the machining process, move the tool to specified (X,Y) with G00.
 - b. Move the tool to the specified R point with G00.
 - c. Move the tool to the Z point at the bottom of the hole with G01.
 - d. Lift up to the initial point (G98) or the R point (G99) with F2 speed . If F2 is not specified or smaller than 0, the lifting speed default is G00 speed.
2. .The action of overlapping

- a. When the program has two consecutive blocks of G81 or the consecutive blocks of G81 and G00, the second block will start when the first block remains a certain distance from the finish. The distance is called "overlapping distance", see Q & D in the figure above.
- b. Overlapping is suitable for the continuous drilling cycle. It does not need Q/D argument in every line of the program but 'overlaps' by a constant distance. In Program example 2, the overlapping distance of the drilling axis and the positioning axis of each drilling command is 2 and 3 respectively. And the overlapping distance will not reset to zero until the disable command G80 is executed.
- c. In order to enable overlapping, set Pr4008 to 1, otherwise the Q and D parameters have no effect; If Pr4008 is set to 1 but the Q and D arguments are not given, the overlapping still won't be executed.

TYPE I : Normal Drilling

1. Effective condition : Pr4008 = 0.
2. Not supporting F2 argument, thus unable to control the lifting speed.
3. MPG simulation, Feedhold, Reset and G01 feedrate switching are valid during drilling process.

TYPE II : Rapid Drilling

1. Effective condition : Pr4008 = 1 and G01 feedrate before drilling should be 100%.
2. Better precision at the bottom of the hole.
3. The machine action is smoother when Z axis reverse at the bottom of the hole.
4. Not able to activate MPG simulation, Feedhold, Reset and change the G01 feedrate during the drilling process.
5. Do not modify the G01 feedrate during the drilling process of multiple holes, it might be switched to the normal drilling and it's unable to switch back.

2.54.3 Note

1. Please start spindle rotation with M code before applying G81.
2. If M Code and G81 are specified in the same block ,the M Code will only be executed once in the block.
3. When specified to repeat K times, M Code will only be executed at first cycle, it won't be executed for others.
4. Before changing the drilling axis, G81 cycle command must be cancelled first.
5. If no axis (X, Y, Z) moving command is included in the block, the drilling action won't be executed.
6. The argument specified R will only be set when executing the block with drilling action, it won't be set in the blocks with no drilling actions.
7. G codes G00, G01, G02, G03 can't be specified in the same block with G81, or the G81 cycle command will be cancelled.
8. In G81 cycle, the tool radius compensation (G41/G42/G40) will be ignored.
9. If the F2 argument is not an integer, the system alarm [COR-045 L Code must be an integer] will be issued.
10. It's unable to apply the overlapping function when C40 is On.
11. The valid version for Pr4008 in drilling mode: 10.116.10J, 10.116.16B
12. Arguments X,Y,Z,A,B,C,U,V,W are recognized as axis command when the parameter 3809 is 0.
13. The increment command for drilling axis is ignored when the increment and absolute command for drilling axis are commanded simultaneously.

2.54.4 Program Example

Example 1

```
N001 F1000. S500;
N002 G90;
N003 G00 X0. Y0. Z10.; // move to the initial point
```

```

N004 G17;
N005 M03 S1000; // start drill CW rotation
N006 G90 G99; // specify the coordinate of R point, Z point and hole NO.1
N007 G81 X5. Y5. Z-10. R-5. F2=1000; // set the lift up speed to 1000
N008 X15.; // hole NO.2
N009 Y15.; // hole NO.3
N010 G98 X5.; // hole NO.4, and return to the initial point
N011 X10. Y10. Z-20.; // hole NO.5, set new z point as -20
N012 G80;
N013 M05; // stop the tool
N014 M30;

```

Example 2

```

N001 F1000. S500;
N002 G90;
N003 G00 X0. Y0. Z10.; // move to the initial point
N004 G17;
N005 M03 S1000; // start drill CW rotation
N006 G90 G99; // specify the coordinate of R point, Z point and hole NO.1
N007 G81 X5. Y5. Z-10. R-5. Q2. D3.; // set the overlapping distance of the drilling axis(Q) and the positioning axis(D)
to 2 and 3 respectively
N008 X15.; // hole NO.2
N009 Y15.; // hole NO.3
N010 G98 X5.; // hole NO.4, and return to the initial point
N011 X10. Y10. Z-20.; // hole NO.5, set new z point as -20
N012 G80;
N013 M05; // stop the tool
N014 M30;

```

2.55 G81.1 : Chopping (ENG)

2.55.1 Command Form

G81.1 Z_ Q_ R_ F_

Z_ : Upper vertex coordinate of the Z axis movement (absolute value, machine coordinates, please specified another axis title if the axis is not Z axis)

Q_ : The distance between upper and lower vertex of the Z axis movement (increment value)

R_ : The distance between upper vertex of Z axis movement and point R (increment value)

F_ : feedrate

G80 : Cancel chopping

2.55.2 Description

- When an axis needs to move back and forth repeatedly, the chopping function can offer a more stable moving method with higher speed. For example, if applied on a grinding machine, the grinding axis will be moving back and forth more smoothly in a fast speed. This function will compensate server lag at the same time, usually the axis won't be moving to the specified position on the ends due to the server lag, running the compensation at the same time can make the axis moving to the correct position.
- The server lag compensation function is only valid by setting up Pr181~ and Pr3956 correctly.

3. The function can be activated in 2 ways, one is to execute G81.1 command in the machining file, the other is to trigger the related C bit with PLC (valid version: **before** 10.118.19 (included)). (For the PLC triggering method, please refer to C86/C87 of PLC Interface.)
4. When Chopping is activated, the tool will move to point R at G00 speed. The feedrate will be using the override register(R19) of Chopping and is limited to be lower than 100%. Moving back to point R after chopping or moving to the upper/lower vertex from point R before Chopping will both be moving at the Chopping speed.
5. As mentioned above, the feedrate is also using the override register(R19) of Chopping, the range is from 0% to 150% or section 1~15. (Please refer to Pr3207 C/S interface version number for further descriptions of feeding ratio)
6. The movement parameter of acceleration/deceleration section plannings is specified by the parameter of moving axis Pr541~ and Pr641~.

2.55.3 **Note**

1. Only Mill, Lathe, eHMC, FC, GlassGrind machines supports the Chopping function.
2. If switched the controller mode while the Chopping axis is moving, the controller will be feedhold and the Chopping axis will be stopped after moving back to the assigned R point till the machining process is executed again (Cycle Start, C0).
3. If paused the controller (C1) during the Chopping process, the Chopping axis will be stopped after moving back to the assigned R point till the machining process is executed again (Cycle Start, C0).
Note: The system will look-ahead, so after triggering (C1), it will not stop or return to R point immediately.
4. If pressed the emergency stop (C36) during the Chopping process, the Chopping axis will be stopped immediately.
5. If reset the controller (C37) during the Chopping process, the Chopping axis will be stopped after moving back to the assigned R point.
Note: The system will look-ahead, so after triggering (C37), it will not stop or return to R point immediately.
6. When a controller alarm issued, the Chopping axis will be moving according to the alarm level :
 - a. Alarms will make the system turn into Servo Off state: the Chopping axis will be stopped immediately.
 - b. Alarms will make the system stop the machining: the Chopping axis will be stopped after moving back to the assigned R point.
 - c. Alarms won't make the system stop the machining: the Chopping axis will be working normally.
7. Other limitations :
 - a. Each path supports a Chopping axis, the later commands will be ineffective if giving G81.1 command continuously.
 - b. When changing the axis or vertex coordinate under the same path, please disable G80 first then enable G81.1.
 - c. It's unable to apply the Chopping function in RTCP mode, or alarm COR-342 will be issued.
 - d. During the Chopping process, do not input any other moving commands of the Chopping axis, or alarm COR-339 will be issued.
 - Moving commands such as: G0, G1, G2/G3, G53, G31.....
 - e. During the Chopping process, do not modify the coordinate system data of the Chopping axis, or alarm COR-340 will be issued.
 - Coordinate system data such as: G54 P1~G54 P100, G92/G92.1, G10 L2, G10 L1300, G68/G68.2/G68.3, # value (#1880~#1933, #20001~#20658), external coordinates offset, MPG offset.
 - f. If the Chopping is in the PLC axis executing state, set as the spindle, or being assigned as the Chopping axis of another path, alarm COR-341 will be sent.
 - g. The Chopping axis does not support the mirror image function.
 - h. For versions before 10.118.19 (included), the PLC activation interface (C86/C87) is retained, do not apply the PLC Chopping with 1st path G81.1 Chopping.

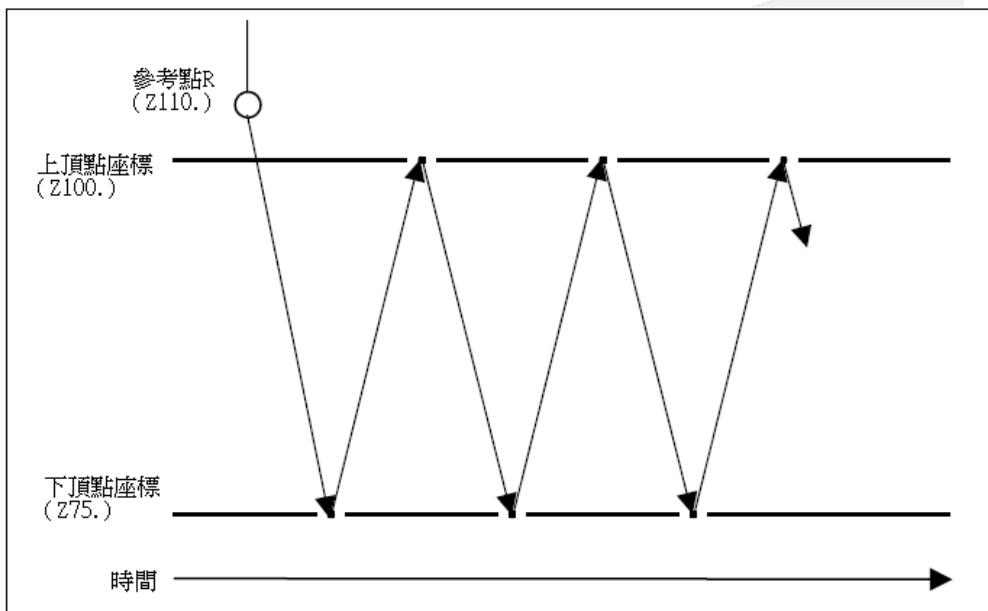
- i. If user want to pause chopping, it can be executed by ON C1 and resume by ON C0, or can be executed by ON C87 and resume by OFF C87(software version: 10.118.19 and **before**). These two ways can not be used at the same time.

2.55.4 Program Example

G90

G81.1 Z100. Q-25. R10. F3000 // the machine platform arrives at point R at G00 speed then moves back and forth between the upper and lower vertex

G80 // disable and return to point R



Note : The reference point is suggested to set outside of the upper and lower vertex to avoid the excessive axial jerk.

2.56 G81.2 High precision advanced chopping function

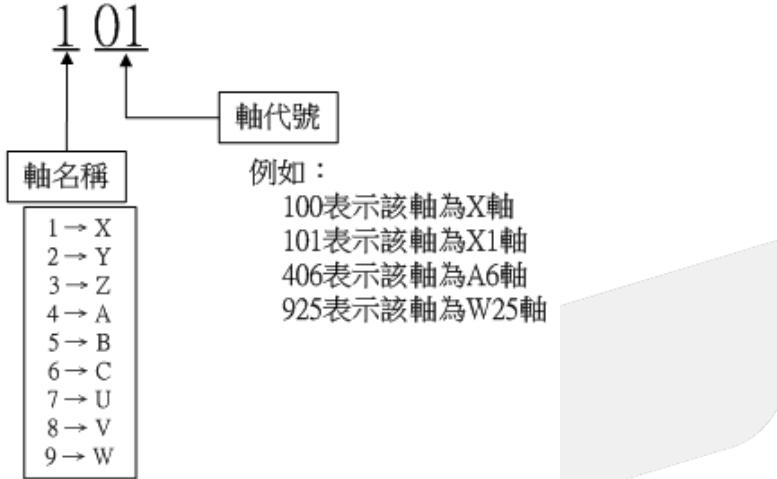
2.56.1 Command Form

G81.2 P_ Q_ [R_] F_ : Activate high precision advanced chopping function

P : The axis number or axis name that the advanced chopping axis correspond to

- e.g. axis name

- P101 set X1 as advanced chopping axis



- e.g. axis number
 - P1 set first axis as advanced chopping axis

Q : Assign the upper point reference Registry number, and the lower point reference registry number will be the upper point reference registry plus one.

- e.g Assign R20000 value as upper point.
 - Q20000 assign R20000 value as upper point, then R20001 value will be the lower point.

R : Reference Point (absolute command, machine coordinates, unit : IU), If the R is undetermined, the current position will set as Reference point.

F : feedrate (mm/min or inch/min)

2.56.2 Description

1. When an axis needs to move back and forth repeatedly, the advanced chopping function can offer a more stable moving method with higher speed.
 - a. Advanced chopping compensated server lag is valid by setting up Pr3808, with the compensation it can make the axis moving to the correct position.
2. The Registry number value represent the machine coordinates which the unit is BLU, the value will affected by Pr17 setting.
3. Motion description :
 - a. When advanced chopping is activated, the tool will move to point R at G00 speed. The feedrate will be using the override register(R19) of advanced chopping and is limited to be lower than 100%.
 - b. As mentioned a. If the Reference point is different from upper point and bottom point, the first move path will be the reference point to the farthest boundary point.
 - c. As mentioned a. the feedrate is also using the override register(R19) of advanced chopping, the range is from 0% to 150% or section 1~15. (Please refer to Pr3207 C/S interface version number for further descriptions of feeding ratio)
 - d. The movement parameter of acceleration/deceleration section plannings is specified by the parameter of moving axis Pr541~, Pr621~ and Pr641~.
 - e. G81.2 will wait for the previous command finished.
 - f. It will execute next command till G81.2 arrived reference point.
4. If the advanced chopping is deactivated by Reset (C37), Feedhold (C1) or Cancel (G80.2), the stop position is determined by Pr3957.

- a. If Pr3957 is set to 0 and the reference point is undetermined, the position at which chopping is activated will be set as the reference point.
- 5. If paused the controller during the advanced chopping process, it will pause advanced chopping till the machining process is executed again. (Cycle Start, C0)
- 6. When a controller alarm issued, the advanced chopping axis will be moving according to the alarm level :
 - a. Alarms will make the system turn into Servo Off state: the advanced chopping axis will be stopped immediately.
 - b. Alarms will make the system stop the machining: the advanced chopping axis stop position will follow the Pr3957 setting.
 - c. Alarms won't make the system stop the machining: the advanced chopping axis will be working normally.
- 7. If pressed the emergency stop (C36) during the advanced chopping process, the advanced chopping axis will be stopped immediately.
- 8. The advanced chopping axis doesn't support MPG Simulation.(C20 ON)
- 9. Single Block(C40), Program Pause (M00) will not stop advanced chopping axis.

2.56.3 **Note**

- 1. Option-48 to activate advanced chopping function.
- 2. Please avoid to use the system Registry.
- 3. If switched the controller mode while the advanced chopping axis is moving, the controller will be feedhold and the advanced chopping axis will be stopped
- 4. Other limitations :
 - a. Each path supports a advanced chopping axis
 - i. if giving G81.2 or G81.1 command continuously will post **COR-370 Failed to enable chopping function.**
 - b. The advanced chopping axis path modification is invalid.(example : the mirror image function.)

2.56.4 **Relative Alarm**

Alarm ID	COR-339	Alarm title	[Chopping axis prohibition movement command]
Description	The chopping axial direction does not accept any movement commands.		
Possible Cause	After using the chopping function (G81.1, C86), give movement command to the axis before closing. Note: C86 valid version: 10.118.19 and previous versions.		
Solution	Check the movement command of the NC program G code, whether there is chopping axis, and it is executed before the chopping function is turned off. The movement command G code is, for example, G0, G1, G2, G3, G31, G53.		
Alarm ID	COR-340	Alarm title	[Chopping axis prohibits changing coordinate system]
Description	The axis in chopping cannot change any coordinate system, and the related functions will be prohibited.		

Alarm ID	COR-340	Alarm title	[Chopping axis prohibits changing coordinate system]
Possible Cause	<ol style="list-style-type: none"> After using the chopping function (G81.1, C86), switch the coordinate system before closing and affect the chopping axis. Simultaneously use of chopping function (G81.1, C86) and tilted work plane machining function (G68.2, G68.3). Simultaneously use the chopping function (G81.1, C86) and the axis exchange function (C133~C136). <p>Note: C86 support version: 10.118.19 and earlier.</p>		
Solution	<ol style="list-style-type: none"> Check if the system operation and programming coordinate system are switched or changed, and whether the chopping axis is affected. Check if the NC program uses the chopping function (G81.1, C86) and the tilted work plane machining function at the same time. (G68.2, G68.3). Check if the NC program has the chopping function (G81.1, C86) and the shaft exchange of the chopping shaft. (C133~C136). <p>Note 1: C86 support version: 10.118.19 and earlier.</p> <p>Note 2: Coordinate system related programming: G54 P1~G54 P100, G92, G92.1, G10 L2, G10 L1300, G68, #value (#1880~#1933, #20001~#20658).</p> <p>Note 3: Coordinate system related operations: external coordinate offset, MPG offset.</p>		
Alarm ID	COR-341	Alarm title	[Chopping axial switching error]
Description	The specified axis cannot be switched to the chopping axis.		
Possible Cause	<ol style="list-style-type: none"> This axis has been designated as the PLC axis. The axis has been designated as the spindle. This axis has been designated as a chopping axis by other paths. 		
Solution	<ol style="list-style-type: none"> Do not specify the PLC axis as the chopping axis. Do not specify the spindle as the chopping axis. Check if the multi-path repeats the chopping function for the same axis (G81.1, C86). <p>Note: C86 support version: 10.118.19 and earlier version.</p>		
Alarm ID	COR-342	Alarm title	[Chopping axis prohibits RTCP mode]
Description	The RTCP mode is prohibited by the path using the chopping function.		
Possible Cause	<ol style="list-style-type: none"> When the chopping function is enabled, the RTCP mode is enabled. The machine type is the RTCP mode. 		

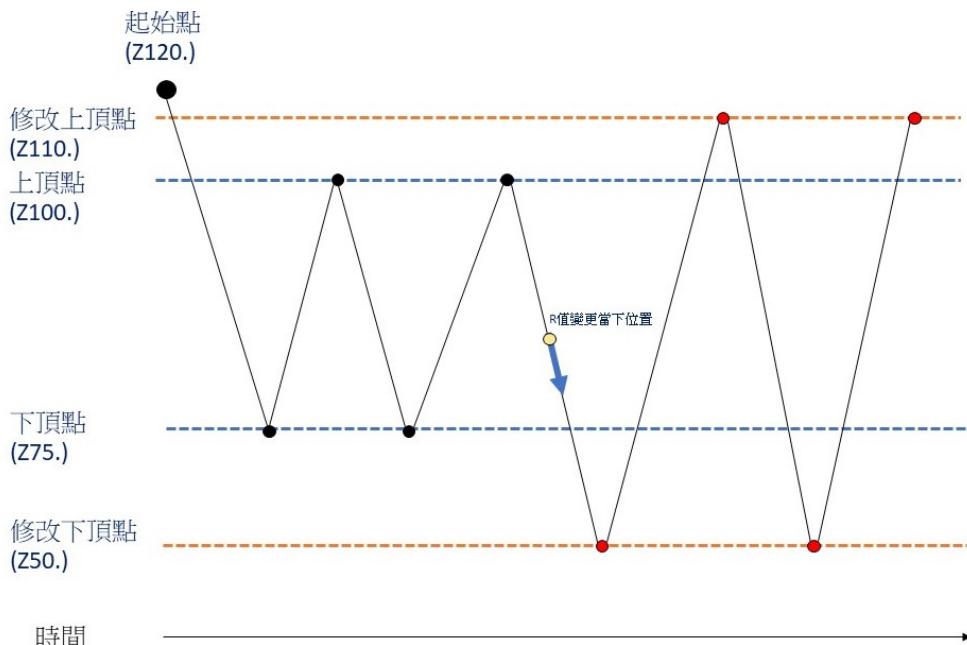
Alarm ID	COR-342	Alarm title	[Chopping axis prohibits RTCP mode]
Solution	<p>1. Check the NC program to make sure that the chopping function (G81.1, C86) is not within the range of the RTCP (G43.4, G43.5). 2. Check the NC program to make sure that the chopping function (G81.1, C86) is not within the effective range of the polar coordinate interpolation (G12.1). 3. The machine configuration used is the RTCP mode, and the chopping function (G81.1, C86) cannot be used. Note: C86 support version: 10.118.19 and earlier.</p>		
Alarm ID	COR-370	Alarm title	[Failed to enable chopping function]
Description	Failed to enable the chopping function .		
Possible Cause	<p>1. G81.2 or G81.1 command repeatedly a. G81.2 command repeatedly. b. G81.1 command repeatedly. c. G81.2 and G81.1 command simultaneously.</p>		
Solution	Please activate or deactivate the chopping function correctly.		

2.56.5 Program Example

```
%@ MACRO
// Pr17 = 2, 1BLU = 0.001mm
@120000 :=100000;           // set R20000 as 100mm
@120001 :=75000;            // set R20001as 75mm
G90;
G00 X0.;

G81.2 P3 Q20000 R110. F3000; // set Z axis as advanced chopping, R20000 value set as upper point, R20001 value set as lower point.
                                // the machine platform arrives at point R at G00 speed then moves back and forth between the upper and lower vertex
G01 X100.;                     // do chopping
                                // change R20000, R20001 value to 110000, 50000
                                // do chopping with new upper/ lower point

G80.2;                         // disable advanced chopping
M30;
```



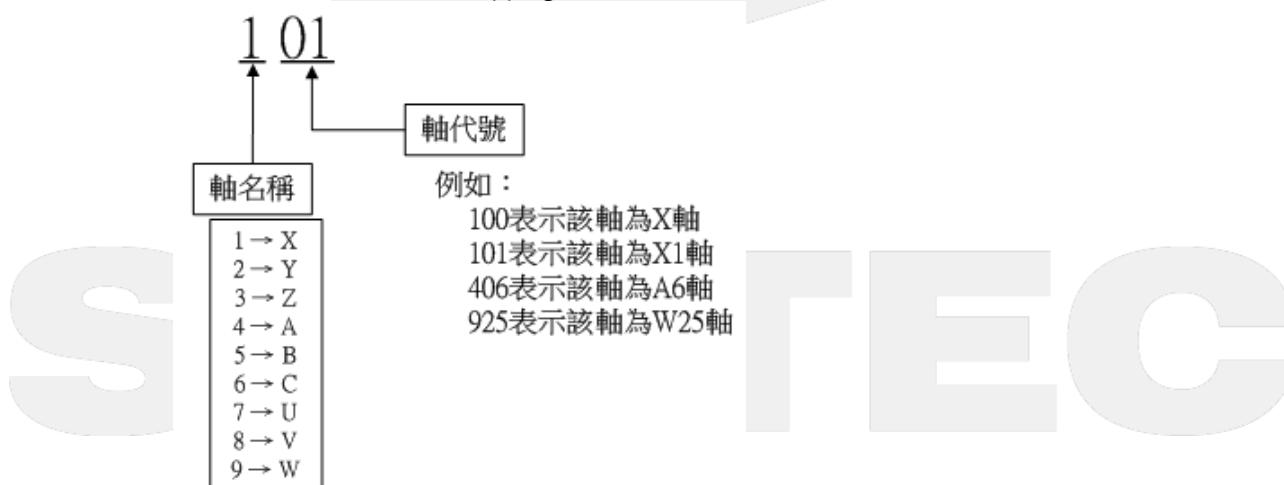
Note : The reference point is suggested to set outside of the upper and lower vertex to avoid the excessive axial jerk.

2.56.6 Command Form

G81.2 P_ Q_ [R_] F_ : Activate high precision advanced chopping function

P : The axis number or axis name that the advanced chopping axis correspond to

- e.g. axis name
 - P101 set X1 as advanced chopping axis



- e.g. axis number
 - P1 set first axis as advanced chopping axis

Q : Assign the upper point reference Registry number, and the lower point reference registry number will be the upper point reference registry plus one.

- e.g Assign R20000 value as upper point.
 - Q20000 assign R20000 value as upper point, then R20001 value will be the lower point.

R : Reference Point (absolute command, machine coordinates, unit : IU), If R is undetermined, the current position will considerate as Reference point.

F : feedrate (mm/min or inch/min)

2.56.7 Description

1. When an axis needs to move back and forth repeatedly, the advanced chopping function can offer a more stable moving method with higher speed.
 - a. Advanced chopping compensated server lag is valid by setting up Pr3808, with the compensation it can make the axis moving to the correct position.
2. The Registry number value represent the machine coordinates which the unit is BLU, the value will affected by Pr17 setting.
3. Motion description :
 - a. When advanced chopping is activated, the tool will move to point R at G00 speed. The feedrate will be using the override register(R19) of advanced chopping and is limited to be lower than 100%.
 - b. As mentioned a.If the Reference point is different from upper point and bottom point, the first move path will be the reference point to the farthest boundary point.
 - c. As mentioned a. the feedrate is also using the override register(R19) of advanced chopping, the range is from 0% to 150% or section 1~15. (Please refer to Pr3207 C/S interface version number for further descriptions of feeding ratio)
 - d. The movement parameter of acceleration/deceleration section plannings is specified by the parameter of moving axis Pr541~, Pr621~ and Pr641~.
 - e. G81.2 will wait for the previous command finished.
 - f. It will execute next command till G81.2 arrived reference point.
4. If reset the controller (C37), paused the controller (C1) or disable advanced chopping(G80.2) during the advanced chopping process, the stop position will follow the Pr3957 setting.In Pr3957 set 0 and without Reference point situation, the position at which chopping is activated will be set as the reference point.
5. If paused the controller durning the advanced chopping process, it will pause advanced chopping till the machining process is executed again. (Cycle Start, C0)
6. When a controller alarm issued, the advanced chopping axis will be moving according to the alarm level :
 - a. Alarms will make the system turn into Servo Off state: the advanced chopping axis will be stopped immediately.
 - b. Alarms will make the system stop the machining: the advanced chopping axis stop position will follow the Pr3957 setting.
 - c. Alarms won't make the system stop the machining: the advanced chopping axis will be working normally.
7. If pressed the emergency stop (C36) during the advanced chopping process, the advanced chopping axis will be stopped immediately.
8. The advanced chopping axis doesn't support MPG Simulation.(C20 ON)
9. Single Block(C40), Program Pause (M00) will not stop advanced chopping axis.

2.56.8 Note

1. Option-48 to activate advanced chopping function.
2. Please avoid to use the system Registry.
3. If switched the controller mode while the advanced chopping axis is moving, the controller will be feedhold and the advanced chopping axis will be stopped
4. Other limitations :

- a. Each path supports a advanced chopping axis
 - i. if giving G81.2 or G81.1 command continuously will post **COR-370 Failed to enable chopping function.**
- b. The advanced chopping axis path modification is invalid.(example : the mirror image function.)

2.56.9 Relative Alarm

Alarm ID	COR-339		Alarm title	[Chopping axis prohibition movement command]		
Description	The chopping axial direction does not accept any movement commands.					
Possible Cause	After using the chopping function (G81.1, C86), give movement command to the axis before closing. Note: C86 valid version: 10.118.19 and previous versions.					
Solution	Check the movement command of the NC program G code, whether there is chopping axis, and it is executed before the chopping function is turned off. The movement command G code is, for example, G0, G1, G2, G3, G31, G53.					
Alarm ID	COR-340	Alarm title	[Chopping axis prohibits changing coordinate system]			
Description	The axis in chopping cannot change any coordinate system, and the related functions will be prohibited.					
Possible Cause	<ol style="list-style-type: none"> 1. After using the chopping function (G81.1, C86), switch the coordinate system before closing and affect the chopping axis. 2. Simultaneously use of chopping function (G81.1, C86) and tilted work plane machining function (G68.2, G68.3). 3. Simultaneously use the chopping function (G81.1, C86) and the axis exchange function (C133~C136). <p>Note: C86 support version: 10.118.19 and earlier.</p>					
Solution	<ol style="list-style-type: none"> 1. Check if the system operation and programming coordinate system are switched or changed, and whether the chopping axis is affected. 2. Check if the NC program uses the chopping function (G81.1, C86) and the tilted work plane machining function at the same time. (G68.2, G68.3). 3. Check if the NC program has the chopping function (G81.1, C86) and the shaft exchange of the chopping shaft. (C133~C136). <p>Note 1: C86 support version: 10.118.19 and earlier.</p> <p>Note 2: Coordinate system related programming: G54 P1~G54 P100, G92, G92.1, G10 L2, G10 L1300, G68, #value (#1880~#1933, #20001~#20658).</p> <p>Note 3: Coordinate system related operations: external coordinate offset, MPG offset.</p>					

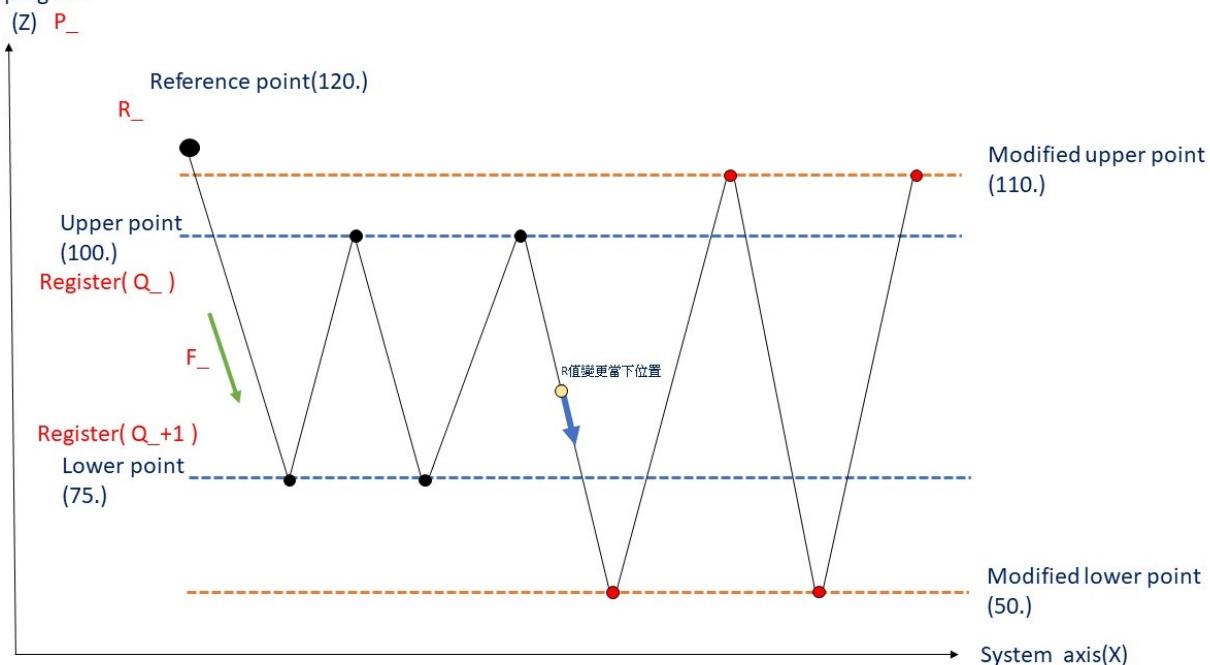
Alarm ID	COR-341	Alarm title	[Chopping axial switching error]
Description	The specified axis cannot be switched to the chopping axis.		
Possible Cause	<ol style="list-style-type: none"> 1. This axis has been designated as the PLC axis. 2. The axis has been designated as the spindle. 3. This axis has been designated as a chopping axis by other paths. 		
Solution	<ol style="list-style-type: none"> 1. Do not specify the PLC axis as the chopping axis. 2. Do not specify the spindle as the chopping axis. 3. Check if the multi-path repeats the chopping function for the same axis (G81.1, C86). <p>Note: C86 support version: 10.118.19 and earlier version.</p>		
Alarm ID	COR-342	Alarm title	[Chopping axis prohibits RTCP mode]
Description	The RTCP mode is prohibited by the path using the chopping function.		
Possible Cause	<ol style="list-style-type: none"> 1. When the chopping function is enabled, the RTCP mode is enabled. 2. The machine type is the RTCP mode. 		
Solution	<ol style="list-style-type: none"> 1. Check the NC program to make sure that the chopping function (G81.1, C86) is not within the range of the RTCP (G43.4, G43.5). 2. Check the NC program to make sure that the chopping function (G81.1, C86) is not within the effective range of the polar coordinate interpolation (G12.1). 3. The machine configuration used is the RTCP mode, and the chopping function (G81.1, C86) cannot be used. <p>Note: C86 support version: 10.118.19 and earlier.</p>		
Alarm ID	COR-370	Alarm title	[Failed to enable chopping function]
Description	Failed to enable the chopping function .		
Possible Cause	<ol style="list-style-type: none"> 1. G81.2 or G81.1 command repeatedly <ol style="list-style-type: none"> a. G81.2 command repeatedly. b. G81.1 command repeatedly. c. G81.2 and G81.1 command simultaneously. 		
Solution	Please activate or deactivate the chopping function correctly.		

2.56.10 Program Example

```
%@ MACRO
// Pr17 = 2, 1BLU = 0.001mm
@120000 :=100000;           // set R20000 as 100mm
@120001 :=75000;            // set R20001 as 75mm
G90;
G00 X0.;

G81.2 P3 Q20000 R110. F3000; // set Z axis as advanced chopping, R20000 value set as upper point, R20001
                               // value set as lower point.
                               // the machine platform arrives at point R at G00 speed then moves back and forth
between the upper and lower vertex
G01 X100.;                  // do chopping
                           // change R20000, R20001 value to 110000, 50000
                           // do chopping with new upper/ lower point
G80.2;                      // disable advanced chopping
M30;
```

Chopping axis



Note : The reference point is suggested to set outside of the upper and lower vertex to avoid the excessive axial jerk.

2.57 G82 : Drilling Cycle with Dwelling at Hole Bottom

i 中文版 Mandarin Version: G82 : 孔底暂停钻孔循环

2.57.1 Command Form

G82 X_Y_Z_R_P_F_K_;

X(U) or Y(V): hole position (absolute/increment, please set parameter 3809 is 1 when using increment value)

Z:

G91: the distance from the R point to Z point (directional)

G90: program coordinate of Z point

R:

G91: the distance from the initial point to R point (directional)

G90: program coordinate of R point

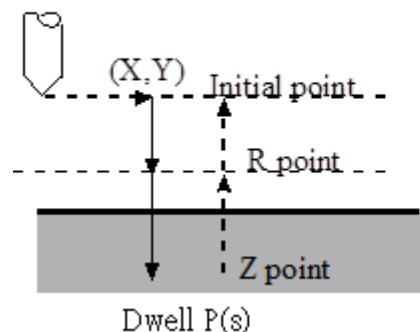
P: dwelling time(sec)

F: feedrate

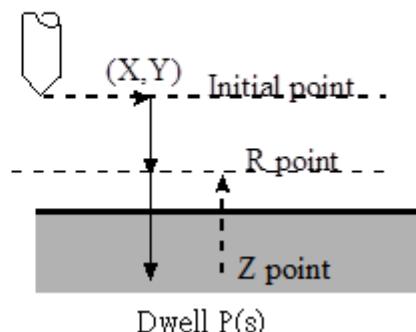
K: repeating times (repeating the moving and drilling, G91 increment input is effective)

Absolute/Increment coordinate of X, Y, Z, R can be specified by G90/G91.

G98



G99



2.57.2 Description

1. Start the machining process, move the tool to specified (X,Y) with G00.
2. Move the tool to the specified R point with G00.
3. Move the tool to the Z point at hole bottom with G01.
4. Dwell for P seconds.
5. Lift up to the initial point (G98) or the R point of the program (G99) with G00.

2.57.3 Notes

1. Please start spindle rotation with M code before applying G82.
2. If M Code and G82 are specified in the same block ,the M Code will only be executed once in the block.
3. When specified to repeat K times, M Code will only be executed at first cycle, it won't be executed for others.
4. Before changing the drilling axis, G82 cycle must be cancelled first.
5. If no axis (X, Y, Z) moving command is included in the block, the drilling action won't be executed.
6. The argument specified by R will only be set when executing the block with drilling action, it won't be set in the blocks with no drilling actions.
7. G codes G00, G01, G02, G03 can't be specified in the same block with G82, or G82 command will be cancelled.

8. In G82, the tool radius compensation (G41/G42/G40) will be ignored.
9. Arguments X,Y,Z,A,B,C,U,V,W are recognized as axis command when the parameter 3809 is 0.
10. The increment command for drilling axis is ignored when the increment and absolute command for drilling axis are commanded simultaneously.

2.57.4 Example

```
N001 F1000. S500;
N002 G90;
N003 G00 X0. Y0. Z10.; // move to the initial point
N004 G17;
N005 M03; // start drill CW rotation
N006 G90 G99;
// specify the coordinate of R point, Z point and hole NO.1, dwelling time 2 seconds
N007 G82 X5. Y5. Z-10. R-5. P2.;
N008 X15.; // hole NO.2
N009 Y15.; // hole NO.3
N010 G98 X5.; // hole NO.4, return to the initial point
N011 G80;
N012 M05; // stop the tool
N013 M30;
```

2.58 G83 : Peck Drill Cycle

Command Form

G83 X_Y_Z_R_Q_F_K_ ;

X(U) or Y(V): hole position (absolute/increment, please set parameter 3809 is 1 when using increment value)

Z:

G91: the distance from the R point to Z point (directional)

G90: program coordinate of Z point

R:

G91: the distance from the initial point to R point (directional)

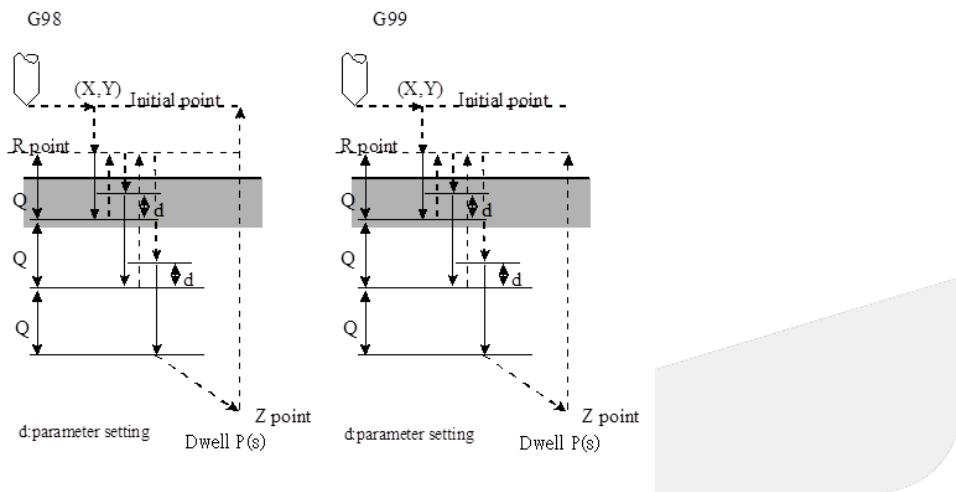
G90: program coordinate of R point

Q: depth of each feed (increment and positive, the minus will be ignored)

F: feedrate

K: repeating times (repeating the moving and drilling, G91 increment input is effective)

Absolute/Increment coordinate of X, Y, Z, R can be specified by G90/G91.



2.58.1 Description

1. Start the machining process, move the tool to specified (X,Y) with G00.
2. Move the tool to the specified R point with G00.
3. Downwards a distance Q relative to the current drilling depth with G01.
4. Lift the tool up to point R on workpiece surface with G00.
5. Move the tool to the position with a retracing distance d above the current cutting depth. (set by Pr4002)
6. Move the tool for a distance of a cutting feed Q from the current cutting depth with G01 again.
7. Lift the tool up to point R on workpiece surface with G00.
8. Repeat the steps above till reaching point at the bottom of the hole.
9. Lift the tool up to the initial point (G98) or point R of the program (G99).

2.58.2 Note

1. Please start spindle rotation with M code before applying G83.
2. If M Code and G83 are specified in the same block ,the M Code will only be executed once in the block.
3. When specified to repeat K times, M Code will only be executed at first cycle, it won't be executed for others.
4. Before changing the drilling axis, G83 cycle command must be cancelled first.
5. If no axis (X, Y, Z) moving command is included in the block, the drilling action won't be executed.
6. The argument specified by Q and R will only be set when executing the block with drilling action, it won't be set in the blocks with no drilling actions.
7. G codes G00, G01, G02, G03 can't be specified in the same block with G83, or the G83 cycle command will be cancelled.
8. In G83 cycle, the tool radius compensation (G41/G42/G40) will be ignored.
9. Arguments X,Y,Z,A,B,C,U,V,W are recognized as axis command when the parameter 3809 is 0.
10. The increment command for drilling axis is ignored when the increment and absolute command for drilling axis are commanded simultaneously.

Program Example

```

N001 F1000. S500;
N002 M03; // start the drill, CW rotation
N003 G90;
N004 G00 X0. Y0. Z10.; // move to the initial point
N005 G17;
N006 G90 G99;

```

```
// specify the coordinate of R point, Z point and hole NO.1, cutting depth 3.0
N007 G83 X5. Y5. Z-10. R-5. Q3.;
N008 X15.; // hole NO.2
N009 Y15.; // hole NO.3
N010 G98 X5.; // hole NO.4, return to the initial point
N011 G80;
N012 M05; // stop the tool
N013 M30;
```

2.59 G84 : Tapping Cycle

Command Form

G84 X_ Y_ Z_ R_ P_ Q_ (F_ or E_) K_ I_ J_;

X(U) or Y(V): hole position (absolute/increment, please set parameter 3809 is 1 when using increment value)

Z:

G91: the distance from the R point to Z point (directional)

G90: program coordinate of Z point

R:

G91: the distance from the initial point to R point (directional)

G90: program coordinate of R point

P: dwelling time; (sec)

Q: depth of each feed (increment and positive, the minus will be ignored)

F: feedrate

E: number of threads per inch (once F and E argument are both specified, then E argument would be ignored), valid version : after 10.116.16B、10.116.18、10.117.19.

K: repeating times (repeating the moving and tapping, G91 increment input is effective)

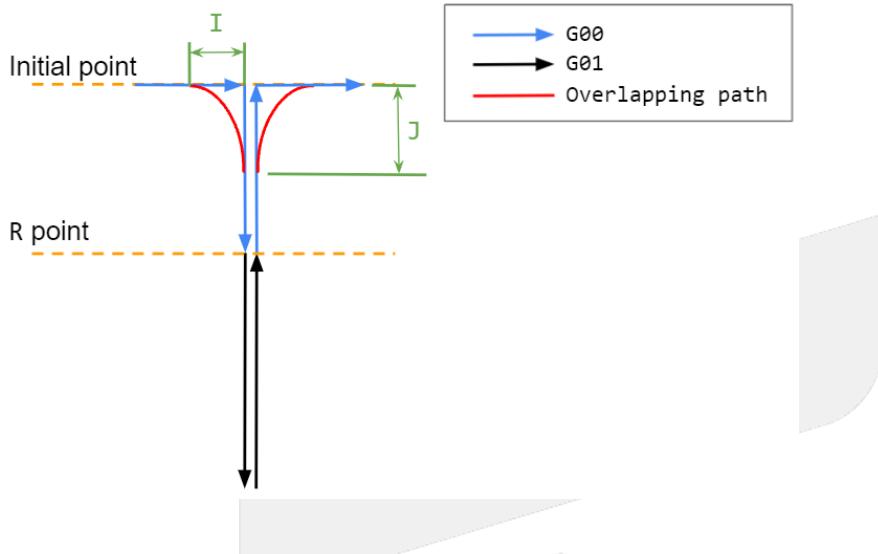
Absolute/Increment coordinate of X, Y, Z, R can be specified by G90/G91.

I : Overlapping distance of the positioning axis (After the previous tapping finishes, it is the overlapping distance from G00 positioning to the current position.). It is valid when Pr4008(setting for drilling/tapping mode) value is 1. Unit: LIU.

J : Overlapping distance of the tapping axis (When the tapping finishes, it is the overlapping distance of exit the hole along the tapping axis to the next G00.). It is valid when Pr4008(setting for drilling/tapping mode) value is 1. Unit: LIU.

1. ~10.118.47: I, J parameters are not supported.
2. 10.118.48A~10.118.48D, 10.118.48~10.118.50: I is the overlapping distance of the tapping axis; J is the overlapping distance of the positioning axis.

3. 10.118.48E, 10.118.51 or above: I is the overlapping distance of the positioning axis; J is the overlapping distance of the tapping axis.



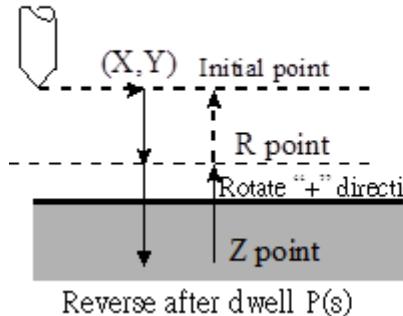
2.59.1 **Description**

1. The process of G84
 - a. When G84 starts, the tool moves to the point of command (X, Y) by G00
 - b. move down to point R by G00
 - c. start tapping
 - d. move up to the initial point (G98) or R point (G99)
2. The action of overlapping
 - a. When the program has two consecutive blocks of G84 or the consecutive blocks of G84 and G00, the second block will start when the first block remains a certain distance from the finish. The distance is called "overlapping distance", see I & J in the figure above.
 - b. Overlapping is suitable for the continuous tapping cycle. It does not need I/J argument in every line of the program but 'overlaps' by a constant distance. In Program example 2, the overlapping distance of the positioning axis and the tapping axis of each tapping command is 2 and 3 respectively. And the overlapping distance will not reset to zero until the disable command G80 is executed.
 - c. In order to enable overlapping, set Pr4008 to 1, otherwise the I and J parameters have no effect; If Pr4008 is set to 1 but the I and J arguments are not given, the overlapping still won't be executed.
 - d. Overlapping will only be valid above R point. Remark: if G84 only moves up to R point(G99), overlapping won't be executed.

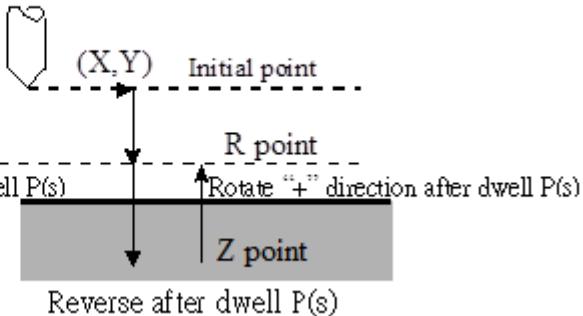
SYNTEC

TYPE I : Without Q argument

G98



G99

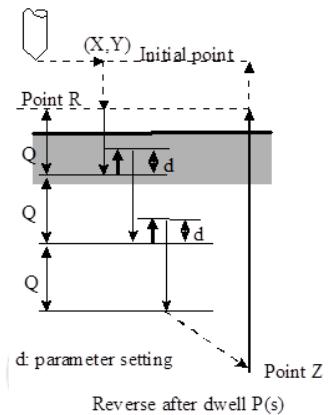


Explanation

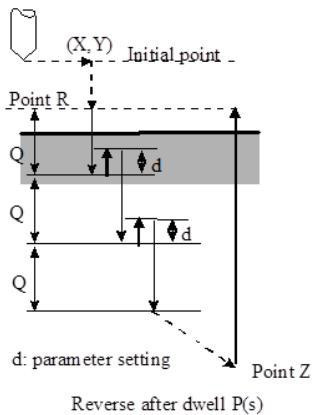
1. Start the machining process, move the tool to specified (X,Y) with G00.
2. Move the tool to the specified R point with G00.
3. Execute spindle orientation (action can be skipped if Pr4007=0).
4. Tapping till the bottom Z point with G01.
5. Dwell for P seconds and reverse the tool.
6. Lift up to R point with G01.
7. Dwell for P seconds and reverse the tool.
8. Return to the initial point (G98) or the R point (G99) with G00.

TYPE II : Rapid Peck Tapping (Pr4001=1)

G98



G99

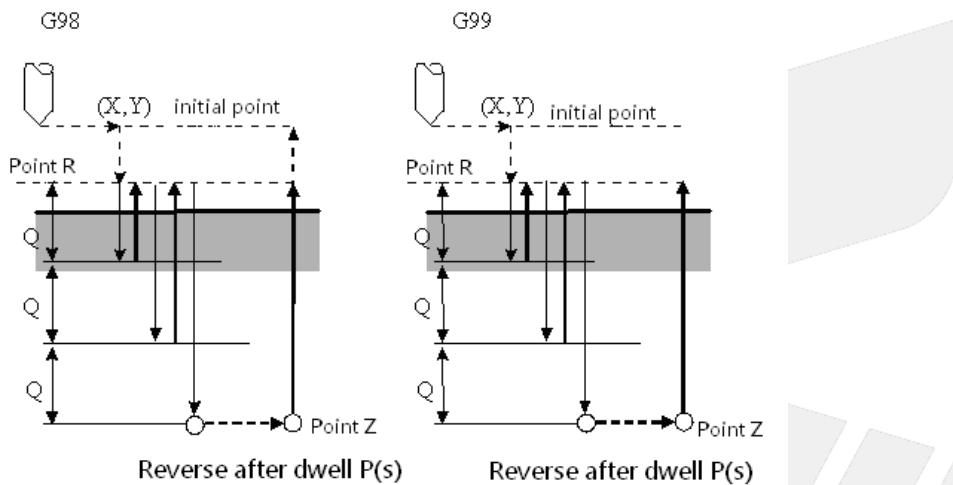


Explanation:

1. Start the machining process, move the tool to specified (X,Y) with G00.
2. Move the tool to the specified R point with G00.
3. Execute spindle orientation (action skippable if Pr4007=0).
4. Tapping downwards for a distance of Q from the current tapping depth with G01.
5. Dwell for P seconds and reverse the tool, lift up a retract depth d with G01 (Pr4004).
6. Dwell for P seconds and reverse the tool, tapping downwards again a distance Q relative to the current tapping depth with G01.
7. Dwell for P seconds and reverse the tool, lift up a retract depth d with G01 (Pr4004).

8. Repeat tapping above till reaches the bottom Z point.
9. Dwell for P seconds and reverse the tool.
10. Lift up to R point with G01 (G99).
11. Dwell for P seconds and reverse the tool.
12. Lift up to the initial point with G00 (G98).

TYPE III : Normal Peck Tapping (Pr4001=0)



Explanation:

1. Start the machining process, move the tool to specified (X,Y) with G00.
2. Move the tool to the specified R point with G00.
3. Execute spindle orientation (action skippable if Pr4007=0).
4. Tapping downwards for a distance of Q from the current tapping depth with G01.
5. Dwell for P seconds and reverse the tool, lift up R point with G01.
6. Dwell for P seconds and reverse the tool, tapping downwards again a distance Q relative to the current tapping depth with G01.
7. Dwell for P seconds and reverse the tool, lift up R point with G01.
8. Repeat tapping above till reaches the bottom Z point.
9. Dwell for P seconds and reverse the tool.
10. Lift up to R point with G01 (G99).
11. Dwell for P seconds and reverse the tool.
12. Lift up to the initial point with G00 (G98).

Tapping Pitch/Machining Speed Calculation

G94 : Machining Speed (F mm/min) = Spindle Speed (S r.p.m) * Pitch (P mm/rev)

G95 : Machining Speed (F mm/rev) = Pitch (P mm/rev)

G84 : During the process, machining speed F and spindle speed S is independent of the override switch (Fixed at 100%)

2.59.2 Note

1. Please start spindle rotation with M code before applying G84.
2. If M Code and G84 are specified in the same block ,the M Code will only be executed once in the block.
3. When specified to repeat K times, M Code will only be executed at first cycle, it won't be executed for others.

4. G84 is a module G Code, effective after being executed. If an (X,Y) coordinate is the only argument given in the next line ,the controller will start the tapping at (X,Y).
5. G84 can be cancelled by G80, or being cancelled automatically when the program executes G00, G01, G02, G03 or other G code cycles.
6. If pressed the pause or reset button during the tapping process, the current hole tapping action will be completed then stops at R point.
7. The angle of the spindle orientation before tapping can be specified by the spindle origin offset value (Pr1771~Pr1780).
8. G codes G00, G01, G02, G03 can't be specified in the same block with G84, or the G84 cycle command will be cancelled.
9. In G84 cycle, the tool radius compensation (G41/G42/G40) will be ignored.
10. The spindle orientation function before tapping is valid from version 10.116.14 and is only provided for serial spindles.
11. Before changing the machining spindle (R791~), please cancel the cycle with G80 to avoid unexpected machining action if the current spindle is in tapping state.
12. Base the model support Rapid tapping, while processing G84 without Q and P argument and spindle is serial or Pr1791=3, then rapid tapping is launched,
13. Arguments X,Y,Z,A,B,C,U,V,W are recognized as axis command when the parameter 3809 is 0.
14. The increment command for tapping axis is ignored when the increment and absolute command for tapping axis are commanded simultaneously.

2.59.3 Program Example

Example 1:

```
N001 F1000. S500;
N002 G90;
N003 G00 X0. Y0. Z10.; // move to the initial point
N004 G17;
N005 M03; // start reverse the tool.
N006 G90 G99; // specify the coordinate of R point, Z point and hole No.1.
N007 G84 X5. Y5. Z-10. R-5.;
N008 X15.; // hole No.2
N009 Y15.; // hole No.3
N010 G98 X5.; // hole No.4, return to the initial point
N011 G80;
N012 M05; // stop the tool
N013 M30;
```

Example 2:

```
N001 F1000. S500;
N002 G90;
N003 G00 X0. Y0. Z10.; // move to initial point
N004 G17;
N005 M03; // start the spindle turning clockwise
N006 G90 G98; // specify the coordinate of R point, Z point and hole No.1.
N007 G84 X5. Y5. Z-10. R-5 I2. J3.; // set the overlapping distance of positioning axis(I) and tapping axis(J) to 2 and 3 respectively, the overlap section only exists above R point.
N008 X15.; // hole No.2
N009 Y15.; // hole No.3
N010 G99 X5.; // hole No.4, return to R point
N011 Y5.; // hole No.5, overlapping won't be executed
```

N013 G80;
 N014 M05; // stop the tool
 N015 M30;

2.60 G84.48 : Tapping Retract

Command Form

G84.48 (F_ or S_ or F_S_);

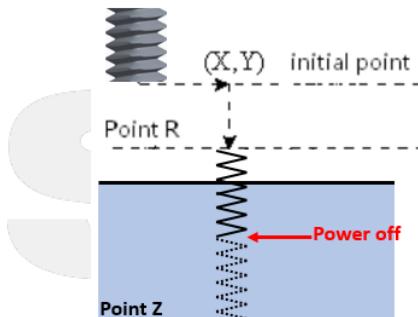
F: feedrate

S: spindle speed

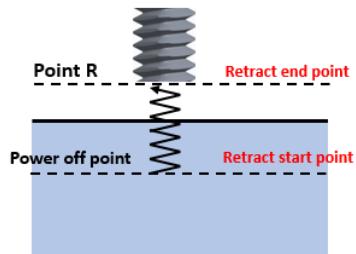
2.60.1 Description

1. If tapping abort is caused by emergency stop or electricity power off, user can follow this command in MDI or Auto mode to execute tapping retraction.
2. The retract G code is G84.48
3. The tool will retract to tapping initial R point.
4. The retraction status is identical to initial tapping status. Do not change the G code status before retraction.
5. When tapping happened as using G84/G74 with E argument(number of threads per inch), the system would automatically turn to save as the F and S argument instead.
6. G84.48 can be used with argument and either without argument. If G84.48 is used without argument, the retraction speed and feedrate follow G84/G74.
7. It is fine to input only F or only S as the G84.48 argument, either input both. If argument F is exist, the unit of F is mm/min.
8. The F to S ratio need to be same as initial when both F and S argument are used, otherwise the system will show alarm.
9. The G code can be used like the instruction bellow:

G84.48 { [F_]
 [S_]
 [F_S_]

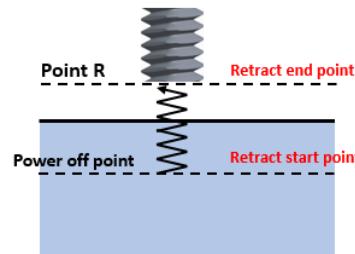


G74/G84



The retract speed is followed G84

G84.48



The retract speed is followed G84.48 argument

G84.48 F/S

2.60.2 Note

1. G84.48 is valid from version 10.118.28S.

2. Only support G84/G74
3. This is standard function for Mill and Glassgrind, other products can use G10L1030 and G10L1031 to implement, please refer Industry Machine Application Manual. for more details.

2.61 G85 : Drilling Cycle

Command Form

G85 X_Y_Z_R_F_K_ ;

X(U) or Y(V): hole position (absolute/increment, please set parameter 3809 is 1 when using increment value)

Z:

G91: the distance from the R point to Z point (directional)

G90: program coordinate of Z point

R:

G91: the distance from the initial point to R point (directional)

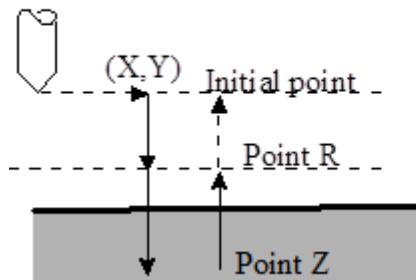
G90: program coordinate of R point

F: feedrate

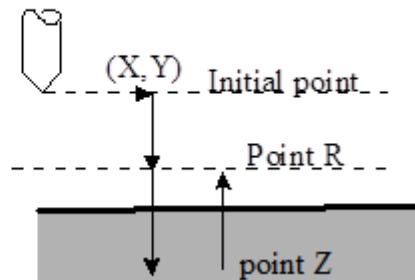
K: repeating times (repeating the moving and drilling, G91 increment input is effective)

Absolute/Increment coordinate of X, Y, Z, R can be specified by G90/G91.

G98



G99



2.61.1 Description

1. Start the machining process, move the tool to specified (X,Y) with G00.
2. Move the tool to the specified R point with G00.
3. Move the tool to the Z point at the bottom of the hole with G01.
4. Move the tool to the R point with G01.
5. Return to the initial point (G98) or the R point (G99) with G00.

2.61.2 Note

1. Please start spindle rotation with M code before applying G85.
2. If M Code and G85 are specified in the same block ,the M Code will only be executed once in the block.
3. When specified to repeat K times, M Code will only be executed at first cycle, it won't be executed for others.
4. Before changing the drilling axis, G85 cycle command must be cancelled first.
5. If no axis (X, Y, Z) moving command is included in the block, the drilling action won't be executed.

6. The argument specified R will only be set when executing the block with drilling action, it won't be set in the blocks with no drilling actions.
7. G codes G00, G01, G02, G03 can't be specified in the same block with G85, or the G85 cycle command will be cancelled.
8. In G85 cycle, the tool radius compensation (G41/G42/G40) will be ignored.
9. Arguments X,Y,Z,A,B,C,U,V,W are recognized as axis command when the parameter 3809 is 0.
10. The increment command for drilling axis is ignored when the increment and absolute command for drilling axis are commanded simultaneously.

2.61.3 Program Example

```
N001 F1000. S500;
N002 G90;
N003 G00 X0. Y0. Z10.; // move to the initial point
N004 G17;
N005 M03; // start the drill, CW rotation
N006 G90 G99; // specify the coordinate of R point, Z point and hole NO.1
N007 G85 X5. Y5. Z-10. R-5.;
N008 X15.; // hole NO.2
N009 Y15.; // hole NO.3
N010 G98 X5.; // hole NO.4, and return to initial point
N011 G80;
N012 M05; // stop the tool
N013 M30;
```

2.62 G86 : High Speed Drilling Cycle

Command Form

G86 X_Y_Z_R_F_K_;

X(U) or Y(V): hole position (absolute/increment, please set parameter 3809 is 1 when using increment value)

Z:

G91: the distance from the R point to Z point (directional)

G90: program coordinate of Z point

R:

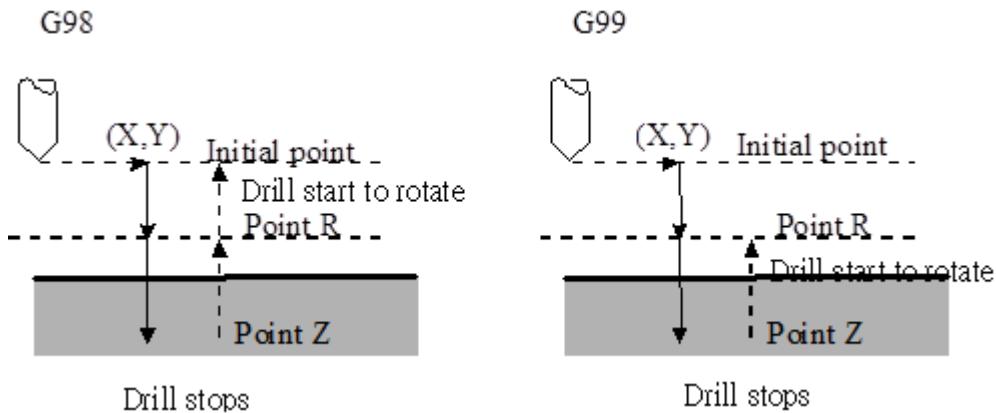
G91: the distance from the initial point to R point (directional)

G90: program coordinate of R point

F: feedrate

K: repeating times (repeating the moving and drilling, G91 increment input is effective)

Absolute/Increment coordinate of X, Y, Z, R can be specified by G90/G91.



2.62.1 Description

1. Start the machining process, move the tool to specified (X,Y) with G00.
2. Move the tool to the specified R point with G00.
3. Move the tool to the Z point at hole bottom with G01.
4. Return to the initial point (G98) or the R point of the program (G99) with G00.

2.62.2 Note

1. Please start spindle rotation with M code before applying G86.
2. If M Code and G86 are specified in the same block ,the M Code will only be executed once in the block.
3. When specified to repeat K times, M Code will only be executed at first cycle, it won't be executed for others.
4. Before changing the drilling axis, G86 cycle cycle must be cancelled first.
5. If no axis (X, Y, Z) moving command is included in the block, the drilling action won't be executed.
6. The argument specified by R will only be set when executing the block with drilling action, it won't be set in the blocks with no drilling actions.
7. G codes G00, G01, G02, G03 can't be specified in the same block with G86, or the G86 cycle command will be cancelled.
8. In G86 cycle, the tool radius compensation (G41/G42/G40) will be ignored.
9. Arguments X,Y,Z,A,B,C,U,V,W are recognized as axis command when the parameter 3809 is 0.
10. The increment command for drilling axis is ignored when the increment and absolute command for drilling axis are commanded simultaneously.

2.62.3 Program Example

```

N001 F1000. S500;
N002 G90;
N003 G00 X0. Y0. Z10.; // move to the initial point
N004 G17;
N005 M03; // start drill CW rotation
N006 G90 G99; //specify the coordinate of R point, Z point and hole NO.1
N007 G86 X5. Y5. Z-10. R-5.;
N008 X15.; // hole NO.2
N009 Y15.; // hole NO.3
N010 G98 X5.; // hole NO.4, return to initial point
N011 G80;
N012 M05; // stop the tool
N013 M30;

```

2.63 G87 : Fine Boring Cycle on Back Side

Command Form

G87 X_Y_Z_R_Q_P_F_K_ ;

X(U) or Y(V): hole position (absolute/increment, please note if parameter 3809 is 1 when using increment value)

Z:

G91: the distance from the R point to point Z (directional)

G90: program coordinate position of Z point

R:

G91: the distance from the starting point to point R (directional)

G90: program coordinate position of R point

Q: tool moving distance (positive value, the minus will be ignored)

P: dwelling time; time stayed at the bottom of the hole (with decimal points, unit: second; without decimal points, please refer to Pr17 and Pr 3241)

F: feedrate

K: repeating times (repeating the moving and drilling actions, G91 increment input is effective)

Absolute/Increment coordinate of X, Y, Z, R can be decided by G90/G91.

G98, G99

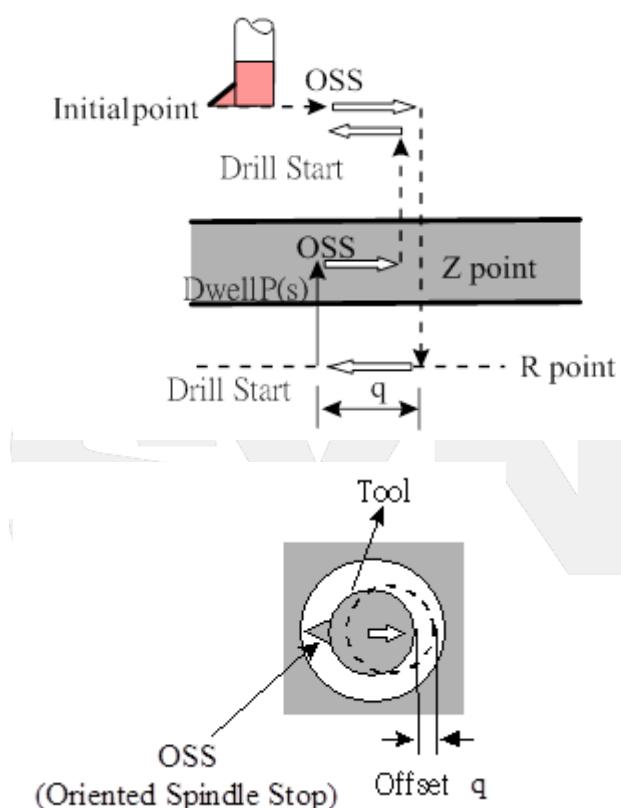


figure : Oriented Spindle Stop(OSS)

2.63.1 **Description**

1. Start the machining process, move the tool to specified (X,Y) with G00.
2. After OSS, shift the distance of a boring offset Q in the opposite boring tool direction set by Pr4020.
3. Move the tool to specified point R with G00, shift the distance of a boring offset Q.
4. Drill CW rotation.
5. Lift up to point Z with G01.
6. Dwell for P seconds and shift the distance of a boring offset Q in the opposite direction.
7. Lift up to the initial point with G00.
8. After the drill starts, shift the distance of Q.

2.63.2 **※Alarm**

- Q is a modal value required in G87 cycle. This Q value must be specified carefully since it is also applied in G73/G83 cycle.
- The OSS(Oriented Spindle Stop) direction is decided by parameter No. 4020:

Parameter 4020	OSS Direction
0	+X
1	-X
2	+Y
3	-Y
4	+Z
5	-Z

Note

1. Please start spindle rotation with M code before applying G87.
2. If M Code and G87 are specified in the same block ,the M Code will only be executed once in the block.
3. When specified to repeat K times, M Code will only be executed at first cycle, it won't be executed for others.
4. Before changing the drilling axis, G87 cycle must be cancelled first.
5. If no axis (X, Y, Z) moving command is included in the block, the drilling action won't be executed.
6. The Q value should be positive, it'll still be defined as a positive value even it's set negative (absolute value).
7. The argument specified by R will only be set when executing the block with drilling action, it won't be set in the blocks with no drilling actions.
8. G codes G00, G01, G02, G03 can't be specified in the same block with G87, or G87 command will be cancelled.
9. In G87, the tool radius compensation (G41/G42/G40) will be ignored.
10. Arguments X,Y,Z,A,B,C,U,V,W are recognized as axis command when the parameter 3809 is 0.
11. The increment command for boring axis is ignored when the increment and absolute command for boring axis are commanded simultaneously.

2.63.3 Program Example

```

N001 F1000. S500;
N002 G90;
N003 G00 X0. Y0. Z10.; // move to the initial point
N004 G17;
N005 G90 ;
N006 M03; // start drill CW rotation
// specify the coordinate of point R, point Z and hole NO.1, moving distance 5.0, dwelling time 4.0 seconds
N007 G87 X5. Y5. Z10. R-30. Q5. P4.;
N008 X15.; // hole NO.2
N009 Y15.; // hole NO.3
N010 G80;
N011 M05; // stop the tool
N012 M30;

```

2.64 G88 : Semiautomatic Fine Boring Cycle

Command Form

G88 X_Y_Z_R_P_F_K_ ;

X(U) or Y(V): hole position (absolute/increment, please note if parameter 3809 is 1 when using increment value)

Z:

G91: the distance from the R point to point Z (directional)

G90: program coordinate position of Z point

R:

G91: the distance from the starting point to point R (directional)

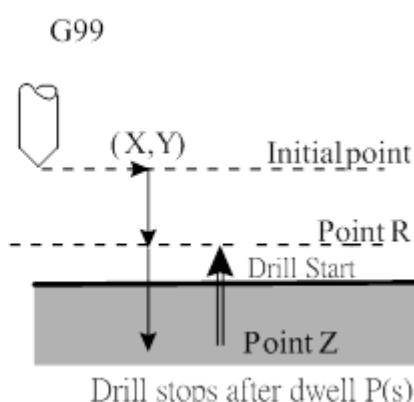
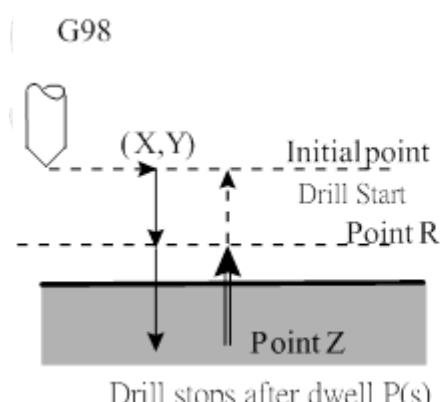
G90: program coordinate position of R point

P: dwelling time; time stayed at the bottom of the hole (with decimal points, unit: second; without decimal points, please refer to Pr17 and Pr 3241)

F: feedrate

K: repeating times (repeating the moving and drilling actions, G91 increment input is effective)

Absolute/Increment coordinate of X, Y, Z, R can be decided by G90/G91.





※ for positioning by manual.

2.64.1 Description

1. Start the machining process, move the tool to specified (X,Y) with G00..
2. Move the tool to specified point R with G00.
3. Move the tool to point Z with G01.
4. Dwell for P seconds.
5. Lift up out of workpiece by manually then restart.
6. Lift up to point R with G01.
7. Lift up to the initial point with G00 (G98) or to the R point of the program (G99).
8. Drill rotation starts.

2.64.2 Note

1. Please start spindle rotation with M code before applying G88.
2. If M Code and G88 are specified in the same block ,the M Code will only be executed once in the block.
3. When specified to repeat K times, M Code will only be executed at first cycle, it won't be executed for others.
4. Before changing the drilling axis, G88 cycle must be cancelled first.
5. If no axis (X, Y, Z) moving command is included in the block, the drilling action won't be executed.
6. The argument specified by R will only be set when executing the block with drilling action, it won't be set in the blocks with no drilling actions.
7. G codes G00, G01, G02, G03 can't be specified in the same block with G88, or G88 command will be cancelled.
8. In G88, the tool radius compensation (G41/G42/G40) will be ignored.
9. Arguments X,Y,Z,A,B,C,U,V,W are recognized as axis command when the parameter 3809 is 0.
10. The increment command for boring axis is ignored when the increment and absolute command for boring axis are commanded simultaneously.

2.64.3 Program Example

```

N001 F1000. S500;
N002 G90;
N003 G00 X0. Y0. Z10.; // move to the initial point
N004 G17;
N005 M03; // start drill CW rotation
N006 G90 G99;
// specify the coordinate of point R, point Z and hole NO.1, dwelling time 2.0 seconds
N007 G88 X5. Y5. Z-10. R-5. P3.;
N008 X15.; // hole NO.2
N009 Y15.; // hole NO.3
N010 G98 X5.; // hole NO.4, return to the initial point
N011 G80;
N012 M05; // stop the tool
N013 M30;

```

2.65 G89 : Boring Cycle with Dwelling at Hole Bottom

Command Form

G89 X_Y_Z_R_P_F_K_;

X(U) or Y(V): hole position (absolute/increment, please note if parameter 3809 is 1 when using increment value)

Z:

G91: the distance from the R point to point Z (directional)

G90: program coordinate position of Z point

R:

G91: the distance from the starting point to point R (directional)

G90: program coordinate position of R point

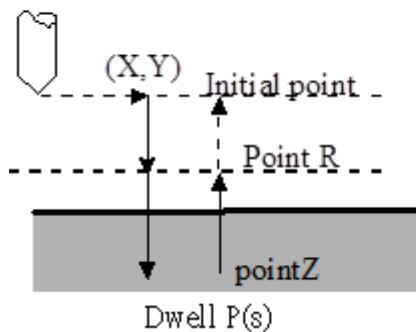
P: dwelling time; time stayed at the bottom of the hole (with decimal points, unit: second; without decimal points, refer to Pr17 and Pr 3241)

F: feedrate

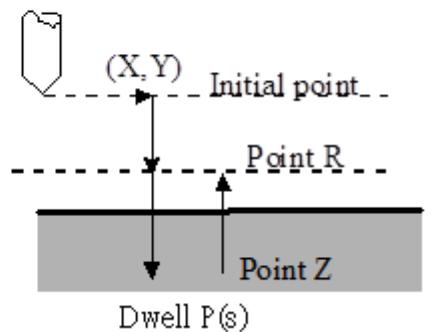
K: repeating times (repeating the moving and drilling actions, G91 increment input is effective)

Absolute/Increment coordinate of X, Y, Z, R can be decided by G90/G91.

G98



G99



2.65.1 Description

1. Start the machining process, move the tool to specified (X,Y) with G00..
2. Move the tool to specified point R with G00.
3. Move the tool to point Z with G01.
4. Dwell for P seconds.
5. Lift up to point R with G01.
6. Lift up to the initial point with G00 (G98) or to the R point of the program (G99).

2.65.2 Note

1. Please start spindle rotation with M code before applying G89.
2. If M Code and G89 are specified in the same block ,the M Code will only be executed once in the block.
3. When specified to repeat K times, M Code will only be executed at first cycle, it won't be executed for others.
4. Before changing the drilling axis, G89 cycle must be cancelled first.
5. If no axis (X, Y, Z) moving command is included in the block, the drilling action won't be executed.
6. The argument specified by R will only be set when executing the block with drilling action, it won't be set in the blocks with no drilling actions.
7. G codes G00, G01, G02, G03 can't be specified in the same block with G89, or G89 command will be cancelled.

8. In G89, the tool radius compensation (G41/G42/G40) will be ignored.
9. Arguments X,Y,Z,A,B,C,U,V,W are recognized as axis command when the parameter 3809 is 0.
10. The increment command for boring axis is ignored when the increment and absolute command for boring axis are commanded simultaneously.

2.65.3 Program Example

```
N001 F1000. S500;
N002 G90;
N003 G00 X0. Y0. Z10.; // move to the initial point
N004 G17;
N005 M03; // start drill CW rotation
N006 G90 G99;
// specified the coordinate of point R, point Z and hole NO.1, dwelling time 2.5 seconds
N007 G89 X5. Y5. Z-10. R-5. P2.5;
N008 X15.; // hole NO.2
N009 Y15.; // hole NO.3
N010 G98 X5.; // hole NO.4, return to the initial point
N011 G80;
N012 M05; // stop the tool
N013 M30;
```

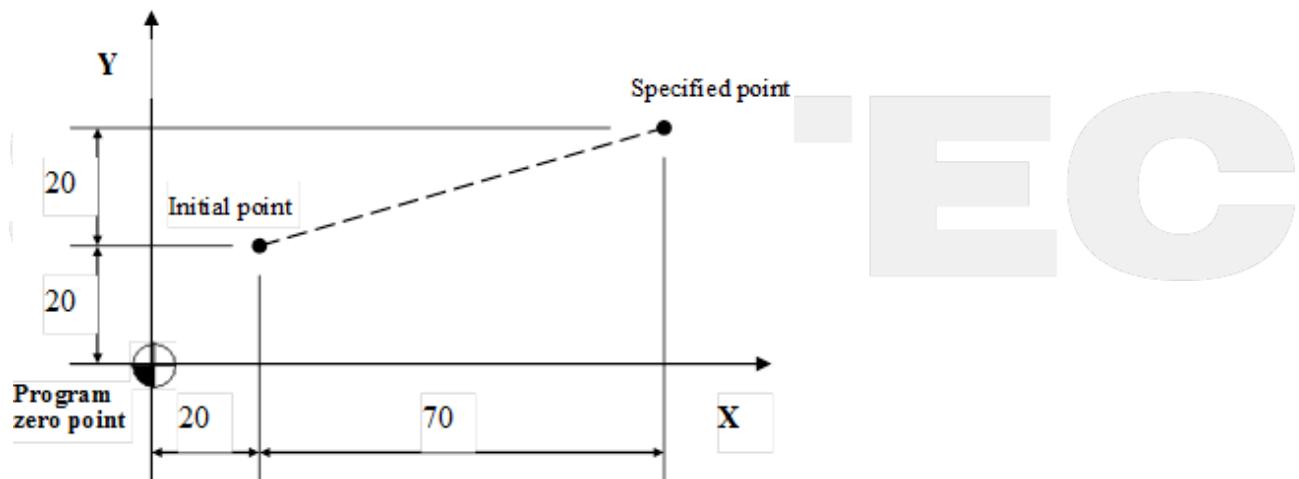
2.66 G90/G91 : Absolute/Increment Command

Command Form

G90;
G91;

2.66.1 Description

G90: absolute command.
G91: incremental command.



2.66.2 Program Example

1. 1st way(absolute): G90 G00 X90.0 Y40.0 ;
// execute the linear interpolation with the distance from program zero point to the specified point
2. 2nd way(increment): G91 G00 X70.0 Y20.0 ;
// execute the linear interpolation with the distance from the starting point to the specified point

2.67 G92 : Absolute Zero Point Coordinate Setup I

Command Form

G92 X_ Y_ Z_;

X, Y, Z: set the program coordinate of current position

For example: The current program coordinate is X10, Y20, Z30, after execute G92 X0 Y0 Z0, the program coordinate will be changed to X0, Y0, Z0.

Description

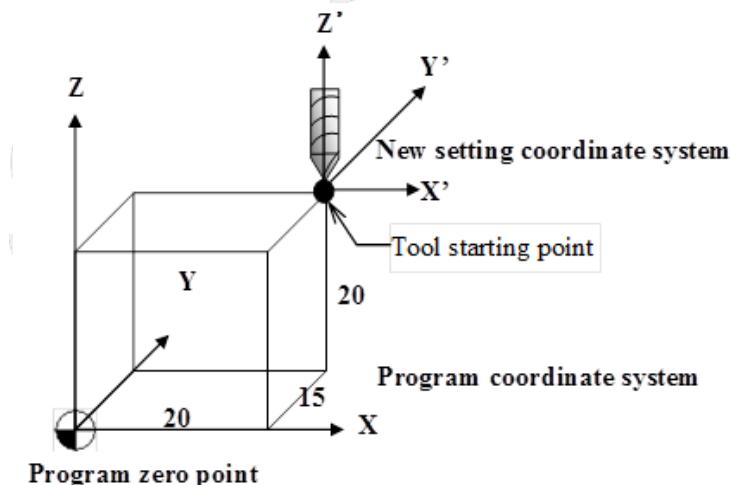
When edit a program, sometimes a new program zero point is required for special situations, thus the G92 command can be used to build a new coordinate system.

This command sets the current tool position as a specified point of the new sub-coordinate system. After setup, the tool will start machining at this point, absolute commands will be computed refer to this new coordinate system.

Note

1. The Machine Coordinate is obtained from the formula : Machine Coordinate = Workpiece Coordinate (G54~) + Program Coordinate + G92 Offset + External Offset + MPG Offset + Tool Length Compensation.
2. G92 Offset is the difference that Program Coordinate before executing G92 minus after running G92.
3. The axial macro-variables of G92 coordinate system's offset are #1901~1918.
4. Please do not use G92 and G92.1 at the same time.

Program Example



G90 // Absolute command

G01 X20.0 Y15.0 Z20.0 // machining to X20.0 Y15.0 Z20.0, the current program coordinate is X20.0 Y15.0 Z20.0

G92 X5.0 Y10.0 Z15.0; // change the program coordinate of current position to X5.0 Y10.0 Z15.0. G92 offset X=15, Y=5, Z=5. The offset value = original program coordinate - new program coordinate

G01 X25.0 Y25.0 Z35.0 // machining to new program coordinate X25.0 Y25.0 Z35.0

2.68 G92.1 : Absolute Zero Point Coordinate System Setup II

2.68.1 Command Form

G92.1 X_ Y_ Z_ I_ J_ K_ R_

X, Y, Z : specified the coordinate as the zero point of program coordinate system;

I : take X axis as the rotation center and rotates the YZ plane.

J : take Y axis as the rotation center and rotates the XZ plane.

K : take Z axis as the rotation center and rotates the XY plane.

R : the rotation angle of the coordinate

2.68.2 Description

G92.1 is similar to G92, both are used to build new coordinate systems. This command sets the specified point (assigned by the command) of current coordinate system as the program zero point of the new sub-coordinate system.

After setup, the tool will start the machining at the specified point and all the absolute commands will be computed with this coordinate system.

Comparing between G92 and G92.1

Command	Description
G92 X20. Y15. Z20.	Set the current position as the X20. Y15. Z20 of new coordinate system
G92.1 X20. Y15. Z20.	Set the X20. Y15. Z20. of current coordinate system as the zero point of new coordinate system

2.68.3 Note

1. The Machine Coordinate of controller is obtained from the formula : Machine Coordinate = Workpiece Coordinate (G54~) + Program Coordinate + G92.1 Offset + External Offset + MPG Offset + Tool Length Compensation.
2. G92.1 Offset is equal to the argument X_, Y_, Z_ in command of G92.1.
3. Argument I_, J_, K_ in G92.1 = The axes of G92.1 coordinate system's rotation center.
4. Argument R in G92.1 = G92.1 coordinate system's rotation angle.
5. The axial macro-variables of G92.1 coordinate system's offset are #1901~1918.
6. The macro-variable of G92.1 coordinate system's rotation angle is #1930. The default value is 0.

7. The axial macro-variables of G92.1 coordinate system's rotation center are #1931~#1933. The default values are 0, 0, 1.
8. #1930~#1933 will be affected by Pr413
 - a. When Pr413 is set to 0, #1930~#1933 will be restored to default value after CNC reset or reboot.
 - b. When Pr413 is set to 1, #1930~#1933 will be restored to default value after CNC reboot. However, these system variables will maintain the value of user's setting after CNC reset.
 - c. When Pr413 is set to 2, #1930~#1933 will maintain the value of user's setting after CNC reset or reboot.
9. Please do not use G92 and G92.1 at the same time.

Program Example

Example 1, comparing between G92 and G92.1 (no external offset, tool length, tool wear compensation)

G92	G92.1
<pre>N1 G90 X10. Y10. //machine coordinate X10. Y10. //program coordinate X10. Y10 //#1901 #1902 coordinate X0. Y0. N2 G92 X20. Y20. //machine coordinate X10. Y10. //program coordinate X20. Y20. //#1901 #1902 coordinate X-10. Y-10. N3 X50. //machine coordinate X40. Y10. //program coordinate X50. Y20. //#1901 #1902 coordinate X-10. Y-10. N4 M30</pre>	<pre>N1 G90 X10. Y10. //machine coordinate X10. Y10. //program coordinate X10. Y10. //#1901 #1902 coordinate X0. Y0. N2 G92.1 X20. Y20. //machine coordinate X10. Y10. //program coordinate X-10. Y-10. //#1901 #1902 coordinate X20. Y20. N3 X50. //machine coordinate X70. Y10. //program coordinate X50. Y-10. //#1901 #1902 coordinate X20. Y20. N4 M30</pre>

Example 2

Program Content	Figure
<pre>N1 G90 G0 X0. Y0. //machine coordinate X0. Y0. //program coordinate X0. Y0. //#1901 #1902 coordinate X0. Y0.</pre>	

Program Content	Figure
<pre>N2 G92.1 X0. Y0. K1. R45. //machine coordinate X0. Y0. //program coordinate X0. Y0. //#1901 #1902 coordinate X0. Y0. //XY plane rotates 45° with rotation center Z axis on program coordinate system, #1930 = 45°</pre>	<p>A 2D coordinate system diagram. The vertical axis is labeled '機械座標Y軸' (Machine Coordinate Y-axis) and the horizontal axis is labeled '機械座標X軸' (Machine Coordinate X-axis). A green dot at the origin is labeled '程式座標X0., Y0.'. A red arrow indicates a 45-degree counter-clockwise rotation from the machine coordinate axes to the program coordinate axes. The text '程式座標旋轉45度' (Program coordinate rotate 45 degrees) is written near the arrow.</p>
<pre>N3 G01 X100. //machine coordinate X70.711 Y70.711 //program coordinate X100.000 Y0.000 //#1901 #1902 coordinate X0.000 Y0.000</pre>	<p>A 2D coordinate system diagram with both horizontal and vertical axes labeled '機械座標' (Machine Coordinate). A green dot is located at coordinates (70.716, 70.716), with dashed lines extending to the axes. The text '程式座標X100., Y0.' is written next to the dot. A red arrow indicates a 45-degree counter-clockwise rotation from the machine coordinate axes to the program coordinate axes. The text '程式座標旋轉45度' is written near the arrow.</p>
N4 M30	

Example 3



Program Content	Figure
<pre>N1 G90 G0 X20. Y20. //machine coordinate X20. Y20. //program coordinate X20. Y20. //#1901 #1902 coordinate X0. Y0.</pre>	
<pre>N2 G92.1 X10. Y10. K1. R45. //machine coordinate X20. Y20. //program coordinate X14.142 Y0. //#1901 #1902 coordinate X10. Y10. //XY plane rotates 45° with rotation center Z axis on program coordinate system, #1930 = 45°</pre>	<p>A 2D coordinate system diagram with both horizontal and vertical axes labeled '機械座標' (Machine Coordinate). A green dot is located at coordinates (14.142, 0), with dashed lines extending to the axes. The text '程式座標X10., Y10.' is written next to the dot. A red arrow indicates a 45-degree counter-clockwise rotation from the machine coordinate axes to the program coordinate axes. The text '程式座標旋轉45度' is written near the arrow.</p>
<pre>N3 G01 X100. machine coordinate X80.711 Y80.711 program coordinate X100. Y0. #1901 #1902 coordinate X10. Y10.</pre>	<p>A 2D coordinate system diagram with both horizontal and vertical axes labeled '機械座標' (Machine Coordinate). A green dot is located at coordinates (80.711, 80.711), with dashed lines extending to the axes. The text '程式座標X100., Y0.' is written next to the dot. A red arrow indicates a 45-degree counter-clockwise rotation from the machine coordinate axes to the program coordinate axes. The text '程式座標旋轉45度' is written near the arrow.</p>
N4 M30	

2.69 G93 : Inverse Time Feed

2.69.1 Command Form

```
G93;  
G01...F_;  
G02...F_;  
G03...F_;
```

2.69.2 Description

This command is the feedrate modal command, used to specify the definition of feedrate. It only needs to be assigned in the program once and will be cancelled till G94/G95 is assigned. This mode only affects the feedrate of G01, G02, G03.

In G93 mode, F only affects the feedrate of it's own block, thus the F argument should be specified in every interpolation block or alarm Cor85: "F argument incorrect in G93" will be issued.

For G01 block in G93 mode, the feedrate is defined as $F * \text{block length}$.

For G02 / G03 block in G93 mode, the feedrate is defined as $F * \text{block radius}$.

2.69.3 Program Example

```
G71  
G93  
G01 X10. F1      // feedrate of this block,  $1 * 10 = 10 \text{ mm/min}$   
G02 X20. R5. F3    // feedrate of this block,  $3 * 5 = 15 \text{ mm/min}$   
G03 X0 R10. F5    // feedrate of this block,  $5 * 10 = 50 \text{ mm/min}$   
M30
```

2.70 G94/G95 : Feed Unit Setup

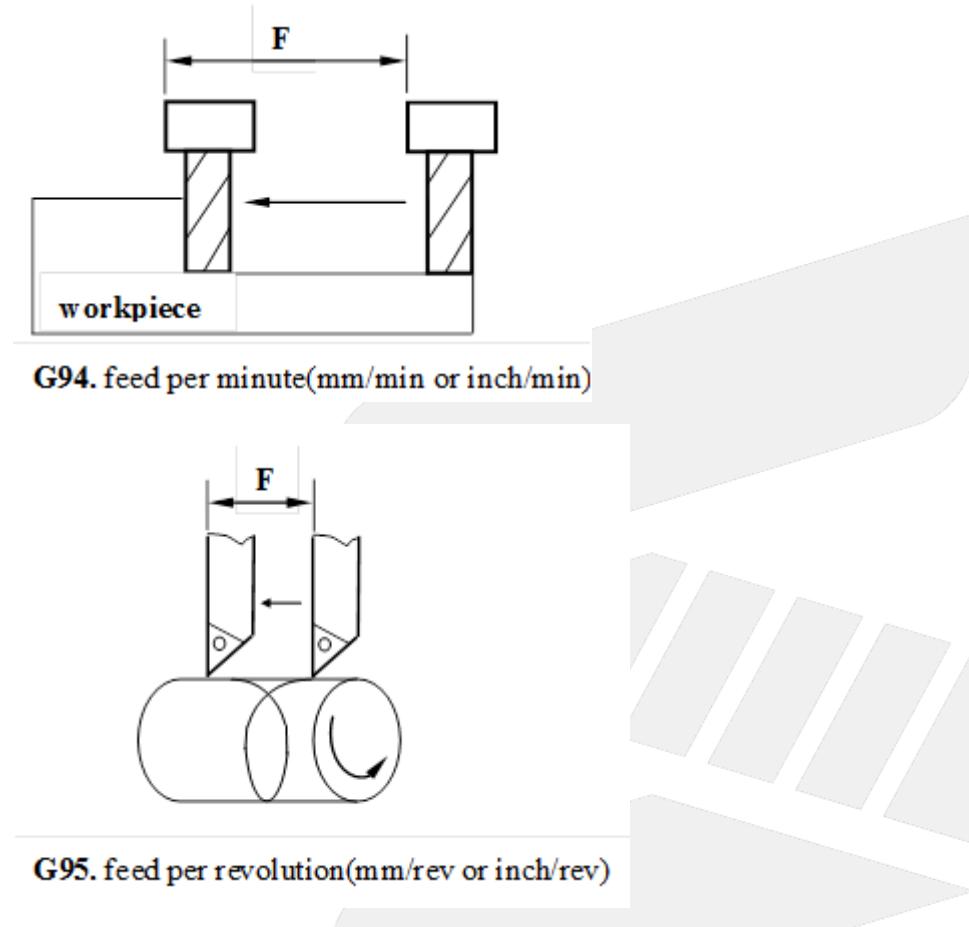
Command Form

```
G94 F_;  
G95 F_;
```

2.70.1 Description

This command set up the feed unit of F_function (tool moving distance per unit time/per revolution). G94 is feed per minute, unit: mm/min, inch/min; G95 is feed per revolution, unit: mm/rev, inch/rev.

2.70.2 Figure



2.71 G96/G97 : Constant Linear Surface Speed Control

Command Form

G96 S_ ; constant linear surface speed control: ON

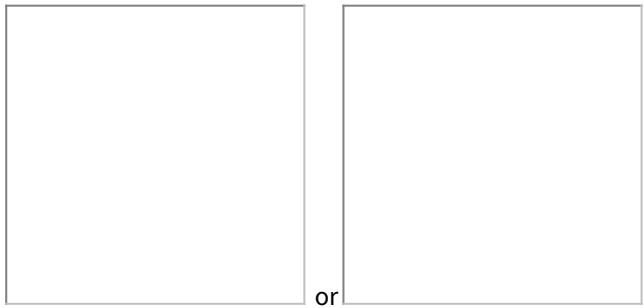
G97 S_ ; constant linear surface speed control: OFF

2.71.1 Description

G96 specified the surface speed, relative speed of the contact surface between the tool and workpiece; G97 command is able to cancel G96 command and it's also the spindle rotation speed command.

- While machining process tools with different radius but requires constant surface speed, applying G96 S_ is able to control the machining speed. For the machining process with constant spindle rotation speed through out the whole process doesn't matter with the tool radius, G97 S can be applied to control the rotation speed.
- When G96 is commanded, the surface speed of spindle will be kept constant in G01、G02/G03. However, the surface speed will be calculated by end position of G00/G53 block, and it will not be kept in constant.

The S value of G96 can be set by :



- V: surface speed, can be specified to a fixed value with G96, unit: m/min or ft/min
- D: effective diameter of the tool, unit: mm/inch
- N: spindle rotation speed, can be specified to a fixed value with G97, unit: RPM.

2.71.2 **Program Example**

Fixed surface speed:

G92 S2000; // restrict the maximum spindle revolution with G92

G96 S130 M03; // the cutting speed is maintained at 130 m/min

Notes: G92 is always applied with G96 to restrict the maximum spindle speed. For the example above, if a 10mm radius tool is applied, then

$$N = \frac{1000 \times 130}{\pi \times 10} = 4140 \text{ rpm}$$

Through restricting the maximum spindle speed to 2000 rpm with G92, it's able to prevent the reduced grip force and workpiece falling caused by the excessive centrifugal force. Therefore, G92 should be applied with G96 for some situations.

Fixed spindle rotation speed:

G97 S1300 M03; // the spindle rotation speed is maintained at 1300 rpm

2.72 G120.1 : Machining Conditions Selection

Command Form

- G120.1 P_ Q _;
P, Q : Call machining conditions, P is the applying state of the machining, Q is the machining condition
- G120.1 P0 or G121
Return to standard parameter, if quick parameters were set (speed, smooth level), the parameter set will be applied automatically.

2.72.1 Descriptions

1. Totally, 9 sets of working conditions are available: P1Q1、P1Q2、P1Q3、P2Q1.....P3Q3。
2. G62/G64 P_ is also available, the range of P argument is expanded to 0~9. For example: P1Q1 → G62P1, P2Q1 → G62P4.
3. Users can select the machining parameter according to the requirements.
4. Able to set up corresponding parameters at the machining condition selection page.

5. Able to apply the default value directly. Before returning to the default value, it's able to see the applying occasions of each default value. (please refer to the picture below)



2.72.2 Note

1. Is only applicable for the 6D/11/21 series milling machine controller.
2. Valid version: 10.116.24B
3. Set up range of P: 0~3
4. Set up range of Q: 1~3
5. No actions will be executed without P, Q arguments
6. If missed the Q argument, it'll be Q1 in default. For example, G120.1 P1 is the same as G120.1 P1 Q1.
7. After Reset, return to standard parameters. If quick parameters are set, the parameter set will be applied automatically.
8. Valid version of Quick parameter: before 10.116.54A (included).
9. Related alarms :
 - a. MACRO-411_wrong command form : alarm issued when missing P argument in G120.1.
 - b. MACRO-412_argument set out of range : P argument range 0~3, Q argument range 1~3, alarm issued when argument set out of range.
 - c. COR-103_incorrect HSHP parameter setup : before setting up multiple parameter sets or applying the default value, the parameter sets might be empty, alarm will be issued if the blank parameter set is called.

2.72.3 Program Example

```
G90
G54
G43 H1
G0 Z0
X0 Y0
M3 S15000
G120.1 P1 Q2 // call machining conditions selection before G01 machining, can also enable at the beginning
```

```
G01 X50. Y50. F3000
```

```
.....
```

```
G00 Z0
```

```
G121 // disable machining conditions selection after the machining process is finished
```

```
G49
```

```
M5
```

```
M30
```

2.73 G134 : Hole Drilling Cycle on Circumference

Command Form

```
G134 X_Y_I_J_K_;
```

X, Y: center position of circumference, affected by G90/G91.

I: radius of circle(r), the unit is specified with G70/G71, must in positive value.

J: angle of first drilling hole.

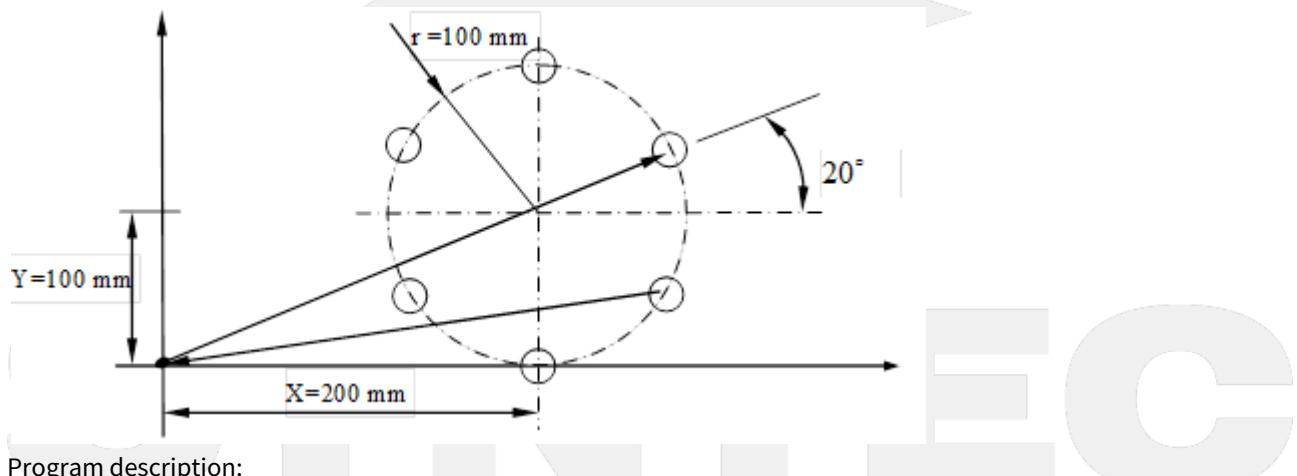
K: number of holes. Range 1~9999, can't be set zero. Positive for CCW, negative for CW.

2.73.1 Description

For the circle center formed at (X,Y) with specified radius, divide it into n parts and drill n holes with the specified angle from X axis.

2.73.2 Example

PIC:



Program description:

```
N001 G92 X500.0 Y100.0; // set up absolute zero point coordinate system
```

```
N002 G91 G81 Z-10.0 R5.0 K0 F200;
```

//execute the drilling cycle, feedrate 200 mm/min, depth 10 mm, and return to initial point after finished

```
N003 G134 X200.0 Y100.0 I100.0 J20.0 K6;
```

//execute the hole drilling cycle on circumference, drill the first hole at X=200mm,Y=100mm, radius 100mm, starting angle 20 degrees, total 6 holes

N004 G80; // cancel the cycle

N005 G90 G0 X0.0 Y0.0; //return to the zero point of coordinate system

2.74 G135 : Linear Hole Drilling Cycle with Angle

Command Form

G135 X_ Y_ I_ J_ K_;

X, Y: coordinate of starting point, affected by G90/G91.

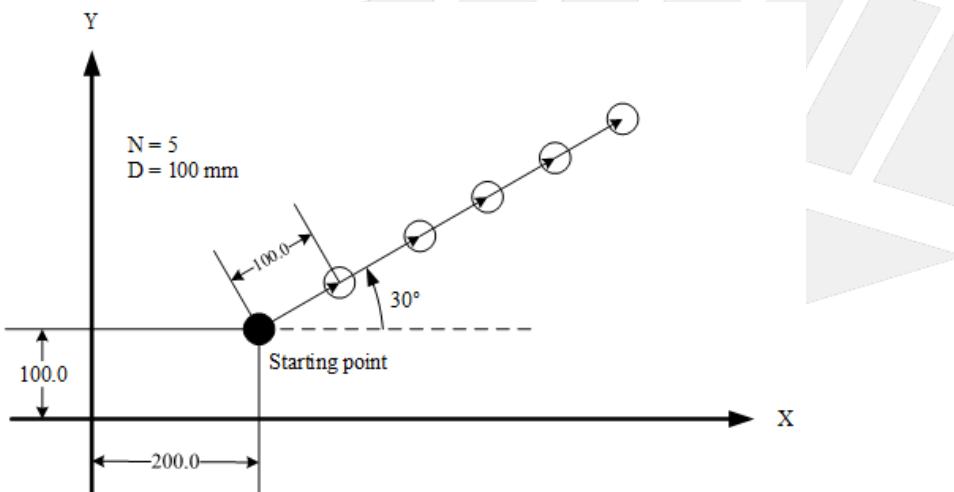
I: interval, the unit is specified with G70/G71, if it is negative then the drilling cycle will be executed in the opposite direction related to starting point.

J: angle between horizontal direction, positive for CCW.

K: number of holes, include the starting point, range 1~9999.

2.74.1 Example

Start the action at the specified (X,Y) coordinate, then execute the drilling cycle in the direction with the specified angle from X axis and drill after each interval.



N001 G91; //enable increment mode

N002 G81 Z-10.0 R5.0 K0 F100 ;

//execute the drill cycle, feedrate 100mm/min, depth 10 mm, return to starting point after finished

N003 G135 X200.0 Y100.0 I100.0 J30.0 K5 ;

//execute the linear hole drilling cycle with angle, starting point: X=200mm, Y=100mm, gap: 100mm, angle between machining direction and X axis: 30 degrees, total 5 holes

2.75 G136 : Hole Drilling Cycle on Arc

Command Form

G136 X_Y_I_J_P_K_ ;

X, Y: center coordinate of arc, affected by G90/91.

I: radius of arc, the unit is specified with G70/G71, positive value.

J: angle of first drilling hole, positive in CCW.

P: interval of angle, positive in CCW.

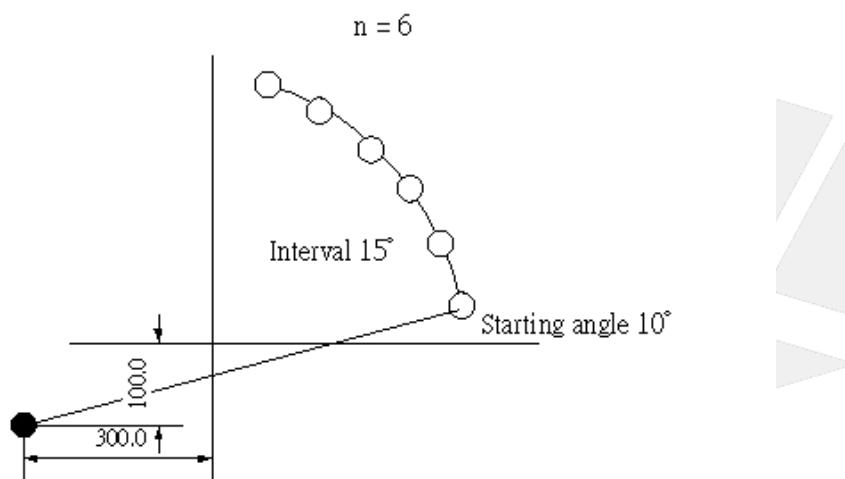
K: number of holes.

2.75.1 **Description**

For the arc formed with center at specified (X,Y) with assigned radius and starting angle, start the drilling cycle from the starting point and drill after moving each interval.

2.75.2 **Program Example**

PIC:



Program description:

G91; // enable increment mode

G81 Z-10.0 R5.0 K0 F100 ;

//execute the drilling cycle, feedrate 100mm/min, depth 10 mm, then return to the start point

G136 X300.0 Y100.0 I300.0 J10.0 P15000 K6 ;

//execute the hole drilling cycle on arc, arc center: X=300mm,Y=100mm, radius 300mm, starting angle 10 degrees, interval angle 15 degrees, total 6 holes

2.76 G137.1 : Matrix Hole Drilling Cycle

Command Form

G137.1 X_Y_I_P_J_K_;

X, Y: coordinate of starting point, affected by G90/91.

I: X axis interval, the unit is specified by G70/G71, if the value is set positive ,then executes in the positive direction from starting point, if it is set negative ,then executes in the negative direction.

P: number of holes in X axis direction, range 1~9999.

J: interval in Y axis direction

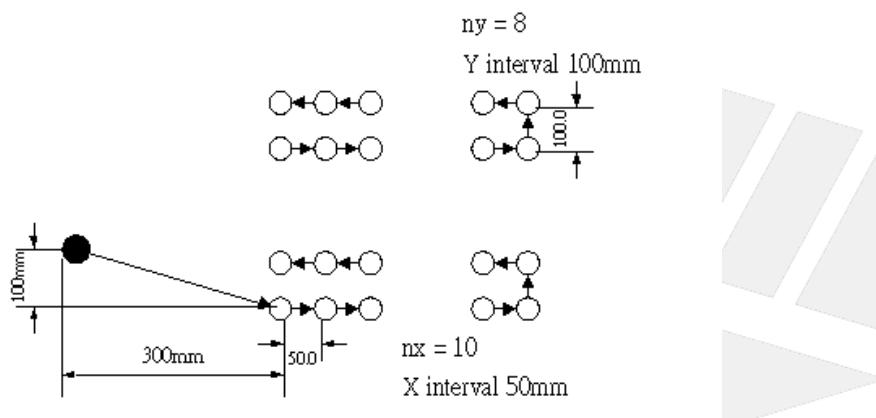
K: number of holes in Y axis direction, range 1~9999.

2.76.1 **Description**

The starting point is specified at (X,Y). Drill nx holes with specified interval in the direction parallel to X axis and drill ny holes with specified interval in the direction parallel to Y axis.

2.76.2 **Program Example**

PIC:



N001 G91; // enable increment mode

N002 G81 Z-10.0 R5.0 K0 F20;

// execute the drilling cycle, feedrate 20mm/min, depth 10mm

// then return to the start point

N003 G137.1 X300.0 Y-100.0 I50.0 P10 J100.0 K8;

// execute the matrix drilling cycle, staring point X=300mm,Y=-100mm

// X axis interval 50mm, total 10 holes, Y axis interval 100mm, total 8 holes

2.77 Cycle Machining Function:

G Code	Drilling Action	Action at Hole Bottom	Retract Action	Application
G73	Intermittent Interpolation feed	----	Rapid movement	High Speed Peck Drill Cycle

G74	Interpolation feed	Spindle CW rotation after dwelling	Interpolation feed	Left Hand Tapping Cycle
G76	Interpolation feed	Spindle orientation stops and shift an offset value	Rapid movement	Fine Boring cycle
G80	----	----	----	Cancel Cycle
G81	Interpolation feed	----	Rapid movement	Drilling Cycle
G82	Interpolation feed	Dwell	Rapid movement	Drilling Cycle with Dwelling at Hole Bottom
G83	Intermittent Interpolation feed	----	Rapid movement	Peck Drill Cycle
G84	Interpolation feed	Spindle reverse after dwelling	Interpolation feed	Tapping Cycle
G85	Interpolation feed	----	Interpolation feed	Drilling cycle
G86	Interpolation feed	Spindle stop	Rapid movement	High Speed Drilling Cycle
*G87	Interpolation feed	Spindle CW rotation	Rapid movement	Fine boring cycle on Back Side
*G88	Interpolation feed	Spindle stop after dwelling	Manual movement	Semiautomatic Fine Boring Cycle
G89	Interpolation feed	Dwell	Interpolation feed	Boring Cycle with Dwelling at Hole Bottom

2.77.1 Address and Meaning of Fixed Cycles:

Address	Address meaning
G	Selection of fixed cycle
X	Specify the drilling coordinate (increment or absolute)

Y	Specify the drilling coordinate (increment or absolute)
Z	Specify the hole bottom coordinate (increment or absolute)
P	Specify the dwelling time at hole bottom
Q	Specify the interpolation feed in G73, G83, or movement (increment) in G76/G87
R	Specify the R position (absolute or increment)
F	Specify the feedrate
K	Specify the cycle repeating times 0~999

The drilling axis can be set with G17, G18, and G19, list as below:

G Code	Orientation Plane	Drilling Axis
G17	X-Y plane	Z axis
G18	Z-X plane	Y axis
G19	Y-Z plane	X axis

2.77.2 Return to point R

When the tool reaches the hole bottom, the tool can return to the initial point or R point, which is decided by G98/G99. G98: back to initial point, G99: back to R point.

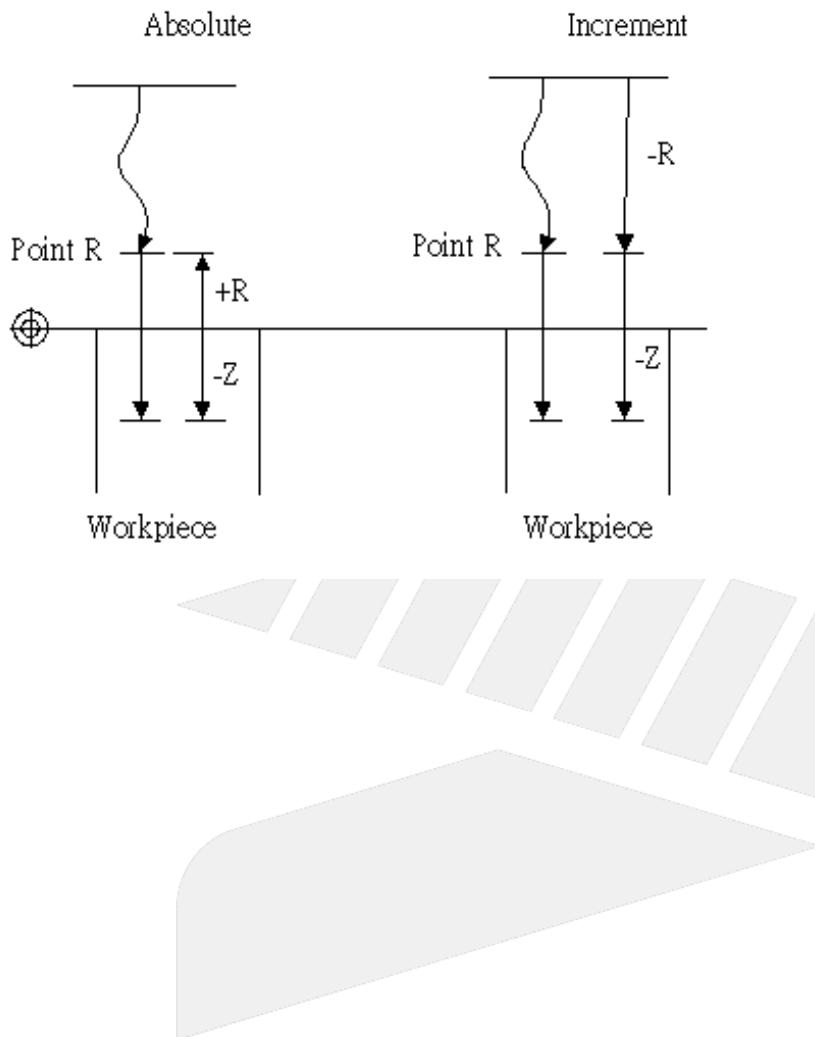
2.77.3 Repeating Times K

When machining multiple holes with constant distance, it's able to specified the number of holes K, the range of K is 0~9999. But for the position of 1st hole, it must be specified with increment mode (G91), or the drilling cycle will be executing at the same position repeatedly. When K=0, the argument of drilling will be set, but the X,Y moving command set in the block won't be executed, neither the drilling action.

2.77.4 Cancel Cycle

Cycles can be cancelled with G80 or group NO.01 G codes (G00/G01/G02/G03...etc.)

2.77.5 Increment (G91)/ Absolute(G90) mode:



SYNTEC

3 M Code Command Description.

The supporting function is used to control the ON and OFF of the machine function. There are two digit numbers in its command form; the corresponding number and functions are listed below :

M Code Function Table

M Code	Function
M00	Program Pause
M01	Optional Stop
M02	Program Ends and Return to First Line
M30	Program Ends and Return to First Line
M96	Interrupting Subprogram Call Function ON (Depend on Pr3600)
M97	Interrupting Subprogram Call Function OFF (Depend on Pr3600)
M98	Call Subprogram
M99	Subprogram Return to Main Program
M198	Call External Subprogram

1. M00 : Program Pause

When CNC executes the M00 command, the spindle rotation will be paused, the feed will also be paused and the cutting fluid will be turned OFF so the examination and compensation could be done more properly. It's able to control the program pause by the "M00 signal switch" on the operation panel.

This M code can't be logged as Pr3804 workpiece counting.

2. M01 : Optional Stop

The function of M01 is similar to M00, but M01 can be controlled alternatively. When the switch is ON, M01 is valid and will stop the program; when the switch is OFF, M01 is invalid.

This M code can't be logged as Pr3804 workpiece counting.

3. M02 : Program End

The same as M30.

4. M30 : Program End

M30 command is the end of program, all the actions will be stopped when the program executes M30 and return to the initial position of the program.

5. M96/M97 : Interrupting Subprogram Call Function

Please refers to [M96/M97- Interrupting Subprogram Call Function](#)

6. M98 : Subprogram Calling, need to be applied with M99

Command Form : M98 P_H_L_

P : The number of the subprogram calling (When P is skipped means the program itself can only be operated in the memory or in MDI operation mode)

H : The serial number of the subprogram calling (N) (Starts from the beginning if it's skipped)

L : Repeating times of the subprogram

- Description

- i. The content of subprogram are usually fixed machining process or commonly used parameters, The subprograms are prepared first and saved in the memory, can be called by the main program when needed. The calling of subprogram is executed by M98 and finished with M99.
- ii. If M02 or M30 command is included in the subprogram, the subprogram will be defined as finished then return back to the main program and proceed the machining.

The M code won't be counted in the Pr3804 workpiece counting

7. M99 : Return to Main Program

Command Form : M99 P_

P : The serial number of the executing block (N) before returning to the main program, if there is no P argument then it means the machining proceeds from the next line of M98 or M198 after returning to the main program.

8. M198 : Subprogram Control need to be applied with M88

Command Form : M198 P_ H_ L_

P : The number of the subprogram calling (P must be specified, or alarm COR-052【Subprogram calling P code must be an integer】will be issued)

H : The serial number of the subprogram calling (N) (Starts from the beginning if it's skipped)

L : Repeating times of the subprogram

The subprogram content will be opened and read again when executing M198, thus the subprogram content will always be the latest version.

The M code won't be counted in the Pr3804 workpiece counting

3.1 M96/M97- Interrupting Subprogram Call Function

M96/M97: Interrupting Subprogram Call function

- Command Form

Take Pr3600 = 96 for example

a. M96 P_[I_][Q_][R_][L_] : Start interrupting subprogram call function

- i. P argument

Argument Description	Specifies the subprogram number of the call when the interruption triggered.
Argument Unit	-
Argument Range	[1 ~ 9999]
Precaution	<ul style="list-style-type: none"> a. Subprogram name should start with 'O'. b. P argument cannot have filename extension, and the called subprogram cannot have filename extension either. Ex: When argument P1234 is executed, the subprogram O1234 is called instead of O1234.txt since these two files are treated as different files in kernel.
Input Example	<ul style="list-style-type: none"> a. Subprogram name O1111, use argument P1111. b. Subprogram name O1213, use argument P1213. c. Subprogram name O0001, use argument P1.

- ii. I argument

Argument Description	Interruption signal source
Argument Unit	-
Argument Range	[1 ~ 3]
	1: the interrupted signal is R-bit
	2: the interrupted signal is I-bit
	3: the interrupted signal is A-bit

iii. Q argument

Argument Description	Interrupt signal number
Argument Unit	-
Argument Range	Varies according to I argument.
	I=1(R-bit): [0 ~ 65535][00 ~ 15]
	I=2(I-bit): [0 ~ 511]
	I=3(A-bit): [0 ~ 511]
Input Example	a. Interrupted signal R49, use argument I1 Q4900. b. Interrupted signal R51.1, use argument I1 Q5101. c. Interrupted signal R50.11, use argument I1 Q5011. d. Interrupted signal A350, use argument I3 Q350.

iv. R argument

Argument Description	Trigger method
Argument Unit	-
Argument Range	[0 ~ 1]
	0: Upper-edge triggered
	1: Lower-edge triggered

v. L argument

Argument Description	Signal maintenance time
Argument Unit	ms
Argument Range	[0 ~ 2,147,483,647]

b. M97: Close interrupting subprogram call function

- Trigger Signal

- If the trigger signal source is not specified, then the preset signal is C49. If the trigger signal source is specified, only the specified trigger is valid, and C49 is ignored.
- If M96 P__ is used and the trigger source is C49, the executing program stops immediately and calls the interrupting subprogram when this C Bit On.
- If the multiple axis groups use M96 P_I_Q_R_L to specify the signal source, each axis group can specify different signal sources, and each axis group only refers to the signal source specified by the respective axis group.

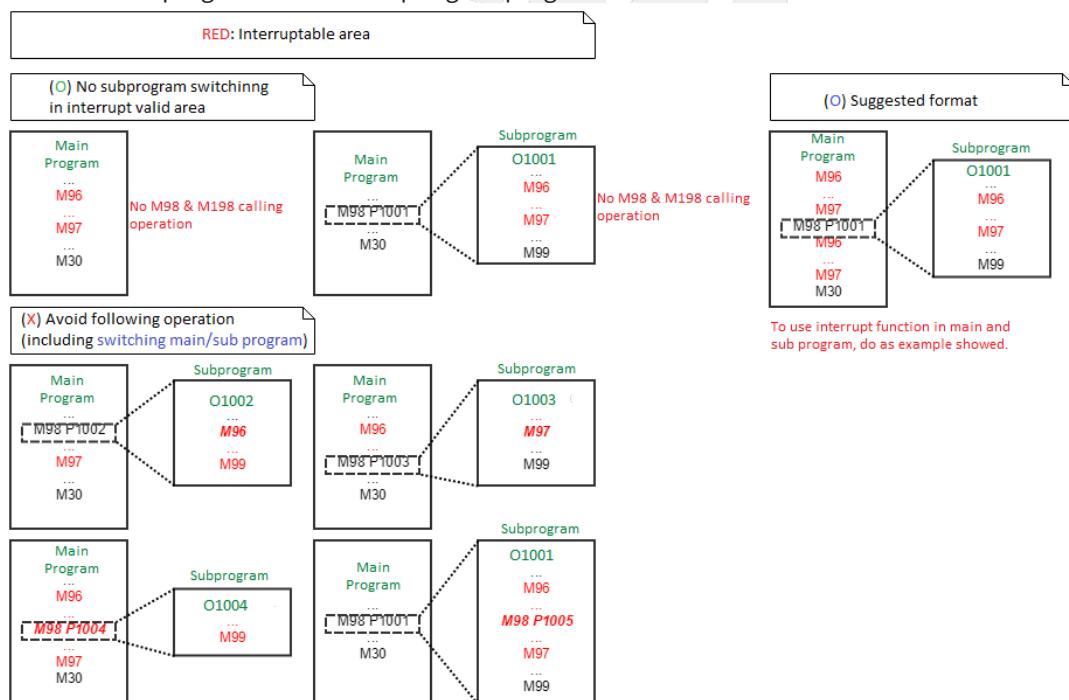
- Description

- Return to main program method: Use M99 in the interrupting subprogram, where:
 - M99 (without argument): Return to the coordinates of the of interruption with G00 and reinterprets it from the interrupted block.
 - M99 PXXXX: Return to the specified N-number block to start interpretation (no G00 return action). If the specified return block number (N) does not exist, trigger alarm COR-017.
 - M99 QXXXX: Return to the specified row number to start interpretation (no G00 return action). If the specified return number does not exist, trigger alarm COR-018.

- The interrupt signal is not supported within the subprogram; triggering in the subprogram may cause wrong interruption return row number problem.

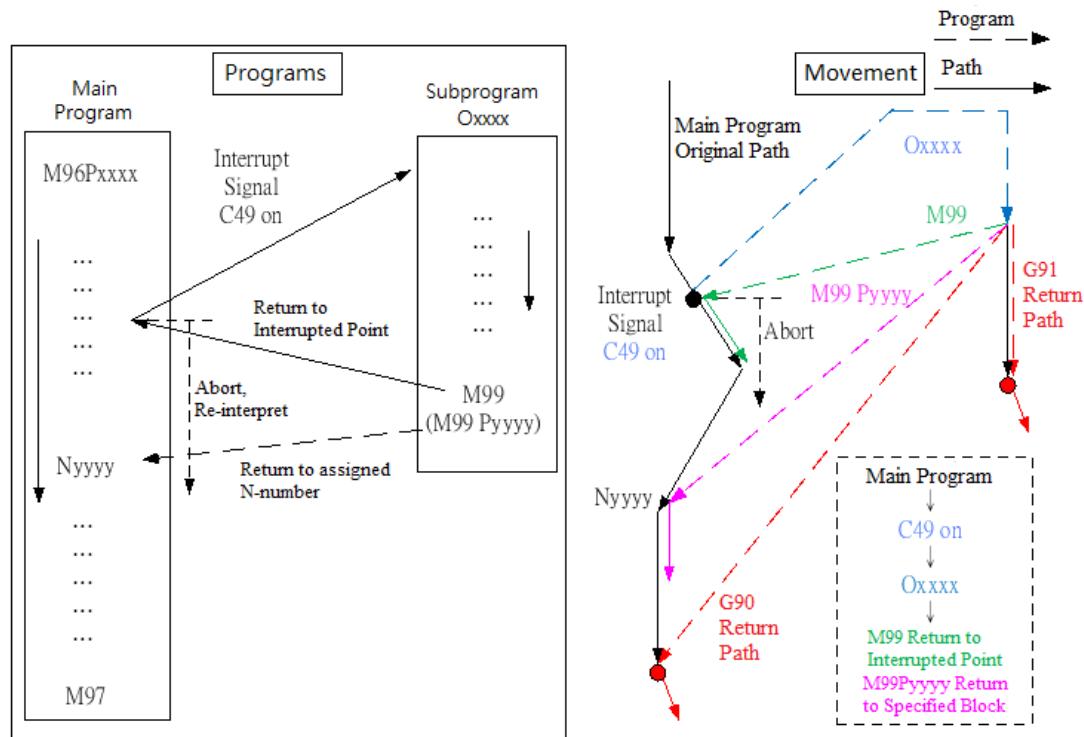
PS: That is, calling the subprogram with M98 or M198 or returning from a subprogram within the rows between M96 and M97 are not allowed. Please refer to the figure below for how to use it.

PS: Please note that the above limit refers to the interruption trigger valid area; M98 can be used to call another subprogram in an interrupting subprogram.



- The M96 M97 command will block the pre-interpretation so that axes are decelerated to zero.
- If there are multiple M96 in the program, the interrupt subprogram number is determined by the M96 P argument that is closest to the interrupt signal.

- e. If M96 is issued in the program, M97 must be issued to cancel the function before the main program ends, otherwise alarm COR-117 [Interrupt type subprogram does not issue M code] will be triggered.
- f. When M96 P_[I_][Q_][R_][L_] is issued in the program, if the I, Q, P, R, L arguments are beyond the specified range, the alarm COR-330 [illegal interrupt signal format] will be triggered.
- g. G02(G03) I_J_K_ and ,A_,R_,C_, these geometric related functions will be affected by the starting point of the single block. Therefore, if the interruption occurs in the block, an alarm will be triggered or may have different path from the original.
- h. The interrupt subprogram inherits the state of the main program interrupt point, including G, S, T, and so on.
 - i. S, T, etc, these pre-interpretation blocking commands will correctly inherit the status when entering interruption point.
 - ii. G, F, etc., these pre-interpretation commands **will inherit the pre-interpreted status**, please be careful when programming.
PS: For example, if the main program receives an interrupt signal in the execution of G00X50., the initial interpolation state is not necessarily G00 when entering to the subprogram.
- i. In the figure below, main program's G00 Z100. (starting position 0.) block is interrupted and stopped at the position of Z35. When returning from interrupting subprogram Oxxxx, system returns to interrupt point Z35 and then execute G00 Z100. If using G90 mode to return to interrupted block at Z35, it will move to Z100.; If using G91 mode, it will move to Z135 after returning to interrupted block. (Only C type needs to be notice this description).



Whether M99 returns to the main program interrupted point or M99Pyyyy returns to the main program block Nyyyy, it is both re-interpreted. So if G91 mode is used, need to pay attention to whether the processing path meets the requirements (Only Mill and Lathe C type need to notice this description).

- j. The interrupting subprogram function can't work when using followings function

G5: High speed high precision function

G5.1: Path smoothing function

G12.1: Polar coordinate interpolation

G16: Polar coordinate transformation

G41(G42): Tool radius compensation

G51: Scaling function

G51.1: Mirror function

G51.2: Polygon cutting

G114.1: Spindle synchronization

G114.3: Spindle bearing function

When the interrupting function is executed, if the controller is in these modes, the interrupt function (C49, and the specified trigger signal under the command) will not be enabled.

- k. The M96 is canceled when Emergency Stop, M30, or Reset are triggered.

- l. When executing Feedhold or the Block stop (M00/C40), the M96 trigger will be paused. The signal hold time will be paused and not be cleared until the restart (Cycle Start) and then resumed M96 to interrupt Trigger and continue timing.

- Program example

```
// main program
M96 P1111
G00 X0 Y0 Z0
G01 X10. F500
Y10.
X0
Y0
M97
M30
// O1111 (interrupting subprogram) simulate the Z-axis ascend to check tool and descend back
%@MACRO
#30 := #1000; //mode backup: G00/G01/G02/G03
#31 := #1004; //mode backup: G90/G91
G00 Z100.; // rapif positioning to the Z-axis tool checkpoint
G#30 G#31; // mode restore
M00; // After entering M00, it is allowed to switch to manual mode for axial movement.
M99; // return to interrupted point
```

- Precaution

- Versions after 10.116.10 support M96/M97 as "interrupting subprogram call function M code".

- Versions after 10.116.24Y/10.116.36E (included) provide Pr3600 *Define interrupting subprogram call function M code to set the value of subprogram call M code.

- As mentioned above, when the Pr3600 is set to the same value as the extension M code parameter (Pr3601~) or the part count M code (Pr3804), the OP-020 alarm will be triggered. Please correct accordingly.

- Versions after 10.118.10 support M96 P_[I_] [Q_] [R_] [L_] command which can specify trigger signals other than C49.

- After 10.118.12E, 10.118.15, if M96 has specified a trigger signal, the interrupted subprogram only follows the specified trigger signal and ignores C49 signal.

4 T Code Command : Tool Function

Command Form

T

4.1 Description

The tool function is also called T function, mainly used to assign the tools. Normally is applied with tool switching command (M06) to switch the tool automatically.

4.2 Program Example

T03 M06; // represents switch to tool No.3

SYNTec

5 F Code Command : Feed Function

5.1 Command Form

5.2 F

5.3 Description

When cutting a workpiece, the tool moving speed is called feedrate. There are two units to set up the feedrate, mm/min (G94) and mm/rev (G95). For G94, F300 represents the feedrate is 300 mm/min; For G95, F0.5 represents 0.5mm/rev.

5.4 Program Example

```
G94 G01 X100.0 Y100.0 F300 ;  
// linear cutting, feedrate 300mm/min  
G95 G01 X100.0 Y100.0 F0.5 ;  
// linear cutting, feedrate 0.5mm/rev
```

SYNTec

6 S Code Command : Spindle Rotation Speed

Command Form

S_

6.1 Description

S function is the spindle speed command, used to specify the spindle rotation speed in rpm or assign cutting speed by G96/G97.

6.2 Note

When switching between different spindle, if the current machining spindle is 2nd spindle and 1st spindle is assigned to run 150 rpm CW rotation, apply M03 S1=150 to avoid the delay of machining spindle changing, otherwise rotation speed will thus affect 2nd spindle.

6.3 Program Example

```
G96 S150 M03 ; // machining surface speed remains at 150 m/min  
G97 S500 M03 ; // spindle speed remains at 500 rev/min
```

SYNTEC